Pulsonix Design System

V4.0 Update Notes

Contents

Chapter 1. Getting Started With V4.0

About these notes

These update notes are provided for existing users as a supplement to the existing Pulsonix Users Guide. These notes are to highlight new features in version 4.0 and to briefly describe their use.

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

Installing the New Version of Pulsonix

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

As with any other installation of Pulsonix, insert the CD and wait for a short time. The *Autorun* facility will start the installation procedure. Follow the on-screen messages from the install wizard. You can install Pulsonix version 4.0 on top of your existing installation, you do not need to uninstall any old version first.

New License

Version 4.0 requires a new license that will be supplied to you with the update or by email.

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

New System Files

There are no new system files for version 4.0. However, if upgrading from an older version than version 3.0 then there is a change to the system files that are installed. If you have this older version, then during installation, from the **Change Components** page of the installer, then the **System** box should be checked.

New Menu Options

Although detailed later on under *Customise Dialog Changes*, you should note that the standard menus and toolbars have been changed. If you wish to modify yours to match the new standard then you can use the **Reset** options from within the **Customise** dialog (on the **Settings** menu) for **Toolbars** and **Menus**.

Installation Notes for existing Pulsonix Spice Users

File locations

The simulator files are now in the \Pulsonix-Spice folder beneath the main Pulsonix folder. Pulsonix Spice version 1.0 files remain in the \Spice sub-folder, as it may contain some of your own model files.

The following files now live in the \Application Data folder on your computer, in a \Pulsonix\Pulsonix-Spice sub-folder.

- out.cat

- user.cat

- map_symbols.txt

- oem_symbols.txt

If you don't know how to get to the application data folder on your computer use the utility program **FindAppDataDir.exe** to show the folder path. This is located in the ...\Pulsonix-Spice\Support\Help folder.

Installing your own models

After installation, the simulator will contain the newly installed set of model libraries. You will need to re-install your own model files into the Simulator. You can install your model files or a folder containing them by picking them up in windows explorer and dropping them into the Pulsonix Spice command shell.

If you have made changes to the categories or Parts used by the models using the **Associate Models & Parts** option, you'll also need to copy the **user.cat** file to the new installation folder. To do this, copy **user.cat** from the 3.1 ...\Spice\Script folder to the Pulsonix Spice application data folder (described above).

Pulsonix Parts

If you had changed the part names for the simulator's built-in models (e.g. NPN) using the Pulsonix 3.1 **Simulator Setup** dialog. You will either have to do this again with version 4.0,or copy the **map_symbols.txt** file from the 3.1 ...\Spice\Script folder to the Pulsonix Spice application data folder (described above).

Use the Pulsonix **Library Manager** to make sure the Pulsonix 4.0 program is pointing to your own parts libraries. This should already be set up as Pulsonix library paths are retained from 3.1.

If you have changed the standard spice parts (Spice.lib) or have new parts that contain spice information, you will need to run up the Pulsonix **Simulator> Simulator Setup** dialog from the **Simulation** menu and use the **Update Simulator Parts List** button to recreate the **oem_symbols.txt** file to inform the simulator about your parts.

Simulator Scripts

If you have your own scripts, either copy them to the ...\Pulsonix-Spice\Script folder, or change the simulator to point to your scripts folder using the **File Locations** page of the Simulator's **File** menu and **Options**, **General** dialog.

Chapter 2. General Options

Updated GUI

Application Look

The appearance of the Pulsonix user interface can be tailored using a new option named **Application Look**, this is located on the **Customise** option.
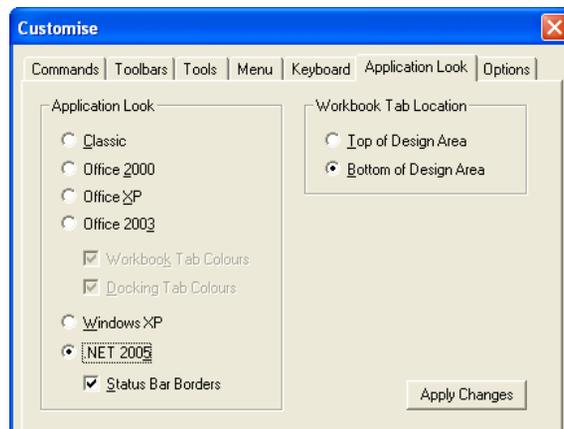
By default the .NET 2005 look will be adopted at installation with the workbook tabs at the bottom of the design area.

Changing the Application Look will affect the appearance of items such as windows, toolbars and menus. Use the **Apply Changes** button to apply the changes to the design.

XP Look

If the Windows Classic theme is selected on the XP platform, Pulsonix will revert to its previous style interface. On all non-XP platforms, the interface will remain unchanged.

XP Themes and Appearance

When running under Windows XP, Pulsonix will now make use of the new XP visual contents such as Themes and Appearance settings, which are available on the **Desktop** via the **Display Options** in the **Control Panel**. Using the Windows XP theme will adopt the current theme colour and display controls, such as buttons, combo-boxes, and lists will appear with the new XP look and feel.

Auto-hide Dockable Windows

All dockable windows, such as World View, Layers etc. may now be switched to be an Auto-hide window. This means they can still be docked but now you can also make them docked but visible as a button to the side of the Pulsonix framework (see below – this example of the three windows are docked on the right side of the design area). When you hover the mouse over one of these buttons, the appropriate window 'slides' into view ready to use.

There are three small icons along the top of this window, (moving left to right) these indicate a drop down shortcut menu (also available as a shortcut menu in the window by right clicking the mouse), a pin to enable the dialog to be sliding to conceal it or docked to be permanently displayed, and a **Close** button to dismiss the dialog altogether.

Tabbed Windows

The window can also contain tabs within any other dockable window options. You can switch between any of these windows by clicking on the appropriate tab.
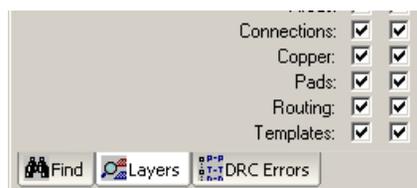
A tabbed window contains multiple dockable windows and may be manipulated like any other dockable window in that it may be resized, docked, floated or auto-hidden. Additional dockable windows may be dragged into the tabbed window and existing tabs may be dragged out to become individual windows once again. Tabs may be re-ordered simply by dragging them into the desired order.

Depending on the application 'look', Pulsonix will now also provide you with some extra features to aid the docking of tabbed windows (these are not available in the XP Look).

Docking Indicators

When running the latest application look, docking indicators appear to show the potential dock positions which can then chosen by releasing the drag with the mouse pointer over the appropriate indicator. The docking indicator is different when over an existing docked window.

To reposition a docked window, press and hold down the left-hand mouse button on the window title bar border and drag the window frame to the desired docking indicator. Any of the four 'outside' indicators will allow the window to be docked. By dragging it over the centre indicator, it will be added to the existing window as a new tab.

When used on a dockable window it looks like this:
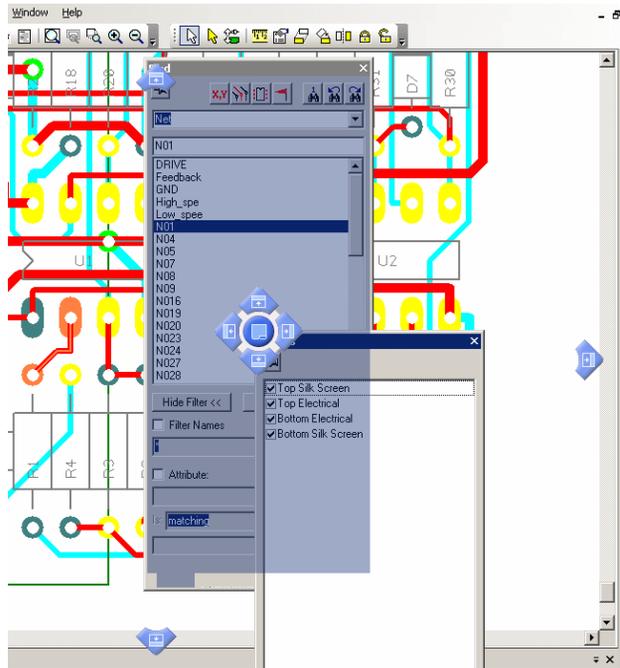
Outside the main docking indicator are four other position indicators. These show that the window can be docked to the adjacent edge of the design area of main Pulsonix framework.

New tab dialog style

Dialogs that use tabs to select pages have been replaced with the new style dialog showing the individual pages along the left hand side of the dialog. (Note: not all tabbed dialogs have been changed).
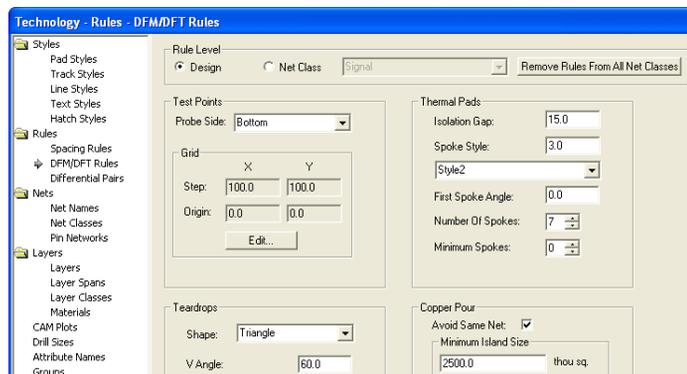
The tabs are categorised (such as Styles, Rules etc.) and sub-categorised into functional areas (such as Pad Styles, Track Styles etc.), using a tree structure.

Each category can be expanded or contracted using the small 'folder' icon next to the category (in the example below, *Rules* is the folder name).

The page that is open is shown with a small arrow next to the category.

Customise dialog changes

The **Customise** option (on the **Tools** menu) has been changed to allow new functionality. You can customise menus, add User tools to menus and toolbars, set up shortcut keys (**Keyboard**) in this dialog, create your own buttons and change some toolbar options.
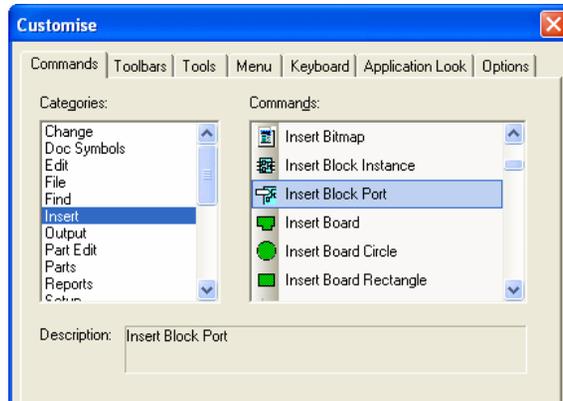
Customise menus

Menus can be customised and named using your own names, they can contain your own set of commands. This is similar to toolbar customisation.

From the list of **Categories**, you can select the **New Menu** option. This will allow you to add a new menu item to the menus.
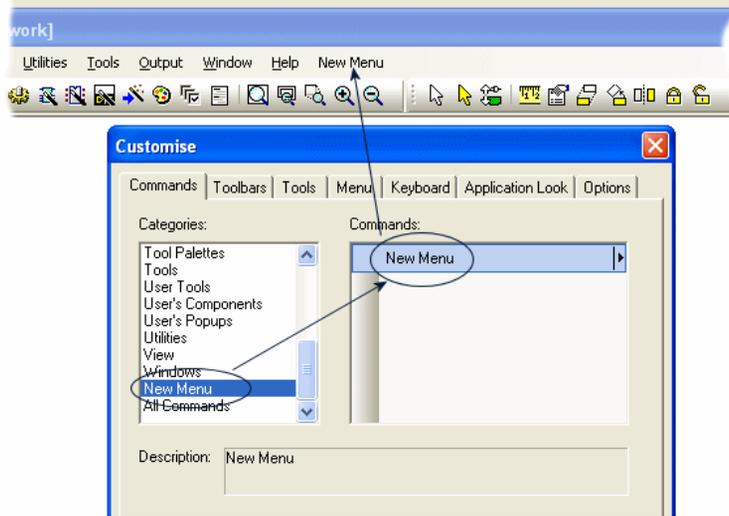
When in the **Customise** dialog, dragging a new menu item over a menu bar item will cause the corresponding drop down menu to be displayed. While in a specific application (such as the PCB Design Editor), if the relevant menu isn't displayed use the **Menu** tab and select the menu for the application required. All Pulsonix application types are available in the this drop down list. (see Menu Customisation later on).

By right clicking on the **New Menu** name (while still in the **Customise** dialog), you can change the name of the menu item.

You can edit the menu name by using the edit field **Button Text** on the **Button Appearance** dialog. This dialog also allows you to customise the button appearance.
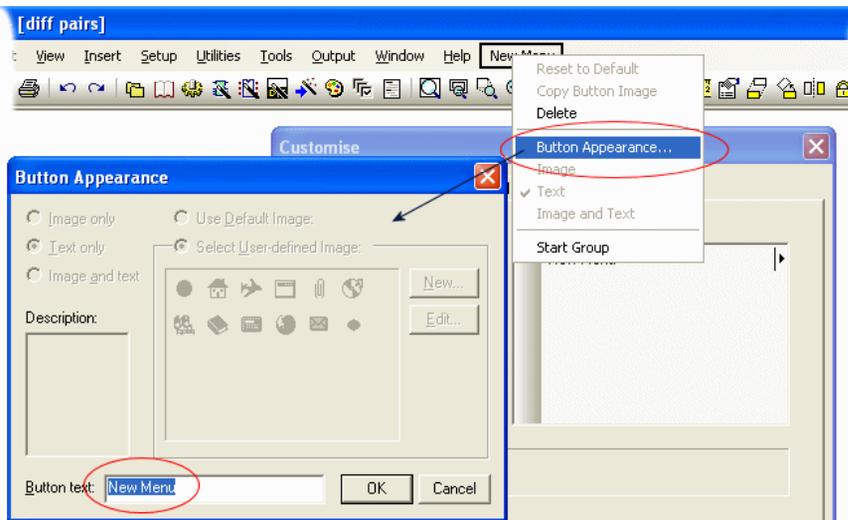
The dialog is split into four sections:

Choice of Appearance - At the top left of the dialog use the radio buttons to choose between **Image**, **Text** or **Both**.

Description - Shows a description of the tool the button represents. This is not editable.

Button Text - Enabled if you have chosen to show text on the button. Leave as the default option name, or type in your own text to display on the button. To reset the button to use the default option name, use the **Reset To Default** option from the shortcut menu.

Button Image - Enabled if you have chosen to show an image on the button. Choose between using the default image (shown next to this option), or using a user defined button image, in which case the user image palette will be enabled to allow you to select one. An initial set of images are provided, but you can alter these or add new images using the **New** and **Edit** buttons next to the image palette. When either of these buttons are pressed the **Button Image Editor** dialog is displayed to allow you to create your own image.

User defined icons for commands

With the **Customise** dialog open, the button options on the shortcut menu for a selected option are used to configure the appearance of the button on the toolbar or menu. The **Delete** option is an alternative way of removing the button. The **Start Group** option is an alternative method of creating groups of buttons, when selected it will add a separator. It will be ticked on the menu if the button has a preceding separator to define the start of a group.
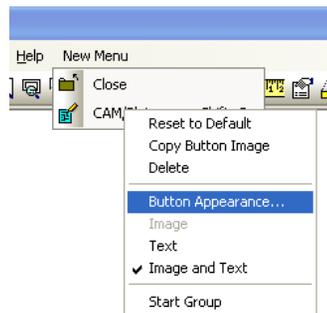
Once the **Button Appearance** option has been selected, you can add and edit the icon as required.

Menu Customisation

The **Menu** page on the **Options** dialog allows the menus for a different type of design to be displayed, allowing them to be customised. You can add and edit menus to contain your own selection of commands and options. Each menu design type can be selected for editing regardless of which application you are currently in.
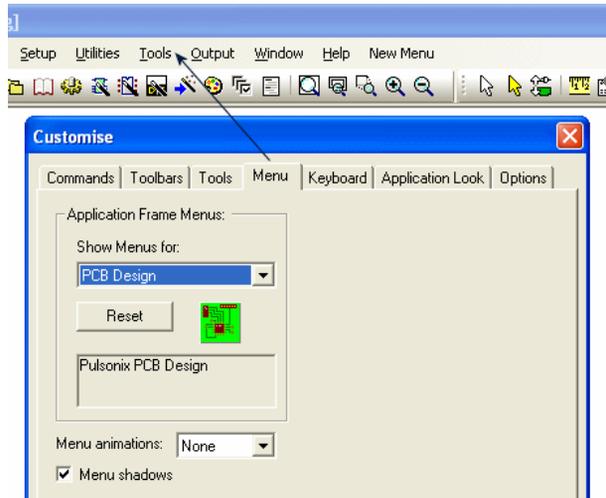
The **Reset** button will reset the selected menu back to its original default settings. Any user customisation will be lost.

The **Menu animations** list box specifies the type of animation effect to be employed when a menu is displayed, for example, the menu can be shown to slide out when selected.

The **Menu shadows** check box specifies whether or not a menu is displayed with a shadow.

Adding User Tools

You can use the **Tools** tab to add external tools, such as your text editor, to a **User Tools** sub-menu at the bottom of the **Tools** menu. Adding external tools allows you to easily launch other applications while working in the integrated Pulsonix environment. You can specify arguments and a working directory when launching the tool. User reports can also be run using this mechanism.
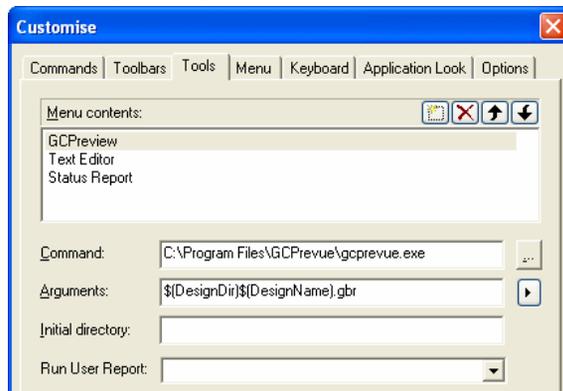
The small buttons across the top of the dialog adjacent to the **Menu contents:** text have the following functionality:
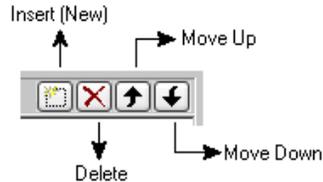
Use the **New** (**Insert**) button to add a new tool, this will add an empty entry at the end of the list. Edit this entry in the list to specify the name of the tool. This name will appear on the **User Tools** sub-menu on the **Tools** menu.

The order in the list is the order the tools will be presented on the **User Tools** sub-menu. Use the **Up** and **Down** buttons to re-order the list.

Use the **Delete** button to remove a tool from the list.

Select an entry in the list to change its tool details. Use the **Command** box to provide the name of the application to be run. Enter the path to the .exe, .com, .bat, .cmd, or other file that you intend to launch. Use the browse button (a box shown as …) to find the required file using a standard file browser.

Use the **Arguments** box to provide any command line arguments that you need to send to the program when launched. Use the right arrow > button to the right of the argument edit box to drop down a list of plug in arguments. The chosen plug-in argument will be added to the argument field at the current cursor position. This argument will be expanded when the tool is run to contain the relevant information. In the example above the command line is "MyGerberViewer $(DesignDirectory)\$(DesignName).gbr". When the tool is run the command line will be changed to include the current design. e.g. "MyGerberTool c:\myfolder\board1.gbr". Use the **Run Report** argument to place the name of the user report file , specified below, into the command arguments to be used to launch the tool.

Use the **Initial Directory** box to specify an optional working directory for the tool.

The **Run User Report** field allows you to select a user report from the drop-down list to run before the tool is launched. if you leave the **Command** field blank, only the user report is generated. This is a way of putting your user reports on the **User Tools** menu and assigning shortcut keys to them.

By using the **Run Report** plug in argument with your tool, you can write a user report to generate the format required by your tool and automatically generate the report prior to running the tool and pass its filename as an argument to it.

