



Pulsonix Design System

V7 Update Notes

Vertrieb für Deutschland, Österreich und Schweiz:

PESCHGES VARIOMETER GmbH

D-52078 Aachen, Zieglerstr. 11

Tel: 0241 - 56 30 23
Fax: 0241 - 56 39 13

www.peschges-eda.de

E-mail: vertrieb@peschges-eda.de

2 Pulsonix Version 7.0 Update Notes

Copyright Notice

Copyright © WestDev Ltd. 2000-2011

Pulsonix is a Trademark of WestDev Ltd. All rights reserved. E&OE

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

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

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

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

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

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

All other trademarks are acknowledged to their respective owners.

Pulsonix, a division of WestDev Ltd.

Printed in the UK Issue date: 20/04/11 iss 2

Pulsonix

Oak Lane

Bredon, Tewkesbury

Glos, GL20 7LR

United Kingdom



Contents

CHAPTER 1. VERSION 7.0 UPDATE SUPPLEMENT	5
Installing the New Version of Pulsonix.....	5
Licensing	5
Windows 7 Support	5
Changes to Folders	6
Support for Macros.....	6
Active-X Scripting Language Support.....	10
Support for STEP Format.....	13
New Base Auto Router	18
Using the Auto Router	19
Design Rule Check – Track Layer.....	20
Changes to Technology dialog	21
Layer Class – Exception Only	21
Layer Class – Draw Component Mounting Holes.....	21
Layer Class – Physical Copper Layers.....	22
Layers – Materials	22
Layers – Thickness	23
Layers – New Track Layer Bias Type.....	24
Design Rules – Minimum Pad Land	24
Net Classes – Edit dialog changes	25
Net Classes – Track Size Limits	26
Net Styles – Track Styles	28
Spacing Rules – Match Net Class Pair.....	28
DFM Rules – Thermal Rules.....	29
DFM Rules – Testpoint Rules	29
DFM Rules – Teardrop Rules.....	30
Design Rules – Drill Spacing	30
Pad Styles – Restrict Usage of Pad Styles.....	30
Advanced Technology Features.....	31
Layers Classes.....	31
Layers	31
Layers Now Have a ‘Facing’ Flag	32
Layer Spans.....	32
Composite Layer Spans.....	32
Micro-vias / Laser Drilled Vias	34
Micro-via Keep In/Out Areas	35
Board Outline and Areas on Layer Spans.....	35
Component Spans.....	37
New Items in Colours dialog.....	38
Changes to Bond Pad Options	39
Select All Bond Pads	39
Move Chip Body	39
Change Layer of Bond Pads.....	39
Insert Track - Show Legal Completion Path.....	40
Insert Track - Hug Mode Improvements	41
Insert Track - Follow Construction Lines.....	42
Changes to Interactive High Speed Routing Option.....	43
Show Track Limits	43

4 Contents

Serpentine Differential Pair Tracks.....	46
Copy & Paste of Differential Pair Routing.....	46
Draw Hollow Segments.....	46
Testpoint Components.....	47
Improvements to Copying Cutouts.....	48
Drawing Dimmed in Uniform Colour.....	48
Number of Drill Symbol Shapes Increased.....	49
Change of Angle Resolution to 1/100,000 th Degree.....	49
Rotation to Polar Grids.....	50
Full Optimisation After Move.....	52
Find Library Items.....	53
Find – Count Find Items.....	54
Find – Via Spans.....	54
Find – Footprint in Schematics.....	54
Report Maker Command Additions.....	55
Is This User Report command.....	55
Report Maximum Track Length and Minimum Track Length.....	55
Report Start and End Positions of Oval Slots.....	56
Report Drill Shape, Slot Width & Slot Length.....	56
Is Micro-Via command.....	56
Spice Netlist Exporter.....	56
Save Copy As.....	58
Auto-Rename - Exclude Locked Items.....	58
Type Coordinate – Get From Clipboard.....	58
‘Align’ options available in Define Pad Shape.....	59
Variants Improvements.....	59
Filled Shapes with Dashed Line Styles.....	59
Name base for Variant Reports.....	59
Cadstar 12 support.....	59
Library Changes in V7.0.....	59
Mounting Holes Placed on Non-Electrical Layers.....	60
Cam Plot Changes.....	60
Alter Main Plot Area.....	60
Step and Repeat offset of 0.....	60
Import Spacings Rules using CSV Format Files.....	61
Using the Import CSV feature.....	61
Using the dialog.....	64
The Import CSV Report.....	64

Chapter 1. Version 7.0 Update Supplement

Installing the New Version of Pulsonix

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

To install Pulsonix, insert the CD or double-click on the download executable and wait for a short time. The *Autorun* facility will start the installation procedure. Follow the on-screen commands from the install wizard. You can install Pulsonix 7.0 on top of your existing installation or along side if you prefer; however, you do not need to uninstall the old version first.

Licensing

If you are using a version of Pulsonix earlier than Version 7.0, you will require a new license. This will be supplied to you by email under the contract conditions of your current maintenance agreement.

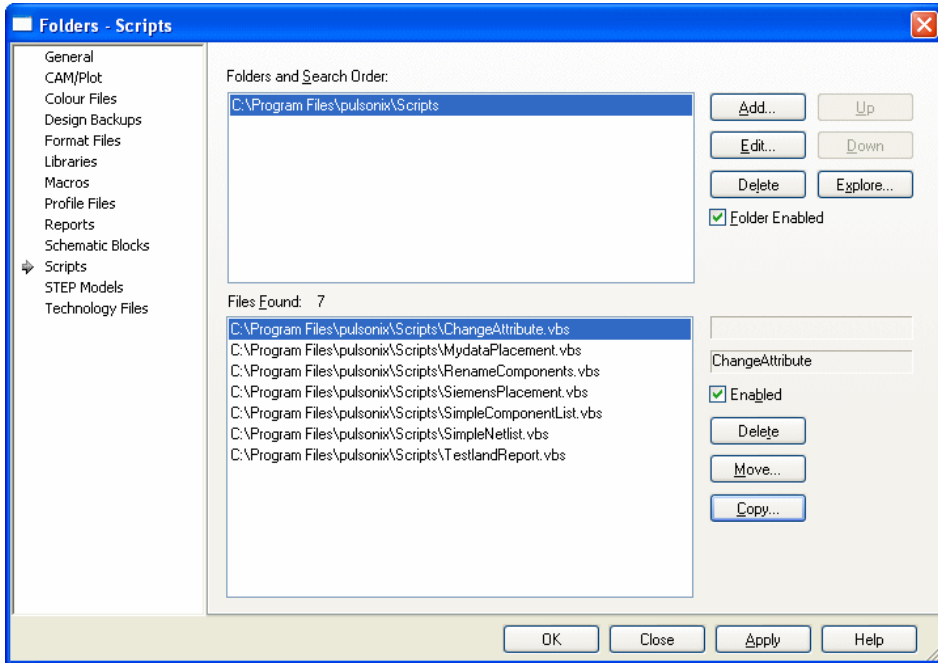
For existing users upgrading from a previous version, it is recommended that you save the new license and overwrite the existing one. When requested during installation, simply click the **No Change In Licensing** check box on the licensing page of the installation wizard. The **License Manager** can be used to add new licenses and make changes to network licensing after the installation has been completed.

Windows 7 Support

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

Changes to Folders

The Folders dialog has been changed for Pulsonix V7.0 to additionally allow the management of **Macros**, **Scripts** and **STEP Models**. Each of these is discussed below.

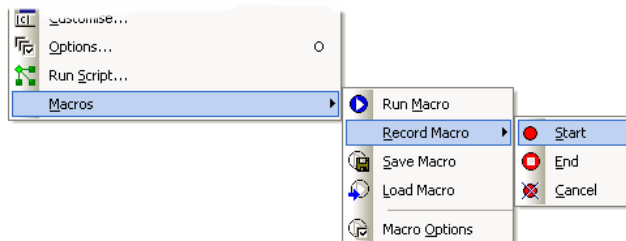


Support for Macros

You can now create and run macros within Pulsonix. This will record your keystrokes and dialog selections, then give you the ability to subsequently save and run them again to repeat the actions that were performed. Recorded macros can be saved so they may be reloaded and re-used at a later date.

Creating a new Macro

From the **Tools** menu, select **Macros** and **Record Macro**> then **Start**.

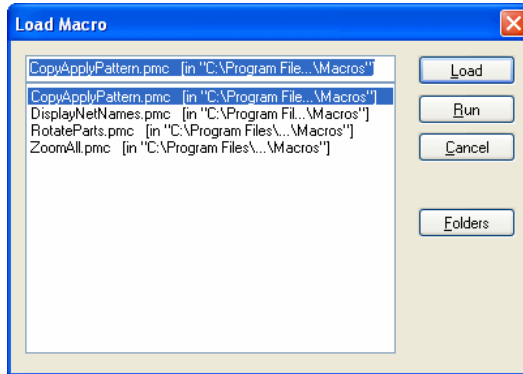


A new macro is now being recorded. You must use the **End** command from the **Macros** menu to finish recording it. If you change your mind and decide you don't wish to keep the macro you started, use **Cancel** from the **Macros** menu.

Once the macro has been recorded using Start and Stop it becomes the current macro and you can then replay it using the **Run Macro** command.

If you wish to save this macro, use the **Save Macro** option from **Macros** menu.

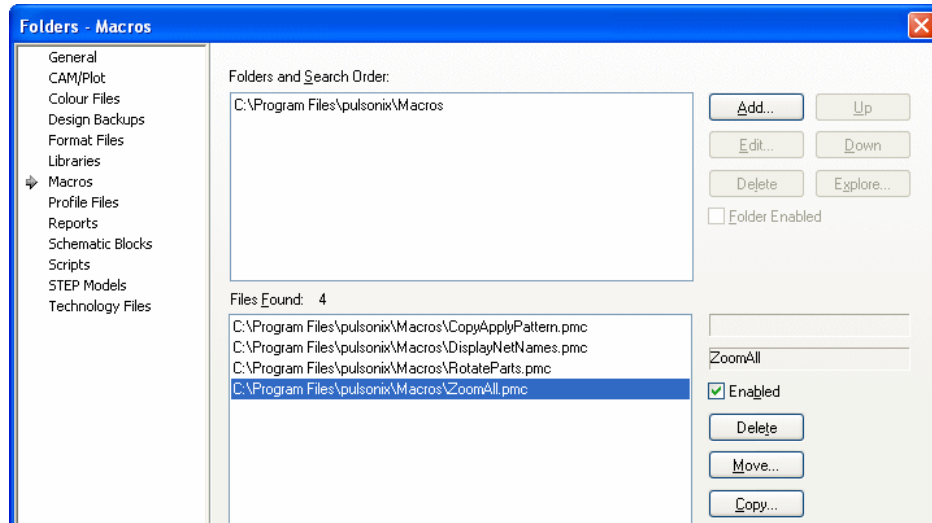
For any previously saved macro that you wish to run, use the **Load Macro** option from the **Macros** menu.



Once loaded, use the **Run Macro** button to run it (from the **Macros** toolbar), it won't automatically run after using **Load Macro**. If you pressed the **Run** button on a selected Macro, it will load and run at this point.

Macros Folders

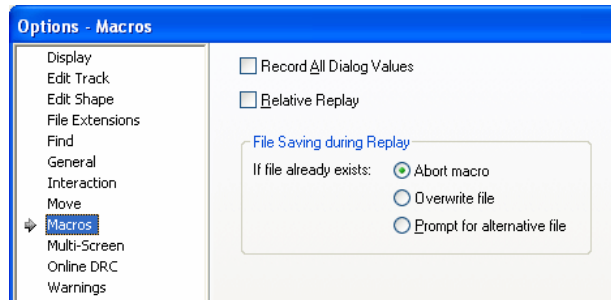
Macro files are located in a defined list of Macros folders managed from the **Macros** tab within **Folders**.



Running in Absolute or Relative mode

Within a design view, macros can be run in absolute or relative mode. Running in absolute mode means it will replay exactly the same as when it was recorded, a copy of whatever was created previously would be created at exactly the same position as before. A macro run in relative mode will replay relative to cursor position when the macro is run.

Macros can be switched to run in Absolute or Relative mode by selecting the **Relative Replay** check box on the **Options** dialog under **Macros**.



If **Relative Replay** is enabled and **Run Macro** is invoked while the mouse cursor is outside the design window, such as when selecting the Relative Replay option from the menu or toolbar, you are required to set the relative start position for the macro, using the mouse, before it will run. You will also see that the cursor has changed and is now a model cursor to indicate that a relative reference point is required.

Normally when a macro is recorded, only the dialog values that are changed are recorded. Enabling the **Record All Dialog Values** option will result in all values being recorded for a dialog. This can change the effect of the macro when it is replayed.

For example, consider recording the addition of a text item where in the Insert Text dialog the only item changed is the text string, to 'Example Text', which is then added on the default text layer 'Silkscreen Top'. If the macro is subsequently run in a design where the default text layer is set to 'Silkscreen Bottom' the 'Example Text' item will be added on that layer instead.

However, if the Record All Dialog Values option is enabled before the macro is recorded, although the Layer setting in the Insert Text dialog was not specifically changed its value will also be recorded. So when the macro is replayed, the text item will be added to the 'Silkscreen Top' layer rather than the design's default 'Silkscreen Bottom' layer.

File Saving During Replay

If a recorded macro contains a command to save out an item, such as the current design, when it is replayed it is possible that the target file may already exist. This option allows you to specify how you want a macro to behave in such an event.

Abort macro will cause a macro to stop immediately without overwriting, **Overwrite file** will cause an existing file to be silently overwritten by a macro and **Prompt for alternative file** will result in the **Save As** dialog being displayed to allow an alternative file name to be chosen for the save, after which the macro will resume replaying as normal.

Macros toolbar

The Macros toolbar is available to provide quick and easy access to all the Macro commands. If you intend recording macros from within a dialog, the Macros toolbar will need to be floating and not docked.

Initially, the toolbar allows you to **Start** a new macro, **Run** the current macro in memory, select macro **Options**, **Load** a macro and **Save** the current macro. Hovering your cursor over each of the toolbar buttons will display its function as normal.



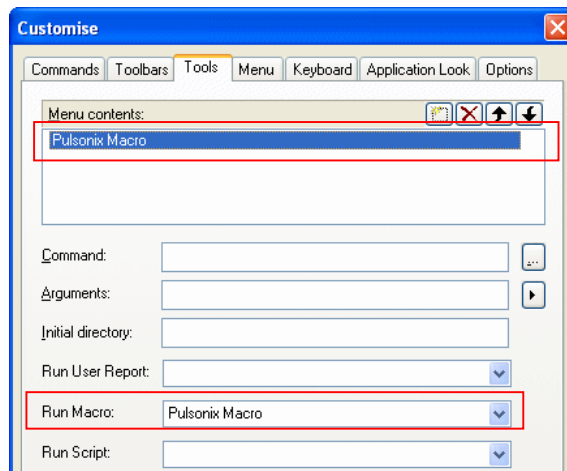
Once macro recording has been started the **Stop Macro** button and **Cancel Macro** button are available.



Assigning Shortcut Keys to your own Macros

Shortcut keys, **Toolbar** buttons and **User Tool** options can be assigned to macros using the **Customise** option.

You should first create a **User Tool** to assign the Macro to. This is done using the **Tools** tab and after assigning a name, select the macro using the **Run Macro:** option.



Once a User tool has been created using the Keyboard or Commands tab respectively, it can then be assigned to a shortcut key placed on a toolbar as a button.

Limitations of Macros

There are some small limitations when using macros. For example, you cannot create a macro for the **Customise** dialog or functionality within it.

Active-X Scripting Language Support

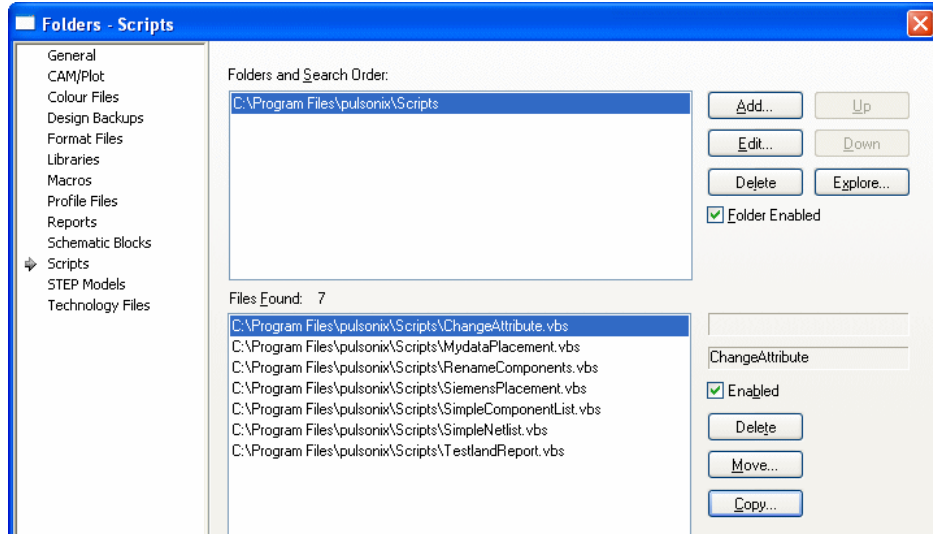
Pulsonix 7.0 supports Active-X Scripting languages. You can write scripts for a range of activities and functions. These can be run from within the Pulsonix environment or externally from a command line.

