PULSONIX

Pulsonix Design System

V7.5 Update Notes

Copyright Notice

Copyright © WestDev Ltd. 2000-2012
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: 24/01/12 iss 2

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

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

Contents

CONTENTS **3**
CHAPTER 1. VERSION 7.5 UPDATE SUPPLEMENT **5**
Installing the New Version of Pulsonix 5
Licensing 5
Windows 7 Support 5
Report Maker Enhancements 6
Snap Cursor to Grid 23
Multiple File Extensions 24
CAM Plot Changes 25
Enable a Named Group of Plots from Context menu 25
Move Additional Plot Items In CAM Plot Option 25
Copy Plot 25
Crop Plot Output 26
PDF Output Bookmarks 26
PDF Output Variants 27
Gerber Verification Plot to PDF 27
Associated Parts – Execute Hyperlink 28
Local Pin Types 28
Component Variant Spreadsheet 30
Delete Variant Changes 35
Attribute Editor shows Variant ID 35
Swap Component Positions 36
Part Edit Changes 36
Footprint Dialog Filter 36
Ability to Resize Footprint/Symbol Dialogs 37
Change Connector Part Type option 37
Delete Segment on Unextended Delete 38
Net Class - Mark All Nets As One 38
Replace Part Changes 39
Insert Multiple Item Changes 39
Auto-Rename in Schematic Editor 41
Schematic Bus can have Own Colour 41
Show Net Name on Change Net 42
Pad Style Spacing Shapes 43
Changes to 3D Viewer 46
New 3D Package Shapes 46
Axial Components - Rounded Ends 46
Colour-coded Bands on Components 46
3D View - Print With White Background 47
3D View - Save As Image 47
Layer Type Choice for Component Body 47
Find Bar – Drill Sizes in Table Units or Design Units 48
Spacing Wildcard Match 48
Wildcard Strings 49
Spacing Rule Class to Class Match 49
Copy Pad Style 50
Suppress Pads on Powerplanes 50

Highlight Locked Tracks....51
Draw Pads in Top Most Layer Colour....51
Net Styles Dialog Changes....52
Via Protection Against Automatic Changes – Default Settings....53
Add Track Option – Use Style From Picked Track....53
New Context options in Layers Bar....54
New Commands for Change Layer....54
Component Pushing uses Accurate Shapes....55
IDF Output Enhanced....55
DXF Import Enhanced....57
Scale Import Plot....57
Align DXF 0,0 to Origin....57
Apply to Entire Column....57
Copy Shape Layer Map to Text Grid....58
STEP I/O Improvements....58
Design Rules Check – Check for Missing Teardrops....60
Online DRC – Check Via in Pad....60
Online DRC – Check Comp to Comp....60
High Speed Option - Differential Pairs – Spurs Allowed Check....61
Changes to Calculators....62
Ariadne Import (Cost Option)....62

Chapter 1. Version 7.5 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 7.5 on top of your existing installation or along side if you prefer; however, you do not need to uninstall the old version first.

Licensing

Version 7.5 uses a new license. This will be supplied to you by email under the contract conditions of your current maintenance agreement.

If you are using a version of Pulsonix earlier than Version 7.x, you will require a new license.

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.

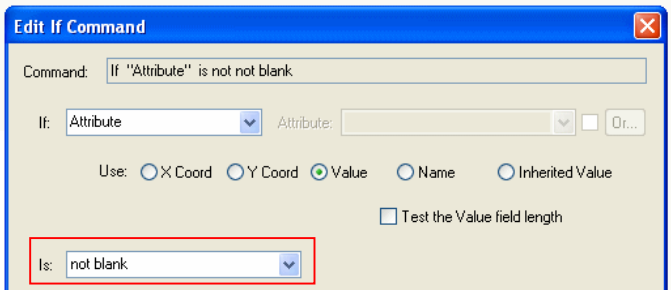
Windows 7 Support

As further confirmation, since the release of Pulsonix 6.1 (and for later versions) the release version of the Windows 7 operating system in both 32 and 64 bit variants is fully supported. No earlier Pulsonix versions can or will be supported. Also, Beta or pre-release versions of Windows 7 cannot be supported.

Report Maker Enhancements

New commands – Is Blank and Is Not Blank

When using an **If** Command, it can now have the test **Is Blank** and **Is Not Blank**. These only work on text fields. These commands also exist when using the **While** command on a text variable. They can be used to test for empty fields.

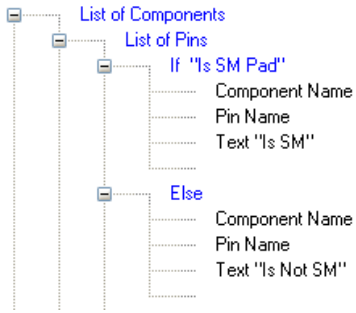


New command - Is All SM Pads

The new **Is All SM Pads** command is used on the **List Of Components** command to test if all pads on the component are surface mounted, it reports True or False. It can also be used within an **If "Is All SM Pads"** command.

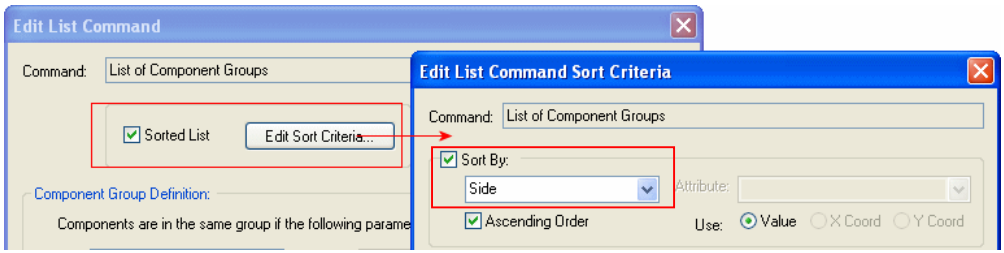
The new **Is SM Device** command is similar to the new **Is All SM Pads** command but looks at Mounting Holes as well.

The new **Is SM Pad** command can be used on a single component pad to test if it is surface mount or not. This would be used within a **List of Pins** command or **If "Is SM Pad"** command to test the pad condition.



List of Component Groups by side

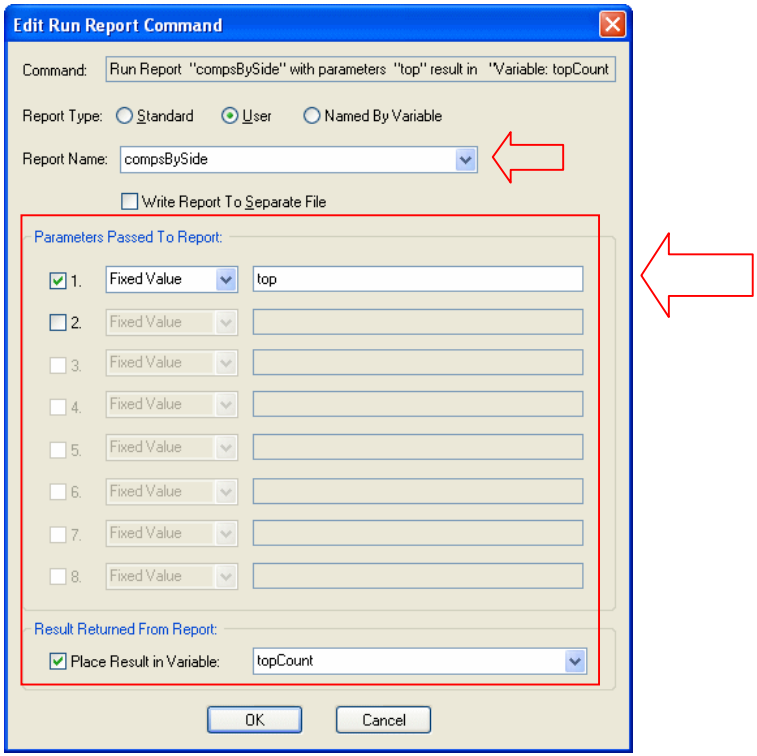
You can now sort **List of Component Groups** by **Side**. The **Side** command appears under the **Sort By:** drop down list on the **Edit Sort Criteria** page. If you want to do this, ensure that Side is also part of the Component Group definition. This way, all the components on the top side will be reported separately to all the components on the Bottom side.



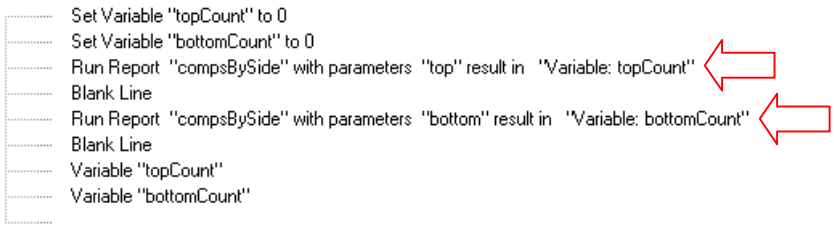
Ability to pass variables from one report to another

New commands allow common report format files to be shared or run as sub-routines.

To do this, the **Run Report** command dialog now has 8 optional parameters (supplied either by variable or fixed value) that can be passed to the report being run. There is also an optional variable to contain a return value from the run report (**Place Result in Variable**).

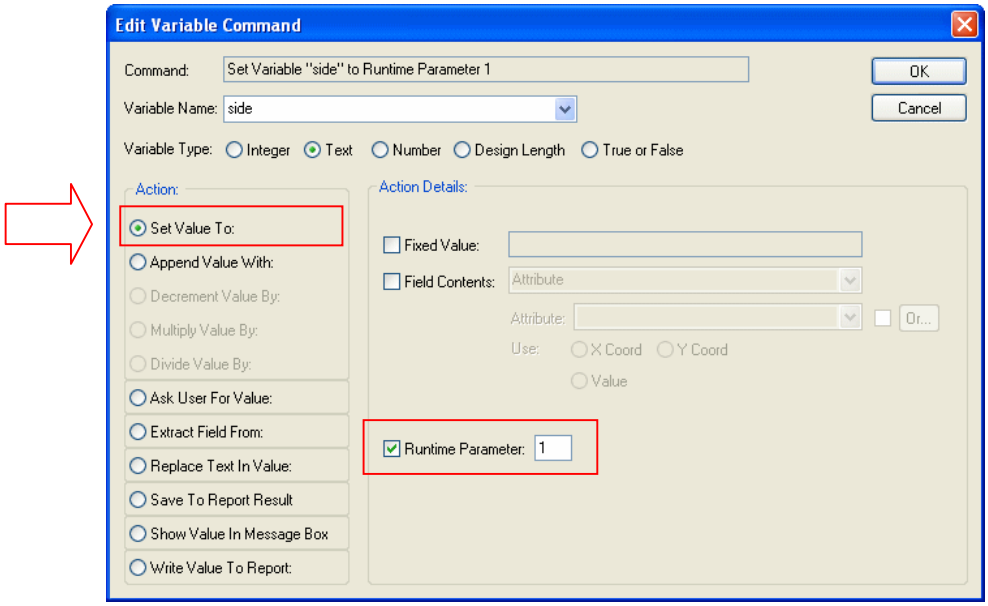


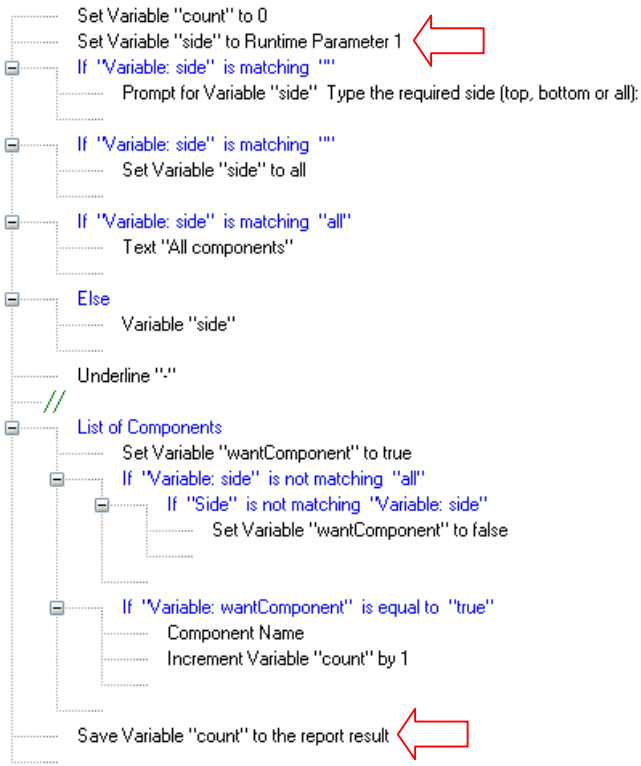
An example Report Maker script below running this (*compBothSides*) demonstrates how this command may be used:



In the report being run (*compBySide* in our example above), the parameters can be picked up by setting a Variable to their value. e.g. **Set Variable 'side' to Runtime Parameter 1**, and you can save the value of a Variable to the report result for passing back to the parent report. For example, **Save Variable 'count' to the report result**.

The example script being called from the first script (*compBySide*) demonstrates how these new commands may be used:





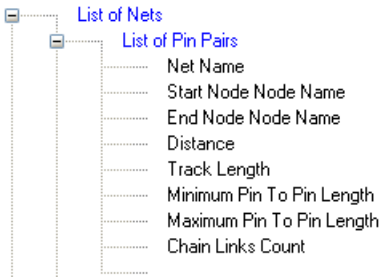
If the script above is run stand alone, i.e. does not have the Variable Side passed in from a prompt report it prompts the user for a a side value. This new command is documented further down in the Report Maker changes section *User Input Prompted with Drop Down List.*

The **Save To Report Result** selection within the **Variable** command is used to save the current value of this variable as the report result. This is only relevant if the report has been run by a parent report using the **Run Report** command. The parent report can then retrieve this result and place it into its own result variable to use in its report.

Report pin to pin track lengths

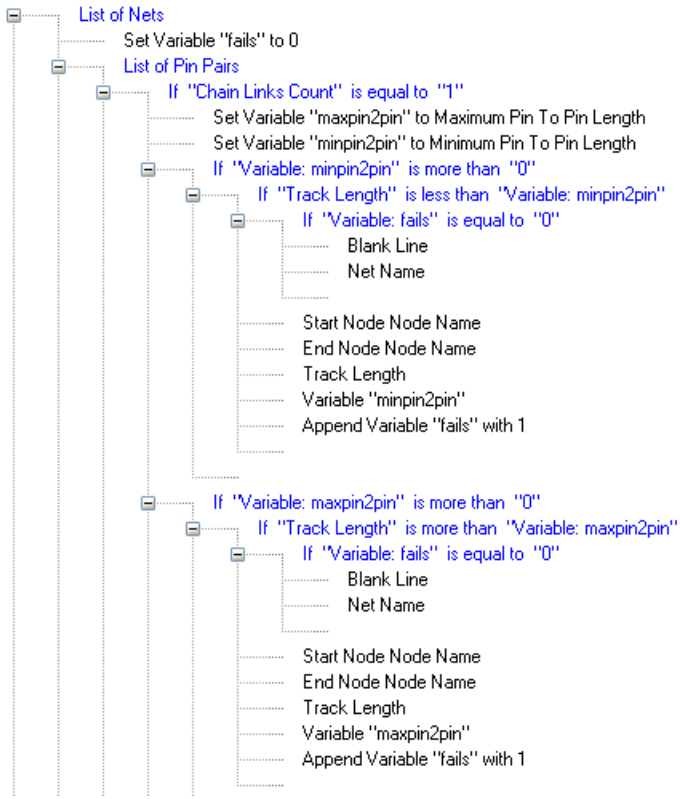
The ability to report pin to pin track lengths has been added. For a net you can now report or place into a variable the **Maximum Pin to Pin Length** and **Minimum Pin To Pin Length** value from its net class.

You can now report a list of the component pin pairs for a net using the **List of Pin Pairs** command. Each unique pair of pins will be reported.



The **Chain Links Count** for a **Pin Pair** is the number of pads you pass through to get from one to the other, plus 1. This enables you to test for the shortest path between pins and report it.

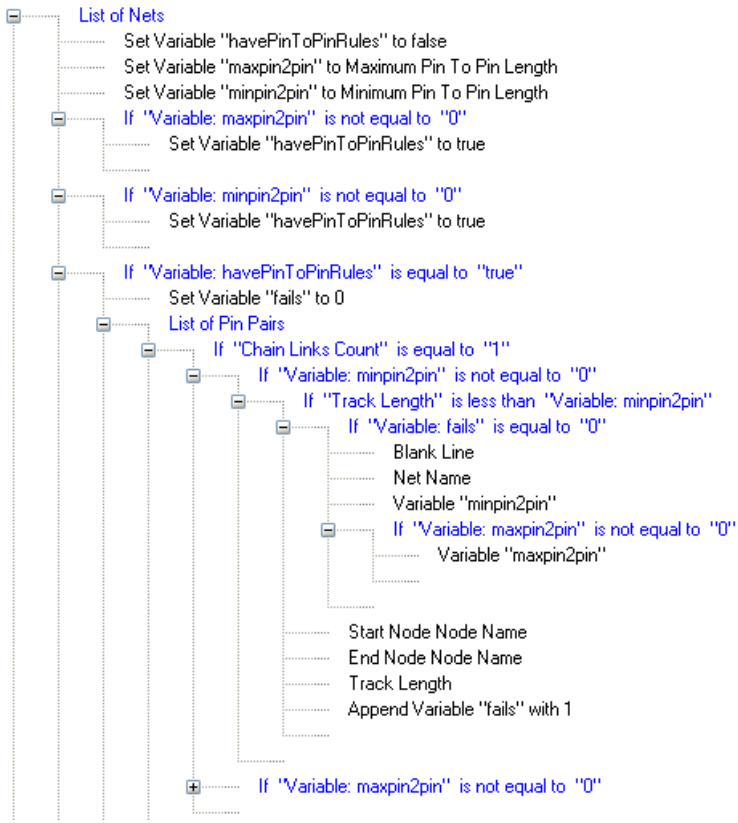
In the example report below, the Chain Link Count is tested to see if it is equal to 1, this means there is a direct link between the pins. If it does (in the design) then it sets two variables for the Max and Min Pin to Pin Length. With this set, the first variable (Min Pin to Pin Length) is checked to be greater than 0. The Track Length for that net is then checked to be less than the variable just set and if it is (equal to 0), then it reports the Net Name and start Node Node Name etc. It then spins through the Max Pin to Pin length to run the same check.