Personalised menus

An **Options** tab in the **Customise** dialog allows you to change options specific to menus and toolbars.
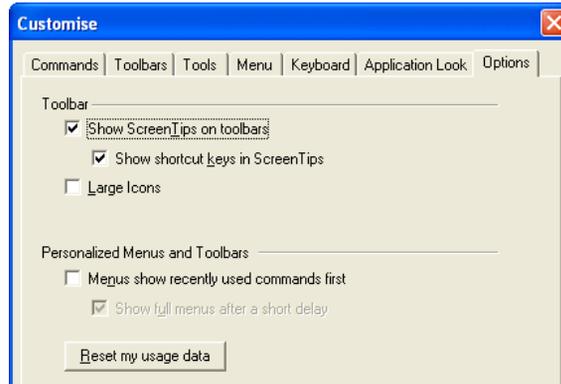
Show ScreenTips on toolbars specifies whether or not tooltips are to be shown when the mouse pointer is over toolbar buttons.

Show shortcut keys in ScreenTips on toolbars specifies whether or not the tooltips should include equivalent shortcut keys for toolbar buttons where defined.
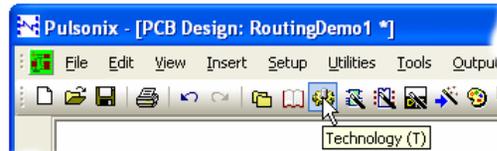
Large Icons enables toolbars to be displayed with the larger size icons.

The default setup will be to not show full menus (see below), and so the initial menus will have some of the more advanced functions missing. These can be accessed by pressing the last entry of the menu or by un-checking the **Show full menus after a short delay** switch.

The menus can be forced to be full size all the time by switching the check box **Menus show recently used commands first** off.

Use the **Show full menus after a short delay** option to automatically change to the full menu after a short wait.

Pulsonix now has a choice between **full** and **short** menus.
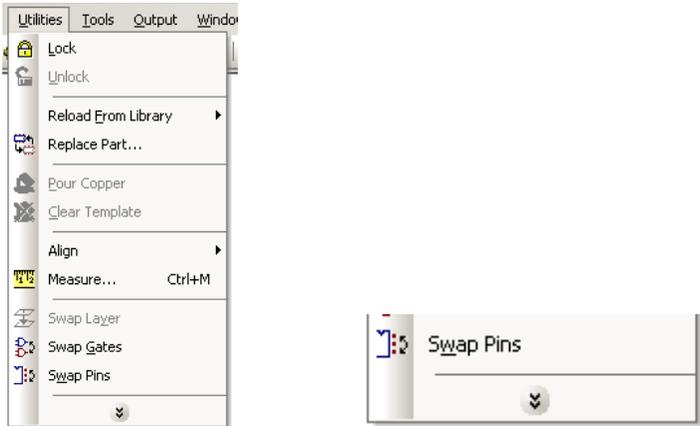
Short menus are chosen from the **Customise** dialog on the **Setup** menu. Use the check box, **Menus show recently used commands first** on the **Options** page.

A short menu just shows the most recently used commands from the full menu. If a command is not used for a long time it will eventually disappear from the short menu.

By clicking on the small circled arrows at the bottom of a short menu, the full menus will be revealed.

Shortcut Keys

The old **Shortcut Keys** option (in version 3.1) is now integrated into the **Customise** dialog, and is named **Keyboard**. This means that all interface customisation is located in one dialog.

The shortcut keys report can now be sorted **By Command** or **By Key**:
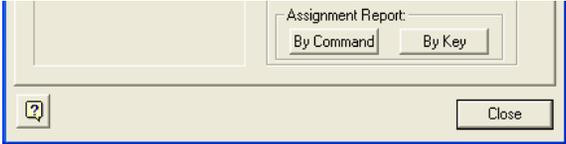
Standard Menu changes

There are minimal changes to the menus to declutter ones that may have become congested with the new options introduced.

Menu declutter

To declutter the menus, if only part of a **Component** or **Doc Symbol** is selected, the **Reload Component** and **Edit Symbol/Footprint in Library** options now appear on a sub-menu.
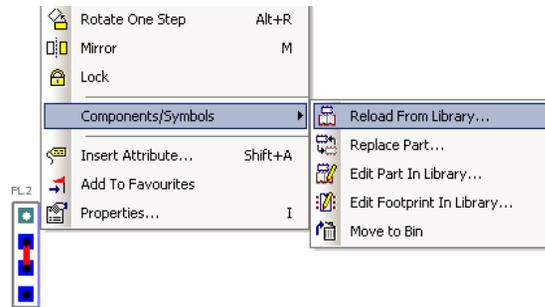
Rotate One Step now appears back on the shortcut menus if the rotation step is not set to 90 degrees.

The shortcut menus have been tidied up, so for example, **Add To Net** and **Remove From Net** have been added to the **Nets>>** section.

Within the **PCB design** application, **Optimise Nets**, **Unroute Nets** and **Add Teardrops** have been moved from the **Utilities** menu to the **Tools** menu.

In both the **Schematic** and **PCB** applications, **Rename** has been moved from the **Tools** menu to the **Utility** menu.

All dockable windows are now on a sub-menu named **Dockable Windows** on the **View** menu.

Shortcut Menu changes

Insert Spiral

Insert Spiral is a new option and is available on the **Insert Breakout** and **Insert Track** shortcut menus. This option is discussed later in the Spiral Tracks section in the PCB Design chapter.

Insert Rectangle

On the **Insert Rectangle** shortcut menu, it now says **Square** instead of **Free Movement** when in this mode.

Cross probe

With an item(s) selected, the **Cross Probe** option now appears listed on the shortcut menu.

Active Status Bar

The previous Status bar has now been modified so that it now becomes an active status bar. This means that by double-clicking on a pane within the status bar, relevant information can be modified. For example, clicking on the **Abs** word will toggle between **Absolute** and **Relative** modes. Other modes, such as **Grid:** will display the **Grids** dialog where a change can be made.

The Status bar items that may be modified are:

- Layer, Layer Span, Net, Style (text style, pad style, line style) also editable by clicking on Width, Size, Line Width, Side, Delta X, Delta Y, Angle and Radius (for arc).

- Units (mm/thou toggle),

- Units (double-click on coords when not editing/moving item),

- Typed coords (double-click on coords)

- Absolute/Relative coords (toggle),

- Current Grid

Find

Item Finder renamed

The **Item Finder** has been renamed to the **Find Bar**. This also uses the new style sliding when selected.

New Find Items
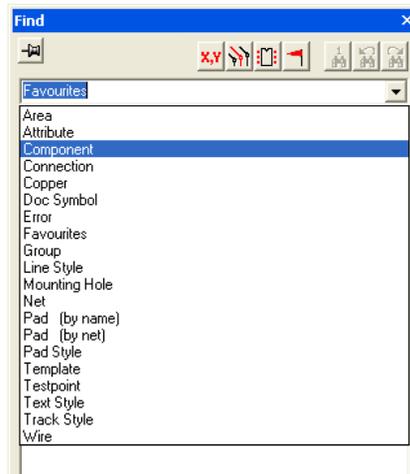
Find can now be used to locate **Named Areas, unnamed Areas** and **Named Busses**. Unnamed areas will only be found using the <Search entire design> switch but cannot be listed (because it isn't named). Use the first and next button on the top of the option to locate them.

You can now use Find to locate **Doc Symbols** (by Symbol Name and by using First/Next).

Find Net in Schematics now also finds closed busses that include the net name selected.

Find Vias is available in PCB either by net or all vias in design. Works in a PCB design and PCB Footprints. Using the shortcut menu, it allows you to select all vias in a net.

Favourites

Find Favourites has been added to the find bar. You can use Favourites for locating commonly used items in the design.

The finder bar maintains a list of *Favourite* items, this list is saved with the design so you can keep track of items you often want to return to. You can add items to this list in two ways. Firstly, you can explicitly add an item by selecting it and using the **Add To Favourites** command from the shortcut menu. Secondly, you can check the **Add Found Item to Favourites** option in the find

bar shortcut menu. This will automatically add any items found using the find bar to the favourites list. You can limit the maximum length of this list using the **Options** dialog and **Find** page, **Favourite List**.

You can remove items or clear the whole favourites list at any time using the commands available on the find bar shortcut menu.
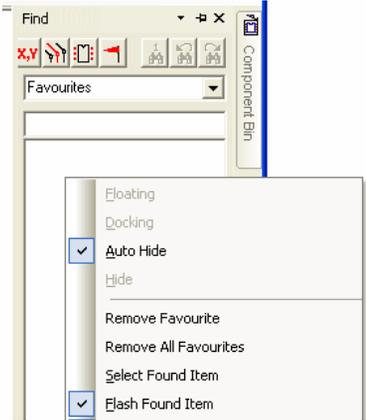
Refresh Option

New commands **Refresh Find Bar** and **Refresh Part Browser** on their bars shortcut menus.
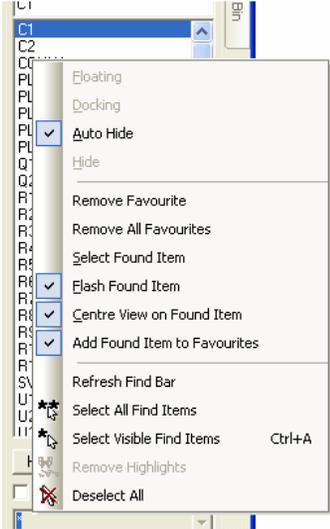
This option is needed because not all ways of introducing design items update the Find bar. This is also available in the Parts browser because it does not always know of new and changed Parts. You can assign these options to shortcut key using Customise.

New Find options

Find switches are available on the **Options** dialog and **Find** page.
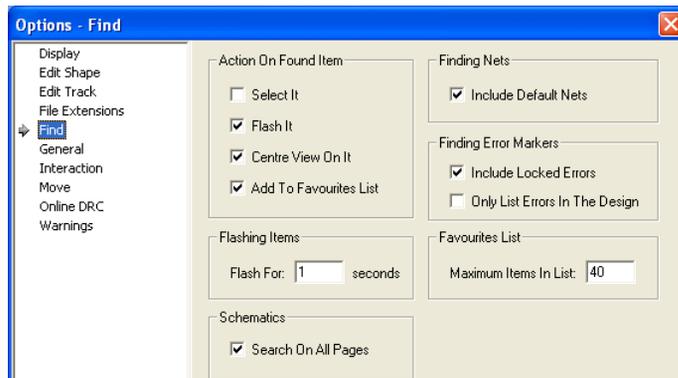
Other than some obvious actions on found items, one new choice is the ability to be able to **Flash** items that are found, and the length of time that the item is flashed.

You can also can use shortcut menu in the **Find** bar to enable flashing of each item as you find it. Each of these options can be changed on this menu, these change the settings in the **Options** dialog.
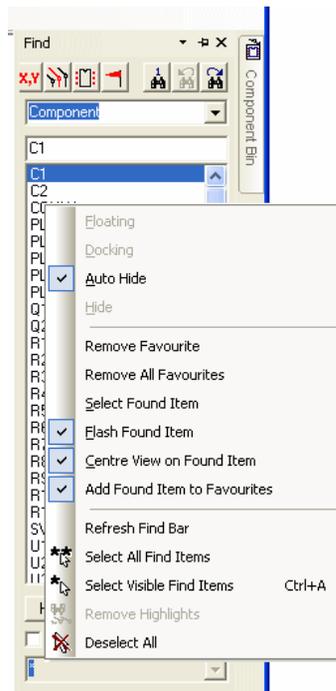
Note: Don't make the flash item time duration too long as you will not be able to do anything else while they are flashing.

Flash Find Items Commands

Three new functions have been added to help you find items in crowded designs when using the **Find** options.

Flash Errors, **Flash Highlights** and **Flash Selection**, these flash the appropriate items for the length of time specified in the **Find** options dialog.
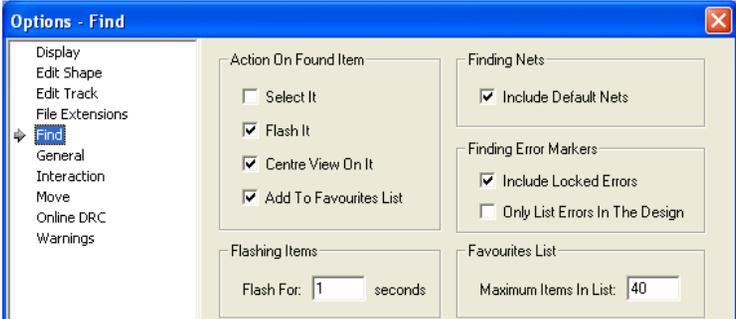
Report Maker Update

Report Maker Dialog changes

Colours in the command tree have been changed to reflect the different states: Comments are shown as green, Errors are shown as red, Disabled items are shown as grey and Control Commands are shown as blue.
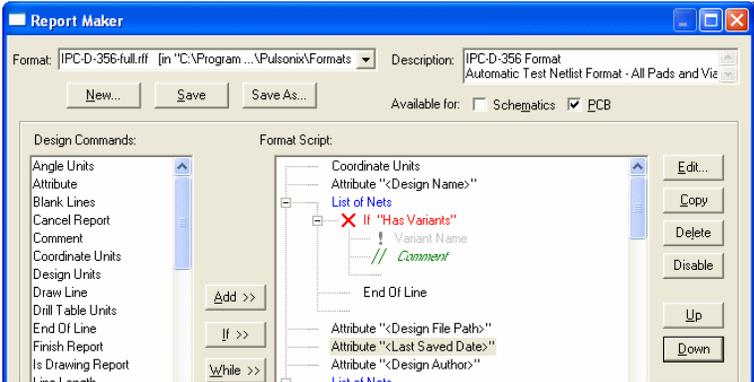
At the end of each command, a blank line is inserted to indicate the end of each level (as indicated below between the // Comment field and Else commands).

Drag and drop

The command structure in the **Format Script** window now allows you to move items about by dragging and dropping the command. This significantly speeds up the use of the command structure.

New and updated commands

The **Report Maker** option has been updated to include new list items.

List Of Layers and **List Of Layer Spans** have been added for use in manufacturing reports.

New fields have been added for Error Marker **Codes** and **Annotation**. The code has been removed from the end of the **Type** field. If your report uses the existing Type field, you will need to modify it to add the new Code field as well.

An **Area** field has been added for use with Pad Styles, Pads, Vias and Mounting Holes. Also added for these pad types is the **Set Layer** command. This has been added so that the pad area and total pad area can be calculated for paste mask pads but it can be used for other pad area calculations as well.

List of Drills has been added. This can be used as an alternative for creating a user defined drill table. Other commands used to support this feature are **Drill Table Units** to change the report units to those of the drill table, **Drill Table** with sub-commands **Step**, **Tolerance** and **Units Name**.

A new command of **Design Units** has been added to set the units back to the ones defined in the design.

A new command for **Drill Position** has been added for Pad, Mounting Hole and Vias.

A new command for testing **Is Complete** has been added for **PCB Nets**. This works the same way as the standard **Net Completion** report.

CSV Command

In the CSV command, you can now specify the separation **character**. Even though the option is called CSV, the output separation character will be the one defined in this dialog. By default the comma is used.
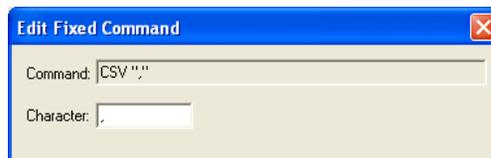
Angle & Coordinate Units

On the **Angle Units** and **Coordinate Units** dialog, you can now specify the **Decimal Point Character**. The default will be a full stop but it provides you the ability to change this for your own.
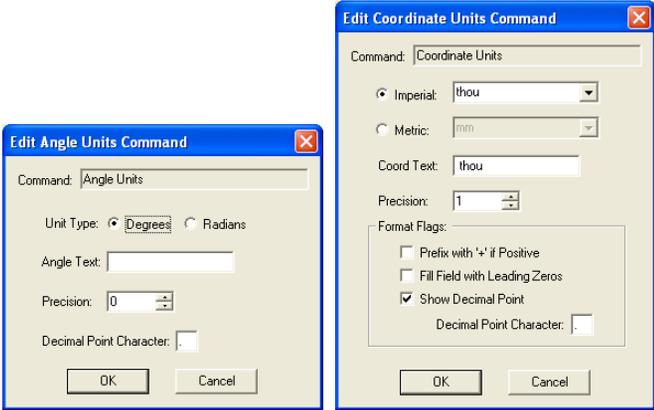
New commands for use with Embedded Components

New fields have been added to the **Report Maker** to support the new embedded Component functionality.

New fields for **Layer** - **Are Comps Allowed**, **Are Comps Mirrored**, **Is Essential Class** and **Associated To Layer** have been added.

New fields for **Components** - **Is Mirrored**, **Is Embedded Component** and **Layer** have been added.

New fields for **Vias** - **Component Name** has been added.

List of Vias is now available on **Components**.

Insert User Report

There is a new option on the **Insert** menu to allow you to add a copy of a user report into the design. The report script is held within the design so that the report can always be updated without the actual format file having to be located.
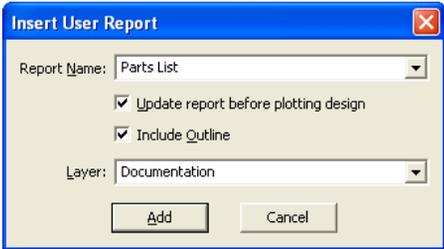
User reports added to the design use the same defaults as the **Drill table**, both now use a new **Default** in the **Design Settings** dialog – **Report Symbol**. This uses layer, line style and text style for the defaults.

To facilitate the insertion and management of user reports some new options have been added:

Update All User Reports – located on the **Utility** menu, this option allows you update all the user reports when the design has changed. A similar command, **Update User Report** is available on the shortcut menu for a selected user report in the design. These commands are automatically run when the **CAM Plot** option is entered as well.

Reload User Report – this is used to load a user report format file that has been modified. Care must be taken when using the **Insert User Report** feature, reports that are added to the design are added for the design in its current state and for the format file as it is. Both of these items are not updated automatically.

Nets Reports

The **Nets Report** for both Schematic and PCB now includes the number of pins in each net.

For the SCM Nets Report, the report indicates if a net is in a closed bus, this is indicated as **In Closed Bus**.

Select Mode

Path Select

There is a new option on the shortcut menu in select mode which allows you to select all segments in a shape or a track path between two selected points. If the selected points are in the middle of segments, they are split.
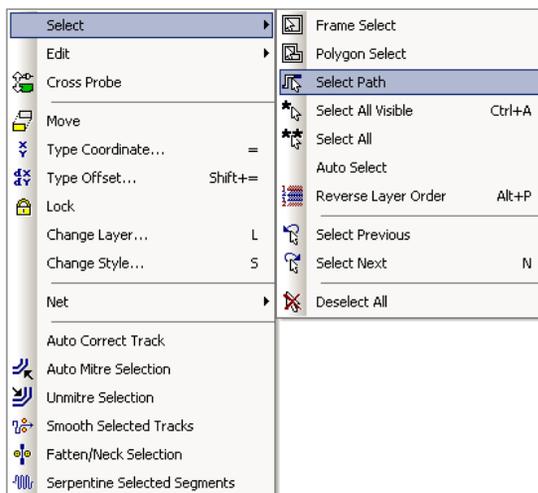
The path select mode can also be accessed by simply dragging along the line or track segment (provided you have this mode enabled in the **Options** dialog – see below).

The process for inserting segments is to drag along a segment which selects part of the segment, and then drag the selected segment to create a 'top hat' shape as shown below.

If you wish to select more of the shape, then simply drag further along the shape. Once the drag has been initiated and the shape segment picked up, the mouse movement will mimic the shape of the segment without having to follow it accurately.

If you don't want to drag to select the path, the **Select Path** option can also be used from the shortcut menu.

