Pulsonix Design System

V5.0 Update Notes

Copyright Notice

Pulsonix, a division of WestDev Ltd.

Printed in the UK Issue date: 20/05/08 iss 1

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 V5.0

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 5.0 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 5.0 on top of your existing installation, you do not need to uninstall any old version first.

Licensing

If you are already using Version 4.6 or an earlier version you will require a new license for Version 5.0. This will be supplied to you by email under your current maintenance contract.

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**.

System Files

There are no new system files for this release.

New Additional Supplementary Files

Details of all additional files supplied are supplied in Appendix A at the end of this document.

Pulsonix Spice Installation

During installation of Pulsonix Spice, if updating from a previous version, there are changes to the installation folder which you should be aware of.

For existing users who have not added their own models to Pulsonix, you can ignore the comments below.

Pulsonix-Spice, in common with most applications, needs to store a number of values that affect the operation of the program. These are known as configuration settings. Included among these are the locations of installed model libraries, font preferences, colour preferences and default window positions.

Some changes to the file structure have been made especially the structure of the files located in the application data path. Some of these changes have been made to accommodate Windows Vista **User Access Control** which write protects the **Program Files** tree. In particular the following has changed.

1. The Application data directory path name has changed to include the version number "<your_application_data_folder>\Pulsonix\Pulsonix-Spice540".
2. The configuration file has moved to a subdirectory \Config under the application data path.This means you will need to set up your **Options** again in the simulator program, and set up your model paths again.
   The configuration file from the previous version will still be on your system in the text file "<your_application_data_folder>\Pulsonix\Pulsonix-Spice\v52\Base.sxprj".
   If you cannot remember the options and model paths you had set up, they can be found in this text file in the [Models] and [Options] sections.
   If you have your own models installed into the Pulsonix environment, you need to be aware of this:
   To set up your model libraries again, from within the Pulsonix-Spice command shell. From the **File** menu, select **Model Library** and **Add/Remove Libraries**, or simply drag the folders from the Windows Explorer to the Pulsonix-Spice command shell.
3. The default user script directory has moved to a location under the "My Documents" folder. This directory is not created by the installer but will be created by Pulsonix-Spice when one of the script options under the **File** menu and **Scripts** is run.
   Either copy your own existing scripts to this folder, or change the scripts file location in the general options dialog.
4. Because of the application data path name change, existing users will need to re-inform the simulator about the model to Pulsonix Part associations.
   After you have installed the new Pulsonix program and have updated the Pulsonix-Spice model library locations, you need to run the Simulator Setup facility as follows:

- Run up Pulsonix and open any schematic design (or create a new one).
- From the **Simulation** menu, use the **Simulator** and **Simulation Setup** option.
- If you had changed the Part names for built-in models then reset these to the correct names using the top part of the dialog. It is unlikely that you changed these, but if you did and you cannot remember the correct names, you will be able to find them in the old text file "<your_application_data_folder>\Pulsonix\Pulsonix-Spice\map_symbols.txt".

- Press the **Update Simulator Parts List** button and exit the dialog by pressing **OK** when this is complete.

5. If you created your own model to part associations with the previous version, you will have a file called “<your_application_data_folder>\Pulsonix\Pulsonix-Spice\user.cat”. If you want to keep these associations with the new version you will have to copy this file to the “<your_application_data_folder>\Pulsonix\Pulsonix-Spice540” folder, and rename the file to “user_v2.cat”.
   Now from the **File** menu, select **Model Library** and **Rebuild Catalog** option to update the catalog files contain the latest models and part associations.

The new simulator will now be ready to use.

Network Licensing Changes

Network licensing has undergone a major rewrite to include new functionality. In addition to existing functionality, new options have been added to enable:

- Loan licenses
- Named domains and groups
- License usage logging
- Multiple and redundant servers

If you are using the floating licensing scheme, new and additional documentation can be found in the *Pulsonix Network Licensing Guide* available on the product CD or on the Pulsonix web site.

Chapter 2. General Options

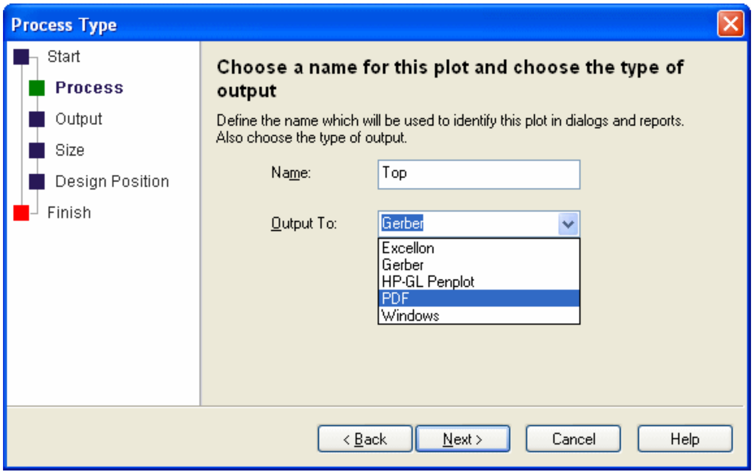
Direct Printing to PDF

The new plot to PDF output is used to write PDF format file with active annotation. Once the PDF file is created, you can hover your cursor over design items and properties of that item will be displayed.

Writing the PDF files

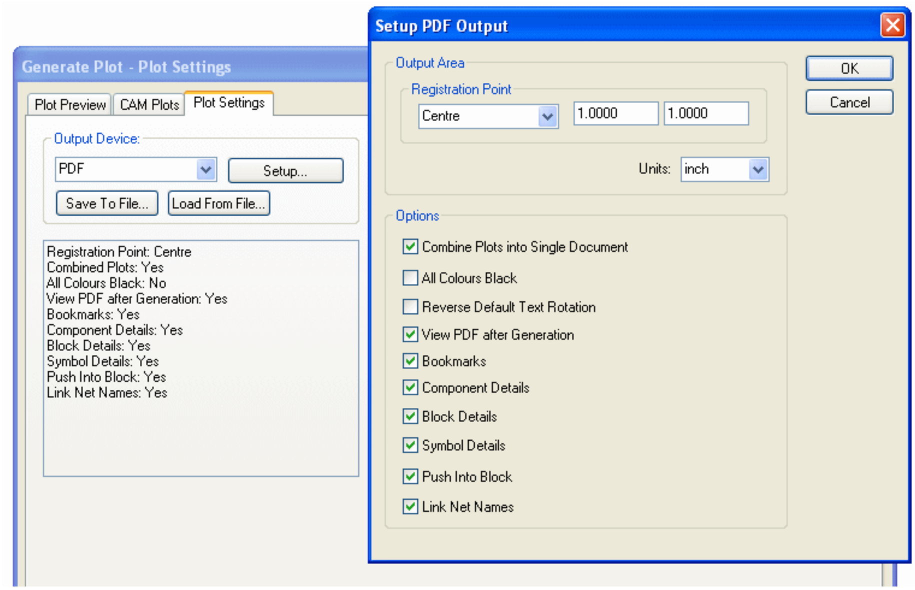
The PDF output is additional to the standard Windows driver and you do not need to install a separate driver for it. If you have a PDF Windows driver already installed, you can still use it. The Pulsonix PDF driver has the added ability of the active annotation which your standard PDF output will not have.

The **PDF** output is accessed through the **CAM Plot** dialog and **New** or **Edit** when you create a plot. Simply select it from the drop down list of available output types.



Setting up the PDF output

You can configure the PDF output using the **Device Setup** option from within the **CAM Plot** dialog.



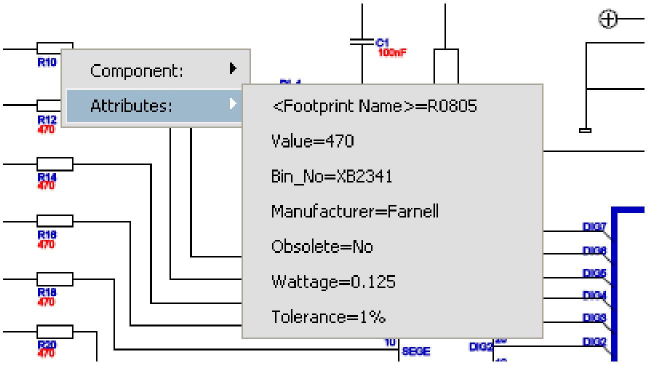
Active Annotation options available

Active annotation options are available but only for Schematic designs. PCB designs are always ‘flat’ with no annotation.

In a Schematic design, all pages in the design and hierarchy are plotted (including lower level hierarchical blocks).

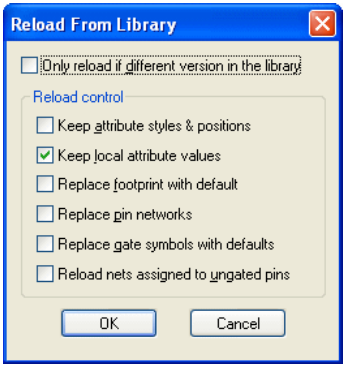
Using the PDF output

Once the PDF file has been created and it is open, you can hove your cursor over design items, such as Components or connections. A small context menu is displayed from which you can view Properties of the item chosen.



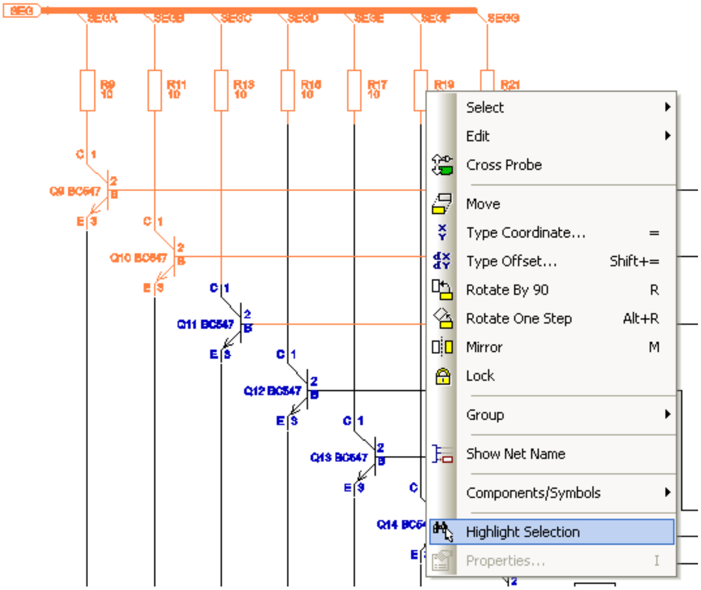
Reload & Replace Part

When performing a **Reload**, the option has a check box to allow you keep or remove local component attributes, attributes that have been added to that component instance.



Highlight Selection

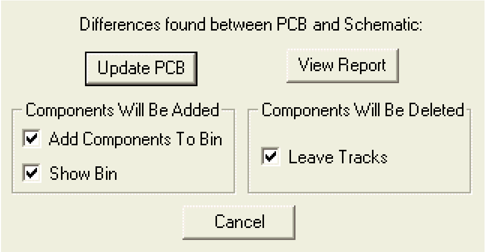
Once a selection has been made in the design, from the context menu you can now **Highlight** the **Selection**. When you select away from this, the selection will still be displayed in the selection colour. Use **Unhighlight Selection** from the context menu to remove it.



Synchronise Design Changes

Rearranged dialog

The **Synchronise Designs** dialog has been rearranged to make synchronise choices more visible.

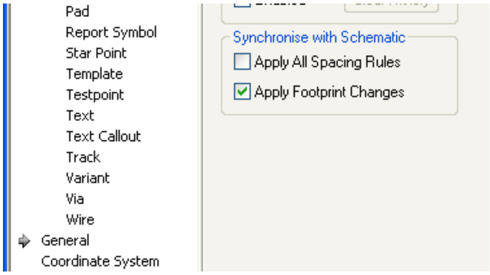


Leave Tracks

If components are to be deleted from the PCB, you will be presented with the option to **Leave Tracks** to the components as *dangling tracks*. Select the check box to leave them, or uncheck it to delete the connected tracks.

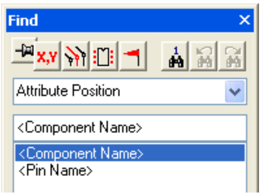
Footprint Settings

In the **PCB Design Settings** dialog under **General**, you can now choose to ignore specific footprint settings in the Schematic. This means if you define the footprint in the Schematic but then change it from the default in the PCB Design, you can use this check box to ignore the change.



Find by Attribute Position

In **Find**, changes have been made to the category **Attribute** to **Item (by attribute),** and a new category **Attribute Position** has been added.



The new category lists all attribute names (including in-built ones like <Component Name>) that have attribute positions in the design.

Note: some of these attribute positions are hidden on items (like when a component name is unchecked in the properties dialog). A message will be shown if a hidden attribute position is found. Hidden attribute positions are found only after finding all visible ones.

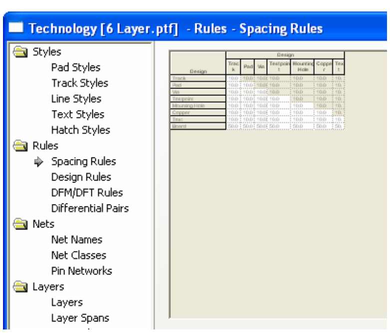
This category is required to make it easy to make global changes to similar attribute positions. For example, to change component names to use the same text style, or to change all testpoint names to the same layer.

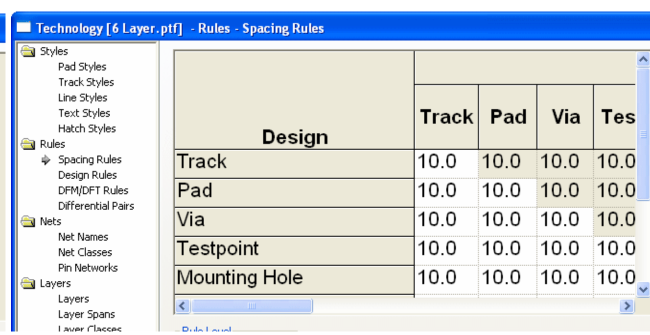
Mouse use in Dialog Grids

Changes have been made to the functionality available within grids used in some dialogs, **Technology** and **Layers** for example.

Using the wheel button (on the Intellimouse), you can now scroll up and down the list of items displayed where the list is longer than the grid visible.

Using the **Ctrl** key on the keyboard, pressing this and rolling the wheel button, you can zoom in and out of the grid, like you can with Microsoft Excel for example as shown below.

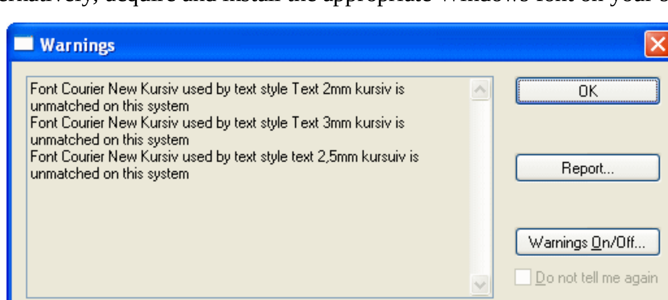




If you press and hold the wheel button pressed you can use the auto-pan option to scroll up/down and left/right depending on the mouse movement direction.

Warning message if Type Font is not defined on the system

On the opening of a design, you are now warned if a text style uses an unknown true type font. The warning message will tell you the font expected and the text style which uses it. If you wish to make a font replacement, use the **Technology** dialog and **Text Styles** to do this. Alternatively, acquire and install the appropriate Windows font on your system.

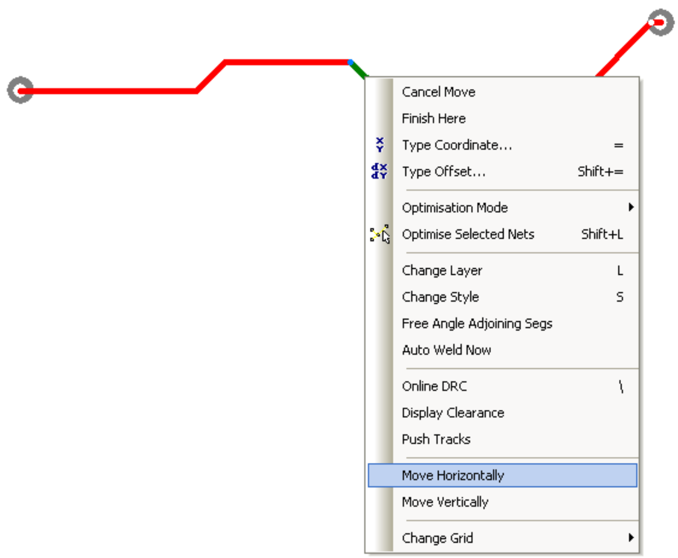


Move Horizontal/Vertical

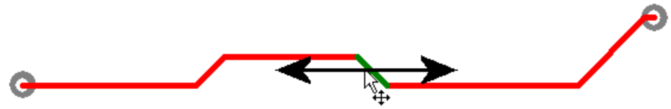
During **Move** and **Move Corner**, two new commands **Move Horizontal** and **Move Vertical** are available on the context menu. Selecting one of these restricts the item movement to that direction only.

These options can be added to a shortcut key or can be used from the **Move** context menu. The commands can also be used directly on a selected item (so that you don't get the jump to grid when it is first moved).

Note: This option will not work if you are moving on a special grid (Polar grid, snapping to Placement Sites or Construction Lines).



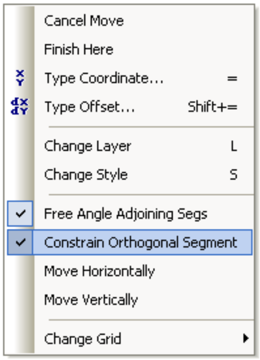
From the context menu, select **Move Horizontal** or **Move Vertical**.



A model cursor is displayed. During **Move Horizontal**, the movement of the cursor is restricted in the horizontal plane.

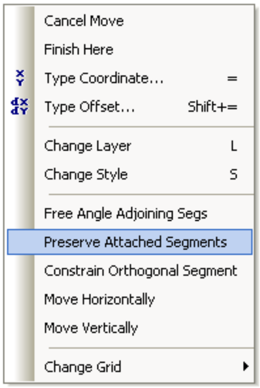
Constrain Orthogonal Segment

In previous versions of Pulsonix, single selected orthogonal segments were always constrained to move only in their perpendicular direction. There is now an option included –**Constrain Orthogonal Segment**, on the **Move** context menu to switch this off. Once off, this allows movement in any direction.



Preserve Attached Segments

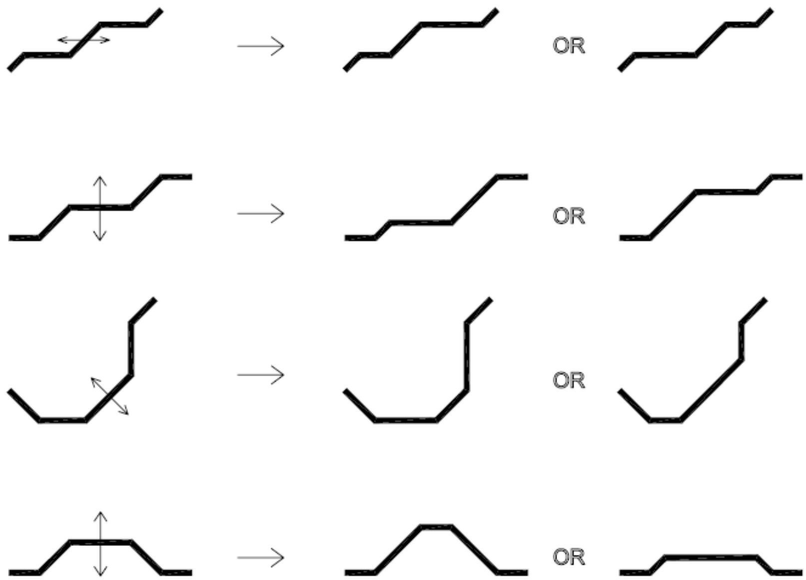
During **Move**, a new option is available to describe how unselected segments attached to moving selected segments are altered. This is available on the context menu when **Free Angle Adjoining Segs** is switched off.



If you leave this option unchecked, it will work as before in previous versions of Pulsonix, i.e. the attached segments are matched to a segment mode and altered thus.

Select it to preserve the angle and far end point of the unselected attached segments. This is usually achieved by changing the length of the selected segment.

This is demonstrated during track editing.



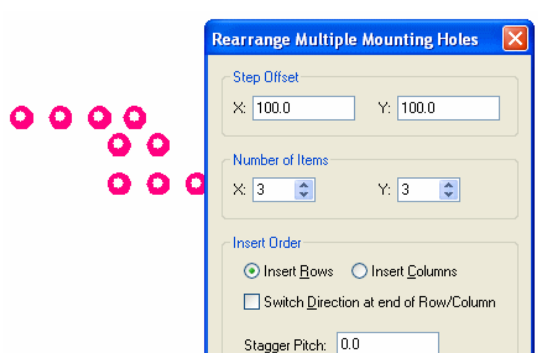
Rearrange Multiple Items

If you select multiple free pads, vias, mounting holes (all with the same style) or Components (with the same footprint), the context menu displayed the **Rearrange Multiple Items** option. You can enter this mode by **selecting** the items or from within the **Move** option.

Note: this option is not available if the items have track attached.

If you wish to assign a shortcut to this option, it shares it with the **Insert Multiple Items** command/shortcut.

This new option will allow you to rearrange the items in a grid that you specify with the **Insert Multiple Items** dialog using the controls provided.



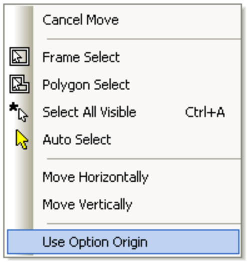
Once multiple items have been selected, the option sets all items' mirror, rotation and layer flags to be the same as the item in the selection that your cursor is over.

Option Origins

When working with option with the latch mode selected, you can now use two commands **Use Option Origin** and **Place Option Origin**.

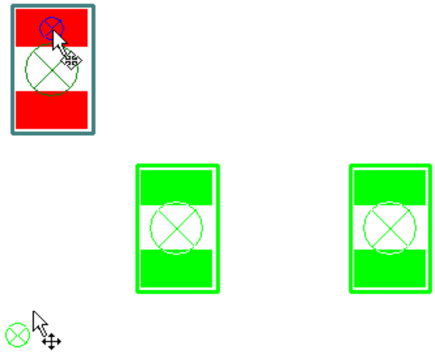
This option is used to create a temporary origin as an alternative to the symbol origin for use when Moving, Rotating or Mirroring the item.

Once a latch mode has been selected from the toolbar, select **Use Option Origin** from the context to display a temporary origin. Use **Place Option Origin** from the context menu to change its location after it has been placed or if it needs to be moved. The **Use Option Origin** can be selected from the context menu to hide it again. If the temporary origin is displayed, the rotate or mirror option will perform their task about this origin. You can also use **Alt-Click** as a shortcut to define the origin while in latch mode.



Using the Option Origin in Move mode

You can now use the **Use Option Origin** feature in the latched mode of **Move**. This works the same way as the Mirror and Rotate options shown above. This allows you to move any item, or multiple items, and define the origin for the move.

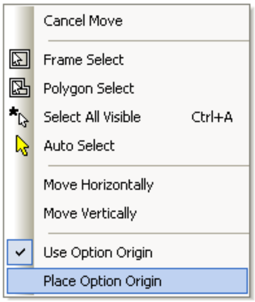


The temporary origin stays visible during the move to make it easier to place it over existing items.

This can be used, for example, to place a component by its first pad when its origin is already defined at its centre. This mode also allows you to reposition some imported shapes so that a particular point on the shape aligns up exactly with a known point in the design.

Show Placement Origin

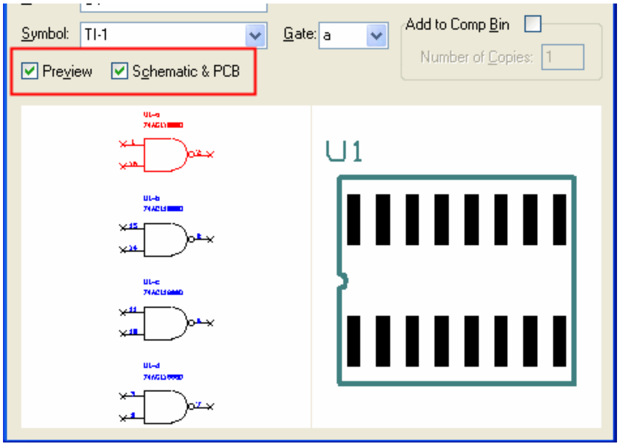
You can now show the placement origin whilst moving items. A new option command **Show Placement Option** is available on the context menu whilst moving.



The origin displayed is a fixed size and shows the position displayed on the Status bar.

View SCM & PCB Symbol Previews in Insert Component dialogs

You can now optionally preview both **Schematic & PCB** symbols by selecting the check box on the **Insert Component** dialog. This preview check box is available in both Schematic and PCB design editors.

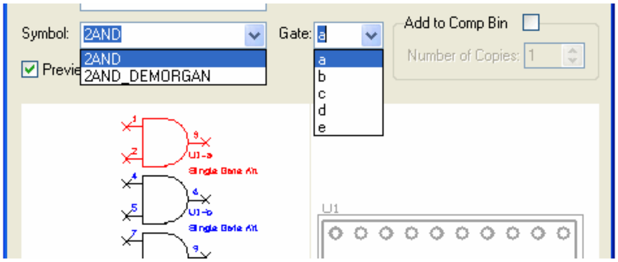


Zoom in Preview Windows

You can now use the wheel button on the mouse to zoom in and out of each of the Preview windows. Note, this functionality works in any Preview window.

View Symbol Alternates in Preview Windows

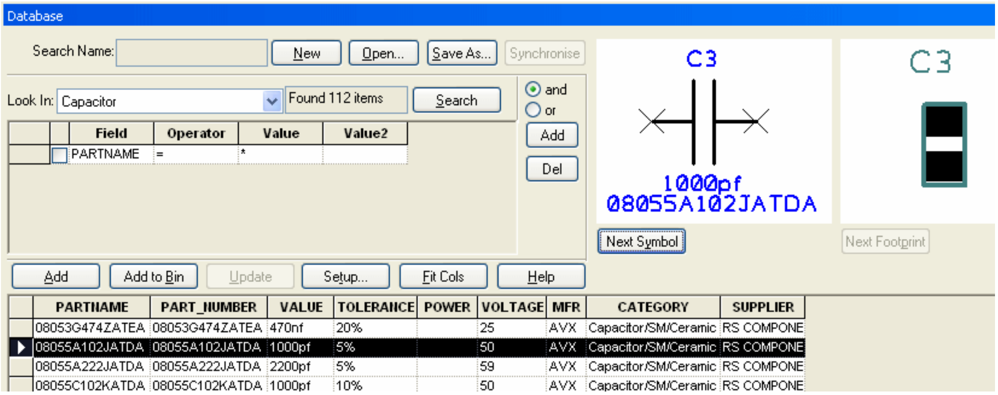
You now have the ability to select a different symbol alternate on the **Insert Component** dialog for the current symbol type, i.e. Schematic symbols in Schematic editor etc. You can also select different footprints while in the Schematic editor, this was available in previous versions but you can now see its preview using a drop down list of **Footprint symbol alternates**.



The **Symbol:** drop down list will show you the symbol for the current **Gate:** (Gate is shown when there is more than one gate for a Part). Where the Part is constructed of multi-gated symbols, the current selected gate is shown. Using the drop down list you can select another gate by letter or number and the alternate symbol for that gate shown.

Next Symbol in PDC Window

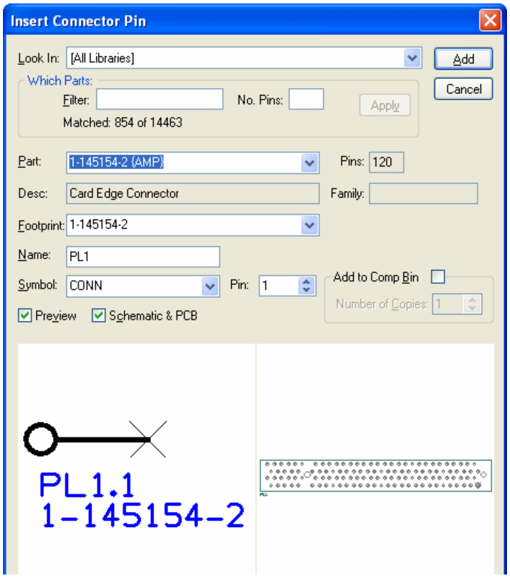
If you have purchased the **Pulsonix Database Connection (PDC)** product, you now have the ability to show symbol and footprint alternates in the preview window. If symbol alternates are available for the selected database Part, the **Next Symbol** and **Next Footprint** buttons will be highlighted. Selecting this will display the next symbol for that Part.



Preview Windows in Insert Connector Pin

When using the **Insert Connector Pin** option, you can now check the **Preview** box to view a preview of the currently selected connector. As with the Insert Component dialog, you are also able to select the **Schematic & PCB** box to view both the symbol types.

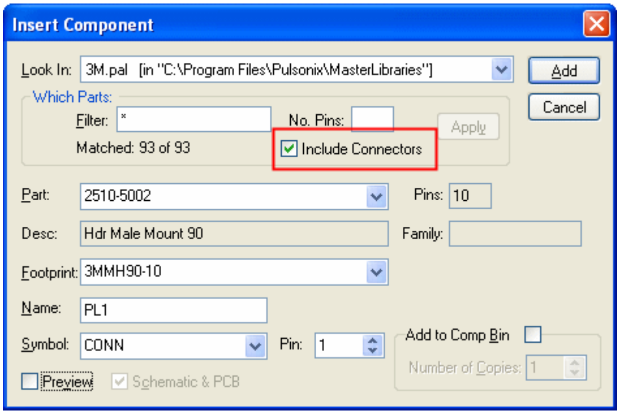
The **Insert Connector** dialog has been changed and is now the same style as the **Insert Component** dialog.



Changes to Insert Connector Option

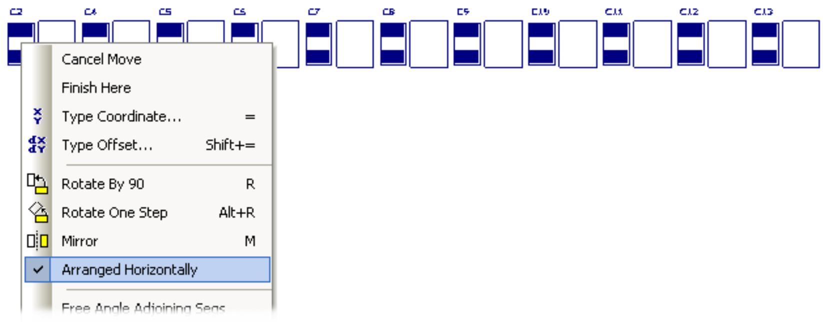
Insert Component

As well as using the existing Insert Connector dialog to add a connector, it is now also possible to add a connector from the **Insert Component** dialog. There is a **Include Connectors** check box in the filter section.



Arrangement of Components From Bin

When dragging multiple components from the bin you can now switch the arrangement between horizontal and vertical placement. The **Arranged Horizontally** check appears on the context menu; uncheck it to arrange the components vertically. This option is remembered for the next time it is used.



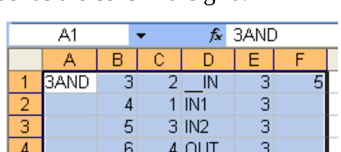
Part Editor Changes

Improved Cut and Paste in Part Editor

Significant improvements have been made to the Cut and Paste features within the Part Editor Pins and Gates pages. Improvements have been made to the cell selection and selection ranges.

Using Copy and Paste To Edit Values in cells

The values in this grid can be manipulated using the normal Copy and Paste techniques associated with spreadsheet applications such as Microsoft Excel. This includes the ability to copy and paste multiple rows and/or columns of data. One possible use would be to copy the pin data into an external spreadsheet to take advantage of advanced features such as sorting and filtering, then paste the modified data back in to the Part Editor. When pasting from an external source it is important to ensure that the data is arranged in the same order as the cells in the grid.



The **Copy** and **Paste** commands are available from the **Edit** menu on the top menu bar or via the context menu. Alternatively, the standard **Ctrl+C**, **Ctrl+V** shortcut keys may be used.

Selecting cells to copy or paste

A single grid cell or a range of grid rows and/or columns may be selected for copy or pasting. By default, clicking on a cell causes that cell to enter *edit mode* indicated by a flashing cursor and any copy or paste will be directed to the edit control or dropdown list for that cell. If, rather than clicking, the mouse is dragged while the left button is held down, a range of cells may be selected indicated by being shown in reverse highlight. This range of cells can now be copied or pasted as a whole.

Gate	Symbol	Symbol Pin	Pin Name	Logic Name	Pin Swap	Gate Swap
a	3AND	1	1	IN1	2	0
		2	2	__IN	2	
		3	3	IN2	1	
		4	4	OUT	1	
b	3AND	1	5		0	0
		2	6		0	
		3	8		0	
		4	9		0	
c	3AND	1	10		0	0
		2	11		0	

If a single cell has already been selected by clicking, it is possible to extend this into a range of cells. First, if the cell is in *edit mode,* press the <Enter> key to exit *edit mode* so the cell is simply shown as selected, outlined in bold. The selection may now be extended by dragging the mouse as described or by clicking with the <shift> or <control> keys held down. The <shift> or <control> key can also be used with the keyboard arrows keys to extend the selection. To quickly select an entire column click in its header cell.

Pasting Values into the Grid

When data is pasted into grid the values are subject to the normal constraints of the column into which they are being inserted. Any invalid values will not be inserted. For example, a value pasted into the Pin Name column must be a valid Pin Name defined in the Pin Names column of the **Pins** page.

The **Gate** and **Symbol Pin** columns are read only and as such may not have values pasted into them although they may be selected and copied.

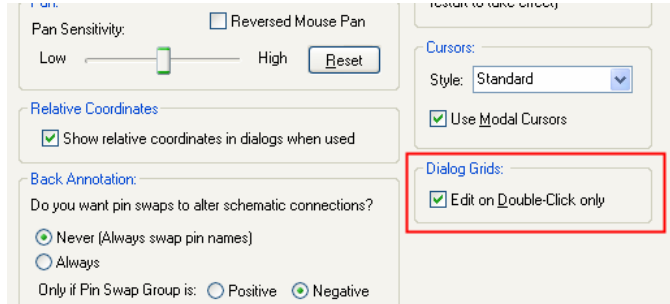
The cells affected when pasting is dictated by the range of the cells copied not by the range of cells currently selected in the grid. So copying a 2 by 3 range of cells and pasting into the grid will paste a 2 by 3 range of cells starting at the top left cell of the currently selected range, subject to the constraints of the columns affected.

One exception to this is when pasting a single value into multiple cells. All cells in the current range of cells will be set to the pasted value, again subject to the column constraints.

Edit on Double-Click

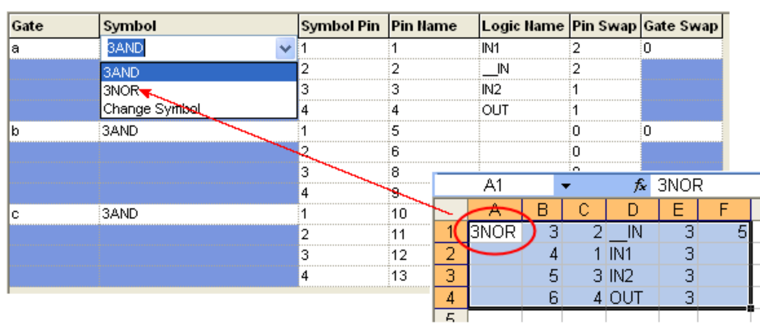
Throughout the system many dialogs use two-dimensional data grids to display information. By default, when interacting with one of these grids, clicking in a cell will immediately causing it to enter edit mode. On some occasions it may be preferable for the grid to behave more like a traditional spreadsheet and checking the **Edit on Double-Click only** option will cause a cell to only switch into edit mode when it is double-clicked rather than single-clicked. This can make it easier to select ranges of cells for example when using copy and paste in the **Part Editor**.

To enable this, check the **Edit on Double-Click** box in the **Options** dialog under **General**.



Symbol Alternates

If the **Copy** selection in the Spreadsheet includes the **Gate** name where the Gate name is the alternate Symbol, provided the Part Editor Gate definition has the alternate listed, when the Paste of the selection is made, the Symbol Name will now switch to the alternate Symbol Name.



Cell operation changes

You can now use the **Delete** key in non-editing **Part Editor** grid cells where applicable. It will work for a single cell or a range of cells.

Check Pin Mappings

Within the **Parts Editor** there is now a **Check Pin Mappings** option on the **Tools** menu. This will report pins that have not been mapped to a gate or that have been mapped more than once and is useful for Parts with large pin counts such as FPGAs.

CSV Import file format into Part Editor

The CSV import mechanism allows you to create Pin data for Parts in an external editor, Microsoft Excel for example, and to import this data into the Part. This may be beneficial if there are many pins or if the data needs to be manipulated first.

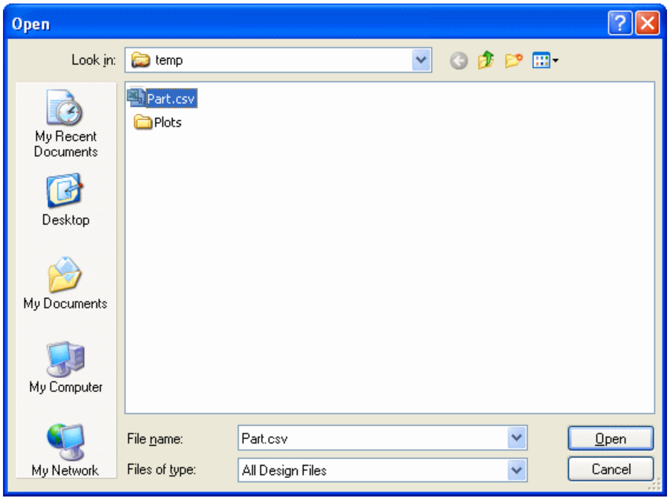
CSV files are not Part specific, the data within one single CSV file can be applied to many Parts in the Parts library.

In addition, for Import and Export of CSV files, also see the next chapter on new Schematic Options and FPGA support section.

Opening the CSV file

Once a Part has been initially created in Pulsonix, open it and then run **Import Pin Data** from the **File** menu.

From the **Open** dialog, select the file for import.



The normal rules of importing files applies, you must not have the file open at the time during import otherwise a ‘fail’ message will be displayed.

File Format

The CSV file can contain any of the following Part editor headings in any order: Pin Name, Logic Name, Pin Swap, Net Name, Pin Type, Pin Networks and user defined attributes. During import, the Pin Name is matched with the corresponding data, this means the Pin Name field is mandatory but it doesn’t have to be the first text entry in the CSV file though.

The first line in the CSV is ignored as it is assumed to be a header. The Import option will read the headers to give it its structure. The headers can be in any order but their values must match the column titles in the Part editor. Column headers that don’t match the Part editor titles will be added as new Pin Attributes and the field entries added as the per pin values.

Using two commas will provide you with an ‘empty’ column field on import if you wish to miss out a field of data.

As with normal text in Pulsonix, doubling the barring character will display names with a bar over the top of them.

If a **Pin Type** is defined as **No Connect**, but the **Net Name** field is also defined, the Net Name will be ignored during input (the Pin Type will still be added though).

If your CSV file contains an empty Pin Type field this will be defined as <Undefined> in the Parts Editor. You may also define the field especially as <Undefined> in the CSV file if you wish to define a pin type for all the pins, to help identify the pin type field for clarity.

Basic Example

Below is a basic example of the CSV input file:

```
Pin Name,Logic Name
1,LOGIC1
2,LOGIC2
3,RESET
```

Complex Example

Below is an example of a more complex CSV input file, this shows the addition of the Pin Order attribute field as well as Pin Swap, Net Name, Pin Type and Pin Network fields:

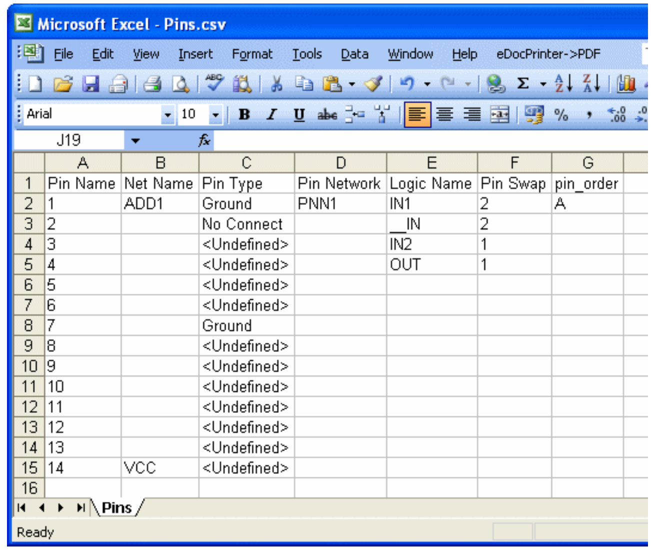
```
Pin Name,Logic Name,Pin Swap,Net Name,Pin Type,Pin Network,Pin_Order
1,IN1,2,ADD1,Ground,PNN1,A
2,__IN,2,ADD2,No Connect
3,IN2,1
4,OUT,1
7,,,GND
14,VCC,,VCC
```

CSV Export Option

You can also export pin information to a CSV file using the **Export Pin Data** option from the **File** menu.

Note: on Export, all Pin Type fields shown in the Parts Editor as <Undefined> will be output as this value into the CSV file.

If your system is set to automatically run CSV files in Excel for example, once the CSV file has been exported, it will show in your spreadsheet:



	A	B	C	D	E	F	G
1	Pin Name	Net Name	Pin Type	Pin Network	Logic Name	Pin Swap	pin_order
2	1	ADD1	Ground	PNN1	IN1	2	A
3	2		No Connect		__IN	2	
4	3		<Undefined>		IN2	1	
5	4		<Undefined>		OUT	1	
6	5		<Undefined>				
7	6		<Undefined>				
8	7		Ground				
9	8		<Undefined>				
10	9		<Undefined>				
11	10		<Undefined>				
12	11		<Undefined>				
13	12		<Undefined>				
14	13		<Undefined>				
15	14	VCC	<Undefined>				
16							

At this point, you could edit the spreadsheet and re-import it back into the Part editor overwriting the existing data in that Part.

FPGA Support in Pulsonix

Overview

Pulsonix includes additional features to aid the use of Field Programmable Gate Arrays (FPGA) and the increasing large pin counts associated with this type of device. Most of the major FPGA manufacturers provide their own tools for designing the logic and performing the pin assignments necessary with an FPGA. Interface features are provided that allow the integration of these tools and the information they generate directly in to the Pulsonix system.

In phase I of this interface, Pulsonix will directly support formats for Altera's Quartus II development system and the Xilinx ISE development system. Other system interfaces will be developed based on demand. Please contact our technical support desk or your local distributor if you would like to register your system. Pulsonix V5.0 will also write and read Part Pin CSV format files, this generic format can also be used in the FPGA environment.

The FPGA interface is supplied as a standard interface within the Pulsonix environment free of charge.

The flow of the process

FPGA pin assignment data generated by manufacturers' tools can be imported into a part in the Pulsonix library. From there, it can be included with the part into the schematic and propagated forwards to the PCB layout. You can also start from within the Pulsonix Part editor and export the Part pin data to a CSV format file for use with the FPGA tool.

In parallel, you can develop your FPGA internal functionality using the Altera and Xilinx development tools. Once completed, or even part completed, write out an ASCII file. Changes to the FPGA pin out can be quickly reloaded into Pulsonix reducing the need for error prone manual editing.

Reports about pin swaps performed in the Pulsonix PCB will include additional FPGA information to assist with the process of updating the corresponding pin assignments in the FPGA design system.

Multiple FPGA implementations may be retained as separate Parts in the Pulsonix library.

The Pulsonix Part used in the design is then replaced using the new pin mappings to complete the process. This can be an iterative process and run multiple times to completion of the finished FPGA device.

Importing FPGA Pin Data

FPGA pin information is read into a Pulsonix Part using the **Import Pin Data** option.

As some FPGA manufacturers provide pin assignments output suitable for loading in to a spreadsheet, an alternative to using the Import Pin Data option is to use the copy and paste facilities in the Part Editor to manually construct the FPGA pin data. If this method is used, is it important to remember to check the FPGA setting on the Part to take advantage of the additional FPGA features provide by Pulsonix.

The Part to gate pin mappings may be checked by using the **Check Pin Mappings** option.

FPGA Pin Data Formats Supported

Altera PIN file format

The Altera Quartus II software will automatically generate a PIN file as Part of the FPGA design process. The PIN file is an ASCII text file which contains pin assignments and other pin information for an FPGA design.

Importing a Altera PIN file will assign the *Pin Name/Usage* values from the file to the **Logic Name** fields of the Pulsonix Part pins by mapping the *Location* values from the file to the Pulsonix Part pins' **Pin Name** fields. All other fields in the PIN file are currently ignored.

Xilinx PAD file format

A PAD file can be generated by the Xilinx ISE software as part of the FPGA design process. It is an ASCII file containing the I/O pad assignments and other properties.

Importing a Xilinx PAD file will assign the *Pin Name* values from the file to the **Logic Name** fields of the Pulsonix Part pins by mapping the *Pin Number* values from the file to the Pulsonix Part pins' **Pin Name** fields. All other fields in the PAD file are currently ignored.

CSV file format

It is also possible to import (and export) Part pin data in standard CSV format. If the CSV includes the Logic Name field the Part will be set as a FPGA.

Other formats

The list of formats supported by Pulsonix is constantly being extended. If you have a format which is not currently supported, please contact our technical support desk to check if the format is under development or now available.

FPGA Pin Names as Pulsonix Logic Names

Once loaded as Logic Name values attached to the pins on a Pulsonix Part, the FPGA pin names can be used as an aid to making the correct connections in the schematic and laying out the PCB.

When you add a component using the Part to a schematic (or PCB) you can choose to display the Logic Name in the design as an attribute alongside the pin. If you prefer, you can choose to show the Logic Name as Part of the tooltip information displayed when you hover the cursor over the pin. The Logic Name is also shown in the Properties dialog for the component pin.

Additional FPGA specific behaviour

A part that is marked as an FPGA will trigger the following additional FPGA specific behaviour:

Reloading revised Pin Data into a FPGA Part

A second and subsequent import of the FPGA pin data will not simply add the FPGA pin names to the Pulsonix Logic Name fields again in the same manner as the original import. Rather, where possible, it will switch the Pulsonix pin name values, e.g. "A3", mapped to the Pulsonix symbol pins by comparing the FPGA pin names in the revised pin data file against the previous set of FPGA pin names held in the Logic Name fields of the Pulsonix part. This means that the logic representation in the schematic symbol can remain constant even though the pin out for the FPGA has changed. Only FPGA pin names that are unique will switched in this way, those that are common to multiple Pulsonix pins will be loaded using the same method as the original import.

Reloading a revised FPGA Part into a design

Pin out modifications loaded into an FPGA part must be subsequently applied to the schematic into which a component using the part has be loaded. This can be done using **Reload from Library** or **Replace Part** (depending on whether a new version of the Pulsonix part was created for the revised FPGA). A component that uses an FPGA part will be recognised as such and treated differently when mapping the pins (and their connected nets) to the revised Part.

Normally a pin that is connected to a net is mapped to the new part using the Pulsonix pin name. For FPGA parts, the net pin will be mapped to the new part using the FPGA pin name, i.e. the Logic Name field on the Pulsonix part pin, thus if the FPGA pin out has changed on the new part although the schematic symbol representation remains constant, the net connectivity will change accordingly. For example, if the pin IC1.A3 (Logic Name:IO_L16P_2) is connected to net IOB, after the Reload the pin IC1.A5 (Logic Name:IO_L16P_2) could be connected to net IOB, with the Logic Name IO_L16P remaining the constant factor.

The connectivity changes caused by the FPGA Part Reload/Replace in the schematic will be propagated to the PCB layout as normal by the **Synchronise Design** command.

It is also necessary to Reload/Replace the FPGA part in the PCB design so the revised FPGA pin out is reflected on the component pads. However, unlike in the Schematic, the Reload/Replace will map net pins in the normal manner as the connectivity changes have already been applied via **Synchronise**.

The sequence of **Import Pin Data**, **Part Reload/Replace**, and **Synchronise** may be repeated as many times as there are revisions of the FPGA design.

Additional Pin Swap information for a FPGA Part

During PCB layout some pin swaps may occur. The Pulsonix **Back Annotation** report shows what pin swaps have been made. For an FPGA part, this will additionally show the pin Logic Name representing the FPGA pin name as a guide to making the equivalent change in the FPGA design system.
e.g.

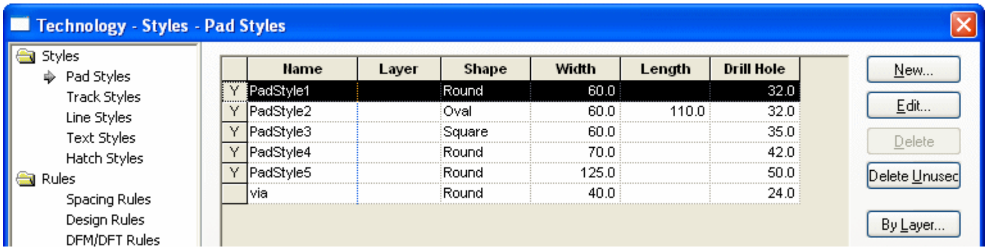
```
Swap Pad IC1.A3 (IO_L16P_2) and IC1.A5 (IO_L39P_3)
```

Technology Changes

Delete Unused command

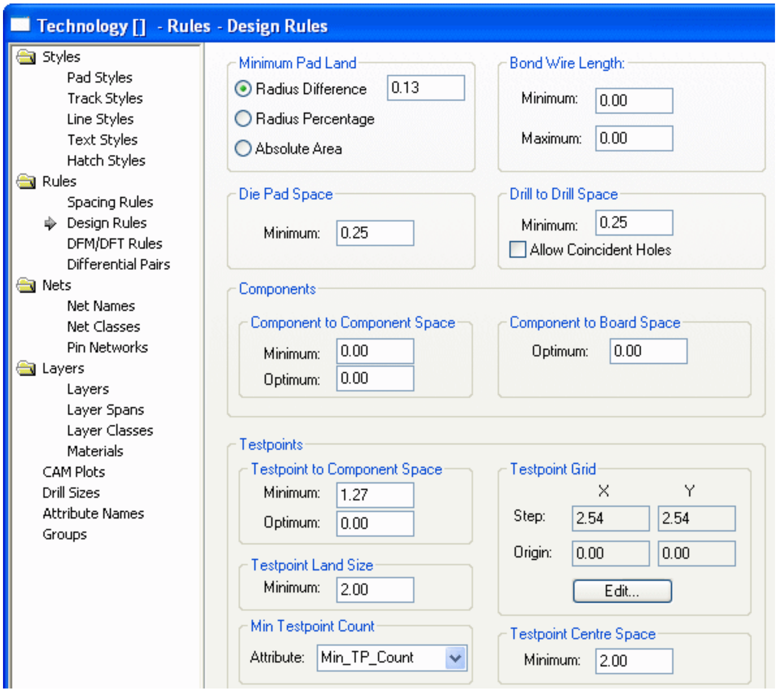
There is a new check box in **Technology** dialog to **Delete Unused** styles from the page.

Available on Pads, Tracks, Text, Line, Hatch styles, Net Names, Net Classes, Pin Networks, Layer Spans, Layer Classes, Materials, Attributes and Groups pages.



Design Rules Page

In Version 5.0, a new tab has been added to the Technology dialog. **Design Rules** contains some new rules and takes some existing functionality to declutter other pages of the Technology.



Moved Features

Testpoint Grid has been moved from **DFM/DFT Rules** page.

All other rules on this page (other than the new ones) have been moved here from the **Spacing Rules** page.

Die Pad Space – used when using the Chip Packaging Toolkit option. This is only available in the footprint editor and defines the minimum distance between die pads.

New Features

Testpoint to Component Space – used when testpoints are added to the design.

Testpoint Land Size – used when testpoints are added to the design.

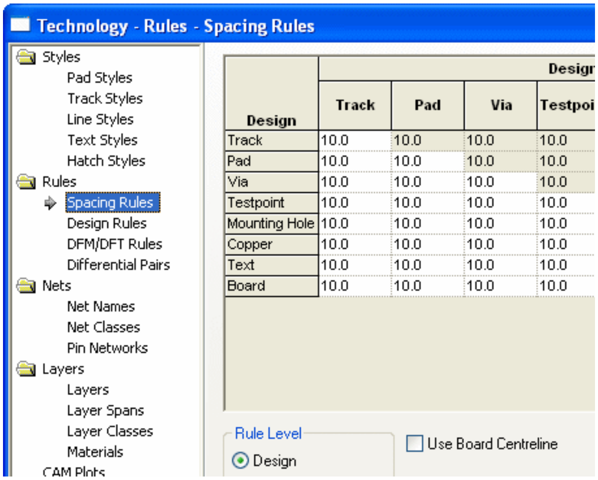
Min Testpoint Count - new rule, used in Testpoint analysis; see later in this section under *Testpoint Analysis*.

Testpoint Centre Space – new rule, used in Testpoint analysis; see later in this section under *Testpoint Analysis*.

Spacing Rules Page

New Features

Use Board Centreline has been added. When checked, it is used to ignore the real width of all board outlines. Checking would then be done up to the centreline of the board outline shape.

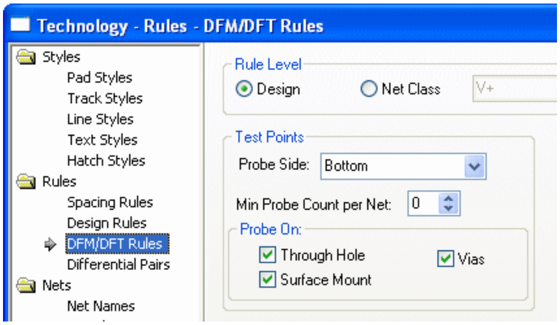


DFM/DFT Rules Page

New Features

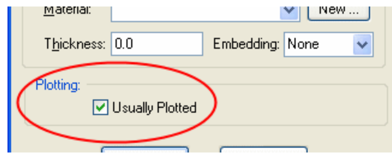
Min Probe Count per Net – used in Testpoint analysis; see later in this section under *Testpoint Analysis*.

Probe On - used in Testpoint analysis; see later in this section under *Testpoint Analysis*.



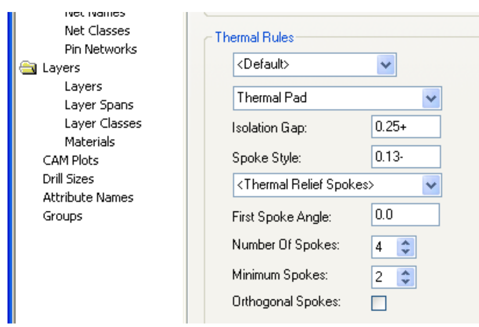
Layers Page – Usually Plotted Option

In the **Technology** under **Layers**, the new **Usually Plotted** check box under **Plotting** defines if the current layer would normally be plotted, if you uncheck this option, a plot will not be generated for this layer, or appear in the list of layers requiring a plot. It also will not appear in the **Layer Stack Preview**. It does not prevent you from manually creating a plot.

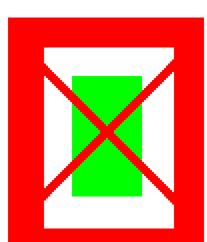


Thermal Rules - Orthogonal Spokes

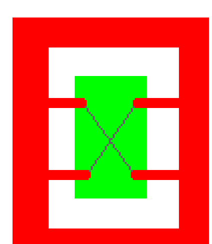
When the **Orthogonal Spokes** check box is checked it causes spokes to be orthogonal to the pad shape. This has no effect on round pads, on other shapes the spokes take the shortest distance across the gap from the same point where the angled spoke would have started. When unchecked, the spokes simply follow the line from the pad centre



The pads will be displayed like this:



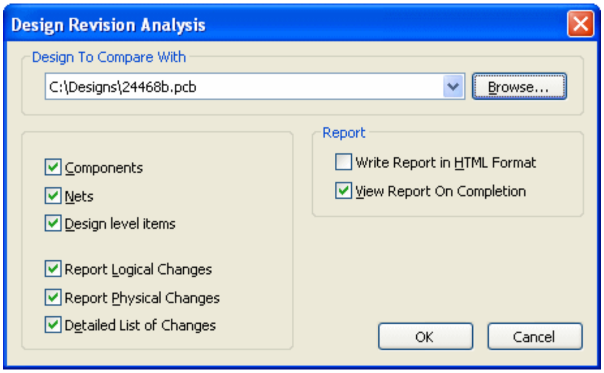
‘Normal’ spokes



Orthogonal Spokes

Design Revision Analyser

The **Design Revision Analyser** is used to compare two 'versions' of the same design to check for significant differences or changes between them. It can be used for example to help you identify what might have been changed between two revisions of a board when the original revision has been signed off for EMC purposes.



Your current open document revision is known as the 'Master'. With this document (Schematic or PCB) open, choose the **Design Revision Analyser** option from the **Tools** menu.

You then need to select the second document, the one to compare against. This is known in the report as the 'Other' document.

Choose from the options on the dialog to specify what checks should be done, and how you want the report to appear.

Component Logical Changes

This checks for components being added, deleted or renamed, and component values being added, deleted or modified, in the same way that the **Synchronise Designs** option does when comparing a PCB against its corresponding Schematic.

Net Logical Changes

Checks for nets (and net 'nodes') being added, deleted or renamed, in the same way that **Synchronise Designs** does when comparing a PCB against its corresponding Schematic.

Component Physical Changes

Checks for corresponding components being moved, rotated, flipped, or having a different footprint or pads in different locations, or pads with different styles.

Net Physical Changes

Checks for changes in total track length, track segment positions and width changes, total copper area, copper segment positions and width changes, via positions, via layer spans, via styles.

Design Level Physical Changes

Checks for changes to the total area of boards, the 'segments' of all boards, the area and segments of design level (not net) copper, and free pad positions, layers and styles.

The Report Output

With the **HTML Format** check box ticked, the report is generated as an HTML (web page) file, which should then open automatically in your default web browser if you also have the **View Report** check box ticked. This HTML report includes links between the sections to make it easy for you to navigate up and down the report.

With the **HTML Format** check box unchecked, the report is written as a plain text file in the normal way that other reports are written. This will open in the application that is assigned on your computer to handle text files.

Dynamic Attribute Value Substitution

The attribute substitution capability has been extended to allow dynamic attribute value substitution based on the value of another attribute. A 'nested' attribute substitution value can now be indicated by adding an extra 'Attribute Substitution Character', e.g. %%% rather than the usual %%.

This can be useful for achieving attribute switching based on the value of another attribute such as the built-in <Current Variant>.

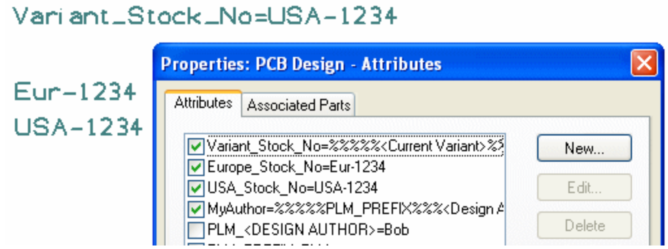
For example, in a design that is defined to have two variants; **USA** and **EUROPE**, the following **design level attributes** could be defined:

USA_Stock_No="USA-1234"

EUROPE_Stock_No="EUR-1234"

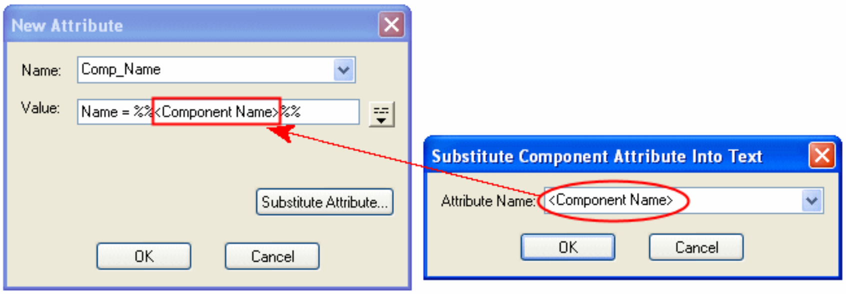
Variant_Stock_No="%%%%%<Current Variant>%%%_Stock_No%%"

Then, if the **Variant_Stock_No** attribute is displayed in the design, it would show the appropriate stock number value for which ever variant was currently active at that time. In the example below, the USA variant is shown and the values from the USA_Stock_No value plugged in.



Attribute Substitution check box

On all edit attribute value dialogs, there is now a **Substitute Attribute...** button. Use this to pick from a list of attribute names the attribute you want to substitute into the attribute value being edited. This prevents you needing to remember the substitution character. The substituted string will be pasted and therefore will replace any text already selected in the value being edited.



This can be used for Text Callouts for example.

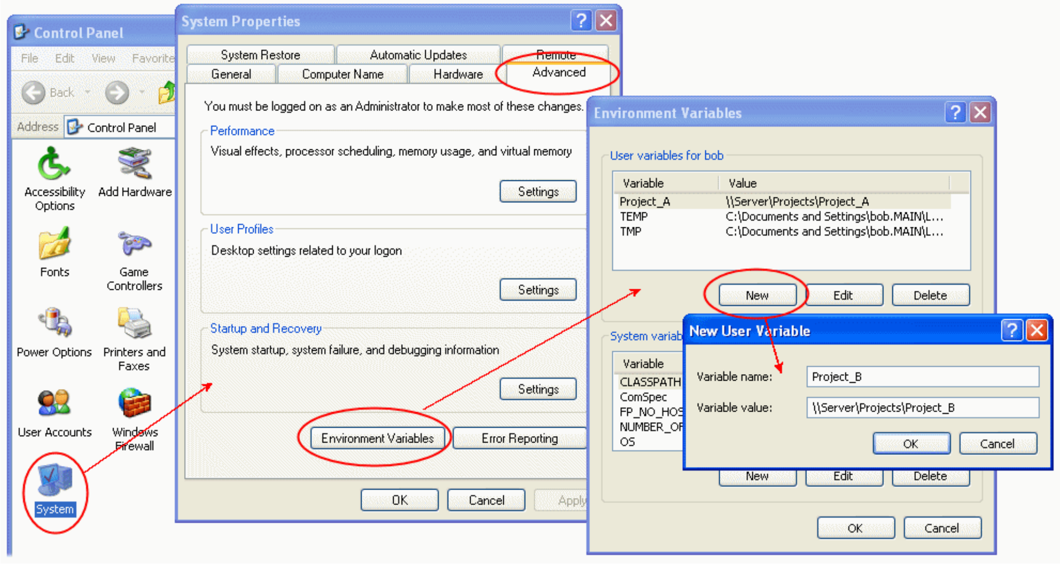
Environment Variables in Paths

You can now use environment variables in paths in the folders dialog for use with the Library, Technology, etc. paths. This means that by switching environment variable values, you can switch all or some paths in one easy step.

Adding Environment Variables

Before being able to use environment variables in Pulsonix you will need to define them first. To do this you will need to edit the environment variables in the **Control Panel**.

From the **Control Panel, Systems** page, click on the **Advanced** tab. Select the **Environment Variables** button. This page displays user variables and system ones. You should add local user variables.

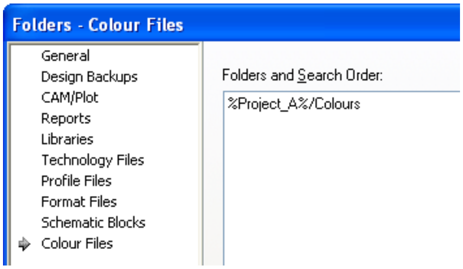


Use the **Add** button to add a new **Variable name** and **value**. The variable name will be the name used in Pulsonix to change the environment, the value will be the path for that environment.

Click the **OK** button to add the variable. If you wish to edit it, select the name in the list and click the **Edit** button.

Using Environment Variables in Pulsonix

In the **Folders** dialog in Pulsonix, choose the folder type to change and click the New or Edit button. Type in the name of the environment variable. Use the **%** character to delimit the variable, for example: %Project_A%/Colours

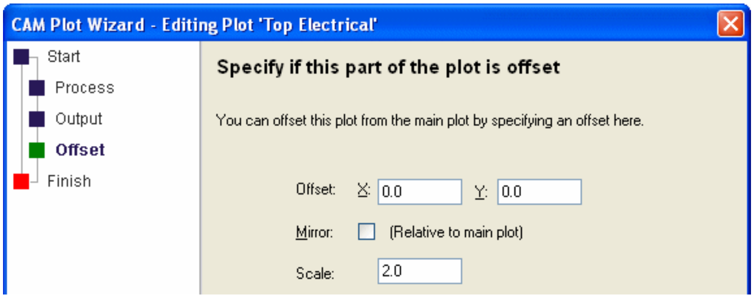


Switch file locations using environment variables, this can be done using the Control Panel or by using system scripts or batch files. If different locations use different servers or paths, this customisation can be achieved by changing one or more Windows environment variables, for example "ECAD_SERV" and "ECAD_ROOT". All Pulsonix paths or filenames are able to store and resolve such variables. These variables could contain UNC paths to avoid the use of fixed drive letters.

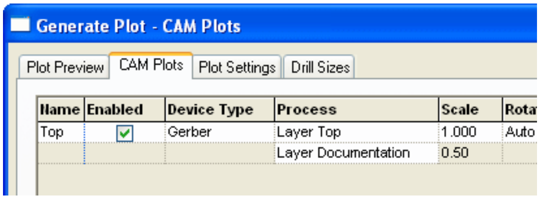
CAM Plot Changes

Scale Combined Plots

You can now scale each part of a combined plot relatively. Previously this was fixed so that the 'master' plot had the scale factor applied. The plot scale is relative to the 'main' plot scale and not relative to it's own true size.

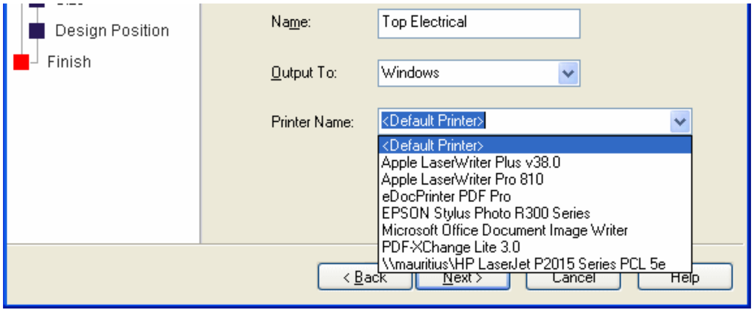


Once the combined plot has been scaled, the main CAM Plot tab page will show the scale in the cell under the main plot scale. This is only shown when plot scale is not 1.

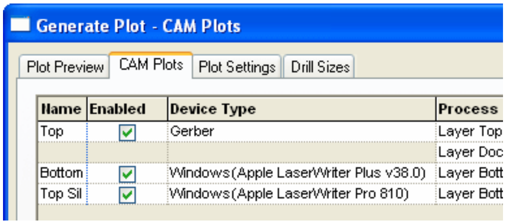


CAM Plot to named Windows Printer

In the plot wizard you can specify a specific printer to use by selecting the **Printer Name:** from the drop down list. By leaving the list set to **<Default Printer>** it will use the printer currently selected in the Setup dialog. You are warned if the selected printer is not available when you print.



On the CAM Plot tab of dialog, the Windows printer name is shown next to the Windows printer, like Windows (Apple LaserWriter Plus 810) or Windows (eDocPrinter PDF Pro) for example so you can see which one is selected.



The settings for the Windows printers are taken from the **Windows setup** page on the **Plot Settings** tab.

Controls in Combined Plot

You can now independently specify the style and variant on a combined plot. So, for example, you could plot the sheet outline in **artwork**, but the design as **outline** on the same plot.

Formats Option to Named Folder

In the **Cam Plot Wizard** under **Process Type** page, the old **Insert Output Into Plot Report** check box has been replaced with a **Output Location** list box containing three options:

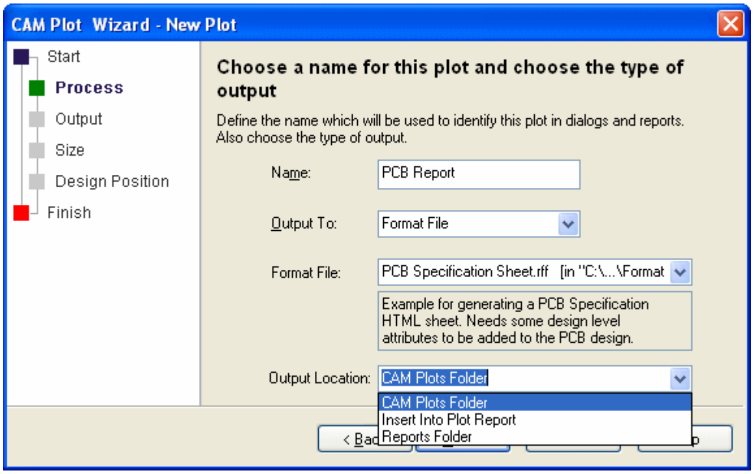
CAM Plots Folder

Insert Into Plot Report

Reports Folder

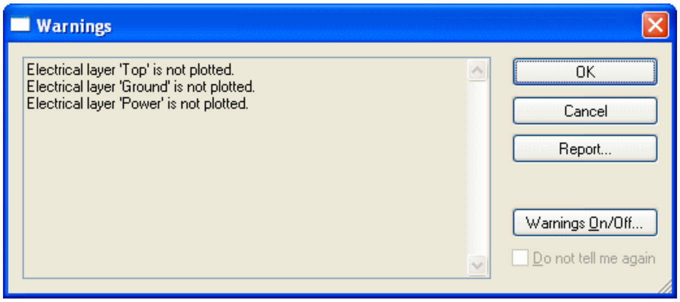
This allows you greater flexibility when running report files from the CAM Plot option. It also allows you to append reports into the main CAM Plot report so that you have one single report.

The **Reports** folder is the new option allowing the format file plot output to be saved with the other reports. Note, it will always take the plot name (not the format file name).



New Warning if Electrical Layer is not plotted

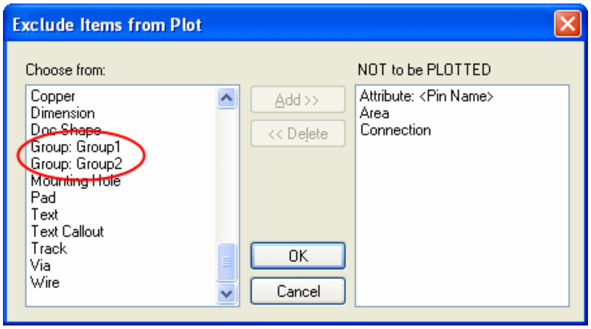
On entry to the CAM Plot dialog, it now warns you if an electrical layer is not being plotted at all. This is a warning if the electrical layer is flagged in the **Technology Layers** dialog as **Not Usually Plotted**.



Once in the **CAM Plot** dialog, a plot can still be generated using the **CAM Plot Wizard**. If the option is exited and re-entered, the warning for this layer will no longer be displayed.

Exclude Items By Named Group

In the **CAM Plot Wizard**, you can now exclude items by named **group**.



Report Maker

New/Changed Commands

List of Boards

Used on a PCB Design, this lists each board in the design. For each board you can report the following:

Area

Extents

Is Circle – this is a new command and allows you to test for the shape being a circle. It reports a value of True or False, you can then test this condition using the **IF** command. For example, if the board is a circle (True), report the Circle Radius.

Is Selected – this reports True or False depending on whether a board is selected in the design or not, you can test this condition using the **IF** command.

Length – this gives the total length of the perimeter centre line around the board

List of Segments

This new command lists each segment of the board.

Is Arc - If writing out a shape segment that is an arc, this can test the arc condition and reports True or False. You can then test this condition and report the segment differently, output the arc centre for example.

Start Position

End Position

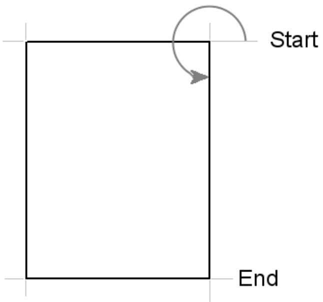
Arc Angle - when testing **Is Arc** for example, you can then use **Arc Angle**, **Centre** or **Radius** instead of the **Start** and **End** positions as alternatives.

Arc Centre

Arc Radius

Is Arc Clockwise – The command reports True or False (True means the arc is clockwise). On the response of the this test, you can run a command, **Text: Clockwise** or **Counter-Clockwise** for example.

Line Angle - This reports the angle of the end points relative to the start point of the segment. In the example below, the angle reported for this particular segment would be 270 (degrees).



Length - this reports the length of the segment (if it is on an arc, the distance around the arc).

Is Selected – this reports True or False depending on whether a segment is selected in the design or not, you can test this condition using the **IF** command.

List of Cutouts

List of Cutouts works through each board cutout within the current board (including special board cutouts, defined as areas). For each cutout, you can report the following:

Area

Is Circle – As with **List of Boards**, this is a new command and allows you to test for the shape being a circle. It reports a value of True or False, you can then test this condition using the **IF** command. For example, if the board is a circle (True), report the Circle Radius.

Is Selected – this reports True or False depending on whether a cutout is selected in the design or not, you can test this condition using the **IF** command.

Length – this reports the length of the perimeter centre line of the cutout.

Plated Through – Area Cutouts can be plated, this command will report a True or False status, you can then test this condition using the **IF** command.

List of Segments – A new command, see above.

List of Attributes

List of Attributes is now available for **Boards**. You can report the Attribute Name, Value, X Coord, Y Coord and (X, Y) coordinates.

List of Drill Holes

List of Drill Holes is used in **List of Drill Sizes** to report the position of each hole of that size. Within this list you can report the following:

Drill Diameter

Drill Shape – This will report **Round** unless a special drill hole shape has been used.

Position

Length - only used for special drill holes where a hole length is required.

List of Segments - as above, use “IF Drill Size | Drill Diameter Is Equal To Special” to spin through the drill shape and output the individual segments.

Type

Type is now available on **List of Testpoints**. The report results are **part**, **doc symbol**, **star point**, **pad**, **mounting hole**, **via**, **component pad**, **component mounting hole** and **component via**.

Units Name

Units Name reports the name of the current coordinate units.

Selection

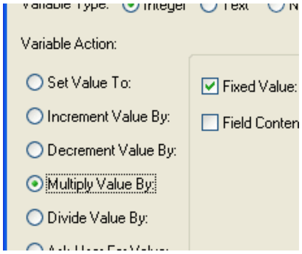
The new **Selection** command allows you to select items in the design. This has sub-commands for **Select Item**, **Deselect Item**, **Deselect All**, **Select Item If Visible** and **Deselect Item If Visible**. Useful for difficult selection situations, for example to select all components on side top with a certain attribute. Inserted reports in the design ignore the select command.

Is Selected

Is Selected can now be used on selected nets in the design. This allows you to report details of the selected nets.

Changed Functionality:

Variables – The Variables dialog now has **Multiply By** and **Divide By** as well as increment and decrement as possible actions available.



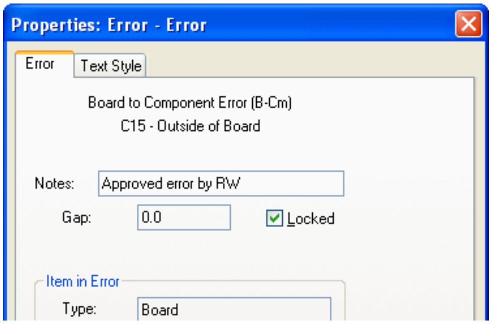
Pin Count Report

A new **Pin Count** report is supplied. This provides you with details of the number of pins, testpoints, etc. and in particular, reports the pin limit and how close the design is to that limit.

```
Component count 139
Component Pad count 509
Design Pad count 916
Pin Limit Count 1425
Unlimited PCB Pin Limit
```

Notes Field for Error Markers

There is a **Notes** field which can be edited in the **Properties** of an error.

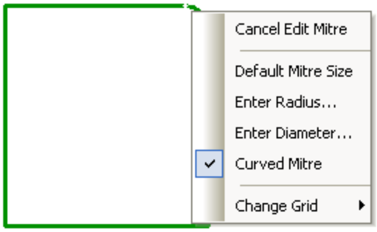


This is especially useful for reporting a decision for locking an error marker when signing off a design with errors still in it for example. The Notes field is available in both Schematic and PCB designs.

These notes can be reported in the **Report Maker** using the **Notes** command on **List Of Errors**.

Arc Radius/Diameter Improvements

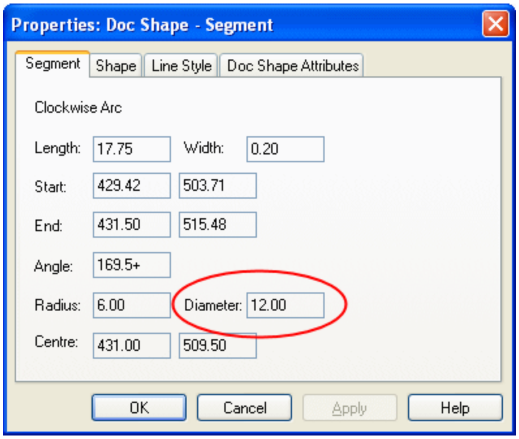
A new command, **Enter Diameter** is available whenever editing a circle, arc or fillet.



When editing a **Mitre** or **Fillet** you can now type the exact radius or diameter (for fillets) using **Enter Radius** and **Enter Diameter**. This is available on all modes of this type.

Properties of Circle & Arc

Properties of a selected **Circle** or **Arc** segment will also now show the circle **Diameter** next to the **Radius**.



New Cost Option

Access for Product Lifecycle Management (PLM) Systems

Feature Overview

Pulsonix V5.0 introduces the Pulsonix PLM product interface. The license enables PLM systems to communicate with Pulsonix using an interface mechanism and proprietary commands within the Pulsonix environment.

The initial release phase will include support for to the ***Integrate*** PLM interface system.

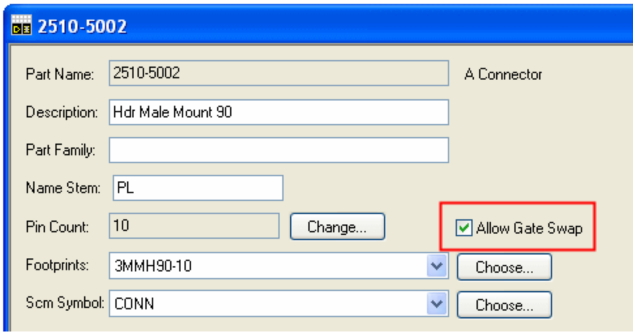
Future product releases will include additional systems support on request. If you currently have a PLM and wish to interface to Pulsonix, please contact our technical support desk or your local distributor.

Chapter 3. Schematic Options

Connector Gate Swaps

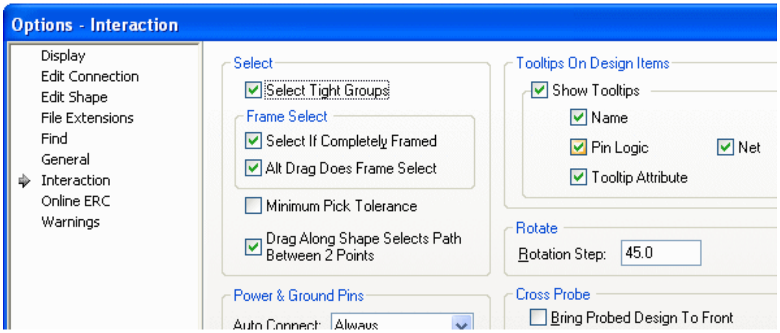
You can now perform a gate swap on a connector. You can already do pin swap, but this does not let you swap pins between components.

For connectors in the **Part Editor**, on the **Details** page, you can now set Gates to be swappable using the **Allow Gate Swap** check box..

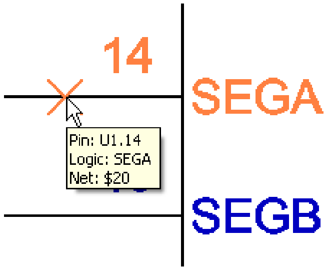


Pin Logic Tooltips

Within the **Options** dialog and **Interaction**, there is an additional tooltip switch for showing the **Pin Logic** name on selected component pins.

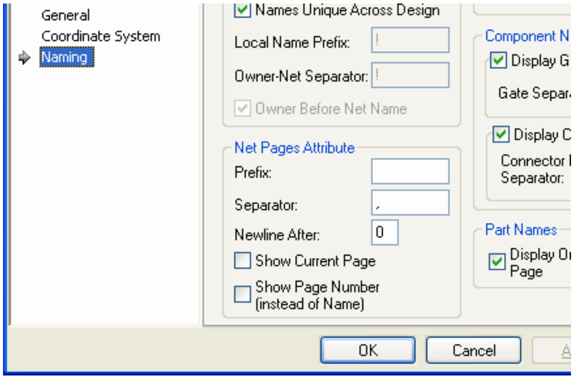


The **Logic:** name is shown on the tooltip when the cursor is hovered over the pin.



New Line in Net Pages Attribute

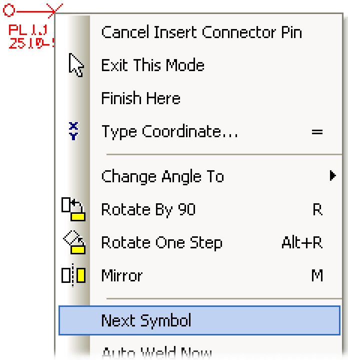
For additional formatting of the **Net Pages** Attribute, from within the **Design Settings** dialog and **Naming**, you can now set the **Newline After** parameter to a value greater than 0. This gives a new line after that number of pages (specified in the value), i.e. 3 will give you a new line after Page1,Page2,Page3 and then after Page4,Page5,Page6.



Alternate Symbols for Connector Pins

Connector Parts can now have alternative symbols defined for them, similar to gates in normal Parts.

The alternative symbols can be selected using the **Next Symbol** option from the context menu when using the **Insert Connector Pin** option. You can also select any alternative symbols using **Properties** of the connector once added to the design.

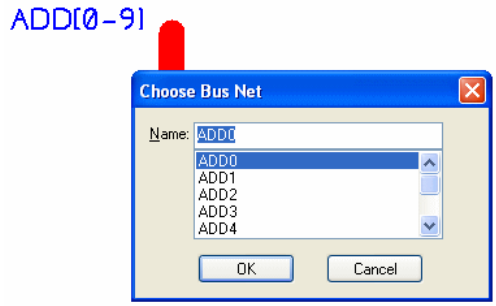


Selection of Buses

Shift-select will now expand to select all connected buses.

Mark Net on a Closed Bus

You can now use the **Mark Net** option on a closed bus. When the bus is selected you will be prompted to select which net to mark using the **Choose Bus Net** dialog.



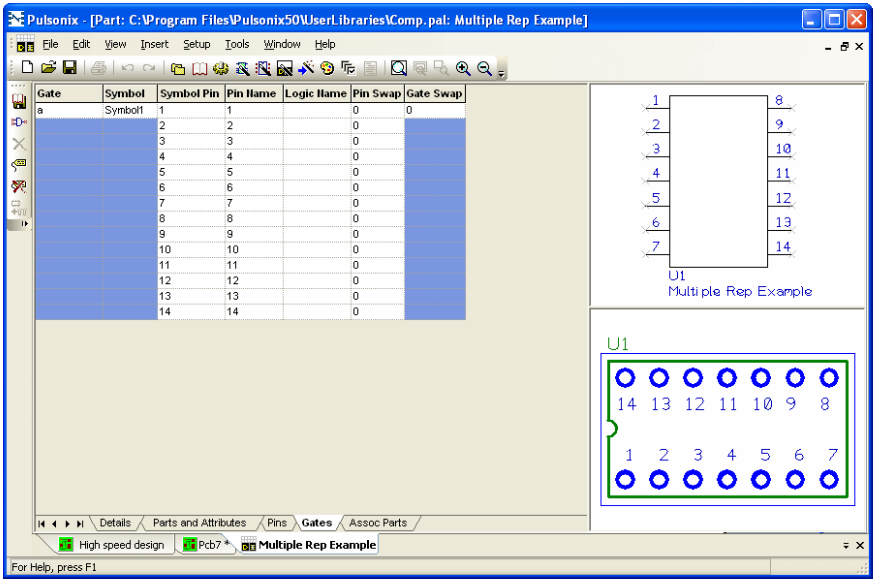
Multiple Symbol Representations in Parts

You can now have multiple Symbol Representations in Parts.

A Part Representation is a set of gates which can be added to a schematic. A Part can have more than one representation. For example, a pack of four gates could be represented by a single *top level* symbol, or as four individual gates. Each representation is independent of the others. There is no cross checking, so representations do not have to conform to a hierarchical structure, although this is likely to be the case. The first representation is the default, used when a component is first added to a schematic.

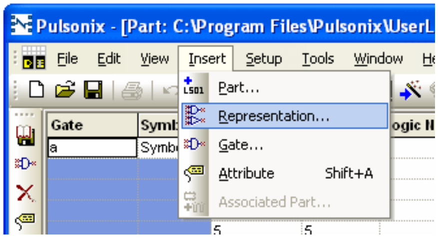
Adding Part Representations

From within the **Part Editor**, add your normal Symbol(s) to the Part.

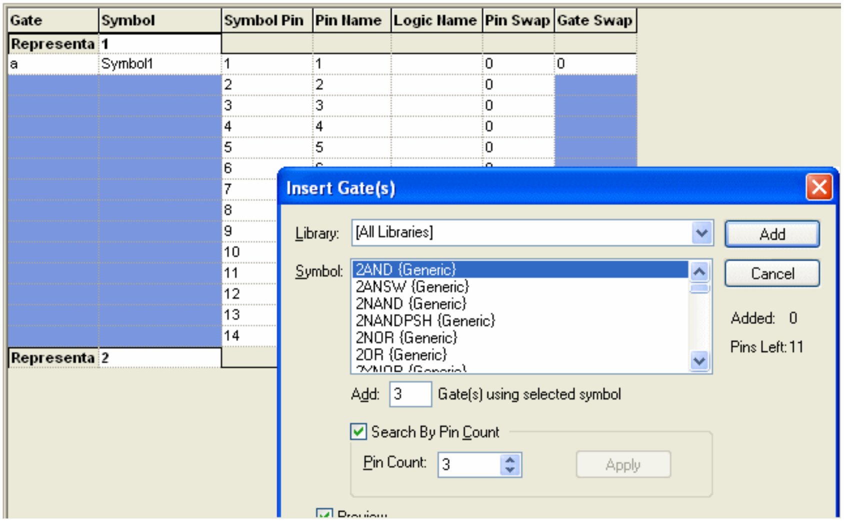


Select the **Gate** to add the representation to.

From the **Insert** menu select **Representation**.



A 'blank' Representation is added to the Part. At the same time, the **Insert Gates** dialog is also displayed.



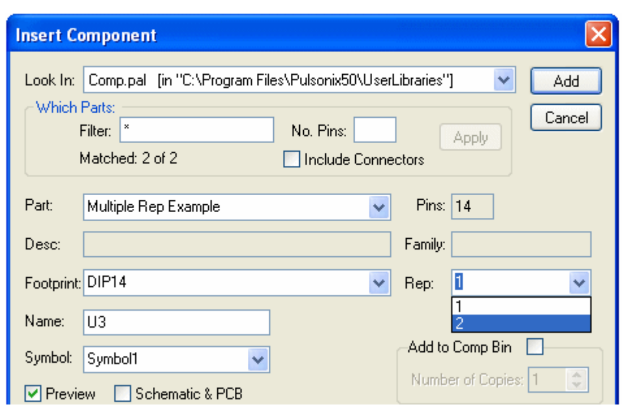
From the dialog, select the **Symbol:** (2AND name or whatever yours is) and the number of gates you require and click the **Add:** button.

The Representation now shows the four 'internal' gates of the main symbol.

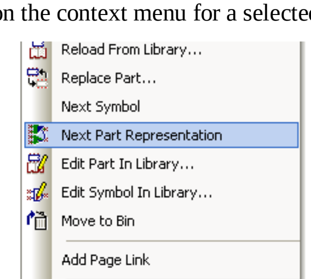
Gate	Symbol	Symbol Pin	Pin Name	Logic Name	Pin Swap	Gate Swap
Representa	1					
a	Symbol1	1	1		0	0
		2	2		0	
		3	3		0	
		4	4		0	
		5	5		0	
		6	6		0	
		7	7		0	
		8	8		0	
		9	9		0	
		10	10		0	
		11	11		0	
		12	12		0	
		13	13		0	
		14	14		0	
Representa	2					
a	2AND	1	1		0	0
		2	2		0	
		3	3		0	
b	2AND	1	4		0	0
		2	5		0	
		3	6		0	
c	2AND	1	7		0	0
		2	8		0	
		3	9		0	
d	2AND	1	10		0	0
		2	11		0	
		3	12		0	

Using Parts with Multiple Representations

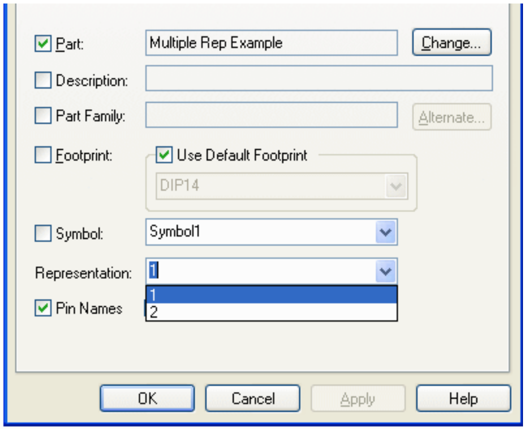
When adding to the design, you can use the **Insert Component** dialog to select the alternative representation of the Part.



Once added, you can change the **Representation** using the **Next Part Representation** button on the context menu for a selected Component.



You can also change the **Representation** using the **Properties** dialog.



New Cost Options

DxDesigner Import

Pulsonix V5.0 introduces the import filter cost option for the Mentor DxDesigner format of Schematic designs and Schematic Symbols and Parts libraries.

This is available as a cost option on a license. Please contact your local sales office for more information.

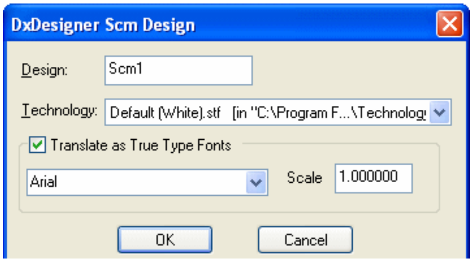
Other variants of the DxDesigner format are also available for import, for example, the older ViewLogic ViewDraw.

Import Designs

Use the **File** menu and **Open** dialog to import Schematic Designs. To import DxDesigner format files, you must open the sub-folder **\sch** and select the **top level design file** with the file extension **.1** Note: the Data Transfer Wizard cannot be used to import designs of this type.

You are presented with the **Import** dialog confirming the format selected. Type the **Design** name if you wish to change the one presented. Under normal circumstances, the Technology file name will not be required.

Check the **Translate as True Type** Fonts box if you wish to import the DxDesigner fonts as True Type fonts. You are also able to rescale the fonts during import if required using the **Scale** box (during import only and not afterwards in the design). Click **OK** to proceed.



Importing Libraries

The Pulsonix DxDesigner Import option is looking for the **.1** file extension when it imports Schematic symbols. Under the normal file structure of DxDesigner files, the symbols are stored in the **\sym** folder.

Use the **Import** button on the **Library Manager** dialog to import the schematic symbols.

The same schematic symbol file is used when creating Parts. The symbols or the schematic designs do not contain footprint information, this would need to be added from a separate source.

Zuken System Designer SCM Import

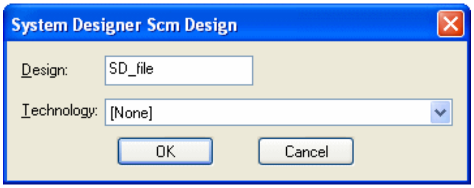
Pulsonix V5.0 introduces the import filter cost option for the Zuken System Designer format of Schematic designs and Schematic Symbols and Parts libraries. The Zuken SD EDIF format should be exported from System Designer for import into Pulsonix. The normal Pulsonix import mechanisms of **Open** and the **File Transfer Wizard** are used.

This is available as a cost option on a license. Please contact your local sales office for more information.

Import Designs

Use either the **Data Transfer Wizard** or the **File** menu and **Open** dialog to import schematic designs. To import System Designer format files, you must open the design files with the file extension **.eds**

You are presented with the **Import** dialog confirming the format selected. Type the **Design** name if you wish to change the one presented. Under normal circumstances, the Technology file name will not be required.



Import Libraries

Use either the **Data Transfer Wizard** or the **Import** button on the **Library Manager** dialog to import the schematic symbols.

The same schematic symbol file is used when creating Parts. The symbols or the schematic designs do not contain footprint information, this would need to be added from a separate source.

Pulsonix Spice Changes

There are minor changes to the Pulsonix Spice product. The changes include some new models, examples and scripts (detailed at the back of this document in Appendix A). There are also changes to the simulator interface and simulator engine itself to introduce improvements.

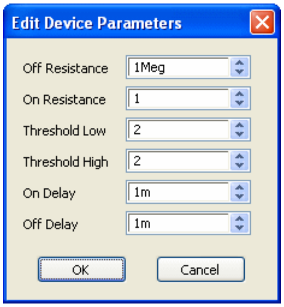
Pulsonix Interface Changes

New Parameterised Device

A new parameterised device is available:

Delayed Switch

Use the **Part Browser**, with the **Spice Category** set, or use **Insert Component** to add the part named **Delayed Switch** device. Editing the device with <**F7**> will bring up a dialog with 6 parameters:



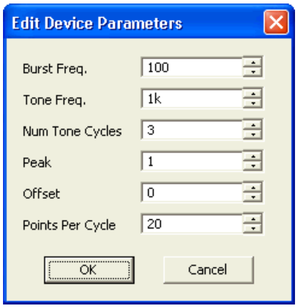
New Signal Sources

Two new Signal Sources are available:

Sine Tone Burst

Generates a sequence of sinusoidal bursts with a user defined number of cycles per burst, burst frequency and tone frequency.

Use the **Part Browser**, with the **Spice Category** set, or use **Insert Component** to add the part named **Sine Wave Burst Voltage Source**. Editing the device with <**F7**> will bring up a dialog with 6 parameters:

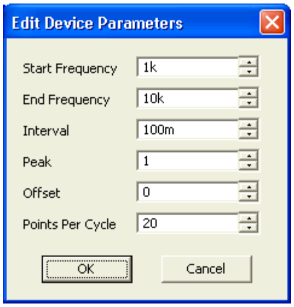


Parameter	Description
Burst Freq.	Burst frequency
Tone Freq.	Frequency of the sinusoidal tone
Num Tone Cycles	Number of sinusoidal cycles in each burst
Peak	Peak voltage
Offset	Offset voltage
Points Per Cycle	Minimum number of time steps in each sinusoidal cycle. Increasing this number will improve the accuracy of the simulation at the expense of simulation speed

Swept Sine

Generate a sinusoidal signal with linearly increasing frequency.

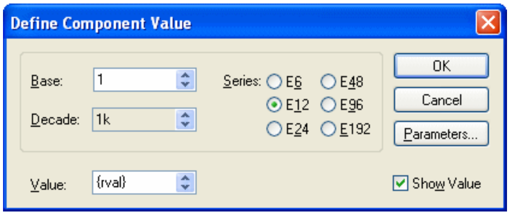
Use the **Part Browser**, with the **Spice Category** set, or use **Insert Component** to add the part named **Sine Wave Sweep Voltage Source**. Editing the device with <**F7**> will bring up a dialog with 6 parameters:



Parameter	Description
Start Frequency	Starting frequency
End Frequency	Frequency at the end of the ramp Interval Time taken to ramp from start frequency to end frequency
Peak	Peak voltage
Offset	Offset voltage
Points Per Cycle	Minimum number of time steps in each sinusoidal cycle. Increasing this number will improve the accuracy of the simulation at the expense of simulation speed.

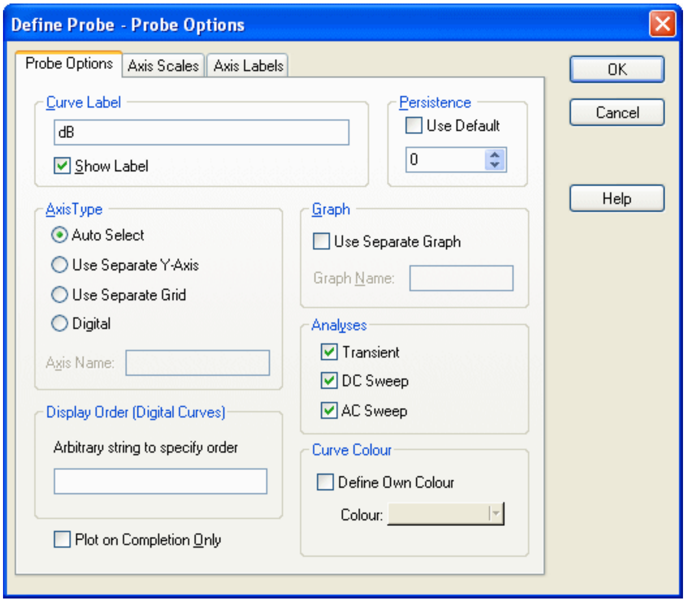
Editing Passive Devices

You can enter Expressions for Passive devices directly. You can add curly brackets in normal dialog, the need for Shift-F7 is no longer required. This takes the parameter from this value at run time and uses it.



dB and Phase Fixed Probes Improvements

The **Edit Probe** dialog for dB and Phase fixed probes has been improved and now has full editing capabilities.



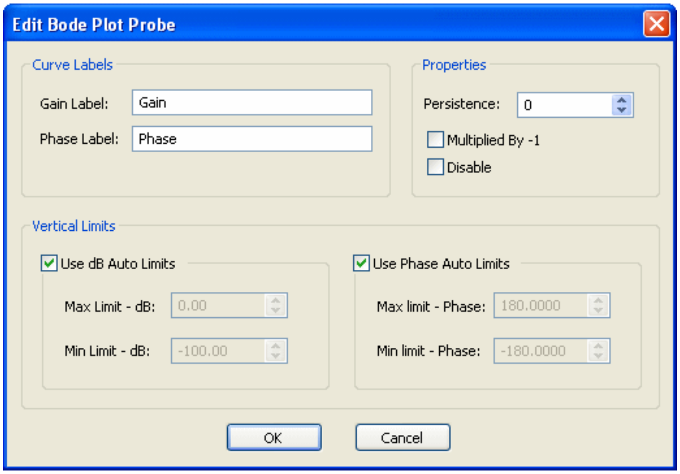
There is a new **Default** box in the **Persistence** section. If **Use Default** is checked, the value used will be the value set in the general options in the simulator. This is changed using the **File** menu and **Options**, **General...** Select the **Graph/Probe/Data Analysis sheet.** Under **Fixed probe** global options the **Default** persistence setting is the value used here when default is checked.

Curve Colour - If **Define Own Colour** is checked, the colour will be chosen automatically in a manner that tries to minimise duplicate colours on the same graph. Alternatively uncheck this box then use the **Colour** selection select a colour of your choice. In this case the trace will always have the same colour.

Display Order (Digital Curves) - Enter a string to control display order for digital curves. Normally digital curves are ordered according to their title. The value supplied here will be used instead if not empty. To force the curve to be placed above other curves that don't use this value, prefix the name with '!'. The '!' character has a low ASCII value. Conversely, use '~' to force curve to be displayed after other curves.

Bode Plot Improvements

db and Phase limits have been added under **Vertical Limits** and an additional check box to **Disable** Properties has been added.



Bode Gain & Bode Phase plots

New probes have been added for Bode Gain Plot, Bode Phase Plot and Bode Phase Plot (X -1). The Bode Phase Plot (X -1) probe selects a phase probe with 180 degree phase offset.

db and Phase limits have been also been added to this probe (see above).

If you want even more bode plot options you can use the new individual dB and phase bode plot probes.

Note: unlike the standard Bode plot probe the above only create a single curve either dB or phase as specified, but you have the full range of options available to all other probes.

Simulator Interface Changes

The section below documents specific changes to the simulator interface:

Save & Restore Session

This saves the current schematics, graphs and simulation data for later retrieval. Schematics that have been edited will be saved in their current state even if the original file has not been updated.

This feature allows the Pulsonix Spice simulator to be shut down and the machine switched off without the need to save work.

Gain and Phase Margin Functions

Two new functions - GainMargin and PhaseMargin - have been added. These can be used with **Performance Analysis** and **MC Histograms**. These functions are implemented using the script based user defined function system.

Stacking Curves

This feature will separate curves so that they all have their own grid. Two options on the **Curves** menu are available for this, one operates on all curves in a graph (**Stack All Curves**), the other operates only on selected curves (**Stack Selected Curves**).

Stimulus Specification

A new feature has been added to voltage and current sources allowing much more sophisticated stimulus specification. This uses the PSpice derived REPEAT and ENDREPEAT syntax allowing PWL definitions to be specified with a repeating sequence. The original PSpice syntax has been extended to allow the specification of Sines and Pulses within the PWL sequence. This allows wave-shapes such as tone-bursts to be easily created. Further the Sine specification used in this manner has some additional parameters not available in the standard Sine source. These are “minpoints” which specifies the minimum number of points that may be used to fabricate the sine signal, and “ramp” which specifies a frequency gradient. This allow the implementation of swept sine sources.

The new feature uses this syntax:

`Vxxx n1 n2 PWLS` *specification*

specification will accept the PSpice PWL syntax using REPEAT, ENDREPEAT and FOREVER along with the scaling parameters. Note: currently the PWL syntax (as opposed to PWLS) only accepts the fixed standard SPICE PWL syntax.

‘M’ for subcircuits

The ‘M’ multiplier parameter may now be applied to subcircuit instances. This is implemented by scaling all devices within the subcircuit and works for all analysis modes including noise. There is therefore no performance penalty specifying this parameter. However, not all devices support it and an error will be raised if a subcircuit containing unsupported devices is instantiated with M not equal to unity. Currently unsupported devices include all digital devices and the lossy transmission line. All other analog devices support this including the new PSP and Hicum models.

In addition ‘M’ may be applied individually to all devices except those exceptions mentioned above.

Important Note: because ‘M’ is now an implicit subcircuit instance parameter, its use as a regular parameter can introduce problems. For this reason you cannot use ‘M’ as a regular parameter unless it is also defined with the subcircuit definition, e.g.:

```
.SUBCKT subname 1 2 3 params: M=1
```

If you use the above, ‘M’ as a multiplier will be disabled and instead it will treated as a regular parameter. If the definition of M in the .SUBCKT line is omitted, an error will be raised if any attempt to access ‘M’ in an expression is made.

SCRIPTS in Netlists

The Pulsonix Spice script engine has now been incorporated into the simulator. This allows a script to be called from the new simulator statement **.post_process**. This will call a specified script when an analysis successfully completes. In addition, the script code can be embedded in the netlist using the .FILE/.ENDF syntax.

The above will work even if the simulation is called in non-GUI mode.

SOA Enhancements

The **Safe Operating Area** (SOA) feature has been extended as follows:

1. Limit values may now use parameter expressions
2. A per definition de-rating factor has been added and this can be defined using a parameter expression.
3. It is possible to specify limits on the mean value during the course of the run.
4. A report can be generated showing the margin within which all devices are operating even if they did not violate the absolute limits. This information is available via a script function if required.
5. A .option setting can specify start and end times during a run over which SOA is active.

Expressions in Graph Annotations

You can put expressions enclosed in curly braces in graph annotation labels as well as plain text and angle bracket enclosed symbols. This permits more complex live displays based on cursor positions or curve marker positions. A simple example might be to display 1/Time in an x-axis cursor display to show frequency instead of period.

Histogram Stepped Style

Histograms are now displayed in a stepped style which shows the bin widths more clearly. This can be disabled in the **Options General** option if required.

Small Graph Cursor

The small cursor that was available in early versions of Pulsonix Spice has been resurrected. To define this, use the **Cursors** menu and **Cursor Style...**

Additional Cursors

You can now add additional cursors as well as the standard 2. To add more, use the **Cursors** menu and **Add Additional Cursor...**. There is no theoretical limit to the number that can be added, but practically the graph becomes difficult to manage with more than about 2.

Data Point Markers

Data point markers can now be displayed. To display them, from the **Curves** menu, go to| **Show/Hide Points**.

Colour maintained for Persistence=1 Curves

When a probe has persistence set to 1, the curve displayed remains the same colour.

Freeze Curve Feature

A curve can be frozen so that it is not deleted in a persistence purge operation. That is when a fixed probe's persistence is set to non-zero, a frozen curve will not be deleted to satisfy the persistence setting.

There is no user interface for this feature, so set a curve as frozen, double click it, then in the **Properties** box double click **Frozen** and set to **TRUE**.

CSDF Import (from PSpice)

The waveform viewer can now read CSDF ASCII format data files which both PSpice and Hspice can export in this format.

BITMAP Image Export

It is now possible to export the schematic image in the bitmap formats PNG, JPG and BMP. Also, the method for writing out image files has changed. Previously this was done using the **Save As...** option from the **File** menu then select format using **Save as type**. This is now done using a new **Save Picture...** option devoted to saving exported image files. The same changes and additions have been made to the waveform viewer.

Simulation "Started" and "Complete" command shell messages

You can enable messages to be displayed in the command shell stating when a simulation was started and completed. This is off by default, to enable it type the following at the command line:

```
Set DisplaySimProgressMessage
```

Chapter 4. PCB Options

Construction Lines

Construction Lines have been added to Pulsonix. This gives you the ability to add infinite lines and circles that other design items can snap to and can be used to easily construct accurate shapes. These lines can be created using a variety of functions available to aid, fore example, the centre of a triangle or arc or creating temporary intersects that you can use for creating complex shapes. Construction lines can be added to any design layer or can be given their own layer, and their own colour.

Construction lines can be used in the Schematics editor as well as PCB designs. They can also be used in the Symbol editors and in the user defined Pad Shape Editor in PCB.

Construction lines are not selected in Frame Select and are not counted in the total board area. They are also not plotted using an CAM plot outputs.

Note: if using Construction lines in the user defined Pad Shape Editor it does not remember the construction lines after exiting; in this mode they are transient.

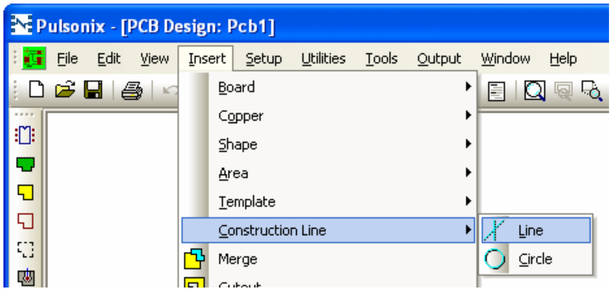
What to use Construction Lines for

Construction lines can be simply used as visible guide lines to see if items are in line, or can be used by options to add shapes, or snap items onto exact positions.

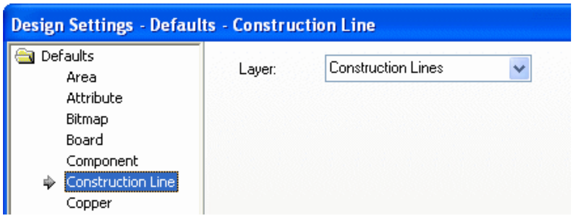
- In **Align** you can align items using a construction line.
- When using **Add Polygon** you can create a shape by using **Use Construction Regions** to merge together shapes defined by their surrounding construction lines and circles.
- When editing a shape you can create a shape by using **Follow Construction Lines** to automatically place segments exactly on construction lines or circles.
- In the **Insert Shape** functions you can snap the points in a rectangle, circle, triangle, line or polygon onto construction lines, or onto the intersection of construction lines using the **Snap To Item** option.
- The above three methods can also be used when creating cutouts in shapes, or when merging shapes together.
- You can use multiple horizontal and vertical construction lines to define a irregular matrix of points to be used as a grid in **Move** when using its **Snap To Construction Lines** option. This makes the construction line intersection points act as magnets to attract the moving item when it is near to them. If the item is not near to a construction line it is gridded normally.
  The snap distance that is used to determine if the moving item is close to a construction line, uses the value defined in the **Placement Sites** option (**Settings** menu and **Design Settings**, **Placement Sites**).
  Using construction circles for this enables you to define irregular polar grids.

How to Insert Construction Lines

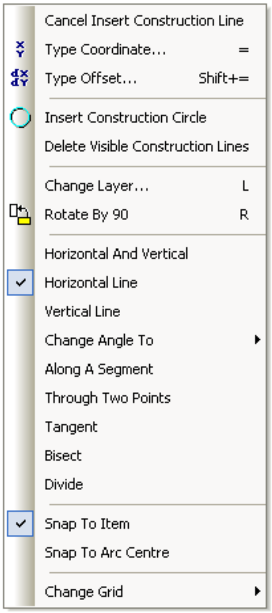
Construction lines are added to the design using the **Insert** menu and either **Construction Line** or **Construction Circle** options.



Default construction lines settings are taken from the **Settings** menu, **Design Settings** and **Construction Lines**. You can set the default **Layer** on which the lines will appear. You can change this as you add the lines using **Change Layer** as you use them.



Once in this option you can select different modes of operation from the context menu. The context menu can be used to switch between modes.

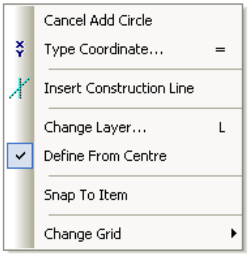


Swapping between Insert Construction Lines and Circles

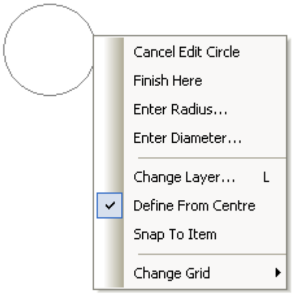
Without having to go back to the main menu, you can select the **Insert Construction Circle** option from the context menu (while adding a line) to swap modes to now add a circle.

How to Insert Construction Circles

After selecting **Insert Construction Circle** option from the main menu or swapping mode to this from the context menu, the context menu then changes to 'circle' modes of operation.

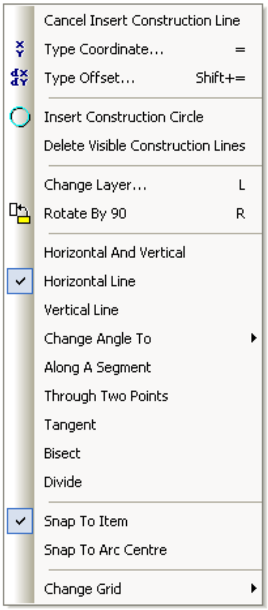


Once the first point of the circle has been added, the context menu then shows more options.



Modes of creating Construction Lines and Circles

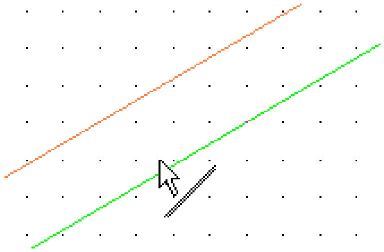
All modes of operation are available from the context menu while in **Insert Construction Line** and **Insert Construction Circle**. From within one of these modes, right clicking the mouse reveals options available for construction lines.



Adding Additional Construction Lines

As well as using the Insert Construction Line command, you can also pick an existing construction line in the design and copy it. This can be done using the Copy command (from the menu or shortcut key Ctrl-C). You can also select an existing line and with the Ctrl key pressed, drag off the line to create a copy using the same angle as the original.

If you copy an existing construction line using **Ctrl-pick** or the **Copy/Paste** or **Duplicate** commands, the 'original' line will be highlighted to indicate that it is the origin.



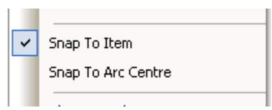
The new line can be positioned with an offset using the **Type Offset** command (see below). With the original line highlighted, the copied line can be moved either side of the original. The Type Offset command will then work in that direction using the values you type in. This means the offset is always typed in positive values.

Snapping Modes

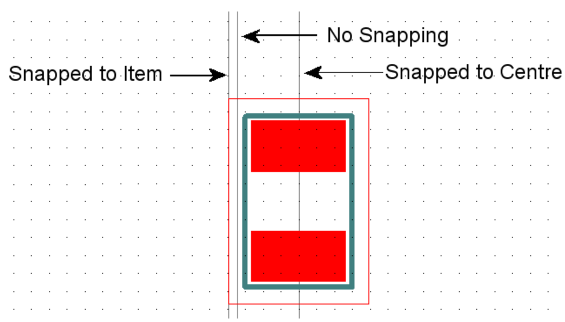
To aid the addition of construction lines/circles, you now have different 'snapping' modes available to you. These snapping modes are also available while in other options, such as Move and Shape Editing etc.

Snap to Item

Whatever mode is being used, when picking a position for the construction line to pass through there are two options available to snap the picked point onto an existing design item. - **Snap To Item** and **Snap To Arc Centre.**



Check the **Snap To Item** option on the context menu to force the picked point be snapped to the closest part of the item beneath the cursor. If snapping to a construction line, the snap point will be further refined to use a construction line intersection point if one is near.

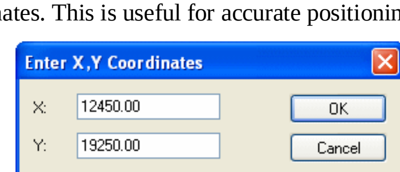


If **Snap To Item** is being used, check the **Snap To Arc Centre** option to cause the construction line point to snap to the centre of a picked arc, rather than the nearest point on it.

If the **Snap To Arc Centre** option is not being used, and an arc or circle is picked whilst adding a construction line, the picked point on the arc will first try to snap to the top, bottom, left or right most point on the arc if it is close enough. If this fails it will try to snap to the tangent point where the line would touch the arc (again if close enough to the picked point). Otherwise it will snap to the nearest point on the arc.

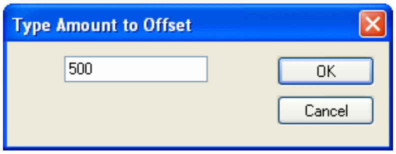
Type Coordinate

The **Type Coordinate** command displays the dialog into which you type the **X** and **Y** coordinates. This is useful for accurate positioning to construction lines.



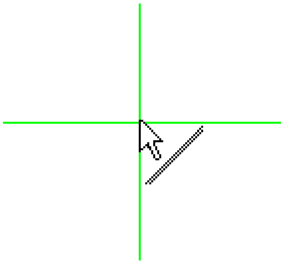
Type Offset

The **Type Offset** command displays the dialog into which you type the **X** and **Y** offset from the delta position, normally displayed on the status bar. The delta position is dependent on the operation being performed, but is usually the starting point.



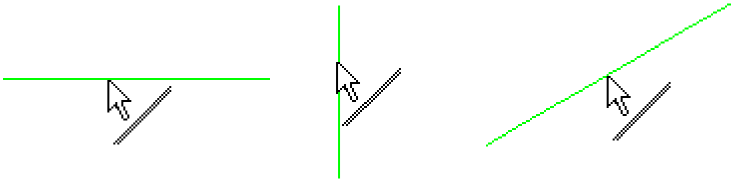
Horizontal and Vertical

This mode adds two construction lines at once, shown as horizontal and vertical lines through the cursor position. Left click to pick the required position for the intersection of the lines. The new lines will be added to the design and drawn in the highlight colour. The next two construction lines will appear, showing the offset from the intersection of the highlighted lines on the status bar.

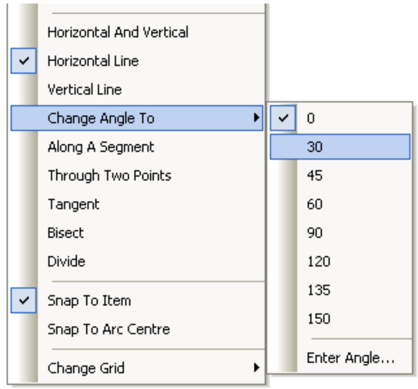


Single Construction Lines – Horizontal, Vertical and Change Angle To

The next three modes, **Horizontal Line**, **Vertical Line** and **Change Angle To**, add a single construction line at a specified angle through the picked position. As each line is added, it is highlighted and the next line is displayed on the cursor using the same angle as the previous line. The status bar shows the distance from the highlighted line.



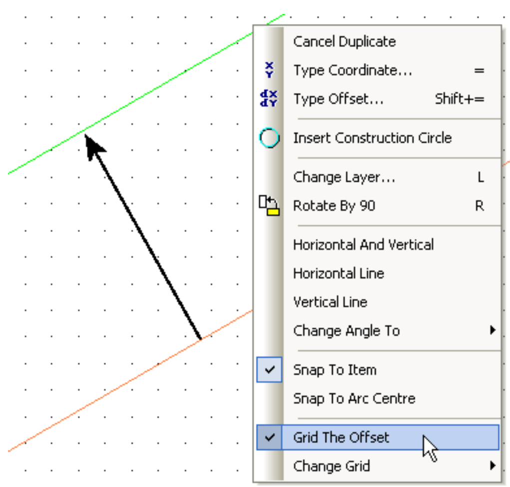
Use the **Change Angle To** sub-menu to alter the angle of the line currently being added. The menu shows all the thirty degree steps commonly used, or use the **Enter Angle** option to type the exact angle required. You can also set the construction line angle to match any straight line segment already in the design. Do this by using **Ctrl-Click** on the existing segment. The moving line will change its angle to the picked segment, and if the segment was a construction line it will be highlighted and the status bar altered to show the distance from it.



Use the **Rotate By 90** option if you want a line that is perpendicular to the current line angle.

If a previously added construction line is highlighted you can use the **Type Offset** option to set the exact distance from it for the line currently being added (see above for Type Offset).

The line being added moves on the current chosen grid, which can be changed using the **Change Grid** option on the context menu. If a highlighted line is shown, you can use the **Grid The Offset** option to change the gridding so that the moving line moves in exact grid steps away from the highlighted line. This enables you to place two lines exactly a set amount apart, and is especially useful when adding angled lines.

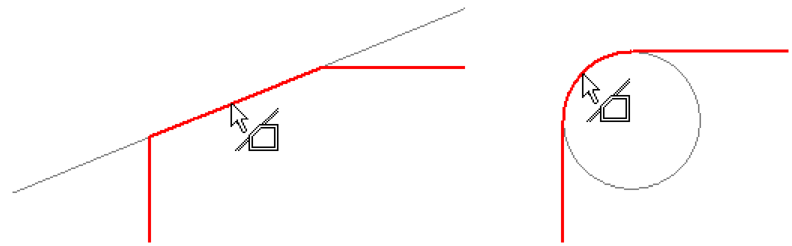


Along A Segment

This allows you to select a shape segment to create a construction line along it. If the segment is an arc, a construction circle is created through the arc.

This mode allows you to create construction lines, or construction circles, that travel along the centreline of the picked shape or track segment. Each time you pick a segment in the design, a construction line, or circle if an arc is picked, is added to the design.

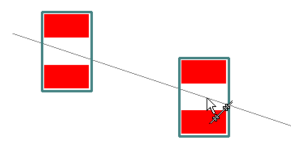
This can be useful if you want to recreate the construction lines that were used to create a complex shape, and then use them to correct, or alter the shape.



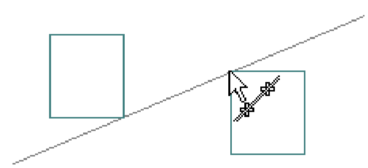
Through Two Points

This mode allows you to define a construction line that passes through two picked points, usually used with **Snap To Item** checked to snap to two specific points on items in the design.

For example, between two component centres:

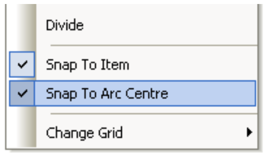


Or between the corners of two shapes:

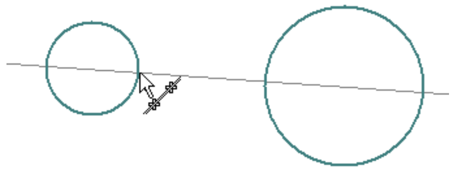


This is a two stage process as follows:

- Click to pick the first point for the construction line to pass through.
- A dynamic line will be drawn from the picked point to the current cursor position, and the status bar will show the current angle of the moving line.
- Click to define the second point, and the construction line will be constructed through the two picked points and added to the design. You are now ready to add the next line.
- By using the **Snap To Item** and **Snap To Arc Centre** options, you can add a line directly between two circle centres.

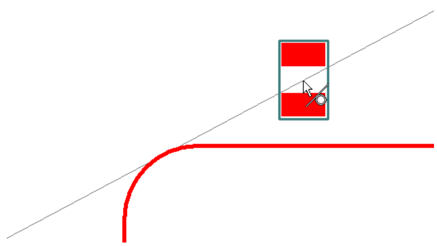


Regardless of where you select on the circles, this mode will select both circle centres.



Tangent

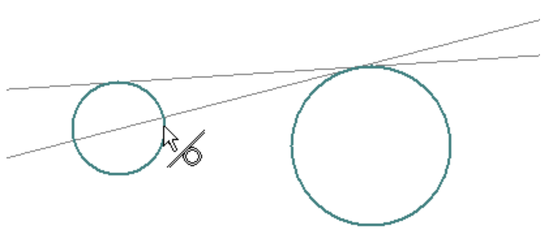
This mode helps you create a construction line that is a tangent to a picked arc or circle from a picked point.



This is a two stage process as follows:

- Click to pick the segment arc or circle for the construction line to become a tangent to.
- A dynamic line will be drawn from a tangent to the picked arc, to the current cursor position, and the status bar will show the current angle of the moving line. If the cursor position is within the arc sector the tangent is drawn at the point where a line from the arc centre passing through the cursor position touches the arc.
- Click to define the point that the tangent is to pass through. The construction line will be constructed from the picked items and added to the design. You are now ready to add the next line.
- If you want to add a tangent between two circles, first make sure that **Snap To Item** is on and **Snap To Arc Centre** is off. Then pick one of the circles, and move the cursor close to the required tangent point on the other circle. Click again and the picked point on the second circle will be adjusted to the actual tangent point.

The example below shows both a tangent to circle edges and a tangent between the circle edge and a circle centre.

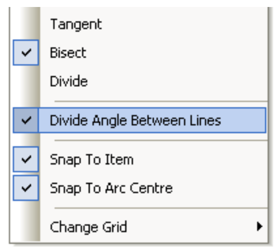


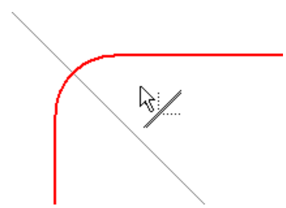
Bisect

Use this option to add a construction line that bisects an angle, or bisects the line between two points. This option can be used on design items and other construction lines.

If an angle is to be bisected.

6. Check the **Divide Angle Between Lines** option on the context menu. The cursor will change the show you are dividing an angle.





7. You can use **Ctrl Click** on an existing **arc** segment to immediately bisect its angle. In this case the construction line will be added and you will be ready to bisect another line.

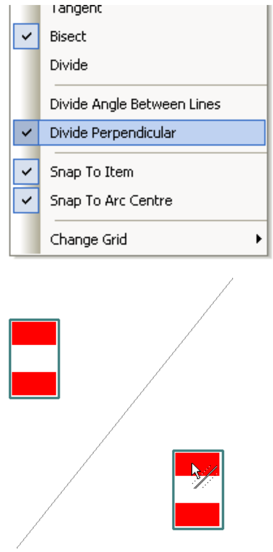
8. If not using the Ctrl key, click on an existing line segment to define the first line for the angle to be bisected. A temporary line will drawn through the picked line. Now pick the second line of the angle. A construction line will be added bisecting the angle between the two picked lines. If the picked lines are parallel then the space between them is bisected. You will now be ready to pick the next angle to be bisected.

If you creating a bisecting line perpendicular to the line between two points:

1. Uncheck the **Divide Angle Between Lines** option on the context menu in order to indicate it is a line you wish to bisect rather than an angle. The cursor will change the show you are not dividing an angle.

2. Check the **Divide Perpendicular** option on the context menu to indicate the angle of the resultant line is to be perpendicular to the line between the picked points.

In the example below, the component centres were selected and the perpendicular line added through the centre.



3. You can use **Ctrl Click** on an existing **straight line** segment to immediately bisect it. In this case the construction line will be added and you will be ready to bisect another line.

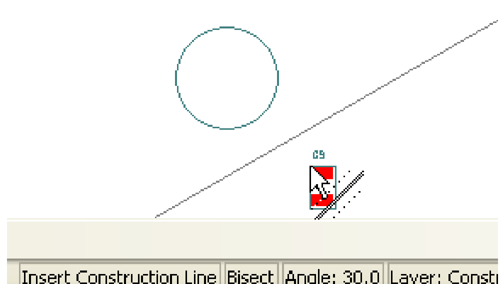
4. If not using the Ctrl key, click on an existing point to define the first point on a line to be bisected. A temporary line will drawn through the picked line, perpendicular to the moving cursor.

5. Now pick the second point. A construction line will be added perpendicular to, and bisecting, the line between the two picked points. You will now be ready to pick the next points to be bisected.

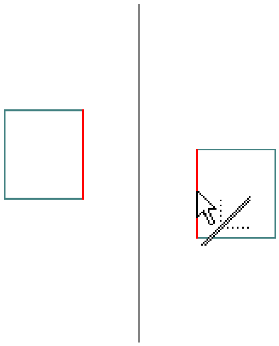
If you creating a bisecting line between two points at a fixed angle:

1. Uncheck the **Divide Angle Between Lines** option on the context menu in order to indicate it is a line you wish to bisect rather than an angle. The cursor will change the show you are not dividing an angle.

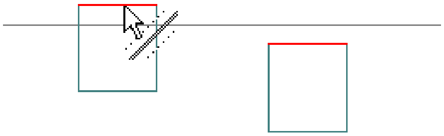
2. Uncheck the **Divide Perpendicular** option on the context menu. The bisecting line will use the current angle displayed on the status bar.



3. Click on an existing point to define the first point on a line to be bisected. A temporary line will drawn through the picked line at the current angle.



While in Bisect mode, if you wish to switch the line mode (between vertical and horizontal for instance), you must select the mode required from the context menu, then re-enter Bisect mode again from the context menu.



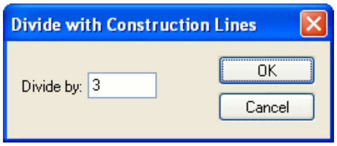
4. Or you can use **Ctrl Click** on an existing **straight line** segment to change the angle of the bisecting line to the same angle as the picked line.

5. Now pick the second point. A construction line will be added at the current angle, bisecting the line between the two picked points. You will now be ready to pick the next points to be bisected.

Divide

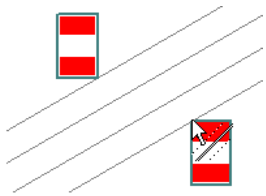
This option uses the same interactive procedures as the **Bisect** option above, but adds multiple construction lines to divide the angle or line into multiple parts. As with the Bisect option, it can be used on design items and other construction lines.

When the option is chosen, the following dialog is given to get the number you want to divide by. If you are already in the **Divide** option, simply select the **Divide** option again from the context menu to change the divide number.

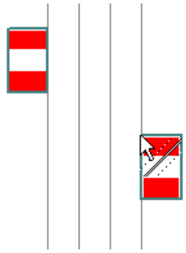


When the construction lines are added to divide the picked angle or line, unlike **Bisect**, lines will also be added at the extents of the angle or line.

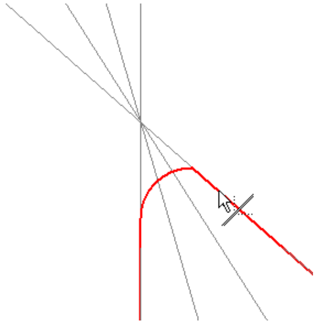
The example below shows two corners of the two components divided by 3 with the **Divide Perpendicular** switch selected.



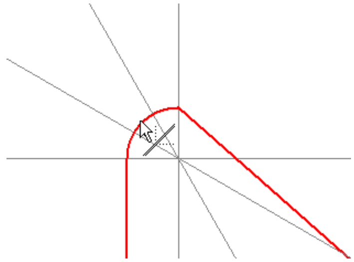
The example below shows the same selection but with the **Divide Perpendicular** switch unchecked. The original line mode selected was **Vertical** before using **Divide**.



The example below shows Divide by 3 on two selected straight line segments.



In the same mode, if you Ctrl-pick the Arc instead of the two straight line segments, the divide will be performed on the arc centre.



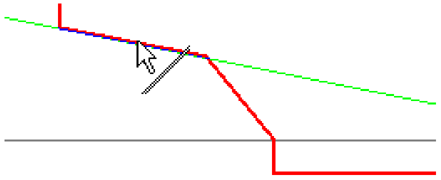
Finding an Arc or Mitre Centre

If you need to find an Arc or Mitre Centre, there is a simple method you can employ.

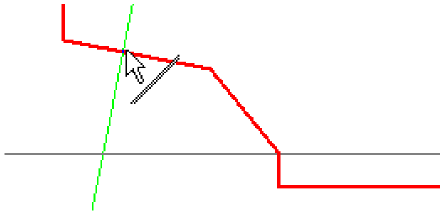
In the picture below, we've already added one horizontal construction line (using either **Horizontal** mode or **Select Along A Segment** mode then rotating it by 90 degrees as the shape line is at 90 degrees and snapping it to the corner).

We now need to add another construction line at 90 degrees to the other shape segment. The difficulty with this other line is that it is not orthogonal.

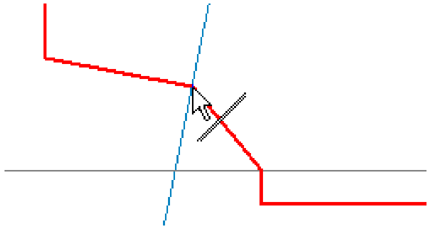
In the Construction Line option, using the **Along A Segment** mode again, Ctrl-pick the angled segment. The construction line being added will snap to this line and mimic its angle:



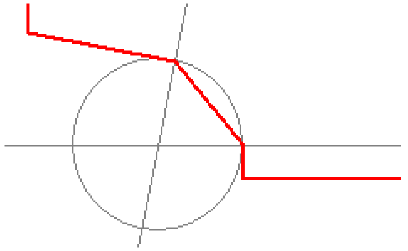
Now that the angle is the same as the segment, Rotate it by 90 degrees (use the <R> shortcut key).



Click on the corner of the two shape segment to snap the construction line to it.



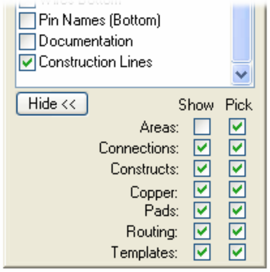
The two construction lines cross at the arc or mitre centre point. We can show this by adding a circle to the construction line intersection:



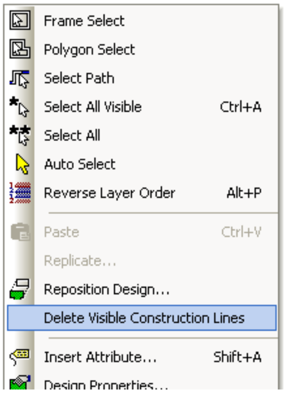
Viewing/Deleting Construction Lines

Construction lines can be added to any layers but it can be beneficial to create a separate layer to add them to. Using a unique layer, the layer name becomes available in the **Colours** dialog and on the **Layers Browser**. When defining a unique layer, the **layer type** should be set to **Documentation**.

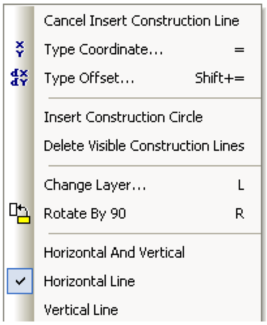
Construction lines are available as a display item on the **Layers Browser** list at the bottom of this dialog. This is shown under **Show>>/Hide>>** under **Constructs:**.



You can delete all visible construction lines in the design using context menu command **Delete Visible Construction Lines** and as a command or shortcut key.



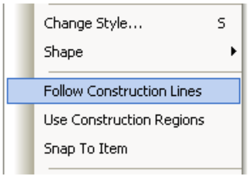
You can also use the **Delete Visible Construction Lines** option whilst in the **Insert Construction Line** option from the context menu.



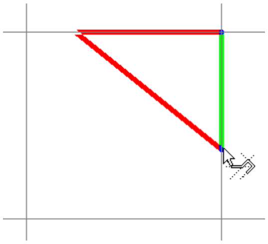
Using Construction Lines to create shapes

Follow Construction Lines & Use Construction Regions

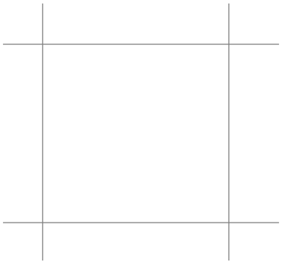
When adding shapes, with construction lines in the design, from the context menu you can select that item to **Follow Construction Lines** or to **Use Construction Regions**. These options can be used when adding **Shapes**, **Merging Shapes** and adding **Cutouts**.



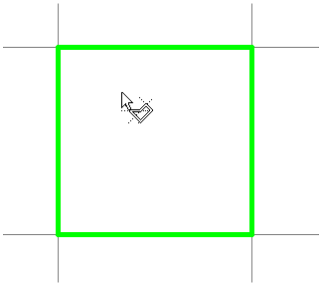
Follow Construction Lines will allow the shape to snap to the nearest construction line and then follow the shape as you move the cursor along further construction lines.



Use Construction Regions will allow a shape to mimic enclosed construction line shapes created, for example if you cross four lines to make a 'box', this will create you a box shape region.

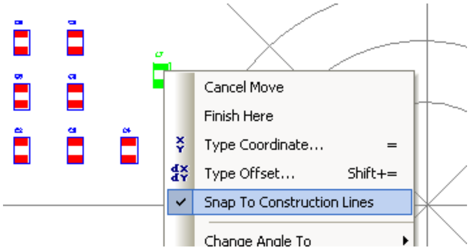


When adding a board outline for example and selecting this option, if you then click inside the 'region' it will automatically add you a box shape snapping to the construction lines.



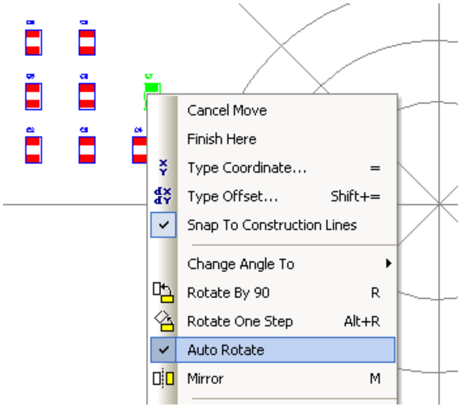
Snap to Construction Lines

Construction lines work like placement sites during placement when placing components. From the context men, you can select the **Snap To Construction Line** option. Components will then use the **Placement Site** distance **value** defined in the **Options** dialog under **Placement Sites**.

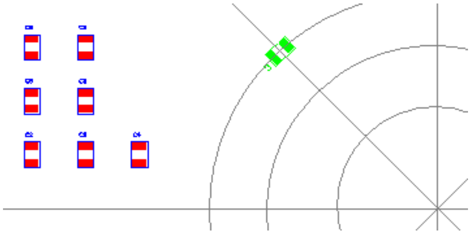


Moving item on Construction Lines

When moving components, you can also use the **Auto Rotate** mode to snap to intersections between circles and lines, this is available on the context menu.

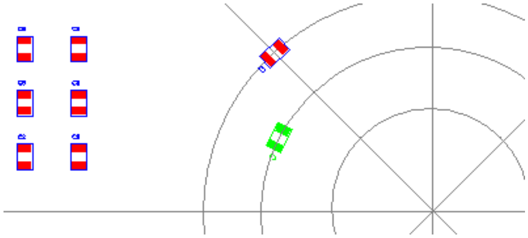


With **Auto Rotate** selected, components now snap to construction lines. This can be an intersection of lines:



Or it can be along the lines themselves, auto-rotate adjusting the component's rotation accordingly.

If you move off the construction line, the component will snap to the nearest grid point and back to its original orientation.



DXF Import of Construction Lines

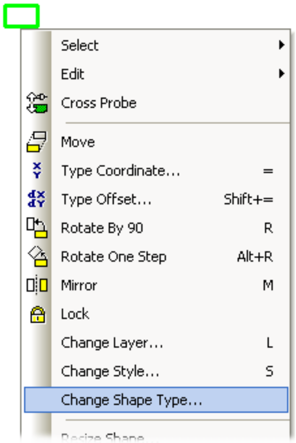
If construction lines exist in the DXF file, they will be automatically imported. They are included if they appear in the file and would have been defined in AutoCAD.

New Shape Functions

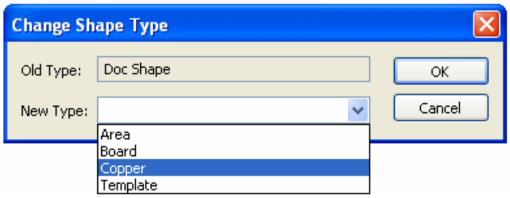
Change Shape Type

When you select a shape, you can swap from one type to another. For a shape type where **Closed** shapes are mandatory and the original shape is not closed, changing it from one type to another will close the shape.

Select the **Change Shape Type** option from context menu or by using a shortcut key.



Choose the New shape Type from the drop down list and click **OK**.



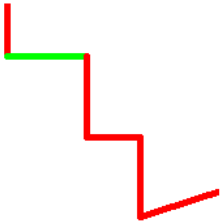
Change Shape Type for Text

Change Shape Type is available on the context menu for some text items. It allows you to convert true type font text to copper or doc shapes.

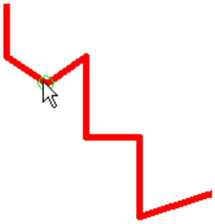
This allows conversion of company logos to copper for example, logos that were done with a special font. It will then also allow pour copper to pour up to true text outline.

Add Corner

You can add a corner into a line segment and do not want to use the **Edit Segment** operation, it can be performed by selecting the segment and using the **Add Corner** option from the context menu.



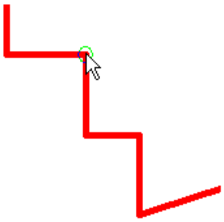
Pick a segment and select **Add Corner** from the context menu.



A corner will be added, and presented at your cursor for you to place. Click to drop the corner at the required location.

Delete Corner

Similarly you can remove a corner from a shape by selecting it and using the **Delete Corner** option from the context menu



Pick a corner and select **Delete Corner** from the context menu.

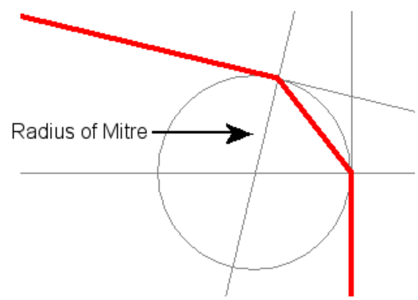
The corner is removed and a straight line added between the previous corner points.

Mitre / Unmitre / Fillet Corner

Orthogonal corners can be converted to mitres or fillets by double clicking on a (90 degree) corner or on an existing orthogonal mitre or fillet. The interactive **Edit Mitre** mode will be entered to drag the mitre or fillet to the required size.

Alternatively you can now select a single corner, mitre or fillet and use the **Mitre Corner** or **Fillet Corner** options from the context menu. This is a single shot operation, but has the advantage that **any angle corners** can have a mitre or fillet generated. You will be presented with a dialog:

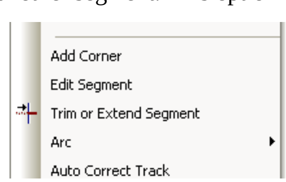
Type the value for the radius or diameter of the mitre or fillet to be created. This is the radius of the circle that is tangent to the two lines that meet at the corner.



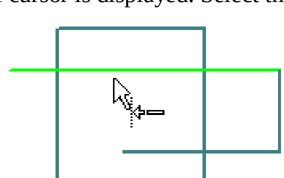
You can remove any fillet or mitre back to a sharp corner by selecting it and using the **Unmitre Corner** option from the context menu.

Trim & Extend Segment

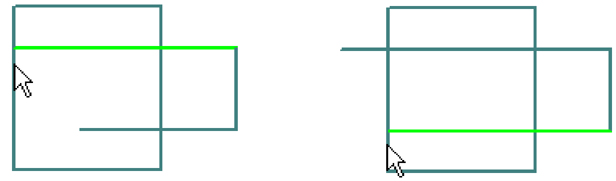
With a shape segment selected you can use the new **Trim** and **Extend** option from the context menu to trim the line or arc back to where another segment crosses it, or extend it to reach another segment. This option is also available on the **Utilities** menu.



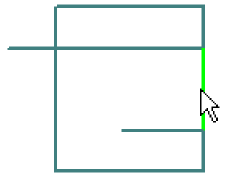
Once a track has been selected, from the context menu, select **Trim or Extend Segment**. A modal cursor is displayed. Select the segment you wish to trim or extend to.



These examples show how the one segment has been trimmed back to the shape and the other segment extended to meet the shape (in the right hand side example).



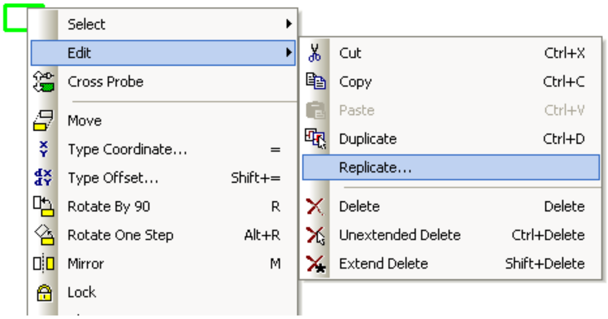
You can also select segments to trim back to, to make them parallel:



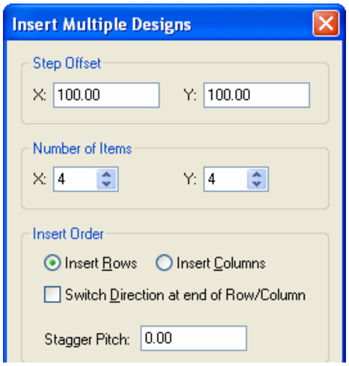
Replicate (Array Placement)

Use **Replicate** to add multiple copies of selected items within the design (in an array placement).

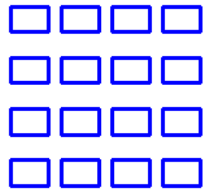
Select the item to replicate and from the context menu, select **Edit**> and **Replicate**. This option is also available on the main **Edit** menu and can be allocated to a shortcut key.



Enter the required parameters into the **Insert Multiple** dialog.

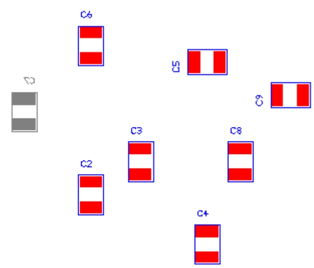


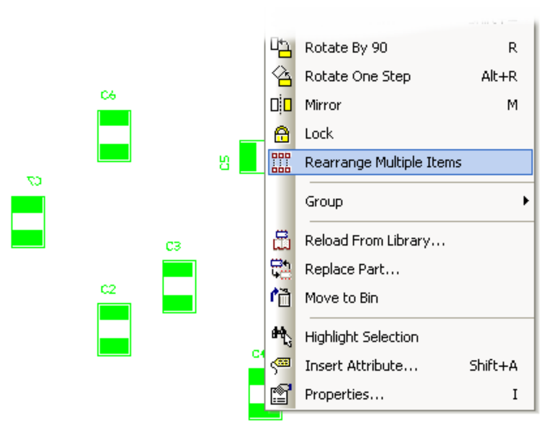
The items are replicated:



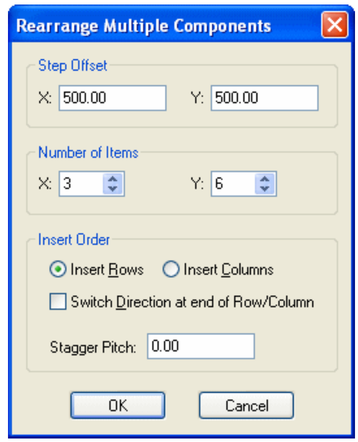
Replicate in Move

Select a number of (same) items and from the context menu select **Rearrange Multiple Items**. The items can be with different rotations and mirror status. When using this option, whichever one the cursor is hovered over, the option will take the Rotation and Mirror status and apply it to all the selected items.

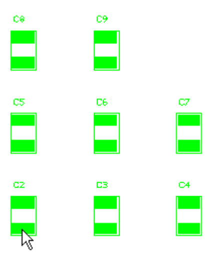




Enter the parameters into the **Rearrange** dialog.

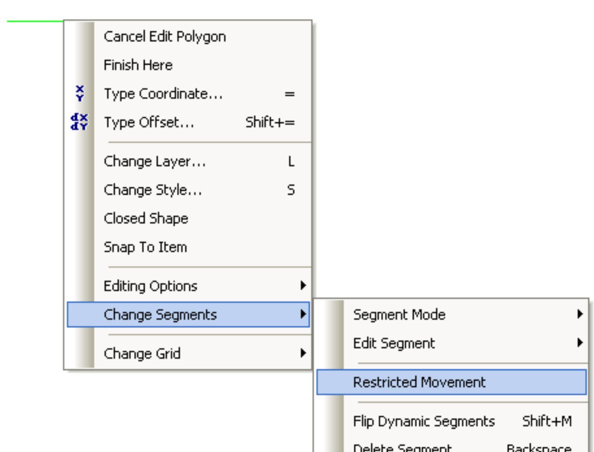


The items are rearranged and based on the status of the selected item.

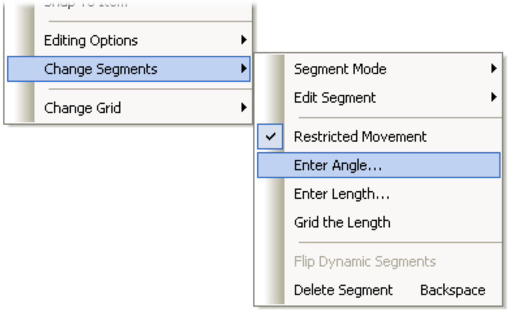


Restrict Movement

While in **Edit** shape or **Insert Track** there is a new option is called **Restrict Movement**. This is a special single segment mode for use whilst editing a shape, the default is 45 degree mode.



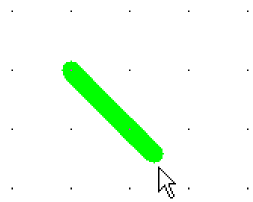
Once enabled, there are additional features to **Enter Angle**, **Enter Length** and **Grid the Length**.



Type Angle – this restricts the next segment to angle or perpendicular to angle.

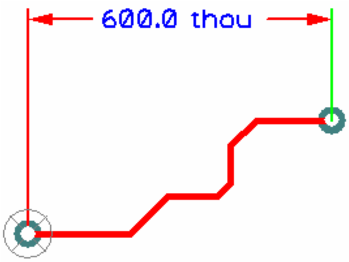
Type Length – this restricts the next segment to a fixed length.

Grid the Length will lock the length of the moving segment to the same length as the grid spacing. This is noticeable when moving in non-orthogonal directions, i.e. 45 degrees.



Attached Dimensions & Attached Callouts

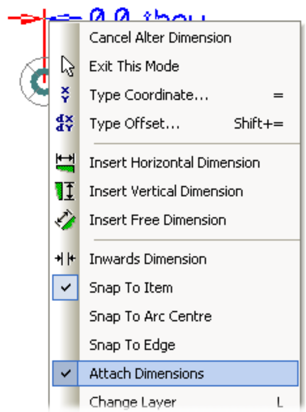
You can now attach the end of **Dimensions** and **Callouts** to items. When the item is moved, the dimension moves with it and its displayed size adjusts without further user intervention. You are able to attach dimensions to most design items except for Text, Text Attributes, another dimension, Construction Lines, Callouts, junctions (like a 'T' route), unrouted connections, Vias, Teardrops and Origins.



Attached Dimensions

There is a new command **Attach Dimensions**, available from the context menu whilst adding and stretching **Dimensions**.

Check this so that when a dimension is snapped to an item it will be attached to it (if it can).

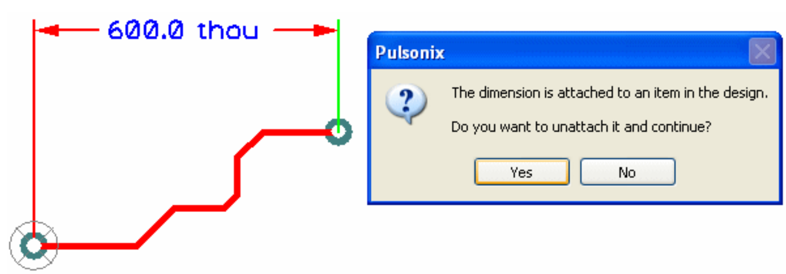


Move

Moving or changing the item will cause the dimension to be stretched (and have its measurement altered).

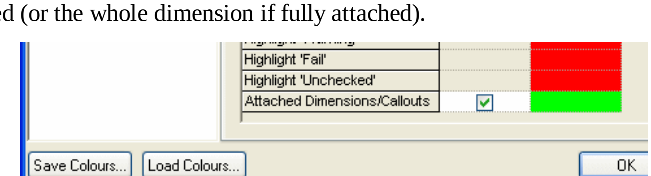
Unattach

Dragging the dimension line attached to an item will present a dialog asking if you want to un-attach the dimension.



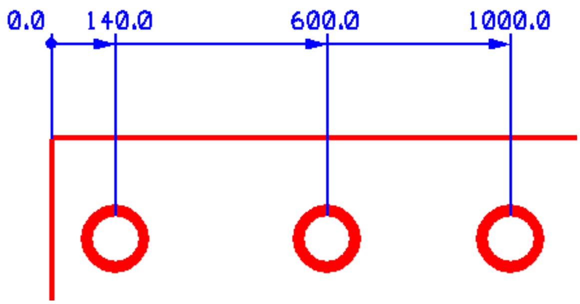
Highlight

There is a new highlight colour in **Colours** to highlight the half of the dimension that is attached (or the whole dimension if fully attached).

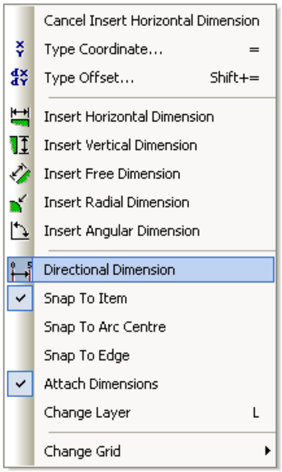


Directional Dimensions

You can now add horizontal or vertical dimensions with a **directional** appearance, for documenting multiple positions relative to a common point.

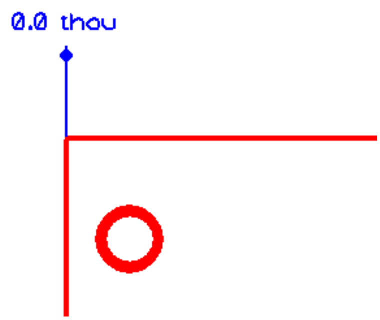


When adding these types there is a new command, **Directional Dimension** available from the context menu.



As soon as you have selected the dimension type (horizontal or vertical), select **Directional Dimension** from the context menu and then select the position of the start of the first leader line.

You are now presented with the first 'blob' and dimension text. You can position the (0,0) first line length, or select the first item to be dimensioned, in which case, the zero line will be automatically made. If you move your cursor, you will notice the next dimension at the end of your cursor ready for positioning. This makes it easy to add a sequence of directional dimensions without having to pick the start point each time. You can position this over the existing line or move it to view it as required. Only one 'arrow' is displayed on the dimension line in this mode.

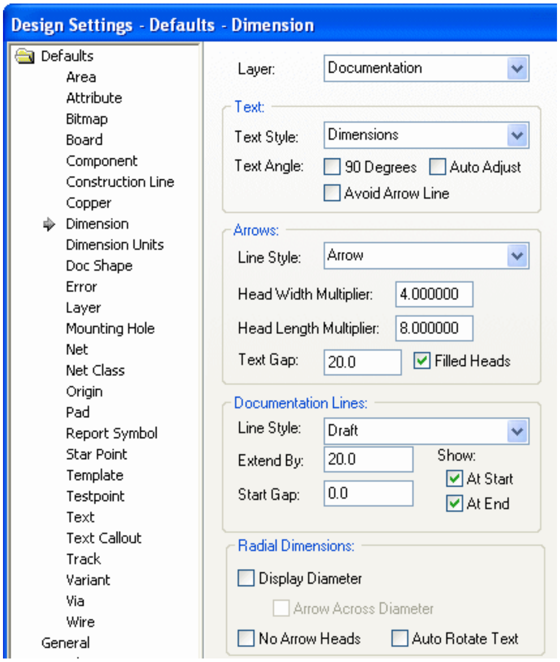


A directional dimension has no start arrow or start doc line and its default text position is next to the end arrow. If the dimensions measurement is **0** the end arrow is draw as a small filled circle the width of the arrow to indicate the common point.

General Dimension Changes

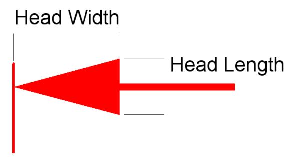
Dimension Arrow Size

You can now specify the dimension arrow size by providing a width and length multiplier, multipliers of line style width used.



The arrow line width is multiplied by these values to give the arrow head size. If the values are zero, no arrow heads will be drawn.

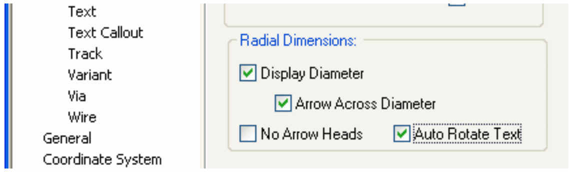
The **Design Settings Dimension** page now has extra controls in the **Arrows**: section to set the default **Head Width Multiplier** & **Head Length Multiplier**.



Individual arrows can have their arrow head size changed by changing the multipliers using **Properties** dialog.

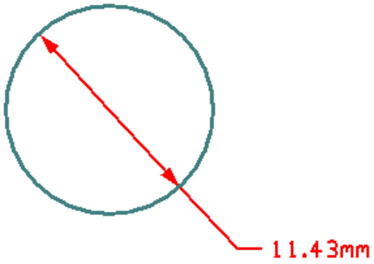
Radial Diameter Dimensions

Radial Dimensions can now display the diameter of the dimension (instead of the radius).



If the radial dimension is set to display the diameter, then you can also specify that you want the arrow lines to cross the whole diameter (instead of just the radius). These new switches are in the Dimension defaults in a new section called **Radial Dimensions**.

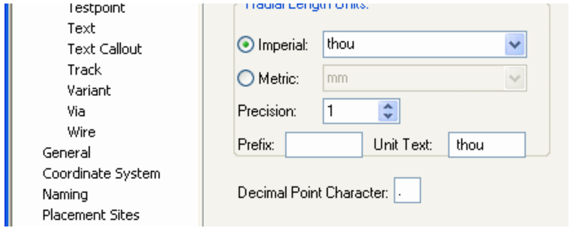
There is also a default switch for ensuring that radial dimensions have no arrow heads (to fit some standards).



Individual radial dimensions can have these values changed using the dimension **Properties** page.

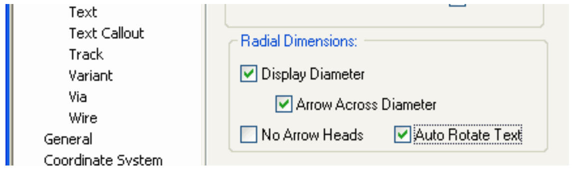
Dimension Units Decimal Point

You can now set the Decimal Point character used in Dimensions. This can be set in the **Dimension Units** defaults dialog, or individually on dimensions using **Properties**.



Auto Rotate Text

Dimensions can now be set to have their "default" text auto-rotated to the angle of the dimension.



There is a switch in Dimension defaults for radial dimensions only, but you can change it in the dimension text Property page for any dimension.

If the auto-rotated text angle becomes greater than 180 degrees the text is adjusted by 180 degrees to make it more readable.

Attached Callouts

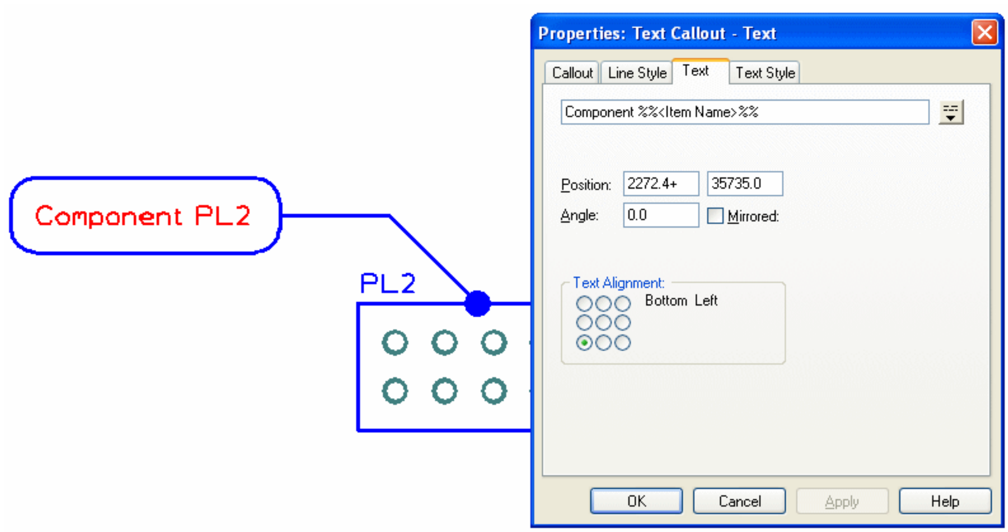
There is a new command **Attach Callouts**, this is available from the context menu whilst adding and stretching callouts. Check this so when the end of a callout is positioned over an item it will be attached to it (if it can). **Shift-select** on a shape will attach a callout to the nearest point on a shape.

As with Attached Dimensions, if you attempt to move the callout near where it is attached, a warning dialog is displayed.

Attributes

You can substitute any of the **attributes** of the attached item in the callout text. For example, when attaching to a component the text can contain the **Component Name**.

For the **Callout Text Properties**, enter your text (*Component*, in our example) plus the substitute attribute name (*%%<Item Name>%%* in our example). The text callout will then take on the item properties.



Hyperlinks

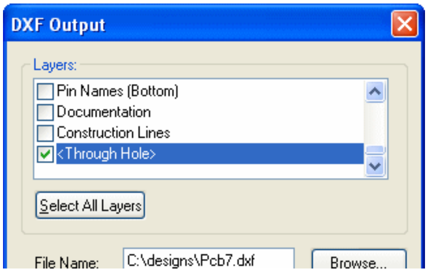
If you select the callout text, the **Execute Hyperlink** command will use the text as the link. If the text contains **c:\temp\document.pdf**, it will run this PDF file.

If you select the callout lines, **Execute Hyperlink** will use the hyperlink attributes from the attached items. For example, if attached to a Component and the component contains an attribute with the value **http:/www.datasheets.com/12345.pdf** then when run, it will execute this link. The attribute must be flagged as available to be **run** as a **Hyperlink** in the **Technology** and **Attribute Names** first though.

DXF Export of Drill Letters and Symbols

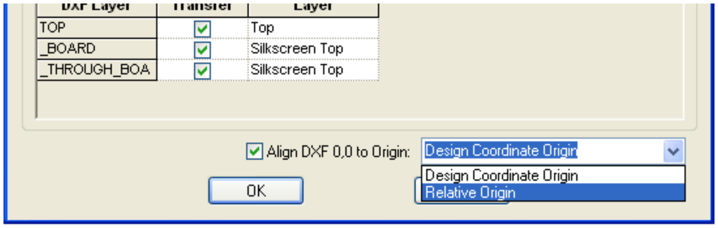
The DXF output can now export **drill letters** and **Symbols** using a selection on the DXF dialog.

If you select the layer span layer i.e. <Through Hole>, you will get a drill drawing layer in the output (called *_THROUGH_HOLE_DRILL_DRAWING_*). This layer will contain any drill symbols and letters defined for the appropriate drills in the **Technology** file and **Drill Sizes** dialog.



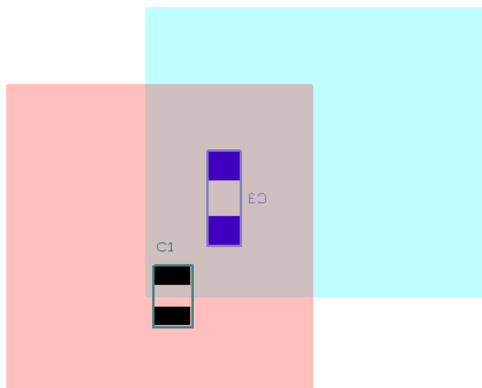
DXF Import Changes

The dialog for **DXF Import** now includes the command **Align DXF 0, 0 to Origin** as a checkbox. This allows you to choose to align the 0,0 point of the incoming DXF data with either the **Design Coordinate Origin** or the current **Relative Origin** in your design.

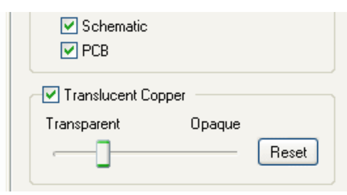


Translucent Copper

Pulsonix now contains the ability make copper transparent. This aids the visualisation of overlaid copper areas and other items under those copper areas, SMD components for examples.

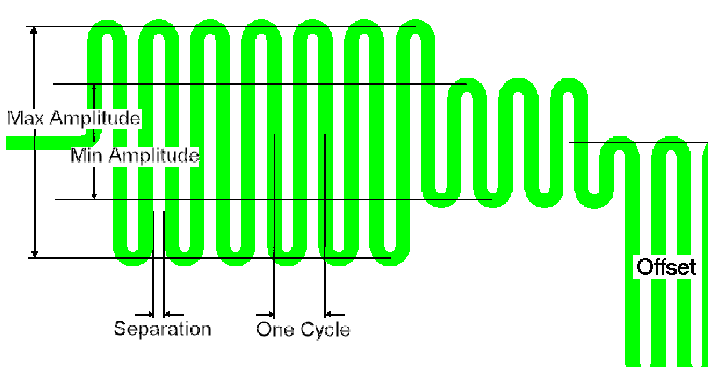


Control for **Translucent Copper** is available on the **Options** dialog and **Display** page. The slider on this dialog allows required level of translucency to be chosen. Once enabled, all copper will be drawn at the specified translucency. The Reset button will reset the slider to an optimum translucency position.

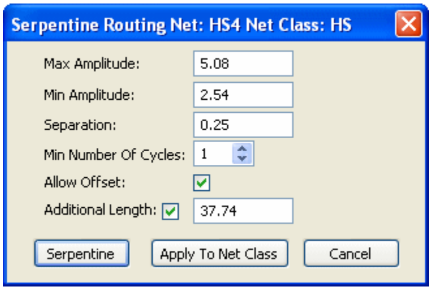


Offset Serpentine

You can now **Allow Offset** on a serpentine, which means that the serpentine section may be offset from the centreline of the track (in either direction. It will only be offset if it is not possible to add the track along the centreline.

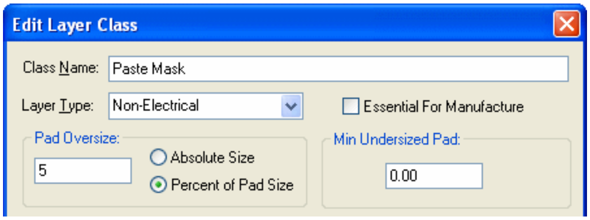


A check box on the **Serpentine** routing dialog and in the **Technology** file and **Net Class, Routing** dialog will enable this.



Pad oversize as a percentage

In the **Edit Layer Class** dialog, you can now specify the **Pad Oversize** as a **Percentage** of the pad size as an alternative to specifying an Absolute size. Check the **Percentage** radio button and type in the value required. You do not need to add a % sign in the value field.



Insert Multiple Items

The **Insert Multiple Items** command has been added to the context menu for use when adding **Mounting Holes** and **Footprint Vias** in the footprint editor.

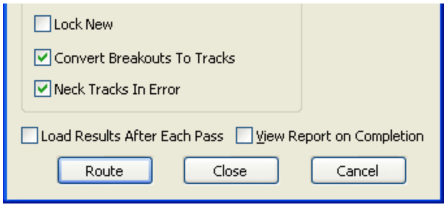
Online DRC Update

When using the **Online Design Rules Check**, the option now checks for a via in a pad when moving a component pad over a via or a via over a pad.

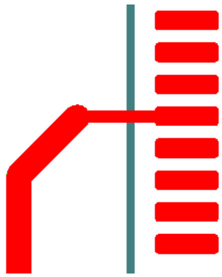
Auto-necking into SMD Pads

When using the autorouter, you can now specify whether the router will automatically use the alternative track style width to ‘neck’ down into SMD pads.

In the router dialog, a check box **Neck Tracks In Error** is available. If a ‘fat’ track to be routed will cause a DRC error when entering an SMD pad, with this check box selected, the ‘thin’ track style can be used and the routing completed successfully.



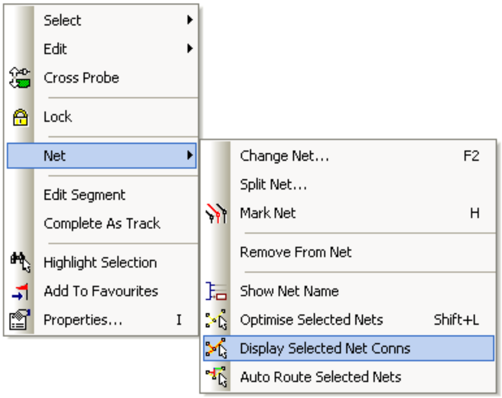
The resultant tracks will look like this:



Toggle Display Conns

The existing option, **Toggle Display Conns**, is now called **Display Selected Net Conns**. This change has been made to make the name of the option more meaningful.

This option is available on the context menu under **Nets>** and is available for selected components or connections. It will display the connection selected or all the connections attached to a component.

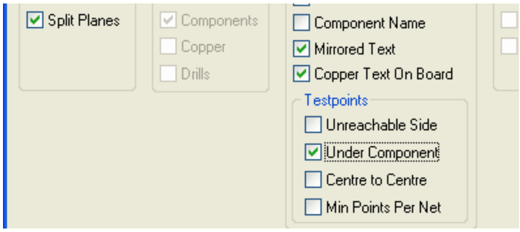


New DRC Checks

There are a number of new DRC checks available from the **Design Rules Checks** dialog:

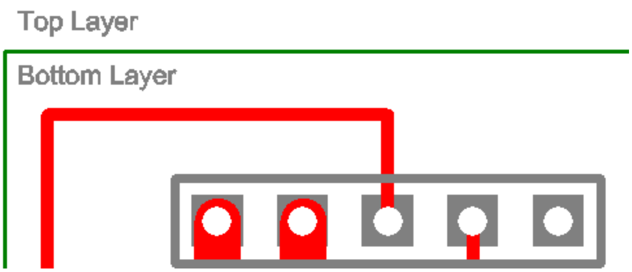
Testpoint under Component

There is now a check for Testpoints under Components.



DRC checks on text

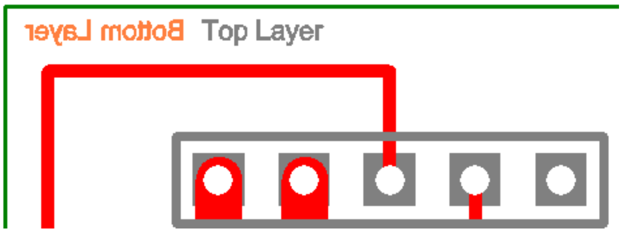
You can now check for copper text outside the board outline and for mirrored text on a non mirrored layer (or non mirrored text on a mirrored layer).



In the above example, the resultant error messages will be:

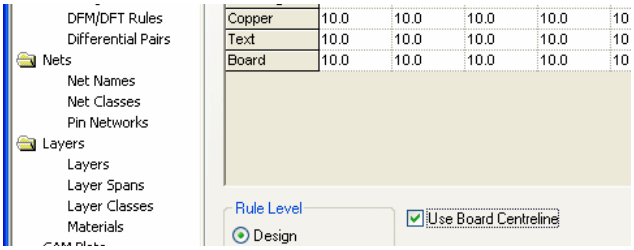
- Copper Text Outside Board Outline (CT) At (31.24 87.63). Layer 'Top Electrical'.
- Incorrect Text Mirroring (MX) At (30.73 85.85). Layer 'Bottom Electrical'.

The design below shows both of these errors corrected.



Use Board Centreline

There is a new check box on the **Technology** dialog and **Spacing Rules** tab – **Use Board Centreline**. Use this option to ignore the ‘real’ width of board outlines when using Design Rules Checking. Checking would then be done up to the centreline of the shape.



Testpoint Analysis

Testpoint Rules

You can now specify additional constraints on testpoints (such as minimum pad size, distance between testpoint centres, whether you can test on vias, SMD pads, Through-hole pads, etc.). You can define the number of testpoints to be added to nets in the design.

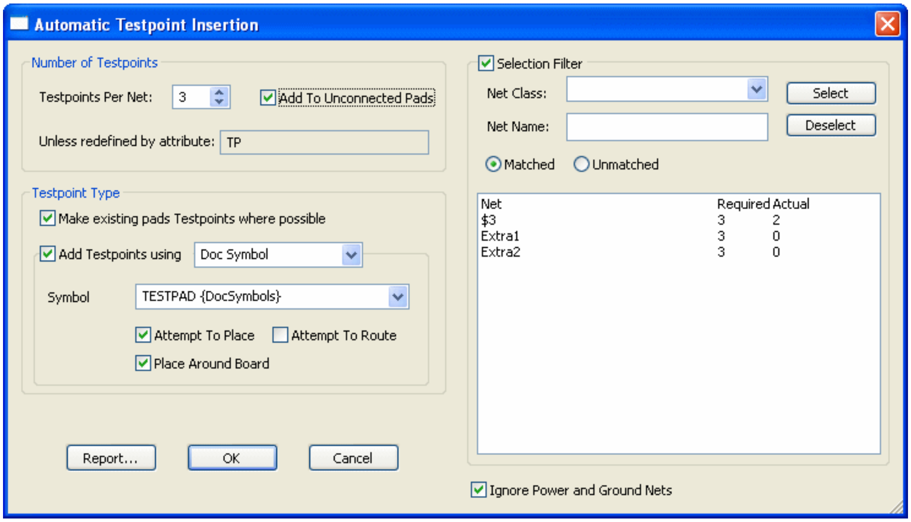
You can also specify an attribute which defines the minimum number of testpoints required on a connected net (you can already specify this on a net class). This attribute can be on a net, pad or component even if the net class rule is set.

If any of these rules (min number of TPs required) specifically define 0, then no testpoints are required on that net, otherwise it is the maximum of these minimum numbers (non-numeric values have no effect).

Automatic Insertion of Testpoints

Using the **Auto Insert Testpoints** option from the **Tools** menu, will automatically allocate additional testpoints on a net. These can be added to existing pads, vias or new testpoint symbols, vias or parts can be added.

The **Automatic Testpoint Insertion** dialog allows these testpoints to be defined and added.



Using the **Add To Unconnected Pads** check box will allow you to add testpoints to pads which are not currently connected.

Testpoint Type allows you to choose which type of testpoint you will add to the design and where they should be positioned. Unplaced testpoints are added to the Component Bin or placed **Around The Board** if this option is checked.

Testpoint Insertion Report

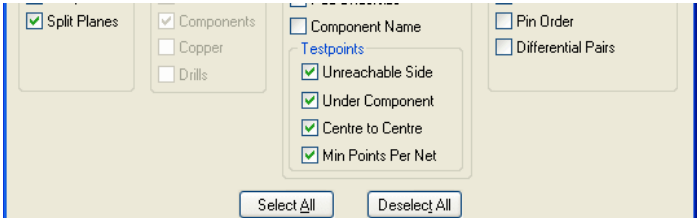
Run from **Automatic Testpoint Insertion** dialog, the Testpoint Insertion **Report** lists nets which do not have the required number of testpoints (using the rules above).

It suggests suitable existing nodes which could become testpoints and any DRC rules they would break.

```
Summary
-------

  48 net(s) missing testpoints
   0 net(s) with extra testpoints
 144 testpoint(s) need to be inserted

Nets With Missing Testpoints
----------------------------

Net        Actual  Required

DIFF1      1       2
DIFF2      0       3
DRIVE      2       1
E1         0       3
E12        0       3
FAT        0       3
GND        0       3
High_speed 0       3
HS         0       3
```

New DRC Checks for Testpoints

There are new checks available in the **Design Rules Checking** dialog under the **Testpoints** section.



Testpoints – Unreachable Side – is an existing check but now categorised under the **Testpoints** section.

Testpoints - Under Component - Checks that Testpoints are not placed under or too close to a Component. This check uses the component placement shapes to determine the extents of each component. Each testpoint must be more than the minimum distance away from all components on the probe side.

Testpoints – Centre to Centre – Checks that the testpoint centres are not too close using the Testpoint Centre space rule defined on the **Technology Design Rules** page.

Testpoints - Minimum Points Per Net - Checks that the net has enough Testpoints, as defined in the Net Class Rules section of the Design's Technology data.

Testpoint and Testability Report

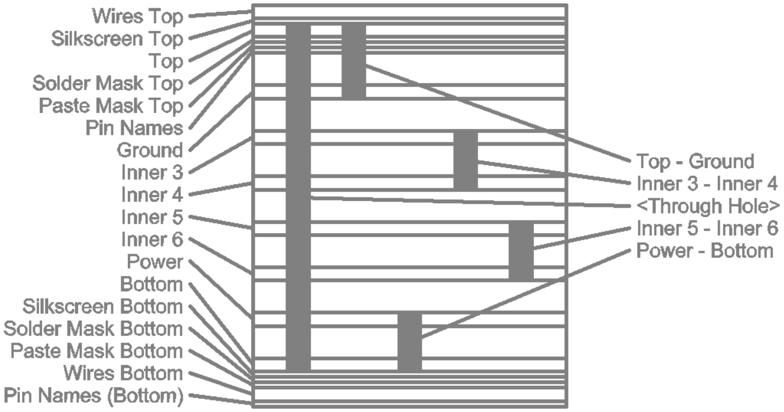
Run from the **Output** menu and **Reports**, two reports are available to report information about Testpoints:

Testability Report – this is an analysis report of the testpoints in the design. It also includes Testpoint DRC checks defined in the **DRC Acceptance** tests. This enables you to include this report in your **CAM Plots** batch as a final sign-off report for Testpoints.

Testpoint Report – this reports the X and Y position, Side, Name and Net Name of each testpoint in the design.

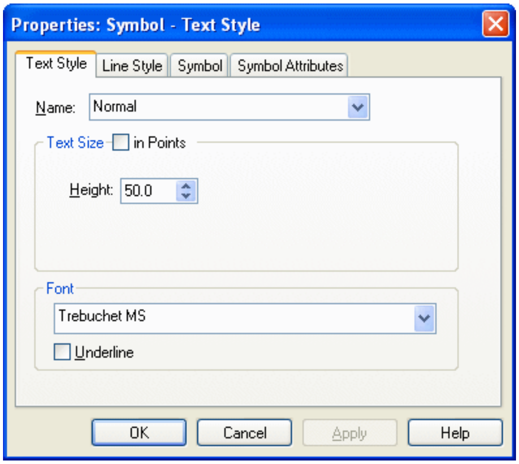
Insert Layer Stack Preview

Within the **Technology** dialog under **Layers** and **Layer Spans** is the **View Layer** feature. This enables you to view the layer stack and via spans. In version 5.0 you are now also able to insert this into a PCB design as a documentation symbol using the **Insert Layer Stack Preview** option on the **Insert** menu. Where the **Material** and **Thickness** have been defined in the **Technology**, these values will also be displayed in the Layer Stack Preview.



Default values for Layer:, Line Style: and Text Style: are defined in the **Settings** menu and **Design Settings** and **Defaults**, **Report Symbol** dialog.

Once in the design you can change the properties of the preview using the **Properties** dialog.



The size of the overall layer stack preview doc symbol is determined by the text size used for the layer names within it. If you change the text style and size using the **Properties** dialog, the doc symbol used for the layer stack will be automatically resized bigger or smaller as appropriate.

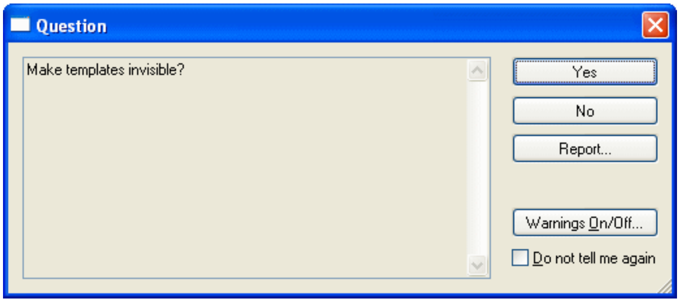
If you change the layer stack or via stack in the **Technology** dialog, you can update the doc symbol in the design by selecting it and clicking **Update Layer Stack Previews** from the context menu. You may have more than one layer stack preview in your design.

Display Layer Stack Preview Command

There is a new command to display the **Layer Stock Preview** without going through the **Technology** dialog. This is available as a shortcut key command and a command from the **Run Command** dialog - **Display Layer Stack Preview**.

Toggle View Powerplane Templates Off/On

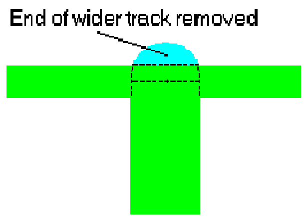
When using the **View Powerplane** option, Pulsonix now automatically switches off **Templates** on powerplane layers. Templates are switched back on again after using this option (**View Powerplane> Hide**), again with a question dialog.



Warning check boxes for both modes are available to pre-select in the **Design Settings** and **Warnings** option.

Square Ended Tracks and Backoff

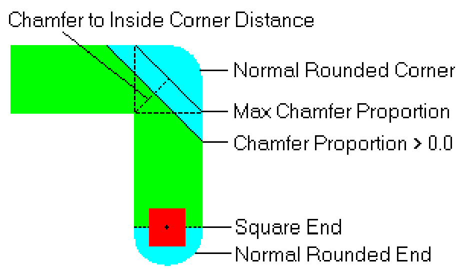
When using thick tracks into a thinner track using a 'T' route, if you wish the thick track to be backed off, you can check the **Square Ended Tracks** box on a **Net Class** under **Routing**. This now backs off at width changes and junctions as well as on pads and at dangling ends. Enabling **Backoff Track Ends** will cause tracks to end squared off. This is only applied to tracks which have sufficient length for the track to be backed off and the end squared.



Design Rule Checking will take the square end into account, allowing tracks to end more closely to other obstacles than would normally be the case. At width changes and T-Junctions, track ends are squared off, so there is no overshoot. Again, this allows obstacles to be placed slightly closer to the track end.

Copper pour also takes these fully into account.

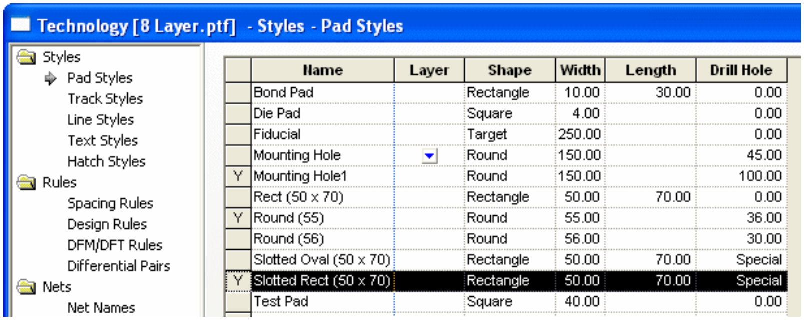
Note that there is a drawing performance penalty when using square ends.



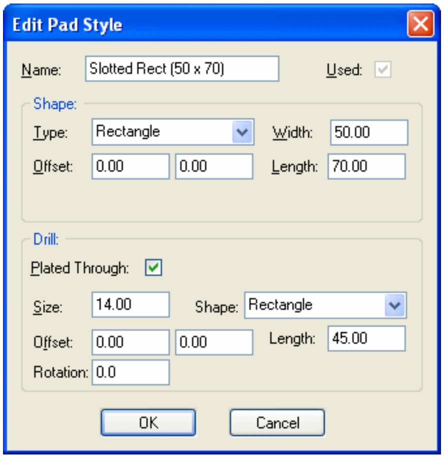
Non-round Drill Holes - Slotted Pads

Slotted Pads

You can now define non-round drill holes. You can do this using the **Edit Pad Style** dialog from the **Technology** dialog and **Pad Styles** page or the using **Define Pad Shape** editor. **Drill Holes** then show as **Special** in the Technology file.



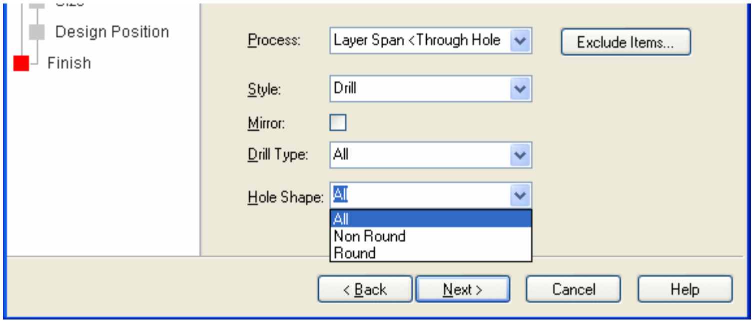
When editing pad styles, you can choose from the list of **Drill Shapes** available from the drop down list or you can create your own shape using the Define Pad Shape editor.



Outputting non-round drills

When post processing a layer span, you can specify if round and non-round holes will come out. Non-round holes will be milled.

The **Cam Plot Wizard** now allows you to specify the **Hole Shape:** when the **Excellon** output is selected.

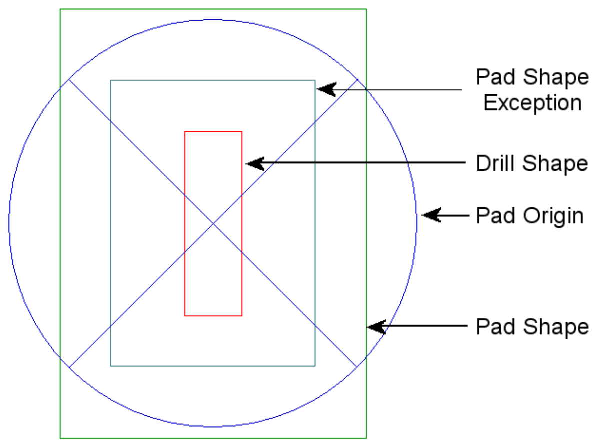


Finger (Oval) shaped holes are milled using a single line of the appropriate width, rounded rectangles are milled using a drill of the corner radius.

If you require Rectangle shaped holes to be punched, your normal selection would be for **Hole Shape: Round** and for no additional output. It is assumed you will not mix punched and milled slotted holes. Punched holes would then be output through the **Report Maker** using the new **List of Drills** command and the commands within it.

Pad Shape Editor Changes

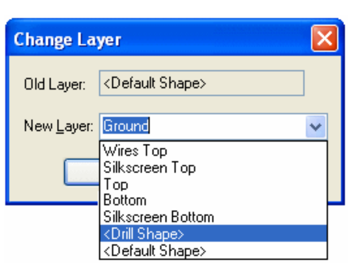
The Pad Shape editor now shows the pad exception shapes as well as the default shape and drill hole shape, you can edit, add, delete any of these shapes.



In V5.0, the ability to see and modify the drill hole which you couldn't previously do has been added.

Edit an existing pad style in the Pad Shape Editor and you can delete, resize, move or edit the red circle drill shape. You are able to delete it first then add a new shape (a Rectangle, for example) to draw in a slotted pad. If you then select it, you can use **Change Layer** to select a new special layer of **<Drill Shape>**, this specifies the shape to be the drill hole.

If editing the pad shape, use one of the other layers to add a pad exception. You no longer choose a layer to exist but can add exceptions at the same time.

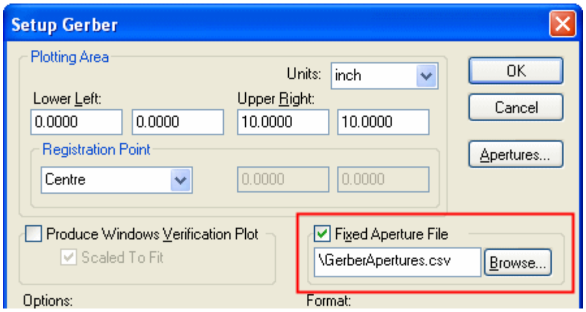


The **<Default Shape>** entry shown means that the selected shape will appear on all the layers in the layer stack unless changed to a specific layer.

CSV Import of Gerber Apertures

From within the **CAM Plot** dialog and **Setup**, **Gerber**, you can now opt to use a fixed format aperture table from a CSV file. Any attempt to use apertures that are not defined by this file will cause an error to be reported during plotting.

To use this option, select the **Fixed Aperture File** check box and choose your CSV file using the **Browse** button. The file name selected will be displayed.



Clicking on the **Apertures** button will display a list of the fixed apertures that will be used.

When the Gerber output is run, if the aperture required doesn't exist, the plot is not output and the report file will contain a list of missing apertures.

CSV File Definition

The CSV file should contain aperture definitions in a particular format. The first line can show the field names for ease of reference when editing the CSV file. The fields required are shown in this example:

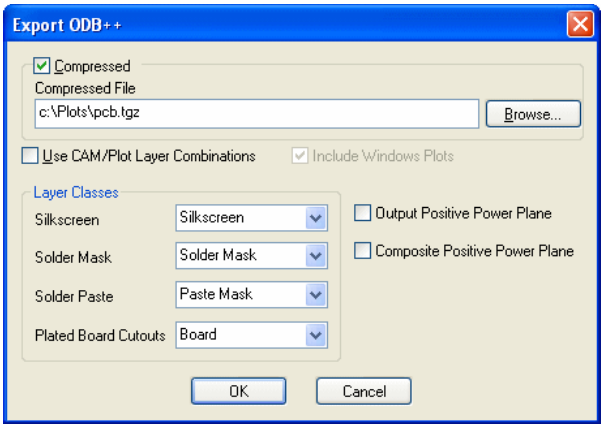
```
dcode,shape,width,length,cornerradius,rotation,hole,offsetx,offsety
72,round,0.6mm,0.6mm,0,0,0,0,0
73,round,1.4mm,1.4mm,0,0,0.9mm,0,0 10,square,60,60,0,0,32,0,0
```

The default units are thou/mils, but you can use mm by appending "mm" to each value as shown in the example.

Change to ODB++ Output

The ODB++ Output has been modified to allow slotted pads and board cutouts to be exported to be processed for routing.

Select the layer required from the drop down list under **Plated Board Cutouts**. The drop down list will display **Non-electrical Layer Class** layers available for selection.



When processed, a new layer will appear in the ODB++ file which will have a milling profile created for it.

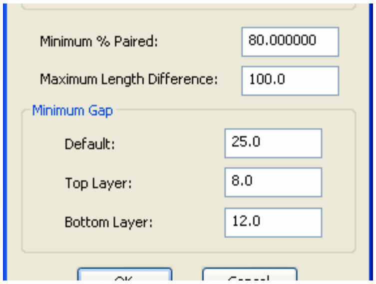
Changes to the High Speed Cost Option

Diff Pairs - by layer Gap and Width

Within the High Speed option, you can now define the gap used on Inner layers and Top and Bottom. Previously, you could only define the gap used for all layers occupied by the differential pair tracks. A change has also been so that when changing layers, not only is the gap changed but also the track width. The track width change relies on a new switch being enabled in the Net Class dialog.

Differential Pair Gaps

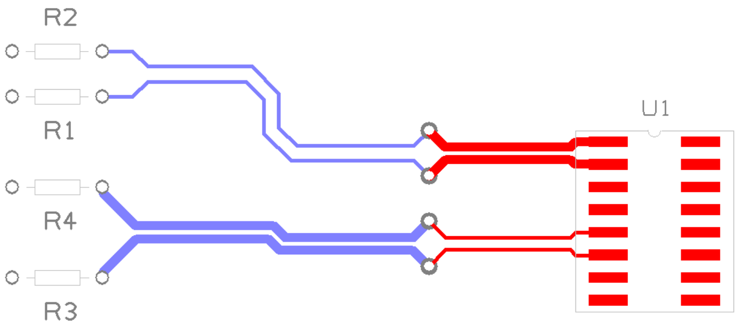
Within the **Differential Pairs** option of the **Technology** file, you can now define the pair Gap for the Top and Bottom layers, plus the Default. The **Minimum Gap** is the minimum distance the pair of tracks can be. This can be less than the normal Spacing rules for the nets (which would otherwise be the value used). This is the gap used by the Differential Pair Paired Track routing feature. You can define different values for the **Top** or **Bottom** Layer, if either of these values is *Undefined*, then they take the **Default** value. If the **Default** value is *Undefined*, then it takes the appropriate Track to Track spacing rule value.



Alternate Track Style on Inner Layers

There is a check box on **Net Classes** dialog in the **Technology** file to use the alternate style when routing on inner layers (**Alternate Style on Inner Layers**). When a layer change is performed, with this switch set, the track width will automatically change to the alternate style.

An example of a design using differential pairs might look like this:



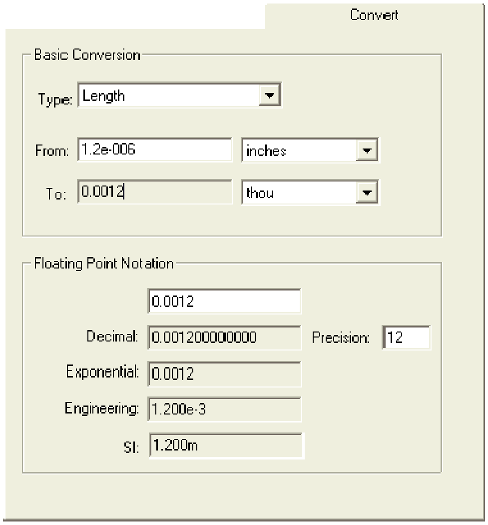
Design Calculators

The **Design Calculators** option can be accessed from the **Utilities** menu. This new option contains seven calculators, each accessed from a tab at the top of the dialog. Each of these calculators is explained in its own section in this document.

Conversion Calculator

This is a basic tool for converting numbers from one unit or notation to another.

The **Convert** dialog appears when the tab is selected.



Basic Conversion

Choose the **Type** of value you are converting, **Length** or **Angle** for example.

Enter the original value to be converted into the **From:** edit box, and select the units you are converting from using the drop down list to its right. Select the units you are converting to from the drop down list to the right of the edit box labelled **To:**.

The converted value will be displayed in the To: edit box. It will also be entered automatically into the *Floating Point Notation* edit boxes to show the value using different numeric notations.

Floating Point Notation

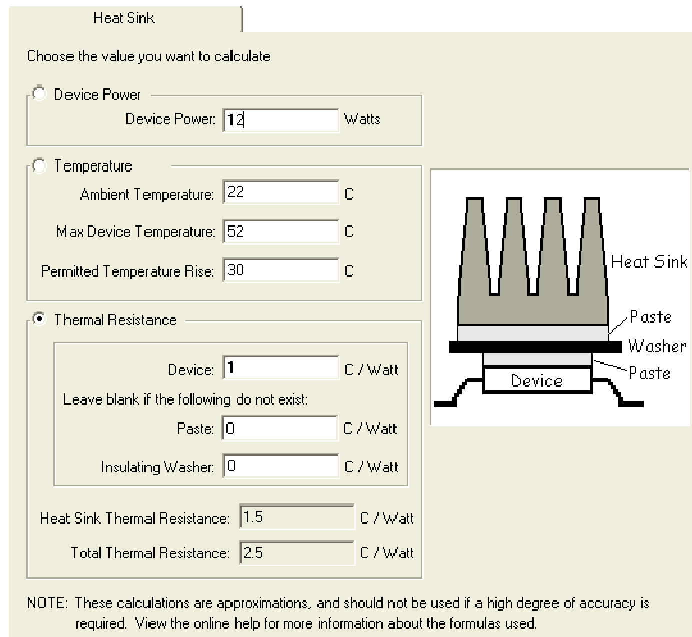
Enter the number in the edit box using any notation you require. The number will then be shown in each of the different floating point notations. The Precision for the decimal point notation can be entered to set the number of decimal places shown after the decimal point.

Note: Several of the other calculators will automatically set these values to their result when used.

Heat Sink Calculator

This is a calculator based around the calculation of thermal resistance in a heat sink. It can be used to help with the selection of a suitable heat sink for a transistor or other semiconductor device that is mounted in a case that is then mounted on the heat sink.

The **Heat Sink** dialog appears when the tab is selected.



This calculator uses the basic formula:

Total Thermal Resistance = Temperature Rise / Device Power

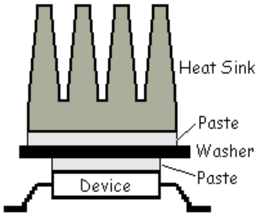
The **Temperature Rise** is the rise from the ambient air temperature to the maximum device temperature, often called the junction temperature of the device.

The **Total Thermal Resistance** is made up the following parts:

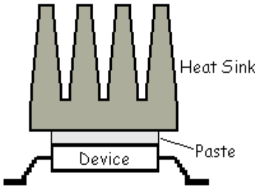
- The "Junction to Case" thermal resistance for the device.
  This figure can usually be obtained for the device from the manufacturers data, or can be calculated from the device case temperature.

The "Case to Sink" thermal resistance of the thermal interface material between the device and the heat sink.

This is normally either a layer of thermal paste (thermal compound):



or more likely an insulating washer with thermal paste either side:



- The thermal resistance between the heat sink base and the ambient air.

Using the dialog

First select the value you want calculated using the radio button on the appropriate section.

- Device Power
- Temperature Rise
- Heat Sink Thermal Resistance

The value being calculated will become a read only box. Enter the values in the other sections to produce the result. The result will be blank if not all the required values are entered.

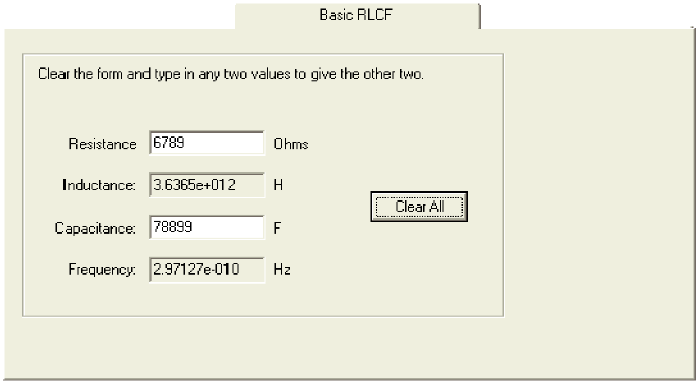
Once you have calculated the required result, you can switch to the **Conversion Calculator** tab to see the result in engineering or scientific notation.

Note: These calculations are approximations, and should not be used if a high degree of accuracy is required.

RLCF Calculator

This calculator page represents the relationship between Capacitance, Resistance, Inductance and Frequency in a basic RLC circuit. An RLC circuit (also known as a tuned circuit or a resonant circuit) is an electrical circuit consisting of a resistor (R), an inductor (L), and a capacitor (C), connected in series or in parallel. You can use this calculator to calculate any two of these values given the other two.

The **Basic RLFC** dialog appears when the tab is selected.



Press the Clear All button to clear all four values, ready for you to use. Enter the two values you know and the other two values will be calculated in read only boxes.

These calculations are based around the following two generally known equations:

$$f = \frac{1}{2\pi\sqrt{LC}}$$

and

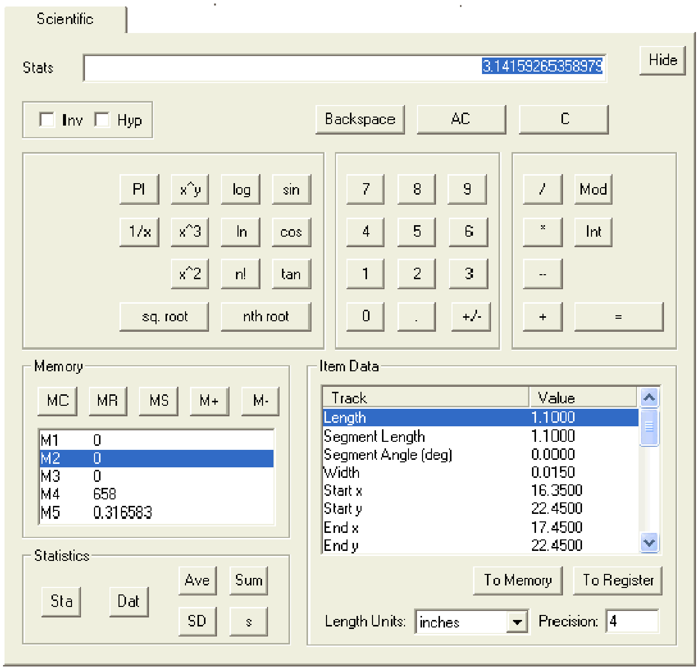
$$R = 2\pi f L$$

where R is the resistance, C is the capacitance, L is the inductance and f is the natural frequency of the tuned circuit.

Scientific Calculator

This is a scientific calculator with the ability to use values from picked PCB design items. The calculator performs basic arithmetic, such as addition and subtraction, as well as functions found on a scientific calculators, factorials and logarithms for example.

The **Scientific** dialog appears when the tab is selected.



Basic Use

The calculator works the same way as a hand held calculator.

▶ To use the calculator

1. Click on the digit, sign and decimal point buttons to enter a number. The number will be displayed in the **Register** at the top of the calculator. Use the **Backspace** key if you make a mistake whilst entering the number.
2. Click on an operation button, **+** to add, **-** to subtract, **/** to divide, or ***** to multiply.
3. Enter the next number in the calculation.
4. Continue to press operators followed by numbers until all numbers have been entered.
5. Click on the equals button to give the result in the register. For example, 4 + 5 + 2 * 3 = should give the result 33.

You also can enter the digits and operation keys directly from the keyboard, and copy and paste numbers from elsewhere in this application, or from other applications, to the register using standard windows cut/copy/paste operations.

At any time use the C (Clear) button to clear the register before entering numbers, or use the AC (All Clear) button to clear the register, all memory and the current operation.

Scientific Operations

The group of buttons to the right of the number buttons perform scientific calculations on the numbers you enter. Some of them are single click calculations on the number in the register, and some need another number to complete the operation followed by an equals to present the answer. PI enters the pi constant into the register, ready for the next operation. The trigonometric functions operate on numbers in degrees.

Some examples:

- Pressing "2" followed by "x^3" performs 2 to the power 3 and the result 8 is placed in the register.
- The sequence "5", "x^y", "4", "=" performs 5 to the power 4 and the result 625 is placed in the register.
- The sequence "6", "4", "nth root", "3", "=" performs the cube root of 64 and the result 4 is placed in the register.
- Pressing "4", "5" and "tan" calculates the trigonometric tangent of 45 degrees, and the answer 1 is displayed.

Trigonometric Operations

Check the "Inv" box to change the trigonometric operations to perform the inverse, or arc functions. For example, use the sequence "Inv" "1" "tan" to perform the arctangent of 1, to find which angle gives a tangent of 1, so the answer 45 is displayed.

Check the "Hyp" box to change the trigonometric operations to perform hyperbolic operations. For example, use the sequence "Hyp" "1" "tan" to perform the hyperbolic tangent of 1 giving the answer 0.761594 (which is pi/4). Use "Inv" and "Hyp" to calculate the inverse of the hyperbolic function.

Note, if you know an angle in radians and want to perform a trigonometric function on it, you can convert it to degrees using the **Conversion Calculator** and paste the converted value into the register to perform the operation.

Logarithms

Use "log" to find the base 10 logarithm of a number and use "ln" to find the natural logarithm. Checking the "Inv" box makes "ln" perform the exponential of the number in the register.

Memory Operations

The group of buttons in the Memory box allow you to save numbers to one of five memory locations, and recall these numbers when they are required at a later stage in your calculations. First select which of the five memory fields you will be using, and then press one of the memory buttons as follows.

- Click **MS** (Memory Store) to copy the number from the register to the chosen memory field.
- Click **MR** (Memory Recall) to copy the number from the chosen memory field back to the register.
- Click **MC** (Memory Clear) to set the current memory field to zero.
- Click **M+** (Memory Add) to add the number from the register to the chosen memory field.
- Click **M-** (Memory Subtract) to subtract the number in the register from the chosen memory field.

Statistical Operations

The group of buttons in the **Statistics** box allow you to enter data in the form of a series of numbers, and calculate basic statistical information on the data.

▶ To use the statistical operations

1. Press the **Sta** button to enter Statistics mode. In this mode Stats will be displayed to the left of the register, and all the statistics buttons are enabled.
2. Enter a number into the register and press the **Dat** button to add it to the data list. Repeat this for each number in the data series.
3. When all numbers have been entered you can use one of the statistical operations.
4. Click the **Sum** button to show the sum of the data.
5. Use the **Ave** button to display the average of the numbers.
6. Use the **SD** button to show the standard deviation of the data.
7. Use the s button to show the sample standard deviation of the data.
8. Press the **Sta** button again to exit the Statistics mode. The data will be cleared, ready for next time.

Design Item Data

This is available when editing PCB designs or PCB symbols. The Item Data box has a list of numerical information from the selected design item. To use this data, click on the item of interest in the list and use the To Memory button to store the selected value in the currently selected memory field, or use the To Register button to copy the number to the main register at the top of the calculator.

Use the **Length Units** and **Precision** controls to change the way the design data is displayed in the list only.

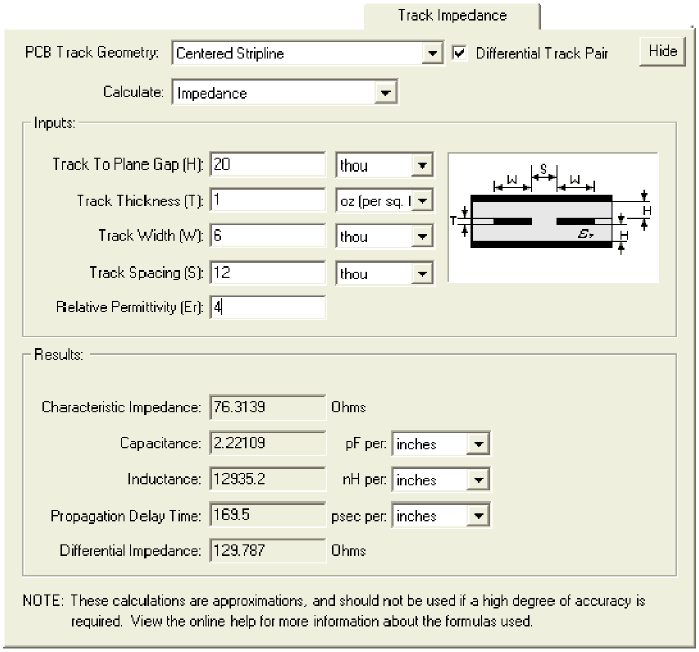
To change the item that information is displayed for, move the cursor out of the dialog and you will be presented with a calculator cursor. Use this to pick on the next item from the design. Keep the shift key pressed down to select whole shapes or tracks, or keep the control key down to just select a position instead of a design item. If you can not see enough of the design, use the Hide button at the top right of the dialog to hide the dialog until the mouse key is clicked. Pressing the Escape Key, or the cross at the top of the dialog will exit the calculator picking mode back to normal select mode.

You can use the picking to measure the approximate distance between two picked points. After the second pick you will find Picked Distance and Picked Offset in the item list. The distance is the direct line between the picked points and the offset gives the X and Y coordinates. Keep the control key down to avoid items while picking if you only want to see the point information.

Track Impedance Calculator

This calculator calculates the characteristic impedance of a unit length PCB track for a set of common track geometries, or it can be used to calculate the required track width given a known approximate impedance value.

The **Track Impedance** dialog appears when the tab is selected.



Track Geometry

There are two controls that define the track configuration to be used in the calculations.

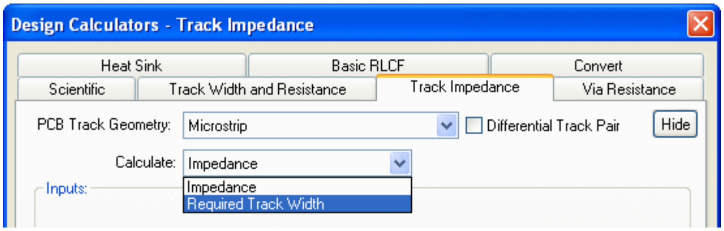
First drop down the PCB Track Geometry list at the top of the dialog and choose the geometry that matches your track situation.

Then use the check box below the list to select whether you want the calculation for a Differential Track Pair or a single track.

The diagram in the dialog will be changed to show the track configuration you have chosen.

What to Calculate

Drop down the **Calculate** list at the top of the dialog and choose which value you would like to calculate. The controls in the Inputs section will alter to provide the inputs for the calculation you have chosen and the Result section will display the chosen value.



Inputs

The controls in this section will be changed to match the chosen configuration and each will include a label in brackets referring to a label on the diagram.

Enter the correct values to generate the required result. Make sure the units drop down list is correct for the values you have entered.

If you are calculating the track impedance, you can enter the **Track Width** by picking on a track in the PCB design. To do this, move the cursor out of the dialog and you will be presented with a calculator cursor. Use this to pick on the track in the design and its width will be automatically entered into the dialog. If you use shift click to select a whole track, the largest width will be used. If you can not see enough of the design, use the Hide button at the top right of the dialog to hide the dialog until the mouse key is clicked.

The Track Spacing is only presented for differential paired tracks and is the gap between the track pair. This cannot currently be extracted from the design.

If you are calculating the required track width, you will be presented with the Characteristic Impedance control to enter the known value.

Results

This section shows the results of the calculations using the formulas listed below. Some calculations have constrains on the relationship between parameters and, if these are not met, an error message is shown on the top line of the Results section. The message will show the relationship and state a minimum or maximum value.

Once you have calculated the Characteristic Impedance, you can switch to the **Conversion Calculator** tab to see the result in engineering or scientific notation.

Formulas

The formulas used are taken from "The Design Guide for Electronic Packaging Utilising High Speed Techniques" IPC-2251 document.

NOTE: These calculations are approximations, and should not be used if a high degree of accuracy is required.

Microstrip

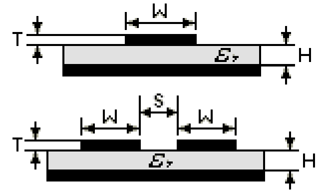
$$Z_0 = \frac{87}{\sqrt{\varepsilon + 1.41}} \ln\left(\frac{5.98H}{0.8W + T}\right) \text{ohms}$$

$$C_0 = \frac{0.67(\varepsilon + 1.41)}{\ln\left(\frac{5.98H}{0.8W + T}\right)} \text{ pF/inch}$$

$$L_0 = C_0 \times Z_0^2 \text{ nH/inch}$$

$$T_{pd} = C_0 \times Z_0 \text{psec/inch}$$

$$Z_{diff} = 2 \times Z_0 \times \left(1 - 0.48 \times \exp\left(\frac{-0.96 \times S}{H}\right)\right) \text{ohms}$$



Embedded Microstrip

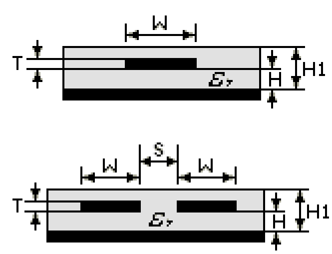
$$Z_0 = \frac{87}{\sqrt{\varepsilon_r + 1.41}} \ln\left(\frac{5.98H}{0.8W + T}\right) \times \left(1 - \frac{H_1 - T - H}{0.1}\right) \text{ohms}$$

$$T_{pd} = 84.75 \times \sqrt{0.475 \times \varepsilon_r \times \left(1 + e^{-1.55 H_1 H}\right) + 0.67} \text{psec/inch}$$

$$C_0 = \frac{T_{pd}}{Z_0} \text{pF/inch}$$

$$L_0 = C_0 \times Z_0^2 \text{ nH/inch}$$

$$Z_{diff} = 2 \times Z_0 \times \left(1 - 0.48 \times \exp\left(\frac{-0.96 \times S}{H_1}\right)\right) \text{ohms}$$



Centred Stripline

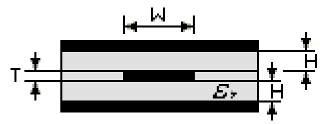
$$Z_0 = \frac{60}{\sqrt{\varepsilon_r}} \times \ln\left(\frac{4 \times (2H + T)}{0.67\pi \times (0.8W + T)}\right) \text{ohms}$$

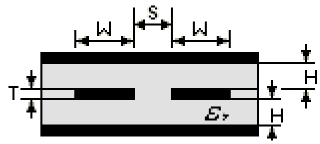
$$T_{pd} = 84.75 \times \sqrt{\varepsilon_r} \text{ psec/inch}$$

$$C_0 = \frac{T_{pd}}{Z_0} \text{pF/inch}$$

$$L_0 = C_0 \times Z_0^2 \text{ nH/inch}$$

$$Z_{diff} = 2 \times Z_0 \times \left(1 - 0.347 \times \exp\left(\frac{-2.9 \times S}{(2 \times H + T)}\right)\right) \text{ohms}$$





Asymmetric Stripline

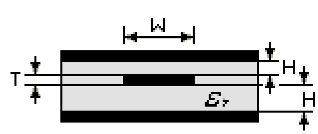
$$Z_0 = \left(1 - \frac{H}{4 \times H_1}\right) \times \frac{80}{\sqrt{\varepsilon}} \times \ln\left(\frac{4 \times (2H + T)}{0.67\pi \times (0.8W + T)}\right) \text{ohms}$$

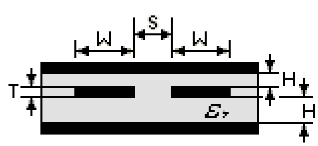
$$T_{pd} = 84.75 \times \sqrt{\varepsilon} \text{ psec/inch}$$

$$C_0 = \frac{T_{pd}}{Z_0} \text{pF/inch}$$

$$L_0 = C_0 \times Z_0^2 \text{ nH/inch}$$

$$Z_{diff} = 2 \times Z_0 \times \left(1 - 0.347 \times \exp\left(\frac{-2.9 \times S}{(H \times H_1 + T)}\right)\right) \text{ohms}$$





Dual Stripline

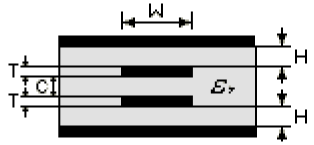
$$Z_0 = 0.5 \times \left(\frac{60}{\sqrt{\varepsilon_r}} \times \ln\left(\frac{8H}{0.67\pi \times (0.8W + T)} \right) + \frac{60}{\sqrt{\varepsilon_r}} \times \ln\left(\frac{8(H+C)}{0.67\pi \times (0.8W + T)} \right) \right) \text{ohms}$$

$$T_{pd} = 84.75 \times \sqrt{\varepsilon} \text{ psec/inch}$$

$$C_0 = \frac{T_{pd}}{Z_0} \text{pF/inch}$$

$$L_0 = C_0 \times Z_0^2 \text{ nH/inch}$$

$$Z_{diff} = 2 \times Z_0 \times \left(1 - 0.48 \times \exp\left(\frac{-0.96 \times S}{H_1} \right) \right) \text{ohms}$$



Note: These calculations are approximations, and should not be used if a high degree of accuracy is required.

References

"The Design Guide for Electronic Packaging Utilising High Speed Techniques" IPC-2251 document

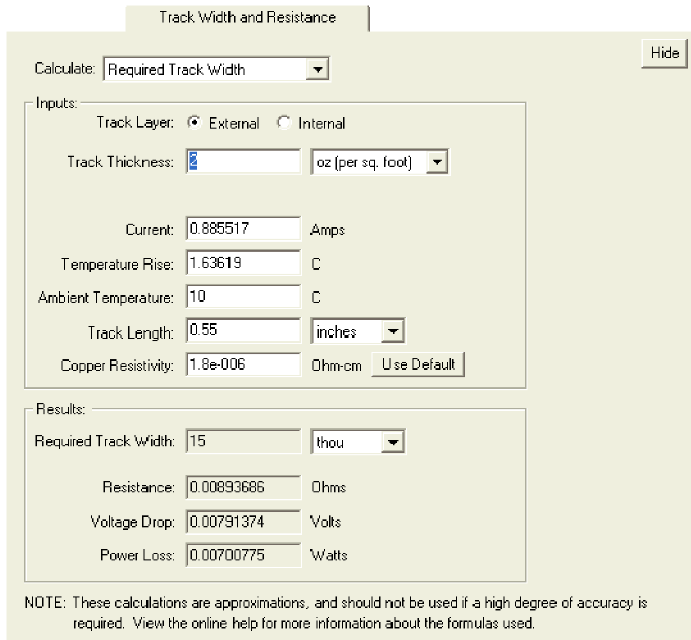
"PCB Impedance Control: Formulas and Resources", by Doug Brooks.

Based on the "PCB Trace Impedance Calculator" from the University of Missouri-Rolla EMC Laboratory.

Track Width Calculator

This calculator represents the relationship between the track width, the current applied to it and the temperature rise when that current is applied. Given any two of these values the third can be calculated. You can enter the track width, length and layer by picking on a track in the design.

The **Track Width and Resistance** dialog appears when the tab is selected.



What to Calculate

Drop down the Calculate list at the top of the dialog and choose which value you would like to calculate. The controls in the Input section will alter to provide the inputs for the calculation you have chosen and the Result section will display the chosen value.

Inputs

The values in this section are required to perform the calculation.

Select the Track Layer. Choose whether the track is on an outer layer, or an inner layer of the board. The formula uses curve fitting following the IPC-2221 guidelines, which uses a different constant for tracks on outer and inner layers.
You can also enter this value by picking on a track in the PCB design. To do this, move the cursor out of the dialog and you will be presented with a calculator cursor. Use this to pick on the track in the design and its side will be automatically entered into the dialog. If you can not see enough of the design, use the Hide button at the top right of the dialog to hide the dialog until the mouse key is clicked.

Enter the Track Thickness and select the units of the value being entered. This is the thickness of the copper used for the track. You can choose to define the thickness using ounces per square foot, which we convert to thickness using 1.378 thou per oz.

If calculating the Current or Temperature Rise the Track Width is required. Enter the value and select the units of the value being entered. You can also enter this value by picking a track segment in the design using the method mentioned above. If the whole track is selected (by holding the shift key down) the maximum width on the track will be entered.

If calculating the Track Width or Temperature Rise the Current to be applied to the track is required. Enter the value in amps.

If calculating the Track Width or Current the expected Temperature Rise the track will go through is required. Enter the value in degrees Celsius.

Resistance Inputs

The next three parameters are only required if the Track Resistance, Voltage Drop or Power Loss are required.

Enter the Ambient Temperature of the board in degrees Celsius.

Enter the Track Length and select the units of the value being entered. You can also enter this value by picking a track segment in the design using the method mentioned above. If the whole track is selected (by holding the shift key down) the total track length will be used, otherwise the length of the selected track segment will be used.

Enter the Copper Resistivity. If you don't know the Resistivity of the copper, it can be set to the default value of 1.7e-006 by pressing the Use Default button.

Results

Once legal values have been entered for all input sections the result will be automatically calculated. The fields will be left blank if not enough information has been entered. The formulas used to generate these results are detailed below.

Formulas

The formula used to represent the relationship of the three values (Current, Width and Temperature Rise) is as follows:

$$I = k * \Delta T^{B1} A^{B2}$$

where:

I is the Current applied to the track in amps.

T is the Temperature Rise above the ambient, in degrees C.

A is the cross section area of the track in square thou. This is the Track Width times the Track Thickness.

k, B1 and B2 are constants determined by applying least squares fit and multiple regression techniques to actual measured PCB data for different track widths and currents, such that the difference between the actual temperature rise and the calculated value using the constants in the above formula is minimised. The Doug Brooks paper on "Temperature Rise in PCB Traces" goes into this technique in more detail. The constants we will use are taken from the IPC-2221 design standard for PCB track widths, which uses curve fitting to estimate the following values:

B1 = 0.44
B2 = 0.725
k = 0.024 (for internal layers)
k = 0.048 (for external layers)

The formula for track resistance used in this calculator is:

Resistance = ((Resistivity * Length) / Area) * (1 + TC * (Temp - 25))

Voltage Drop = Current * Resistance

Power Loss = Voltage Drop * Current

where:

Resistivity is the copper resistivity entered in the dialog, and defaults to 1.7e-006.

Length is the Track Length entered in the dialog.

Area is the track cross-section area mentioned above.

Temp is the Ambient Temperature plus the Temperature Rise, both from the dialog.

TC is the linear temperature resistance coefficient of copper and we use the value 3.9e-003 ohm/ohm/C taken from the last reference below.

NOTE: These calculations are approximations, and should not be used if a high degree of accuracy is required.

References

"Constructing Your Power Supply - Layout Considerations", by Robert Kollman

"Temperature Rise in PCB Traces", by Doug Brooks

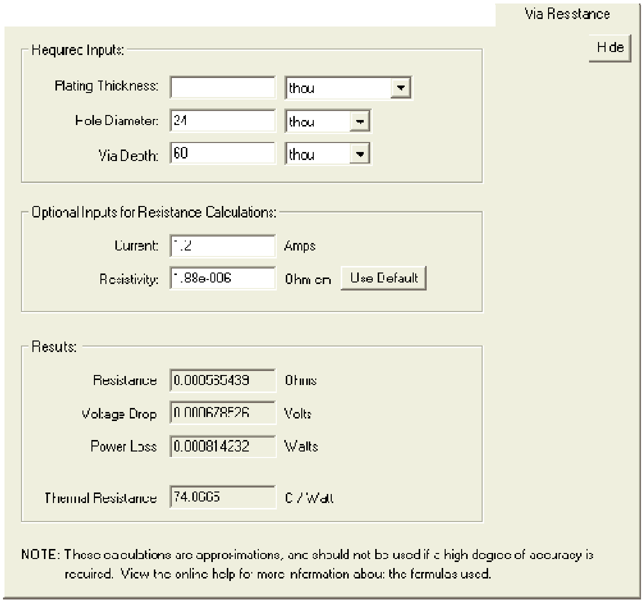
The IPC-2221 design standard for PCB track widths.

"Temperature coefficient of resistance" - by John N Fox

Via Resistance Calculator

This is a calculator to calculate the resistance, voltage drop, power loss and thermal resistance of vias in a PCB design. It has the ability to take the hole diameter from a via selected in the design.

The **Via Resistance** dialog appears when the tab is selected.



Required Inputs

The values in this section are required to calculate the thermal resistance and the resistance for the via.

Enter the Plating Thickness of the copper the via is plated with, and select the units the value being entered is using. You can choose to define the thickness using ounces per square foot, which we equate to thickness using 1.378 thou per oz.

Enter the Hole Diameter of the via. This is the drill diameter before plating is applied. You can also enter this value by picking on a via in the PCB design. To do this, move the cursor out of the dialog and you will be presented with a calculator cursor. Use this to pick on the via in the design and its drill size will be automatically entered into the dialog. If you can not see enough of the design, use the Hide button at the top right of the dialog to hide the dialog until the mouse key is clicked.

Enter the Via Depth, i.e. the length of the via through the board.

Optional Inputs

The Resistivity in this section is required for calculate the via Resistance, and the applied Current is required to calculate the expected Voltage Drop and resultant Power Loss.

If you don't know the plated copper Resistivity, it can be set to the default value of 1.9e-006 by pressing the Use Default button.

Results

Once legal values have been entered for all input sections the results will be automatically calculated. The fields will be left blank if not enough information has been entered. The formulas used to generate these results are detailed below.

Formulas

The formula for thermal resistance used in this calculator is:

Thermal Resistance = (Thermal Resistivity * Via Depth) / Via Area

where:

Thermal Resistivity for plated copper is assumed to be 0.249 cm-K/W (at a temperature of 300K)

Via Depth is the length of the via through the board, entered directly in the dialog.

Via Area is the cross section of the plating, defined by (PI * (Drill Diameter - Plating Thickness) * Plating Thickness)

The formula for via resistance used in this calculator is:

Resistance = (Resistivity * Via Depth) / Via Area

Voltage Drop = Current * Resistance

Power Loss = Voltage Drop * Current

where:

Resistivity is the plated copper resistivity entered in the dialog, and defaults to 1.9e-006.

Via Depth and Via Area are as mentioned above.

Current is the current applied to the via in amps, entered directly in the dialog.

NOTE: These calculations are approximations, and should not be used if a high degree of accuracy is required.

References

"Constructing Your Power Supply - Layout Considerations", by Robert Kollman

"Current Carrying Capacity of Vias", by Doug Brooks and Dave Graves

"Appendix B", Materials Science and Engineering - An Introduction, by William Callister

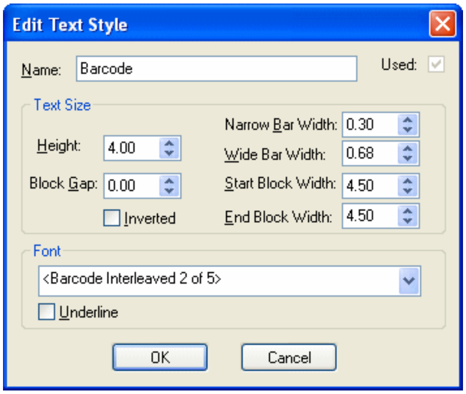
Additions to V4.6 but not previously documented

There are a few features which were introduced late into Pulsonix V4.6 and were not previously generally documented.

Barcode Text

You can add barcodes into you design using a special text font.

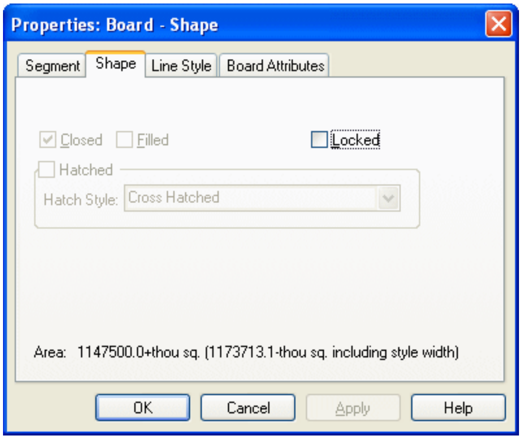
From within the **Technology** file and **Text Styles** you can create a **font style** as normal. When you select the font however, choosing the built-in font of **<Barcode Interleaved 2 of 5>** from the drop down list, it will present you with a list of parameters that can be assign to that font. These parameters describe the barcode features.



Once the font has been defined, in the design, use the **Insert Text** option. In the text box presented, type in the barcode in numbers, for example 05505223. Choose a suitable layer for it to be added to. Once added to the design, use **Change Style** and select the newly created Barcode text style. The text will now be displayed as a barcode.

Properties Shows Calculated Area

For a selected shape, if the shape is closed, even if just one segment of that shape is selected, the **Properties** dialog will now show the total **Area** in your current design units.



New Component Mirroring Defaults

You can default mirroring of SMD and through-hole components separately using the two check boxes in the **Design Settings**, **Defaults** and **Component** dialog.

When Components are added to the design, this will force their ‘side’ automatically.