When the **Maximum Pin To Pin Length** and **Minimum Pin To Pin Length** commands are used on pin pairs they take into account the net class subgroups defined within the net class by pin attribute. By comparing these values against the pin to pin track lengths you can report length fails.

The Pin to Pin Connection rule defines the **Maximum Track Length** and **Maximum Number of Vias** in the shortest path between each component pin and the nearest other pin in the net.

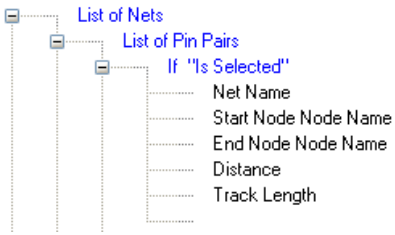
An alternative method to check the Max and Min Pin To Pin Lengths is by running the script below. This sets up three variables for Max and Min Pin to Pin length, and a rule check for whether the rule is set (havePinToPinRules). It then checks to see if the rule is not equal to 0, if it isn't then it sets the havePinToPinRules to True. If this is true, the List of Pin Pairs command is run. Each rule check (Max Pin to Pin and Min Pin to Pin) is run after checking the variable maxpin2pin and minpin2pin.



When run, the report looks like this:

```
net: HSE1
  A1.2 U13.1 len: 5.652 min: 8.000
net: HSE2
  A1.1 U13.2 len: 5.828 min: 8.000
```

You can also use the **Is Selected** on a pin pair to only report the track to track lengths on selected pins in the design.



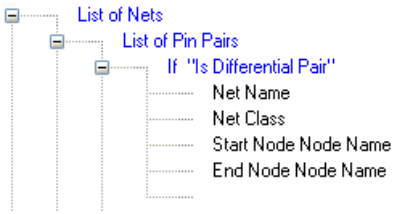
If Differential Pair

There is now a **Has Differential Pairs** command that can be used in a design (at the top level) or within a **List of Nets** command.

This example might be reported like this with a **true** or **false** status:

```
$11 false
$12 false
+5V false
GND false
HSE1 true
HSE2 true
NPRESET false
OUTPUT false
```

Has Differential Pairs can also be used within **If** to test if Nets have Differential Pairs. It can also be can be used within a **List of Pin Pairs** command.

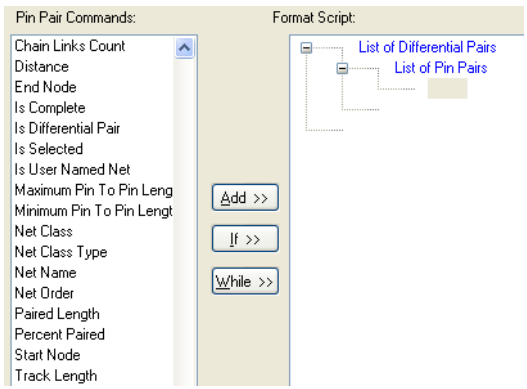


This example might be reported like this:

```
Net=HSE1 Net_Class=HSE U13.1 U13.9
Net=HSE2 Net_Class=HSE U13.2 U13.8
```

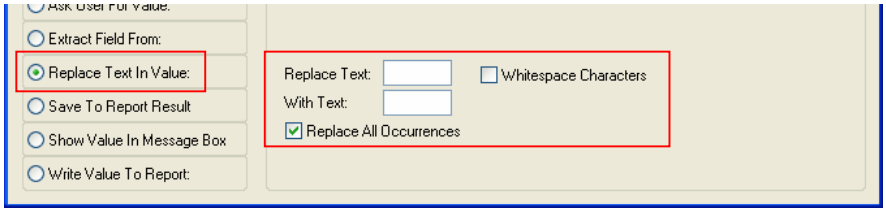
Report Maker List of Differential Pairs

You can now output using the **List of Differential Pairs** command for PCB and Schematic designs. Within this, you can also do a **List Of Pin Pairs** to report each of the two pin pairs in each differential pair, and can report the paired length rules and actual paired lengths using additional commands.



Ability to replace substrings of text items

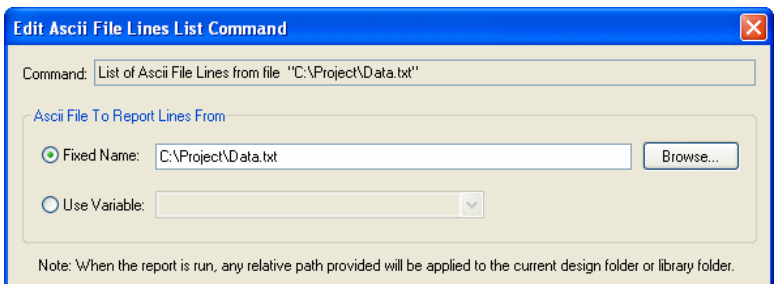
The ability to replace substrings of text items has been added, e.g. to replace spaces in part names (which old pick-and-place machines don't like) with another character before they are reported.



For a variable command, there is a new **Replace Text In Value** action. Type in the text sequence to be replaced, or check the **Whitespace Characters** box to easily replace all spaces, tabs and newline characters. You can then provide the text sequence to replace them with, or leave this blank to simply remove them. If the **Replace All Occurrences** box is left unchecked, only the first occurrence of the text will be replaced.

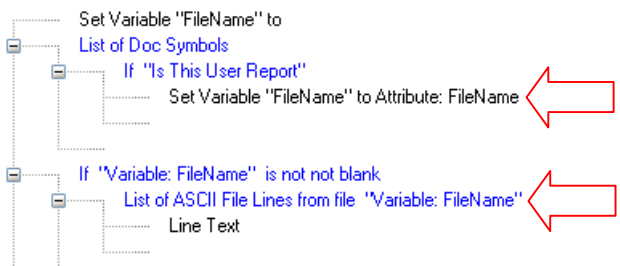
Ability to insert information into reports from external file data

The ability to insert information into reports from external file data has been added. To do this a new **List of ASCII File Lines** command has been created. This command can have the file name provided by a fixed text string, or by the contents of a text variable at run time. This file name may contain a relative path, in which case it will be applied relative to the current design folder or library folder when run.



Each ASCII file line can then be reported or saved to a variable using the **Line Text** command.

This command could be used in an inserted report to display the current contents of an ASCII file in the design. The file path could be provided by an attribute of the inserted report symbol.



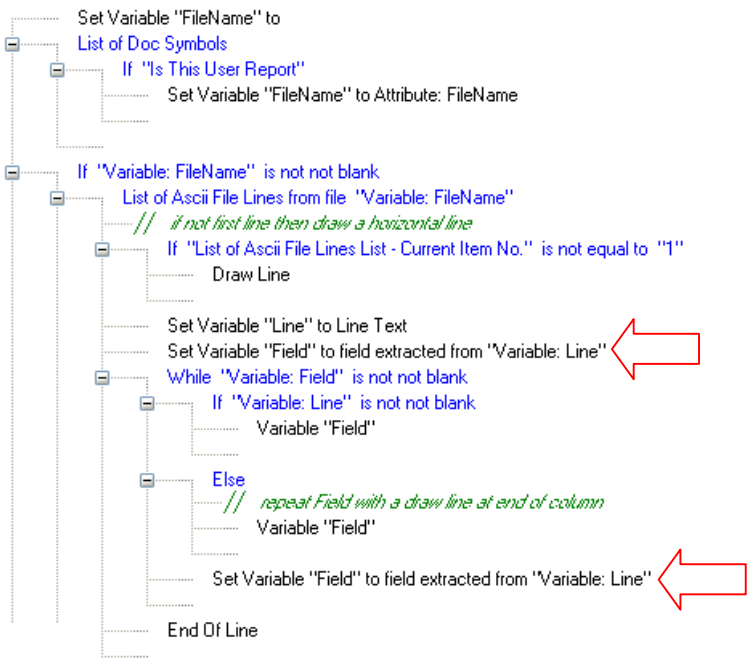
This feature can also be used along with the new feature **Extract a String into sub-strings** (documented below). This is used to insert a table from an excel spreadsheet into your design. See the example below.

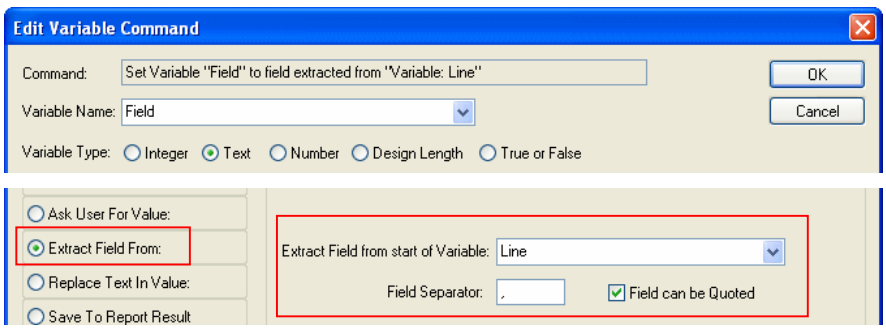
Ability to extract a string into sub-strings

The ability to extract a string into sub-strings and loop with a variable set to each sub string has been added. For example the first part of the report may build up a comma separated list of fields in a variable based on some criteria. Then the second part of the report could extract each field in turn and report it.

To do this, add a **variable** command and set its action to **Extract Field From**. Choose the variable the field is to be extracted from and the separator character (or string) that defines the end of the field. When this command is run, the variable will be set to the field text, and the variable it is extracting from will have the field (and its separator) removed from it.

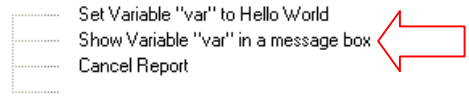
The example below uses this feature to insert a CSV file into your design as a table.

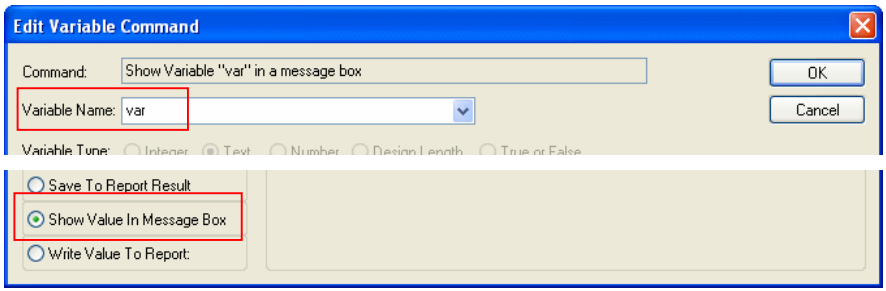




Ability to show a simple message box

The ability to show a simple message box has been added. For a **variable**, there is a new action **Show Value in Message Box**. When the report is run this command will display the current contents of the variable in a message box to the user.

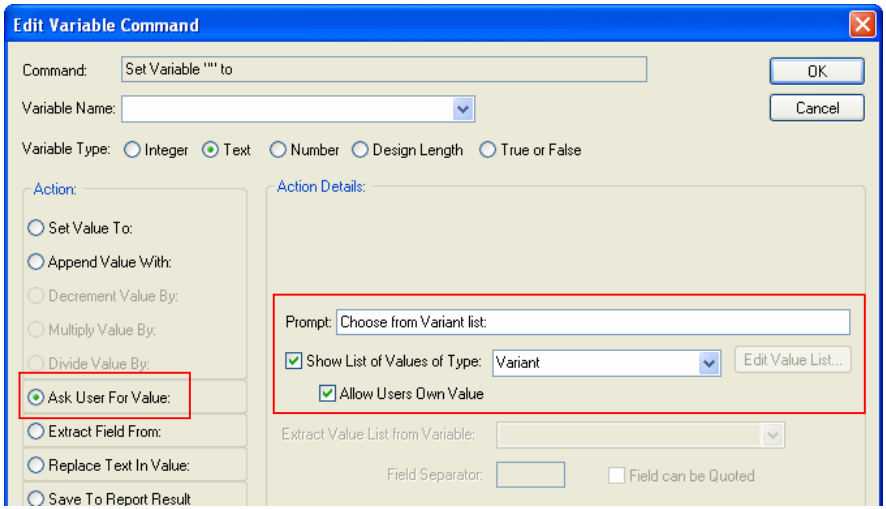




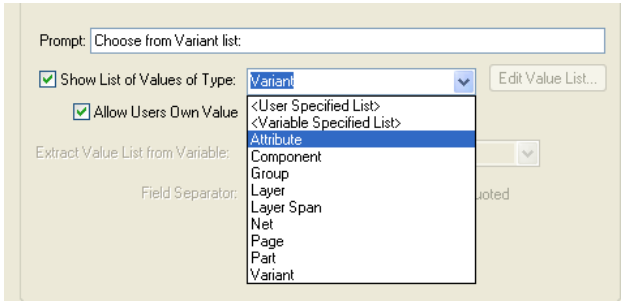
The report output continues when you press the **OK** button. In our example, this would display like this:

User Input Prompted with Drop Down List

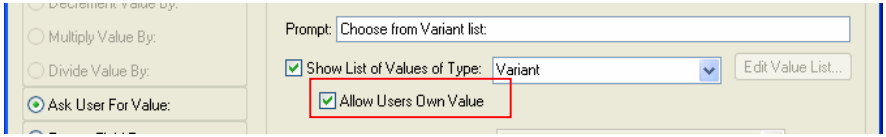
You can now offer a drop down list of possible values when prompting for user input for the report. For example, a list of variant names in the design. Offering a prompt is achieved using a **variable** command with action set to **Ask User For Value** like before, but now you can check a box to **Show List of Values of Type** and choose a type of design item to present in a list when the prompt command is run. Names will be extracted at runtime from the design (if there is a design and it is of the correct type).



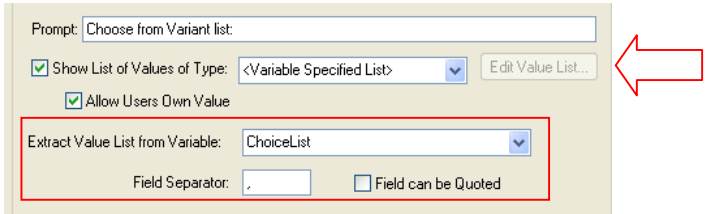
There are two special value types.



Choose **<User Specified List>** and use the **Edit Value List** button to add your own list of acceptable values. Or choose **<Variable Specified List>** to extract a list of acceptable values at runtime from another variable (using a specified **Field Separator** between acceptable values). This is useful to present a list of items that satisfy a certain criteria, e.g. all components on the top side



When **Variable Specified List** is used, you can select the variable name from which to extract the data:



You can also use a check box to specify whether to **Allow Users Own Value** or to only let them choose a value from the drop down list.

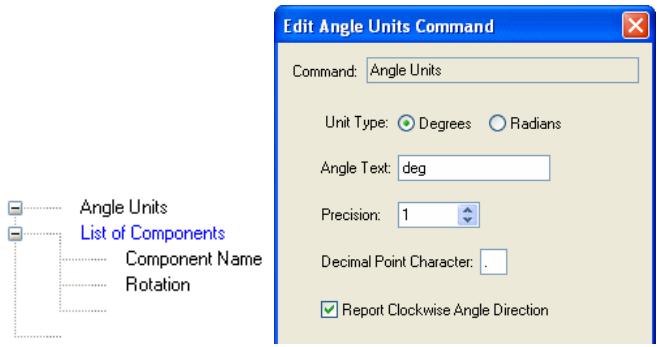
Ability to make a report not-insertable into a design

You can now make a report not-insertable into a design. To do this insert the new command **Not Insertable** at the top of the report script. The interactive **Insert User Report** option will not list it.

Note: If an already inserted report is changed to be not insertable - it will still run..

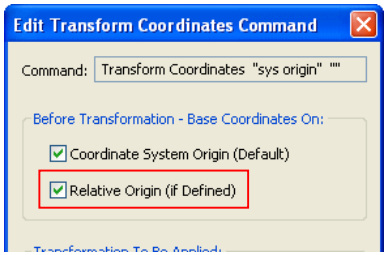
Ability to output angles as clockwise rotation

The ability to output angles as clockwise rotation has been added. A switch has been added to the **Angle Units** command **Report Clockwise Angle Direction**.



Ability to output coordinates relative to the relative origin

The ability to output coordinates relative to the Relative Origin has been added. The **Transform Coordinates** command now has new parameters **Coordinate System Origin** and **Relative Origin**.



If **Relative Origin** is checked and the relative origin is placed in the design the coordinate will be output relative to it. Else, if the **Coordinate System Origin** box is checked and the system origin is defined it will be output relative to the system origin, otherwise it will be output using its exact value.

Note: use of this command stops any CAM Plot transformation and if the relative origin is at (0, 0) and not switched on, it will not be used.

New commands for reporting the placement area for each footprint in a library

Enabled commands for reporting the placement area for each Footprint in a library. This allows you to run a report on a library which gives you the Footprint Name and the total sq mm (or sq inch) size of any body Area and placement Area. To do this **List of Symbols** now has the following field commands enabled:

Is All SM Pads

Is Embedded Component

Is SM Device

Position (symbol origin)

Extents Area

Top Placement Extents Area

Bottom Placement Extents Area

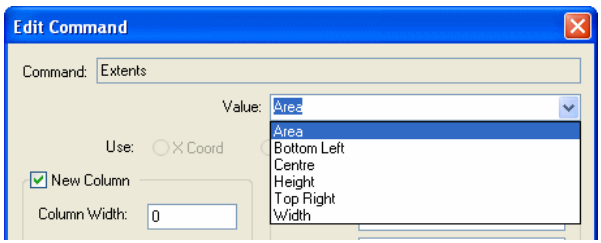
Top Placement And Pads Extents Area

Bottom Placement And Pads Extents Area

Is All SM Pads and **Is Embedded Component** report **True** or **False** as the test status.

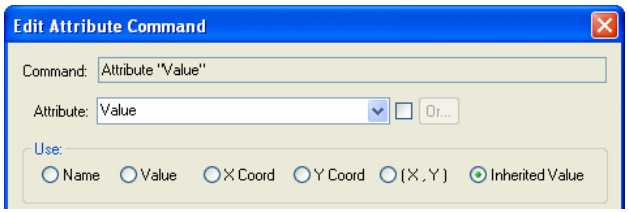
The **Position (symbol origin)** reports the position of the Symbol origin in the Footprint relative to the coordinate system origin.

Each of the **Extents** commands allow you to choose the **Value** from a drop down list:



Extra field on an attribute to get the "original" value from the part or symbol

There is now an extra field on an attribute to get the "original" value from the Part or Symbol. This allows you to produce a report of the attributes that have been changed in the design. When you edit an **Attribute** command, or an **If** command on an attribute field or a variable command using an attribute field, there is now an extra **Use:** option of **Inherited Value**. It reports an empty field if there is no inherited value.



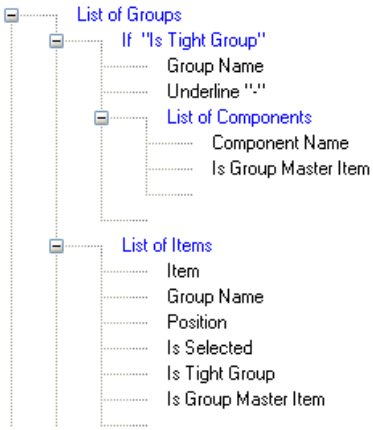
*Note: A test of **Exists** on the inherited value will be true if there is an inherited attribute, even if it has no text.*

When run, it could report like this:

```
A1 10uF Inherited Value: 1uF
A2 100uF Inherited Value: 1uF
A3 1uF Inherited Value: 10uF
```

Ability to report groups

The ability has been added to report **Groups**.



Use **List Of Items** within **List of Groups** to report each item in the group. For each item you can use **Item** to report a description of it and can test whether it **Is Group Master Item**. You also can use the following commands:

```
Group Name
Is Tight Group
List Of Components
        Comp Name, etc
        Is Group Master Item
List Of Gates
        Gate Name, etc
        Is Group Master Item
```

Is Group Master Item - Use this command within **List of Items** to report whether the item is the groups master item. This can also be used within **List Of Components** and **List of Gates** commands when they themselves are within a **List Of Groups** command.

Use **List Of Items** within **List of Groups** to report each item in the group. For each item you can use **Item** to report a description of it and can test whether it **Is Group Master Item**.

```
List Of Groups
     List Of Items
            Item – [reports textual description of the item]
            Position
            Is Group Master Item
            Is Selected
```

When run, the report could report like this below, you could also test and change the conditions reported:

```
Component: A2 Group1 (524.000,535.000) true false
Component: A3 Group1 (524.000,530.000) false false
Copper: Layer:Top Group1 (518.500,533.500) false false
```

Used within **List Of Components**, **List Of Groups** can be used to show components which belong to multiple groups.

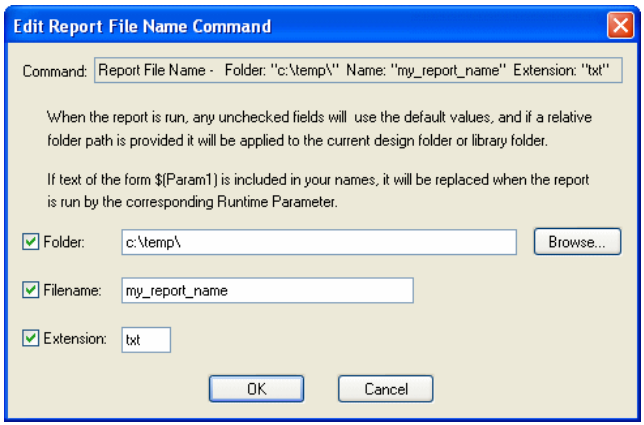
```
List Of Components
       Comp Name, etc
       List Of Groups
              Group Name
```

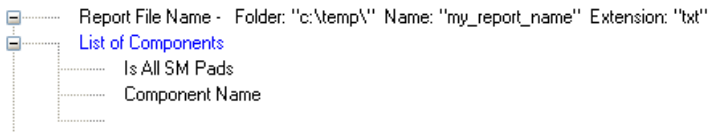
You can also use **List Of Groups** within **List Of Gates**.

```
List Of Gates
       Gate Name, etc
       List Of Groups
              Group Name
```

Added ability to set the report file name and folder within the report script

The ability to set the report file name and folder within the report script has been added. There is a new command **Report File Name** that replaces the old **Report File Extension** command. This now has optional Folder, Filename and Extension parameters. If a relative folder path is provided, it will be applied to the current design folder or library folder.

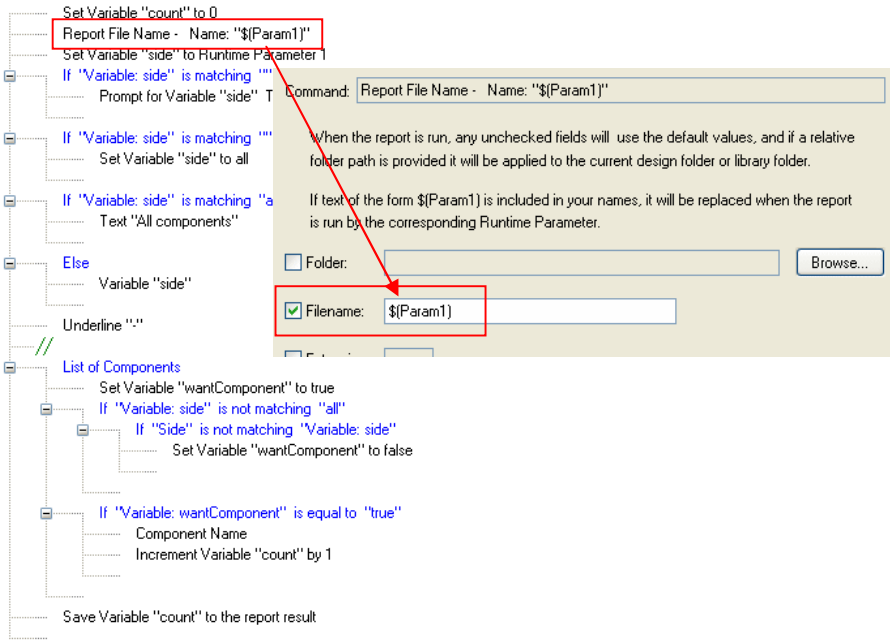




Text Substitution In Report Filenames

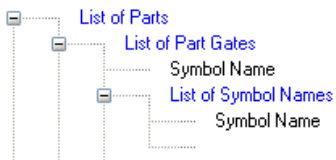
The **Report File Name** command also allows runtime substitution of text of the form $(Param1) with its corresponding runtime parameter.

If you are using nested reports, you can build up your filename using information passed through to this report from its parent report. If you include text of the form $(Param1) anywhere in the report name fields provided in this dialog, it will be replaced at run time by the corresponding runtime parameter passed from the parent report. If the runtime parameter was not set up, the text will simply be removed from the name. Note, there are only eight parameters so $(Param8) would be the highest that can be used.



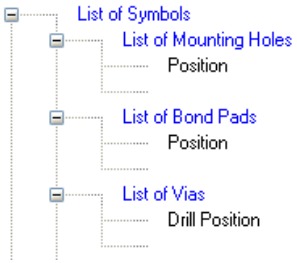
Added List of Symbol Names inside List of Part Gates

A **List of Symbol Names** command has been added for use inside the **List of Part Gates** command. This allows access to alternative symbol names when running a **Library Report**.



List of Symbols contains additional List commands

For use within a Library, the **List of Symbols** command now has **List of Mounting Holes**, **List of Bond Pads** and **List of Vias** enabled for **Footprints** (in a library).

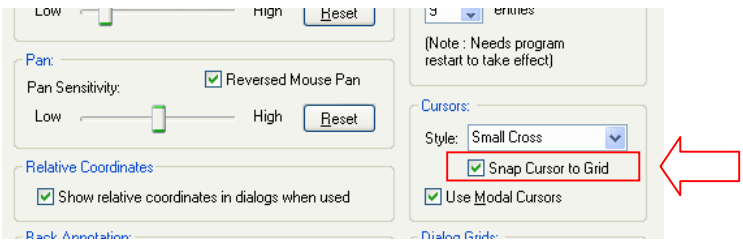


New command for Is Default for Symbol and Footprint Names

Within the **List of Symbol Names** and **List of Footprint Names** commands, you now have an **Is Default** command to report if the current symbol name or footprint name is the default that will be used by the Part Gate (in Schematics) or Part (in PCB). You can use this to report if a Part in the design is using the default symbol or not. The following example lists the alternative Footprints for a Part:

Snap Cursor to Grid

There is a new check box on the **Options** dialog and **Preferences** page named **Snap Cursor to Grid**. Select this option to make the cursor move only on the current grid when in the design area.



Gridded cursors will still be shown when modal cursors are displayed.

This option can only be used with the three cross cursors, it will not work with the standard windows cursor.

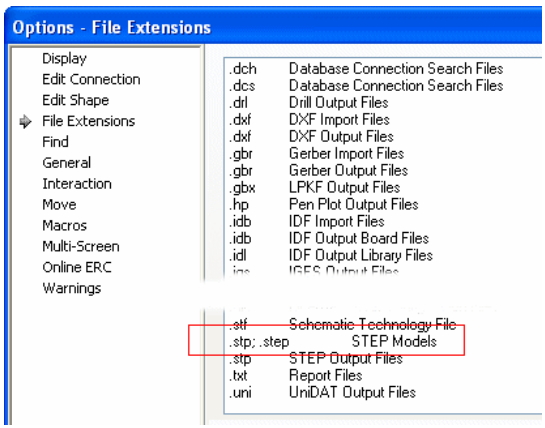
Toggle Cursor Gridding Command

There is also a new command available in the **Customise** option and **Keyboard** named **Toggle Cursor Gridding**.

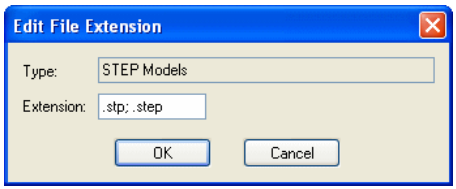
Multiple File Extensions

The ability to have multiple extensions associated with a file type has been added to Version 7.5. For example for STEP Models, you could specify .stp and .step.

From within the **Options** dialog and **File Extensions**, use the **Edit** button to add multiple file extensions as required.

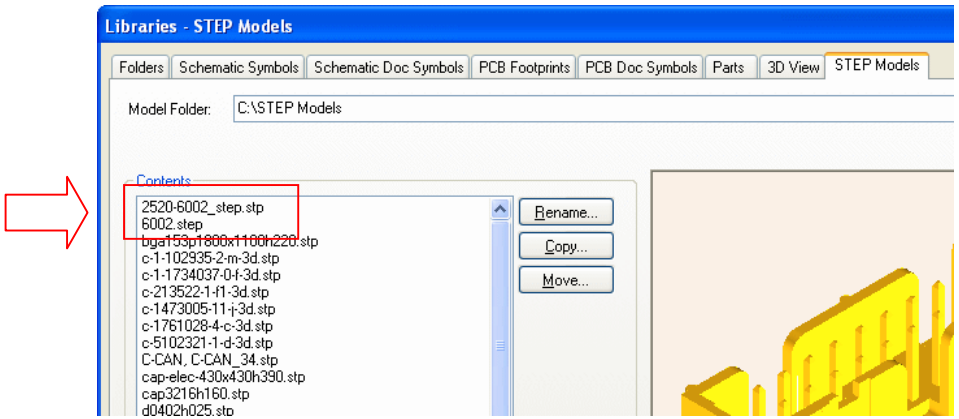


The **Edit** dialog allows you add the extensions using the semi-colon ; character as the extension separator.



Multiple file extensions are used for both browsing of data and writing data out. These multiples would be shown under the **File Of Types** as legal extensions to use for writing data.

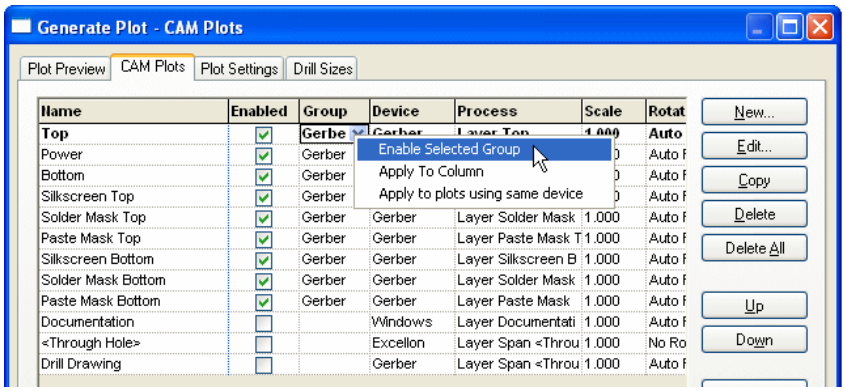
An example of multiple file extensions would be where browsing for STEP models. Models with the .stp and .step extension would be selected in the Contents: list.



CAM Plot Changes

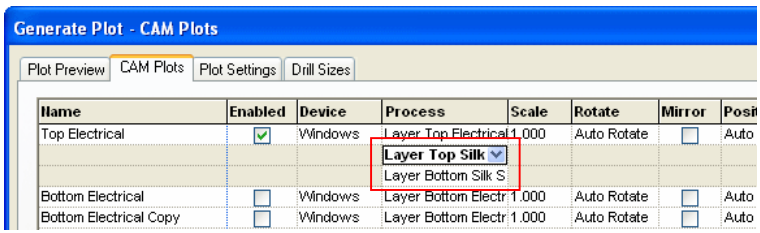
Enable a Named Group of Plots from Context menu

From within the **CAM Plots** dialog, you can now enable just a named group of plots with a single context menu command **Enable Selected Group**.



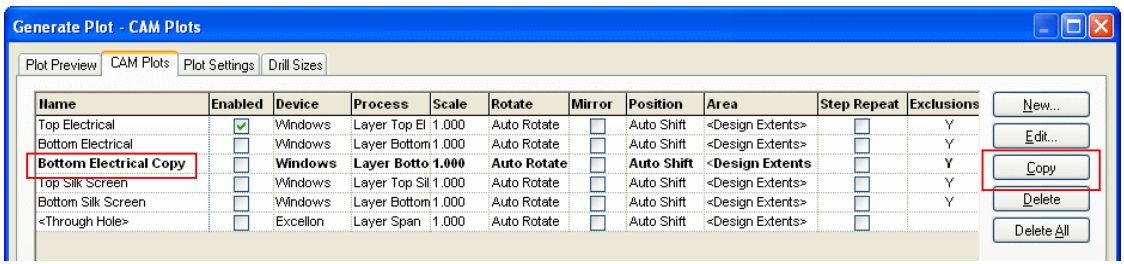
Move Additional Plot Items In CAM Plot Option

You can now move additional plot items within a plot on the CAM Plots grid. The currently selected row is now drawn in bold and can be moved **up** or **down** the grid using the buttons on the right side of the dialog.



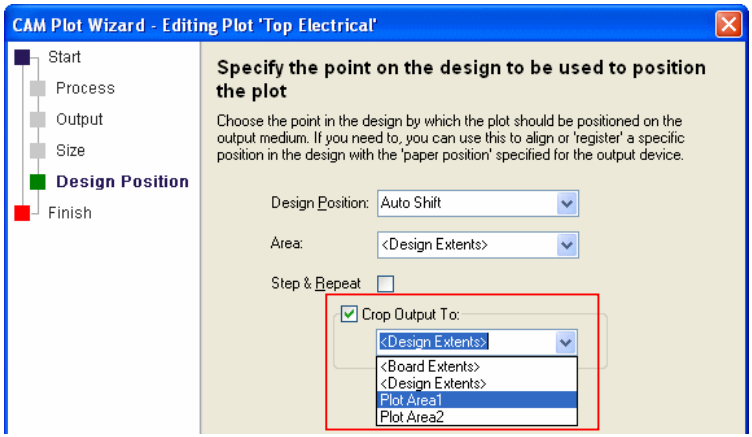
Copy Plot

From the **CAM Plots** dialog, you can now copy a plot using the **Copy** plot button. This copies the currently selected plot and all additional plots. The copied plot will be displayed with the original name plus the word **Copy** next to it, as shown below with **Bottom Electrical Copy**.



Crop Plot Output

In the **CAM Plot Wizard**, you can now define a **Crop area** which will suppress items totally outside the given area.

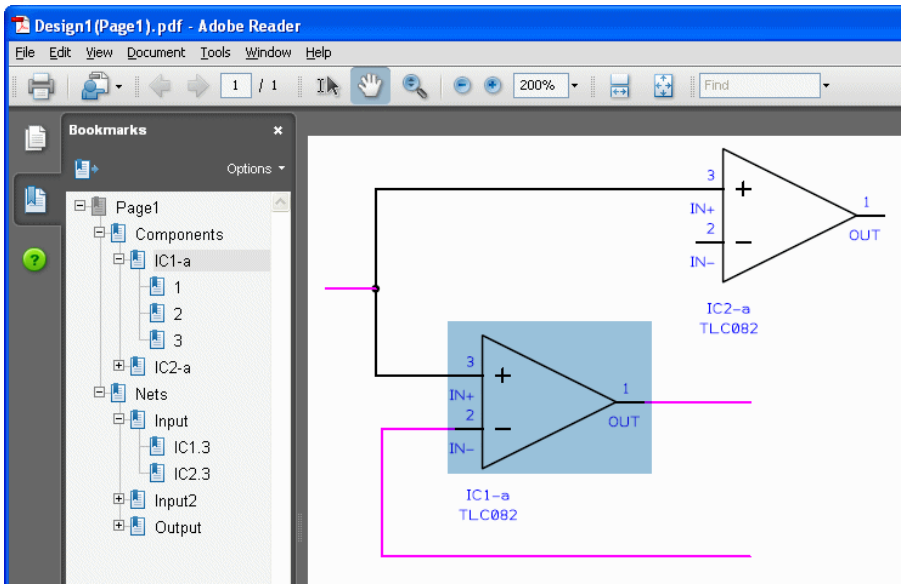


Use **Crop Output To:** a defined area to prevent items outside this area from being plotted. The cropping is only done crudely, so items that cross the area will be plotted. In general, this would be used for a set of items which form a coherent subset of the design which you would want to position within the overall plot.