New Options selection

Within the **Options** dialog you can select the **Interaction** tab and enable or disable the ability to **drag along shapes**. You may wish to deselect this if you find that you are regularly dragging items which you do not intend to.
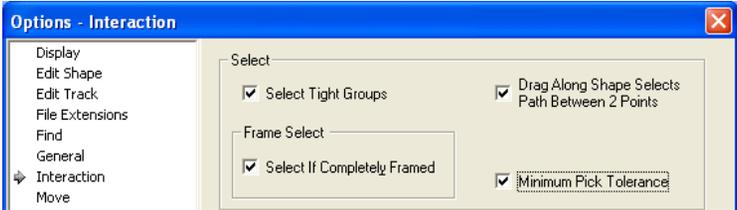
Possible usage

Uses of this could include:

- Dragging segments in a track or line shape to insert a 'top hat'

- Unrouting a track between two points

- Necking between two points (using the Select Path command followed by <S> Style to change the segment thickness)

- Changes layer between two points

Type Coordinate / Type Offset

The two options, **Type Coordinate** and **Type Offset**, are now available on the selected items from the shortcut menu or by using a shortcut key when in select mode. These options can be used to reposition selected items directly without having to enter a move operation.
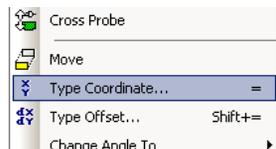
Free Angle Segments

The **Free Angle Adjoining Segments** is now available on **Move**. The switch to enable this option is on the Options dialog and Interaction for both Schematic and PCB applications.

Component Bin Changes

From within the **Component Bin** you can now make multiple selections of **Components**. Multiple selections can be made using the **Shift** and **Ctrl** key combinations. When multiple selections are made in the bin, no Component preview picture is drawn.

From the design, if you drag multiple items into the bin then these will all be automatically selected in the bin.

If using **Move From Bin** with the Component Bin containing multiple selected Components, each of the selected Components will be available sequentially as the current one is released. If nothing is selected in the bin or a single item selected, you get **all** of them sequentially on the end of your cursor.

For multiple selected items in the bin, if you drag them out to the design they will all be on the end of your cursor. The actual item dragged will be on the end of the cursor and the others will be positioned along side it.

With the changes made, if you use the **Find Bar** and use **Select All Find Items** using any selections or criteria in the find window, items in the design and in the Component Bin are selected. Previously, items in the Bin wouldn't have been selected.

In **Cross Probe** mode, items selected in the Schematic design can now be probed even if in the Component Bin. This should make item selection and placement much easier.

TrueType Font Preview in Selection

Text in the **Change Style** drop down list is drawn in the TrueType font selected to preview it.
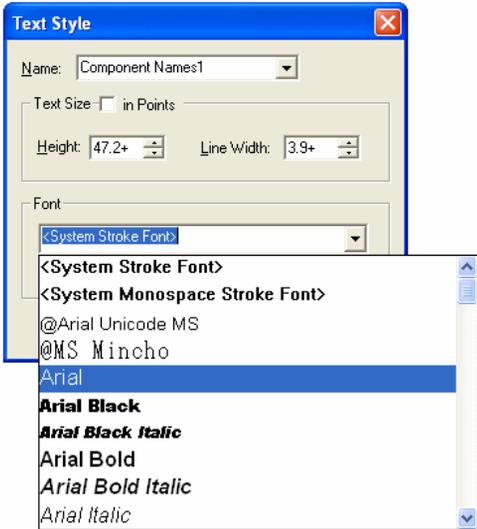
Simulated True Type Fonts (True Scale)

The **Simulated True Type Fonts (True Scale)** switch specifies how True Type text is to be drawn. This mode is setup in the **Options** dialog under **Display** and **Simulated True Type Fonts (True Scale)**.
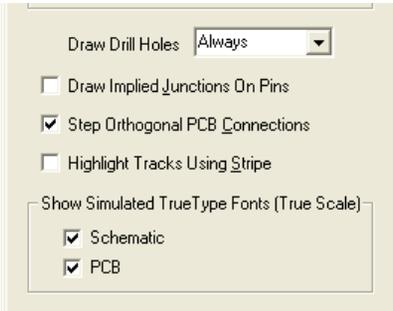
The default system drawing of fonts is often not very accurate and can result in text strings which are drawn much wider at some scales. Checking this option ensures that the true type text is drawn and plotted at a consistent width. This may result in loss of detail particularly on intricate fonts. You can use different options for Schematics and PCB because you may want better detailed fonts for Schematics, but more accurately scaled fonts for PCB where the relative

positioning of items is more important. If you output to Gerber or Pen Plot, then the simulated true type will be used to produce the shapes required to plot the text.

Control of Display of Part Name from Part

You can now force the display of **Part Name**, **Component Name**, **Pin Names** and **Logic Names** from the **Part**.

There is a new command in the **Part editor** on the **Edit** menu, **Force Attribute Display** which provides you with a dialog to force the display of the these values.
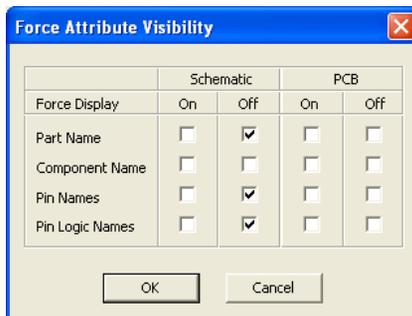
Replace Part

There is a new option on the **Utilities** menu to replace a Part used by selected Components. This is also available on the shortcut menu when the selection contains Components.

Features

- You can replace a Part on just the selected Component(s).

- You can replace a Part on all similar Components in the design, page or block.

- You can change the footprint to an alternative on Components using the chosen Part.

You can get a report prior to performing the replace as verification.

Repeat Last Command

A new command to repeat the last command run has been added (**Repeat Last Command**). This is not in the user interface but is available as a default shortcut key assigned to <**F4**>. It is also available in the **Run Commands** dialog. For example, this is useful when having to re-enter an option dialog if you made a mistake, e.g. entering the colours dialog again.

Groups

Set Group Master Item

You can now select a grouped item in a loose group and make it the master item. This means that moving the master item will cause all the other items to be selected and move with it (similar to a tight group). However, the other items in the group can be moved independently.

Add To Group

You can now select multiple items and use **Add To Group** from the shortcut menu to add them all to a chosen group. This only previously worked for single items.

You can also use this option in latch mode operation (select the option first then select the items to add to the group).

Design Item Tooltips

Tooltips are available when hovering over a design item to display information about the item.
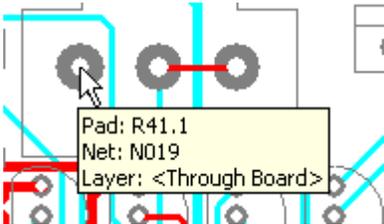
This is an optional feature and is controlled from the **Interaction** page of the **Options** dialog using the switches provided.
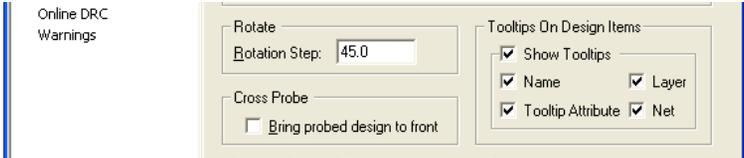
Using the options available, you can check boxes to say what to include in the tooltip. There are separate options for Schematic and PCB applications.

One option on this dialog is to include any **attribute** that is marked as a tooltip - **Tooltip Attribute**. Use the **Attribute Names** page on the **Technology** dialog to mark attribute names as tooltips (**Use as Tooltips**).
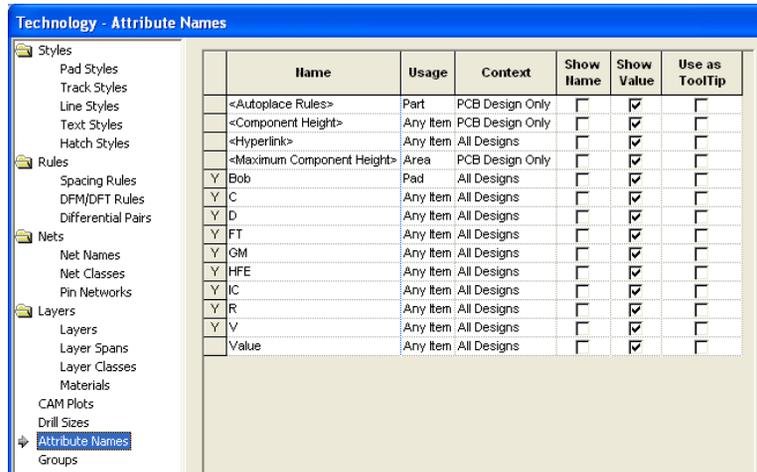
There is a new command, **Design Tooltips On/Off** to use for toggling tooltips on and off, this can be assigned to a shortcut key.

Change Styles

In select mode, you can now choose to apply track or via style changes to every track/via in the net. There is a new switch in the **Change Style** dialog called **Apply to Whole Net**.
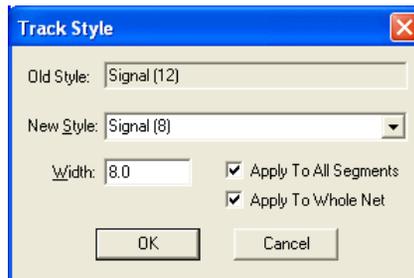
Library Creation from Part Information

There is a new option **Make Libraries** to copy existing Schematic Symbols, PCB Footprints and Part libraries from the selected Part definitions in the **Parts** page of the **Library Manager**.
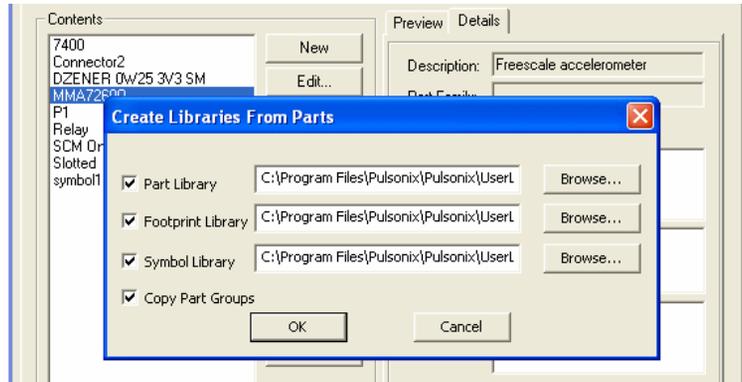
You can select an existing Part or Parts in the Part library listing and then using the **Make Libraries** option, can copy those Parts to a new Parts library (using the named list) along with the corresponding Schematic Symbols and PCB footprints.

Library Contents Report

New Report Options

When running the new **Library Report** option from the **Library Manager**, it allows you to select from **List Contents of Libraries** to give you a list of the selected Library items (Symbols, Footprints or Parts), or to report a list of selected Symbols/Footprint items that are used in Parts libraries (use the **Find Parts Using Selected Items** radio button).
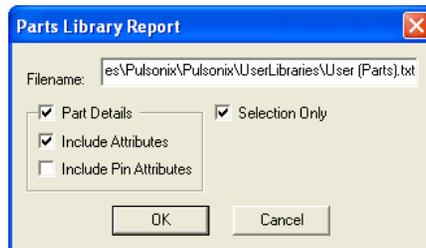
Last Saved

It will also now report the **Last Saved** time and date of the selected library or all libraries (depending on the selection).

Single Line Attribute Value Editing

When editing a **Value Attribute**, the value 3.3K on a resistor for example, you can now type in the new value, 3.6K, then press Enter <CR> for the value to be accepted and the dialog closes as though you had typed it and pressed OK. This is more widely used on generic Components in the Parts library where the value attribute changes.

Properties: Part Attribute - Attribute

Attribute | Text Style | Component | Comp Attributes

Name: D

Usage: Any Item

Value: Height

If you require a multi-line attribute then click the drop down button on the right of the list and type in the text into the extended text box.

Usage: Any Item

Value: Height

Note: This change also applies to normal Text.

Picking Tolerance Value

You can now set the minimum picking tolerance, this makes picking much easier in dense areas of a design. With this switch set, you will have to pick much closer and more accurately to select items in the design.

This is set in the **Options** dialog and on the **Interaction** tab.

Options - Interaction

Display
Edit Shape
Edit Track
File Extensions
Find
General
→ Interaction
Move

Select

☑ Select Tight Groups

☑ Drag Along Shape Selects Path Between 2 Points

Frame Select

☑ Select If Completely Framed

☑ Minimum Pick Tolerance
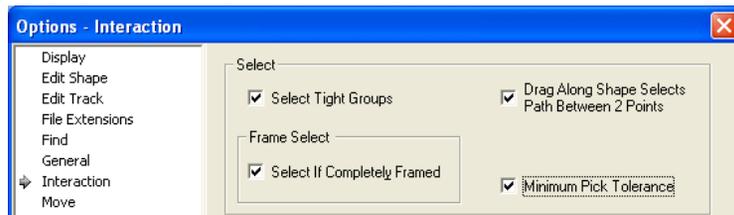
Hyperlinks

Some small changes have been made to the existing hyperlinks feature to make them more usable:

Double clicking on a hyperlink attribute now launches the browser to follow the link.

If a Part has the <hyperlink> attribute, the shortcut menu contains the **Execute Hyperlink** option.

Integra Design and Library Import

Using the **Open** or **Data Transfer Wizard** options, you can now import the Mentor® Integra format files. When you open a .txf file, this will open and import both the Schematic and PCB design files.

Using the same files, you can also import .txf files into the library manager to create Symbols, Footprints and Parts.

Insert Circle & Insert Rectangle Changes

The existing **Insert Circle** and **Insert Rectangle** options have been enhanced with additional functionality. When adding the circle or rectangle to the design, you can select the **Define From Centre** option from the shortcut menu. This will 'grow' the shape being dragged outwards from the centre rather than from the start point edge. The **Define From Centre** option toggles between this and the normal mode.
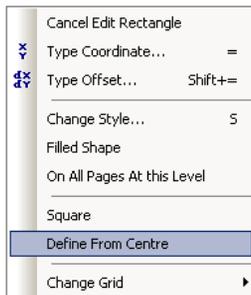
Set System/Relative Origin

New options from the shortcut menu allow you to set the **System** or **Relative** origins on the selected item. The **Set Symbol** and **Set Relative Origin** commands are located on the **Select>** sub-menu for a selected item.

Within the footprint editor these appear on the main shortcut menu but Set System Coordinates is replaced with **Set Symbol Origin**.
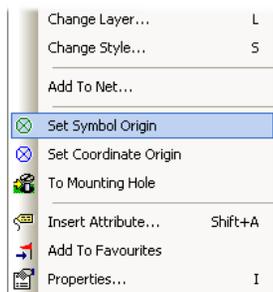
These are also available as commands to assign to shortcut keys:

Set Coordinate Origin

Set Relative Origin

Switch Between SCM and PCB Designs

A new command allows you to switch between a Schematic and PCB design, opening the design if necessary. The command is called **Switch to Schematic/PCB** and can be assigned to a shortcut key.

Chapter 3. Schematic Options

Bus Changes

Closed busses can now be named if required. All busses that have the same name will have the same allowed net name set. Busses can be named through the **Bus Properties** dialog.
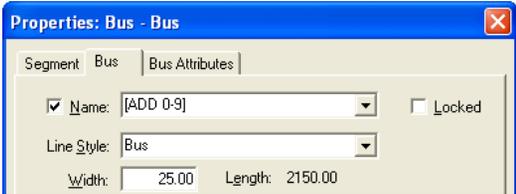
When adding a name in the Properties dialog, if the bus is an unconnected open bus then you can choose from a list of closed bus names.

If a bus is open and unconnected, a new option will allow you to select it and from the shortcut menu, select **Change Bus**. The selected bus will be changed to match a closed bus, taking it's name and net names.

The **Change Bus** option is also available in the **Insert Bus** and **Edit Bus** operations.

One other small change to busses, you can now use the **Find** bar to locate busses within a design. Where a closed bus doesn't have bus name, the bus will displayed using the net name range. You can use the **Find Bus** option to cycle through any separate sections of the same bus in a multi-page design.

Net Page Attribute

A new default attribute **<Net Pages>** was added to the list of available attributes in Pulsonix during V3.1. This displays the pages that the net appears on and has **Net Pages Attributes** configuration switches in the **Design Settings** option naming the page (see below).

When you use the attribute <**Net Pages**> anywhere on a net (using the **Insert Attribute** option), it will show a list of the pages that net appears on.

You can configure how this attribute will be presented using the **Design Settings** option and **Naming** page. You can define a **Prefix** and the **Separator** between each page. You can choose if the current page is to be included in the list using the **Show Current Page** option. The page can be shown as a name or a sequence number using the **Show Page Number** option. For example 'Pages: Page1 + Page2 + Page4'.
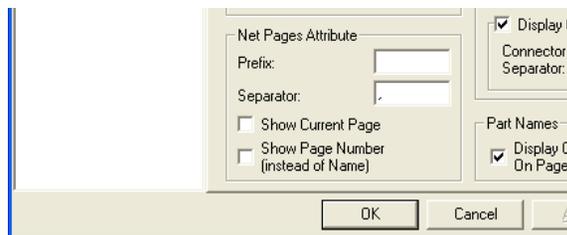
Pulsonix version 4.0 allows you to check that any net which has the <Net Pages> attribute displayed, must have it displayed on every page that the net appears on. The check is called **Unlabeled Net Pages**.

Display of Part Names on each Gate

On the **Design Settings** option, there is a new switch to allow **Part Names** to be displayed on each gate of a page or just on one of the gates on each page that the Part is on. When a Part name is displayed on one gate only, the first gate used in the design on that page will display the gate name. If the first gate is in the Component bin, then the next available one is used.

ERC Changes

Pin type Warnings

You can now add an error marker to the design to show **Pin Type warnings** using the **ERC** option. **Pin Type errors** are already flagged but this allows the warning status to be flagged as well.

Default Block Port Symbols

A new default on the **Design Settings**, **Defaults** and **Block** page has been added to define the Symbol for a bi-directional port using the bi-directional port symbol entry. The **Add Block Port** dialog now uses the defaults for port symbols when the pin type changes.
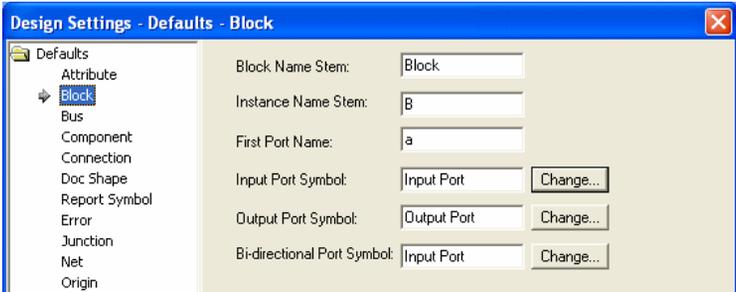
Part Editor - Default Logic Names

From within the Schematic Symbol editor, you can now specify **Pin Logic Names** for the schematic symbol. Previously, only the attribute place holder could be defined. When defined, these names are the defaults when the symbol is used in the Part editor. The **Pin Logic Names** can be edited from the **Properties** dialog of the selected **Logic Name**.

Apply Symbol Logic

From within the Part Editor, there is a command **Apply Symbol Logic Names** available on the shortcut menu on the **Logic Names** column which resets the logic names to those specified in the schematic symbol.

Pin Networks

A Pin Network allows you to define additional Components to be connected to a pin. For example, you can automatically add bypass capacitors (like in the example below) or terminating resistors that are associated with another Part.

There are two main processes to make the association:

- For the Part that requires the association - add the **Pin Network** name on the **Pin** page of the **Part** in the **Part editor**.

- In the **design** and in the **Technology** file, define a **pin network name** and add the **Part Name** to be used as the secondary Part. The Terminating Net name can also be defined if required.

Terminology Used

We would consider the main Part (the Part to be decoupled) to be the 'host' and the Part used for decoupling, to be the 'secondary' Part. This is how they are referred to in this text and in the online help pages.

Reasons for using pin networks

- To ensure you have enough bypass capacitors defined in your Schematic.

- To aid checking the PCB to ensure that these capacitors are close enough to the 'host'.

- To aid laying out the PCB to keep associated Components close to the 'host'.

Creating Pin Networks

▶ **To create a pin network**

1. You must add the **Pin Network** name on the pins of the 'host' **Part** (the main Part in the **Part Editor** that will use the secondary Part).