Supported Programs

Pulsonix scripting supports VB Scripts, Python script, JAVA script and Perl script to mention a few popular ones. This list includes almost any Active X scripting language that can be used by an Active X host.

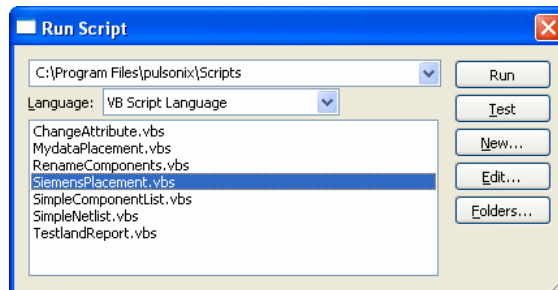
Scripts Save Folder

Scripts can be saved and loaded from the Scripts Folder. This is defined within **Folders** on the **Scripts** tab.



Running Scripts

From the **Tools** menu inside a Pulsonix design, you can use **Run Script** to get access to existing Scripts. This will allow you to select a script from the list and run it. Scripts shown on this list are from whatever is defined in **Folders** under **Scripts**.



Choose the **Run** button to run the script. This runs the script and closes the dialog.

Test allows you to try running the script without leaving the dialog.

New will open the **Edit Script** dialog ready for you to type in your new script. Once created, use the **Save** or **Save As** button to save it. You can also edit scripts using your favourite text editor or development tool, you must ensure you save the script to one of the scripts folders.

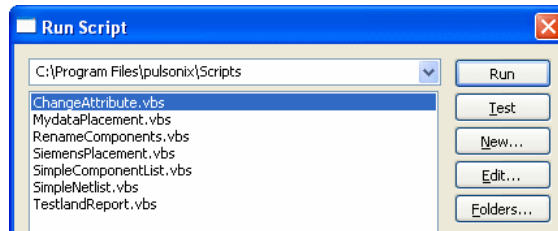


Edit will open the existing script in the **Edit Script** window ready for editing.

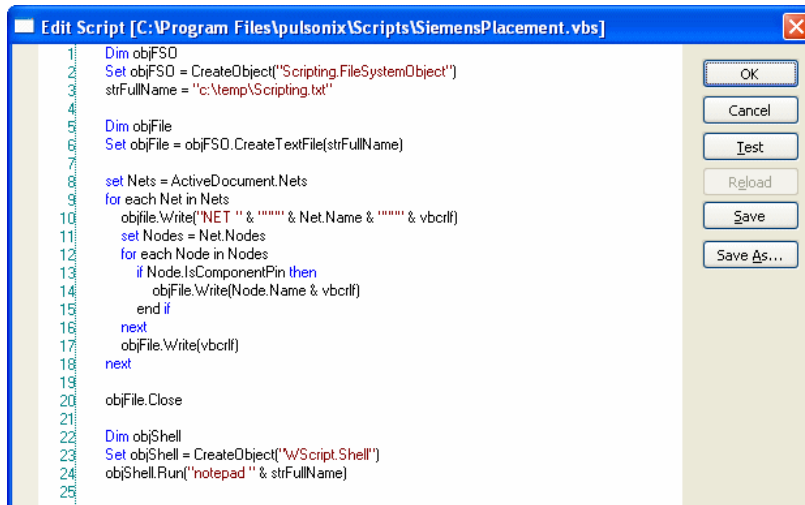
Folders opens the **Script** page of the **Folders** dialog.

Writing Scripts

To create a new script or edit an existing script, use the **Run Script** option from the **Tools** menu.

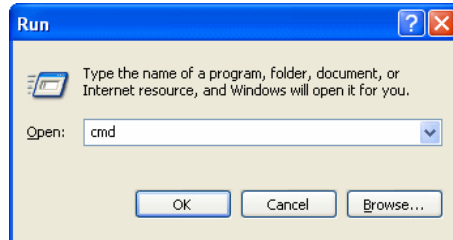


Use the **New** or **Edit** buttons to access the Edit Script dialog.



Running Scripts from a command Line

Command line driven scripts can be run outside the Pulsonix environment on a Pulsonix design or library. Pulsonix must be installed on the machine running the script in order to be able to do this. To run a command line script - run a command shell from the **Taskbar**. The command to use will vary but generally it will be **cmd**. This will display a command prompt window.



The scripts used from the command line, or within the program are the same, it is only the method of launching them that changes.

From within the command prompt window, type for example

```
Pulsonix /scriptfile myscript.js <Enter>
```

More information about running command line scripts is available in the Online Help.

Help for product and command line scripts

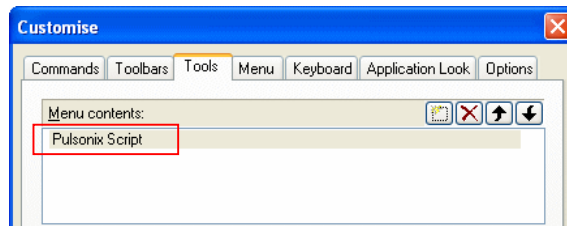
In the Online Help under Index, type **Scripting** and select the sub-category **Overview**. This will display help pages with an overview of scripting. For a full syntax of scripting available click on the **Scripting Reference** link at the bottom of the page, this will open a separate Scripting reference Help system with full grammatical references.

When running command line driven scripts, there is also Online Help available, please refer to the Index entry **Scripting** and **Command Line Scripts**.

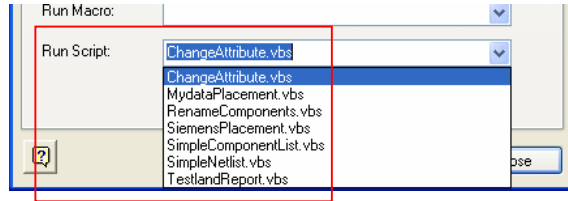
Assigning Scripts to shortcut keys or toolbars

Scripts can be assigned to both shortcut keys and toolbar buttons. To do this, use the **Customise** option from the **Tools** menu.

From the **User Tool** tab select the **Run Script**: drop down list and select your script name.



You will need to assign the user tool a name and then it can be used as a User Tool, a toolbar button or as a shortcut. You can only assign shortcut keys to a script defined under User Tools.



Items that can be read by scripts

Read access is available for many design and library items. Please refer to the online Help to check availability.

Items that can be written by scripts

There will be some limited write access back into designs and libraries. For example, reading and writing Attributes back into library items, where the script searches for all attributes which match 'cost' and '0805' and make these £0.20

Range of script activity

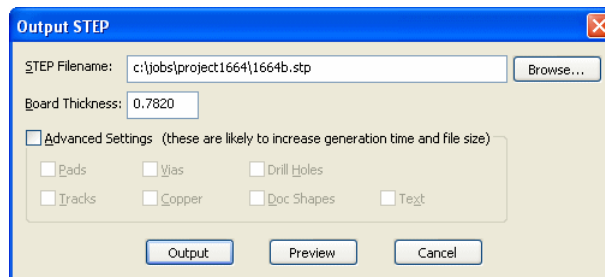
There will be limited access to Pulsonix options within the application itself. For example, you could run **Design Rules Check**, check that the number of errors equals 0 and then run **CAM Plot** to plot the design. You can run any command that is 'in context', the same as you get from the **Edit** and **Run Command** option.

Support for STEP Format

Where mechanical systems don't support or have limited support for the DXF or IDF formats, you can now export your designs in STEP format. STEP is an ASCII based data exchange format widely used by mechanical CAD systems. It can contain 3D component model information, these models are often available from a manufacturers web site.

Exporting STEP format

There is a new **STEP** option on **Output** menu which allows you to produce a STEP model of your board. It can also include component models if available.

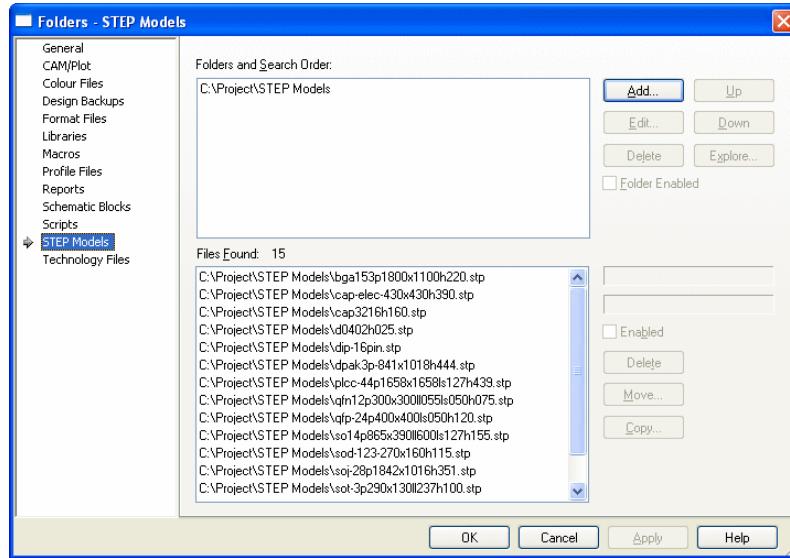


The **Board Thickness** is taken from the **Layers** dialog in the **Technology**. If the **Board Thickness** hasn't been setup, you can type your own value in to be used.

Use the **Advanced Settings** button to add additional design items to the STEP file. Be aware though, selection of these could make your STEP files very large.

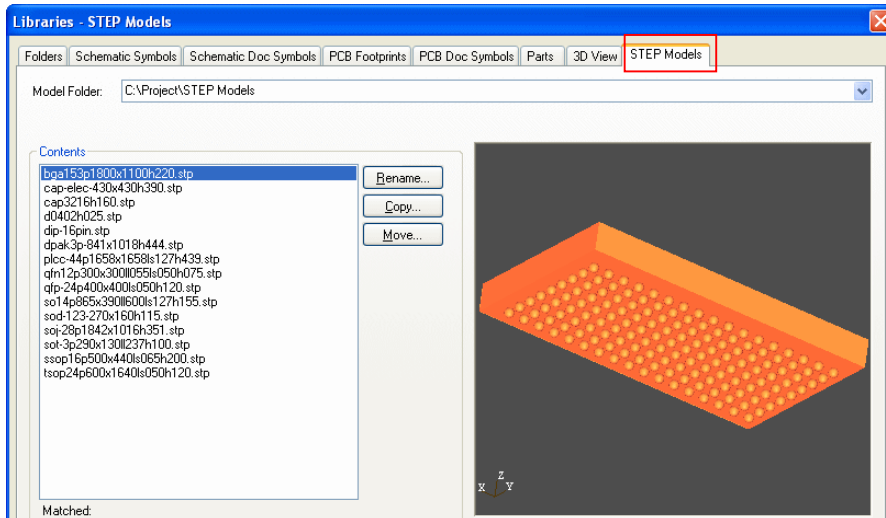
Folders

Models are located using the **Folders** options and **STEP Models**. Models are not provided but can be downloaded from the manufacturers web sites.



Viewing STEP Models in the Library Manager

STEP Models available to Pulsonix can be viewed in the **Library Manager** using the **STEP Models** tab.



In the footprint editor you can view the model if the footprint has the <STEP Filename> attribute. Once the attribute has been added and a model name provided including the file extension (.stp), you can use the **View STEP** option from the **View** menu. This will display the STEP model and the footprint to enable you to check their alignment and verify the model for that footprint.

Using STEP Models

When outputting the STEP format, it will use existing STEP models if they are available. If not, it will use standard Pulsonix Package models defined in the Pulsonix library.

Models are attached to a Part, Component or footprint definition using an attribute showing the model name, see below under *The STEP Attributes*.

Within the PCB Editor, there is an option to **Generate Heights** from the STEP models on the **Utilities** menu. This produces **<Component Height>** attributes on the Components which have STEP models associated. If there are no STEP models associated, then this attribute cannot be extracted automatically. The Component Height attribute is required for DRC checking because the STEP Model is only a reference to an external model name and not something available directly in the design.

Manufacturers Models and Model File Size

Some manufacturers have two flavours of models - basic and fancy. The fancy ones are normally a lot more detailed but are also a lot larger in size. Generally, STEP files and model files are very large.

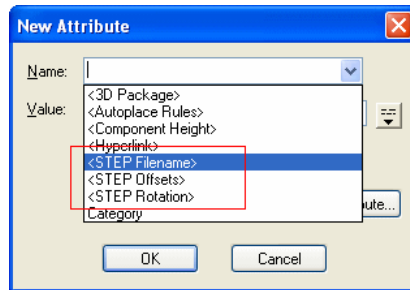
The STEP Attributes

There are three attributes which can be used on footprints or Parts for STEP models:

<STEP Filename>

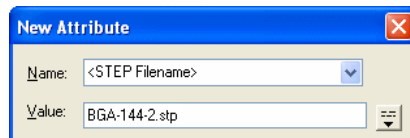
<STEP Offsets>

<STEP Rotation>



These attributes can be assigned at any level – in the footprint or Part library definitions, or within the design on a component. The attributes are all available as system attributes and are shown in brackets and are available on the Attribute **Name:** drop down list.

<STEP Filename> The **Value** will be the model name, you do not require the path as this is taken from the **STEP Folders** list. The Value must include the .stp filename extension.



The name used for the value will usually be defined from the manufacturers own footprint model name, but it can be any name you like.

Model to Footprint alignment

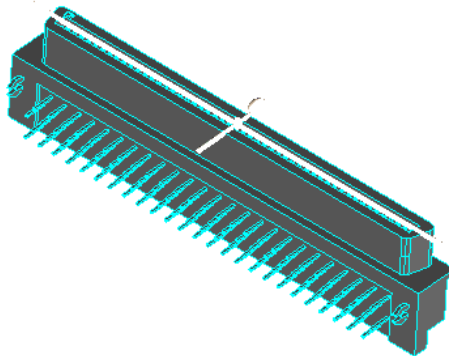
The orientation of the component STEP model and the footprint may be different. To make the best possible use of existing STEP models, you should create your Pulsonix footprint in the same orientation as the model, or vice versa.

You can add a **Model Placement** area to the footprint to aid alignment. Pulsonix will use the relative height/width of this area to compare with the height/width of the component to decide whether the model needs to be rotated or not. You can preview the model to check the footprint alignment with the model. Use the **View STEP** option from the **View** menu. If the model filename attribute has not been defined, a warning message will be displayed.

For an Area on the footprint, you should check the **Model Placement** check box on the **Area** tab of the **Area Properties**. This will be used by the output to best ‘describe’ the position that the model should adopt. The STEP model will then be oriented and/or shifted to fit within the Area you have defined.



For some STEP models, the model contains an origin with a **Z-axis** seating plane reference.

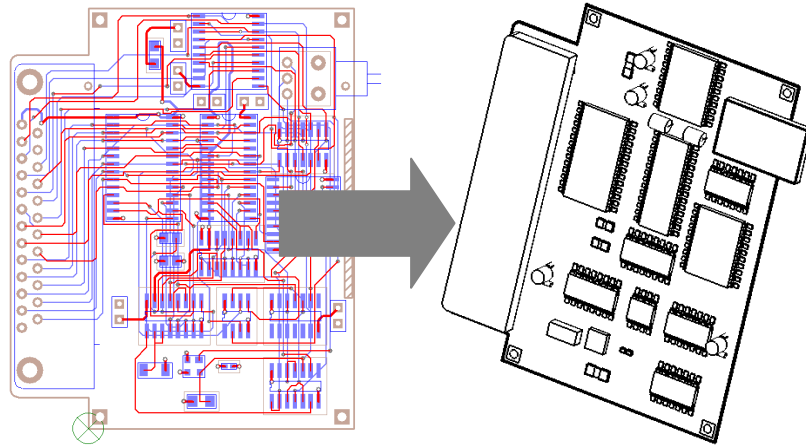


When positioning the STEP model in the Z-axis, Pulsonix will by default assume that the model is defined with $Z=0$ on the seating plane of the component. If this is not the case, you can apply an adjustment in the Z-direction by adding the **<STEP Offsets>** attribute to your component. Simply supply the distance in mm by which the model should be offset in the Z-direction to bring the seating plane in line with the board surface. The adjustment value is added to the current board Z-coordinate, so a positive adjustment will increase the Z position and hence move the component upwards away from the board surface. You can also make adjustments in the X and Y axes. To make multiple offsets in different axes the attribute value takes the form of **X=10;Y=5;Z=2**. If you do not want an offset in an axis just miss it out. The default units are millimetres. If you miss out the axis letter then Z is assumed.

Similarly if you want to apply a rotation in any axis use the **<STEP Rotation>** attribute. This uses a similar notation as the offset attribute. **X=90.0** will rotate the model in the X axis. If you miss out the axis letter then X is assumed.

STEP File Preview

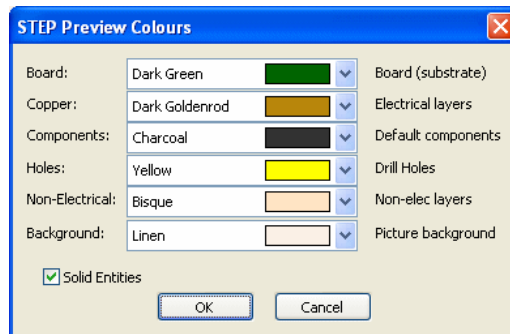
An option on the **Output STEP** dialog allows you to **Preview** the STEP file in it's own 3D Previewer.



Previewer Colours and Solid Models

The colours used in the **Preview** window can be changed to suit your own requirements. Use the **Colour Setup** dialog from the **Setup** menu.

If you wish to view your models as solid shapes, select the **Solid Entities** option from the **STEP Preview Colours** dialog.



View Model option in Footprint Editor

While editing your footprint, once a STEP Model filename attribute has been added (using the <STEP Filename> attribute and the **Model Placement Area** has been added, you can preview the model to check the footprint alignment with the model. Use the **View STEP** option from the **View** menu. If the model filename attribute has not been defined, a warning message will be displayed and you get the option to preview it using the Pulsonix 3D Package instead.

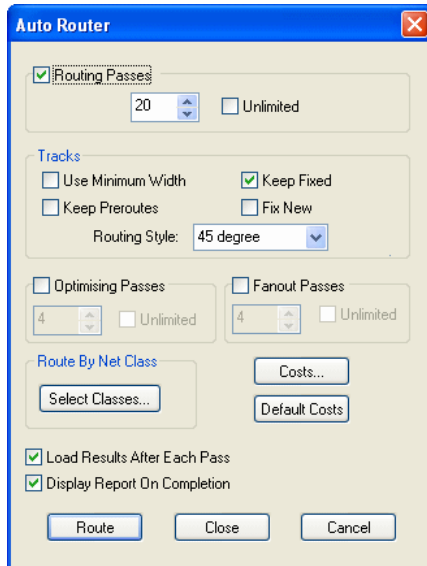
Import Board Outlines from STEP files

From the **File** menu you are able to import STEP files using the **Import Design Data** option. From the **File of Type:** drop down list, select **STEP Model.stp** You are able to import Board Outlines in STEP format, but currently not Board Cutouts, Areas or Component positions.



New Base Auto Router

All versions of Pulsonix now contain our new base level autorouter. The 'high-end' Advanced Autorouter is still available as a cost option where the base router isn't suitable. The new **Auto Router** is available on the **Tools** menu.



The name on the menu will be **Auto Router**, the existing Autorouter (**Auto Route>**) will be renamed on the menu to **Advanced Router**.

Using the Auto Router

Routing Passes

- **Routing** This invokes a number of routing passes which will route either all nets in the design or the specified nets, depending on which option started this dialog. You can specify the number of passes that you want. The larger your design the more passes you will want to give it. If it routes to 100% before the number of specified passes then no more passes will be attempted.
- **Optimising** This includes re-routing passes which attempt to reduce the overall track length and via count once the Routing passes are complete. You can specify the number of optimisation passes that you want. Optimisation can be a slow process, however it can significantly reduce the overall track length and via count. For larger boards, this may be an over night process. If you already have a fully routed board, you can disable the Routing phase and just use the Optimiser.
- **Fanout** This invokes a number of passes that route short escape tracks to vias from SMD pads. This is done before the Routing passes. Fanout passes are not necessary in order for the router to function, and is more useful as a initial phase before manual routing. You can specify the number of passes that you want.

Any of the above pass options can be disabled by unchecking the appropriate box. You can also specify Unlimited passes, in which case, the router will continue until the board is complete, or until it is cancelled.

Tracks

- **Use Minimum Width** means that if a net has a net class defined the router will use the minimum track style as opposed to the nominal track style.
- **Keep Preroutes** means that existing routing is protected against being ripped up by the Autorouter.
- **Keep Fixed** means that only routes that are already fixed will be protected against being ripped up by the Autorouter.
- **Fix New** means that tracks routed by the Autorouter will become fixed and will remain unaffected by the Autorouter if a subsequent routing pass is made, changing the route parameters.
- **Routing Style** allows you to specify the angle of tracks, either *90 degree* (orthogonal), or *45 degree* (angled tracks).

Check **Load Results After Each Pass** if you want the display refreshed with the latest results after each pass.

Check **Display Report On Completion** if you want the Auto Router to produce and display a status report when it finishes.

Costs will display the Costs dialog which you can use to change the cost factors used by the Auto Router. You can use this dialog to change the costs involved with ripping up existing tracks and inserting vias. You can also restart the Router at a specified Pass.

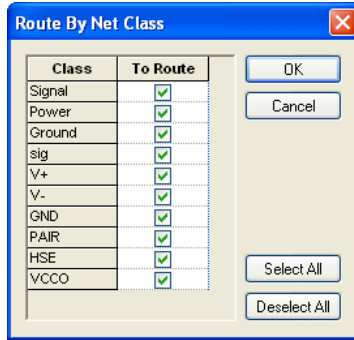
Default Costs changes the costs back to their default values.

The **Route** option will start the Auto Router using the values specified in this dialog. **Close** will exit this dialog without starting the Auto Router but will store any changed values.

Routing By Net Class

The **Routing By Net Class** dialog is used to control the nets which are selected for routing.

Nets which are power or ground, or which have specific properties e.g. high speed or memory etc. should be assigned their own net class if they are to be excluded from the routing.



By using the **To Route** switch, net classes can be selected or deselected for use.

Route Selected

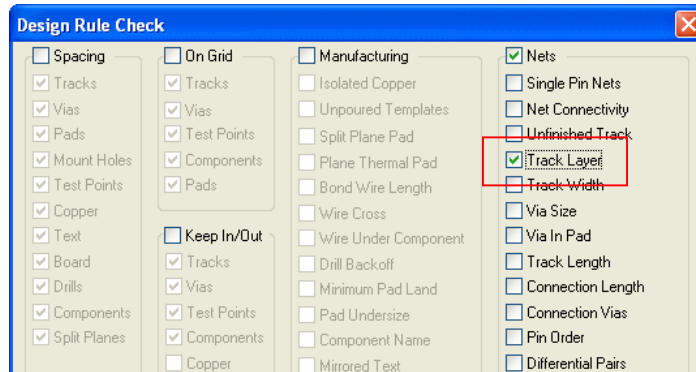
For a selected connection or component, you can select the **Route Selected** command from the context menu or from the **Tools** menu and **Auto Router> Route Selected** (or **Browsed Nets**). This will route the nets selected using the parameters already defined in the **Auto Router** dialog.

Previous Original Autorouter

Anyone using the original autorouter will no longer be able to use it under Windows 7. If you have this situation, please contact our sales office or your local Pulsonix supplier for advice on how to resolve this. This does not affect the Advanced Router.

Design Rule Check – Track Layer

Within **DRC**, the **Track Layer** option will now also add errors for tracks on layers that have their routing bias set to **No Tracks**.

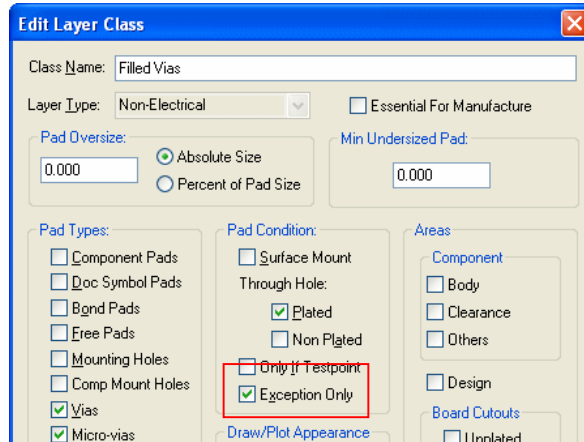


Designs that previously had **No Tracks** set will be changed to **No Autoroute**.

Changes to Technology dialog

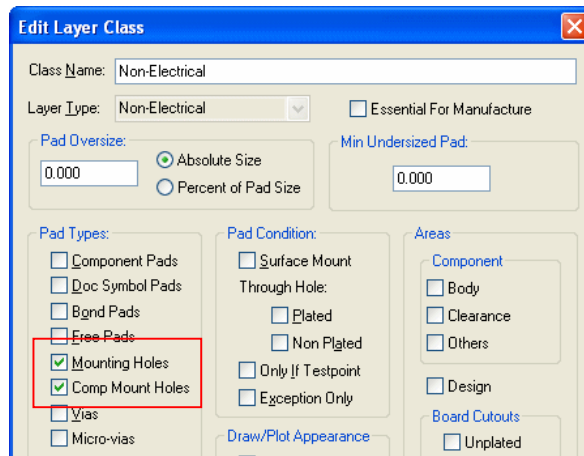
Layer Class – Exception Only

You can now create a **Layer Class** which will only show pads which have an explicit pad style exception assigned to that layer. For example, if a layer is required for manufacturing where specific vias are to be ‘filled’ or ‘tented’, these can be defined as being on a layer exception. These exceptions can then be extracted during plotting by specifying their **Pad Condition** as **Exception Only** in the **Layer Class**.



Layer Class – Draw Component Mounting Holes

You can optionally display **Component** and **Design level Mounting holes** separately from within the **Layer Class** dialog in the Technology file. There are two check boxes to differentiate the two Mounting hole types.

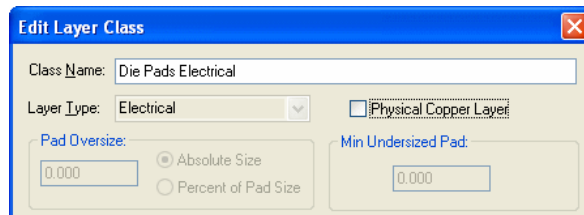


Layer Class – Physical Copper Layers

Layer Classes have been changed to allow three build status to be available.

Essential for Manufacturing selected – available for **Non-electrical** layers. If a Footprint contains items on an Essential layer, it cannot be added to a design which does not contain a matching layer.

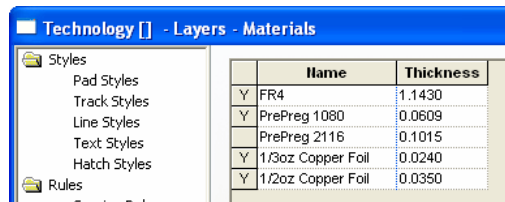
Physical Copper Layer check box checked. This check box is only shown if the type is **Electrical**. Most electrical layers must have this box checked as they represent a physical copper layer in the construction of the final board. Uncheck it if you need an electrical layer that represents something else, for example an inner layer to display **embedded component die pads** on but not allowing copper tracks etc.



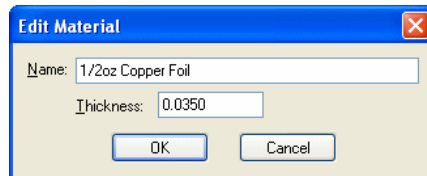
This feature and the physical Layer count are discussed further in the **Advanced Technology** section.

Layers – Materials

Within the **Technology** file and **Materials** page, the material **Thickness** can now be added. This will be used as the default value when the Layer uses that Material. You can also leave the material thickness as zero if you are unsure of its thickness or do not wish to show this.



If you select the **Edit** button, you can add a Material and its **Thickness**.

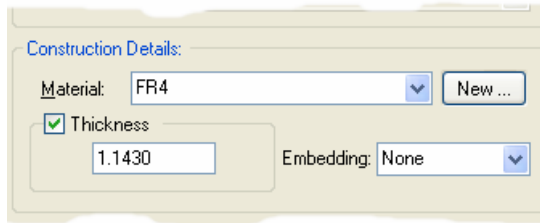


Materials and Thickness in Layers dialog

As stated above, the Materials can be selected during creation of a new layer or can be added to an existing layer.

Y	Pin Names	Non-Electrical	Top	None			0.0000
	PrePreg1	Construction		None		PrePreg 1080	0.0609
	Prepreg2	Construction		None		PrePreg 1080	0.0609
Y	Ground	Electrical	Inner	Power Plane	GND	1/2oz Copper Foil	0.0350
	FR4	Construction		None		FR4	1.1430
	+1.8V	Electrical	Inner	Power Plane	+1.8V	1/2oz Copper Foil	0.0350
	PrePreg3	Construction		None		PrePreg 1080	0.0609
	Prepreg4	Construction		None		PrePreg 1080	0.0609
	Bottom	Electrical	Bottom	Y		1/3oz Copper Foil	0.0240
	Silkscreen Bottom	Silkscreen	Bottom	None			0.0000

If the Material has a Thickness value defined, this will be used by default. It can also be overridden by editing the selected layer and by checking the Thickness box. The new value can then be typed in. You cannot directly edit the Thickness value in the Layers dialog unless the Thickness check box has been selected.



Layers – Thickness

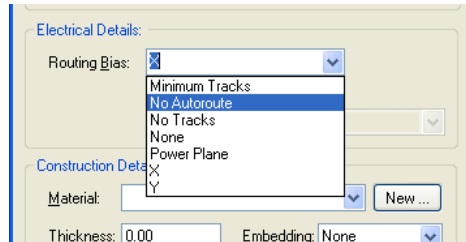
Within the **Technology** file and **Layers** page, the total **Board Thickness** is now shown. This will be combined thickness of any layer which contains a thickness value, this includes all Electrical and Construction layers but also includes Silkscreen layers for example. It will also include unused layers where that layer has a thickness defined, Embedding is taken into account.

Name	Associated Layer	Class	Side	Bias	Net	Material	Thickness
Y	Silkscreen Top	Silkscreen	Top	None			0.0000
Y	Top	Electrical	Top	X		1/3oz Copper Foil	0.0240
	Paste Mask Top	Paste Mask	Top	None			0.0000
Y	Solder Mask Top	Solder Mask	Top	None			0.0000
	Wires Top	Wire Link	Top	None			0.0000
Y	Pin Names	Non-Electrical	Top	None			0.0000
	PrePreg1	Construction		None		PrePreg 1080	0.0609
	Prepreg2	Construction		None		PrePreg 1080	0.0609
	Ground	Electrical	Inner	Power Plane	GND	1/2oz Copper Foil	0.0350
	FR4	Construction		None		FR4	1.1430
	+1.8V	Electrical	Inner	Power Plane	+1.8V	1/2oz Copper Foil	0.0350
	PrePreg3	Construction		None		PrePreg 1080	0.0609
	Prepreg4	Construction		None		PrePreg 1080	0.0609
	Bottom	Electrical	Bottom	Y		1/3oz Copper Foil	0.0240
	Silkscreen Bottom	Silkscreen	Bottom	None			0.0000
Y	Solder Mask Bottom	Solder Mask	Bottom	None			0.0000
	Paste Mask Bottom	Paste Mask	Bottom	None			0.0000
	Wires Bottom	Wire Link	Bottom	None			0.0000
	Pin Names (Bottom)	Non-Electrical	Bottom	None			0.0000
	Documentation	Documentation		None			0.0000
	Construction Lines	Documentation		None			0.0000

Number of Electrical Layers: 4 Board Thickness: 1.5046

Layers – New Track Layer Bias Type

When editing a **Layer** within **Technology**, a new routing **Bias** value of **No Autoroute** has been added, this will still allow interactive routing. A routing bias can now be **No Tracks** or **No Autoroute**.



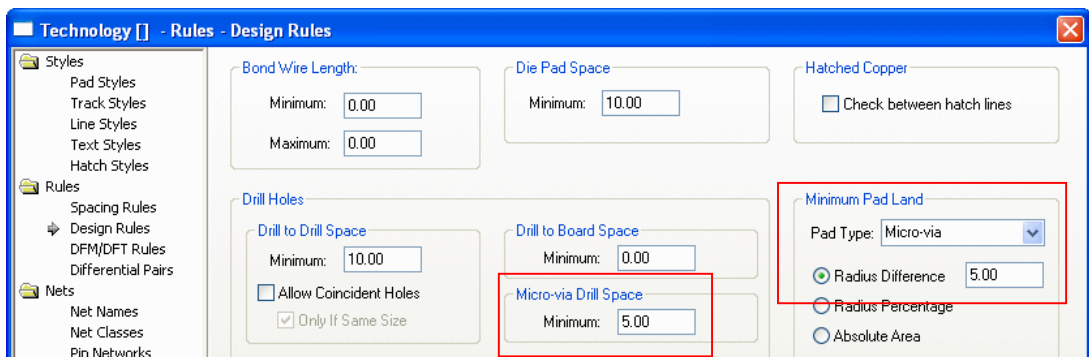
No Autoroute means no Automatic Routing on this layer. You can use this to disable a layer for Automatic Routing.

No Tracks also means no Automatic Routing on this layer, but in addition any tracks added interactively on this layer will be flagged as in error by the **Design Rule Check - Track Layer** option.

*Note Existing designs that used a bias of **No Tracks** will have that value changed to **No Autoroute**.*

Design Rules – Minimum Pad Land

There are now different values for the **Minimum Pad Land** size for **Through Hole Pads**, **Vias** and **Micro-vias**, (**Vias** and **Micro-vias** have been added).

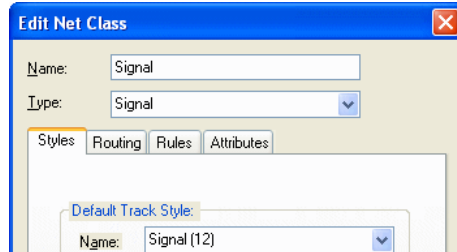


The **Micro-via Drill Space** defines minimum distance between the outer edge of micro-via drill holes, regardless of the pad shape. The minimum between micro-via drill holes and other drill holes is the maximum of this and the drill to drill spacing above.

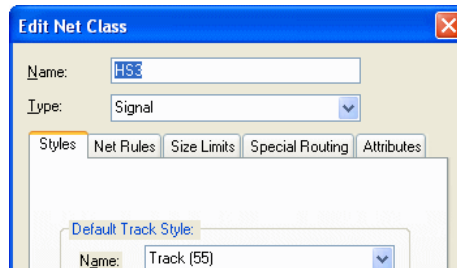
Net Classes – Edit dialog changes

The Edit Net Class dialog has been changed for version 7.0 to allow for the new rules.

Version 6.1 looked like this:



In version 7.0, it looks like this:

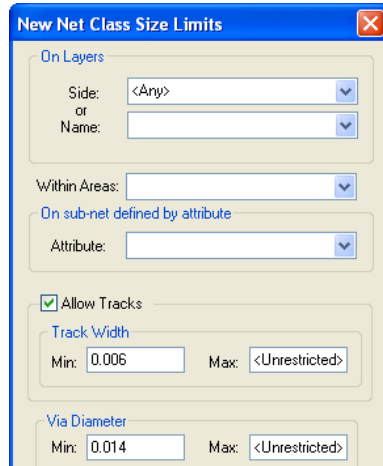


Routing now becomes **Special Routing** (this contains Serpentine Rules etc.). **Rules** becomes **Net Rules** but is also modified to include the new changes. **Size Limits** has been added for Version 7.0.

Each of these dialogs will be discussed below.

Net Classes – Track Size Limits

Size Limit rules for track width and via pad size can now be defined to match by layer name or side and within a matched area, and attribute defined sub net.



The **Net Class Size Limits** define restrictions on the widths of tracks and diameter of vias. They would normally be defined for critical nets in the Schematic then passed through to the PCB. This gives the Schematic engineer some control over how the net is routed. These rules are applied in terms of actual sizes, regardless of the Track and Pad Styles used.

Track length rules are defined on the **Net Rules** tab.

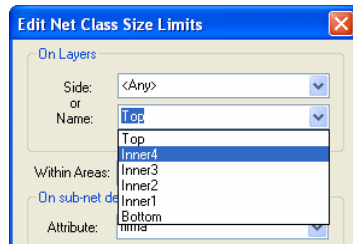
Rule values default to **<Unrestricted>**, in other words, no rule is applied. You can define a rule simply by typing in the appropriate value. Clearing a value will set it back to **<Unrestricted>**.

Rules are checked by selecting the appropriate options in the **Nets** section of the **Design Rule Checker**. None of these rules are currently applied during the design process, and are only checked when explicitly selected in the **Design Rule Checker**.

How to define a size limit rule

To create a size limit rule, press the **New** button. This takes you into a dialog where you define which layers, areas and sub-net this rule applies to (for the current Net Class).

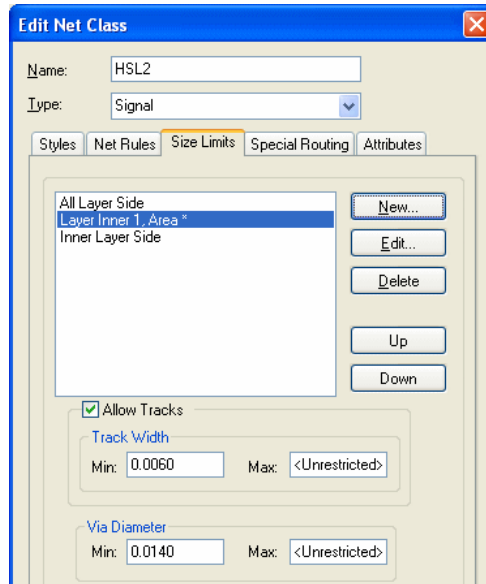
The electrical layer can either be a **side** (**<Any>**, Top, Inner, Bottom), or a **layer** name. The layer name can be a wildcard matching string, so 'ground *' would match layers 'ground 1' and 'ground 2').



Similarly, a wildcard string can be defined to match **areas**. The matching areas must be marked as a **Style Override Area**. A sub-net is defined by selecting an **attribute** which all pins in that sub-net should have. The value of the attribute is unimportant for this purpose.

Size Limits

Once a new **Size Limit** rule is defined, it will appear in the list of Size Limit rules.



These rules are applied in the order defined in this list, until the first match is found. You should therefore have the more specific rules (in terms of layer, area and sub-net) defined at the top of the list. You can move a rule up and down the list by selecting the rule in the list and using the **Up** and **Down** buttons.

A rule can be deleted by selecting it in the list and pressing the **Delete** button.

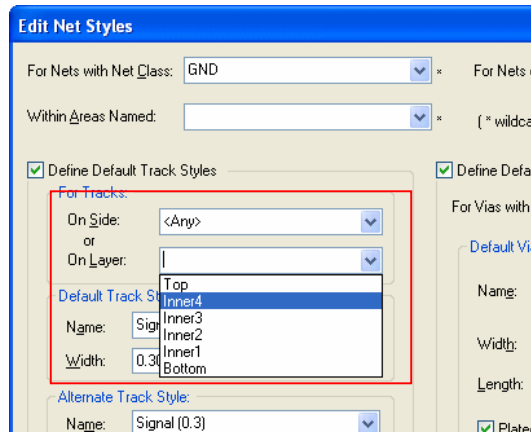
Minimum & Maximum Track Width defines the limits of the width of all track used on a net. Any number of different track styles can be used, but they must all fall between these limits.

Unchecking the **Allow Tracks** box means that track will not be allowed at all. So you can effectively disallow tracks for this net class on a specific layer, or within certain areas.

Minimum & Maximum Via Diameter defines the limits of pad styles used on vias on a net. The limits apply to the minimum and maximum dimensions of any shape, and includes all shapes explicitly applied to any electrical layer.

Net Styles – Track Styles

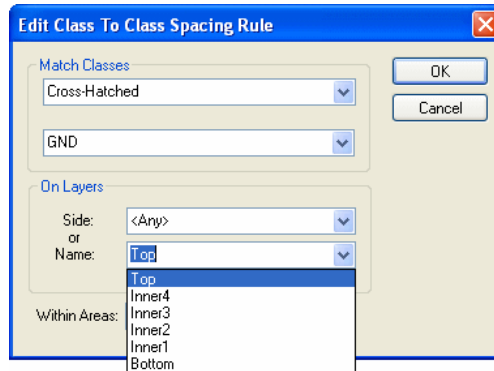
Track Styles can now be matched by **Layer Side** or **Name**.



You can define the default track styles for specific layers or side. If you choose to specify a layer name, this can be a wildcard string which may match several actual layer names. Select a **Track Side** if the track styles in this entry are only to be used for tracks that are on a specific side of the board, or on all inner layers. This allows you to specify different default track styles to be used on the top or bottom of the board for particular net classes. Use **Any** if the side does not matter.

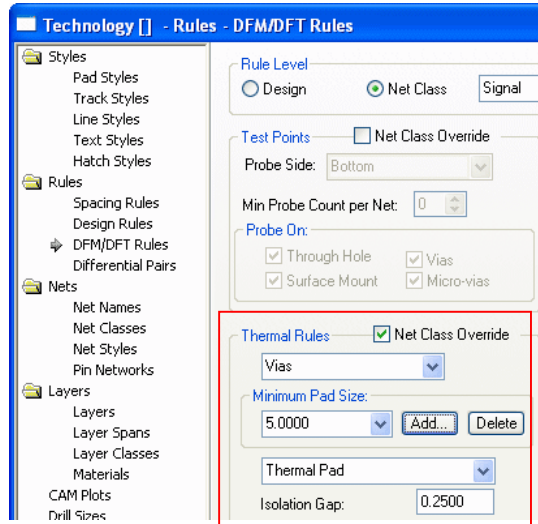
Spacing Rules – Match Net Class Pair

Can now match layer names instead of layer side. Dialog space enlarged to accommodate longer match rule names. Matching against a class of '*' also matches items not on a net, and if used, allows you to specify spacings for text & board against a specific net class.



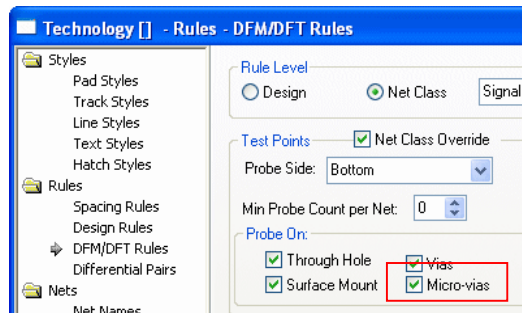
DFM Rules – Thermal Rules

You can define **Thermal Rules** which apply to pads greater than a **minimum size**. This now lets you define Thermal Rules for specific pads by using an attribute that is not tied to a **Net Class**. This says, for a given Minimum Pad Size, the Thermal Rules *below* it will be applied. This applies when the **Net Class Override** check box is selected.



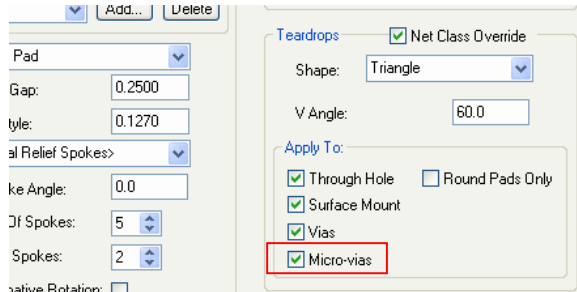
DFM Rules – Testpoint Rules

Micro-vias have been added to the list of available pad types for applying **Test Points** to (or more likely, not apply Testpoints to).



DFM Rules – Teardrop Rules

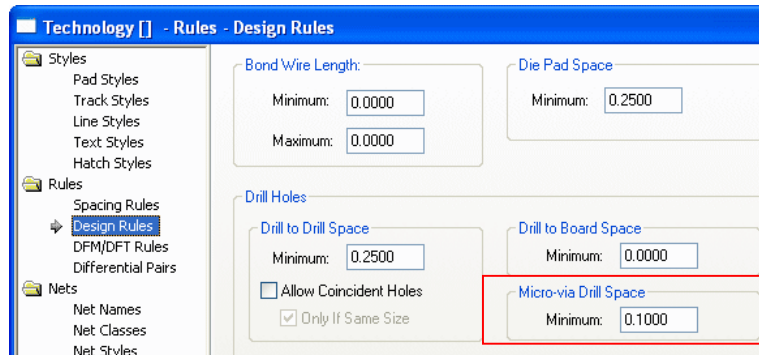
Micro-vias have been added to the list of available pad types for applying **Teardrops** to.



You can specify what types of pads can receive teardrops. You can choose **Through Hole** pads (including mounting holes), **Surface Mount**, **Vias** or **Micro-vias**. You can also decide if only **Round** pads will receive teardrops, this applies in addition to the pad type restrictions.

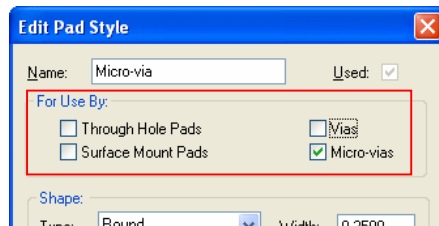
Design Rules – Drill Spacing

An explicit **Micro-via Drill** spacing has been added. The greater of the **Micro-via Drill** spacing and the **Drill** spacing is taken as the **Drill to Micro-via rule**.



Pad Styles – Restrict Usage of Pad Styles

You can restrict the usage of a **Pad Style** by unchecking the **For Use By:** boxes on the **Edit Pad Style** dialog from within the **Technology** dialog. So, for example, only pad styles that have the **Micro-vias** check box selected will be shown when you change the style of a Micro-via in the design.



Advanced Technology Features

These features are available in the Advanced Technology cost option. Please note that the previous options for Embedded Component and Chip On Board design have been combined and are now included in this new (Advanced Technology) option.

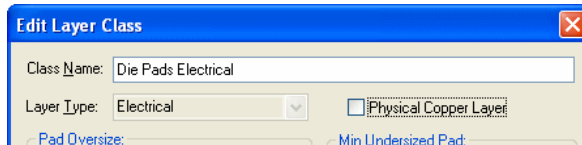
Pulsonix 7.0 now supports Micro-vias, these are laser drilled (usually small) vias between same facing layers with the ability to stack them using composite layer spans. Version 7.0 also supports Flexi-rigid boards, board cavities and physical embedded components by placing outlines and cutouts on layer spans.

Layers Classes

Layer Classes have been changed to allow three build status to be available for Layer Class definitions.

Essential for Manufacturing selected – available for **Non-electrical** layers. If a Footprint contains items on an Essential layer, it cannot be added to a design which does not contain a matching layer.

Physical Copper Layer check box checked. This check box is only shown if the type is **Electrical**. Most electrical layers must have this box checked as they represent a physical copper layer in the construction of the final board. Uncheck it if you need an electrical layer that represents something else, for example an inner layer to display **embedded component die pads** on but not allowing copper tracks etc.



Layers

The **Number of Physical Copper Layers** reported at the bottom of the **Layers** dialog now reflects the real physical layers in the design. This will depend on the status of the **Physical Copper Layer** check box in the **Layer Class** dialog, Physical Copper Layers must be selected for it to be counted.

Y		Pin Names	Non-Electrical	Top	None	
Y	Top		Electrical	Top	None	
	Prepreg		Construction		None	
Y	Layer 2		Electrical	Inner	None	
		Assembly ECP L2	Assembly ECP	Inner	None	
		Cavity ECP L2	Cavity ECP	Inner	None	
		Glue spots ECP L2	Glue spots ECP	Inner	None	
Y	Die Top Core		Die Pads Electrical	Inner	No Tracks	
	Core		Construction		None	
Y	Die Bot Core		Die Pads Electrical	Inner	No Tracks	
		Assembly ECP L3	Assembly ECP	Inner	None	
		Cavity ECP L3	Cavity ECP	Inner	None	
		Glue spots ECP L3	Glue spots ECP	Inner	None	
Y	Layer 3		Electrical	Inner	None	
	Prepreg 2		Construction		None	
Y	Bottom		Electrical	Bottom	None	
		Soldermask bottom	Resist	Bottom	None	
		Solderpaste Bottom	Solderpaste	Bottom	None	
Y		Silkscreen Bottom	Silkscreen	Bottom	None	
	Documentation		Documentation		None	

Only Show Used Entries Number of Physical Copper Layers: 4 Board Thickness: 0.434

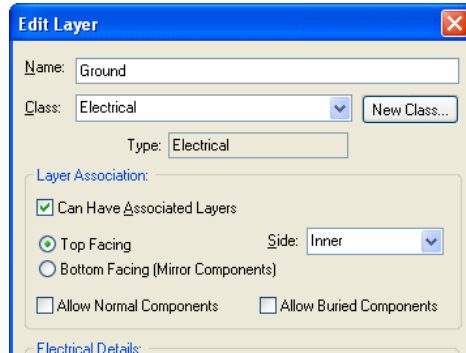
If a layer is marked as **Electrical** and not **Physical Copper Layer** then the layer Bias will automatically be selected and fixed as **No Tracks**. Also, the **Usually Plotted** check box will be unselected and greyed out as unavailable for edit.

Layers Now Have a 'Facing' Flag

The **Mirror Components** flag has been replaced with a **Facing** switch for selection when **Can Have Associated Layers** is selected in the **Edit Layers** dialog.

Not only is this used to define the **Mirror** components status, for a Bottom Electrical layer for example, but is also used to define the face for the layer so that it can be used for Micro-via spans.

For Micro-vias, the face must also be defined if the layer is to be used as a **Stop layer** in the layer span definition.



Layer Spans

Layer Spans are (still) defined in the **Layer Spans** dialog from within the **Technology**. It is from here you can define layer spans to use for Vias, Micro-vias, Board Outlines, Areas and Components.

A layer span becomes a **Micro-via** type by having the same **Facing** side on its Top and Bottom **Layers**. To use a micro-via technology you should define the span between **same facing layers**.

	Name	From Layer	To Layer	Type
Y	<Through Hole>	<Top Side>	<Bottom Side>	Through Hole
Y	T - L2	<Top Side>	<Layer 2>	Micro-via - top facing
Y	L2 - Core Top	<Layer 2>	<Die Top Core>	Micro-via - top facing
Y	Core Top - Core B	<Die Top Core>	<Die Bot Core>	Buried
Y	Core Bot - L3	<Die Bot Core>	<Layer 3>	Micro-via - bottom facing
Y	L3 - Bot	<Layer 3>	<Bottom Side>	Micro-via - bottom facing

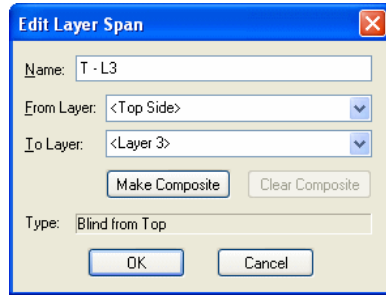
Composite Layer Spans

Composite Spans are used for connecting multiple layer spans together.

Using a combination of **Micro-via** and **normal spans**, you can define a **Composite Layer Span**.

A Composite Layer Span is specifically for creating a via 'stack'. That is, a stack of vias which together complete the span and which move together as one unit, this will usually involve a set of micro-vias and sometimes includes a buried via.

To do this, you must first define each span of the composite. Once created, define the Composite Span by selecting the **From Layer:** and **To Layer:** layers using the drop down list. If these are legal layers which can create a composite span. The **Make Composite** button will be available.



The **Add Composite** or **Edit Composite** button will offer you a dialog showing each possible combination of spans which will cover this composite overall span (if there is more than one, press **Next** to cycle through each one). Press **OK** to define the composite. You can change a composite back to a normal span by pressing the **Clear Composite** button from within the Edit dialog.

Once you have a composite span defined, you must then define the **Pad Style** to use on each via in the stack. Only nets which have composite styles defined using **Net Styles** can use the composite span to create stacked vias.

Net Styles

To use a **Micro-via layer span** or **Composite layer span**, you will require an entry in the **Net Styles** for a net (**Net Styles** is enabled from the **General** page of the **Design Settings** option). When composite spans are used it causes a via stack to be added rather than a single via, each via in the stack using the pad style defined in the net styles entry.

Using Composite Layer Spans

Composite via stacks can be added using **Add Via**, or added automatically by changing layers of tracks. If using **Add Via**, then the same restrictions to the function will apply as when using **Insert Multiple** (no finish markers, test point, design clearance etc).

Selecting a via in a stack selects all vias in the stack, and the status bar tells you so. This is so the stack can be moved as one, but the individual vias in the stack can be selected using **Select Next**. Once a via has been moved away from the rest in the stack, the stack is no longer complete and will not be selected as one item.

Composite span stacks will be shown in the **Layer Stack Preview**, and in the **3D Viewer**.

Micro-vias / Laser Drilled Vias

A Via using a **Layer Span** between **two same facing layers**, is defined as a **Micro-via**.

Entry Pads and Stop Pads

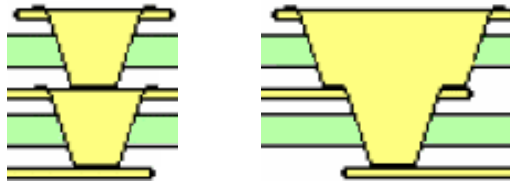
There are two special **Pad Style** exceptions (defined in the **Technology**) for **Micro-via Entry Pad** and **Micro-via Stop Pad**. These define the pad sizes used on the entry and stop layers when the style is used for a Micro-via (unless there is an explicit exception for that layer, which will be used instead). These provide a simple method of defining specific Entry and Stop pad sizes, **they are not mandatory for Micro-vias**.

This pad definition is used for creating ‘tapered’ Micro-vias, where the Entry and Stop pads are different diameters to match the ‘taper’ effect caused by the laser drilling.



A Micro-via showing a tapered hole

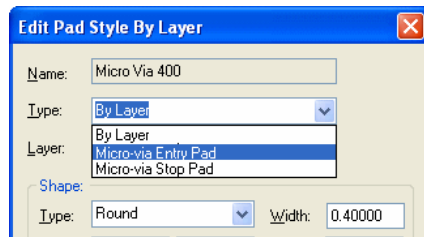
It is also possible to create a set of ‘stacked’ Micro-vias using a **Composite Layer Span**. Without Stop pads, the laser will simply ‘cut’ to a depth or until it reaches a Stop pad. Multiple ‘tapered’ via spans are also possible using Composite Layer Spans using different **Via Pad Styles**.



Two examples of Stacked Micro-vias (Composite)

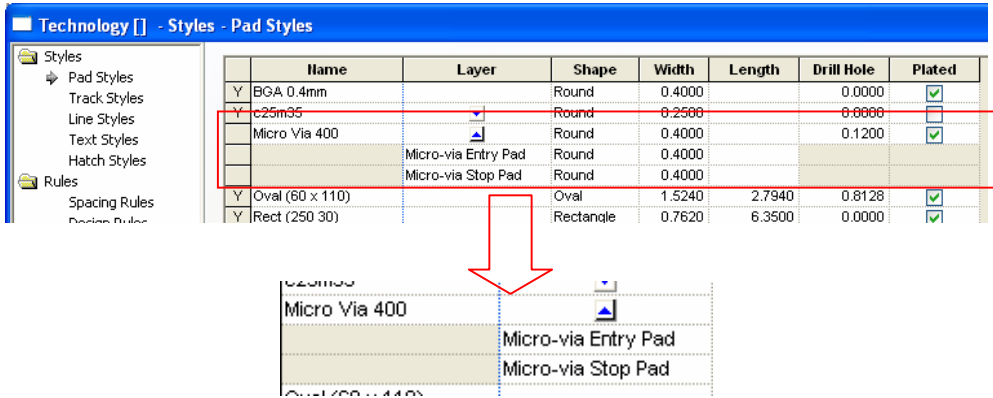
Creating Entry and Stop Pads

Edit the **Pad Style** and use the **By Layer** option, then for **Type**: select **Micro-via Entry Pad** from the drop down list. The **Layer**: box will now be greyed out as un-selectable. Type a value for **Width**: if it needs to be different to the default stack value and then press **OK** to exit and save the exception.



Normally, you would also do the same for the **Stop Pad** so the laser knows where to start and where to stop drilling, although this isn't mandatory for some processes. The Stop pad is displayed as a solid pad even though the default pad stack is defined with a drill hole, and the Entry pad is displayed with a drill hole shown (depending on Layer Class settings).

The **Pad Style** entry for that pad would then look like this (shown for **Micro Via 400**):

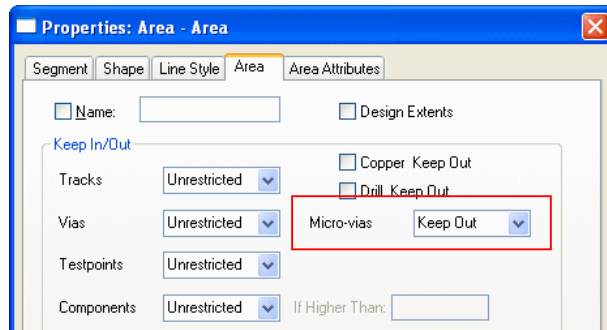


Drilling Micro-vias

Micro-vias are ‘drilled’ (using a laser or plasma ‘drilling’ machine) using the standard **Excellon NC Drill** output. Special customised reports can be generated using the **Report Maker** option on the **Outputs** menu. All pad styles on any layers and the drill position can be output if required, *see new Report Maker commands for details*.

Micro-via Keep In/Out Areas

Areas can now be set to **Keep In/Out** for **Micro-vias**, this is a completely separate option from an area being set to **Via Keep In/Out**.



Board Outline and Areas on Layer Spans

You can now add a **Board Outline**, **Board Cutout** or **Area** to a layer span. You can also add a combination of board outlines or areas on different layer spans to represent how they will be used for those layers, a flexi-board outline on an inner layer span for example.

To add board outlines, cutouts or areas to a span, you must create a suitable layer span in the Layer Span dialog. This has previously been exclusively used for via spans, but is now used for shapes and components as well. *Component Spans are discussed below*.

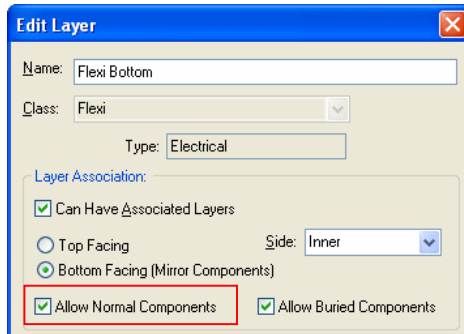
Board Spans

A **Board** on a layer span redefines the outline for that span. For example, it is possible to define a flexi-board on an inner layer span, which overhangs the outer board outline. It is possible to add normal components to the overhangs. For normal boards, the board outline would remain on <Through-board> Layers.

In the **Layers** dialog, create the **Layers** and **Layer Classes**, in our picture above for two ‘inner’ layers, Flexi Top and Flexi Bottom.

Y	Top		Electrical	Top
		Solder Mask Top	Solder Mask	Top
		Paste Mask Top	Paste Mask	Top
Y		Pin Names	Non-Electrical	Top
Y		Inner 2 - shape	Non-Plot	Inner
Y	Inner 2		Electrical	Inner
Y		Flexi Silk Top	Silkscreen	Inner
Y	Flexi Top		Flexi	Inner
Y	Flexi Bottom		Flexi	Inner
Y		Flexi Silk Bottom	Silkscreen	Inner
	Inner 5		Electrical	Inner
		Inner 5 - shape	Non-Plot	Inner
Y	Bottom		Electrical	Bottom

You also need to edit the layer to **Allow Normal Components**.



The **Layer Spans** are defined once the **Layers** and **Layer Classes** have been created, in our example below the span is across <Flexi Top> and <Flexi Bottom>.

	Name	From Layer	To Layer
Y	<Through Hole>	<Top Side>	<Bottom Side>
Y	Flexi layers	<Flexi Top>	<Flexi Bottom>
Y	Top to Flexi Top	<Top Side>	<Flexi Top>
	Top to Inner2	<Top Side>	<Inner 2>

When selecting a Board outline, Area or Board Outline Cutout, use **Change Layer** to change to a Layer Span selected from the drop down list, a list created from the legal Layers Spans available.

Board Area Cutouts

You can also define holes and cavities using a **Board Cutout Area**, and components can be placed within these cavities (if you have the Advanced Technology option). **Board Area Cutouts** are defined as holes or cavities through the use of **Layer Spans** from the Technology dialog, the same as **Board** outlines and **Areas**. Where a ‘cavity’ is to be created in a board, this is simply a **Board Area Cutout** spanning the required layer span.

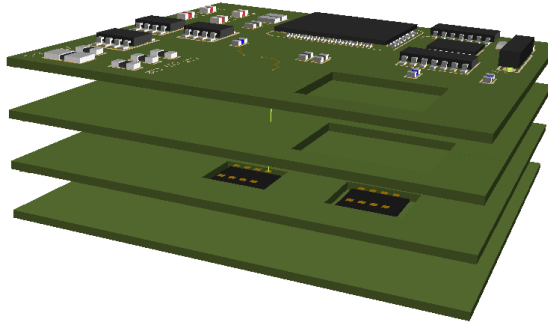
Component Spans

Placing Components on Flexi Boards and within Cavities

Surface mounted components can be placed onto flex board and in cavities by changing their layer to the inner layer they will sit on. The layer must have **Allow Normal Components** enabled to be able to do this. Through-holes components can also be placed on the surface of an inner layer span of a flexi part of the board, or in a cavity within the board. You must create a **Layer Span** to cover the span required to be able to move the component onto.

Define the span that the component's pads will need to reach the other side of the board from the layer the component is being placed on. The component's Through-hole pads, mounting holes, component vias and areas will be changed to use the chosen layer span.

The Layer Span selected must have been first defined in the **Layer Span** dialog of the **Technology**. Choose the span which the component will fit across. For example, a component might span Layer 3 to Layer 4 but sits inside the cavity that spans these layers; a cutout in the Prepreg or other construction layer.



When creating components, cavities or holes in the board outline can be defined up-front as part of the footprint. These are created as Areas and with the Board Cutout check box selected. Obviously, the Area needs to be defined to span the appropriate layers within the design.

Moving Components to a layer span

To move a component onto the new Layer Span required, select the component and from the context menu, select **Change Layer**. Select your span choice from the drop down list. Only layer spans that start or end on layers that can have components will be available. A **Mirror Component** check box will be available to say which side of the span the component will be placed on.

The **Design Rules Check** (DRC) option will check that pads, mounting holes etc. will fit the span defined.

Also, Mounting Holes in a PCB design can now be assigned to a Layer Span. This allows you to place construction holes into the flexi board without having to use a via.

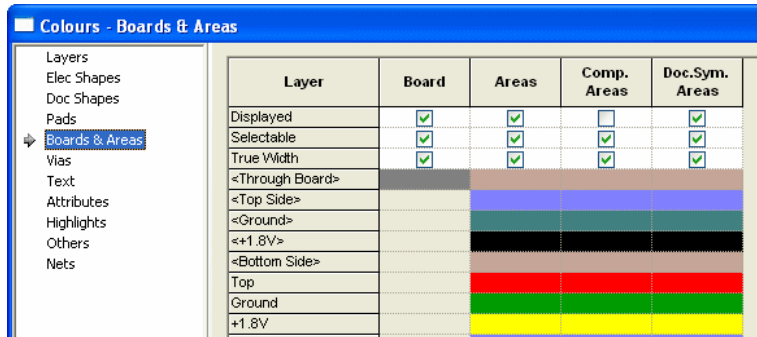


Components which are to be used on any other layer span other than conventional <Through Hole> do not need any special set up requirements but you should be aware of any supporting layers which may or may not be subsequently required, for example Silkscreen layers.

New Items in Colours dialog

Due to the **Board** and **Areas** on **Layer Span** changes, the **Colours** dialog has been changed as follows:

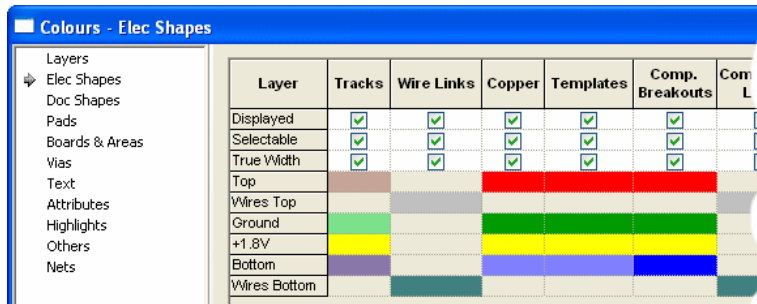
There is now a new **Boards & Areas** tab.



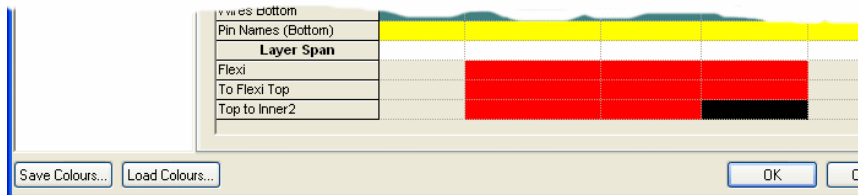
Areas has been moved from the **Elec Shapes** tab and **Board** has been moved from the **Others** tab. Both now appear on the new **Boards & Areas** tab that contains Layer Spans.

The **Tracks** tab has been removed completely.

Tracks and **Breakouts** have been moved to the **Elec Shapes** tab.



The **Pads** page has also been modified to display **Layer Spans**.

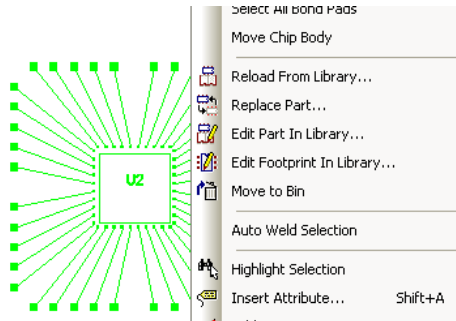


Changes to Bond Pad Options

There are two new options available on the context menu when part of a single component that contains bond pads is selected in a PCB design.

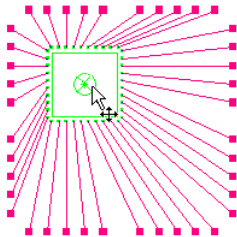
Select All Bond Pads

Use the **Select All Bond Pads** option to select all of the bond pads on the component. Once all the bond pads have been selected, this could be used for changing their layer or style for example. This option is part of the Advanced Technology cost option.



Move Chip Body

Use the **Move Chip Body** option to move the chip body without moving any of the bond pads. This option is part of the Advanced Technology cost option.



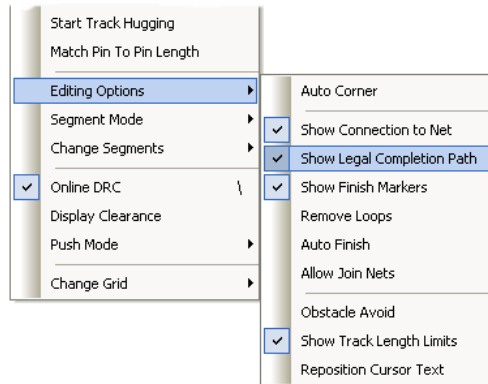
Change Layer of Bond Pads

Using **Change Layer**, you can now set the layer of an individual bond pad in a PCB design to any reference layer that is facing the same way as the layer the component is on. This is to support the placing of the body of bare die components into cavities and to have their bond pads still on the top side.

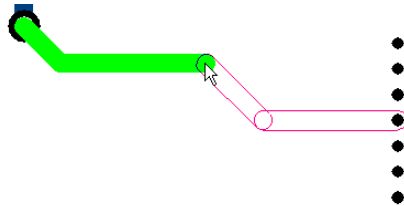
You can also change the layer of multiple bond pads on one component using the **Properties** dialog. Both operations are part of the Advanced Technology cost option.

Insert Track - Show Legal Completion Path

There is a new **Insert Track** option (**Show Legal Completion Path**) to display the natural completion path to the target item if it is legal to complete in two or less segments to the target without introducing any spacing errors. It will display the potential segments as hollow tracks of the correct width and in the **Unfinished Track** colour. This option is also available from the context menu when editing the track.



If **Complete As Track** is used, either from the context menu or by using double-click, and the completion path is shown it will be used to complete the path quickly, without the need to check the track or push any more tracks away.

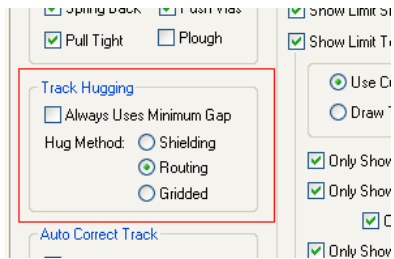


Insert Track - Hug Mode Improvements

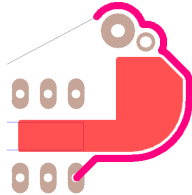
When hugging with **Online DRC** switched on, any obstacle you hit will be added to the hug path so you can hug around them, this includes **Copper**, **Pads**, **Mounting Holes** and **Vias**. This allows hugging to 'jump' to the next obstacle in its way.

Hug Mode Options

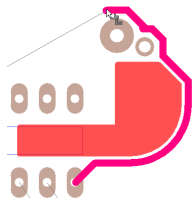
There are now three hug methods on the **Edit Track** page of the **Options** dialog.



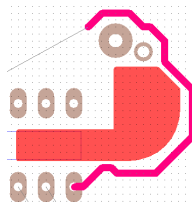
Shielding adds tracks that hug the obstacles as tight as possible, introducing arcs around pads and sharp corners (see below). This can be used to create shielding around tracks and other copper obstacles for RF protection for example.



Routing is like shielding, but will add 45 degree lines around pads and sharp corners. It will only use Arcs to hug around arcs in copper shapes.



Gridded will add a track with all segments a multiple of 45 degrees and snapped to the next track grid position away from the obstacle.



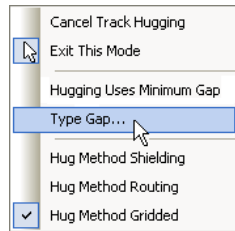
Check the **Always Uses Minimum Gap** option to force track hugging to always hug to the minimum gap allowed by the **Spacing Rules** in your **Technology**. Leave unchecked to allow track hugging to hug at the initial distance from the track to the shape, or to type a required gap.

Add Track

When in **Add Track**, and before clicking to start the track you can now select **Start Hugging** from the context menu. If you are directly over a shape on the current layer it will start hugging it, otherwise it will enter a special mode to pick the item to hug around.

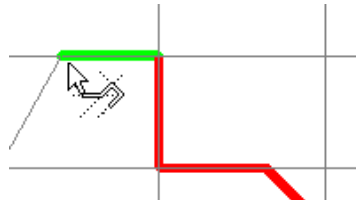
Type Gap

When in the special mode to pick the obstacle to hug, you can now type the required gap.



Insert Track - Follow Construction Lines

When editing a track (an existing track or new track) or breakout, you can now choose to use **Follow Construction Lines** from the context menu. The track or breakout will snap to the nearest construction line (if it can) and you will enter the **Follow Construction Lines** mode. A special cursor will be shown and wherever you move the cursor the track will follow along the construction line or circle and can corner when you reach an intersection with another construction line. You use a left click to exit this special mode and return to normal track editing.



When in **Insert Track**, and before clicking to start the track, you can now select **Follow Construction Lines** from the context menu. A new track will be started on the nearest construction line.

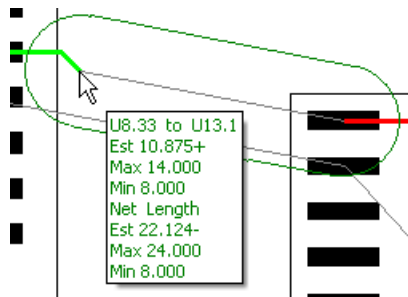
Changes to Interactive High Speed Routing Option

Show Track Limits

In Pulsonix V7.0, there are improvements to the interactive display of **Pin to Pin** and **Net Length Limits**.

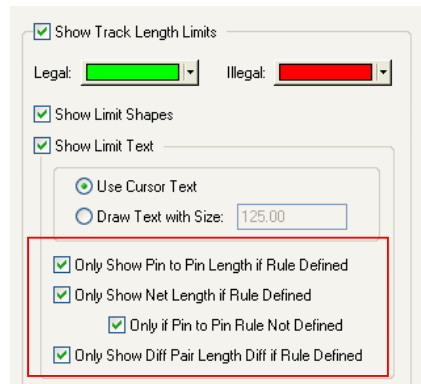
Previously in version 6.1, the display of Interactive High Speed rules worked so that if a Pin to Pin Length rule was defined, it showed that Length and the rule in the 'head-up' display. If there was no Pin to Pin rule, and if it had a Net Length rule, the Net Length and rule was displayed. If it had no rules defined, it would show nothing.

Now in Version 7.0, you have much more control over the Lengths to be displayed. Lengths can now be displayed without rules being defined and you can display **Pin to Pin Lengths** and **Net Track Lengths** at the same time by unchecking the pin rules options in the **Options** dialog and **Edit Track**.



Display the estimated Pin to Pin length, Net length and Diff Pair Length Differences

There are new check boxes on the **Edit Track** page of the **Options** dialog that can be **unchecked** to allow you to display the estimated **Pin to Pin length**, **Net length** and **Diff Pair Length Difference** if there are no limit rules defined.



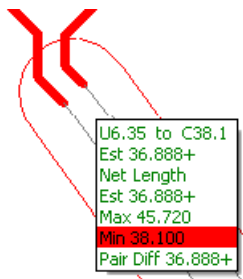
All the rules check boxes above in the red panel have been added.

Additional Net rule display controls

You can now uncheck the new **Only if Pin to Pin Rule Not Defined** check box in **Edit Track** on the **Options** dialog to display both **Pin to Pin Length** and **Net limits**. (before it always showed only one of them). The Net Limits are preceded by the label **Net** or **Sub-Net** depending on where the rule is defined.

Length difference and Max Limit on a Differential Pair

When editing a differential pair, you can now optionally show the **Length Difference** and **Max Limit**.

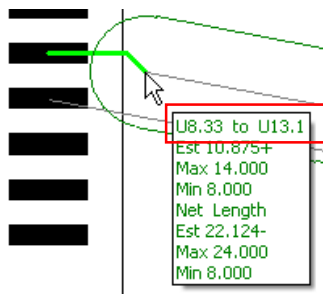


Resolving unrouted connections to obtain the Pin to Pin length

The algorithm for obtaining the Pin to Pin Length for paths including unrouted connections and 'T' junctions has been significantly improved to be more accurate.

Show Pin Names when showing Pin-to-Pin Lengths

If showing a **Pin to Pin length**, the names of the **pins** are shown (unless the limit is defined by a pin network, in which case the **Pin Network** label is displayed).



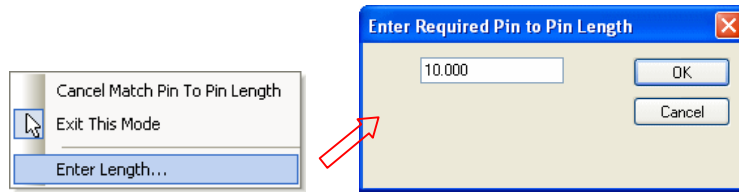
Match Pin To Pin Length

There is a new Interactive High Speed function on the context menu (**Match Pin To Pin Length**) when editing a PCB Track. This allows you to change the head-up track limit display to show the Pin to Pin length from another track that you are trying to match the length to.

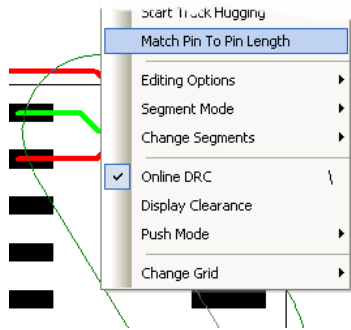
While editing a track, if you right click and select **Match Pin To Pin Length**, if you are over another track the option will be used, otherwise a mode will be entered with a special modal cursor to enable you to select the track to match lengths to.



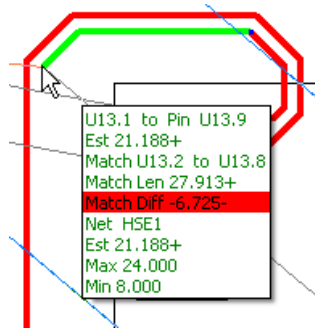
At this point, you can also type in the required length, rather than selecting another track.



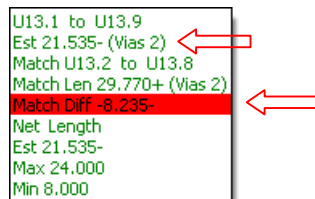
Selecting the option again off the context menu will toggle track matching mode off. Only tracks that are fully routed in a single path from Pin to Pin can be selected.



When matching lengths the head-up display will change to show the names of the pins we are trying to match, the length of the tracks between them, and the difference in lengths. The difference will be shown in the illegal colour if not zero.

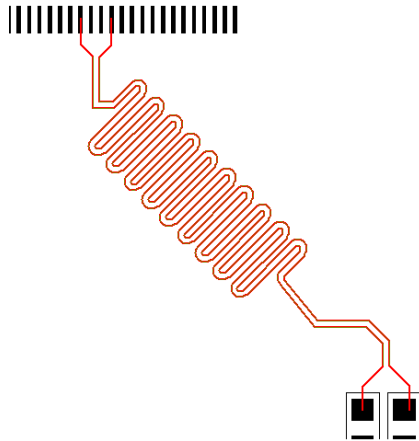


If the matched path has vias, or the current path being edited has or needs vias, the display will include the estimated and required via counts.



Serpentine Differential Pair Tracks

From within the **Interactive High Speed** option, you can now **serpentine a differential pair**. This works the same as when you serpentine regular tracks with the **Serpentine Tracks** option being available on selected **Differential Pair** tracks from the context menu.



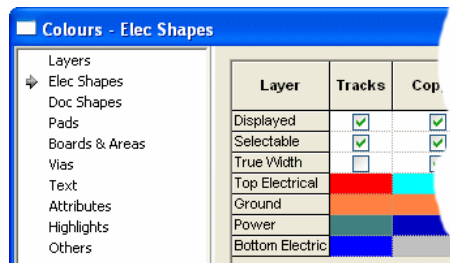
Copy & Paste of Differential Pair Routing

There is no user interface changes, but the **Copy** and **Paste** options now copy differential pairs. If all four pads are being copied the differential pair will also be copied. In **PCB**, paired tracks are also retained when copying the whole differential pairs. In **Schematic** designs this allows you to **Cut** and **Paste** partial designs to another page and not lose the differential pairs.

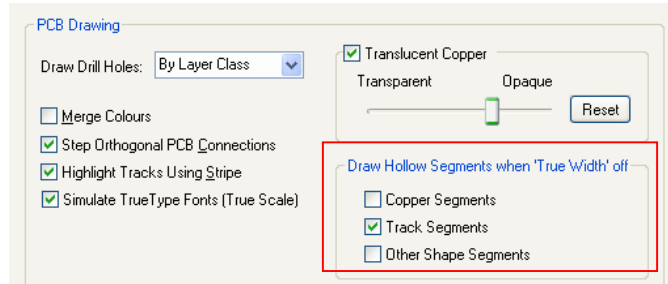
Draw Hollow Segments

There are new display options to draw copper, track and other shape segments that are not 'True Width' as Hollow outlines instead of centrelines.

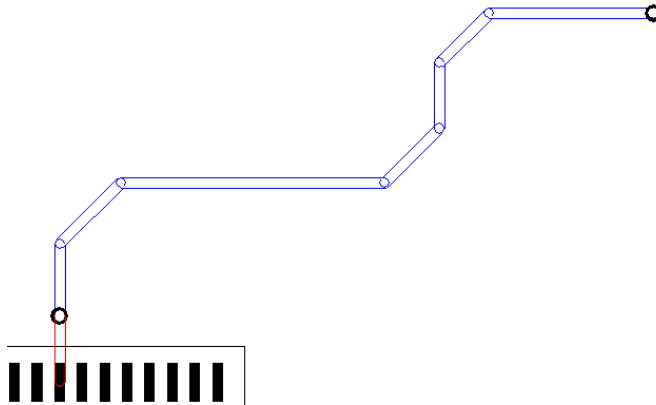
When copper, track and other shape segments have their **True Width** switched **off** in the **Colours** dialog, you can now optionally enable the drawing of those shape as hollow width instead of centreline.



Once the **True Width** is **set off** in **Colours**, from within the **Options** dialog and **Display**, you can select one of the three check boxes to set copper, track and other shape segments to hollow.

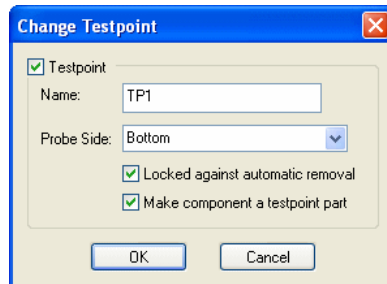


The design will now draw the shapes as hollow as shown below for track segments.



Testpoint Components

You can now use **Change Testpoint** or the **Test Properties** tab on a component that is a single pin component to turn it into a testpoint component. Use the new **Make component a testpoint part** check box. A testpoint component is one where the testpoint name and component name are automatically kept the same.



For a selected component, the status bar and tooltips now show if a component is a testpoint component.

Improvements to Copying Cutouts

Copying Cutouts

There is new option on the **Options** dialog under **Interaction** called **Copy Single Cutout As Shape**. This allows **Copy** to take a single selected cutout and turn it into a single shape on the clipboard. This can then be used by the **Paste** command within the **Insert Cutout** or **Insert Merge** operations to add it back in as another cutout.

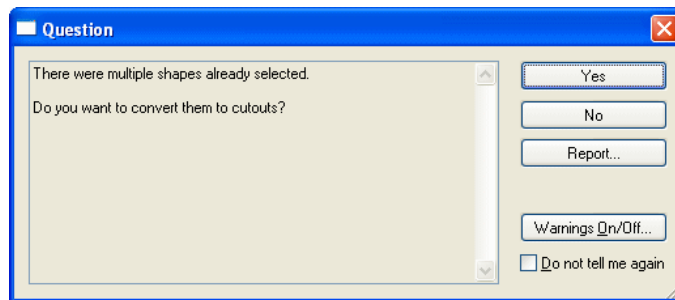
Insert Cutout and Insert Merge changes

Insert Cutout and **Insert Merge** have been changed to stay in their mode after using **Paste**. This makes it easier to paste the same Cutout shape multiple times.

Insert Cutout can now convert multiple shapes to become Cutouts of a selected shape.

This is done by first selecting the multiple shapes you want to become Cutouts, then place them over the master shape. Next, with the shape still selected, use **Insert Cutout** to pick the master shape to receive the cutouts.

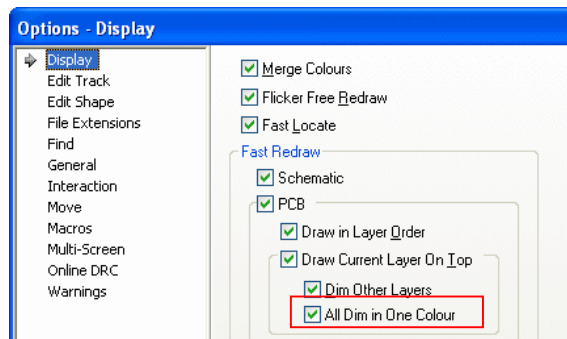
A warning dialog will be displayed, asking if you want to convert all the preselected shapes to cutouts.



If you click **Yes**, the shapes will be cut out of the master shape. If you click **No**, the shapes are ignored.

Drawing Dimmed in Uniform Colour

The ability to dim (lowlight) while editing using a uniform colour for all items has been added. A black background will show all dimmed items as dark grey and a white background shows very light grey, etc. This is controlled by a new check box on **Tools** dialog and **Options, Display**. Select **All Dim in One Colour** to enable this feature.



Number of Drill Symbol Shapes Increased

More drill symbols have been added. Including 3 to 6 point stars, 3 to 5 leg symbol, 3 leg asterisk. The filled property of a shape which is drawn with lines (such as asterisk, target, etc) determines if the shape is drawn with a predefined width, or minimum width (the same as unfilled closed shapes).

Diameter	Plated	Id	Symbol	Symbol Size	Sy
0.5	Y	N	Triangle	0.5	
0.6	Y	A	3 Leg Asterisk	0.6	
0.7	Y	B	3 Leg Symbol	0.7	
0.8	Y	C	3 Point Star	0.8	
0.8	Y	D	4 Leg Symbol	0.8	
0.8	Y	E	4 Point Star	0.8	
0.9	Y	F	5 Leg Symbol	0.9	
1.0	Y	P	5 Point Star	1.0	
1.1	Y	G	6 Point Star	1.1	
1.1		H	Asterisk	1.1	
1.2	Y	L	Cross	1.2	
1.6	Y	I	Diamond	1.6	
1.6		O	Octagon	1.6	
2.0	Y	J	Plus	2.0	
3.2		M	Pointer	3.2	
3.2	Y	K	Round	3.2	
			Square		
			Target		
			Triangle		

These would be displayed in the design on the drill table and in the drill drawing like this:

Diam (mm)	Qty	Id	Plated	Symbol
0.813	165	C	Y	△
0.889	9	D	Y	⋄
1.067	6	E	Y	☆
1.270	3	F	Y	⬠

Change of Angle Resolution to 1/100,000th Degree

The internal resolution of **angles** is now 1/100,000th of a degree. There is no interface change for this.

This has been changed to allow a more accurate calculation of the ‘crossing’ position when lines intersect with arcs for example.

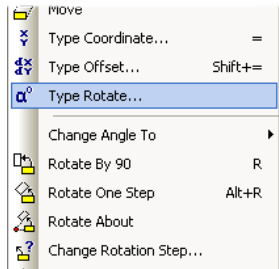
For existing users who had an accuracy problem snapping items to intersections, between circles and straight construction lines, you might have to reposition the lines to more accurately define the intersection points.

Rotation to Polar Grids

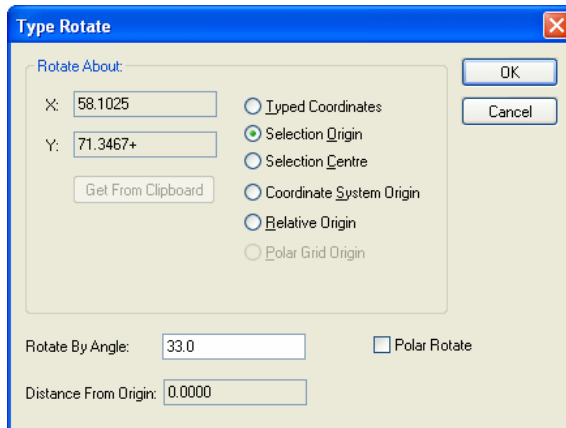
There are new rotation functions available:

Type Rotate

This new mode is available from the **Select Mode** and **Move** context menus.

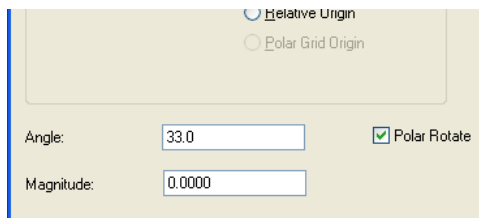


Use this to rotate the current selection about a particular point and by a specified amount by typing the values into a dialog.



If used in **Move**, the rotation is applied to the selection in its *original* position unless you are adding the first item. This way you can position subsequent items rotated relative to the first item added. Also if you are using **Duplicate** or **Apply Layout Pattern**, you do not have to use **Cancel Move** before rotating into position.

The dialog can also be used in Polar form by supplying the **Angle** and **Magnitude** of where you want the origin of the selection to be.

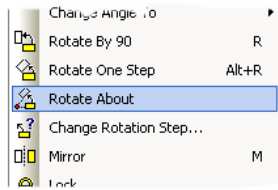


Polar Rotate

Check this box to change the two edit boxes to accept an **Angle** and **Magnitude** to define the position of the selected items using absolute polar coordinates applied about the rotation point. The selected items will be rotated about the rotation point until their origin is at the required angle and then moved towards or away from the origin until the distance from the selection origin to the rotation point is equal to the required magnitude.

Rotate About

This new mode is available from the **Select Mode** context menu. For a selected component(s), you can choose to **Rotate About** from the menu and then set the rotation origin.



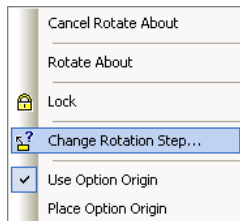
This option allows you to enter the **Rotate One Step** latch mode operation and lets you pick the point that you want to rotate the selection about.



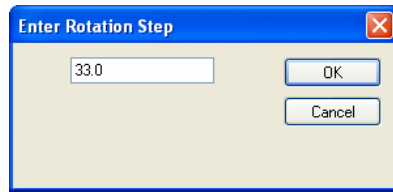
Once this is selected, the 'rotate about' origin is fixed and you click the mouse to make one rotation step.



By right clicking in this mode at any time, the **Change Rotation Step** can also be changed:



From the dialog, type the rotation step required:



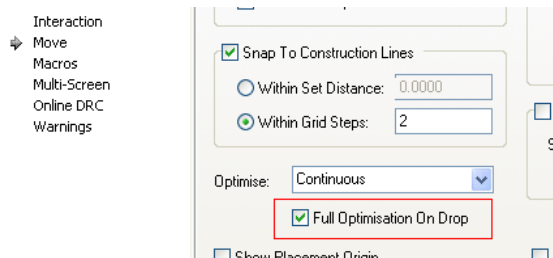
When the point has been picked, left click anywhere to perform the rotation about that point.

Change Rotation Step

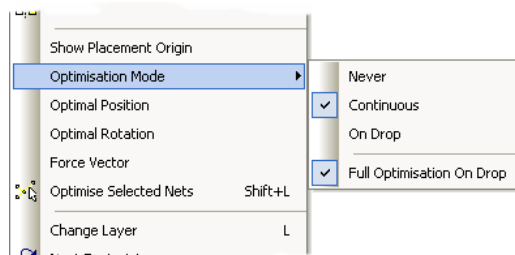
This option is available from **Select Mode**, **Move**, **Rotate Latch Mode** and **Add Construction Line**. Use it to change the rotation step in the interactive options.

Full Optimisation After Move

From the **Options** dialog and **Move**, there is a new move option (**Full Optimisation On Drop**) to perform a full optimisation of all nets involved on finishing the move. This works for **Optimisation** in **Continuous** and **On Drop** mode.

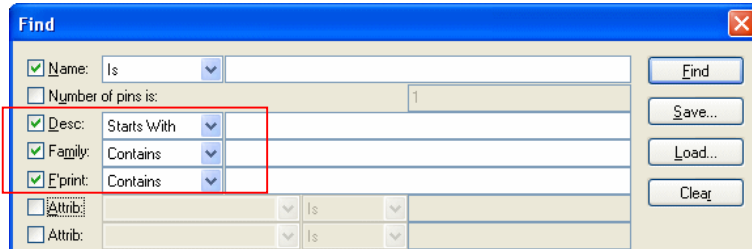
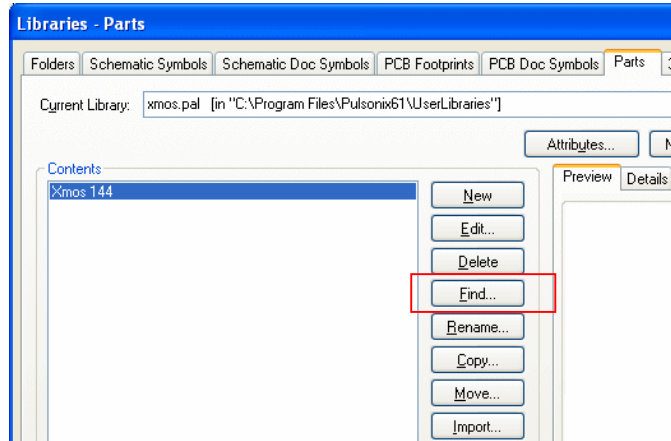


This option is also available from context menu when moving components.



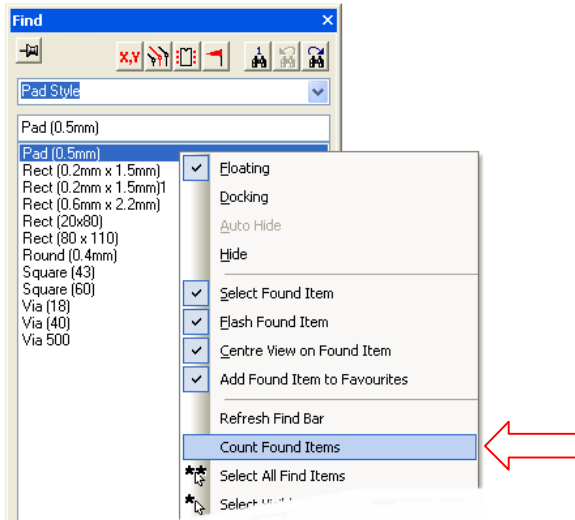
Find Library Items

You can now **Find** in the **Parts Library Manager** and on the **Insert Component** option using fields such as component **Description**, **Part Family** and **Footprint**.

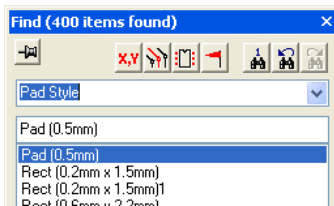


Find – Count Find Items

There is a new option on the Find browser - **Count Found Items**. This is available on the context menu to count all items that satisfy the search criteria. (item type, name filter and attribute condition).



The count results are shown in the Find bar title. Categories of **named** items always show the count whenever their list is refreshed.

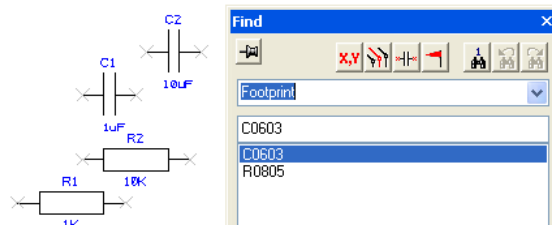


Find – Via Spans

When using **Find Via**, it now allows you to find composite via layer stacks. The layer filter now includes composite spans, and if one is chosen, all vias in the composite stack are found and selected together.

Find – Footprint in Schematics

The **Find Footprint** option is now available for Schematic designs. It finds gates on components that have a explicit footprint name defined, i.e. not using the default footprint from the part.

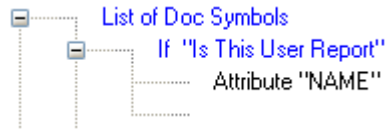


Report Maker Command Additions

New commands have been added to Pulsonix V7.0:

Is This User Report command

Is This User Report is used within **List Of Doc Symbols** to return true if the current documentation symbol is the inserted user report that is currently having its report text updated. This way you can add attributes to the inserted user report containing parameters that influence the contents of the report.



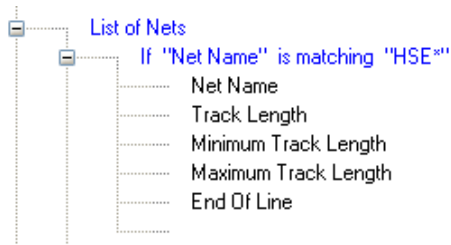
When run, the report will test the doc symbol and plug the requested value back into the report. For example, like this - where the report created is copied twice in the design, both instances containing the attribute NAME but both using two different values (**Att Val 1** and **Att Val 2**):



For example, you could insert a report that lists the ungated power and ground pins on a component by adding the component name as an attribute on the inserted report.

Report Maximum Track Length and Minimum Track Length

From within a PCB report, the **Maximum Track Length** and **Minimum Track Length** can be used within **List Of Nets** to report the min and max track length values from the Net Class rules for the current net. These values can then be used to test against the Track Length field to report failures.



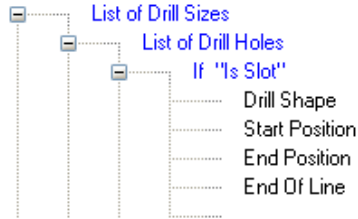
When run, the report could look something like this (using added Pre text to the commands):

```

Units: mm
HSE1   10.000   Min: 8.000   Max: 24.000
HSE2   29.770   Min: 8.000   Max: 24.000
  
```

Report Start and End Positions of Oval Slots

The Report Maker can now report the **Start** and **End Positions** of **oval drill slots**. Used in the **List Of Drills** command, it reports the start or end position of the thick line that would be used to draw the slot.



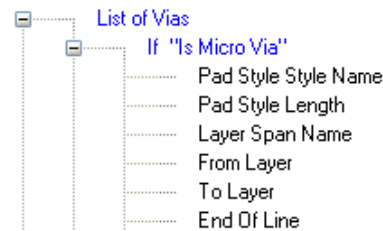
Report Drill Shape, Slot Width & Slot Length

New commands for **Drill Shape**, **Slot Width** and **Slot Length** have been added. These are additional sub-fields of the **Pad Style** command to report information if the pad style represents a slot.



Is Micro-Via command

To support Micro-vias, a new command, **Is Micro-Via** has been added. This is available when using the command for **List of Vias**.



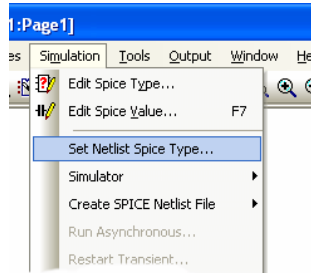
Spice Netlist Exporter

From within Schematics, export to other types of Spice, such as LTSpice, Basic Spice and PSpice. Pulsonix Spice format can also be output, this is also compatible with the SiMetrix and Simplis Spice simulators.

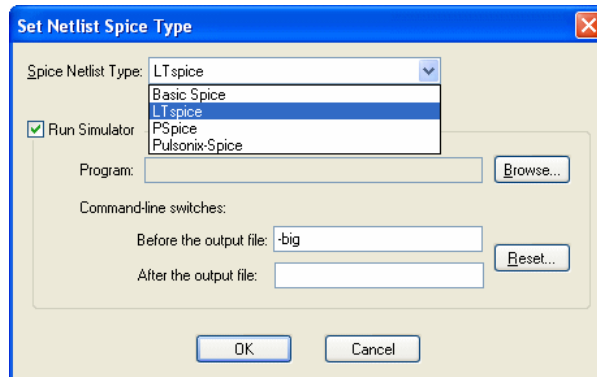
Most of the integrated simulation features available in Pulsonix are specifically designed for use with the Pulsonix-Spice simulator. For example, Random Probing, Bias Annotation and Multi-Step Analysis. If you do not have this simulator, you can still prepare components for Spice output and add circuit stimuli and fixed probes, and generate a Spice netlist.

To set the Spice Netlist Type

Before running a Spice netlist, you must choose the type to export to, each one has its own ‘flavour’ of Spice. Select the **Set Netlist Spice Type** from the **Simulation** menu in Schematics.



First choose the **Spice Netlist Type** from drop down list for the simulation file. The simulator commands in the netlist file will be formatted to suit the chosen simulator.



If available, you can choose to use the **Run Simulator** check box to specify whether you can directly run a simulator as well as producing the Spice netlist.

Use the **Browse** button to locate the program to be run on your computer. Where possible we have added filters to this dialog to suggest a possible simulator program name.

Add any command line switches that are needed to run up the simulator, either before or after the output filename. Initially these will be set to default values for each simulator type. If you change them you can use the **Reset** button to revert back to the default values.

Press **OK** to accept the change. If **Run Simulator** is checked you will be able to use the **Simulate Design** option to pass your Spice netlist directly to your simulator.

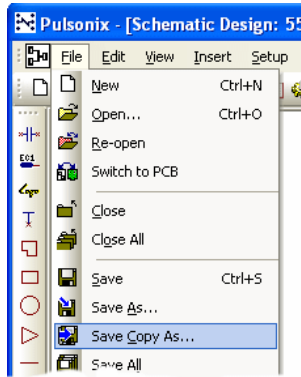
Outputting a Spice Netlist

To generate a Spice netlist in the format you have chosen, If the **Program** has been previously set up, use the **Run Simulation** option from the **Simulator** menu. If this has not been set up, you have to use the **Create Spice Netlist File>** (the **Simulator** menu), then choose one of the page options and the location using the **Save As** dialog.

Save Copy As

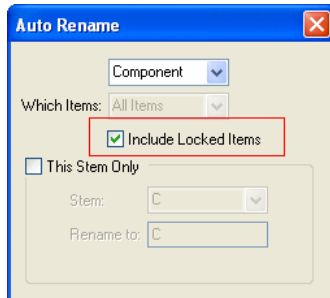
You can save a copy of the current design under a new name, whilst continuing to edit the current design.

From the **File** menu, select **Save Copy As**. From the **Save As** dialog, choose the location and filename to save the file to.



Auto-Rename - Exclude Locked Items

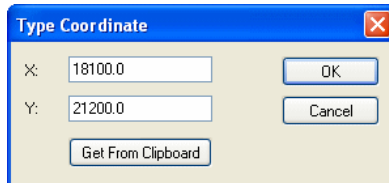
While using **Auto Rename**, you can now exclude locked items from being renamed by unchecking the **Include Locked Items** check box (you cannot lock a net).



Type Coordinate – Get From Clipboard

When using **Type Coordinate** to position an item, there is a new button available - **Get From Clipboard**. This is enabled if the clipboard contains a text item that includes a valid coordinate. The text in the clipboard may have been copied from a report for example.

If pressed, the coordinate X and Y values will be placed into the appropriate edit boxes.



‘Align’ options available in Define Pad Shape

The **Align** options are now available whilst in **Define Pad Shape**. They are available on the **Edit** menu and available as commands. The **Alignment** toolbar is also available for use by selecting it in the **Customise** dialog and **Toolbars**.

Variants Improvements

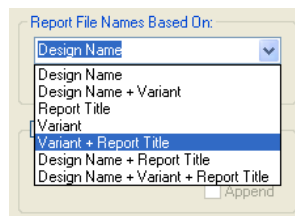
Filled Shapes with Dashed Line Styles

When a shape uses a **Dashed** line style and is also **Filled**, the shape is not drawn filled but is still selectable as a solid shape. Previously, this was not possible, the shape was drawn as solid. This is useful for Variant Components for example.



Name base for Variant Reports

From within the Report option on the Output menu, you can now base your report name on **Variant** or **Variant + Report Title** or **Design Name + Variant + Report Title** using the selections on the drop down list.



Cadstar 12 support

Pulsonix 7.0 can now import Cadstar 12 format ASCII files in .csa and .cpa formats.

Library Changes in V7.0

In Pulsonix 7.0, new Parts libraries have been added and some existing libraries modified, over 20,000 new parts have been added, the additions comprise the following:

Manufacturer	Items added
Altera	1110
LT	1606
Maxim	15884
Microchip	22
NatSemi	572
Xilinx	3264

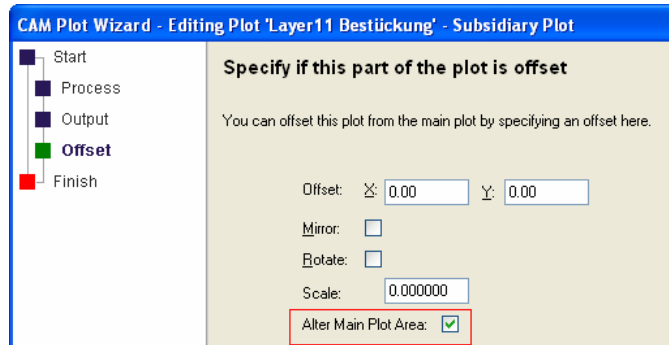
Mounting Holes Placed on Non-Electrical Layers

Mounting holes that are not on a net can now be placed on non-electrical layers. This supports the use of mounting holes in footprints as glue spots, and allows you to change individual instances in a PCB design to have an alternative style for the Mounting Hole to cope with particular manufacturing issues on a per component basis.

Cam Plot Changes

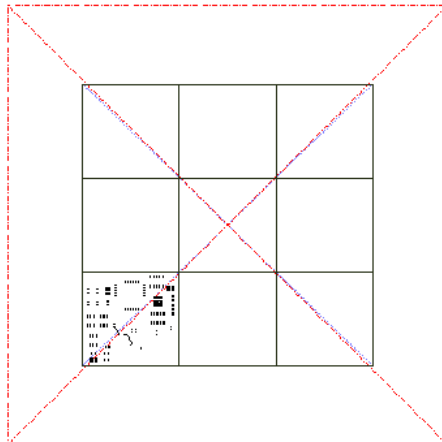
Alter Main Plot Area

There is a new switch on the **Plot Wizard Offset** page named **Alter Main Plot Area**. This is used when you are adding additional plots to a main plot to create a composite. You specify if the offset plot area will be added to the main plot area. If checked, and the main plot is using **Auto Fit** or **Auto Shift**, then the main plot positioning will change to include the appropriate area in this additional plot. If left unchecked the original positioning of the main plot will not be affected by the offsetting of the additional plot.



Step and Repeat offset of 0

A step and repeat offset of 0 is now allowed. This enables you to plot the separation between boards so they can be cut (once manufactured) with a single cut. It will align the centreline of the board outlines. Effectively, the board outlines 'butt' up against each other.

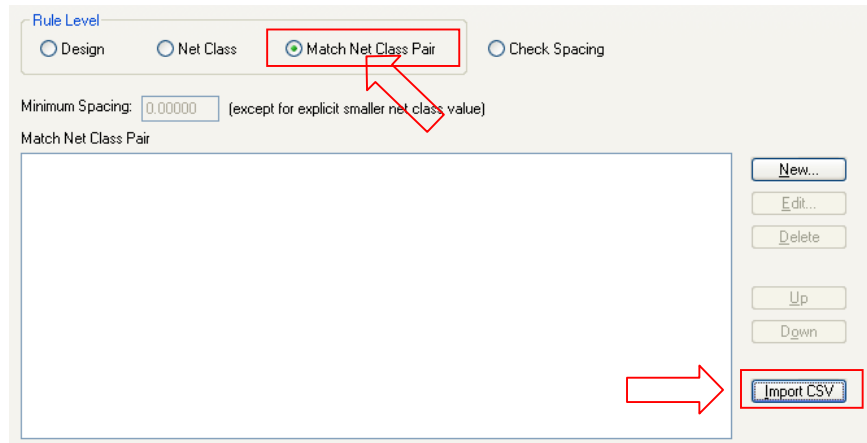


Import Spacings Rules using CSV Format Files

The ability to import **Net Class** names and **Class to Class Spacing Rules** into the Schematic or PCB design **Technology** using a CSV file has been added to Pulsonix.

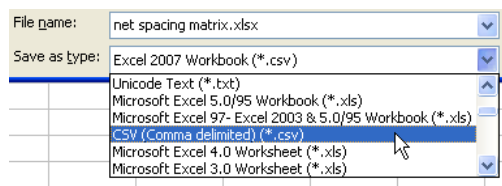
Using the Import CSV feature

The **Import CSV** button appears on the **Spacing Rules** dialog along with **Match Net Class Pair** rule level pre-selected.

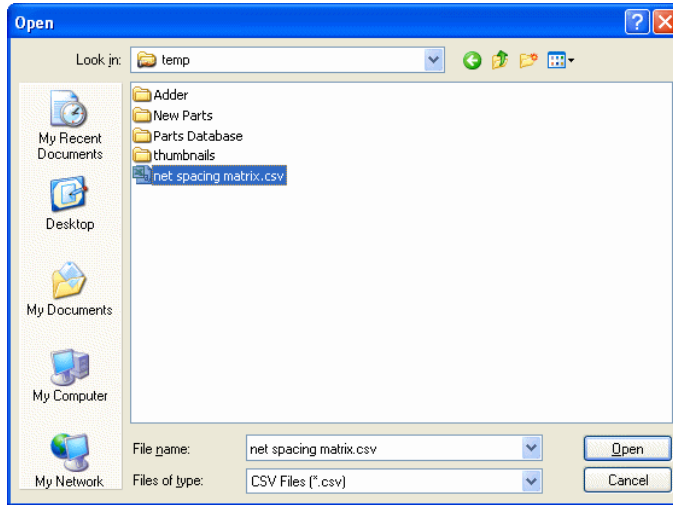


When selected, the **Open CSV** file dialog is displayed. You will be prompted to select the CSV file to import. The **File of Type** drop-down list is set to *.CSV.

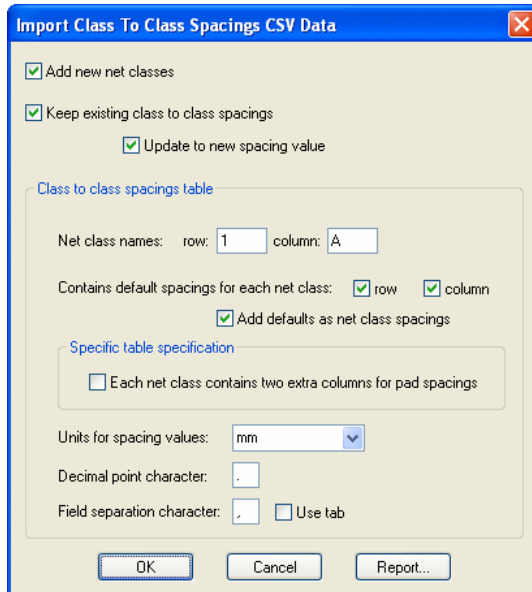
*Note: If you have been editing your spacing rules in Microsoft Excel for example, you will need to save the rules into a CSV file format. To do this, when using the **Save As** option select **CSV (Comma Delimited) (*.CSV)** format from the **File of Type** drop-down list.*



Select the file required from the dialog and press **Open**.



The **Import Class To Class Spacings CSV Data** dialog is displayed. This dialog allows different styles of CSV files to be imported. Currently, this accommodates a number of format 'styles' that have been seen within our development department.



Add new net classes – If new net classes need to be added because they exist in the import file but not in the design, checking this option will allow this. With this not selected, new net classes are not added and any class to class spacings using them will also not be added.

Keep existing class to class spacings – If the design already has class to class spacings, you can choose whether to remove them all and add just the spacings from this import file, or to keep the existing ones. Uncheck this option to remove them all.

If keeping existing spacings the following check box becomes enabled for you to use to decide what to do if the import file contains class to class spacings that already exist in the design.

Update to new spacing values – If during the import, the option detects an existing class to class spacing but the value is different, then you can choose to keep the existing value or update it with the new value. Unchecking this box will mean only new class to class spacings will be imported.

Class to class spacing table

You can tell Pulsonix how to treat the imported data. It is assumed that the spacings table is a grid with net class names in the top row and down the left most column. This spacings table can appear anywhere in the CSV spreadsheet and so the user will have to supply its position. The first option tells the importer which rows and columns contain the net class names. For this, you define the Row name and Column name.

Net Class names Row Column – enter values like **1** for **Row** and **A** for **Column** names. If for example, you had a title at the top of the page on row 1, the net class name may then appear on Row 2, you would then enter Row 2 instead. You can use either numbers, where 1 is the first, or letters, where A is the first and AA and AB are 27th and 28th etc.

Contains default spacings for each net class Row Column – Some spacing tables contain a special row below the net class names row, or a column to the right of the net class names column, or both, containing default spacing values for each net class. These values do not form part of the class to class spacing values grid, and so the importer needs to know about them so it can skip them, or read the default values from them, to reach the main spacing values grid. Check the **row** and/or **column** boxes if you have these values in your table.

If either box is checked the following check box becomes enabled to use to decide what to do with these default net class spacing values.

Add defaults as net class spacings – Check the box if you want to add these default values for each net class they are defined for to the design spacing rules at the **Net Class** rule level. Leave the box unchecked to ignore them.

Specific table specification

The basic spacing table supported will contain values in each cell to be used for all spacings between the net classes defined by its row and column. Usually there will only be values defined for one diagonal half of the grid to avoid duplicating the data, but a full grid of values can also be read.

In order to import a variety of spacing table formats, a section has been included in this dialog to include check boxes for the user to choose any of the specialised table considerations we have been made aware of.

Each net class contains two extra columns for pad spacings – Some spacing tables we have seen from a specific customer contain two extra columns after each net class containing special spacing values to match their manufacturing requirements, the maximum of which is used for the **Pad to Pad** and **Pad to Copper** spacing value. Check this box if this matches your spacing table. If checked, the import report will show this special pad spacing in brackets after the main class to class spacing value.

Units

Units for spacing values – select the units used in the CSV file from the drop-down list. You can specify different units to those in your design and used during import. If different in the CSV file, Pulsonix will convert the imported data into your design units during the import. Only spacing values that are numbers will be imported, blank fields and fields containing text will be ignored.

Using the dialog

Select the options you require to match the CSV file and press **OK** to attempt to import the spacings, or press **Report** to show a report of what spacings it will add.

The Import CSV Report

The import **Report** button can be used before an import to inform you of what will be imported and a report will be created after import to tell you what has been imported. You can also run it a third time to verify the original file against the current Spacing contents, to see if anything has changed since you last imported the file. This way you can use this feature to import the net classes and original class to class spacing values when starting a schematic or PCB design, and then use the **Report** button towards the end of the design to show which class to class spacing sets now contain different values from the original spacings spreadsheet.

Sample Report

Import Class To Class Spacings

Report Written : 21/10/2010 12:02:14 PM
Design Path : CSV Import Sample.pcb
Design Title : Import CSV Example Design
Created : 21/10/2010 11:49:35 AM
Last Saved : 21/10/2010 13:49:35 PM
Editing Time : 12 min
Import File : C:\temp\net spacing matrix.csv
Units : mm

Import Summary

Net classes found in import file row : 12.
Net classes found in import file column : 12.
New net classes to be added : 9.

Default net class spacings found in import file row : 9.
Default net class spacings found in import file column : 9.
Existing net class spacings already with correct value : 0.
Net class spacings that will have value changed : 9.

Class to class spacings in import file : 63.
New class to class spacings to be added : 63.
Existing class to class spacings already with correct value : 0.
Existing class to class spacings that will have value changed : 0.

The following net classes will be added:

LCD safe
LCD zener barrier
pump
segregated -11V supply
segregated -7V supply
segregated 3V switched

sounder

unsafe power

vibrator

The following existing net class spacings will be changed:

"LCD safe" : 1.5
"LCD zener barrier" : 1.5
"pump" : 1.5
"segregated -7V supply" : 2.0
"segregated -11V supply" : 2.0
"segregated 3V switched" : 2.0
"sounder" : 1.5
"unsafe power" : 1.5
"vibrator" : 1.5

The following new class to class spacings will be added:

"LCD safe" to "Ground" : 1.5
"LCD zener barrier" to "Ground" : 1.5
"LCD zener barrier" to "LCD safe" : 1.5
"Power" to "LCD safe" : 1.5
"Power" to "LCD zener barrier" : 1.5
"pump" to "Ground" : 1.5
"pump" to "LCD safe" : 1.5
"pump" to "LCD zener barrier" : 1.5
"pump" to "Power" : 1.5
"segregated -7V supply" to "Ground" : 2.0
"segregated -7V supply" to "LCD safe" : 2.0
Etc. (full list of 63 line items truncated)
Etc. (full list of 63 line items truncated)
"vibrator" to "segregated -7V supply" : 2.0
"vibrator" to "segregated -11V supply" : 2.0
"vibrator" to "segregated 3V switched" : 2.0
"vibrator" to "Signal" : 1.5
"vibrator" to "sounder" : 1.5
"vibrator" to "unsafe power" : 1.5

----- End Of Report -----