You can choose the **Board** and **Design** extents from the list. If there are **named areas** in the design, these are also shown on the drop down list for selection.

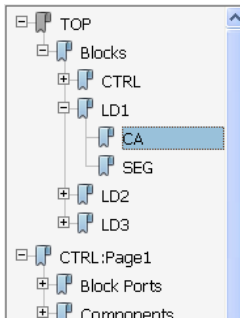
PDF Output Bookmarks

When you output to PDF using the built-in PDF writer supplied with Pulsonix, you can now output a list of Components and Nets as bookmarks.



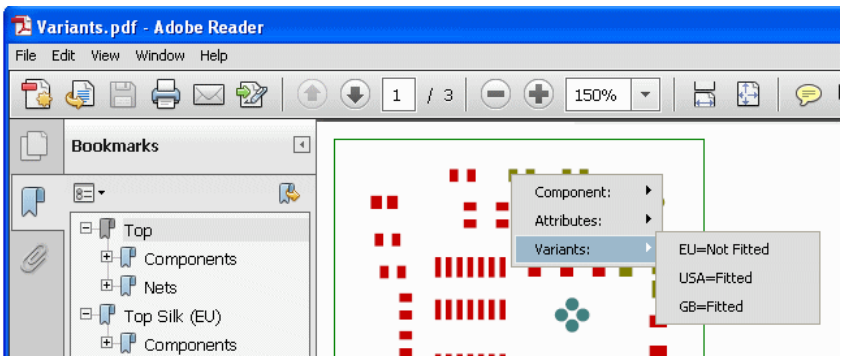
The Components and Nets will be displayed within a list on a per-page basis that can be opened using the small + box. These lists are hierarchical, components show the Component Name, the Gate and the pin numbers on the Gate. Nets will show the net name and the component name with all pins on that net.

The bookmark now also lists Blocks and Block Ports from the Schematic editor.



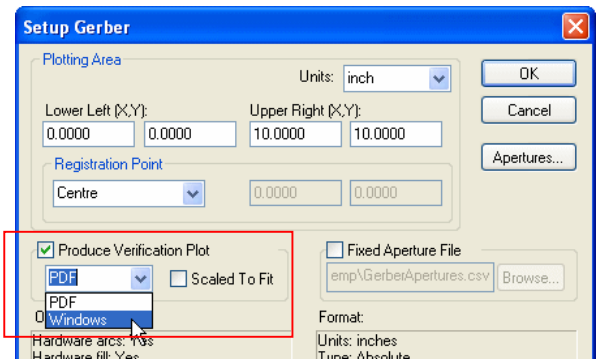
PDF Output Variants

When outputting a PDF plot using the build-in PDF driver within the CAM Plots dialog, you can now list variants and fitted status of each component by selecting it and using options on the context menu.



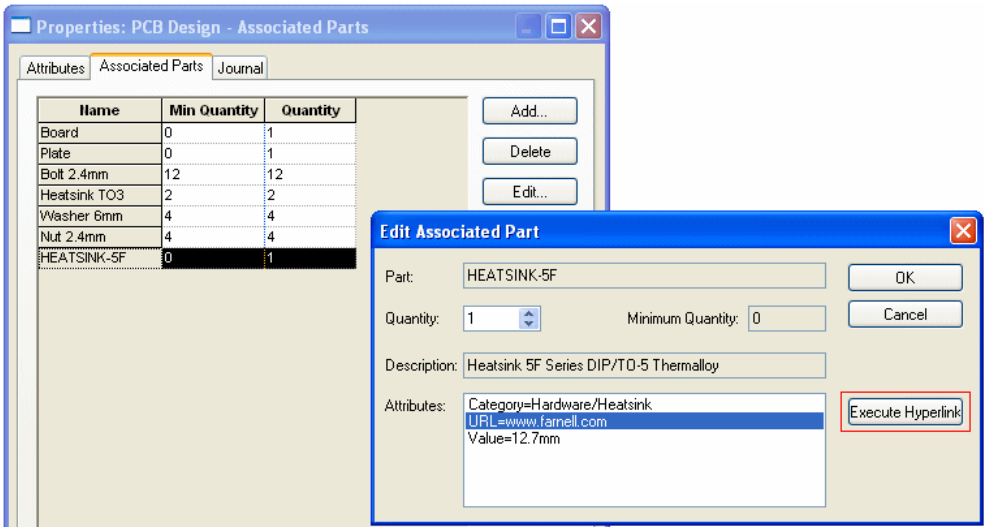
Gerber Verification Plot to PDF

The verification plot can now be output to PDF as well as the existing Windows output. Choose PDF or Windows from the drop down list box presented in the Gerber Setup dialog within CAM Plots.



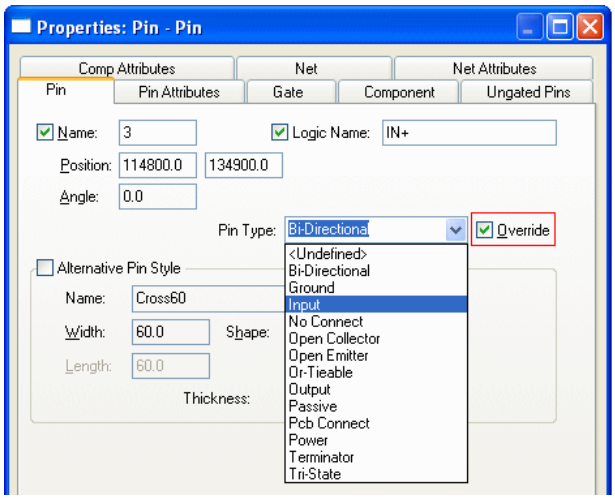
Associated Parts – Execute Hyperlink

In the **Edit Associated Part** dialog from the **Design Properties** dialog, an **Execute Hyperlink** button is available for hyperlink attributes on the associated part. The Associated Part must have a Hyperlink attribute already added in the **Part** definition using the **Part Editor**.

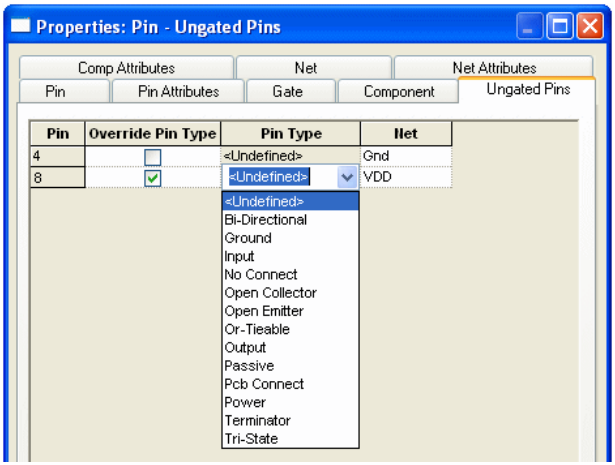


Local Pin Types

The **Properties** dialog can now be used to override a **Pin Type** on a component pin in a Schematic design. On the **Pin** page, there is a new **Override** check box. When checked, it allows you choose a pin type for the pin from a drop down list. This pin type will then be used when running the **Electrical Rules Check** (ERC) option.

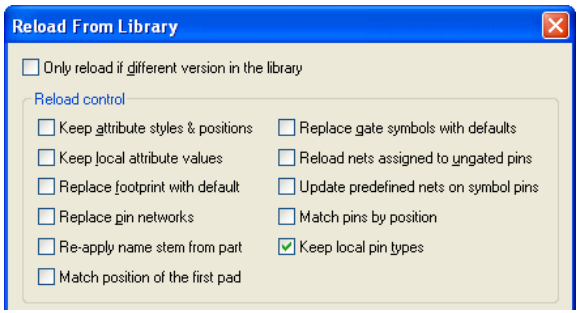


You can also use **Properties** to override the **Pin Type** on **Ungated** pins. Check the **Override Pin Type** check box to then gain access to the **Pin Type** drop down list.



Reloading Local Pin Types

When reloading components from the library there is now a **Keep local pin types** option to prevent the local pin types being changed.

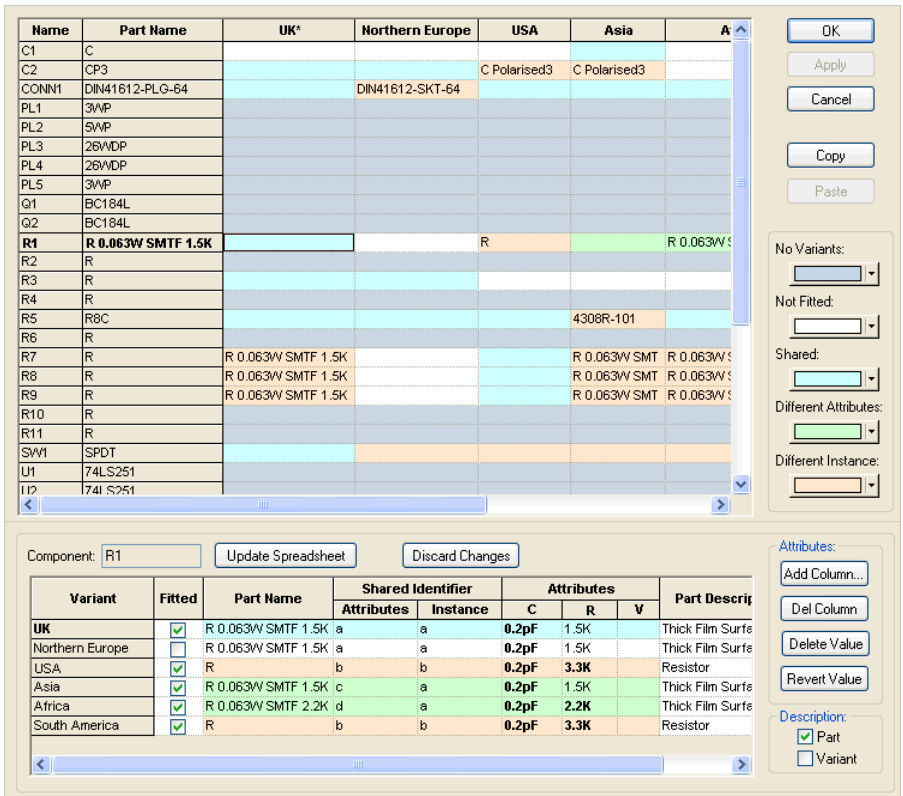


Local Pin Types In Report Maker

In the **Report Maker** option on the **Output** menu, there is now a **Has local pin type** command to be used on a schematic component pin.

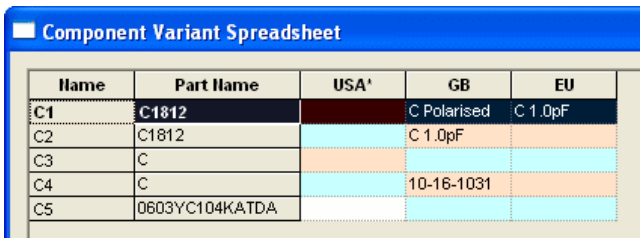
Component Variant Spreadsheet

You can now view and edit variants in a spreadsheet style interface. The **Component Variant** editor is available on the **Edit** menu in both the Schematic and PCB design editors.



The dialog is broken down into two distinct sections with controls for each.

The top section is the component variant spreadsheet, with a row for each component. The bottom section allows you to view and edit the variant details for the component currently selected in the top half.



The main grid in the top left of the dialog shows a row for each component and a column for each variant in the design.

Cells are colour coded to show the state of each component in a particular variant. The colour indicates whether the component is not fitted, is a different part, uses different attribute values or uses

a different physical appearance. This makes the spreadsheet ideal for quickly seeing which components in the design use variants and which are not fitted in certain variants.

The first column shows the component name and the second column shows the main part used for the component. The variant columns only show a part name for a component if it is different to the main part. The part name cannot be edited in this grid, but can be changed by editing its value in the component variant grid at the bottom section of the dialog and then updating the spreadsheet with the changed component. The height of the row may be adjusted if you wish to accommodate longer variant names but keep the width of the columns to a minimum so many variants can be viewed on a single screen.

The **Name**, **Part Name** and **Variant** columns can be sorted by clicking on their column headers. A variant column is sorted by variant **State** or **colour**. Clicking on any cell apart from the column headers will make the component the row represents the current component and will change the lower grid to show its variant details. The current component is indicated in the spreadsheet using a bold text style in the first two columns. When clicking on a different row to change the current component, and the lower grid has variant changes pending, you will be asked if you want to update the spreadsheet with these changes prior to switching the current component.

You can select multiple components in the spreadsheet by dragging the left mouse button vertically in the **Name** column. The only reason for doing this is to use **Paste** to copy the same variant information from another component to multiple components.

Down the right-hand side of the top of the dialog is a set of buttons that allow you to copy variant information between components and define the colours used for the different variant states.

OK - This will Apply any component variant changes to design and exit the dialog.

Apply - If this button is enabled it means there are component variant changes in the spreadsheet that have not yet been committed to the design. Pressing this button will apply these changes to the design, but stay in the dialog for more editing.

Cancel - This will exit the dialog without changing the design at all.

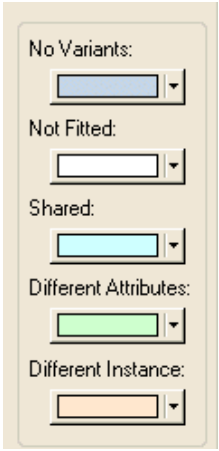
Close - The **Cancel** button becomes **Close** after using **Apply** for the first time. Pressing this button will exit the dialog and throw away any changes you have been made since the last **Apply** was performed.

Copy - This will remember all the information shown in the Component Variant grid for the current component. This information can be subsequently copied to another component by selecting the target components row in the spreadsheet and pressing the **Paste** button. The shortcut keys <**Ctrl-C**> can also be used.

Paste - This will copy the remembered variant information from the previous **Copy** action to the current component, or multiple selected components, in the spreadsheet. Part names, fitted state and sharing information will all be copied, so this feature should only be used on components that are to be the same part and want to appear the same for all variants in the design. The shortcut keys <**Ctrl-V**> can also be used.

Cell Colour Coding

The next set of buttons show the colours that will be used for each variant 'state' in the dialog. Click on a colour block to change its colour. The colours are retained in the system registry, so that your choice of colours is preserved for future use.



No Variants - This colour is used for all cells of a component that does not vary for any variant.

Not Fitted - This colour is used for cells where a component is not fitted in the variant the cell represents.

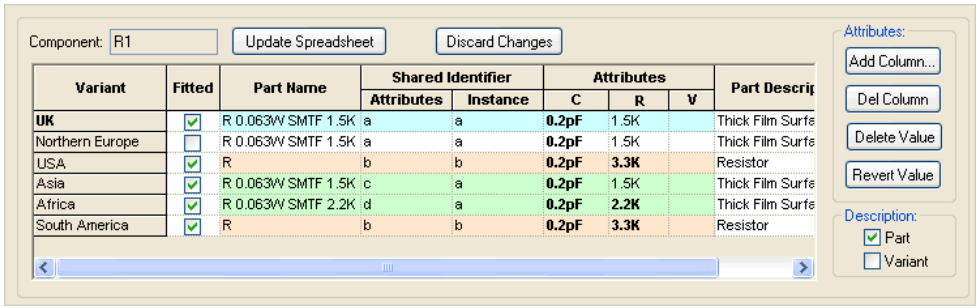
Shared - This colour is used for cells where a component is fitted in the variant the cell represents and it shares the same part name, attributes and physical component instance as the 'normal' component.

Different Attributes - This colour is used for cells where a component is fitted in the variant the cell represents, but it has a different set of attributes to the 'normal' component.

Different Instance - This colour is used for cells where a component is fitted in the variant the cell represents, but uses a different physical component instance to the 'normal' component.

Component Edit Pane

The bottom part of the dialog is occupied by a grid or table showing the variant details for currently selected component in the spreadsheet.



Variant	Fitted	Part Name	Shared Identifier		Attributes			Part Descrip
			Attributes	Instance	C	R	V	
UK	☑	R 0.063W SMTF 1.5K	a	a	**0.2pF**	1.5K		Thick Film Surfa
Northern Europe	☐	R 0.063W SMTF 1.5K	a	a	**0.2pF**	1.5K		Thick Film Surfa
USA	☑	R	b	b	**0.2pF**	**3.3K**		Resistor
Asia	☑	R 0.063W SMTF 1.5K	c	a	**0.2pF**	1.5K		Thick Film Surfa
Africa	☑	R 0.063W SMTF 2.2K	d	a	**0.2pF**	**2.2K**		Thick Film Surfa
South America	☑	R	b	b	**0.2pF**	**3.3K**		Resistor

This is similar to the **Component Variant Properties** dialog for a selected component, but with the added feature of being able to include attribute value information. It contains a row for each variant in the design and columns allowing you to change the components fitted state, part name, whether it has different attributes and whether it has a separate physical component instance. Cells in these columns are coloured depending on their 'state' using the same system as the spreadsheet. The name of the variant currently active in the design is shown in **Bold** text in the appropriate row. The table can be enlarged to show more variants, see **Resizing the Dialog** section at the bottom of this page for details.

Update Spreadsheet - This button becomes enabled when you have changed one of the cells in this grid. Press the button to update the main spreadsheet with the changes for this component. The changes will not be committed to the design until the **Apply** or **OK** buttons are pressed to apply the changes to all the components in the spreadsheet.

Discard Changes – This button also becomes available when you have changed one of the cells in this grid. Press the button to discard the changes to the component and restore it to the values last saved in the spreadsheet.

Column Headers

Variant – This column shows the variant name. The name highlighted in **Bold** is the current variant.

Fitted - This column shows if the current component is fitted in that variant (components are fitted in all variants by default). Use the checkboxes to set the fitted status.

Part Name - This column shows the part used for the current component in that variant. You can use this column to change the part used for this component, but if you want a different part for a particular variant you must first use the **Shared Identifier** columns to give the component a different shared **Attributes** identifier. Changing the part for a particular variant will also change it for all variants that have the same shared attributes identifier. Double click in the cell to change the part used by a component. The **Change Part** dialog will be presented for you to choose the required part. If the chosen part for a particular variant uses a different symbol or footprint to the components main part, then you will also have to change the shared instance identifier to a different value before updating the spreadsheet.

If the Database Connection is enabled, the component's variants dialog will display an additional **Database** column; This displays the component's database key field information for each variant and is useful for seeing if database part variants are being used even if the Pulsonix Library part is the same across variants.

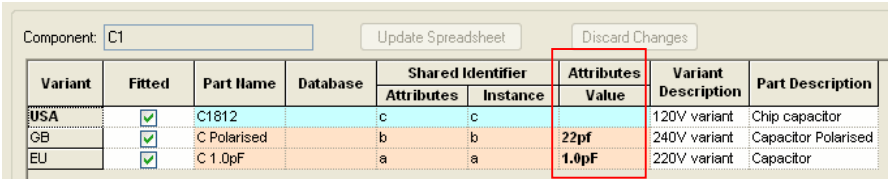
Shared Identifier - An advanced form of variants is to use a different component instance in some variants. You may want the component to have different attributes, but still be represented by the same physical symbol or footprint. Alternatively, you may want a completely different part with a different symbol or footprint, that can be positioned and connected independently. To do this, you need to assign a **Shared Identifier** (also referred to as the Variant Id), this can be any character or word.

There are two columns under the **Shared Identifier** heading. You can share **Attributes** between variants by entering the same identifier in the first column, and you can share physical **Instances** (symbols) by entering the same identifier in the second column. A blank Id is a valid identifier and initially, by default, all columns are blank indicating the same instance and attributes are shared across all variants. You cannot share attributes without also sharing instances across the same variants. Instances can be shared provided they have compatible parts and footprints in the variants in which they are to be shared.

For each Identifier, there must be at least one variant where the **Attributes** and **Instance** Identifiers are the same. The colour coding of the cells mentioned above helps indicate which variants have different attributes and which use different physical component instances.

Attributes:

You can add a column showing a particular attribute name and all the values for that attribute for each variant. This enables you, for example, to change the value of a generic resistor for each variant. If you want a different value for a particular variant you must first use the **Shared Identifier** column to give the component a different shared **Attributes** identifier. Changing the value for a particular variant will also change it for all variants that have the same shared attributes identifier. To change the value, left click in the cell and type the required text.



Variant	Fitted	Part Name	Database	Shared Identifier		Attributes	Variant Description	Part Description
				Attributes	Instance	Value		
USA	☑	C1812		c	c		120V variant	Chip capacitor
GB	☑	C Polarised		b	b	22pf	240V variant	Capacitor Polarised
EU	☑	C 1.0pF		a	a	1.0pF	220V variant	Capacitor

The buttons to the right-hand side of the grid provide you with some tools to edit the attributes and their values. Some buttons may only be enabled if appropriate cells are selected in the grid.

Add Column - This will add another column (Attribute Name) to the Attributes section of the grid, allowing you to change existing attributes more relevant to the current component, or to create a new attribute name to use for attributes be added to the component. An error message will appear if you enter an attribute name that already exists in the grid. The attribute names are retained in the system registry, so that your choice of attribute names for columns is preserved for future use.

Delete Column - Use this to remove an attribute column from the grid. This does not delete existing attributes for the component, removal of the column simply means you will not change the values of this attribute when the spreadsheet is updated with the component changes.

Delete Value - Depending on what is selected this will either be **Delete** or **Reset**. Delete will remove the attribute from the selected cell. Reset will reset the selected cell to its inherited value.

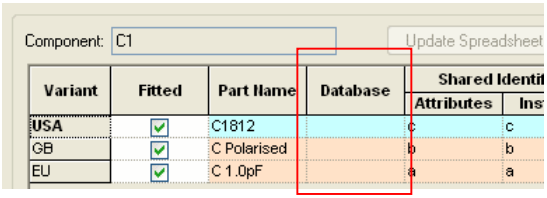
Revert - Use this button if you have accidentally modified the wrong cell but do not want to cancel the whole dialog. Pressing Revert will return just the selected cell to its original value from the design. This button will only be enabled if the selected cell has been modified.

Description:

There are two optional columns that can be used to show the description of the variant and part used in the row. Use the check boxes to the bottom right of the dialog to show or hide these columns.

Additional Database Connection Column

These additional features are available as part of the **Pulsonix Database Connection** (**PDC**) product.



If the **Pulsonix Database Connection** is enabled, the component's variants dialog will display an additional **Database** column; This displays the component's database key field information for each variant and is useful for seeing if database part variants are being used even if the Pulsonix Library part is the same across variants.

*Note that if the Part Name is changed for a variant, unless it is chosen using the **Database Bar**, any existing database information will be lost and it will revert to being a 'local' component for that variant. For a 'local' component, the Database column will be blank.*

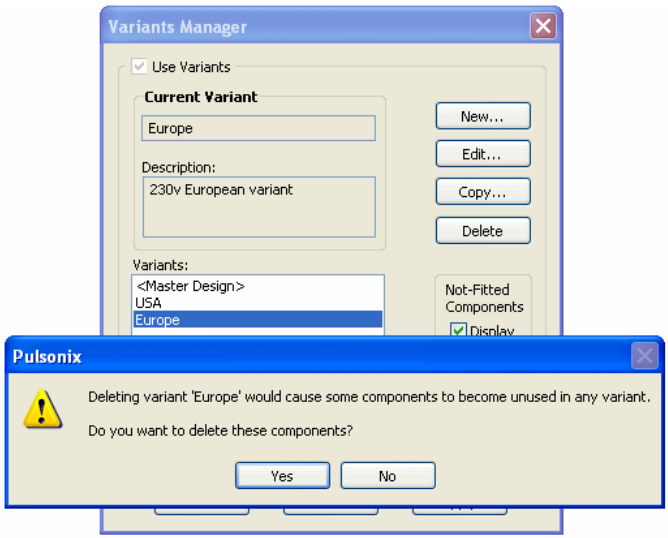
Resizing the Dialog

This dialog can be resized by dragging any of its edges or corners. The spreadsheet grid at the top of the dialog will be resized by the amount you have resized the dialog by, and the other controls will be stretched or moved to fit around it.

If you hold the **Shift** key down whilst resizing, the Component Variant grid in the bottom half of the dialog will be resized, and the spreadsheet grid in the top half of the dialog will only be horizontally altered. This is useful if you have a lot of variants to enlarge the lower grid to be able to see them all at the same time.

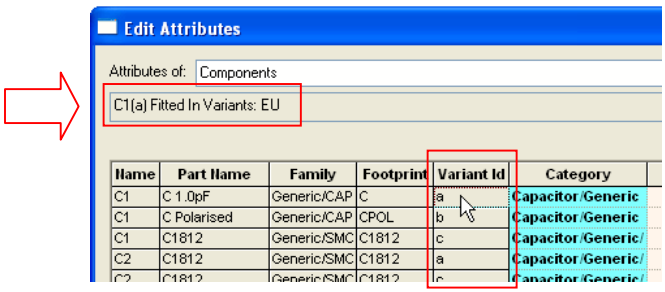
Delete Variant Changes

When using the **Variant Manager**, if deleting the variant would cause a Component to become **Unfitted** in any variant, you are now prompted to decide if these components should be deleted.



Attribute Editor shows Variant ID

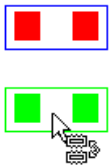
The **Attribute Editor** now shows the **Variant ID** for variant components in the design. Selecting a variant ID cell will display information about the component and variant it is fitted (or not fitted) in.



Swap Component Positions

There is a new interactive mode, **Swap Component Positions**, in the **Schematic** and **PCB** design editors on the **Utilities** menu. In Schematics it can be used on **Gates** and **Connector** pins.

When you enter this mode, components are selected as you pass over them.



If you click to select the two components to swap positions. In PCB there is an option **Swap Routed Components** on the context menu. If enabled the last segment of a track attached to the component will be unrouted prior to swapping the component positions. Component **Rotation** and **Side** is also swapped. If two components are pre-selected they will be swapped in a single shot action.

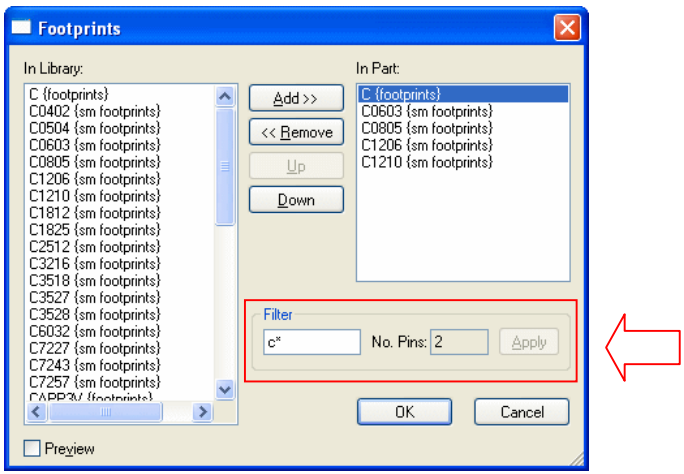
If the components are different sizes, their positions are swapped such that their centres are as close as possible but with their positions still on the component grid.

If the component you want is underneath the currently selected one you can select it by using the **Select Next** or **Select Previous** command on the context menu. When the correct component is selected click on the selected component that you wish to swap.

Part Edit Changes

Footprint Dialog Filter

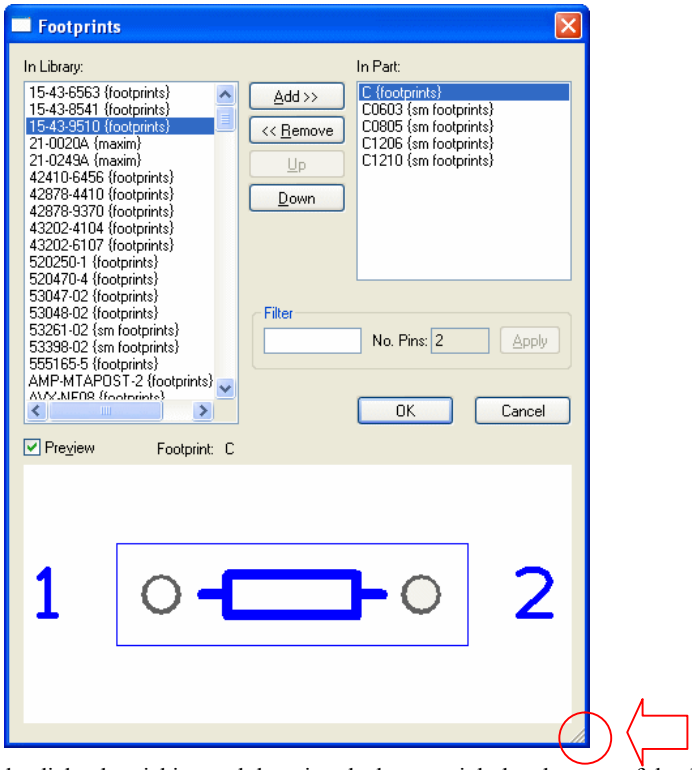
In the **Footprint** dialog in the **Part Editor** it is now possible to use a wildcard filter of the footprint name. The number of pins is also shown but is fixed at the number of pins defined for the footprint unless it is a new footprint for which the number of pins is as yet undefined.



Once the naming and wildcard have been used, use the **Apply** button to update the selection list.

Ability to Resize Footprint/Symbol Dialogs

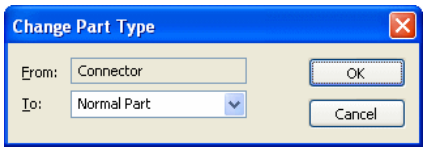
The **Footprint** and **Symbol** dialogs in the **Part Editor** are now resizable. These dialogs are available when choosing a Footprint or Symbol to add to a Part.



Resize the dialog by picking and dragging the bottom right hand corner of the dialog using the gripper.

Change Connector Part Type option

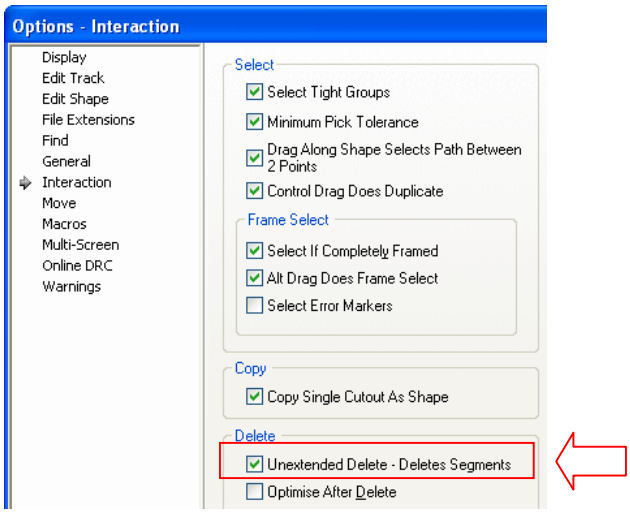
On the **Edit** menu in the **Part Editor,** it is now possible to change an existing **connector** part to be a **normal** part using the new **Change Part Type** option.



Once selected, the **Gates** page appears and normal part functionality within the Part Editor is restored.

Delete Segment on Unextended Delete

Unextended Delete now has an entry to **Delete Segments** on the **Interaction** page of the **Options** dialog.



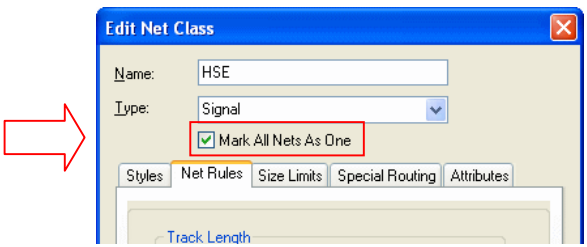
When used, this will remove just the selected segments and leave the rest. This can be used on **Tracks**, **Schematic Connections**, **Busses**, **Copper**, and **Documentation shapes**.

Deleting a segment on a **Closed** shape will make the shape **Open**. Deleting a segment in the middle of an Open shape will split it into two shapes.

When a Schematic Connection is split in two, the normal delete options determine the net naming of the two halves.

Net Class - Mark All Nets As One

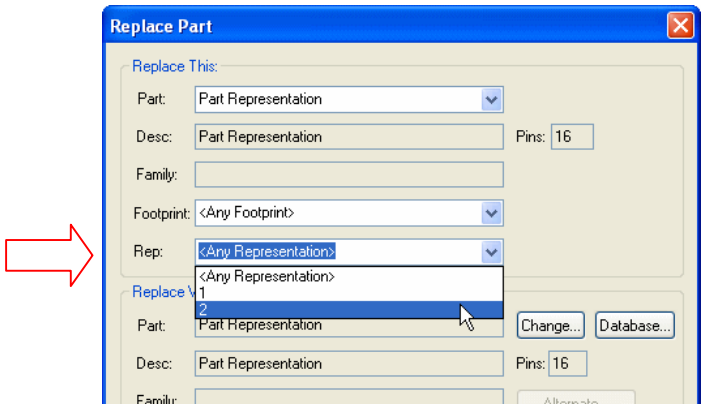
In **Technology**, you can set a flag on a **Net Class** so that when one net using that net class is marked in the design (using the <**H**> key), all nets using that net class are also marked. One use of this is to enable you to mark the full net path even when the path is split over a resistor for example.



This flag is also copied across from the Schematic when synchronising with a PCB design.

Replace Part Changes

You can now **Replace Part** by a specific representation, and also specify the resulting representation during replace part. The footprint and representation are now always used as part of the selection criteria for the components to be replaced (in previous versions, they were only relevant if the Part was not changing).

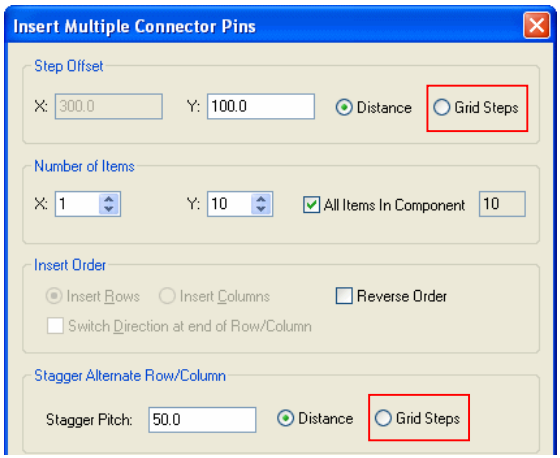


Insert Multiple Item Changes

Some improvements have been made to aid the insertion of multiple items, mostly to ease the adding of **Groups** of Schematic **Connector Pins**. Some of these changes affect **Arrange Multiple Items** and **Replicate** as well.

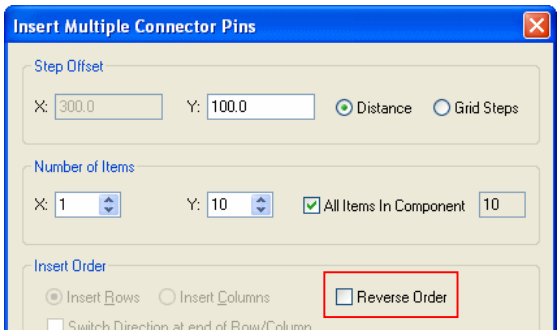
Define Offset and Stagger Values as Grid steps

You can now define the **Offset** and **Stagger** values as **Grid Steps** (using the **current grid** in the current interactive operation).



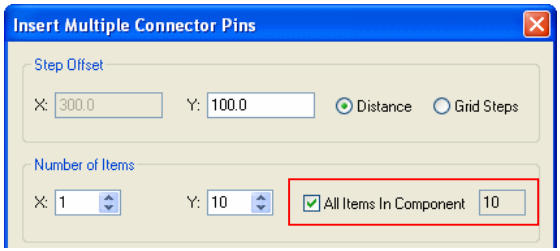
Reverse Order

You can check a box **Reverse Order** to make the offsets reverse. You can still use negative offset values but this makes it more obvious.



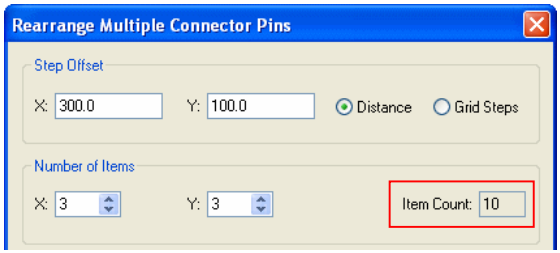
All Items In Component

You can select an **All Items In Component** check box to place all remaining Schematic Gates or Connector Pins in the current component being added. The number of items left in the component is shown in the edit box next to it. Checking this option will change the item counts to have a row or column with the correct number.



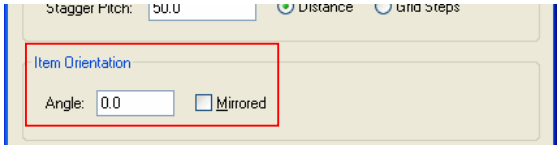
Rearrange Multiple Items – Total Count

Using **Rearrange Multiple Items** you can now see a total **Item Count** of the number of items in the selection.



Change angle and mirrored state

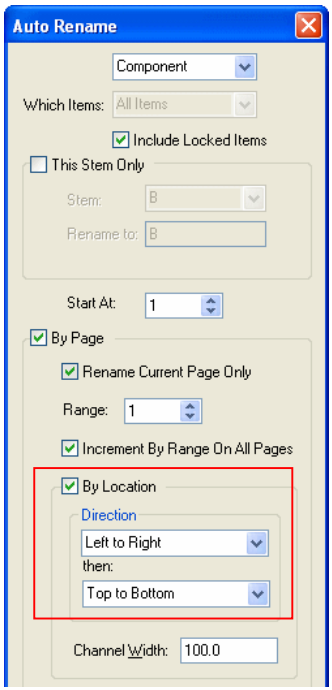
In the **Insert Multiple** and **Rearrange** dialog, you can now change the individual **Angle** and **Mirrored** state of each of the items being added. This allows you rotate the connector pins to be the correct direction.



Auto-Rename in Schematic Editor

The Auto-Rename feature now supports rename **By Location** for a Schematic design as well as a PCB design. This option is available when also using the schematics **By Page** option and will rename items on the same schematic page sequentially according to their position on that page.

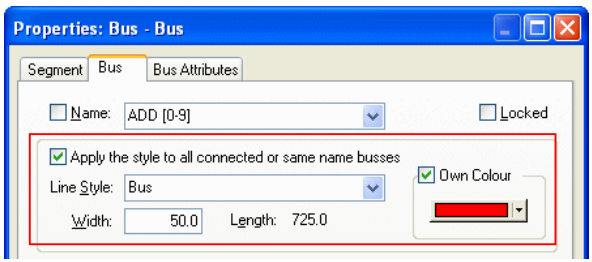
For multi-gate components the most relevant gate, as dictated by the current **By Location** settings, will be used to determine the item's position in the rename sequence.



By checking the **By Location** box, you can rename Components or Test Points by location, within each page. This enables you to produce a name sequence based on the position of these items on the schematic pages. This style of rename is commonly used for the service engineers convenience when attempting to diagnose problems using a schematic diagram as reference.

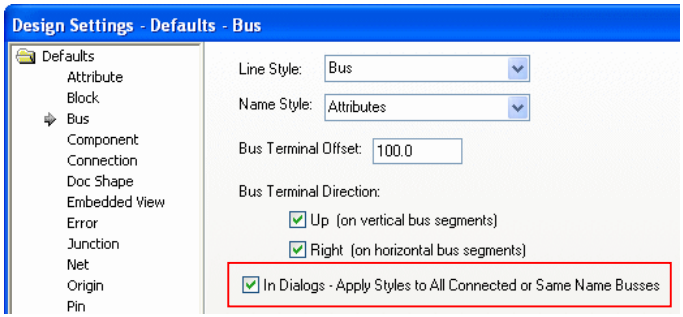
Schematic Bus can have Own Colour

A **Bus** can now have it's own colour. By checking the **Own Colour** check box and selecting a colour, you can override the general Bus colour defined in the Colours dialog.



The **Bus Properties** and **Change Style** dialogs can propagate the **Style** and **Colour** changes to all connected or same name busses by using the new check box.

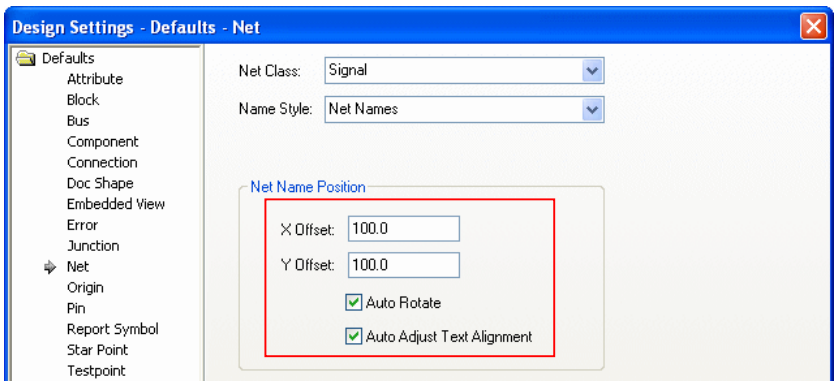
There is a default in the **Design Settings** dialog, **Bus** defaults for this option.



Show Net Name on Change Net

Defaults dialog - Net Name positioning

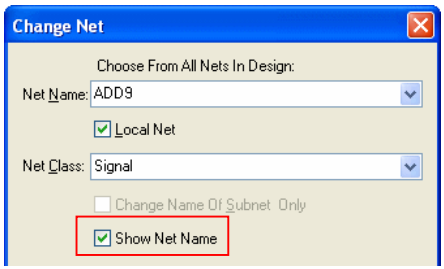
The **Defaults** dialog has a new option on the **Nets** page for **Net Name positioning.** These define the required **Offset** of the name from the pin, junction or bus terminal end, whether it can be **Auto-rotated** to lie along a vertical connection, and if the **Text Alignment** can be altered to be closest corner of the name to the pin.



The **Show Net Name**, **Add To Net**, **Add Connection to Bus** and **Apply Pre-defined Signals** options are other interactive operations that add Net Names, and they will also obey the new default name positioning rules.

Show Net Name on Change Net

A new option on the **Change Net** dialog to **Show Net Name** has been added. This is available if the Net Name is a user defined name. If checked, the name will be displayed on the nearest node to the last picked position on the selected sub-net. If the node already has a hidden net name it will be used, otherwise it will add a new net name and its offset from the pin will be determined by the new net name defaults mentioned above.



If there are no attached connections to resolve the name direction, and the nearest node is a pin, it will work out the direction from analysing the symbol to see which side the pin is on. The new Net Name will have its offset extended if it needs to avoid the displayed Pin and Logic name.

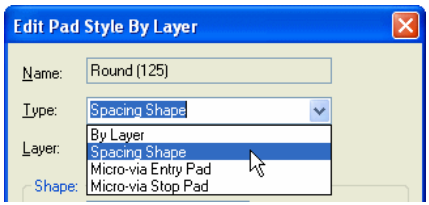
The **Show Net Name** option still has two modes of adding a net name that will vary slightly from **Change Net**; if a single connection is selected it will attempt to add a new net names at the selected point on the connection, and if you pick the free end of a connection, the Net Name will be added to a new junction in a direction away from the connection (rather than along it).

Pad Style Spacing Shapes

New Spacing Shape Pad Exceptions within Technology allow additional spacing around pads to be defined. This could be used, for example, to allow extra spacing around pads on small components which may slip when being placed on the board during manufacturing.

To use this facility, edit the **Pad Style** and add a pad exception using **By Layer:**. Edit its **Type:** to **Spacing Shape**.

Note, in the example below, Micro-Via entries would only be available to you if you have the Advanced Technology cost option.



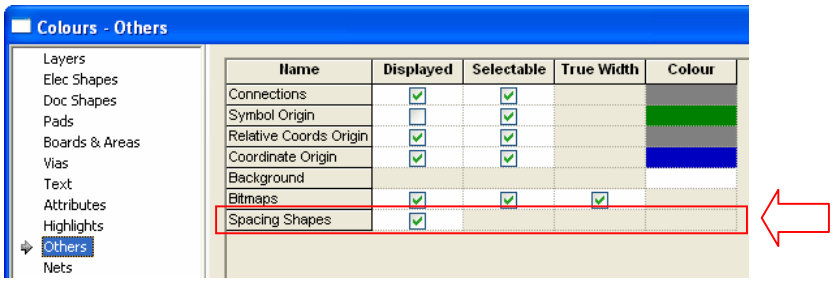
When viewed in the list of **Pad Styles**, this is displayed as a **Spacing Shape** and the layer it represents. You could have two exceptions on the same layer, one for the **Pad Shape/Size**, and one for the **Spacing Shape**.

	Name	Layer	Shape	Width	Length	Drill Hole	Plated
Y	Oval (60 x 110)		Oval	60.0	110.0	32.0	☑
Y	Round (60)		Square	60.0		32.0	☑
Y	Round (70)		Round	70.0		42.0	☑
Y	Round (125)		Round	125.0		50.0	☑
	Spacing Shape	Top	Round	225.0			
Y	Square (60)		Square	60.0		35.0	☑
Y	Via (40)		Round	40.0		24.0	☑
	Via (50)		Round	50.0		32.0	☑
Y	Via (60)		Round	60.0		32.0	☑

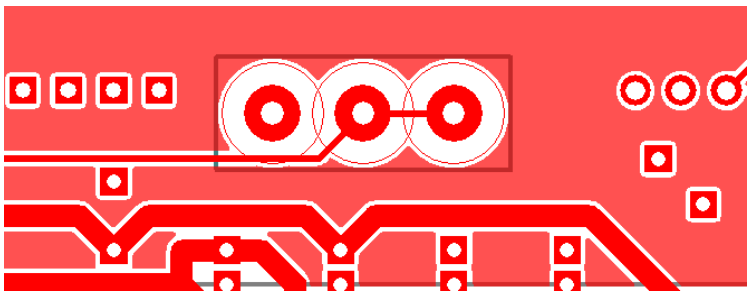
Once the exception has been added, all options that can use additional spacings, such as **Copper Pour**, **DRC** etc. will use the **Spacing Shape** as well.

Spacing Shapes Colours

In the **Colours** dialog, **Others**, there is now a new option for **Spacing Shapes**.

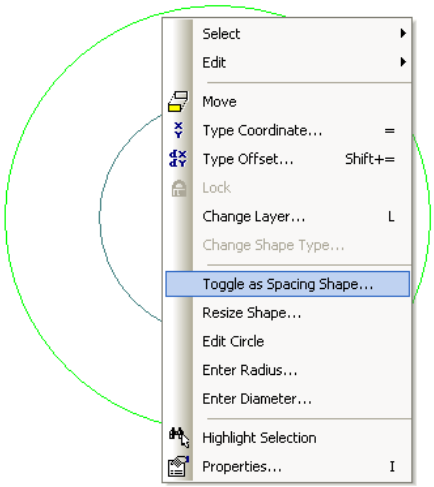


The example below shows the three big pads in the centre of the design after pouring copper and with the Spacing Shapes displayed. Additional clearance (using the Spacing Shape) has been applied to this pad style, copper pour has taken this into consideration.



User Defined Pad Shapes and Spacing Shapes

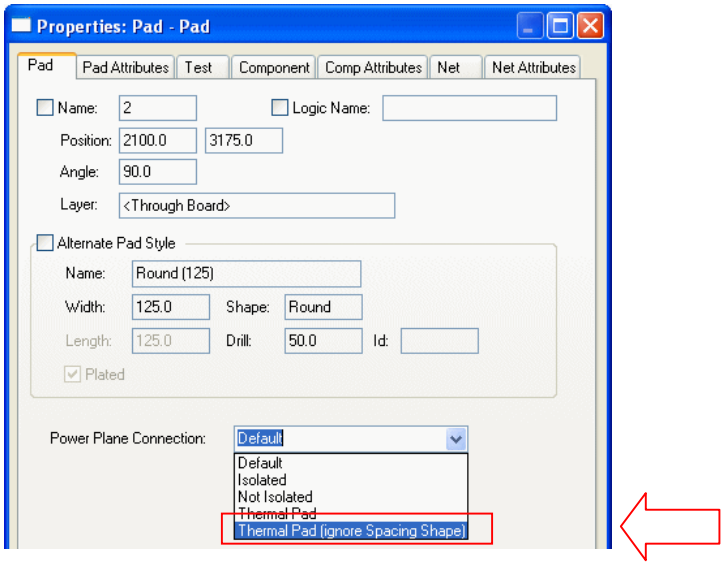
Spacing Shapes can also be used within the **User Defined Pad Shapes** option. For a selected area, from the context menu, select the **Toggle Spacing Shape** function to make the shape a **Spacing Shape**. Spacing Shapes should only be defined on an Electrical layer or the through hole layer, and there should be only up to one shape on each layer. This will be checked when you attempt to save the pad style.



Properties and Spacing Shapes

When using **Properties** of the Pad, Via or Mounting Hole, the **Connect Type** can now be a new item called **Thermal [Ignore Spacing Shape]**.

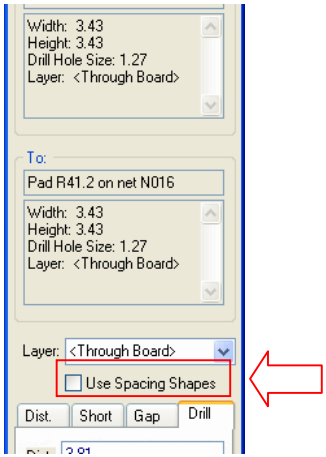
If the **Pad Style** has a **Spacing Shape** defined for the current layer, then this will be used to determine the shape around the pad, instead of the actual pad land shape. This can be disabled by selecting the connection type **Thermal Pad (ignoring Spacing Shape)**, this does not disable the **Spacing Shape** for any other spacing or rule checking process.



This rule is also available in the **Technology** option, **DFM/DFT** page and **Thermal Rules**.

Measure and Spacing Shapes

If one of the items is a pad, via or mounting hole and its style has a **Spacing Shape** defined the **Use Spacing Shapes** check box will be shown. If checked, and the spacing shape is on the current layer being measured, the spacing shape will be used instead of the actual pad land shape. if this is the case, the pad information will also be changed to show the spacing shape size.

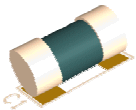


Changes to 3D Viewer

New 3D Package Shapes

New 3D package shapes have been added for BEAD, MELF, DILSwitch and LEDdisplay.

BEAD

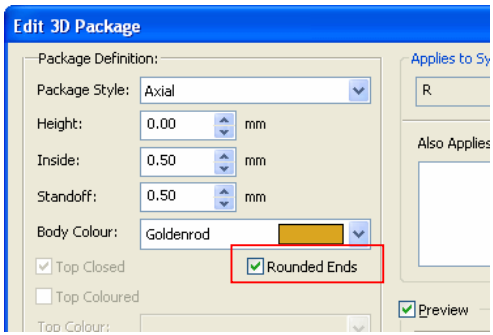
MELF 

DILSwitch

LEDdisplay

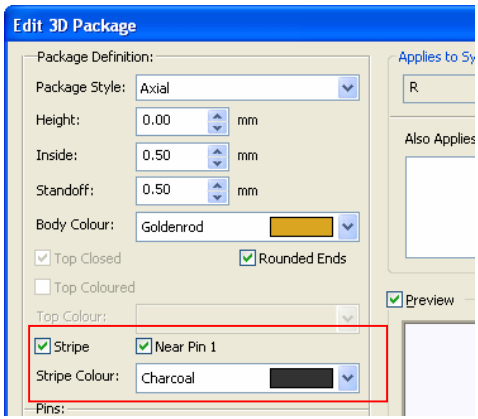
Axial Components - Rounded Ends

For **Axial** components you can choose to round off the ends of the components using the **Rounded Ends** check box.



Colour-coded Bands on Components

Package Styles of **Can** and **Axial** can now be given dynamically colour-code stripes (bands) on the component body. The colour-code bands are based on the Attribute value of the component.



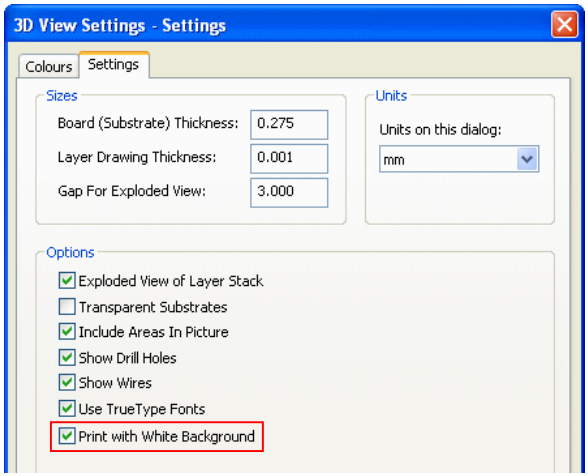
CAN shapes will get a stripe up the side of the body near to pin 1 or pin 2 depending on the pin 1 checkbox. **Axial** components get a single stripe near one end or the other (Capacitors, Diodes etc.), except for **Resistors** and **Inductors** which will get the correct set of coloured stripes depending on the value of each component.

The **Value** attribute is read and converted to a standard colour coding. Values in formats such as 2.2K or 2K2 or 10u are acceptable.

For **Resistors**, a **Value** or **R** attribute is read and converted to a stripe, and for **Inductors**, the **Value** or **L** attribute is read and converted to a stripe. For both types, the **Tolerance** or **Tol** attribute is used for the tolerance band.

3D View - Print With White Background

The background of the image saved (using **Save As Image**) will be white if the setting for **Print with white background** is enabled in the **3D Settings** dialog.

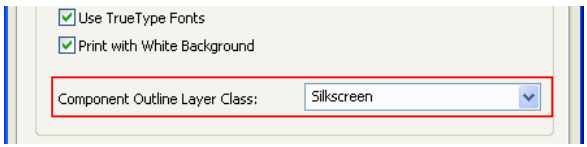


3D View - Save As Image

From within the 3D view screen, you can now write the current 3D view out as an image file using the option **Save As Image** from the **File** menu. **Save As Image** supports Bitmap Files (*.bmp), JPEG Files (*.jpg) and TIFF Files (*.tif).

Layer Type Choice for Component Body

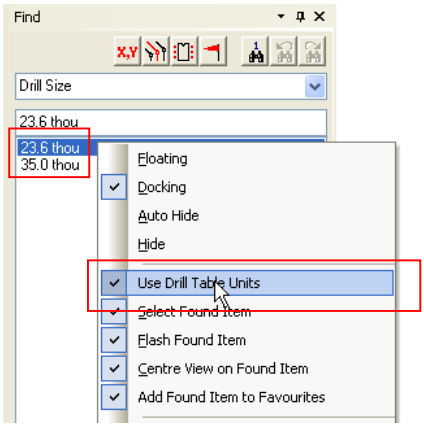
There is a new selection the **Settings** page of **3D Settings** dialog to choose the **Component Outline Layer Class**.



This specifies the name of a **Layer Class** to be used when determining the 'size' of the body of the component. If this Layer Class name is specified, only shapes in the footprint that live on a layer of that class will be considered. The default value is to have no Layer Class name defined, which then allows any shapes to be matched and used for generating the 3D component bodies. To remove a previously specified Layer Class name, simply delete the text from this box.

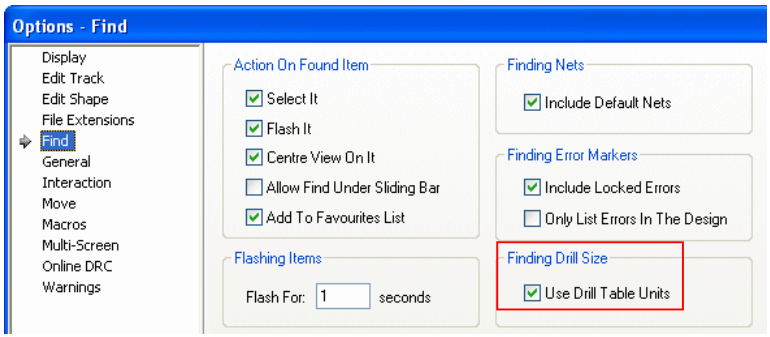
Find Bar – Drill Sizes in Table Units or Design Units

There is a new option on the **Find** bar context menu to choose between displaying **drill sizes** using the **drill table units**, or using the **design units**.



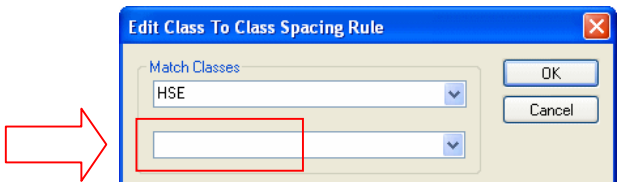
The **units** text is now also shown after each drill size.

This option is also available on the **Find** page of the **Options** dialog allowing it to be pre-set.

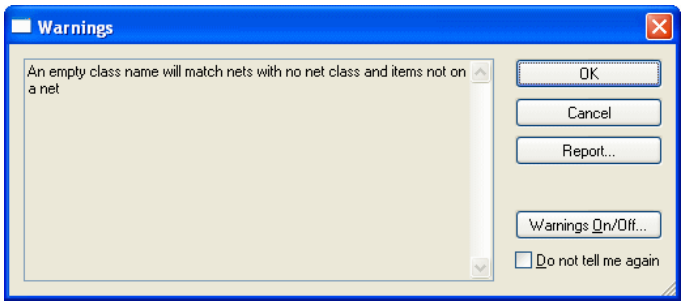


Spacing Wildcard Match

The wildcard match string for **Net Class** names in the **Technology** dialog and **Spacing Rules**, can now be a blank string. This means it matches just items not on a net or on a net with no Net Class. You can therefore define specific rules for items that are on no net, items such as Copper or Free Pads with no Net Name or Net Class.



When you add a rule using a blank string, a warning dialog is displayed:



Spacing Wildcard Match Rules

With the blank string rule now available, below is a reminder of the string rules available to you:

***** - matches everything on a Net or not.

?* - matches everything that has a Net Class.

Blank string – matches anything with no Net or no Net Class.

Named string - matches a specific Net Class.

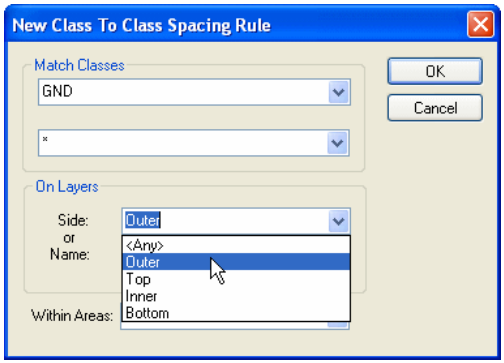
Wildcard Strings

All wildcard strings can now use | for **or**. So when used, **power|gnd*** would match **power** or **gnd***.

Spacing Rule Class to Class Match

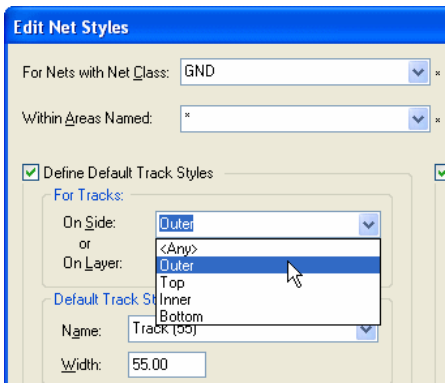
Technology | Spacings

From within the **Technology** dialog and **Spacings**, **Match Net Class Pair**, you can now specify a single rule which matches both **Outer** layers (Top and Bottom).



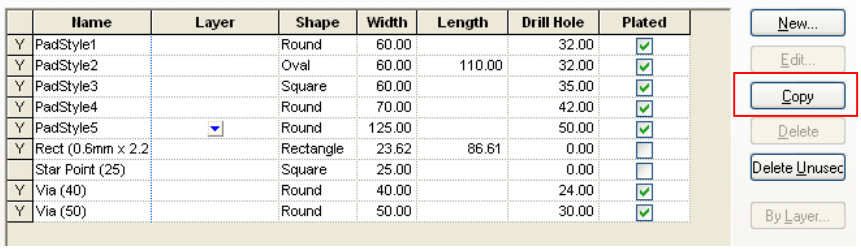
Technology | Net Styles

From within the **Technology** dialog and **Net styles**, you can now specify a single rule for **Track Styles** which matches both **Outer** layers (Top and Bottom) for the **On Side:** style.



Copy Pad Style

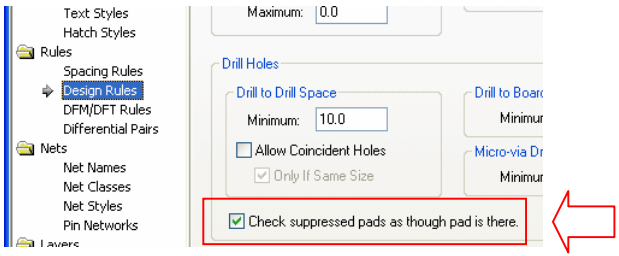
There is a new button on the **Technology** dialog and **Pad Styles**. A **Copy** button has been added, this copies the currently selected pad style and all the **By Layer** exceptions, if the list is expanded.



Suppress Pads on Powerplanes

There is a functional change to be aware of, this effects existing designs as well as new designs but is controlled using a switch. All processing, **Powerplanes**, **Copper Pour** and **DRC** option now take suppressed pads into account. All checking uses the drill size (if there is one) rather than the pad size. This applies even for unmodified existing designs, this will result in a smaller ring around suppressed pads on powerplane layers. There is no interface change as such for this update. Pad suppression is set up on individual layers.

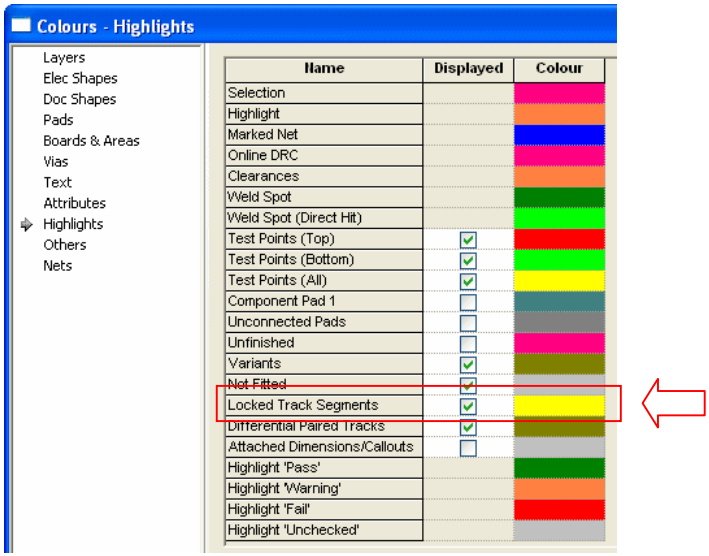
By default, the switch is set to work as it did previously. If however, you uncheck the switch, it works in the new way described above.



Highlight Locked Tracks

There is a new option on the Colours dialog, Highlights tab named **Locked Track Segments**. By selecting the check box and choosing a suitable colour, locked track segments in the design can be highlighted.

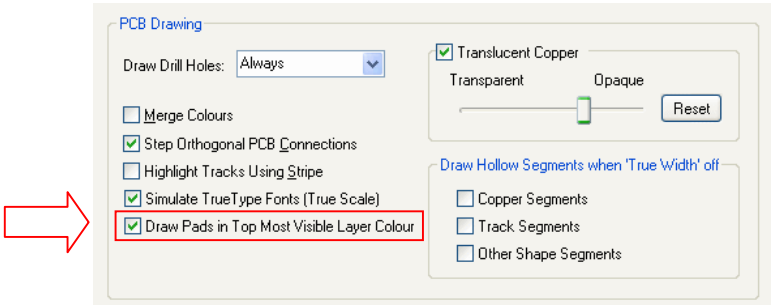
In the Footprint editor, this feature is shown as **Locked Breakout Segments**.



In the Schematic editor, this option is shown as **Locked Connection Segments.**

Draw Pads in Top Most Layer Colour

There is a new switch in the **Display** dialog under **Options**, **Draw Pads in Top Most Layer Colour.** This causes pads to be drawn in the top most visible colour instead of the *Side* or *Through Board* colour. A through-hole pad might be drawn in the Top Electrical layer colour if this were the *top most* visible layer.

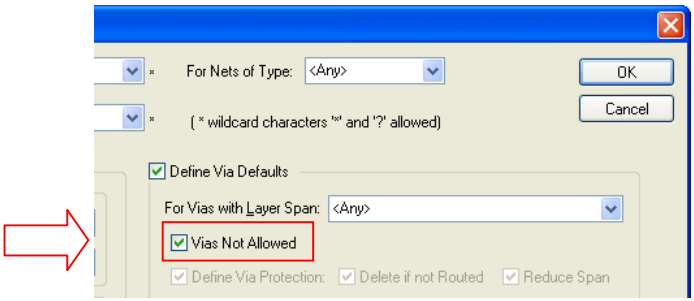


Net Styles Dialog Changes

Stop Spans Being Used For Vias in Net styles dialog

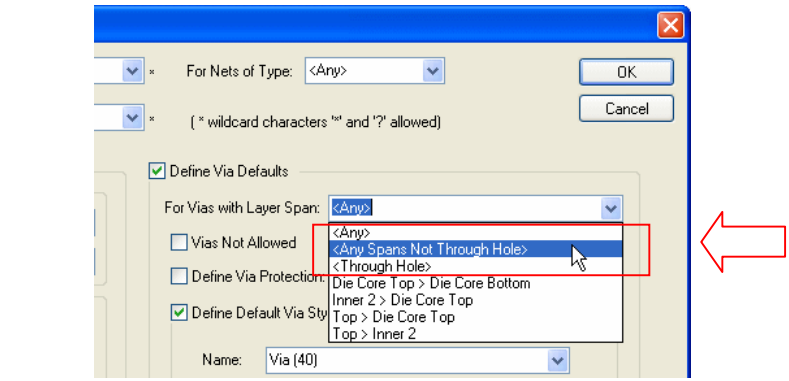
Using the **Technology** dialog and **Net Styles**, you can now specify that a particular layer span should not be used for automatically inserting vias during manual routing.

When editing a **Net Style**, you can check the **Vias Not Allowed** box. This is used when automatically inserting a via (or changing its span) by changing track layers the spans marked with **No Vias** in the **Net Styles** entries will not be used and the next best via span will be used.



Layer Spans set to Any Spans

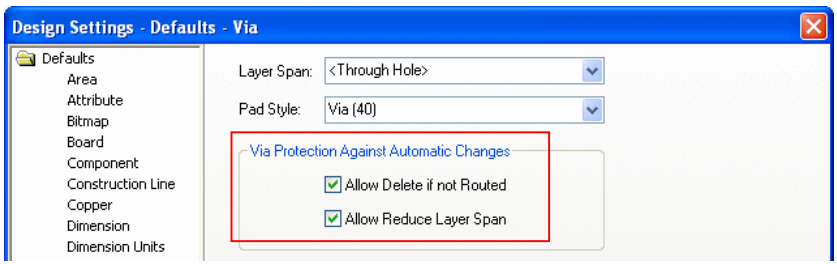
There is also a slight change in **Net Styles** in that the layer span can be set **as <Any Spans Not Through Hole>,** so the via style, via protection flags or **Vias Not Allowed** flag can be applied to all spans except the through hole span in net styles entry.



The combination of these two features enables you to easily specify for a particular net class, or in a particular area, that vias can only be added with through-hole spans.

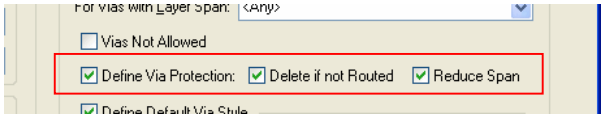
Via Protection Against Automatic Changes – Default Settings

From the **Setup** menu, the **Defaults** dialog and **Via Defaults** page now has a **Via Protection Against Automatic Changes** section with switches **Allow Delete if not Routed** and **Allow Reduce Layer Span**. These will be used for vias added whilst changing track layers.



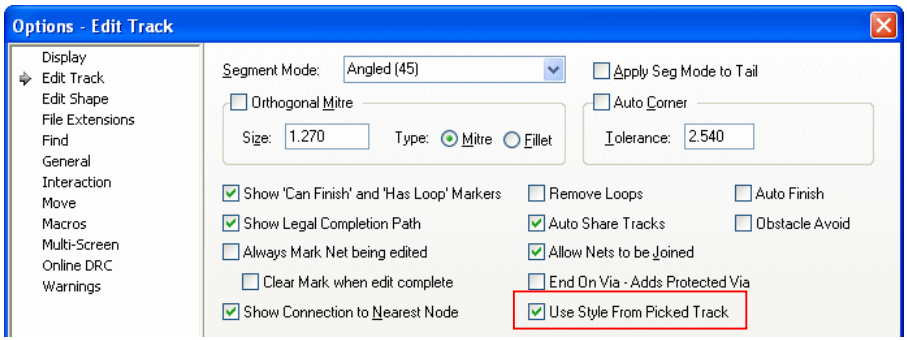
*Note: the **Add Via** and **Add Testpoint** functions will still always add fully protected vias, and there is still an interactive option **End On Via – Adds Protected Vias**, but if you switch it off the new defaults will be used.*

The via protection for new vias can also be overridden for particular **Net Classes**, or for particular **Layer Spans**, or within specified areas using the **Net Styles** dialog. You will see a new **Via Protected** column, which will have text in if the protection has been set for an entry. To change the via protected state **Edit** the row and in the **Edit Net Styles** dialog, check the **Define Via Protection** check box and choose the required protection.



Add Track Option – Use Style From Picked Track

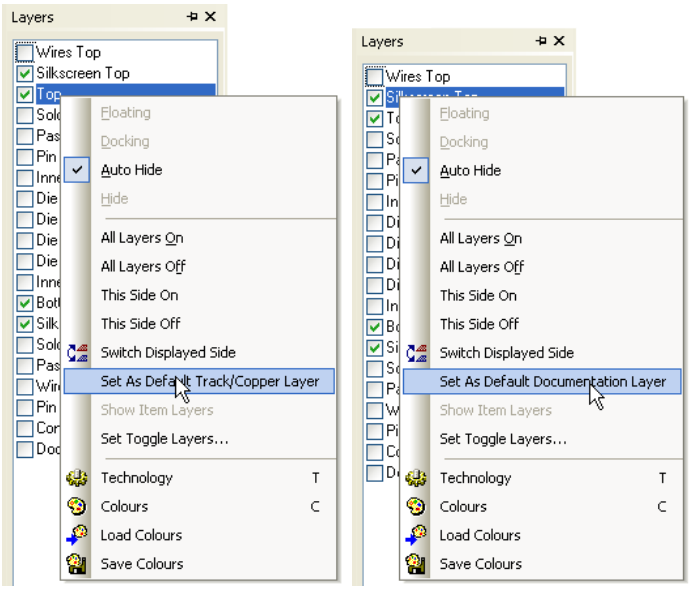
In the **Options** dialog under **Edit Track**, there is a new option **Use Style From Picked Track**.



If enabled, when adding a PCB track and starting on another track, the new track will take its style from the track segment being picked.

New Context options in Layers Bar

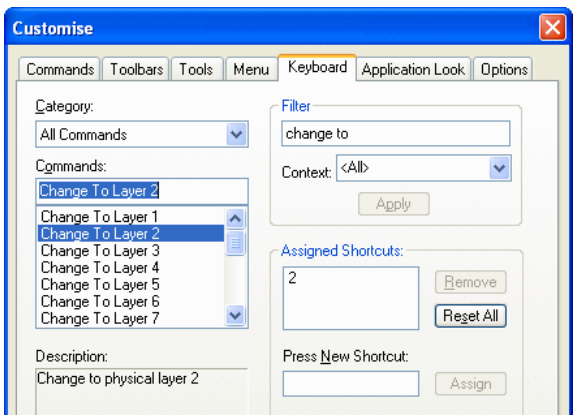
Two new options have been added in the **Layers** browser on the context menu to set default layers.



Use **Set As Default Track/Copper Layer** when an electrical layer is selected. Note it will also set the default template layer. Use **Set As Default Documentation layer** when a documentation layer or non-electrical layer is selected. Setting the default documentation layer will set the default layer of most items that are added for documentation purposes, except **Dimensions** and **Construction Lines** that are usually kept on their own layers.

New Commands for Change Layer

A new set of commands are available to change the layer of the selected item directly to an electrical layer. In the Customise dialog, the commands are labelled **Change To Layer 1** to **Change To Layer 10,** layer 1 being the Top Electrical layer. You can assign keys to these commands (such as 1 through to 9).



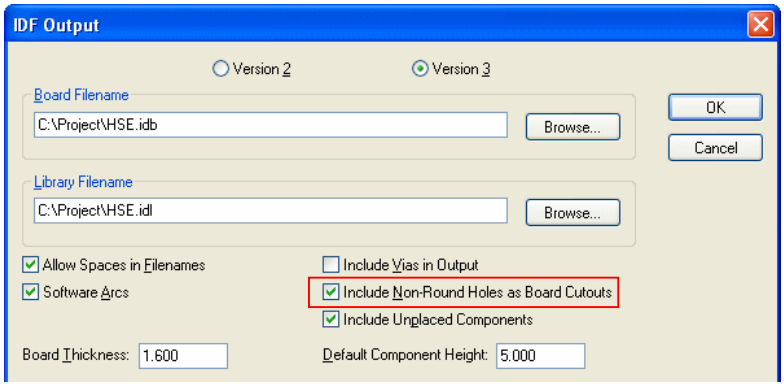
Once assigned to a key, use **Change To Layer 1** to change to the Top Electrical layer, **Change To Layer 2** to change to the next electrical layer and so on. These commands are only available if the selected item can exist on an electrical layer.

Component Pushing uses Accurate Shapes

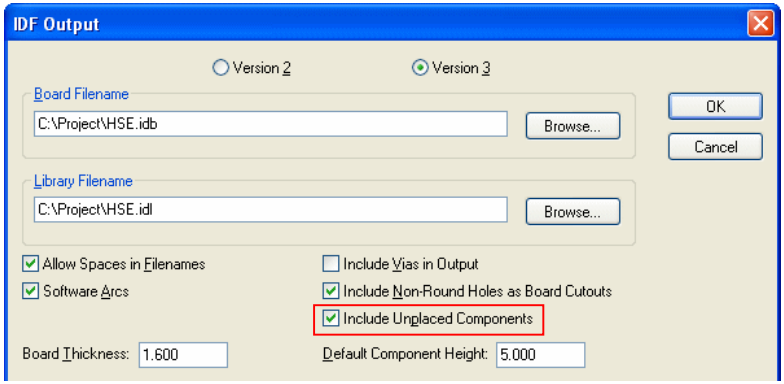
When pushing components, it takes the actual component shapes and pads into account. This is particularly useful for round components, and through-hole versus surface mount components where accurate shapes are used for the pushing boundaries.

IDF Output Enhanced

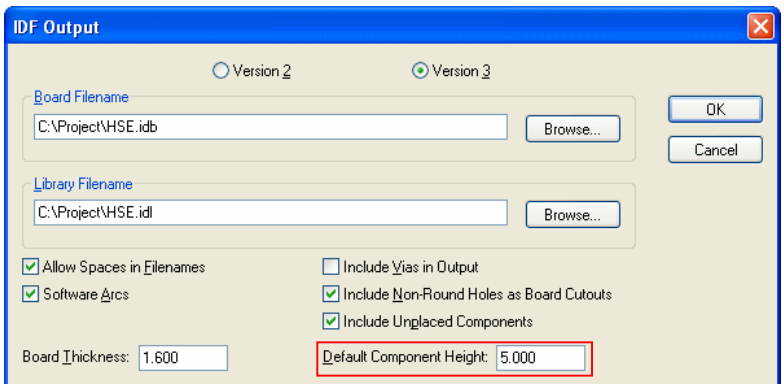
There is a new check box on the IDF Output dialog **Include Non-Round Holes as Board Cutouts**. Check this to output pad, mounting hole and via slots as cutouts of the board outline. Slots in vias will only be output if existing option **Include Vias in Output** is also checked.



There is now an option in the dialog to **Include Unplaced Components**. In previous versions, it always included unplaced components.

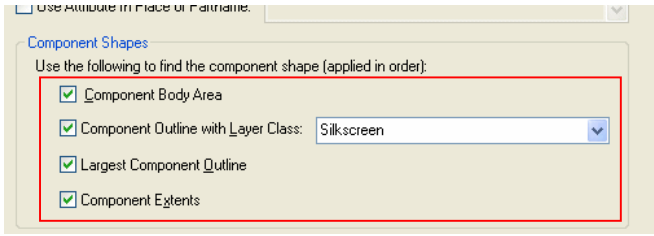


You can now enter a **Default Component Height** value in the IDF Output dialog to be used for components that don't have a height value. In previous versions, the output set the height to zero if it didn't exist in the component. The Units used will be design units.



Component Shapes

There is now more choice for choosing what shape is output for components. It allows you to choose the **Component Body Area**, and lets you disable use of any shapes that you don't want, for example outlines that are not on the **Layer Class** selected.



Multiple Board Outlines

If multiple board outlines, the main one (containing the others) is output in the BOARD_OUTLINE section, and the other boards are output as additional shapes in OTHER_OUTLINE sections with identifier BOARDn.

Overlapping Cutouts

During output, all overlapping cutouts in the board will now be merged together.

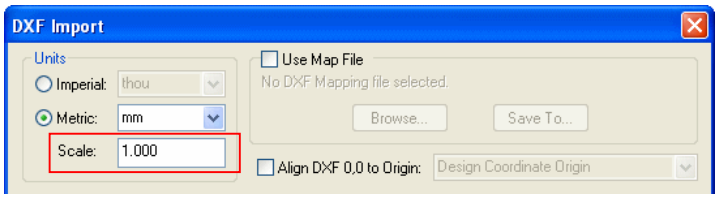
Import IDF Other Outlines

Import IDF will now read OTHER_OUTLINE shapes with identifiers starting with BOARD as additional board outlines.

DXF Import Enhanced

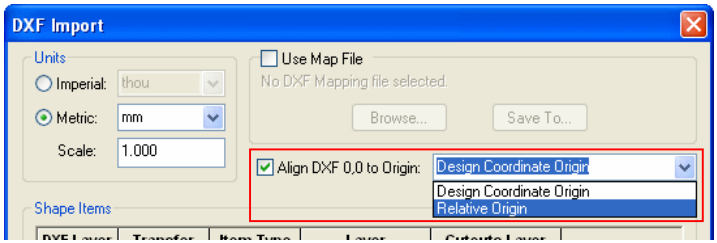
Scale Import Plot

The ability to Scale a DXF during import has been added. This control will help some DXF files where the scale is not as expected.



Align DXF 0,0 to Origin

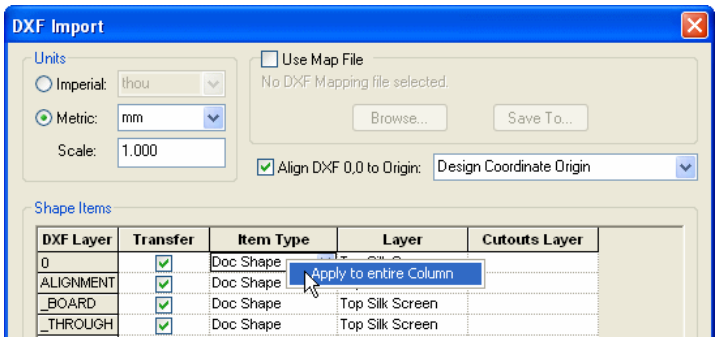
Normally, the imported DXF is placed at the coordinates defined in the incoming DXF file. If you prefer, you can choose instead to align the DXF to the **Design Coordinate Origin** or the **Relative Origin** (if defined).



To do this, simply check the **Align DXF 0,0 to Origin** check box, and choose the origin from the dropdown list. This feature can be used for example to load a DXF board outline into an existing design which already has a sheet outline outside where the board needs to be positioned. By placing your relative coordinate origin at the desired location for the board, the incoming DXF data will be shifted to that point.

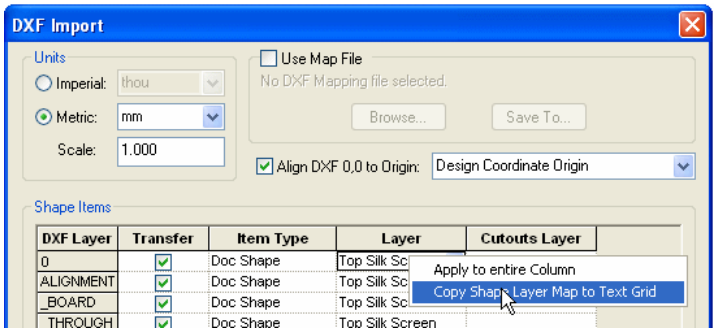
Apply to Entire Column

When the cursor is over the **Item Type** or **Layer** columns, you can choose **Apply to Column** from the context menu to apply the current type or name to the whole column.

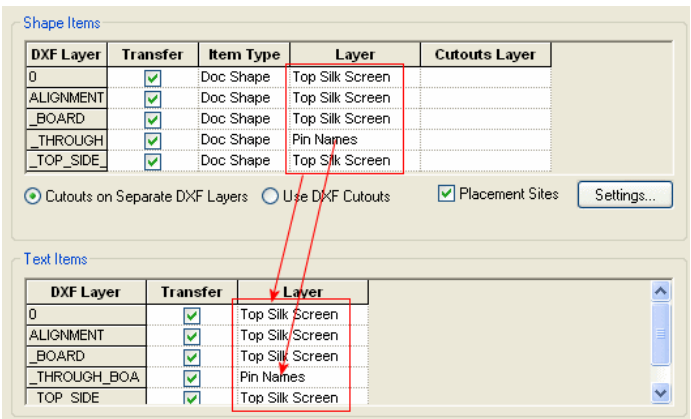


Copy Shape Layer Map to Text Grid

When the cursor is over an item in the **Layer** column, you can choose **Copy Shape Layer Map to Text Grid** from the context menu.



This maps the text layers to be the same as the shape layers.



STEP I/O Improvements

Export Improvements

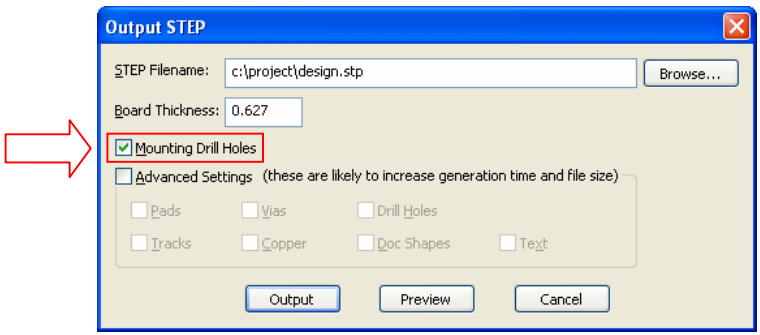
Component identifier tagging is now done automatically in files exported in STEP format from Pulsonix. This allows for identification when used in the MCAD system. Both Component Name and Part Name are exported.

Model Colours used in Preview

If the model you are referencing in the design has its own colours defined, the STEP Previewer will now use these colours. When the STEP file is written, the model colours will also be included in the output file for further use in the MCAD system.

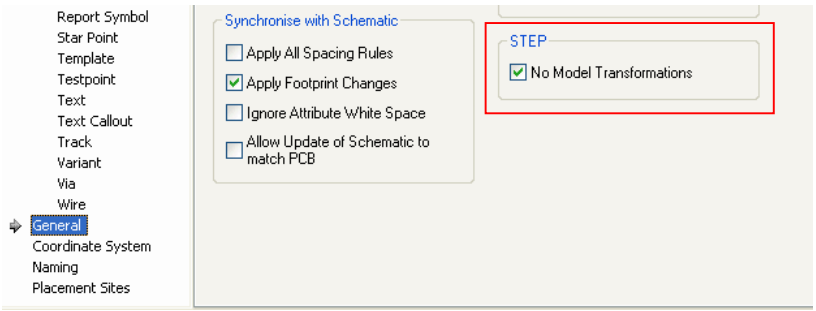
STEP outputting Mounting Drill Holes

There is now an extra check box on the **STEP Output** dialog which allows **mounting drill holes** to be output irrespective of the **Advanced settings**.



STEP Export – No Model Transformations

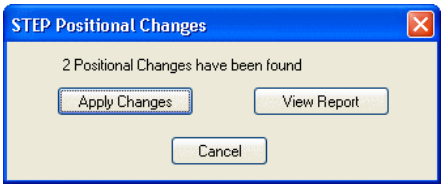
The **No Model Transformations** setting is used if all your STEP models have been previously aligned to match your footprints rotation etc. This allows the transformations to be ignored. This option is selected in the **Settings** menu, **Design Settings** and **General**. This setting is made on a per-design basis.



STEP Positional Changes Imported

When Component positions have been changed in the mechanical system, these positional changes can now be imported back into the Pulsonix PCB design. It will compare the current positions of the components with positions of the components as described in the STEP file.

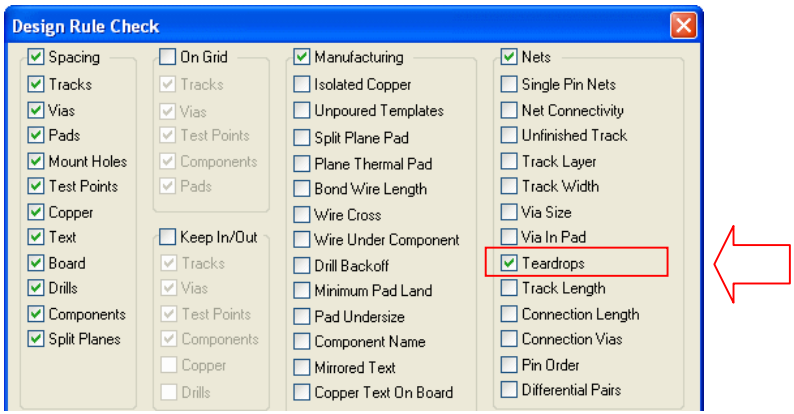
To import positional changes, from the **Utilities** menu select **STEP Positional Changes**. If there are any changes you will be asked if you to apply the changes.



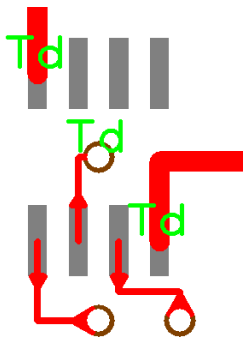
Select **View Report** to see which components have changed. Select **Apply Changes** to move the components.

Design Rules Check – Check for Missing Teardrops

You can now check if a teardrop is missing due to a track corner preventing the teardrop from forming.



For example, teardrops may not be added because a track entering a pad has sufficient space or clearance to add it, or a corner into a via has insufficient length.



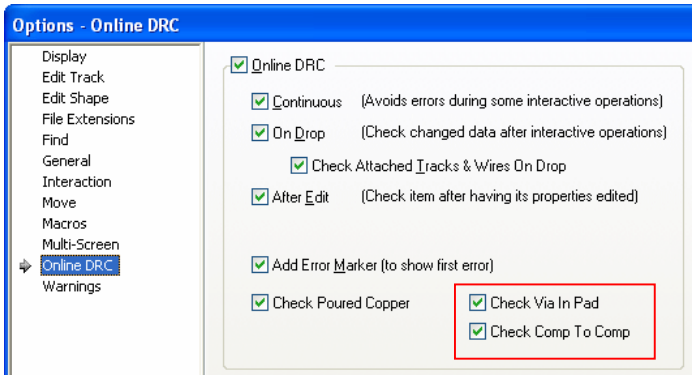
When DRC is run, any errors found will be reported with a Td error marker like this:

```
Missing Teardrop (Td) At (15849.0 4400.0). Layer 'Bottom'.
Missing Teardrop (Td) At (16400.0 5400.0). Layer 'Bottom'.
Missing Teardrop (Td) At (15025.0 5202.4-). Layer 'Top'.
```

Online DRC – Check Via in Pad

Online DRC – Check Comp to Comp

Two new Online DRC Options, **Check Via In Pad** and **Check Comp To Comp** to enable switching off these checks when performing online DRC checking.



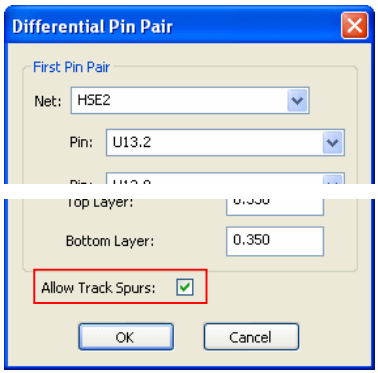
Check Via In Pad - Uncheck this option if you do not want Online DRC to check for vias within pads.

Check Comp To Comp - Uncheck this option if you do not want Online DRC to check the spacing between Components.

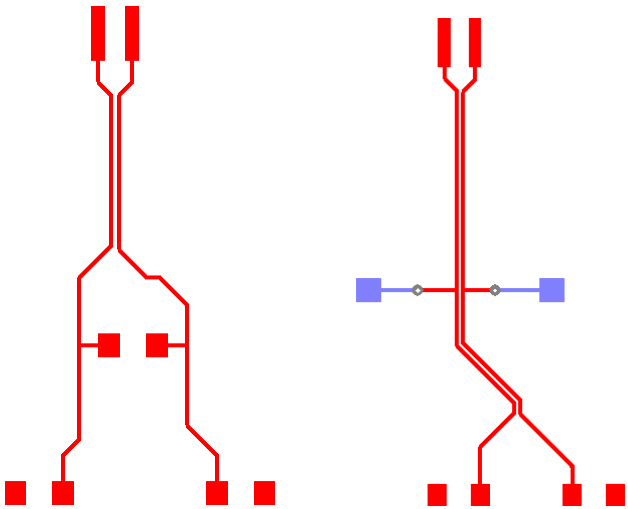
High Speed Option - Differential Pairs – Spurs Allowed Check

Within the **Technology** dialog and **Differential Pairs**, the Differential Pairs dialog now has a flag to say if **spurs are allowed**.

A spur is defined as a single path to an electrical item with no sub-spurs.



This allows you to add spurs off without error as is shown in the two examples below:



When running **Design Rules Checks**, the **DRC** and **Differential Pairs Reports** are more consistent in reporting spur errors.

Changes to Calculators

The **Track Width**, **Resistive** and **Heat Sink** calculators can now take an ambient temperature that can be set below zero, in other words, a negative value.

Ariadne Import (Cost Option)

As part of the Advanced Import cost option, you can now import Ariadne Schematic designs, PCB designs and Libraries. Please contact sales or your local Pulsonix distributor for more information.