2. Edit the host Part and select the **Pins** page.

Footprint Pin	Pin Name	Net Name	Pin Type	Pin Network
1	1		<Undefined>	<None>
2	2		<Undefined>	<None>
3	3		<Undefined>	<None>
4	4		<Undefined>	<None>
5	5		<Undefined>	<None>
6	6		<Undefined>	<None>
7	7	GND	Ground	<None>
8	8		<Undefined>	<None>
9	9		<Undefined>	<None>
10	10		<Undefined>	<None>
11	11		<Undefined>	<None>
12	12		<Undefined>	<None>
13	13		<Undefined>	<None>
14	14	VCC	Power	PN2_VCC

3. The **Pin Network name** is just a name that can be recognised when doing the next process. It is used as a reference when this Part is used in a design.

4. Save the Part. This is all that is required in the Part editor.

5. In the Schematic design, define the **Name** of the **Pin Network Part** in the **Technology** file using the new **Pin Networks** tab.
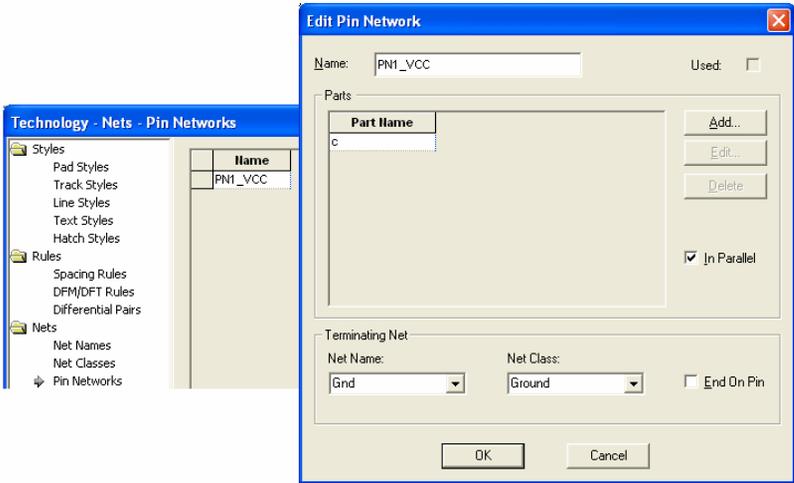
6. When this is done in the Schematic it is automatically transferred to the PCB design during Translate to PCB.

7. The name of the pin network should match the name used on the Part pin, this is essential.

8. Using the **New** or **Edit Pin Network** button, you can define the net which is connected to the other end of the network, either by **name**, by **net class**, or to be prompted for a pin to connect to (**End On Pin**).
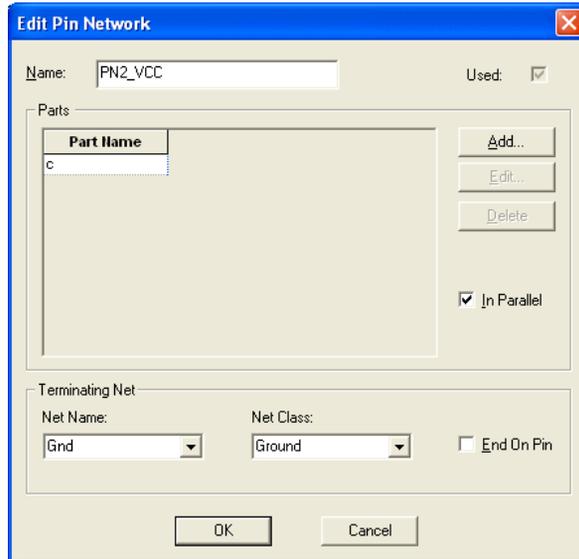
9. When a Component is added to a design, the pin networks are matched with those defined in the Part pins. Any which have not already been defined can be defined as the Component is added.

10. The Components in the pin network are added to the component bin in a special section for connected Components (in the previous release, all Components in the bin were not connected).
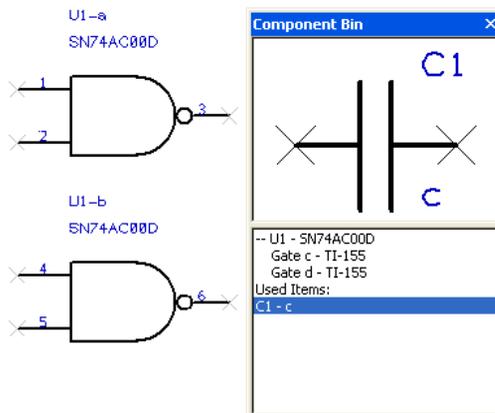
11. You can drag these Components from the bin and they will , provided their nets are in scope according to the parameters set. You cannot place connected Components into the bin.

When the design is transferred to PCB, the pin network Components are also transferred and set up in a loose group with the host Component set as the '**master**' item. This way, moving the host moves the associated Component.

PCB Design Editor

In the **PCB design**, the **Autoplace** option will attempt to place the pin network Components as close as possible to their **master** Component.

The DRC option will report any pin network connection length fails.

When routing a Pin Network connection with a **length restriction**, the **Interactive Net Length Indicators** are shown (*see later in the PCB Design chapter*).

Highlight Bus Net Pins

A new command **Hightlight Bus Net Pins** has been added to highlight the pins on the nets defined on the selected bus. This is available from the shortcut menu of a selected bus.
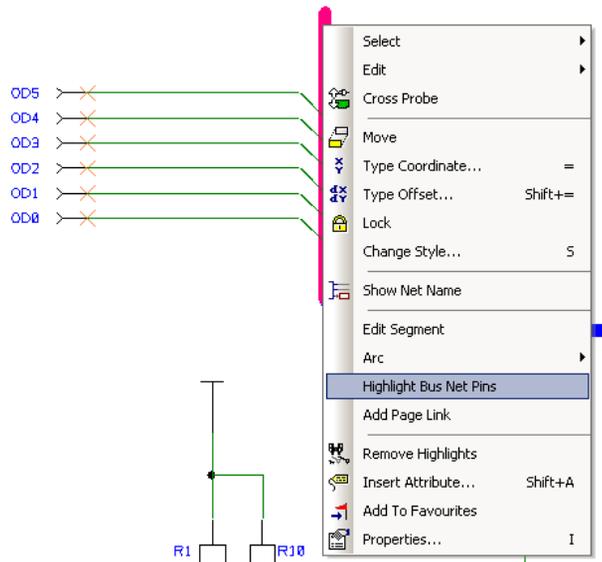
This is particularly useful after using the reverse engineering tool. (See next).

To remove the highlight, click in free space and use the **Remove Highlight** option from the shortcut menu.

Reverse Engineer

Within a PCB design, it is now possible to take your PCB and reverse engineer it back to create a new Schematic design.
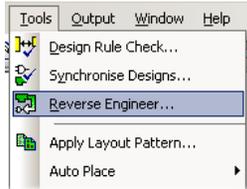
The Reverse Engineer dialog allows you various options and selections to be made when the option is run.
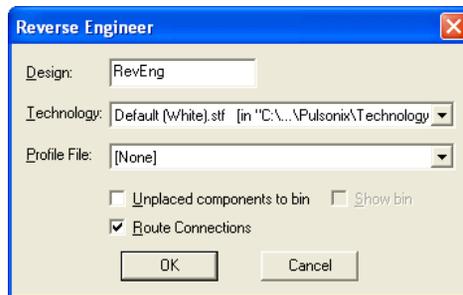
Components can be added to the Component Bin in the Schematic design. The bin is divided between unused and used gates, you should place all the used gates onto the Schematic pages to complete the design.
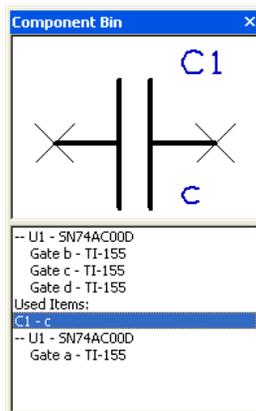
Components in the PCB design that are flagged as PCB Only Parts are not reverse engineered. If you wish to reverse engineer them they need to be edited in the Part editor and Schematic symbols added, then use the Reload option to reload them updated Parts to the design. After this you can use the Reverse Engineer successfully.

For Components that don't have a Parts library entry (because they came from an external source for example), these Components will be reverse engineered but the Schematic Symbols will be automatically created for you. The shapes of these Symbols will be generic boxes and not the real shapes.

Schematic Routing

When the **Unplaced Components to bin** check box is not selected, Components are placed in the design using their relative PCB design positions and using suitable placement spacing.

Having this box unchecked enables you to also select the **Route Connections** check box. Once the design has been placed in the Schematic the connections can be automatically routed using the Schematic connection router.

Sketch Connection

The Sketch connection path mode has been introduced into the Schematic design editor. This allows you to interactively make a connection between two points using a basic 'sketching' mode, or by using a point-to-point mode.

Using the **Sketch Connection** mode from the **Insert** menu, you can add connections to the design. Once in this mode, this is verified by the display of a modal cursor, a cursor which shows a mode.
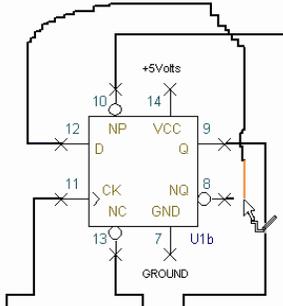
This picture shows the approximate path that you 'sketch' in. The modal cursor is also shown.
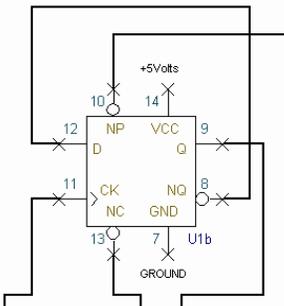
This picture shows the final result after completion and tidy-up.

Immediately after using sketch mode, the tidy routine will snap the connection onto the grid and remove all the rough edges for you. Once added, the connection path can be tidied up using normal connection editing options.

Shortcut menu options

While using the sketch mode, you can use options form the shortcut menu.

Point to Point mode

On the shortcut menu, you can select **Sketch Track Point To Point** mode. This allows you to point and click to add a connection path. After the mouse click, the path will be tidied back to the last click, this gives you a ;guided' routing path which is automatically tidied as it goes along. This may be a preferred mode of operation to the normal sketch mode.

Connection Options

The **Options** dialog and **Interaction** page now allows you to use drag or double click to add a connection from a pin on a net with no connection. This is particularly useful for reverse engineered designs. The default is drag to sketch and double click to use normal add connection. The default sketch mode is point to point.

OrCAD Export

From within the Pulsonix Schematic environment, you can export an OrCAD
EDIF file containing the whole design in graphical format.

Using the **Save As** option on the File menu, select the Orcad EDIF Schematic
Design (.edf) option from the drop down list , the design is output in EDIF V9.2
format. This can be read into V9.2 and later versions of OrCAD Capture.
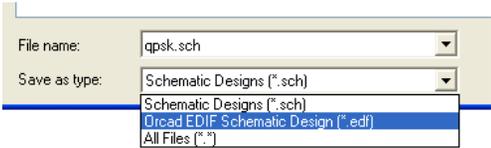
All trademarks acknowledged to their rightful owners.

Show Net Name

For a selected Net in Schematics, the shortcut menu now says **Show Net Name**
or **Hide Net Name** depending if one is displayed on selected item.
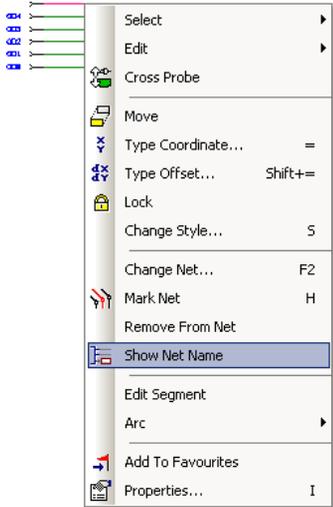
There is also an option on the Schematic **Utility** menu – **Show Net Name**. This
option can be latched down, use it from the **Utility** menu with nothing selected, a
modal cursor is displayed. You can then select items interactively to show or
hide their net name.

Automatically Display Required Net Names

Whilst using this latched mode, from the shortcut menu item you can use **Apply
To All** to ensure that a net name is displayed on each subnet of a user named net.
The net names newly displayed are highlighted to make it easier for you to find
them to reposition.

Cross Probe

You can now select **Block Instance Symbols** in a **Schematic design** to select all the Components in the PCB Design that appear in that block.

Chapter 4. PCB Options

Footprint Creation

Component Vias

You can now add **Component Vias** to footprints. These have been introduced to facilitate the embedded Component technology (see later in the Embedded Components chapter). **Component Vias** can be added to any normal footprint as well.

Once added **Properties** of the **Component Via** will allow you to set a flag for **No Spacing Check Within Footprint** so that two pads on different nets can be joined together (for an embedded resistor for example) through a via without a DRC check rule violation being displayed.
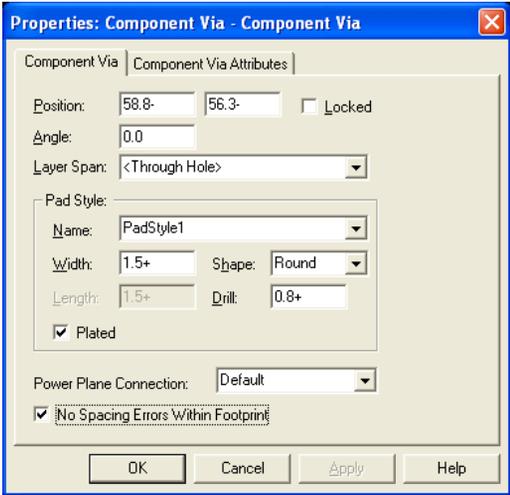
Footprint Copper

Copper within the footprint editor also has the **No Spacing Errors Within Footprint** check box available.

Areas

The **Properties** of **Areas** have been enhanced to add new options. These allow Copper Keep Outs, Drill Keep Out, Power Plane Avoid, Set All Keep In and Board Cutouts.

Named Areas

You can **Name** an Area and select whether the name is displayed on the design. Using the **Find Bar**, you can find a named area. Named areas can also be chosen when using the **CAM Plot** option to only plot that area.

It can also be used in **Autoplace** to place selected Components in the Named Area.

Keep Out Types

Areas can be created so that they avoid power planes, thus creating gaps in the plane (using **Power Plane Avoid**).
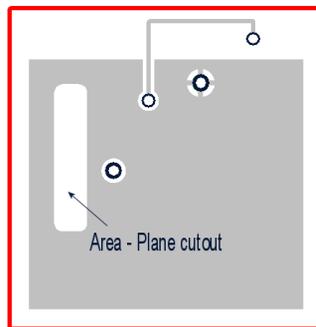
This is used where the **layer** bias is defined as **Powerplane** in the **Layers** dialog. These areas can be defined in a design, in a Footprint and as <Through Board> (for all layers). **Copper Pour Keep Out** is still a separate area type but both can be specified for an area.
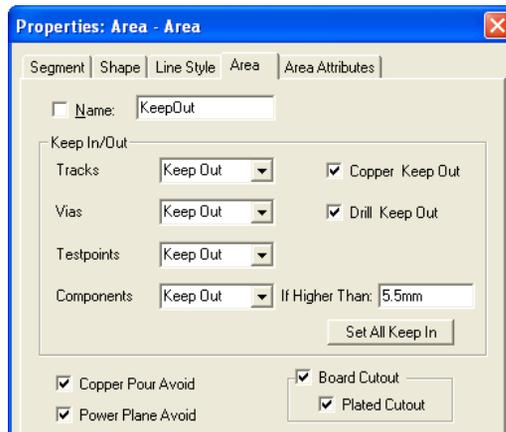
Copper and **Drill Keep-out** areas have been added. **Copper** includes pads and text but not vias, component vias or tracks which have their own area Keep-out type. **Drill Keep-out** areas include all drilled items but only the actual drill hole is checked, not the surrounding pad. These area types are particularly useful

when etched resistors are defined so that holes and copper are prevented from breaking the resistive material.

A single button has been added in **Properties** to allow you to set all area types to **Set All Keep Out** or **Set All Keep In** on this dialog.

Board Cutout Areas

A new area type of **Board Cutout** has been added. This can be used to add a separate cutout to a board. This will be particularly useful if the cutout in a board needs to be processed in a different manner, such as plated through or punched/routed out. This area type is available in **Footprints** so that **slots** can be defined for Components. This area type can only exist on the <Through Board> layer. A Board Cutout can be defined as **Plated** or left unchecked, as not plated.

Board Cutouts Areas can be made plated and/or unplated by using the **Layer Class** option within the **Technology** dialog. These can be used to create slots in Components. These can be selectively displayed as well through this dialog.
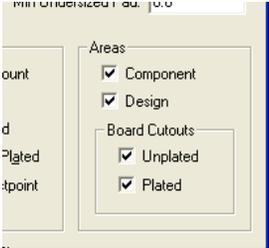
DFM Rule Changes

Minimum number of spokes

It is now possible to specify the minimum number of spokes that are used for a thermal pad connection on a full powerplane (in addition to poured copper).
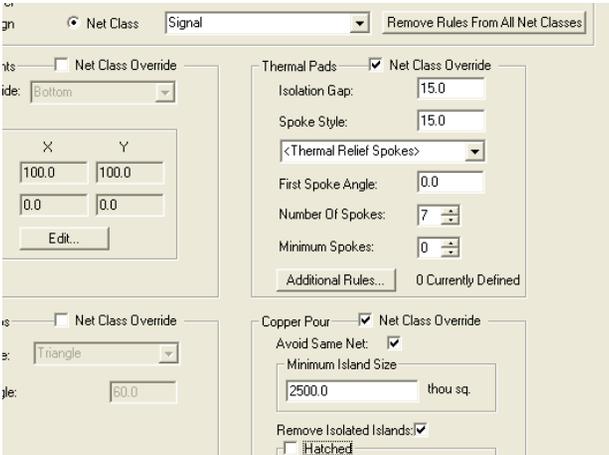
This provides the program with an opportunity to add an exact number of spokes but to also have a tolerance if it cannot add the specified amount.

Thermal pad rules

Thermal pad rules can now be defined for a sub-net of a net class. This can be defined by using an Attribute (a user defined attribute) or by using one of the **Additional Rules** for Vias, Through Hole, and Surface Mount pads.
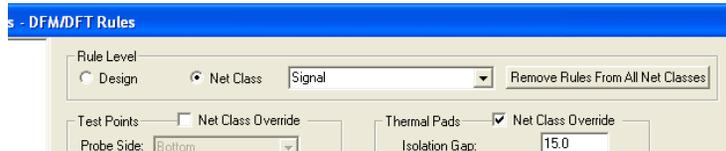
In order to be able to specify a **Thermal Pad Rule**, you must define a **Rule Level Net Class** rule and use the **Net Class Override** check box for the selected Net Class. Once the Net Class Override check box has been selected, you can then use the **Additional Rules** button to add the rules.
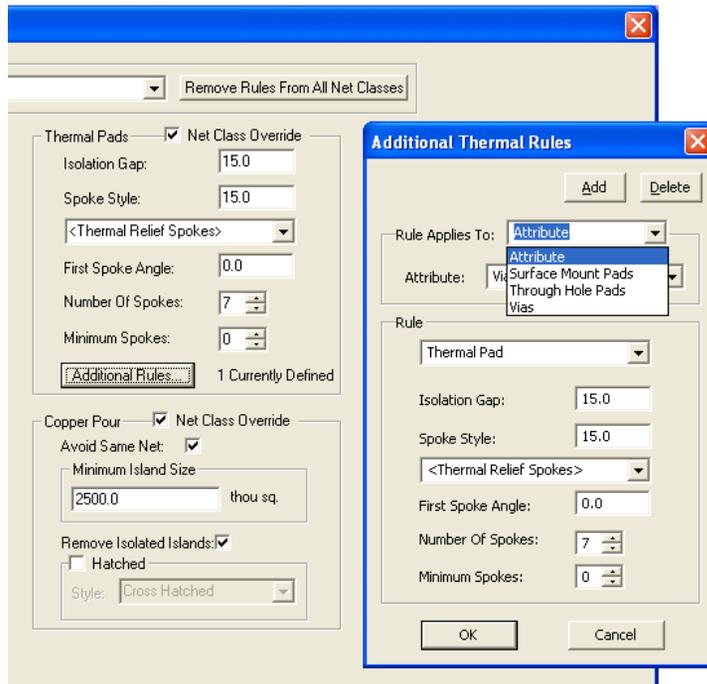
When the **Net Class** is used, any rule defined will be applied to it. This can either be on selected pads or vias on a net where an attribute has been added to apply the rule, or more generically using the rule types (Surface Mount Pads, Through Hole or Vias). Combinations of these types can be used on a net class.

If the rule applies to an attribute, you must select that attribute from the list of attributes that exist in the design. An attribute can be used to define a specific sub-set that requires an additional thermal value.
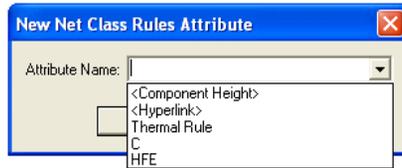
Once a rule has been selected from the list or the attribute selected, use the **Add** button to apply the rules. Clicking the **Add** button will allow you to define the parameters of the rule (these are greyed out before the rule is defined). Create the parameters for the rule and click the **OK** box to save it.

The DFM/DFT Rules dialog will show you that a rule has been defined for any selected net class with the **X Currently Defined** words.

Design Rules Check

New rules have been created to check the following:

Spacing – **Components** (to Board), this checks for Component, Via or Testpoints being completely outside the board.

Manufacturing – **Plane Thermal Pad**, this checks that thermal pads in a power plane have the minimum number of spokes connected to the plane.

Manufacturing – **Component Name**, this checks that the component Name is not closer to another Component than its owner.

Manufacturing – **Minimum Land Size**, this checks that a pad has the correct proportion of pad land to drill hole ratio. This is defined in the **Technology** dialog under **Spacing** Rules.
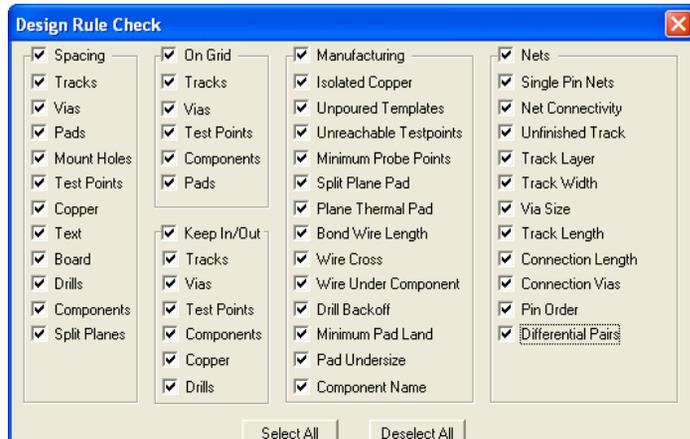
A new check to check that Vias or Testpoints that are completely outside a Board Keep-In/Out area are marked as in error.

In PCB Footprints, a new check will check that the Symbol Origin is on the same grid as the pads. (So the pads stay on the grid when the Component is moved).

DRC changes

You can now DRC items on the same net. This rule is set on the **Technology** dialog under **Spacings** and **Net Class**. This will flag items not explicitly connected which are too close. You must set the **Rule Level** to **Net Class** for this rule to be available.

Only Tracks, Pads, Vias etc. are checked (copper is not checked). Tracks must connect directly to a pad (no corners inside the pad).

You can specify the minimum pad undersize in the **Layer Class** dialog (for example on a Paste Mask layer) using the **Min Undersized Pad** entry. This can be checked with the **DRC** option in the **Manufacturing** checks using **Pad Undersize**.
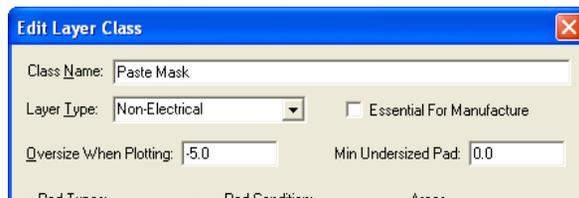
Error checking within a footprint

A copper shape or Component via in a footprint can be marked to be excluded from spacing checks with other items in the same footprint. This allows copper to connect two pads together whilst the pads are on different nets (for example, copper in a spiral inductor or a via in a vertical resistor).

To activate this feature you must check the **No Spacing Errors Within Footprint** box in **Properties** for the selected item.
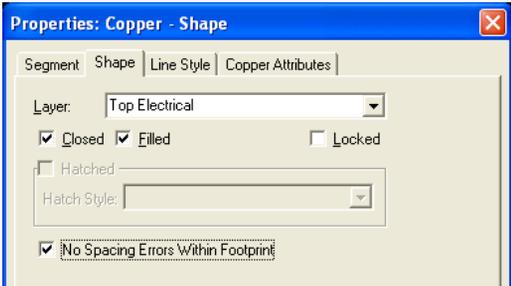
This allows you to still run a DRC check in the Footprint while designing it so that genuine errors can be flagged. There is an option on the DRC dialog to check **Excluded Items**.
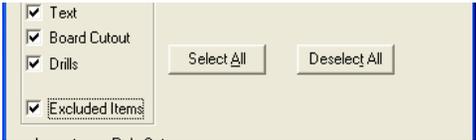
Template Properties

A template can now specify an alternative Net Class for thermal pad and spacing rules.
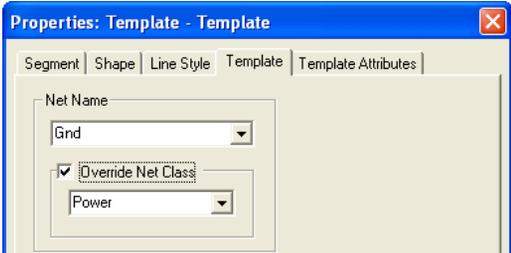
By selecting the **Override Net Class** button, you can specify an alternative net class to use for the selected template.

This can be used for low power or digital sections of a net where different thermal pads are required.

Copper Pour Check

When using the **Copper Pour** option, it now does a check when adding copper that the thermal spokes are added following the new DFM/DFT rule defined for minimum number of spokes.

CAM Plot Changes

NC Drill changes

NC Drill Output

The NC Drill option now reflects the drill table to define the tolerances for drills used.

Drill tolerances

The **Drill Table** dialog has been enhanced so that the you can define drill tolerances either as **Allowed Drill Sizes** and/or by using the existing method of **Tolerances** and **Steps**.
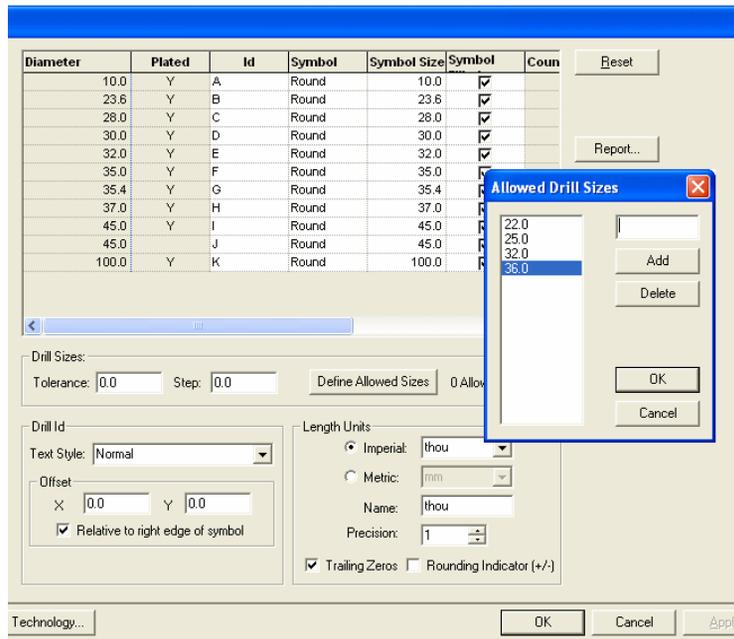
The **Define Allowed Sizes** dialog will enable you to specify drill sizes that are acceptable to your manufacturing process. This enables you to refine the drill sizes based on actual available or recommended drill sizes. The **Reset** button will regenerate the table, setting default values and removing drill sizes which are not referenced in the design.

Drill sizes shown outside the allowed drill sizes will be highlighted in red on the dialog.

The drill table dialog gives you a summary of how many matching drill holes and the range of sizes which match after step and tolerance have been taken into account.

Routing profiles

The Excellon output has been enhanced to allow routing patterns to be output. A routing output is used where a board outline is profiled to remove it from the panel that it is manufactured in. It can also be used to create a routed slot in the board.

This is achieved by selecting a **Process** layer other than **Layer Span <Through Hole>**, Board for example, and by selecting **Output** as **Excellon**. For best results, create a Board layer that has just the option for processing. By only plotting the board it is possible to get a routing output.
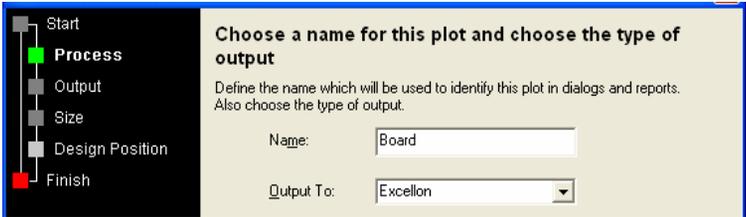
On the **Output** page, select the **Style** as **Outline**. Normally for NC Drill you would select **Drill** as the Style.
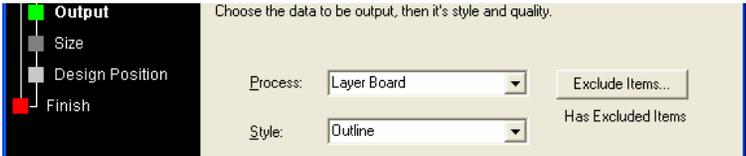
Windows Verification plots

When using a Windows verification plot (selected from the Gerber Setup page), you can specify whether the plot is actual size or auto-scaled to fit the page size. The verification plot is centred on the paper.
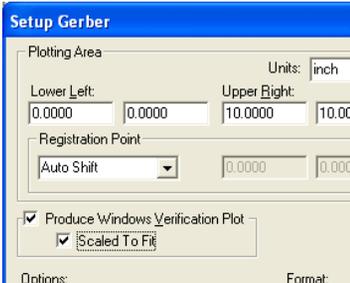
Plot changes

You can now plot using **named areas**. The plot dialog has been changed so that you can specify the alignment with area, design extents, working area and board extents.
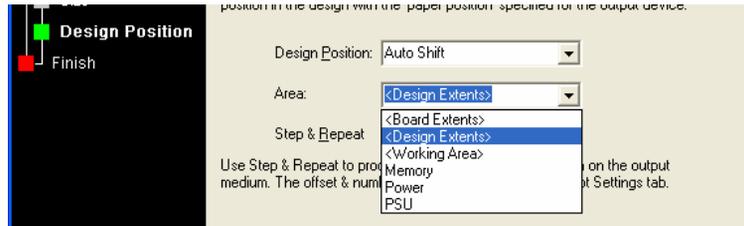
You can include the **Variant Name** in an output file name. You can use any combination of Design Name, Variant Name and Plot Name.

Save Settings

You can Save and Load device settings to file on the **Plot Settings** tab of the CAM Plot dialog. This allows you to create device 'profiles' for the different devices available. For example, the Gerber device can be saved in 3.2 or 4.3 formats using device files. Device files are saved with the file extension of .plt. These files can be sent to your manufacturer to ensure that the settings used are accurately reflected in the plot.

Registration point values displayed

From within the **Preview** window the **Registration** point and **Plot Area** sizes (width and height) are displayed.
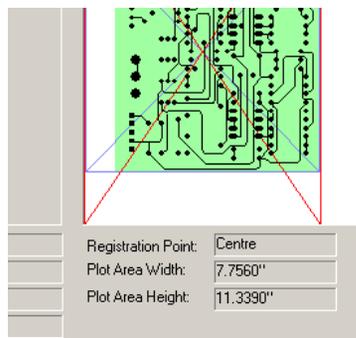
Negative Windows Plot

Windows plots can now be negative. Previously this wasn't available. From within the CAM Plot dialog, on the CAM Plot wizard, **Output page**, you can select between a **Positive Plot** and **Negative Plot**.
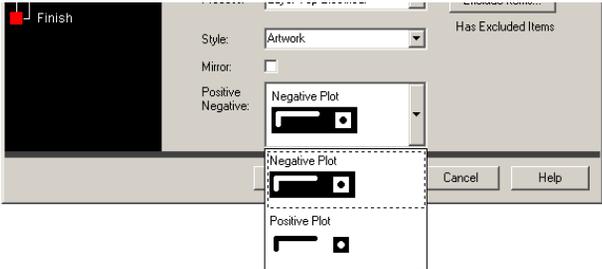
The Negative Plot output would be exactly the same as if for a positive plot except that it is reversed. This may be required for some less sophisticated manufacturing processes.

The resultant plot looks like this:
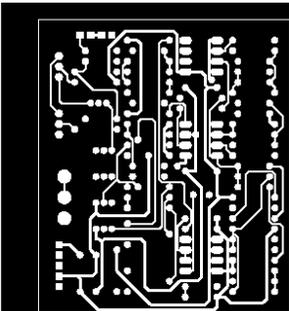
ODB++ Output Changes

Two new check boxes have been added to the ODB++ dialog to enhance its operation.

The **Use CAM/Plot Layer Combinations** check box means that instead of outputting data for every layer in the design, it will output data for each **Gerber** and **Excellon** type plot defined in the **CAM Plot** page of the **Technology** dialog, this includes multiple combine layers.
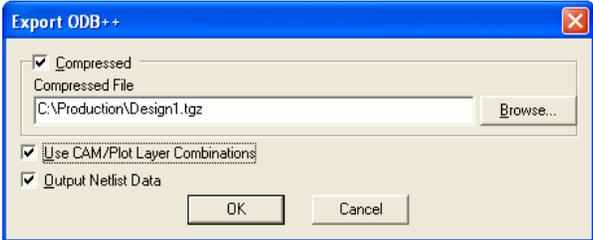
Some viewers have problems with reading the netlist associated data in ODB++ files (a non-Valor viewer). So if the viewer you are using will not read the ODB++ data output (which will include netlist data), uncheck the **Output Netlist Data** option.

GenCAD Output

The GenCAD output uses version 1.4 format. This is a manufacturer specific output for driving test and production but is also used by the Fabmaster product, Polar Instruments products and other systems.
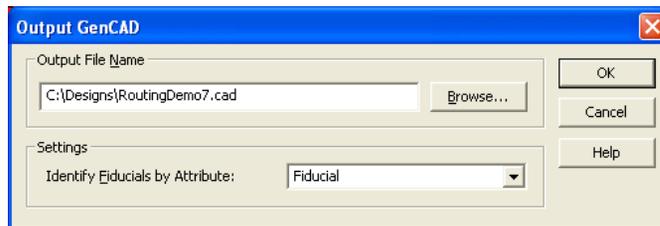
The **Settings** option allows you to specify the attribute used to identify the Fiducial markers in the design. These must be manually attached as attributes by you during the design process.

IDF Output

The IDF Output dialog has been changed to allow you to select the **Component Outlines Layer Class**.
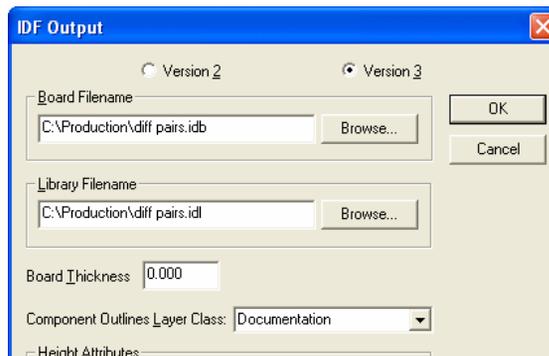
This can be used to select which **layer class** is to be used to choose the Component outline shape. The largest closed shape on a layer using the selected layer class is used as the Component outline for each component. If no layer class is selected, all shapes on non-electrical layers are considered.

Auto Mitre/Unmitre

There is a new option on the **Tools** menu, **Auto Mitre >**. This is used to add and remove mitres in tracks, this is also available on shortcut menu for a selected track.

Mitre

You can set the auto mitre parameters by using the **Auto Mitre All** command, or the **Auto Mitre Parameters** command.

Use **Curved Mitres** to insert an arc instead of a line for the mitre.

Use **Any Angle** to allow mitres to be added to non-orthogonal (not 90 degree) corners. Normally only 90 degree corners are mitred.

Maximum Mitre Size - This is the maximum mitre offset in the X or Y direction, not its length. Mitres are usually added using this size, but if the adjoining segments are short it will add the largest size it can fit in.

Minimum Mitre Size - When the maximum size cannot be used because the 90 degree segment is too short, a smaller mitre will be inserted. This parameter makes sure that the mitres do not become too small.

Note Mitres are only added where they will not cause design errors.

Unmitre

The **Unmitre** tool will attempt to remove mitres. It analyses the corners and will identify segments which could have been added by the mitring tool (it uses the mitre sizes set in the Auto Mitre parameters as a guide). The segments identified as mitres are removed. The segments are only removed if removal will not cause design errors.

Auto Track Smooth

There is a new option on the **Tools** menu, **Auto Smooth>**, this is also available on the shortcut menu for a selected track.

This is used to smooth existing tracks. This command will attempt to remove corners and vias from a track path whilst retaining the connectivity and not introducing any design rule errors.

Changes to Dimensions

Dimensions in footprints

Dimensions can be added and saved in Footprints. When added to the PCB design, if the layer used for the dimensions exists in the design then the

dimensions will be added. If the layer doesn't exist, the dimensions will not be added. The latter may be a useful option for adding additional footprint documentation for use within the Footprint editor only.

Prefix Text in Dimensions

You can now add a prefix to the dimension text used. There are separate defaults for **Angular**, **Radial** and **Length** dimensions. Radial dimensions can now have a different default unit to that of other length dimensions.

When using prefix text, you should add a space character after the text so that when used, it spaces the prefix from the dimension.
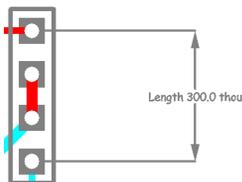
These text prefix can be defined in the **Design Settings** option under **Defaults** and **Dimension Units**.
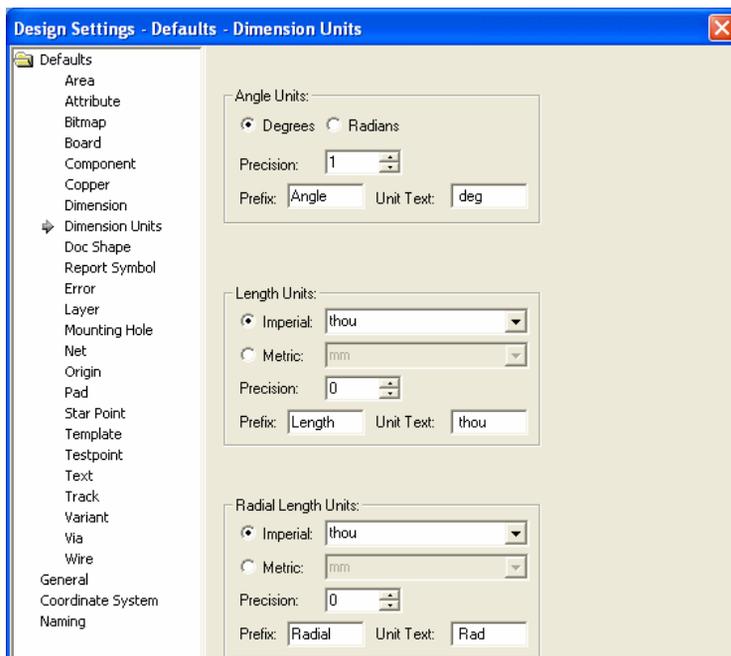
Snap Dimension To Arc Centre

When using linear dimensions, you can now snap to the centre of an arc using the **Snap to Arc Centre** on the shortcut menu. By selecting the arc, the dimension will automatically select the arc centre.
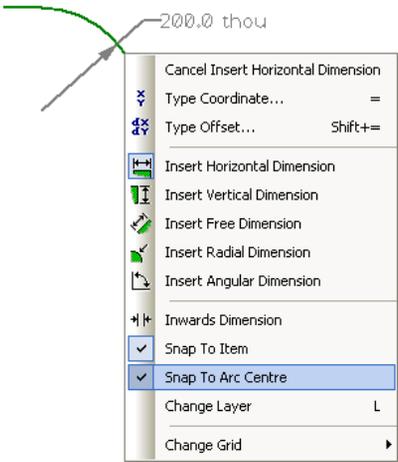
Radial Dimensions

Inward style radial dimensions now behave more consistently when the label is moved to the inside of the arc. The dimension will 'pivot' about the arc centre.

Layers Dialog Changes

Within the **Technology** dialog, the **Layers** page has changed. The dialog is formatted as before but there are separator lines to clarify each class category, e.g. Electrical, Documentation etc.

Changes have been made to the **Layers** dialog grid to support the addition of inner layer '**sets**'. Pulsonix already has <Top Side> and <Bottom Side> layer sets, and you can now add <Inner> layer sets. The Inner layer sets are used for embedded Component technology and allow association of non-electrical layers with inner electrical layers, e.g. resistive material layers.
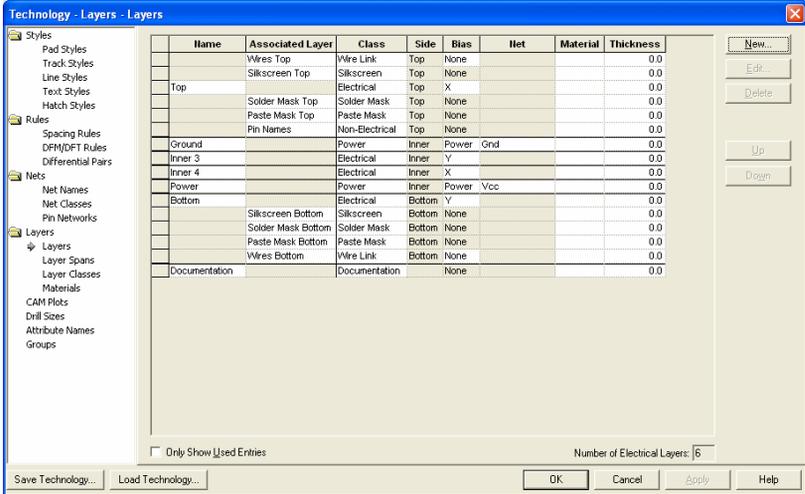
The dialog now shows The **Number of Electrical Layers** at the bottom of the layers page.

The **Colour** column has been removed (layer colours can be changed in the **Colours** dialog as before).

The **Layer Names** column has been split into two columns - **Name** and **Associated Layer.** The second column is used to hold names of layers that are 'associated' with a layer set. So for example, a Top layer might have associated with it, Top Silkscreen, Top Paste, Top Wire etc.

Further changes have been made to enable the Embedded Components technology, this is detailed in a later chapter headed *Embedded Component Technology*.

Layers Bar

Categories button

From within a PCB design, the layers bar contains a **More>>** (**Hide**<<) button.
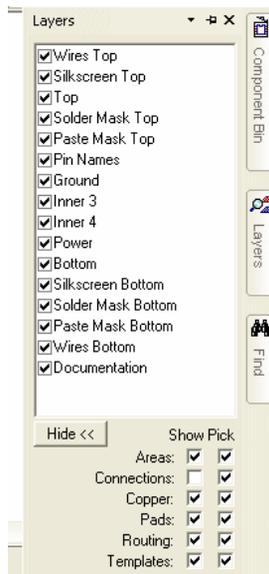
This button displays the **Show** and **Pick** switches for six electrical PCB item categories. If you do not want to use these categories then use the **Hide<<** button to hide the switches from the bar.

Each of the boxes uses a quick method of switching the appropriate options in the **Colours** dialog. These check boxes represent multi items in the Colours dialog, not just the item shown. For example, **Routing** represents **Tracks**, **Breakouts** and **Vias, Pads** switches off all **Pads** and **Mounting Holes**. The state of the switch is based on the 'main' colour item category, e.g. Tracks for Routing, etc.

Show Items Layers

There is a new option on the shortcut menu in the **Layers** bar, **Show Item Layers**. With a layered item selected in the design, this option becomes available.
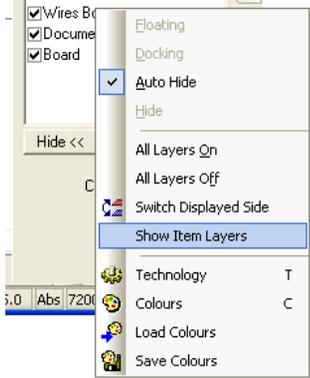
When the option is chosen, the layers on which the selected item resides will be switched on and all other layers switched off. This is also available as a command for assigning to a shortcut key.

Clear All Templates Command

A new command **Clear All Templates** (run from the **Run Command** option on the **Edit** menu) has been added. This means that as a command, you can now add this option to a shortcut key.

Obstacle Avoid Mode

The **Obstacle Avoid** option is available on the shortcut menu for use when adding and editing tracks. When used you are then using a special mode of routing. In this mode the track will automatically add a corner and change direction when you hit an obstacle.
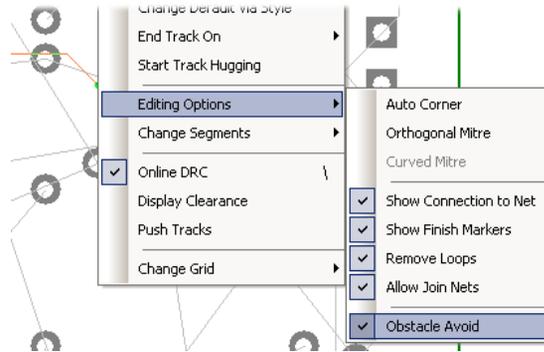
In obstacle avoid mode you can automatically remove any corners that have been added by retracing the cursor back over the track (like in auto corner).

This mode applies to either segment being dynamically moved. This has the result of being able to 'wrap around' obstacles without having to interactively add corners. The track to item spacing rule will be used in this mode.
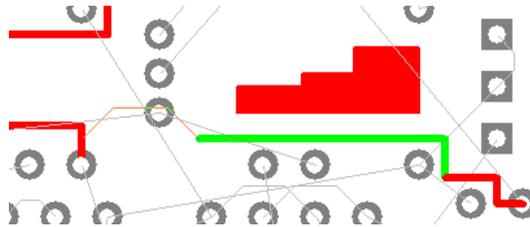
Using this mode you must have the **Online DRC** option enabled as well. The **Obstacle Avoid** mode is available on the shortcut menu under **Editing Options>** when editing a track. Also available on the **Options** dialog under **Edit Track**.

Track Hugging

During track editing, there is a new interactive editing feature that allows you to add a track that hugs another track or design shape as closely as it can.

The feature is particularly useful when adding bus routing patterns and track shielding.

▶ **To use track hugging mode**

1.   Start adding new track segments, or edit an existing track.
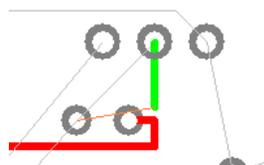
2.   Move the cursor over the track or shape edge that you wish to hug.
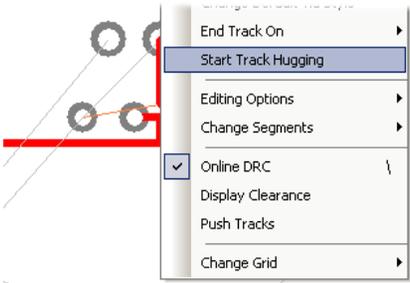
3. Right click the mouse and use **Start Track Hugging** from the shortcut
    menu. A better way to do this is to put the **Track Hugging** command onto a
    shortcut key and press the key when the cursor is over the track to hug.
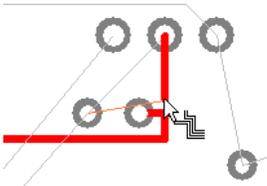
4. A modal cursor is displayed to confirm that the track hugging mode has
    started. If there was no track under the cursor when the mode was selected,
    pick the track to hug at this point.

5. As you move the cursor along the picked track or shape, the new track is
    added alongside it. The spacing rules gap is used to avoid design rule errors.
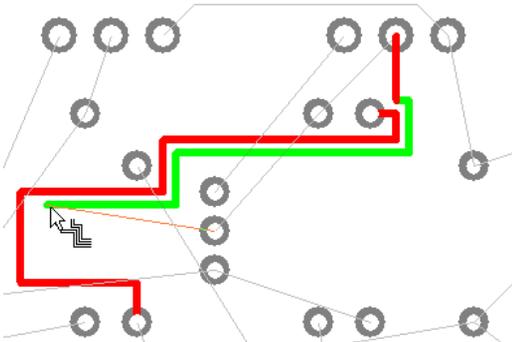
6. Press the left mouse button to add a corner to **End Hugging** and move away
    from the shape. Selecting the **Track Hugging** command again will also stop
    the hugging mode.

Polar Grid Coordinates

Polar grids are used to place items that require a radial grid rather than a regular linear grid.

For a polar grid, check the **Polar Grid** box in the Grids dialog.
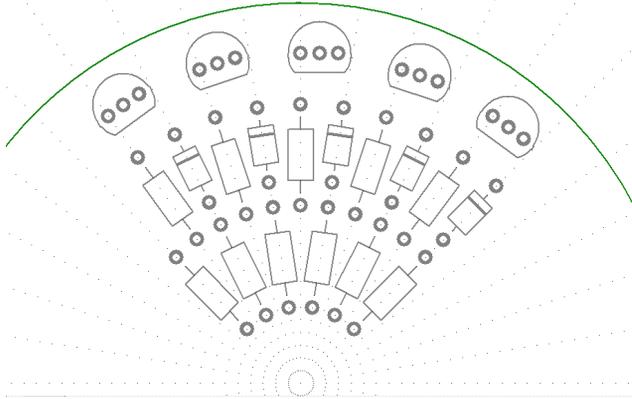
You can define polar grids using the **Radial Steps** count and **Concentric Step** under the **Step** box. Once the **Polar Grid** check box is selected, you should then define the number of **Radial Steps**. For example 8 radial steps would give one every 45 degrees, radiating from the origin. The **Concentric Step** is the gap between each concentric circle around the origin. The origin **Angle** is the angle of the first radial, this effectively rotates the whole grid about the origin. For relative polar grids, only the grid origin position is relative.
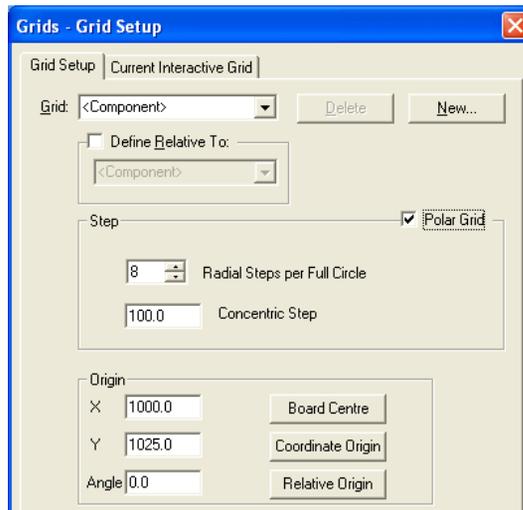
Use the grid **Origin** to position the grid centre in the design, usually this would be the board centre on a circular PCB.

Using the three buttons available, you can choose to set the X and Y grid origins onto the **Board Centre**, **Coordinate Origin** or the **Relative Origin**.

If you zoom out too far, a 'rough' polar grid is displayed. This allows you to still view the polar grid but in a much rougher form until you zoom back in again.
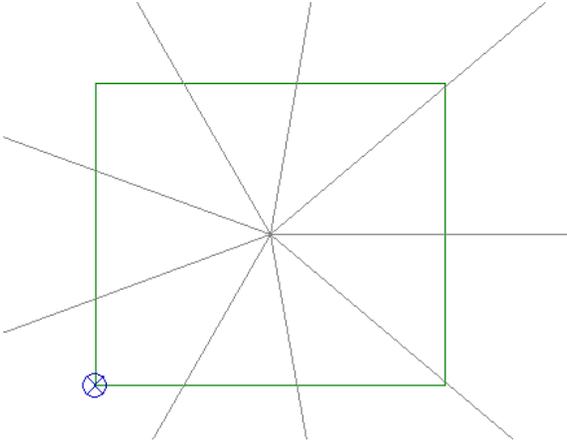
Auto Rotate when moving Components

When interactively moving **Components** and **Text** on polar grids, you can use the **Auto Rotate** switch in the shortcut menu. The next move of a Component remembers this setting. This means you can easily move Components snapping on the polar grid but also rotating them to maintain the grid position.

Placement Sites

To aid the manual placement of critical items, you can now nominate **Attribute Positions** to be placement sites. A new dialog under the **Design Settings** called **Placement Sites** can be used to define the name of the attribute to be used.

The check box **Use Attribute Positions As Placement Sites** activates this special placement option.
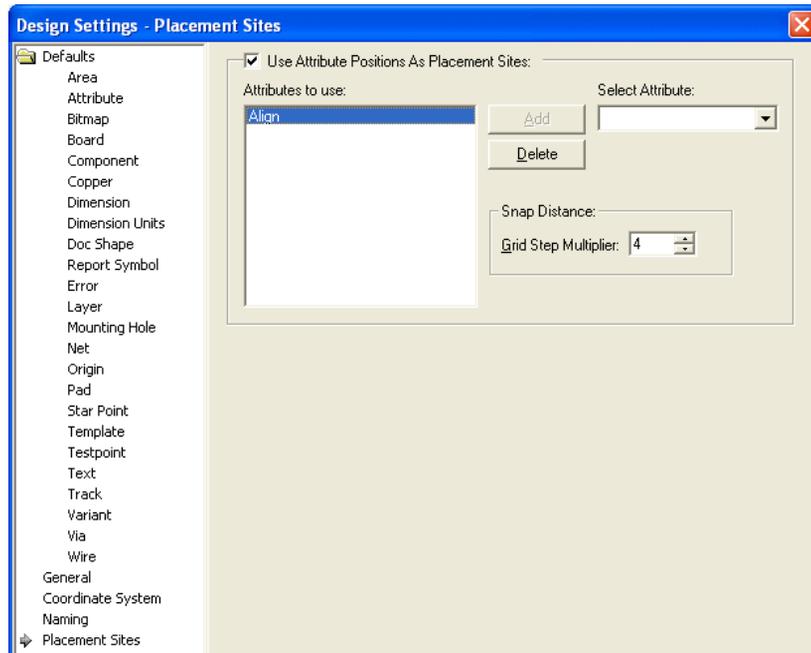
With the PCB design, when moving an item, the option Toggle Placement Sites is available from the shortcut menu.

The Attribute Positions act like extra grid points and the cursor will snap to these.

DXF Import - Placement Sites

When importing into a PCB design, it is possible to generate Placement Sites from 'crosses' in the DXF data. Placement Sites are attribute positions in the design, that can be used to 'snap' Components onto particular locations on the board.
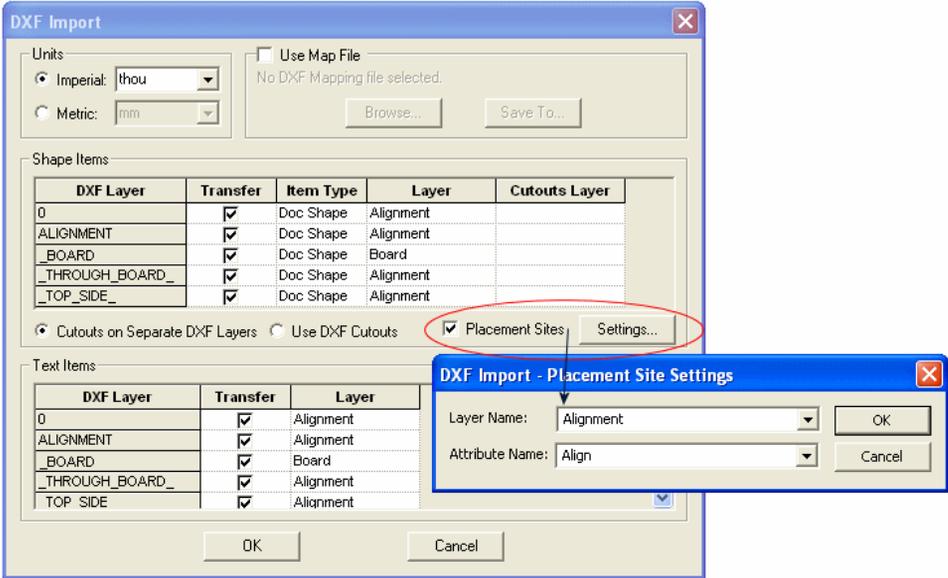
If the **Placement Sites** checkbox is ticked, the design will be examined after import and any crosses (two single segment shapes of the same size that cross at 90 degrees to each other) will be marked with the specified attribute position at the point the lines cross. The **Attribute name** and **Layer Name** are specified in the **Settings** dialog.
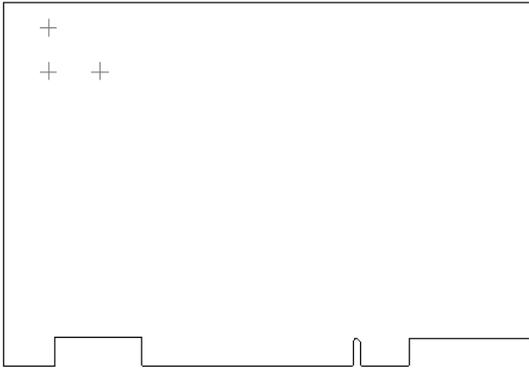
The layer to be checked for crosses, and the attribute name to use, are specified on the Placement Site Settings dialog. The attribute used should be added to the list of **Placement Site attributes** in the **Design Settings** dialog.

Plane Isolation on a Track

For a track on the same net as poured copper or a powerplane, you can specify if it is isolated or not, in a similar way to pads. This overrides the **Technology** option. This is on a per-instance basis for track. The Avoid Same Net option is for all instance.

Use the **Properties** dialog for a selected **Track**. You have the choice of **Default**, **Isolated** and **Not Isolated**.
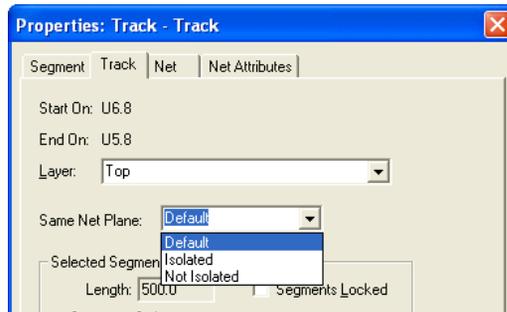
Drawing Drill Holes

You can choose to always draw drill holes, even when the layer classes have turned them off. This means that you can see the holes while designing, without changing the layer classes.

This is available from the **Options** dialog and **Display** page as Draw Drill Holes. You can choose Never, By Layer Class and Always

The **Draw Drill Holes** preference is only relevant to designs which can contain drill holes (e.g. PCB). It only affects drawing to the screen and not post processing. Redrawing speeds may be slightly enhanced by turning off drill holes (Never). The default is to draw drill holes when a displayed layer has a class with drill holes shown (By Layer Class). You can also choose to always draw drill holes when a pad is drawn (Always).

Import Design Data/Export DXF in PCB Profile

From within the **Import Design Data** option on the File menu and while editing **PCB Profiles**, you can now import DXF files into PCB profiles. This information can also be saved in the Profile files.

'Where used' Indicator

The 'where used' indicator has been changed to make it more obvious. Within the **Reports**, the **Drill Table** and **Technology**, a '**Y**' is used to imply used/true, this has changed from an '**X**' used previously.

Plated	Id	Symbol	Symb
Y	A	Round	
Y	B	Round	
Y	C	Round	
Y	D	Round	

Colour Dialog Changes

From within the **Colours** dialog and on the **Layers** page there is a new option on the shortcut menu. **Apply to this side** will allow you to apply the **Displayed** and **Selectable** settings to all pages in the **Colour** dialog.

Colours - Layers

	Layer	Displayed	Selectable	True Width	Colour
Layers	Wires Top	☑	☑	☑	
Tracks	Silkscreen Top	☑	☑	☑	
Elec Shapes	Solder Mask Top	☑	☑	☑	
Doc Shapes	Paste Mask Top	☑	☑	☑	
Pads	Pin Names	☑			
Vias	Top	☑	Apply to entire Row		
Text	New Plane	☑	Apply to entire Column		
Attributes	Plane	☑	Apply to this side		
Highlights	Bottom	☑	☑	☑	
Others					
Nets					
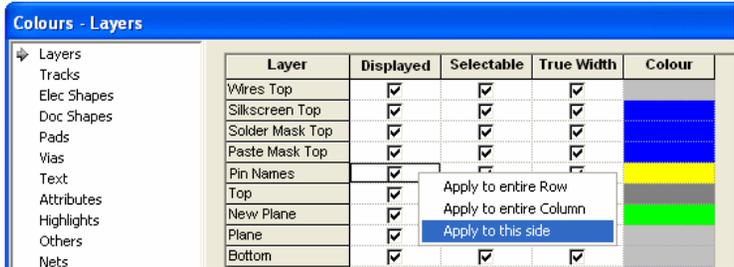
Sketch Track

A new option on the **Insert** menu allows you to very quickly add a PCB track following a particular path. You sketch in a rough track path and it is converted to a tidy track that is as close as possible to the sketch, but avoids producing errors.
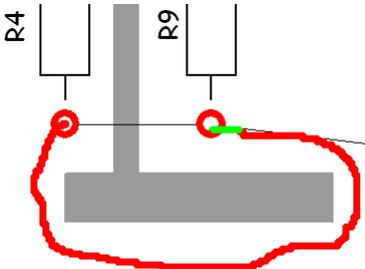
The track is drawn in after selecting **Insert Sketch Track**.
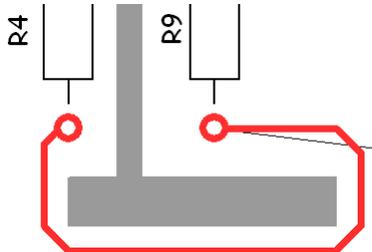
When the track path has been drawn, the mode tidies it up and optionally adds mitred corners.

Shortcut menu options

While using the Sketch mode, from the shortcut menu you can change the options available.
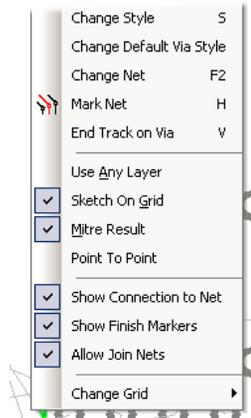
Use Any Layer allows you to sketch a track path which can be routed on any track layer, the current layer is used if possible, but vias can be added as required. Otherwise, the tracks will stay on the layers they were sketched on.

Sketch On Grid controls if the sketch is to be completely free hand or gridded (using the Track Grid).

The **Mitre Result** option causes the resulting track path to be mitred using the **Auto Mitre** tool and the parameters defined.

Rather than sketching a rough continuous path, the **Point To Point** mode allows you to define a more exact path. A track is drawn between the first point (using a mouse click) and the next selected point (also on a mouse click).

The shortcut menu switches are reflected on the **Options** dialog, **Edit Track** page under Sketch Track.
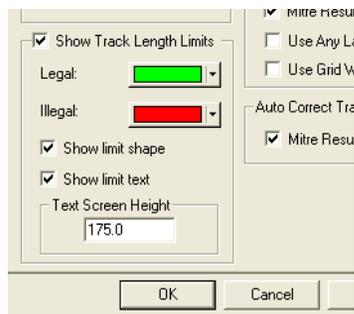
The most effective way to use this mode is to point and click, the click will immediately enable the tidy function. Click at the next point required and another tidy will be done. This process will give you better overall results.

Auto Correct Track

A new option on the shortcut menu when editing tracks, **Auto Correct Track**, allows you to correct a PCB track that is in error. It will automatically move and change the layer of the track segments to avoid obstacles. You can use the Auto Correct Track option in the **Options** dialog under **Edit Track** to decide whether the result is to be mitred or not.
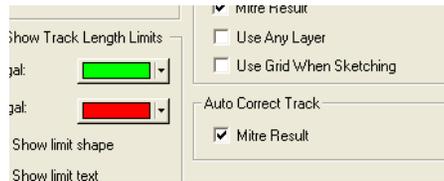
Apply Layout Pattern

There is a new option on **Tools** menu (in PCB) to take the layout pattern (Component positions and tracks paths between Components) from a set of Components and apply it to a different set of Components.

Recommended Uses

This option is ideal for repetitive circuits such as RGB drivers for example, where one 'channel' is defined, placed and routed. Then the other two channels are 'copied' using this tool.

It is also useful where you have a previous layout defined and wish to apply that circuit to the current design. This type of circuit may be particularly EMC or high-speed sensitive where the layout is critical. It may also be that because of compliance testing, the circuit has been approved and hence the layout pattern must be maintained.

How To Apply A Layout Pattern

If you wish to apply the pattern to a particular set of Components, first select them in the design (they must be fully selected using the Shift key), or select them in the Component bin. You do not need to do this if they are all in a named **group**. Note: You can use the **Find Bar** to easily select a set of Components with a common name stem, or sharing a common attribute value. Also, you can use the **Cross Probe** option to select the Components to apply the pattern to in the schematic design. For example, selecting a block symbol in the schematic will find all Components in the block and select them in the PCB design or Component bin.

The **Apply Layout Pattern** dialog is displayed.

The dialog allows you to specify where to get the pattern from and which Components to apply it to.

This includes ability to take pattern from a different PCB design, which allows you to save patterns to PCB files for use later.

Take Pattern From

Choose where to find the completed Components which define the layout pattern (Component positions and track paths) to be used.

- **Selected Components** - Copy the pattern from the selected placed Components.

- **Clipboard** - Copy the pattern from the Components that have been copied to the clipboard. This is a good way to copy the pattern from a set of Components in another design.

- **Group** - Copy the pattern from the Components in the same design that are in a named group.

- **PCB File** - Copy the pattern from all the Components in an external design file. Once you have placed and routed a common block of Components, you can use copy and paste to place a copy of this pattern into a separate design. These layout pattern designs can then be saved in the a directory to form a library of common layout patterns.
  Use the dropdown list to select a pattern file from the directory shown, or use the Browse button to change the folder.

Apply Pattern To

Choose which set of Components you wish to apply the layout pattern to.

- **All Components** - Apply the pattern to all Components in the design. Use this when you are creating a PCB design that is very similar to an already completed design.

- **Selected Components** - Apply the pattern to the selected placed Components in the design.

- **All Components In Bin** - Apply the changes to the Components in the bin.

- **Selected Components In Bin** - Apply the changes to the Components in the bin that have been selected. This is only available if more than one Component has been selected in the bin list.

- **Group** - Apply the pattern to the Components in the chosen group.

Apply Pattern

Once the pattern has been established, the **Apply Pattern** button is used to apply the layout pattern from one set of Components to the other. The method will attempt to match as many Components as possible between the two sets using part names, pin counts, similar nets etc. Any Components not matched will be left in their previous position.

Any tracks between the Components that are having the pattern applied will be removed prior to the operation. Tracks from these Components to other Components not involved will be unrouted, ready to be re-routed by hand after the pattern has been applied.

Each Component will be moved to the location of its matched Component in the layout pattern. The repositioned Components and their new track paths will be left selected after the operation. This makes it easy to use the move option to position them.

If you do not like the result, use **Undo** to revert back to the previous placement and routing pattern.

Fatten/Neck Tracks

There is a new option on the **Tools** menu and the shortcut menu and is used to fatten (neck) tracks. Where possible, this will work from the default track width to the alternative track width, and neck tracks if needed to avoid errors. The option swaps the track back to minimum width and then fattens where it can and remain legal.
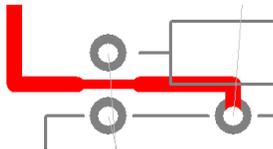
Two options provided on the **Tools** menu are **Fatten/Neck All Tracks** and **Fatten/Neck Selection**. The latter only acts on the selected track segments and is also available from the shortcut menu when a track segment is selected.

The track must have a net class defined for it as well as Values for the Default and Alternative track widths. These values are edited in the **Net Class** option on the **Technology** dialog.
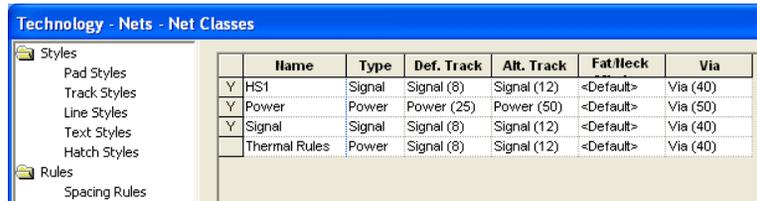
The Fatten and Neck track options will work but looking at the thickness of the Track Style used in the Net Class for that net, and then comparing with the 'other' style. If it is thicker, then it fattens up to it. It is normal to be initially using the **Def. Track** style and to fatten up to the **Alt. Track** style but either way will work.

To avoid the option adding really short segments of track length, there is a minimum necked or fattened track length field to add a value to. The default value <Default> is set to 4 times the 'fat' track width, but you can enter an absolute value for this on a Net Class.

RF Design - Spirals

As part of the increasing support for RF designers, you can now use Pulsonix to insert a **copper** or **track spirals** (and **breakouts** in footprints).

Insert Spiral is a new option available on the **Insert** menu. This feature can be used for designing spiral inductors and planar transformers.
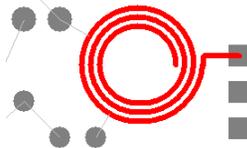
When adding the spiral, various parameters allow you to control its shape.
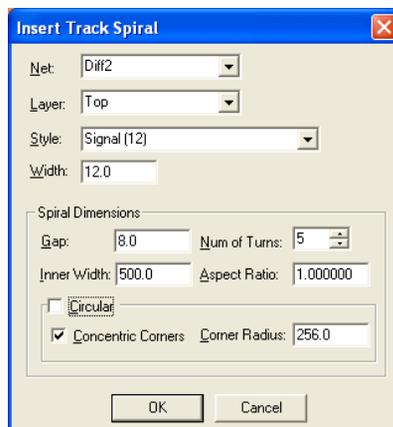
Select the **Net** to connect the spiral to. For tracks you must select an existing net, but Copper does not have to be on a net.

The **Layer** box is used to select the layer to place the spiral on.

Style is used to select the track/copper Style of the spiral. You can also type a **Width**.

The Spiral Dimensions

The spiral always begins at the inner right side and ends at the outer right side (but you can rotate or mirror it later).

The **Gap** is the distance between each *turn* of the spiral excluding the segments **Widths**.

The **Num of Turns** is the number of complete *loops* of the spiral.

The **Inner Width** is the distance across the inner void of the spiral.

The **Aspect Ratio** allows you to create *rectangular* spirals and is the ratio of **Height / Width**. So an **Aspect Ratio** > 1.0 gives a **tall** spiral, and < 1.0 gives a *Wide* spiral.

The **Corner Radius** is the initial inner radius of the corners of the spiral.

Concentric Corners gives you *tight* corners which are properly nested, increasing in radius as the number of turns increases. Otherwise the corners are fixed at the **Corner Radius**.

The **Circular** option fixes **Concentric Corners** on, and the **Corner Radius** to be half the **Width** plus the **Inner Width**. This has the effect of giving near circular spirals. A zero **Corner Radius** and not **Concentric Corners** will give you square corners.
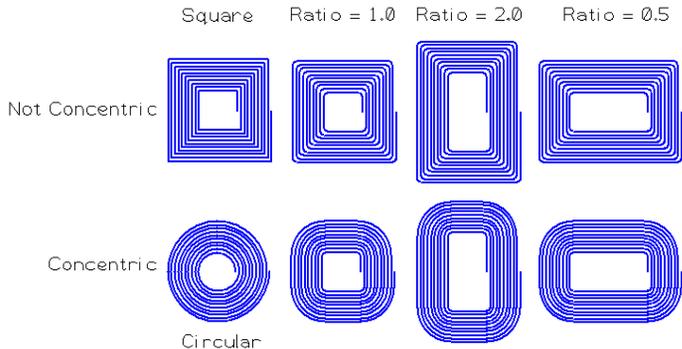
Note: Spirals are added normal track and copper, to modify, you should use delete and then add another one.

Auto Router Changes

Use of DO Files

For users who have the autorouter option, you can now use any existing DO files with the new autorouter. DO files are ASCII command files that can be used to drive the autorouter. These files originate and follow the Specctra auto router format. This file contains a list of route, cost and rule commands which defines the strategy to use.
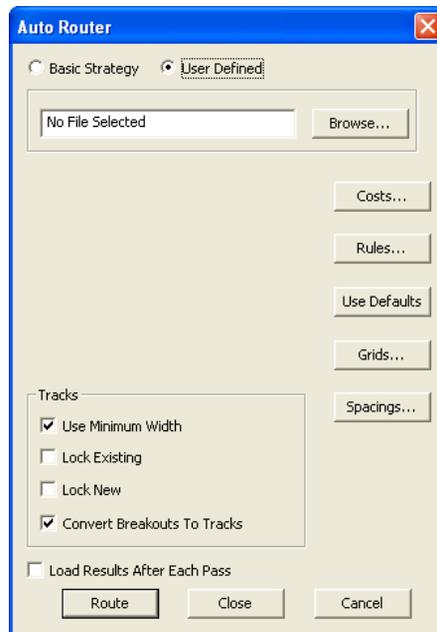
The command file is selected by either typing the name of the file or using the **File Browser** to locate the file. A list of available commands and their syntax can be found in the Pulsonix online help pages.

Auto Route Selected Nets

When using the **Auto Route> Browse Nets** and **Route Selected** options, you can now specify whether the **Router Parameters** dialog is displayed (and acted on). This option is available from the **Options** dialog under **Interaction**.
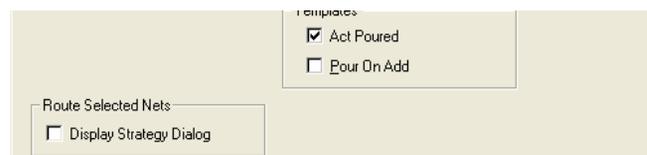
Auto Place Changes

The Autoplace option has been enhanced.

- You can now choose to place Components in a named group.

- You can now place to any selected or named area in the design.

- You can now place Components around any, selected or named board outline or named area in the design.

- You can now place around the board outline using this dialog.

This section of the dialog allows you to choose which Components are to be considered for placement, and where you want them to go. These buttons can be used in various combinations, enabling you to carry out complex placement operations in 'stages' to produce the desired results.
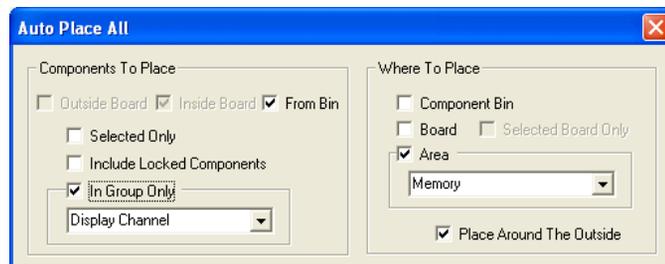
Place Around Board From Bin

If any Components are selected in the bin, the **Place Around Board** option available on the shortcut menu will only place the selected Components around the board. If nothing is selected it will place all the Components in the bin.
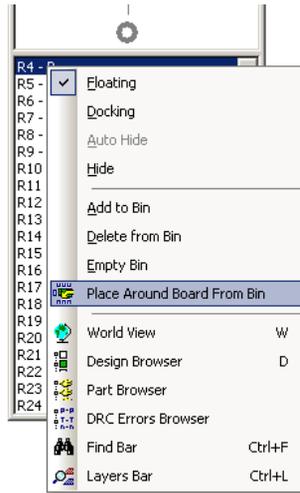
Options dialog change in Version 4.0 Build 2569

A new switch, **Check Poured Copper** in the **Options** dialog on the **Online DRC** page has been added to allow you to force online DRC checking to ignore poured copper.
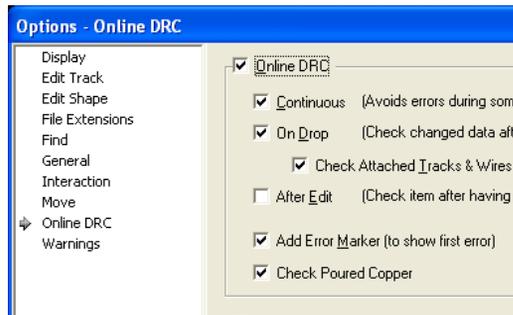
Use this when adding tracks that will be routed into a poured copper shape. This will stop the continuous track to copper errors being displayed. Re-pour the copper after the track has been added to clear the copper around the track.

Chapter 5. New Cost Options

High Speed Option

Interactive Differential Pair Routing

To aid the design of high speed circuits, you can define **Differential Pair Rules** in the design (at both Schematic and PCB stages). The interactive Differential Pair Routing feature is available in the PCB design editor.
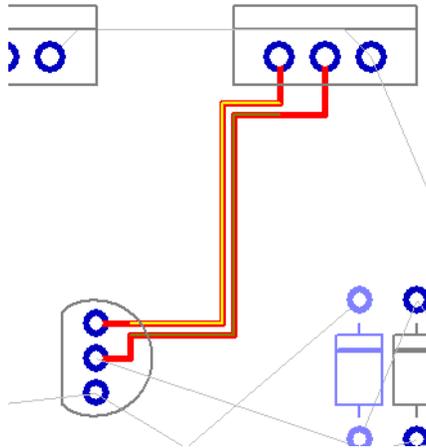
A Differential Pair consists of two pairs of pins on two different nets which should be routed together as close as possible. It is possible to define such pairs in the **Technology** dialog under **Differential Pairs**. The pair may have rules associated with them which define how close the tracks should be and how much they are allowed to differ in length. Once this is defined, a special manual routing mode can be used to route the two track paths together. Paired track sections are locked together using the spacing rules gap. Whilst paired, subsequent editing of the tracks will keep them locked together using the functionality provided.

Differential Pair Rules

The spacing and length rules are defined in the **Technology** dialog. The spacing is either defined explicitly, or is taken from the **Spacing Rule** appropriate to the two nets. **Length rules**, if defined, can be checked as part of the **Design Rule Checking**.

If the two potential differential pair nets are selected when the Technology dialog is opened, these net names will be entered directly as pre-selected in the **Differential Pairs** dialog when the **New** (Pairs) button is used.

For each pair you provide:

- The net and pair of pins for the first master pin pair

- The net and pair of pins for the second pin pair

- The minimum gap allowed between the tracks

- The minimum percentage of how much the tracks are paired

- The maximum allowed track length difference between the pairs

Defining a Differential Pair

A Differential Pair is defined in the **Technology** dialog under **Differential Pairs**.

A Differential Pair requires two pairs of pins on two different nets. In this dialog you can define these two pin pairs by firstly selecting a net. This will automatically select the first two pins in this net. It will normally be the case that the net only has two pins, otherwise you should ensure that the correct two pins are selected. Repeat this for the second pin pair.

You can optionally define rules for this pair.

The **Minimum Gap** is the minimum distance the pair of tracks can be. This can be less than the normal spacing rules for these nets (which would otherwise be the value used).

The **Minimum % Paired** is a rule which is checked by the design rules checker (under Nets, Differential Pairs). It ensures that of the total length of the two Paired tracks, the minimum % value defined for the proximity with each other is obeyed. If 80% is defined, then 80% of the track length must be within the minimum spacking rule.

The **Maximum Length Difference** is the maximum the two track paths are allowed to differ in length.

Routing a Differential Pair

This is the process of routing a differential pair track pair:

▶ To route a differential pair

1. Begin by routing from one of the pins in the set (double click on the net), this will be the *Master* net. Normally you would create a small stub to start the route then enable the differential pair mode.
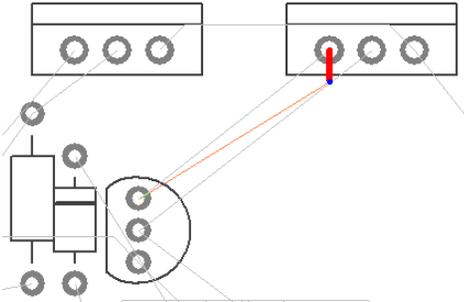
2. From the shortcut menu, use the **Start Differential Pairing** mode
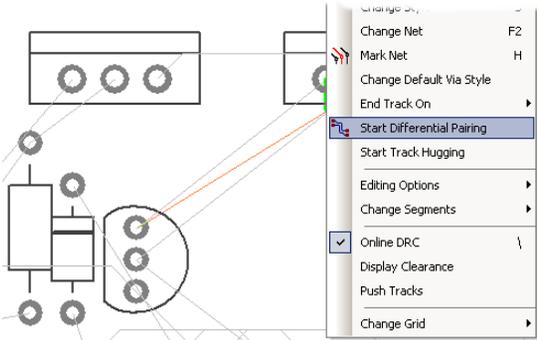
3. You will see that a *clone* track appears parallel to the track you are routing. This is the track on the opposite paired net and will be spaced using the spacing rule defined or the Min. Gap rule.

4. Each track pair has a *master* track and a *clone*. Only the master track can be modified directly, the clone is simply updated to match the master.

5. Continue routing in the normal way adding corners as required. The clone will follow the master track in shape.
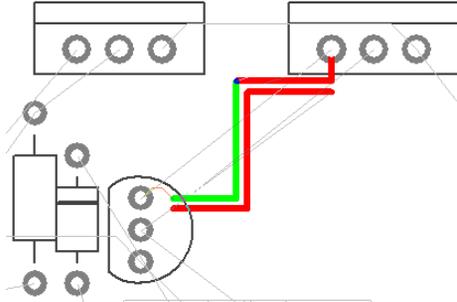
6. On the shortcut menu use the **End Differential Pairing** command to stop the pairing (the **Complete Track** command also ends the pairing).
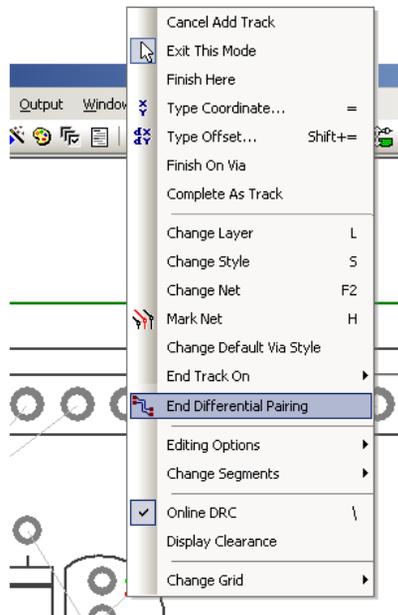
7. Finish the track by editing the remaining connection in the usual way.
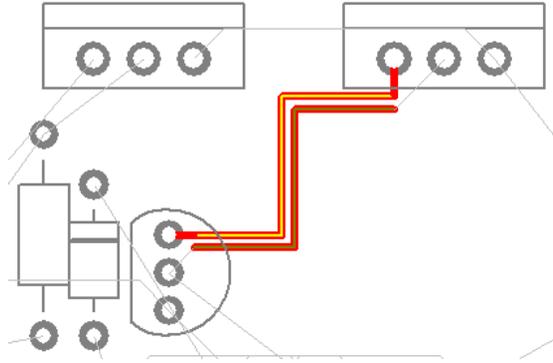
8. Complete the pair by routing the connections at the end of the 'cone' track.

9. This paired tracking is now 'locked' together until impaired.

Remove Differential Pair Routing

A paired section of track (or a selected part of it) can be unpaired using the shortcut menu command **Remove Differential Pairing** whilst in select mode.

You can create any number of paired sections of track along the path but you must complete the gaps manually.

Add Differential Pair Routing

An additional option is also available for the addition of differential pair routing – **Add Differential Pair Routing**.

The process of using this is to route the first paired net, then to select it and from the shortcut menu select **Add Differential Pair Routing**. An exact copy of the first track will appear parallel to it using the spacing rules distance or rules distance defined. As before you would now edit each of the remaining connections to complete the routing on the clone.

Differential Pair Routing Colours

You can draw the *master* and *clone* pairs in different colours (using two entries in the **Colours** dialog and **Highlights** page).
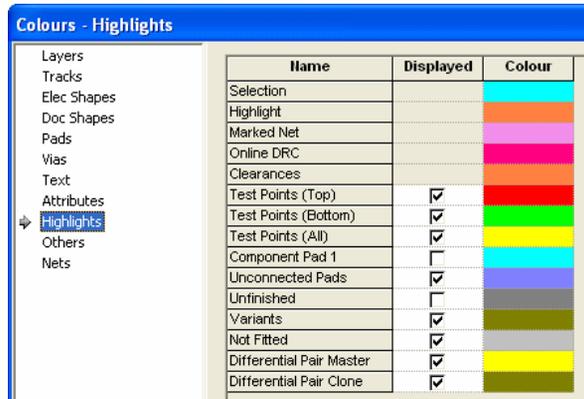
DRC Dialog

The DRC dialog can be used to check **Differential Pairs** using rules defined in the **Technology, t**his is available as a **Nets** check.
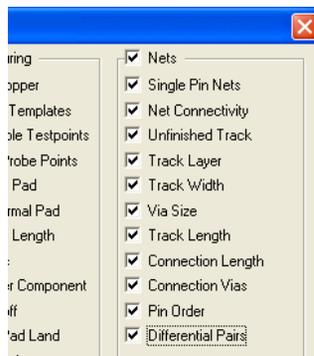
The checker will report an error if:

- The percentage of paired track length to the total track length is less than the minimum defined in the technology.

- The maximum total track length of the pairs is less than the max defined in the Technology.

- The tracks keep the specified distance apart.

A sample DRC error report might look like this:

```
Differential Pair Length Difference (DPL) at (3400.0
3375.0) on Layer "Top Electrical".
Differential Pair Paired Length (DPP) at (3400.0
3375.0) on Layer "Top Electrical".
Total:
     1 Differential Pair Length Difference (DPL)
     1 Differential Pair Paired Length (DPP)
Number of errors found : 2
```

Interactive Net Length Indicators

A new interactive **High Speed** feature has been added to Pulsonix. This allows you to define length rules (in both the Schematic and PCB Designs. If defined in the Schematic, these rules are translated into the PCB) and then for the interactive track editing facility to display the rules as restriction boundaries.

If there is a **Track Length** rule on the **Net Class** for the track being added or edited, or if the track is part of a pin network, then a shape is drawn indicating how far the track can be extended and still be within the total net (or pin network) rule. This is defined on the **Net Class** dialog under the **Rules** tab and **Track Length** Min: and Max:
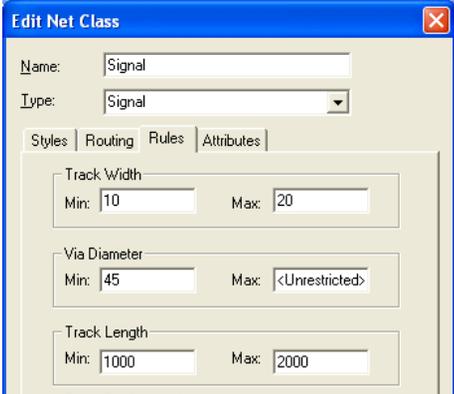
Separate shapes are drawn to indicate minimum and maximum net length rules.
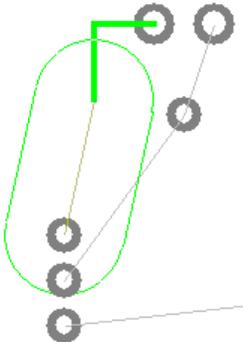
The large oval area shows the length of the track is within the minimum track length set.
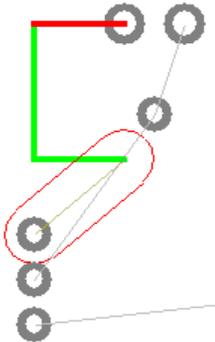
The small oval area shows that the maximum length rule has been exceeded.

The colours for the **Legal** and **Illegal** track lengths can be defined in the **Options** dialog under Edit Track.
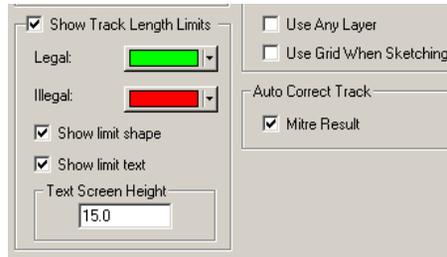
Track Length Limits

As well as the visual indicators for Legal and Illegal values, the rule values used can also be displayed. Using the **Show Limit Text** check box, the text value can be displayed. Again, this is defined in the **Options** dialog under the **Edit Track** tab.
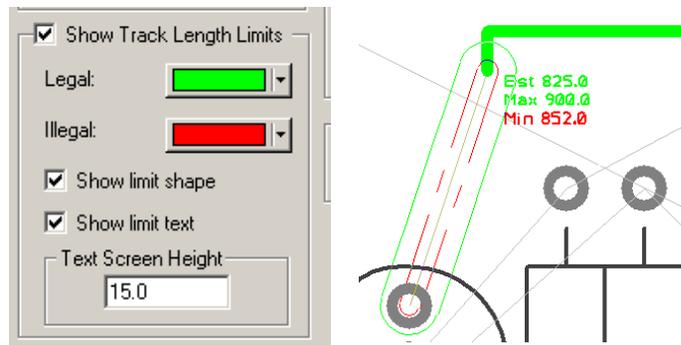
The **Text Screen Height** box allows you to define a text height that will be suitable when viewed. This value is the real screen height and is not related to the design text sizes.

Serpentine Routing

Another new **High Speed** feature for balancing the length of high speed nets is the insertion of track 'length' without introducing spacing errors. This is commonly known as **Serpentine Routing**.
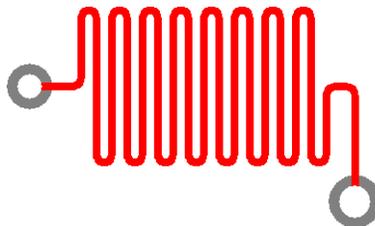
You can select a track segment (or segments) and run the **Serpentine Routing** command from the shortcut menu which prompts for the serpentine parameters.

Parameters are defined for the amplitude and separation of each loop. You can also define the minimum number of loop cycles to insert, and also the amount of additional length required (otherwise it will do as much as possible).
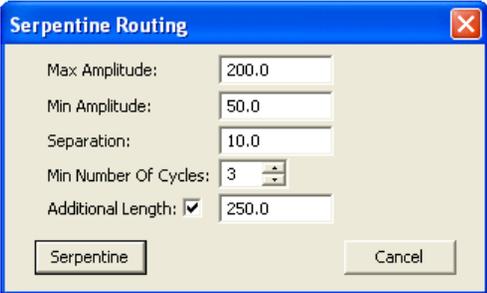
You can define two amplitudes that it can use to automatically reduce the amplitude to avoid obstacles, shown below.
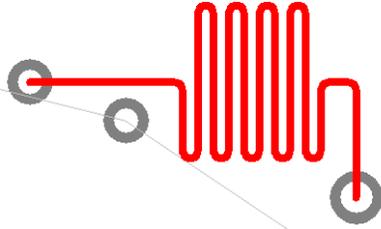
The serpentine parameters can be defined in the **Net Class** dialog, these values are used to prime the dialog. Net Class net and connection rules will be applied to prime the dialog with the required amount of additional track.
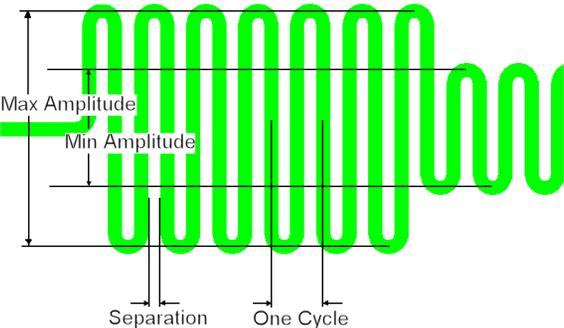
If **Additional Length** is required this can also be defined. If enabled, the serpentine routing option will stop once this length has been exceeded, otherwise the whole of the selected segments will be considered for serpentine routing. If a track minimum length rule has been defined on the Net Class, then the dialog will be primed with the Additional Length required to satisfy this rule.

From the shortcut menu, for a selected track the **Remove Serpentine** feature can be used to remove a selected section of serpentine routing. It can also be used to remove all serpentine routing from the design.

Also on the shortcut menu is the **Reduce Serpentine Routing** feature to reduce a selected section of serpentine routing by one 'loop'. This enables it to be easily trimmed to the correct length without interactively editing the track.

Embedded Component Technology

Technology Overview

New functionality has been added to Pulsonix to support the addition of embedded Components such as printed resistors, etched resistors, embedded capacitors and RF Components.

Some manufacturing technologies allow you to place and embed Components in the layer stack *within* the board. This technology allows much more efficient use of space, but requires additional features in the CAD system to enable this.

In a conventional PCB design, Components can only be added to the Top or Bottom sides of the board. A conventional footprint will have a body which is placed above the surface of the board and pads which fit onto or through the board. You would normally have the following items in a footprint definition: a silkscreen outline which is associated with the top side (mirrored to the bottom when the Component is placed on the bottom); a placement area which represents the space needed to accommodate the body of the Component and is either through the board, or just on the top (and mirrored when on the bottom); pads (though-hole or surface mount); resist shapes, which are often generated from the oversized pad shapes.

In a conventional technology, you would define Silkscreen, Resist and Placement non-electrical layers which are associated with the Top and Bottom electrical layers.

Embedded Components may not have a *body* like conventional Components, the functionality of the Component is built into the board, using copper shapes and perhaps resistive or capacitive materials which are applied to the board during manufacture. There are many different types of embedded Component, but they all have common requirements to enable them to be added to a design.

With embedded Components you may want to place pads and copper shapes on inner layers and create resist or coating shapes on layers which are embedded in the board stack. When you change the layer of such a Component, you would want all the associated shapes to follow the pads.

In a similar way to the conventional technology, you can create additional non-electrical layers which are associated with an inner electrical layer. You can also define an inner layer to be one which embedded Components are allowed to be placed on. For embedded Components to work correctly **it is essential to define the design technology correctly before you begin to define the footprints**.

If you are using embedded Components, it is likely that you may also want to use blind or buried vias which are only partly drilled through the board. You can do this by defining **layer spans** which define the spans which can be drilled.
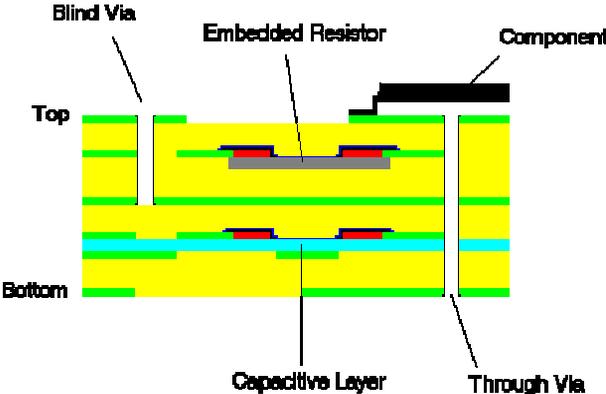
Printed Resistors

A printed resistor consists of two pads, connected by resistive material. Depending on the manufacturing method, a resist mask or encapsulating coating may also be required. The technology would therefore require an electrical layer with associated non-electrical layers for the resistive material, resist and coating shapes. If you wanted to place these embedded resistors on electrical *inner layer 1* you would first define this layer as one which **Can Have Associated Layers** and as one which can **Allow Components**. Then create the appropriate non-electrical layers, and **Associate** them with the inner layer. It would be appropriate to define the layer class for these non-electrical layers as **Essential For Manufacture**, this would prevent the Component from being added to electrical layers which do not have these associated layers present. Using this technology, a footprint can be created with the appropriate shapes. This footprint should be marked as an **Embedded Component** (on **Save To Library**), this tells the system that it is a footprint which can be placed on inner layers. If more than one set of appropriately defined layers exist, **Change Layer** can be used to move the footprint through the layer stack.
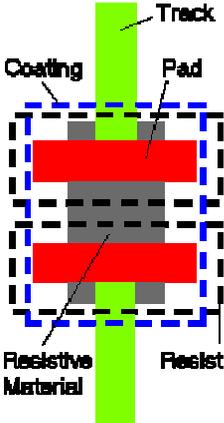
Planar Converter

A second example of an embedded Component shows a planar converter.
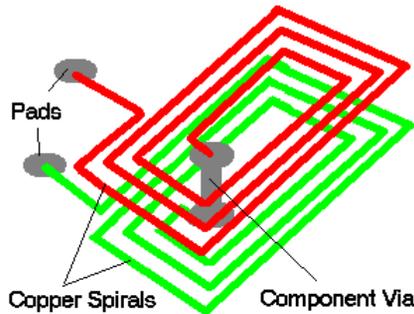
This Component exists on the outer layers and may have a physical body applied to the outer layers. However, part of the footprint consists of copper spirals which are connected by a Component Via, effectively joining the two footprint pads. Although the pads are joined we would want to connect them to different nets. We can do this because it is possible to mark the copper spirals and Component Via as not checked (using **Properties** in the footprint's **Component Via** entry).

A Component Via is a fixed part of a footprint, but unlike a pad or mounting hole, it does not have a pin number and it exists on a layer span, like a normal routing via. If we defined this footprint but did not mark it as embedded, mirroring the Component would not swap the inner electrical layers. By defining the footprint as embedded, you can mirror the Component and all the inner layers will swap as expected.

Overview of Process

To use embedded Components you must follow this process:

- Edit or create the **Technology** file that you wish to use. This will be used for the design and to create the footprint.

- Add **Layers** and **Layer Classes** to support embedded component technology. For layers that are critical to the final manufacturing process, select the **Essential For Manufacturing** check box in **Layers Classes**.

- Add **Inner Electrical** layers and allow **Associated Layers**. Also enable the electrical layer to **Allow Components** (embedded Components).

- Add **Inner non-electrical** layers to support the embedded component manufacturing process.

- Create the footprint and save it using the **Embedded Component** check box on the **Save to Library** dialog.

Application of new functionality

To enable this new functionality, Pulsonix V4.0 has been modified in particular areas:

Layers dialog

Changes have been made to the **Layers** dialog grid to support the addition of inner layer '**sets**'. Pulsonix already has <Top Side> and <Bottom Side> layer sets, and you can now add <Inner> layer sets.

	Name	Associated Layer	Class	Side	Bias	Net	Material
		Wires Top	Wire Link	Top	None		
		Silkscreen Top	Silkscreen	Top	None		
	Top		Electrical	Top	X		
		Solder Mask Top	Solder Mask	Top	None		
		Paste Mask Top	Paste Mask	Top	None		
Y		Pin Names	Non-Electrical	Top	None		
Y		Resistor Coating	Coating	Inner	None		
		Resistor Pad Resist	Resist	Inner	None		
Y	Resistor		Electrical	Inner	None		
Y		Resistor Material	Resistor	Inner	None		
Y	Inner Copper 2		Electrical	Inner	None		
	Capacitor		Electrical	Inner	None		
		Capacitor Material	Capacitor	Inner	None		
	Inner Copper 4		Electrical	Inner	None		
	Bottom		Electrical	Bottom	Y		
		Silkscreen Bottom	Silkscreen	Bottom	None		
		Solder Mask Bottom	Solder Mask	Bottom	None		
		Paste Mask Bottom	Paste Mask	Bottom	None		
		Wires Bottom	Wire Link	Bottom	None		
	Documentation		Documentation		None		

The **Layer Names** column has been split into two columns - **Name** and **Associated Layer.** The second column is used to hold names of layers that are 'associated' with a layer set. So for example, a Top layer might have associated with it, Top Silkscreen, Top Paste, Top Wire etc.

Dark horizontal lines have been added to split the rows into groups of layers associated with the same layer set.

Y	Resistor		Electrical	Inner	None
Y		Resistor Material	Resistor	Inner	None
Y	Inner Copper 2		Electrical	Inner	None
	Capacitor		Electrical	Inner	None
		Capacitor Material	Capacitor	Inner	None

The **Side** column is now read only and does not allow changes to be edited directly in it. