Pulsonix Design System

V4.5 Update Notes

Copyright Notice

Copyright © WestDev Ltd. 2000-2007
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: 06/02/07 iss 2

Pulsonix
Oak Lane
Bredon, Tewkesbury
Glos, GL20 7LR
United Kingdom

Phone +44 (0)1684 773881
Fax +44 (0)1684 773664
Email info@pulsonix.com
Web www.pulsonix.com

Contents

CHAPTER 1. GETTING STARTED WITH V4.5 7
About these notes 7
Installing the New Version of Pulsonix 7
New License 7
New System Files 7
New Additional Supplementary Files 7
CHAPTER 2. GENERAL OPTIONS 9
Automatic inclusion of new Menu/Toolbar options 9
Print Enhancements 9
Windows Plot across multiple pages 10
Auto Weld Improvements 11
Report Maker Changes 12
Load/Save Configuration Files 14
Show Relative Coordinates 15
Recent Used File List now user definable 15
Back Annotation 15
Auto-Hide delay 16
Zoom Factor Control 16
New Command in Move 16
Move Option 16
Select Improvement 17
Colour Files Bar 17
New Cross-Probe Option 18
Font Size in HTML Help 18
World View 18
Changes to Find Browser 19
Merge Shapes 21
Text Callouts 22
Part Editor Improvements 24
Part Editor Attributes 25
Changes to Hyperlinks 26
Default Part Attributes 27
Attributes can be back annotated 27
Change Style Apply to Whole Net 28
CHAPTER 3. SCHEMATIC OPTIONS 29
New ERC Checks 29
Split Net 30
Delete 31
Disconnect Con End / Remove From Net 32
Next/Previous Commands for Insert Doc Symbols 32
CHAPTER 4. PCB OPTIONS 33

Enhanced PCB Footprint Wizard 33
Changes to the Library Manager 34
3D Viewer 35
3D Settings 35
Defining the 3D Packages 37
Changes to Define Pad Shape 42
Layer Stack Preview 42
Intelligent Gerber Import 42
Generate Component 46
Free Pad to Via 48
Layer Toggle 48
Swap Routed Gates & Pins 50
Edit Track Reverse Edit Direction 51
Square Ended Tracks 51
RF Mitres using Chamfered Track Corners 52
Unroute Tracks Change 53
Track Hugging Improvement 53
Drill Names 54
DRC Changes 55
Online DRC Changes 55
Online DRC Change in Options 55
Change to Component to Component Error Markers 55
Router Dialog Change 57
Moving Copper Terminals 57
Thermal Pad Connection Rules 57
Changes to Add Via 58
Change to Protected Vias on Edit Track/Breakouts 58
High Speed Option – Changes to Differential Pair Routing 58
Options Changed in later V4.0 Patches 59
ODB++ Dialog Changes 59

CHAPTER 5. SPICE SIMULATOR CHANGES 61

Simulation Update 61
Changes to Edit Value (F7) dialog for Passives 61
Safe Operating Area (SOA) Testing 61
New Built-In Simulator Parts ‘Watch’ Probe Functions added 62
Edit Device Parameters 63
Full SOA Options Details 63
New Simulator Control Part 68
Simulator Bug Fixes 68

APPENDIX A. SUPPLEMENTARY FILE CHANGES 69

Format Files 69
Updated Libraries 69
New Package Libraries 69

Chapter 1. Getting Started With V4.5

About these notes

These update notes are provided for existing users as a supplement to the existing Pulsonix Users Guide. These notes are to highlight new features in version 4.5 and to briefly describe their use.

Each chapter is broken down into logical functional descriptions based on the application type, Schematic design, PCB design etc.

Installing the New Version of Pulsonix

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

As with any other installation of Pulsonix, insert the CD and wait for a short time. The *Autorun* facility will start the installation procedure. Follow the on-screen messages from the install wizard. You can install Pulsonix version 4.5 on top of your existing installation, you do not need to uninstall any old version first.

New License

Version 4.5 requires a new license that will be supplied to you with the update or by email.

For existing users it is recommended that you simply click the **No Change In Licensing** check box on the licensing page of the Installation wizard. New licenses and changes to network licensing can be made after the installation using the **License Manager**.

New System Files

There are no new system files for version 4.5.

New Additional Supplementary Files

New Package libraries to support the 3D Viewer option are now supplied. Details of all additional files supplied are supplied in Appendix A at the end of this document.

Chapter 2. General Options

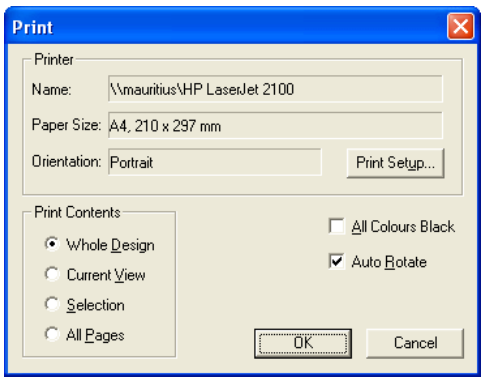
Automatic inclusion of new Menu/Toolbar options

Pulsonix will show new menu options and toolbar buttons without having to explicitly reset menus or toolbars. Any customisation that you might have setup will be retained.

All new commands will be initialised as 'always show' options in the menu, but will revert to hidden 'infrequently used' items over time if not used.

Print Enhancements

The **Print** command now displays its own dialog, rather than the standard windows print dialog. This provides additional options to support Pulsonix design. It has the ability to print the **Whole Design**, and in a Schematic, **All Pages**, as an alternative to just printing the current screen view.



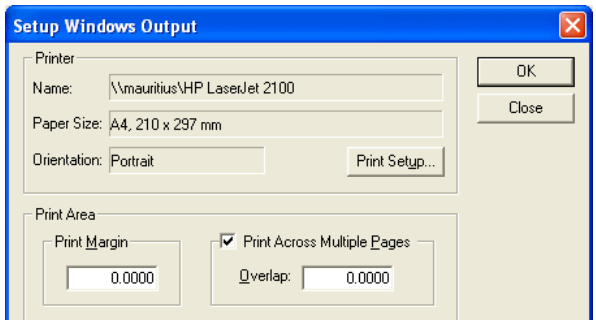
Printing the current selection now prints using design colours rather than printing everything in the selection colour and fits the print around the entire selection rather than printing only the selected items visible in the current view.

There is also the option to select **All Colours Black** and to **Auto Rotate** which will automatically select the best design orientation to fit the current paper orientation.

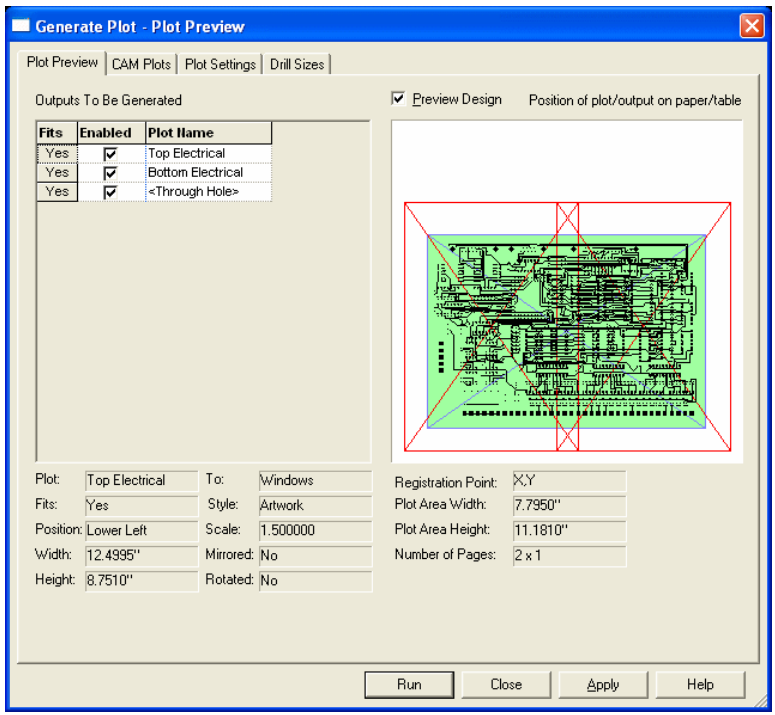
Windows Plot across multiple pages

The **CAM Plot** option has been changed to allow the printing of pages across multiple sheets.

The **Windows Setup** dialog now has an additional option to select **Print across Multiple Sheets** and optionally specify an **Overlap** value. This will allow a single large plot to be spread across several pages, e.g. print A2 across 4 pages of A4. It also now allows you to setup your own additional **Print Margin**, this is in addition to the printers own hardware margin.



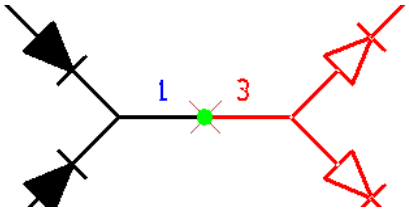
When a print will extend over multiple sheets of paper, the **Plot Preview** shows all pages and how the print will fit on them. The **Number Of Pages** to be used will also be shown as an addition to the plot Fits status.



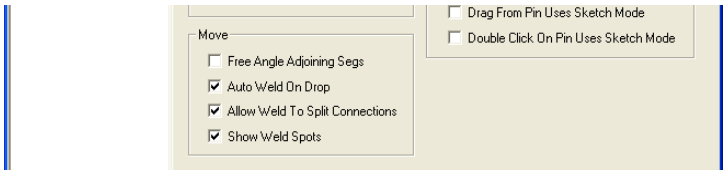
Auto Weld Improvements

Weld Spots

Weld spots are displayed during move to show where a weld would be attempted if you were to finish at this position.

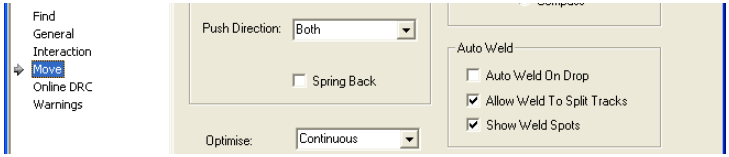


They are drawn using a new highlight colour and are only displayed if the **Show Weld Spots** is enabled on the **Options** dialog and **Interaction** tab for Schematic designs and on the **Move** tab for PCB designs.



Weld to middle of tracks

There is a new option **Allow Weld To Split Tracks** on the **Options** dialog; on the **Interaction** tab for Schematic designs and on the **Move** tab for PCB designs. Check this to specify if tracks can be split by welding a moving pad (or track end) into the middle of them, or leave it unchecked to state pads can only be welded to free ends of tracks.



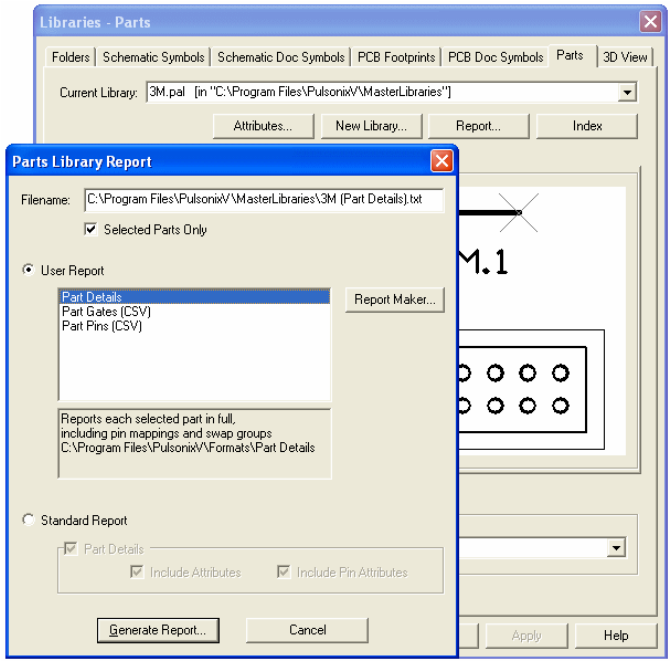
Weld Now on the context menu

This is an option on the context menu when moving components and is now available even if **Auto Weld On Drop** in the **Options** dialog is not enabled. This allows you to weld when you want to rather than every time a move is completed.

Report Maker Changes

Reports On Libraries

You can now run the Report Maker on all types of libraries (except 3D View).



From the **Library Manager** page use the **Report** button. This dialog now offers the choice of running the standard report, or a new user report. The **Report Maker** button takes you to the Report Maker dialog to define a report for the library.

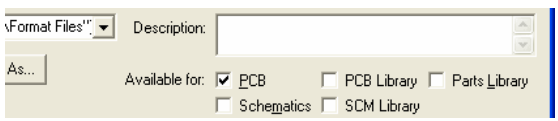
When editing a Footprint, Schematic Symbol or Doc Symbol, from the **Output** menu, the **Report** dialog now has a **User Reports** checkbox. This enables you to run a library report on the single symbol you are editing.

To handle the new library reporting mechanism, the Report Maker dialog now has three new **Available for** contexts:

PCB Library - for running on Footprint and PCB Doc symbol libraries.

SCM Library - for running on SCM symbol and Doc symbol libraries

Parts Library - for running on a Parts library.



Save Inserted User Report

There is a new command **Save Inserted User Report** that can be run using **Run Command** if an inserted report is selected in a design. It will create a format file that contains the commands from the inserted report. This is not on the menu but is useful for retrieving reports from old designs and for support purposes.

New or changed LIST commands

List of Libraries – this is only available when running a report on libraries. It forces you to report the items in each library in turn. If you don't use this, and you are reporting with the lib manager dialog for [All Libraries] then the items will be processed in the order they appear in the library manager dialog.

List of Symbols – this is available on the four symbol libraries, and for PCB and Schematic designs. It reports each symbol in the library or design in turn. Note: this is still required if editing a single symbol but it will be the only symbol reported.

List of Parts – this has been extended for use on Parts libraries as well as on Schematic or PCB designs.

List of Footprint Names – this is used within a **List of Parts** command to report all the footprint names used in the Part.

List of Part Gates – this is a new command used within a **List Of Parts** command. Use it to report information on each gate in the Part.

List of Part Pins – this is a new command used within a **List of Part Gates** command. Use it to report information for each pin on the gate.

List of Attributes - available on all items that can have attributes, including Symbols, Parts, Part Pins Nets, Pins, etc. It lists each attribute on the item in turn. You can use the **Attribute** command for each item to report items name, value and (if present) position.

List of Free Pads – this has been extended for use within **List of Symbols** as well as for a PCB design. This lists all the pins (or pads) in the symbol.

List of Layers - this has been extended for use within **List of Symbols** for a footprint library. Lists all the layers used in the footprint.

New command fields

Library Name - use within a **List of Libraries** or within a **List of Symbols** to report the library file name.

Library Path - use within a **List of Libraries** or within a **List of Symbols** to report the full location of the current library being reported.

Is Associated Part - use on a Part to report if it is an associated part.

Part Group name - use on a Part to report the first Part name in the library group the Part is a member of. If the Part is not in a group, it reports its own name.

Part Group Count - use on a Part to report how many Parts are in its library Part group.

Date, Time and User - Three commands used at the top level (library or design) to report the current date and time and the name of the current user.

Last Saved Date and Last Saved Time - Two commands used on a Part or Symbol to report the Date and Time of the last time the item was saved in a library. In other words, when used together these fields give the library version for the Part or symbol.

Pin PCB Name and Pin SCM Name - Two commands used on a Part Pin, along with the existing commands Logic Name and Pin Name, to report the different names a Part pin has. PCB name is the footprint sequence number, and SCM name is the SCM symbol sequence number.

Swap Group - use on a Part Gate or Part Pin to report the swap group name.

Pin Network - use on a Part Pin to report its Pin Network name.

Name Range Format - used to change the format of name ranges output for Parts and component groups. Adding this command displays an edit dialog that allows you to set the range separators to any string, and set the range start and end values to blank. So you can get "U1-9,U11" or "U[1-9], U11" for example.

Load/Save Configuration Files

It is now possible to save/load current program settings to/from a named configuration file. The Save and Load options are only available from the **File** menu with no designs open. The configuration file can be used to copy a complete configuration to any machine running Pulsonix.

Show Relative Coordinates

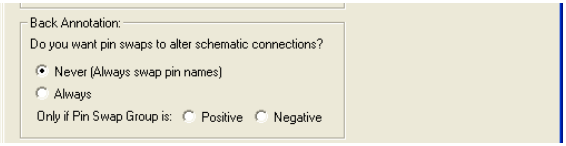
When working in Relative mode, you can now force dialogs which show positional information to show everything relative to the relative origin. This option is set in the **Options** dialog and **General** tab under **Relative Coordinates**. If this option is left unchecked, the positional information will still be shown in Absolute coordinates even when you are working in relative mode.

Recent Used File List now user definable

The **Options** dialog and **General** tab now has an entry for you to define the number of files that will appear on the recent used file list. The list is limited to a maximum of 20 items. You will have to restart the program for the new value to take effect.

Back Annotation

There is now an option for **Back Annotation** to apply pin swaps by swapping the connections between the pins (like in PCB) instead of exchanging the pin names. There is also an option for doing this for only some swap groups, where you do not want the pin names to be moved in the Schematic for some Parts only. These options are defined in the **Options** dialog under the **General** tab.



You have four options as follows:

Never - No change to the existing way in which it works.

Always - All pin swaps are applied by swapping connections.

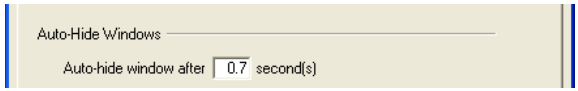
Positive - All pin swaps in swap groups that have a positive id will swap connections.

Negative - All pin swaps in swap groups that have a negative id will swap connections.

Note: Gate Swaps are not affected and will continue to exchange gate id and pin numbers.

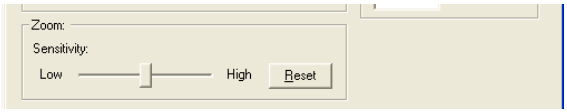
Auto-Hide delay

There is now an option to set the delay period before an auto-hide window slides out of view. This can be found on the **Options** tab of the **Customise** dialog. A delay of between 0.1 and 60 seconds may be specified. By default, this is set to 0.7 seconds.



Zoom Factor Control

You can control the zoom factor for the zoom commands (mouse wheel and shortcut keys). This is available on the **Options** dialog and **General** tab. Changing the slider will make the zoom factor finer or coarser as preferred.

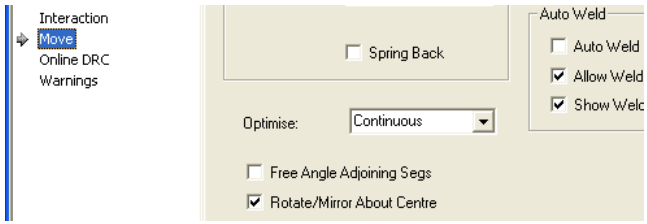


New Command in Move

There is a new command **Reset Delta Coords** available during **Move**. Use this to reset the ***dx*** and ***dy*** on the status bar to 0,0 whilst moving an item. This should be added to a shortcut key to be used, it is not available on the context menu. Use this to position an item relative to a known position, or another item.

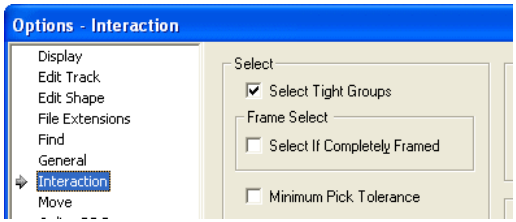
Move Option

This is a new option in the **Options** dialog, under **Move** options for PCB and **Interaction** options for Schematics, called **Rotate/Mirror About Centre**. If checked it works like it did in previous versions and rotates or mirrors about a gridded point at the selection centre. If unchecked, rotate and mirror work like they do in **Select** mode, about the **picked item origin**.



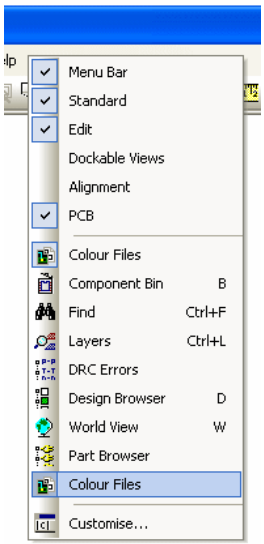
Select Improvement

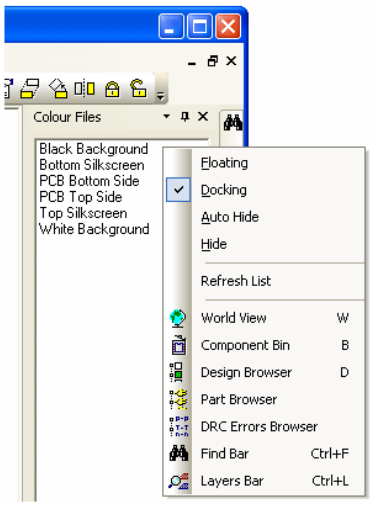
If you drag the left mouse button in select mode whilst holding down the **Alt** key, it will now always go into **Frame Select** whilst dragging. This enables you to frame select more easily in tight areas. This is an option on the **Options** dialog and **Interaction** tab.



Colour Files Bar

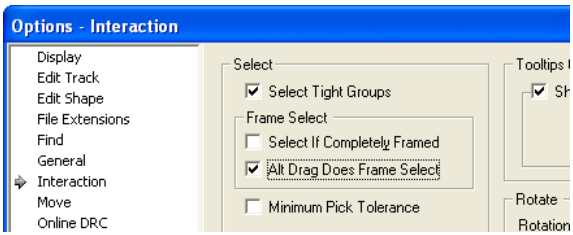
An additional dockable window showing a list of available colour files has been added. Simply clicking on a colour file name will apply it to the current design. The list of files will automatically update if new colour files are saved or the folder list changed. There is also a context menu option **Refresh List** to explicitly update the list (if, for example, colour files have been deleted via the Windows Explorer).





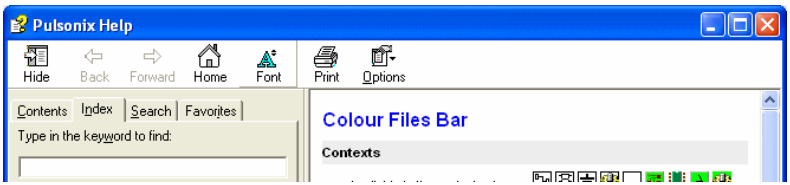
New Cross-Probe Option

A new check box on the **Options** dialog, **Interaction** tab for **Probe Any Design** has been added. With this checked, Cross Probe will probe to any open design. With it unchecked, the option will try and probe to the same named design of the opposite type, i.e. PCB to SCM or SCM to PCB.



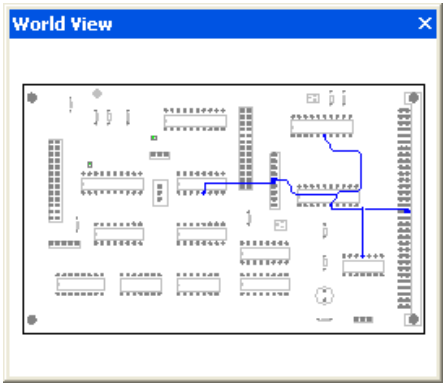
Font Size in HTML Help

It is now possible to select an alternative font size when using the Pulsonix Help Viewer. This is available by clicking the **Font** button on the toolbar.



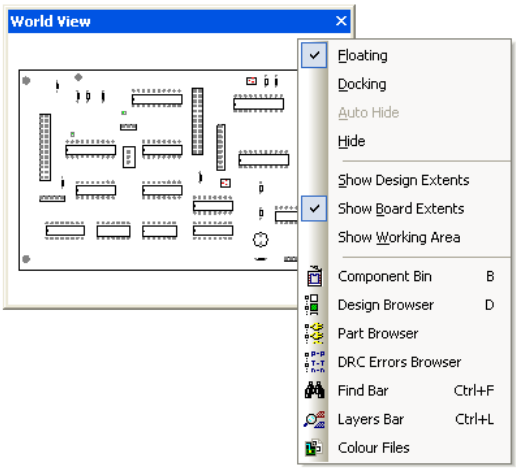
World View

The world view will now show the currently highlighted net. (Note: this is a net that has been located via the Find Bar and is different to a marked net). For this to operate, you must have **Select Found Item** switched off in the context menu for the **Find** Bar.



For PCB, the world view now also includes component outlines and consistently excludes vias.

The world View now includes a context menu option to show **Board Extents** as alternative to the Design Extents or Working Area. In SCM, this works the same as **Design Extents**. This is useful if you are using a drawing profile in your PCB design but wish the World View to focus on the board outline.

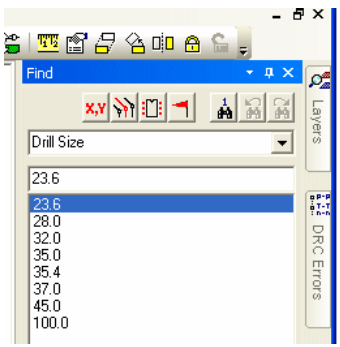


Changes to Find Browser

Drill Sizes

Find has a new category **Drill Size**. Selecting this will list the drill sizes in the design. Clicking on a drill size finds the first item that has this drill size (pad, mounting hole or via). Use the **Find Next** button to find next item with this drill size.

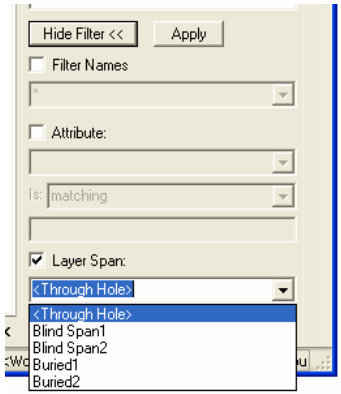
Note: The matching of an item's drill diameter to the chosen drill size will follow the same rules as in the drill table, i.e. it will use tolerances. This means that if a drill table says it has 3 items for a size, the find bar will find the 3 items.



Filter by Layer

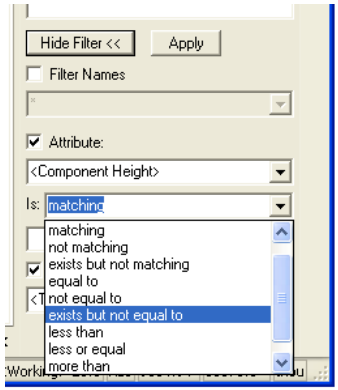
You can now filter by **Layer** or **Layer Span** for some categories. When available, a **Layer** checkbox will appear at the bottom of the filter controls in the bar. Click on the checkbox to use the filter and select the layer that items must have to be found. The item must have this layer to be found. For example, if the layer chosen is **Bottom Electrical** and the item is a through-hole pad, it will not be found.

Note: Via and Drill Size categories present a Layer Span list if you have more than one Span.



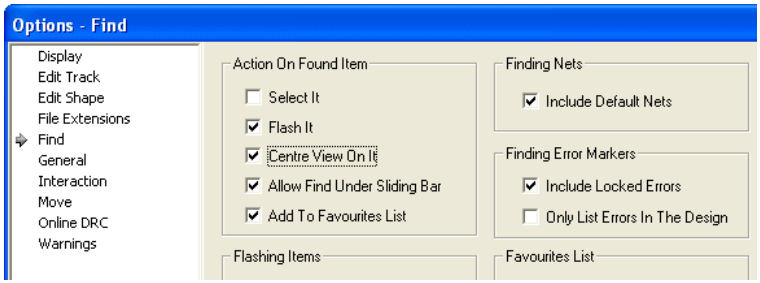
Filter by Attribute changes

Filtering by attribute has changed to include two new operations in the **Is** combo list. These are **exists but not matching** and **exists but not equal**. These differ from the **not matching** and **not equal** operations in that the item must have the attribute to be found.



New Find option in Options dialog

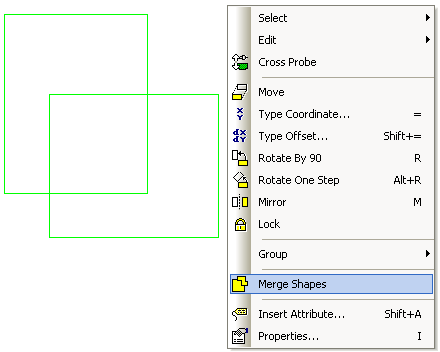
There is a new option on the **Find** page of the **Options** dialog - **Allow Find Under Sliding Bar**. This allows find items to be under the Find bar when it is in Auto Hide mode and not pinned. If unchecked, the item under the bar will be moved to the centre of the view. This option only has any effect if the option **Centre View On Item** is unchecked.



Merge Shapes

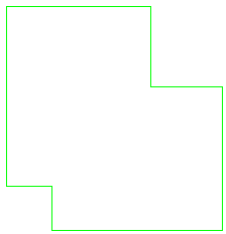
Merge Existing Shapes

You can merge a set of selected shapes using the **Merge Shapes** command (also see **Insert Merge** below).



Add two shapes to the design.

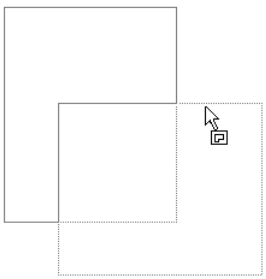
Select both shapes and select **Merge Shapes** from the context menu.



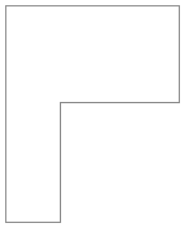
The two shapes are merged as one.

Add Cutout

Adding a cutout now removes the cutout shape from the outline, if they intersect.



Once the shape is drawn, select Insert Cutout. Select the shape. Draw the cutout starting outside the shape. Draw ‘across’ the initial shape and finish.



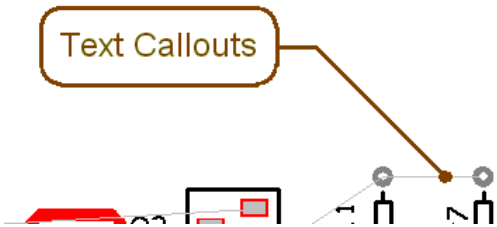
The resulting shape is now minus the shape ‘cutout’.

Insert Merge

A new set of commands allow you to add a new shape to be merged with an existing shape. The function is similar to Add Cutout, except the shape is added rather than removed.

Text Callouts

There is a new composite design item, Text Callout, in both PCB and Schematic designs used for documenting points of interest.

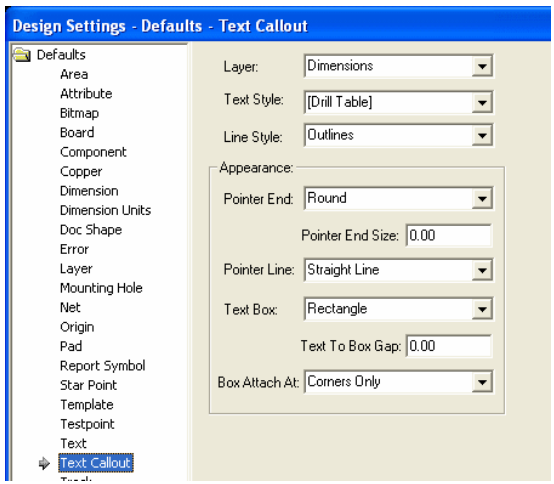


Text callouts consist of a text item, a shape around the text (the text box), a line from the text to the location of interest (the pointer), and a shape at the end of the line (the pointer end). These items are invisibly grouped so that it acts as one dynamic item in the design.

Using Text Callouts

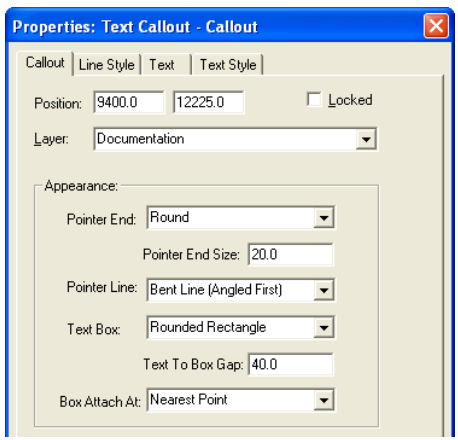
From the **Insert** menu, use **Insert Text Callout**, select the pointer end position, you will be presented with a dialog to supply the text and layer (in PCB design). The layer can only be a non-electrical layer. The callout is displayed on the screen and you can place the text item at the required position.

The defaults for the styles, layer and appearance of new callouts are defined in the **Design Settings** dialog on the Setup menu. You can override this by selecting a **Text Callout** or **Free Text** item in the design before using the **Insert Text Callout** option.

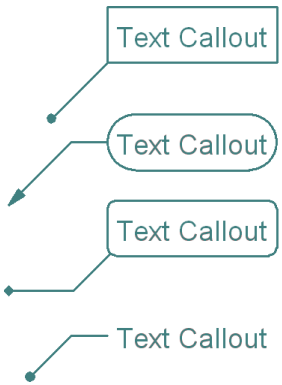


Once added, the text callout can be moved for repositioning by selecting the outer edge and dragging it. The text box alone can be repositioned by selecting just the text. You can also drag the Pointer End to a different position.

You can change the text callout appearance by selecting it and using the **Properties** dialog from the context menu. You can change the **Appearance** of each of the entities.

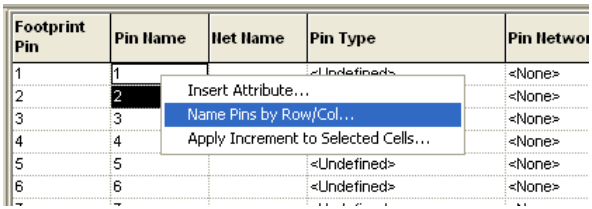


Different styles of text callouts can be achieved by changing the **Properties** of these items.

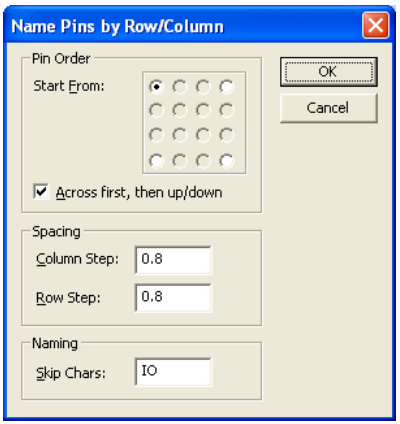


Part Editor Improvements

When editing Parts which use grid based components such as BGA footprints, this dialog helps you to assign pin names automatically.



The dialog presented allows you to define how the pads are named.



Pin Order

These controls allow you to choose the direction in which the pins will be named. The settings you need to choose here will depend on the order in which you have numbered the pins in your PCB symbol. Choose which corner you

want to start from, and the direction to go from that corner. With the selection shown above, the bottom left pin would be A1, the pin to its immediate right would be A2, and so on across the bottom row, then the leftmost pin of row 2 would be B2.

Spacing

These controls allow you to specify the step between the pads that constitute each row and column of your PCB symbol. The initial settings will be automatically determined by examining the arrangement of pads on your symbol, but depending on the order in which the pads were created you may need to adjust these values.

Naming

This control allows you to specify particular characters that will be skipped over when creating the pin names.

The results

Once you have set up the dialog controls and clicked OK, the pin names will be automatically created and displayed in the various pages and views of the parts editor.

The first pin will be named A1, the second A2, and so on until the end of the row or column is reached. The next row or column will start with B1, and so on until all the pins are named.

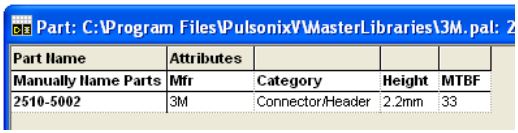
Where there is a break in the sequence of pins, the pin names for the missing pins are skipped and resume with the pin name appropriate to the row/column that the pins re-commence.

Part Editor Attributes

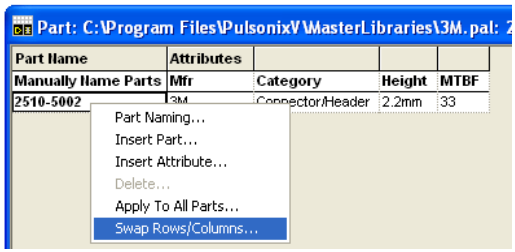
Swap Rows/Columns

In the Part editor you can swap rows/columns in the attribute view using the **Swap Rows/Columns** option from the context menu, this makes it easier to see Attribute Values when you have a lot of attributes.

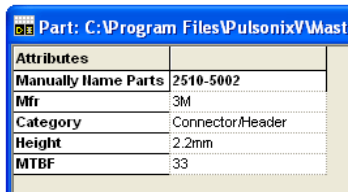
Take this example of a number of attributes for this Part:



Select the **Swap Rows/Columns** option:



The Part view now looks like this:

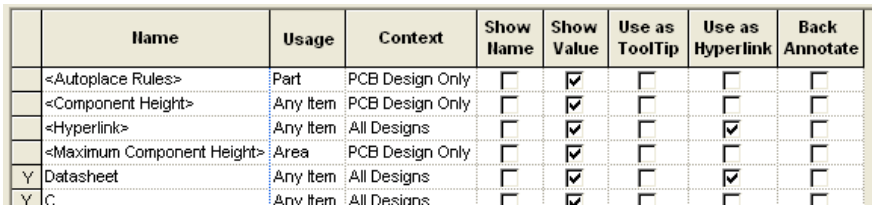


Changes to Hyperlinks

Hyperlink Attributes

Any attribute can now be a hyperlink; this means that you can have more than one hyperlink per item. To set attribute as being a hyperlink, check it's checkbox in the **Attributes** tab of the Technology dialog.

The <Hyperlink> attribute is now redundant, but is still present for items which already use it.



	Name	Usage	Context	Show Name	Show Value	Use as ToolTip	Use as Hyperlink	Back Annotate
	<Autoplace Rules>	Part	PCB Design Only	☐	☑	☐	☐	☐
	<Component Height>	Any Item	PCB Design Only	☐	☑	☐	☐	☐
	<Hyperlink>	Any Item	All Designs	☐	☑	☐	☑	☐
	<Maximum Component Height>	Area	PCB Design Only	☐	☑	☐	☐	☐
Y	Datasheet	Any Item	All Designs	☐	☑	☐	☑	☐
Y	C	Any Item	All Designs	☐	☑	☐	☐	☐

Execute Hyperlink

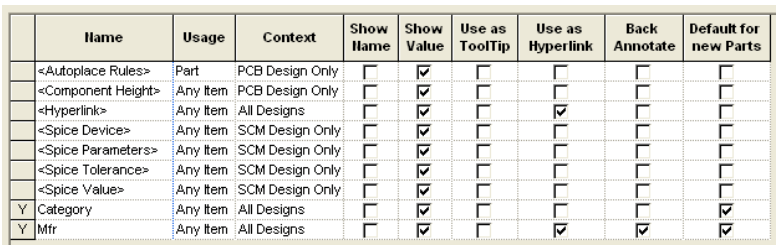
You can now execute any text item (using the shortcut for **Execute Hyperlink**). For example, plain text with the word **Excel** will run the Microsoft Excel spreadsheet program. If the text string includes a path and file extension, such as C:\temp\bob.xls it will run the spreadsheet ***bob*** in Excel (or whatever program is associated with the .xls extension).

If Pulsonix thinks text is a hyperlink it puts the **Execute Hyperlink** command on the context menu as well and will also activate the link on double-clicking.

When executing hyperlinks that don't contain a specific path, for example the text name ***Job99.pcb***, the current directory is set by the relative path of the design that the hyperlink is executed from. If job99.pcb didn't exist in the same folder as the current design, the hyperlink wouldn't run.

Default Part Attributes

In the Part editor, you can set a flag on an attribute to make it a default attribute for new Parts. This status would be saved in the Part Technology file for use when you create a new Part. These unsatisfied attribute names are added to the new Part ready for you to add a value to it.



	Name	Usage	Context	Show Name	Show Value	Use as ToolTip	Use as Hyperlink	Back Annotate	Default for new Parts
	<Autoplace Rules>	Part	PCB Design Only	☐	☑	☐	☐	☐	☐
	<Component Height>	Any Item	PCB Design Only	☐	☑	☐	☐	☐	☐
	<Hyperlink>	Any Item	All Designs	☐	☑	☐	☑	☐	☐
	<Spice Device>	Any Item	SCM Design Only	☐	☑	☐	☐	☐	☐
	<Spice Parameters>	Any Item	SCM Design Only	☐	☑	☐	☐	☐	☐
	<Spice Tolerance>	Any Item	SCM Design Only	☐	☑	☐	☐	☐	☐
	<Spice Value>	Any Item	SCM Design Only	☐	☑	☐	☐	☐	☐
Y	Category	Any Item	All Designs	☐	☑	☐	☐	☐	☑
Y	Mfr	Any Item	All Designs	☐	☑	☐	☑	☑	☑

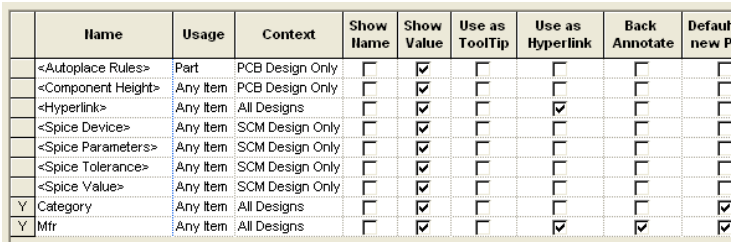
part *

Part Name	Attributes	
Manually Name Parts	Mfr	Category
12345		

When added, in this example, the Mfr and Category attribute names are now added ready to have values assigned to them.

Attributes can be back annotated

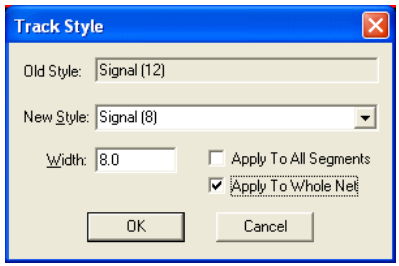
You can now mark an attribute as **Back Annotated**, the values for Design, Components, Nets and Pads are then passed back during back annotation. These are marked in the **Technology** on the **Attributes** page. Effectively, this allows attribute values to be edited in the PCB and back annotated to the master schematic design.



	Name	Usage	Context	Show Name	Show Value	Use as ToolTip	Use as Hyperlink	Back Annotate	Defaul new P
	<Autoplace Rules>	Part	PCB Design Only	☐	☑	☐	☐	☐	☐
	<Component Height>	Any Item	PCB Design Only	☐	☑	☐	☐	☐	☐
	<Hyperlink>	Any Item	All Designs	☐	☑	☐	☑	☐	☐
	<Spice Device>	Any Item	SCM Design Only	☐	☑	☐	☐	☐	☐
	<Spice Parameters>	Any Item	SCM Design Only	☐	☑	☐	☐	☐	☐
	<Spice Tolerance>	Any Item	SCM Design Only	☐	☑	☐	☐	☐	☐
	<Spice Value>	Any Item	SCM Design Only	☐	☑	☐	☐	☐	☐
Y	Category	Any Item	All Designs	☐	☑	☐	☐	☐	☑
Y	Mfr	Any Item	All Designs	☐	☑	☐	☑	☑	☑

Change Style Apply to Whole Net

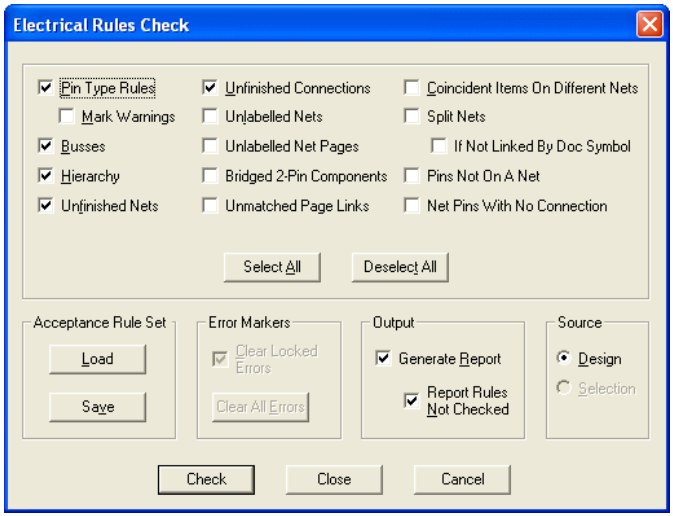
A new option was introduced late in version 4.0 and has now been incorporated into version 4.5. When changing a Track Style, there is a check box to allow you to **Apply To Whole Net** as an option when making the style change.



Chapter 3. Schematic Options

New ERC Checks

New Electrical Rules Checks have been introduced to version 4.5.



Bridged 2-Pin Components

Check this box to verify that every two pin component has its pins on different nets.

Coincident Items On Different Nets

Use this to check that items that are not on the same net do not lie on top of each other. It will check every connection and bus corner, and every pin. Connections on different nets are allowed to cross each other as long as a corner on one does not lie on the other. Pins are checked against all connection and bus segments and against all other pins.

Note: On all but small designs it is a good idea to have the **Fast Locate** option switched on in the **Display Options** dialog.

Split Nets

Use this to add an error marker on all nets that are split into separated subnets. These may be valid, but this allows you to quickly check if a net has been split by error.
Use the **If Not Linked By Doc Symbol** check box to not mark a subnet as being split if it contains a documentation symbol of any type. It assumes that the documentation symbol is an off page reference or signal reference.

Pins Not On A Net

Use this to add a marker on all pins that are unused and have not been added to a net. It will also add markers on a component if its ungated pins are not on a net. Note: Pins with pin type **No Connect** will not be marked.

Net Pins With No Connection

It is possible to select a pin and add it to a net without adding a connection to it. These are not obvious in the design and can now be found using this check box. Use this along with **Pins Not On A Net** to mark all pins that do not have a connection attached in the design.

Unmatched Page Link Check

A new check **Unmatched Page Link** has been added. This creates a **UP** error marker on net nodes that have a page link that is not matched on the page it links to. For example, if net "A1" on page "Page1" has a page link to "Page2", then it is an error if the net "A1" on page "Page2" does not have a link back to page "Page1".

Additional Error Codes

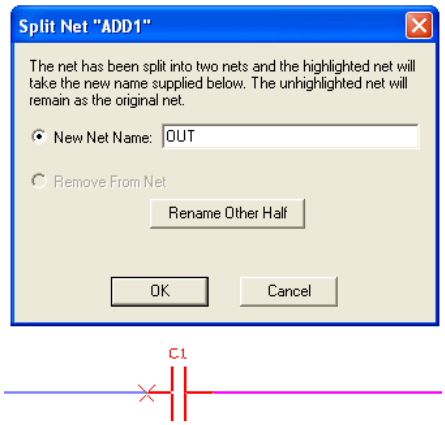
Another ERC change for 4.5 is that error codes have been changed to all be relevant 2 letter codes. Most have changed.

Split Net

The **Split Nets** option is now available in schematic designs. The only difference to PCB is that if pins would become isolated (i.e. they are the only item on the net) after the split then you will be warned that these pads will be removed from the net.

Split Net highlights nets now, rather than marking them. This has the advantage of seeing invisible items on the net.

In schematic, dragging a 2 pin component and dropping onto an existing connection will now display the Split Net dialog. This allows you to choose which net retains the original name and to be able to supply a net name for the split half.

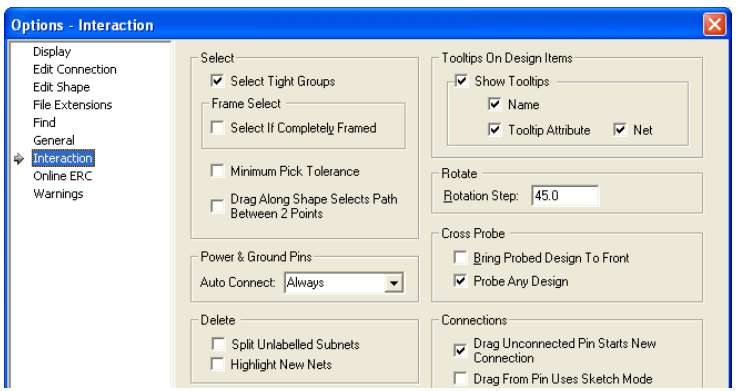


If you select a connection or track and it does not split the net, you now get the chance to OK the deletion question so that as a minimum it deletes the connection.

Delete

You now have the two new options for **Delete** on the **Options** dialog and **Interaction** tab.

Split Unlabelled Subnets. If this is checked, after using Delete, any new subnets that have been created and are unlabelled will be split from their old net to form new unnamed nets. A Warning will be given allowing you to cancel the delete if required. Leave the option unchecked to leave the subnets on the same net, (this is how it worked in previous versions).

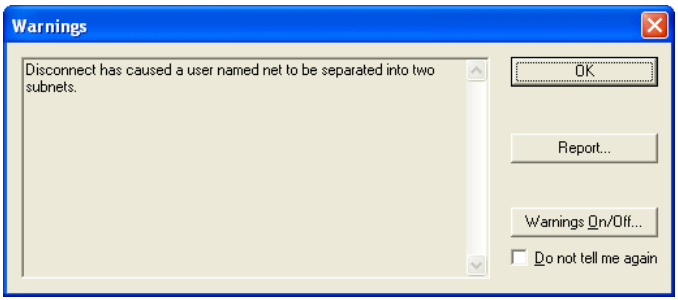


Note: An unlabelled subnet is one that does not display its net name and does not have a signal reference documentation symbol with a predefined net type or name. This is as defined in the ERC check.

Highlight New Nets - Check this to highlight any new nets created by the deletion.

Disconnect Con End / Remove From Net

When using the **Disconnect Connection End** and **Remove From Net** option, these options will now warn you if you separate a user named net into subnets. Default nets will be automatically split into new nets.



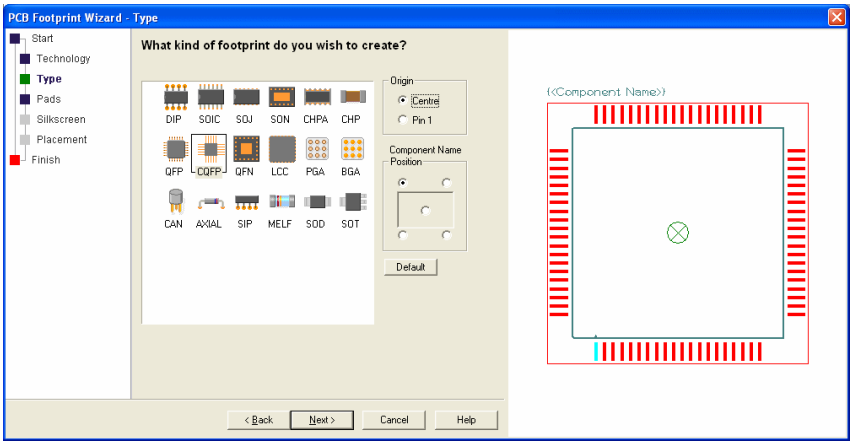
Next/Previous Commands for Insert Doc Symbols

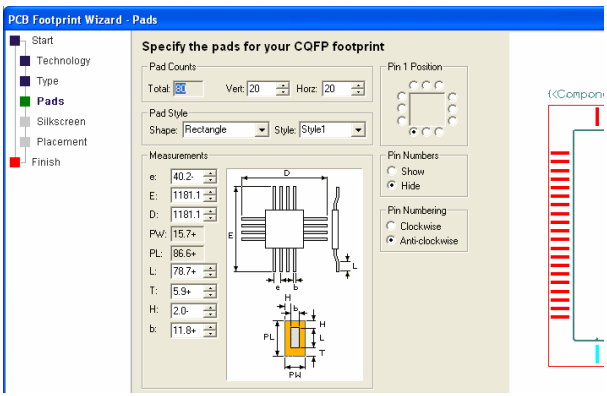
You can now use the commands found on the context menu **Next Symbol** and **Previous Symbol** (and the shortcut key commands) to cycle through next and previous Doc Symbols when using the **Insert Doc Symbol**, **Insert Sig Ref** and **Insert Page Link** options. This saves you having to go back to the Insert Doc Symbol dialog to select the next item in the list. The Next/Previous Symbol command will cycle through the Doc Symbol list and will use filtered symbols if a filter has been applied.

Chapter 4. PCB Options

Enhanced PCB Footprint Wizard

The existing footprint wizard has been significantly enhanced to give you more pre-defined shapes to select from and on the Pads page it now shows the actual dimensions. The dimensions now more closely follow the dimension formats used in the manufacturers datasheets.



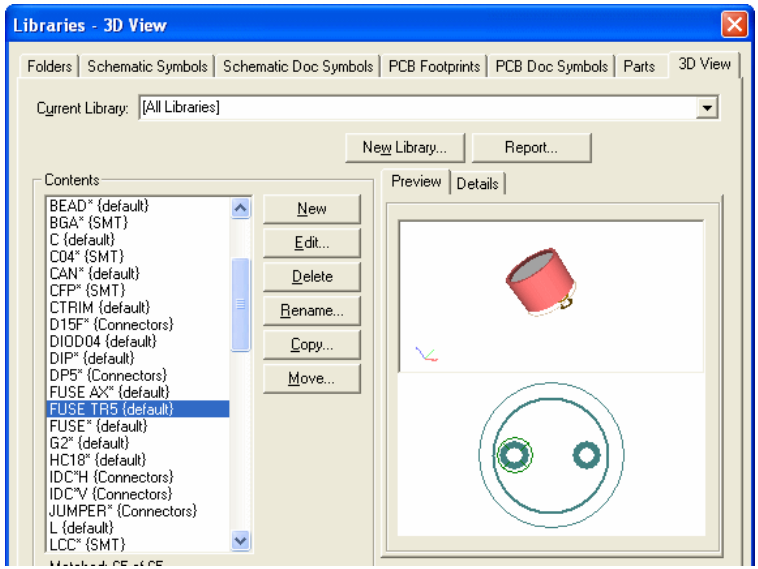


The Preview of the footprint being created has been moved to the right side of the dialog and has been significantly increased in size to make the creation of fine pitch devices much easier.

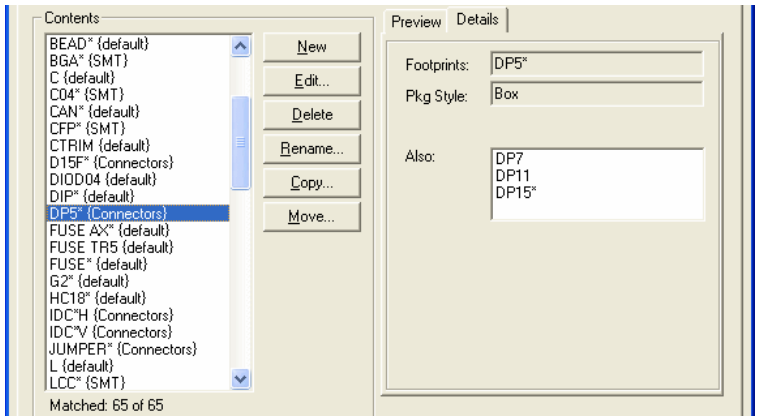
Changes to the Library Manager

There is a new tab (**3D View**) on the Library Manager dialog to support the new 3D Viewer option. The editing of 3D packages and their use is discussed below. Package libraries are saved with the .pkg file extension.

The **Preview** window displays the 3D package view and the 'flat' Footprint library view.

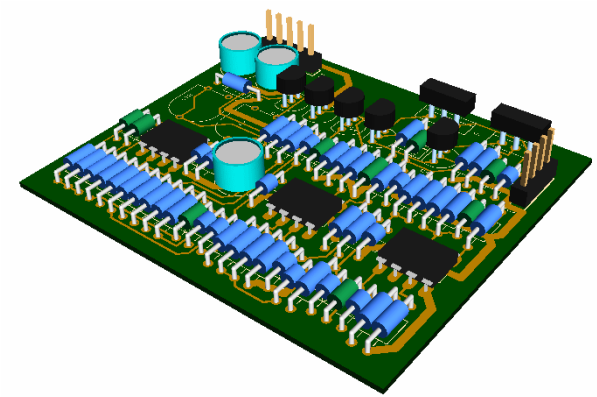


The **Details** tab switches to a detail summary of the package. It shows the Footprints defined for the package, the package style and the list of other Footprints also using this definition.



3D Viewer

The 3D viewer allows the ‘flat’ view of the PCB to be visualised as it would be when populated with components after assembly. A new menu option **3D View** on the **View** menu in the PCB editor is used to display the current design into the viewer. The appearance of components in the 3D view is controlled using a new library type of 3D packages. These can be edited as new footprints are added to your library.



Printing

Once in the **3D View**, you can use the **Print** option from the **File** menu.

Zoom in the 3D View

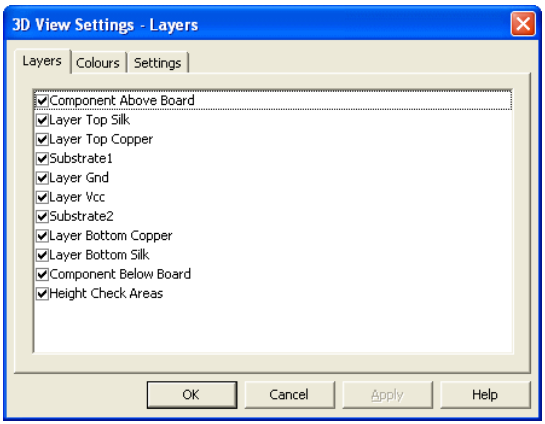
As well as the normal zoom controls on the **View** menu, the mouse wheel works in the same way in the 3D View as it does in the design editor. If you single-click the wheel, a pan is executed. If you hold the wheel down you can pan around the 3D view.

If you hold the right-mouse button down the 3D View picture will spin continuously.

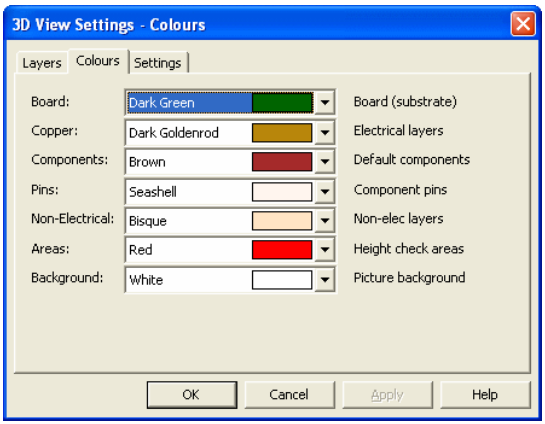
3D Settings

The **3D Settings** dialog is available on the **Setup** menu. The three tabs within it allow you to define how the 3D view will look.

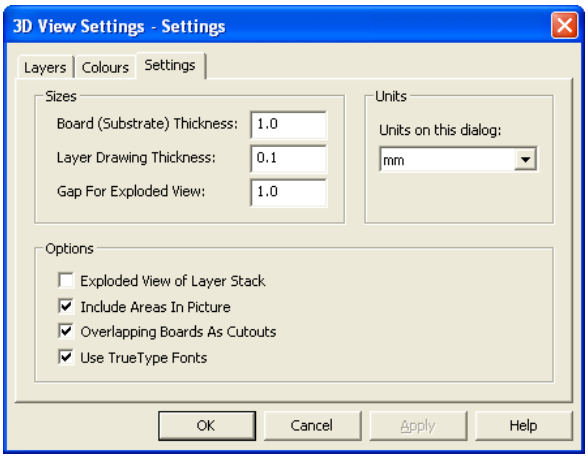
The **Layers** tab will enable you to switch on and off the design layers within the view.



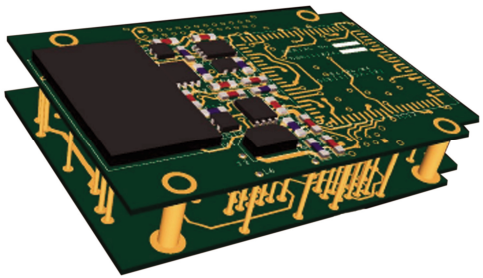
The **Colours** tab allows you to setup preferred colours for the 3D view.



The **Settings** tab is used to define view parameters such as the thickness of the board in the 3D view.



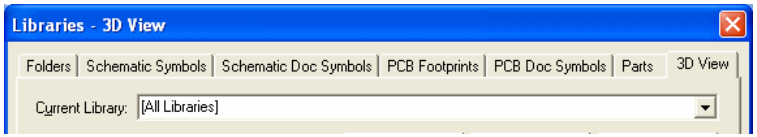
Each of the entries for **Sizes** defines a thickness that can be created and viewed. You may change the **Gap** thickness to show internal layers more clearly, perhaps to show the internal via stacks.



Defining the 3D Packages

From the **Library Manager**, the **3D View** tab enables packages to be edited.

In order to view components in the 3D viewer, you must define the package 'shape' in the library. Use the new **3D View** tab in the **Library Manager** to define package libraries.



The package libraries have the file extension .pkg. The contents of the package libraries have definitions which relate to a footprint or groups of footprints, for example DIP*.

Package Overview

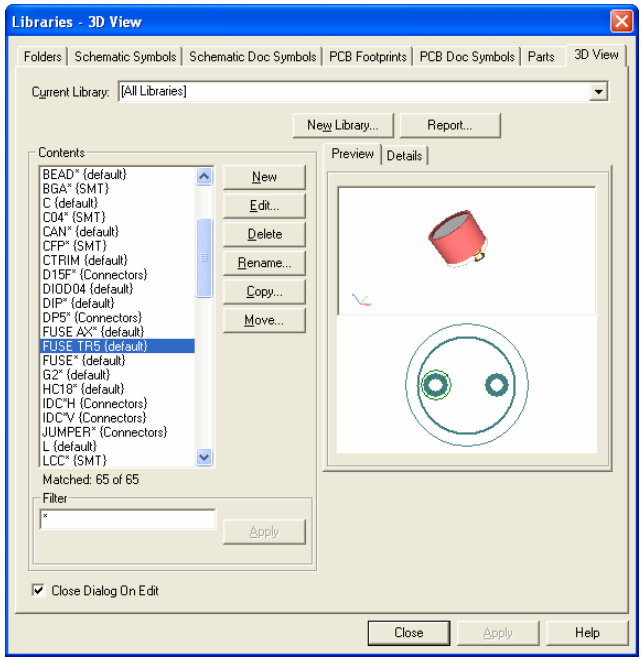
3D Packages define how footprints are represented in 3D View images of your PCB designs. Each 3D Package has a 'style' (DIL, Quad, Axial, and so on), and various settings that control how the 3D image is generated. It also has one or more names that specify which footprints the 3D Package applies to, allowing one package to be used to generate the 3D image for many PCB footprints. These names can be specific footprint names, or they can include 'wildcards' to match multiple footprint names.

To generate the 3D view of the actual components on your PCB design, the set of package libraries is scanned to find the first package that matches each of the footprints used in the design. Because each package can have a basic name and a set of 'aliases' both of which can include wildcards, it is possible to create a few packages that can auto-generate the 3D view for many PCB symbols.

If no package is found, a default representation of the footprint is generated using type ‘Shape’ with the components colour from the Settings dialog.

3D View Dialog

The 3D View tab gives a **Preview** of the 3D package and 'normal' view of a corresponding footprint for the selected item. Even in this view, the picture can be rotated in its 3D axis. The **Details** summary shows the Footprint, shape and alias for the selected package. The footprint that is used for this dialog will be the first one found that matches the rules defined by the selected package.



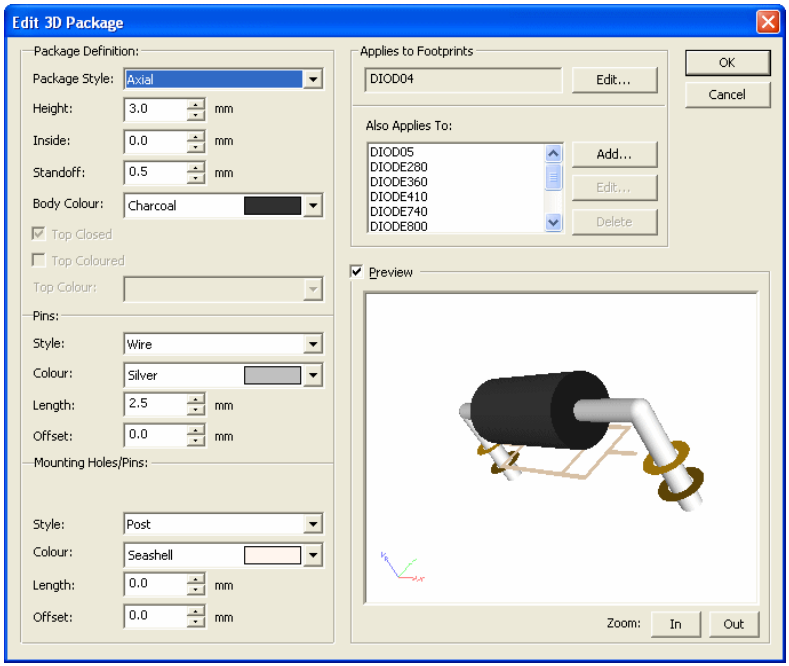
Edit 3D Package

This **Edit** button is used to edit the selected 3D package. Fields on the dialog allow you to specify which PCB footprints this package would be used for, and the style and other properties of the package. Only one package at a time can be edited

Using the ‘Edit 3D Package’ dialog

Choose the package you wish to edit, and click the **Edit** button. The dialog is divided into several sections, each of which is described below. Depending on the type of package, and other factors like the number of pins, some sections or controls may not be available or appropriate, or may not have any effect on the

resulting picture. The best way to see the effects is to try them out and see how the **Preview** picture changes.



Package Style: The main control in this section, which specifies the general type of the package. Available styles are:

Axial	Generally two-pin through-hole components like resistors and diodes.	
Bead	Rounded components like LEDS use this style.	
Box	Rectangular components where the box is taken from the overall size of the component.	
Can	Vertical cylindrical components such as large capacitors.	
Chip	Surface-mount chip-type components such as chip resistors and capacitors.	
DIL	Dual-in-line components such as DILs, SOICs.	

Quad	Components (generally surface mount) with pins on all four sides.	
Shape	General components where the 3D body shape is derived from the 'best fit' shape in the PCB footprint.	

Height: the overall height of the component body. In the 3D PCB picture this value is a default and only used if the component does not have a height defined.

Inside: for some types, this specifies the amount by which the body should be undersized.

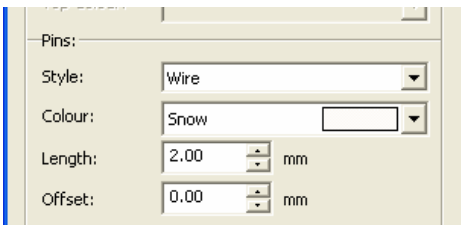
Standoff: the distance between the board surface and the underside of the component body.

Body Colour: the colour of the component body.

Top Closed/Coloured/Colour: Some packages styles allow the top of the body to be open or closed, and/or set to a different colour to the rest of the body.

Pins

This section specifies how the pins should look.



Style: Type of pin. Some of the pin styles are sized and positioned to fit particular styles of package. For example, J-Lead, Gull-wing and DIL pins will only work for package styles such as DIL and Quad where the pins are around the edge of the component.

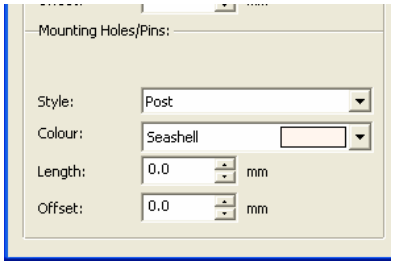
Colour: Pin colour

Length: Overall length of pin. This does not have any effect for some types such as J-Lead and DIL which measure the pin length automatically from the midline of the component body.

Offset: Allows the pin to be offset up or down the component.

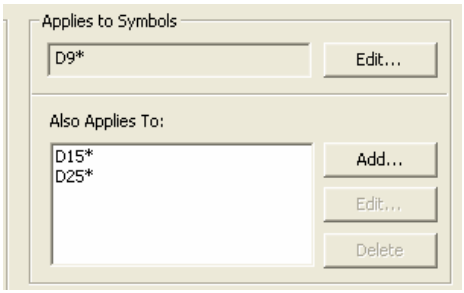
Mounting Holes/Pins

Same as pins, but for mounting holes on the footprint.



Applies To Footprints

This section specifies the footprint names to which this package will apply. The first name is the name by which the package will be listed in the Libraries dialog. The other list shows additional names to which this package will match. Any of these names may contain wildcards, as shown in the example below.



Preview

This section shows a preview of the 3D package you are editing, as long as an appropriate footprint can be located in the available libraries. Just below the preview is shown the name of the footprint that is being used.

You can see the effect of changing any of the controls on the dialog straight away. The picture can be rotated in any direction (in the same way as for the main 3D view itself) by moving the cursor into the preview window, pressing and holding down the left mouse button and then moving the mouse. Buttons are also provided to allow you to zoom In and Out of the preview picture.

Changes to Define Pad Shape

Multiple Shapes During Editing

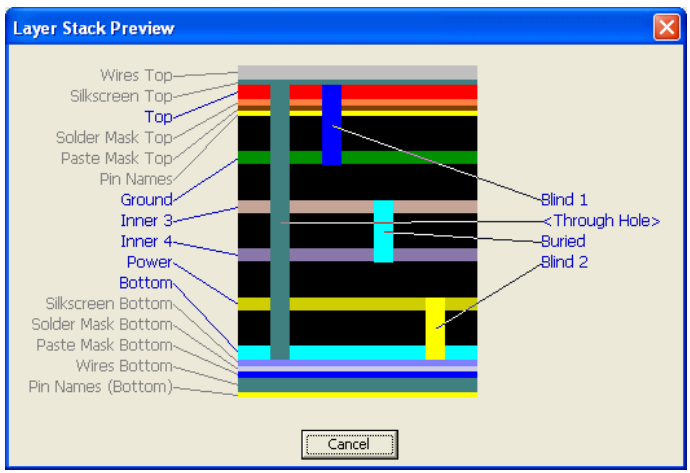
You can now have more than one shape during editing. This is subsequently restricted to only allowing a single shape when saving the pad shape. A warning message will be given if the pad contains more than one shape.

DXF Import

Using the **File** menu and **Import Design Data** option, you can import DXF into a pad shape. The same rules apply as above when saving multiple shapes.

Layer Stack Preview

On the **Technology** dialog, there is a new **View Layers** button on the **Layers** and **Layer Span** pages. This gives a simple view of the layer stack and any layer spans defined for through-hole and blind and buried via stacks.



All layers are displayed with electrical layers labels shown in a more prominent colour for easy identification.

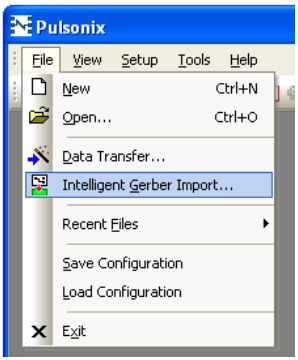
Intelligent Gerber Import

The **Intelligent Gerber Import** option allows you to rebuild a design by importing existing Gerber files, NC Drill files and a bill of materials (BOM). This differs from the existing Gerber Import option as the standard Gerber Importer is primarily only a viewer.

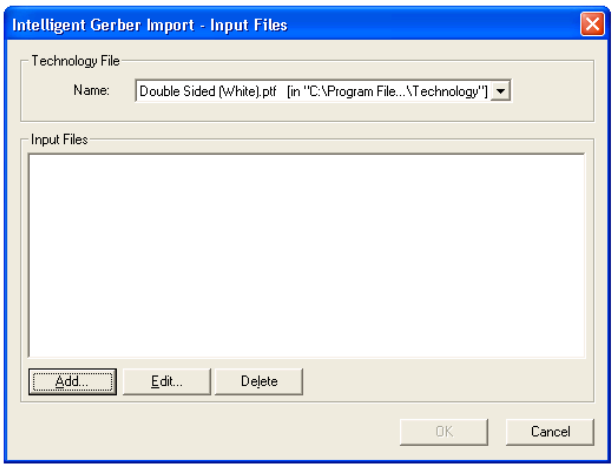
Once the files have been imported you should have a design with Free Pads connected by tracks. The Component Bin will contain any components defined in the Bill of Materials. Use the **Generate Component** tool to convert free pads and silkscreen shapes into components and symbols, and the **Free Pad to Via** tool to create vias.

▶ To start importing and rebuilding Gerber files

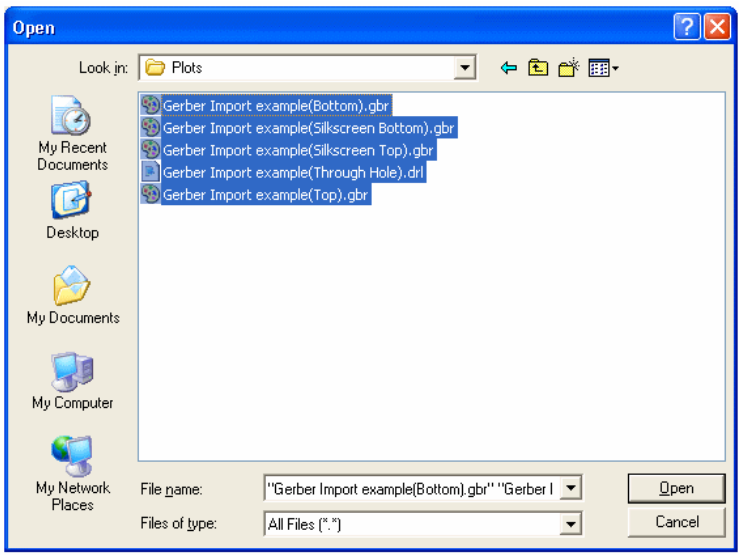
1. From the **File** menu, select the **Intelligent Gerber Import** option**.**
2. You do not need to have the PCB design editor running; you can select this option from the **File** menu with no designs open.



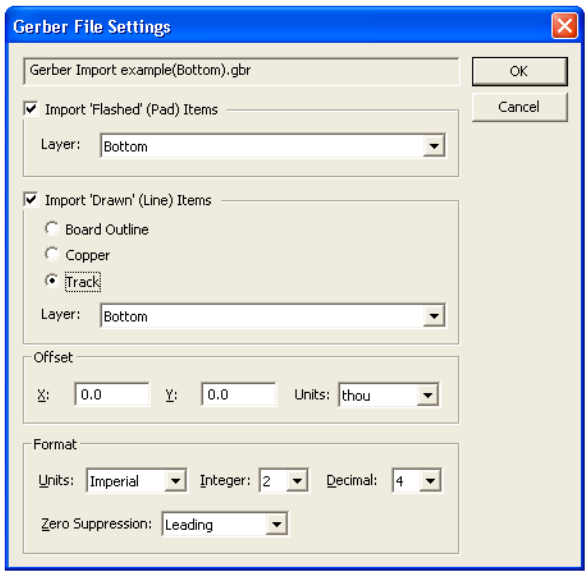
3. After selecting this option, a dialog is displayed:



4. **Technology files** are used during the import for layer selection and to pre-assign Spacings, Styles etc. from an existing file.
5. The **Add** button is used to select the files for import. The **Edit** button is used after the list has been populated and on selection of one of the files in the **Input Files** list.
6. Click the **Add** button. The **Open** dialog is displayed.



7. From the **Open** dialog you can select an individual file or multiple files using Ctrl, Shift and frame select.
8. Click the **Open** button to load the files ready for import.
9. For input files, as each file is being loaded, a settings dialog is displayed for the Gerber, NC Drill and bill of materials file types.
10. From these dialogs you can choose how the file will be imported.

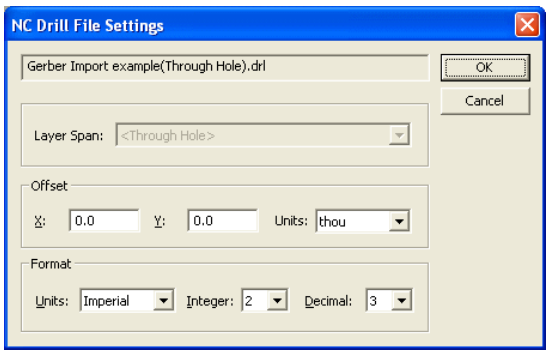


11. This dialog allows you to confirm the format of a Gerber file prior to it being imported. A number of Gerber files can be imported at the same time to rebuild a complete design.

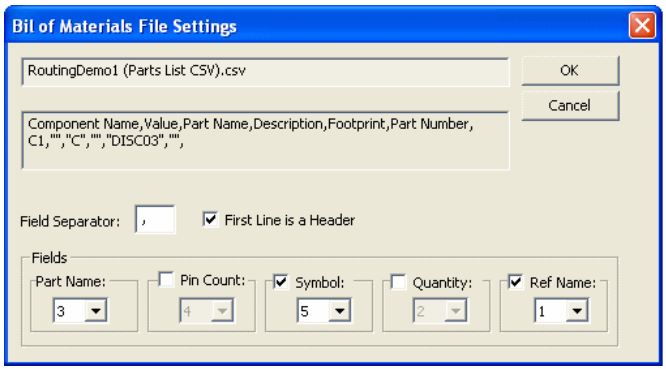
12. A Gerber file consists of line *draws* and shape *flashes* usually of specified sizes. If may also contain filled shapes. There is no electrical, connectivity, text or other information in the file. Typically for our purposes, *flashes* represent pads, and line draws represent tracks. You must decide which layer the *flashes* and *draws* are placed on. The program will attempt to rebuild the *draws* into shapes, and for tracks will create nets joining the pads. At this stage, all pads will be *free pads* which can subsequently be converted to Component Pads and Vias.

13. Other than changing the **Track Layer**, when importing multiple files, the Gerber File Settings option will select the best options for you.

14. The NC Drill Setup dialog allows you to confirm the format of an NC Drill file prior to it being imported. A drill file should be imported along with number of Gerber files, to rebuild a complete design.



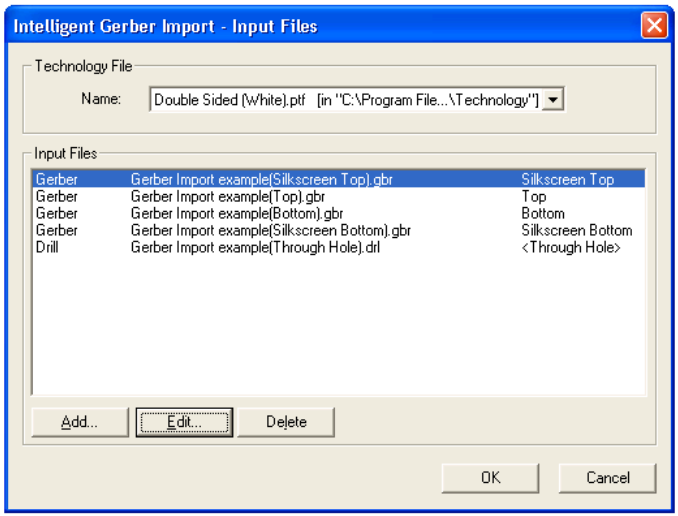
15. The **Bill Of Materials** dialog allows you to confirm the format of a Bill of Materials (Parts List) file prior to it being imported.



16. A Bill of Materials file should contain lines of delimited fields (often a comma separated file) containing a list of the components used in the design (one entry per line). Using this dialog, you can specify which fields represent the values needed. As the file is read, components are added to the **Component Bin**. If the components are defined in the library, they can be added from there, giving a complete definition. Otherwise, the components will be incomplete (no silkscreen shapes, no pins, etc). You can

subsequently complete their definition from shapes and pads in the design (usually imported from Gerber).

17. Once all the files have been selected for input, they appear in the list.
18. The **Delete** button can be used to remove selected files.



19. Once you have selected the data files, press **OK** to start the import process. Once the files have been imported you should have a design with Free Pads connected by tracks.
20. At this point, your design will contain tracks (with default net names) connected to free pads and free pads making up patterns which will represent components. It may also contain shapes which represent the silkscreen outlines on non-electrical layers.
21. The Component Bin will contain any components defined in the Bill of Materials. These components can be moved out of the bin and placed in the design. When placed over existing pads and shapes, they will auto weld themselves back in replacing what is already there.

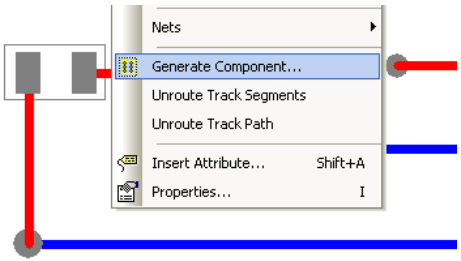
Generate Component

You can replace or recreate a component from its constituent free pads and doc shapes. This is particularly useful when used in conjunction with the **Intelligent Gerber Import** option.

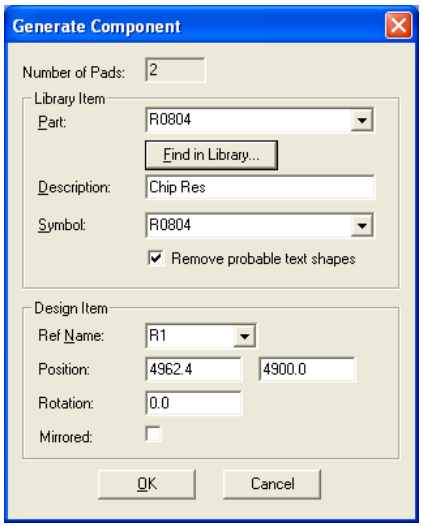
The process for using this is to first select pads in the design using **Frame Select** or **Ctrl/Shift** select. Don't worry if tracks are selected as they will be ignored. After clicking the right mouse button, select the **Generate Component** option from the shortcut menu.

▶ To rebuild Components

1. Use the **Generate Component** tool to convert free pads and silkscreen shapes into components and symbols (use the **Free Pad to Via** tool to create vias, see below).



2. This will display the **Generate Component** dialog:



3. You need to specify what the selected items will be replaced by. There are a number of scenarios.

- The component is already defined in the current design.
  > Select the Part name from the dropdown list. Only part with the same number of pins as you had selected are shown. The program will fill in the remaining fields and estimate the position and rotation from the selected pads.

- The component is already in the current design, but is undefined (such as when created using **Intelligent Gerber Import**).
  > Select the Symbol name from the dropdown list. The program will convert the selected documentation shapes into a symbol outline. All other instances using this component will share this definition.

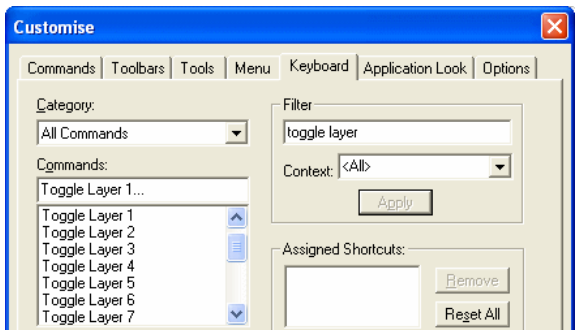
- The component is not defined in the current design and you wish to define it using the selected items.
  > Type the new Part name into the **Part** field. The program will convert the selected documentation shapes into a symbol outline.
- The component exists in a library but is not in the current design.
  > Use the **Find in Library** button to find the component. The program will load the component into the current design; it will also estimate the position and rotation from the selected pads.
- The component instance already exists in the Component Bin and you wish to replace the selected items with it.
  > Select the Reference Name from the list, the program will estimate the position and rotation and place the component. Alternatively, do not use the **Generate Component** option. Just drag the component instance and let **Auto Weld** connect the component.

Free Pad to Via

Select a pad and use the option from the context menu or, with nothing selected, run the command **Free Pad to Via** and from the context menu use the **Apply To All** option. Only free pads on the nets will be converted.

Layer Toggle

In version 4.5 a new mechanism has been provided for 16 new '**Toggle Layer n**' commands that may be used to toggle the display of particular layers on/off. By default, these commands are not present on any menu but are available via the **Customise** dialog, for assigning to shortcut keys (or toolbars/menus).

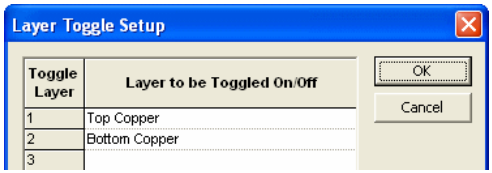


By default, a particular **Toggle Layer** command will toggle the correspond layer in the defined layer order. For example, assigning the shortcut key **Alt+3** to the command **Toggle Layer 3** via the **Keyboard** page of the Customise dialog, will mean that the third layer, as defined in the **Layers** page of the Technology dialog, will be toggled on or off whenever the Alt+3 key is pressed. The defined layer order is also shown in the **Layers Bar** if displayed.

It is possible however, to change the layer or layers to which a particular **Toggle Layer** command applies. Selecting **Set Toggle Layers** from the **Layers Bar** right click shortcut menu will display the **Layer Toggle Setup** dialog.

The Layer Toggle Setup dialog

This dialog shows all the available **Toggle Layer** commands along side a slot for specifying the layer to which each should apply.



Using **Set Toggle Layers** from the Context menu in the **Layers Bar** shows the **Layer Toggle Setup** dialog, Toggle Layer commands may be assigned to chosen layers by name or by characteristics. All Toggle Layer definitions will be saved and restored whenever the application is restarted and are independent of any particular PCB design.

An empty slot implies the default association with corresponding layer in the layer order. Typing a specific layer name next to a particular **Toggle Layer** command will cause it to apply to that layer. Continuing the example from earlier, if "Top Electrical" is entered next to the Toggle Layer command 3 then pressing **Alt+3** now will toggle the "Top Electrical" layer on or off.

All **Toggle Layer** definitions will be saved and restored whenever the application is restarted and are independent of any particular PCB design.

If layer names vary between designs it may be preferable to associate layers by characteristics rather than their explicit names. The format of recognised layer characteristics is:

<Layer Class> <Side> {<Index>}

where <Index> is optional.

So "Electrical Top" would always match the electrical layer on the top side regardless of whether it was named "Top", "Top Elec" or anything else, "Paste Mask Bottom" would always match the bottom paste mask layer if defined. Continuing again with the earlier example, if "Electrical Inner 2" is entered next to Toggle Layer 3, then pressing **Alt+3** now will toggle the second inner electrical layer in a PCB design regardless of what it is called.

A **Toggle Layer** command associated with a un-recognised layer will have no effect.

When a **Toggle Layer** command has been associated with a particular layer it's command name will change to appear as a command for toggling that layer rather than the default '**Toggle Layer n**' command. In the example given earlier

the command will appear as "Toggle Layer: Electrical Inner 2" rather than "Toggle Layer 3".

How to toggle multiple layers simultaneously

It is possible to specify more than one layer against a single **Toggle Layer** command so that a 'set' of layers are toggled when the command is used. Within the **Toggle Layer** entry, each layer definition should be separated by a comma. For example, Toggle Layer 3 could be extended to be "Electrical Inner, Electrical Inner 2" which would toggle both inner electrical layers simultaneously.

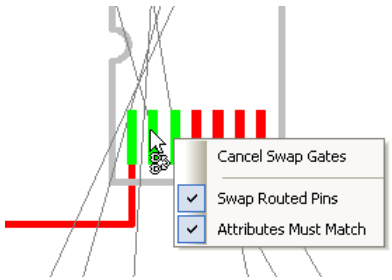
Switching Layers on and off simultaneously

Where multiple layers are specified for a single **Toggle Layer** command, it is possible to simultaneously turn some of them on and some of them off thus allowing 'sets' of layers to be visibly switched. To achieve this, any layer to be switched the opposite way to the first layer specified should be followed by the word **"off"**. For example, defining Toggle Layer 3 as "Electrical Top, Electrical Bottom, Electrical Inner off, Electrical Inner 2 off" will allow the top and bottom electrical layers to be made visible and at the same time, the two inner electrical layers to be made invisible. Using the Toggle Layer 3 command again will have the opposite effect.

Swap Routed Gates & Pins

Two new options exist when attempting to swap routed gates and pins.

The **Swap Routed Pins** option exists on the shortcut menu when using Swap Gates or Swap Pins. Check it to allow selecting of pins that have routes attached. The track segment attached to the pad will be unrouted prior to swapping.

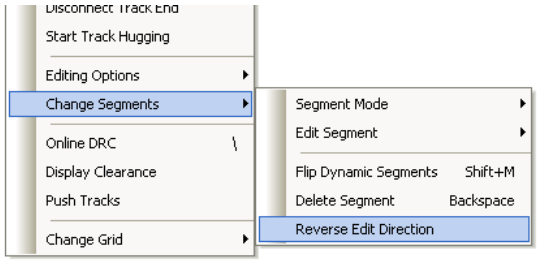


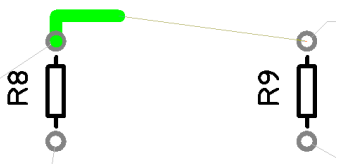
The **Attributes Must Match** option command exists on the shortcut menu when using Swap Gates. Check it to only swap gates between Components that have the same attributes (this is how version 4.0 worked), or leave unchecked to ignore attributes.

Edit Track Reverse Edit Direction

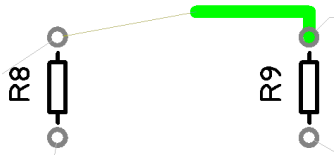
A new command, **Reverse Edit Direction** is now available when editing an unrouted connection to add a track. Selecting it will keep any track segments already added, throw away the dynamic segments and then edit the connection from the other 'end'. If the item at the other end of the connection is off the screen, the view will be panned to position it in the centre of the display.

The command is on the sub-menu under **Change Segments** > and **Reverse Edit Direction**





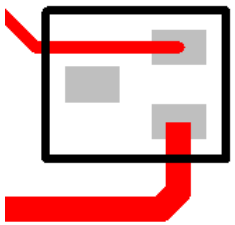
Start editing the connection this way.



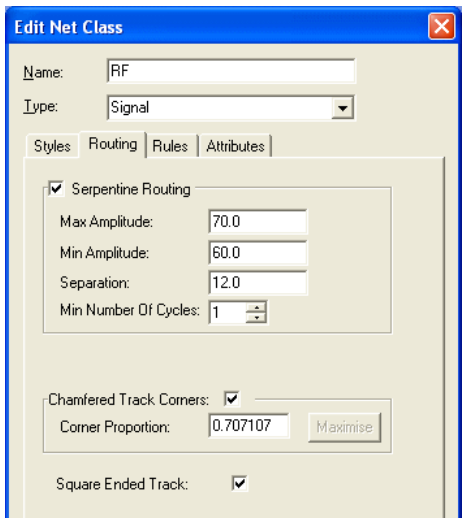
And after the **Reverse Edit Direction** command, the edit direction has been changed to the other end.

Square Ended Tracks

To further support the creation of RF designs, tracks can have square ends instead of rounded ends.

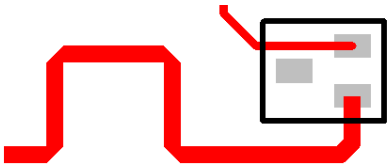


This is defined on a **Net Class** basis in the **Technology** dialog. For each net class that requires this track style, edit the **Net Class** and on the **Routing** page, select the **Square Ended Track** option. The square ends are checked correctly by DRC.

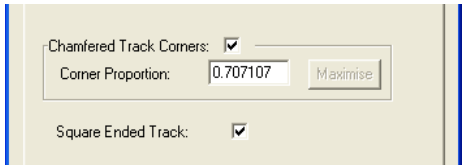


RF Mitres using Chamfered Track Corners

Another additional feature to support RF designs is the creation of RF Mitres. Track corners can be chamfered (45 degree outside corner and 90 degree inside corner).

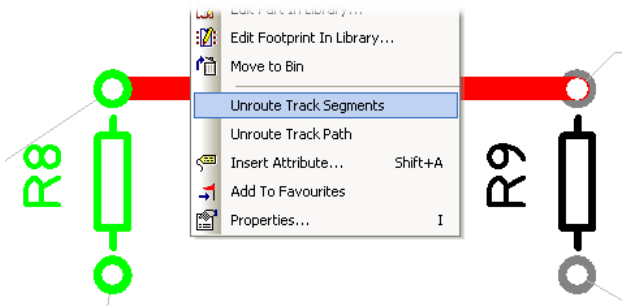


This is defined on a **Net Class** basis in the **Technology** dialog. For each net class that requires this track style, edit the **Net Class** and on the **Routing** page, select the **Chamfered Track Corners** option. The size of the chamfer can be changed using the proportion parameter on the dialog.



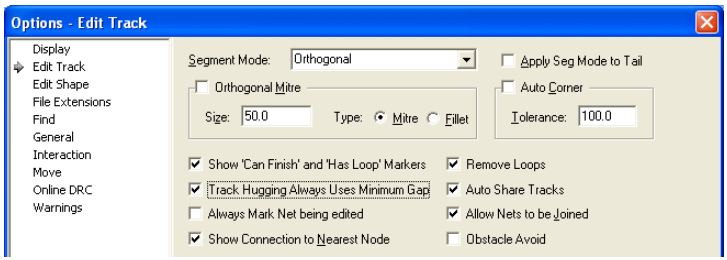
Unroute Tracks Change

The **Unroute Track Segments** and **Unroute Track Path** options are now available on the context menu when items are selected that have track segments attached to them, for example, a component, pad or copper items. The first option will unroute all track segments attached to the selected item(s), and the second option will unroute all track paths attached to the selected item(s).



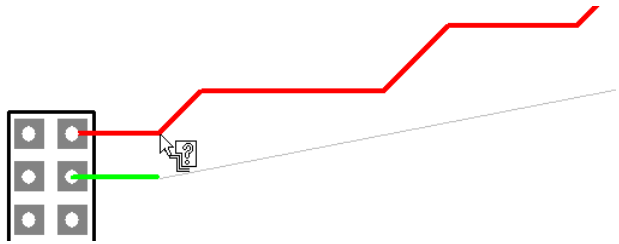
Track Hugging Improvement

Track hugging now allows you to hug at distances greater than the minimum design rule gap. There is an option in the **Options** dialog on the **Edit Track** tab called **Track Hugging Always Uses Minimum Gap**. You can switch this off to maintain the gap from the end of the dynamic track to the shape to be hugged. This switch can also be changed from the shortcut menu whilst in the **Edit Track** stage where you pick the shape or track to hug. Note: The cursor changes whilst in this stage to show a shape with a question mark on it indicating you have to pick a shape.

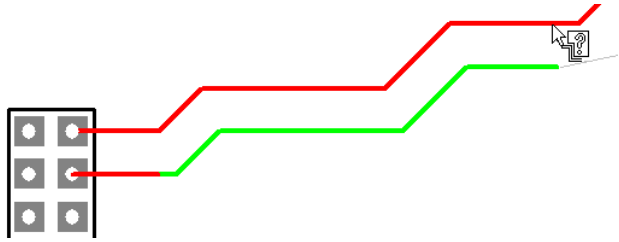


Track Hugging

By picking the ‘host’ track, an equi-space distance will be applied rather than maintaining the minimum track to track distance.

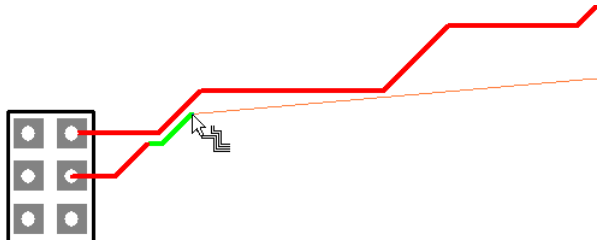


Once the host track is hugged, it follows an exact distance. This could be used where an exact parallel track path(s) is required.



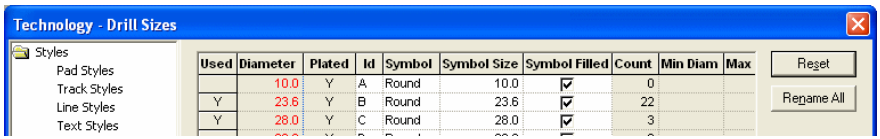
Alternative Method

The original alternative method is still available for use. Selecting the 'host' track in close proximity will enable the hugging mode using the minimum track to track spacing rule.



Drill Names

You can now sort the drills in the **Drill Sizes** dialog (in the **Technology** file and **CAM Plot** option) by clicking on the column header.

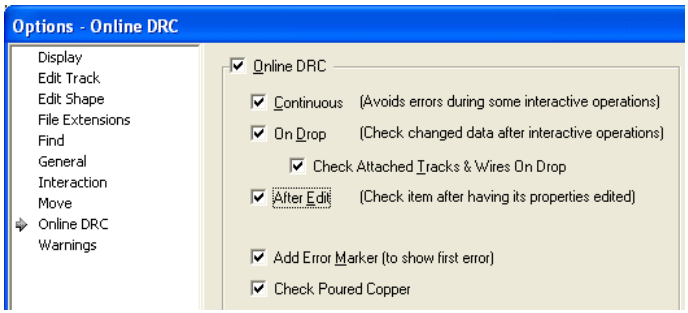


The **Rename All** button will regenerate the names of the drills in the order they appear in the dialog. In particular you can sort the drill by the used state, so consecutively naming the used drills.

DRC Changes

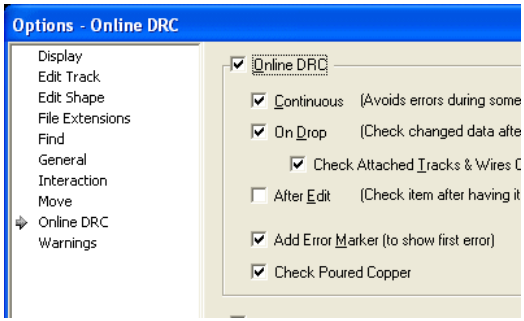
Online DRC Changes

The **Online DRC** checking done 'on drop' now includes Component checking against areas.



Online DRC Change in Options

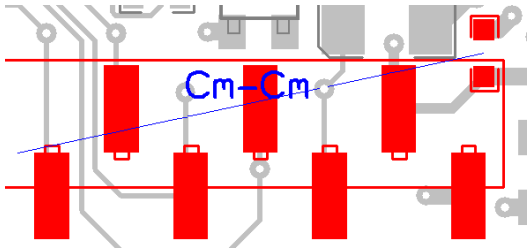
A new switch, **Check Poured Copper** in the **Options** dialog on the **Online DRC** page have been added in V4.0 build 2569 to allow you to force online DRC checking to ignore poured copper.



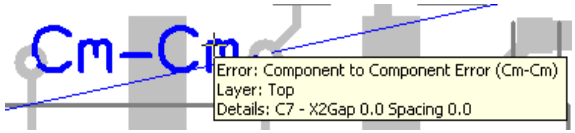
Use this when adding tracks that will be routed into a poured copper shape. This will stop the continuous track to copper errors being displayed. Re-pour the copper after the track has been added to clear the copper around the track.

Change to Component to Component Error Markers

Following a DRC check, Cm-Cm errors now have their start/end positions shown from the centre of the components involved, which then causes a tie-line to be drawn between the component centres.



Design Tooltips for Errors now show the "annotation" (e.g. C2-X3 for Cm-Cm errors) and the spacing and gap.

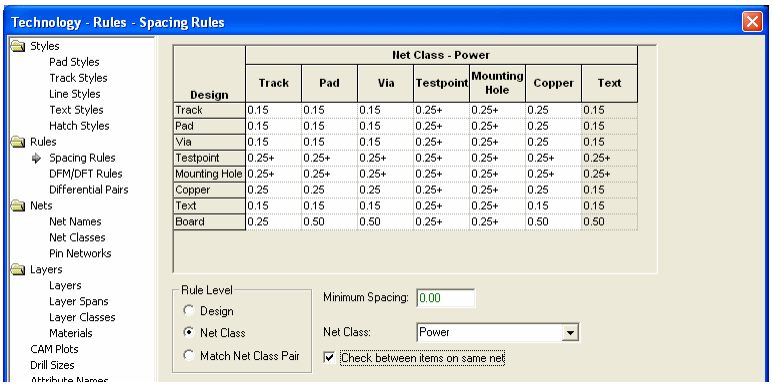


Check between item on same net

A new check had been added to the Design Rules Check. This allows the same net track to pad check. This means that it is an error for pads, tracks or vias on the same net to overlap. Only tracks which connect directly to a pad or via are allowed, and there should be no track corners within the pad shape.

A check box is available in the **Technology** dialog and **Spacing** tab for the **Net Class** rule level.

A check box is available to **Check between item on same net**.



New Track to Star point checks

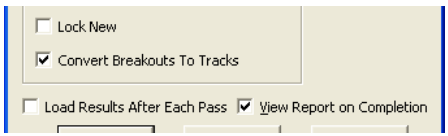
Track to star point checks now allows corners within the pad.

For example, this type of track edit:

This check doesn't allow re-entrant tracks within the pad also.

Router Dialog Change

The Router dialog now has a **View Report on Completion** checkbox. Check this so that the report is now displayed when the router has finished.



Moving Copper Terminals

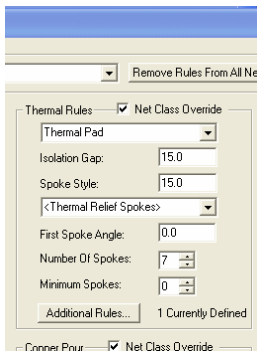
You can now move a copper terminal that has a track attached to it (copper terminals are used to connect a track to a Template for example). The movement is restricted to within the copper or template shape or to the outline if shape is not filled or closed.

Copper terminals cannot be selected so this move can only be performed by clicking on and dragging the copper terminal in one action.

Thermal Pad Connection Rules

In The **Technology** dialog, **DFM/DFT** rules, you can now define the default Thermal Pad connection at the design level or the Net Class level as well as an individual item type.

The choice of **Isolated**, **Not Isolated** and **Thermal Pad** are available. Individual pads and 'classes' of pads can still be changed using the **Additional Rules** dialog.

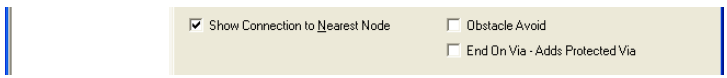


Changes to Add Via

When adding vias and using Change Nets, merge nets is never performed (the vias are moved from their current net to the new net).

Change to Protected Vias on Edit Track/Breakouts

A new check box in the **Options** dialog under **Edit Track** and **Edit Breakout** tabs now allow **End On Via - Adds Protected Via**. Check this to protect vias added against automatic deletion and layer span changes.



High Speed Option – Changes to Differential Pair Routing

Improved Diff Pair Editing

It is now easier to add differential pair definitions in the Technology dialog. The two connections that define the differential pair are selected (using Shift and Ctrl keys), then the **Technology** option, and **Diff Pairs** and **New** dialog are used to get to add differential pairs. The selected connections will be set up in the dialog ready to add.

Improved DRC Checks for Diff Pairs

The **DRC** option now checks the differential pair using the differential pair minimum gap (if defined in the Differential Pairs Technology dialog), even if

this gap is smaller than the spacing rules. DRC now applies this gap to all tracks in the diff pair path. Interactive track pairing will now always use the minimum gap.

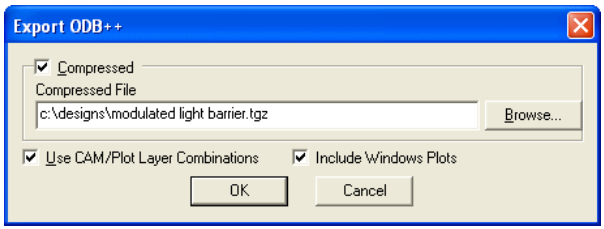
DRC now checks all the track making up the diff pair path for diff pair length even if they are not actually paired. Previously, only the 'Diff paired' tracks were checked.

Options Changed in later V4.0 Patches

These changes were made in V4.0 and are documented here for completeness.

ODB++ Dialog Changes

A new check box has been added **Include Windows Plots**.



Click **Use CAM/Plot Layer Combinations** check box to output the layer definitions used in the CAM/Plot dialog.

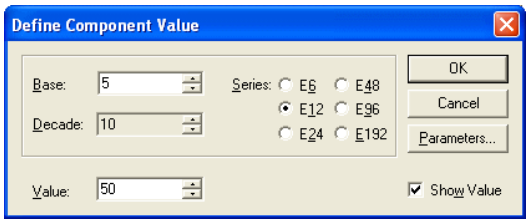
Click **Include Windows Plots** check box to output the Windows plots defined in the CAM/Plot dialog. If this is left unchecked, only plots defined for Gerber and Excellon are output as your layer combination when the above check box is selected.

Chapter 5. Spice Simulator Changes

Simulation Update

Changes to Edit Value (F7) dialog for Passives

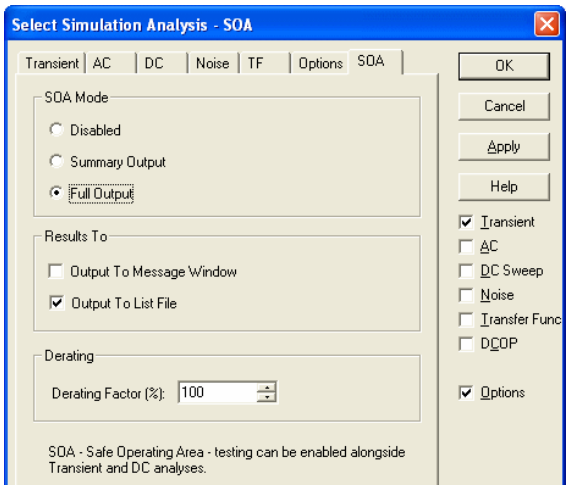
When editing Passive and Reactive devices using the **F7** key (Edit Values), it now includes support for the **E48**, **E96** and **E192** series.



Safe Operating Area (SOA) Testing

In the later version 4.0 release, SOA Testing was introduced as a command driven feature only using the Extra Simulation Data dialog. In Version 4.5, Schematic changes have been made to provide an interface to the new Safe Operating Area (SOA) Limit Testing feature.

A new **SOA** tab on **Simulation Parameters** dialog for enabling and configuring SOA testing has been added.



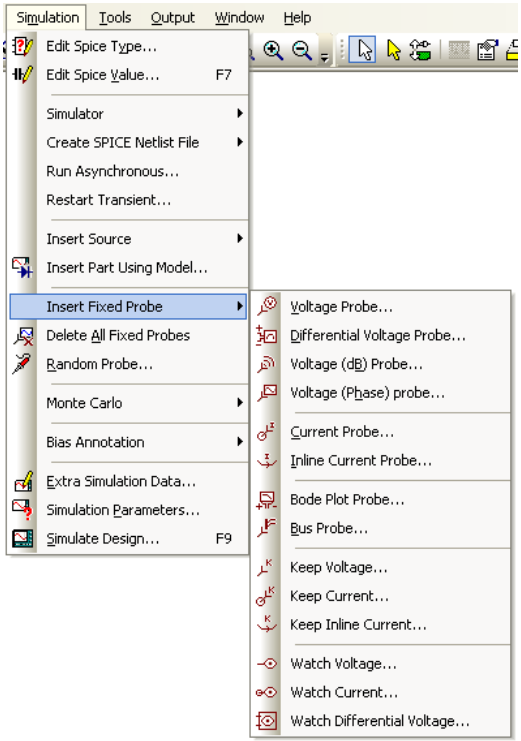
SOA Overview

Safe Operating Area (SOA) testing is a feature that can be used with DC or Transient analyses. With SOA testing, you can set maximum and minimum limits for any simulation quantity and the simulator will display when those limits are violated. It is intended to check that semiconductor devices are operating within the manufacturer's design limits.

New Built-In Simulator Parts 'Watch' Probe Functions added

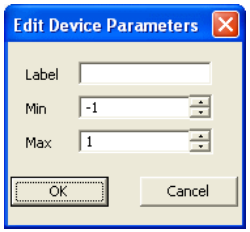
New simulator Parts with built-in functions have been added to support SOA testing: **Watch - Current**, **Watch - Voltage** and **Watch - Differential Voltage**. These are used to create Watch parts for simple SOA limit testing.

These three new Parts have been added to the Spice Parts library. These are also available from the **Insert Fixed Probes** menu and on the **Miscellaneous** Parts popup toolbar.



Edit Device Parameters

When editing Parts created with the SOA functionality, when using F7 key to edit the values, the generic **Edit Device Parameters** dialog is displayed for you to supply the **Min** and **Max** limits and a **Label** to mark any SOA violations.



Full SOA Options Details

Option Name	Default value	Description
SOADERATING	1.0	Scales min and max values used in ".SETSOA" specification. This allows a de-rating policy to be globally applied to SOA limits.
SOAMODE	off	Controls the Safe Operating Area (SOA) test mode. See ".SETSOA" below for details on how to define a SOA test. Can set to: Off — SOA testing is not enabled. In this mode .SETSOA controls will be read in and any errors reported, but no SOA testing will be performed during the run. Summary — SOA testing enabled and results given in summary form with only the first violation for each expression given being output. Full — SOA testing enabled with full results given. Every violation will be reported in this mode.
SOAOUTPUT	list	Can be: msg — Results displayed in command shell message window, or console if run in "non-GUI" mode list — Results output to list file msg|list — Results output to both list file and

command shell message window

none — Results not output to either list file or message window.

Note that all results are always stored for retrieval using the script function GetSOAResults described below. So even if "none" is specified the SOA data is always available.

Setting up SOA limits within a model

It is possible to define SOA limits within the .MODEL statement. To do this, add one or more parameters in the following format:

LIMIT(*name*)=(*min, max, xwindow*)

name — Name of quantity to test. See format for access variables useable when MODEL is specified for a .SETSOA control. This is described in section: ".SETSOA" in the section below. E.g. use 'LIMIT(vcb)' to specify the limits for the collector-base voltage of a BJT.

min, max — As described in ".SETSOA" below.

xwindow — As described in ".SETSOA" below.

Advanced SOA Limit Testing

As well as the simple tests mentioned above, the simulator control .SETSOA allows much more sophisticated definitions for SOA limits. In particular, you can define limits for all devices belonging to a specified model. Suppose that you are using a BJT model that has a Vcb limit of 15V. While you could place a differential voltage watch device across each instance of this model, this would be time consuming and error prone. Instead, you can define a single .SETSOA control that refers to the model name of the device. The simulator will then automatically set up the limit test for every instance of that model.

.SETSOA Definition

.SETSOA [LABEL=*label*] [MODEL=*modelname* | INST=*instname*]

[ALLOWUNUSED] [ALLOWWILD]

expr1=(min1, max1[, xwindow1])

[*expr2*=(min2, max2[, xwindow2]) ...]

Defines a Safe Operating Area (SOA) specification. If SOA testing is enabled the simulator will check simulated results against this specification and record any violations.

The results of SOA testing are output to the list file by default and can optionally also be displayed in the command shell message window, or console window if run in non-GUI mode. They are also always available via a script function GetSOAResults(). See .OPTIONS setting "SOAOUTPUT" for more details.

label — Optional label that will be included in every violation report. You can use the following symbolic values in this label:
%INST% - substituted with the instance name that violated the specification. This is only meaningful if MODEL or INST are specified. (See below).
%MODEL% - substituted with the model name that violated the specification. Only meaningful if MODEL is specified. (See below).
%EXPR% - substituted with the expression that violated the specification.

instname — If specified the expression or expressions supplied in expr1 etc. are applied to the specified instance (e.g. Q23, M10, R56). In this case the expression may refer to node voltages and pin currents of the specified instance. See details under expr1, expr2...

modelname — If specified the expression or expressions supplied in expr1 etc. are applied to every instance belonging to modelname. In this case the expression may refer to node voltages and pin currents for each instance processed. See details under expr1, expr2...

expr1, expr2... — Expression to be evaluated and compared against minimum and maximum specs. This expression can access simulation results using access variables. The format and scope of these variables depends on whether MODEL, INST or neither is specified.
If neither is specified, the expression can use the global access variables defined below:

Syntax	**Function**	**Example**
nodename	Voltage on node	VOUT – voltage on node VOUT
n(nodename)	Voltage on node	n(VOUT) - voltage on node VOUT
instname#param	Instance parameter	M2#vdsat – vdsat value for M2
		Q23#c – current in collector of Q23
paramname	Parameter defined using .PARAM	

If there is a clash between a paramname and nodename, that is if the same name could refer to either a node or a parameter, then the parameter name takes precedence. To access the node in this case, use the n(*nodename*) syntax.

Use the following values if MODEL or INST is specified. In each case (excepting the global access variable) the variable accesses a quantity for the

instance being processed. With INST this will be the single instance specified by instname. With MODEL all instance belonging to the model specified by modelname will be processed.

Syntax	Function	Example
pinname	Current in pin	c - current in collector of transistor
Ipinname	Current in pin	Ic - current in collector of transistor
Vpinname	Voltage on pin	Vc - voltage on collector of transistor
n(pinname)	Voltage on pin	n(c) - voltage on collector of transistor
Vxy Where x = pin name 1, y= pin name 2. Both x and y must be single letters	Voltage between x and y.	Vbc – voltage from base to collector
pow	Power in device	
param	Readback parameter	vdsat - 'vdsat' for MOSFET
#global_name	Global node voltage or pin current	#VOUT – voltage on net called VOUT #q23#c – current in collector of q23
paramname	Parameter defined using .PARAM	

Note that currently the use of V() and I() is not accepted and will result in an error message being displayed.

min, max — Minimum and maximum values respectively. A violation message will be produced if the value of the associated expression is less than min or greater than max. Use '*' if the limit is to be ignored. E.g. (*, 15) will test a maximum value of 15 but the minimum value will not be tested. min and max values may be scaled using the SCADERATING .OPTIONS setting (see below for details). Currently, only constant values are accepted for min and max. Expressions are not permitted.

xwindow — Optional value specifies a minimum window that must be surpassed before limit violations are registered. For example if 10u is specified for xwindow for a transient analysis, then the

limit must be exceeded continuously for at least 10uS before the violation is recorded.

ALLOWUNUSED If INST or MODEL are specified, an error will result if no instances to be processed are found. If INST is specified the error will occur if instname doesn't exist. If MODEL is specified, the error will occur if there are no instances using modelname even if modelname itself is valid.

This error will be inhibited if ALLOWUNUSED is specified

ALLOWWILD If specified, wildcards can be used for modelname and instname. In this case Pulsonix Spice will search for all devices that match the wildcard specification. Use '*' to match any sequence of characters and '?' to match a single character.

New SOA Testing Script

GetSoaResults

No arguments

Return type: string

Returns the SOA results for the most recent simulation.

Return Value

Returns an array of strings, each one describing a single SOA failure. Each string is a semi-colon delimited list with fields defined below.

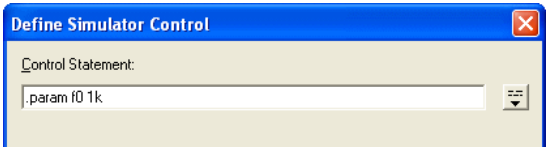
Field	Description
0	SOA Label
1	Start of failure
2	End of failure
3	under' or 'over'. Defines whether the test fell below a minimum limit or exceeded a maximum limit.
4	Value of limit that was violated

New Simulator Control Part

A new generic Part called **Simulator Control** has been added to enable you to add control statements to the schematic rather than placing them in the **Extra Simulation Data** dialog. This way simulation parameters, like .param f0 1K for example, can be seen in the schematic design. The control statements will be added to the netlist when simulation is run.

The new **Simulator Control** Part is also on the **Miscellaneous** popup toolbar. It uses a new **Simulator Control** spice function.

When this Part is edited using the **F7** key the **Define Simulator Control** dialog will be presented for you to enter one or more control statements.



Simulator Bug Fixes

Below is a list of Simulation fixes that you may need to be aware of:

- Big "F" was not being stripped from Spice Value in the netlist when using it in a template.
- Template now uses <SpiceValue> instead of %%<Spice Value>%%. This way the value is not substituted before the "F" is stripped off. Old templates will still work but will not strip off the F.
- From a **Simulation Graph** window, the **Plot** menu option **Create Plot Journal** now works. It used to say this feature was not available.
- Probing nets that start with + or a number now work correctly.
- Now gives warning if you try to plot the ground node.
- Now expands attribute values for <SpiceValue> and <SpiceParameters> when writing the netlist, so you can include values from other attributes using the %%attribName%% format.

Appendix A. Supplementary File Changes

This section details new default files and changes made to existing files.

Format Files

New format file examples are supplied to support the availability of the **Report Maker** in the **Library Manager**.

- **Part Details** — A new Part specific report to output Part details.
- **Part Gates (CSV)** — A new Part specific report to list gates used on a Part. File written in CSV format.
- **Parts Pins (CSV)** — A new Part specific report to list PCB footprint pins on a Part. File written in CSV format.
- **Library Contents** — A new report to list any Symbols or Footprints in a library.
- **Symbol Pad Report** — A new report to list information about pads within the symbol.
- **PCB Specification Sheet** — A new report provided as an example to generate an HTML format file that can be read in a HTML browser such as Explorer.

Updated Libraries

New Package Libraries

New package libraries to support the **3D Viewer** option are supplied. These are installed into the \MasterLibrary folder.

- **Connector.pkg** — New package library for connectors
- **SMT.pkg** — New package library for surface mounted components
- **Default.pkg** — New package library for all other devices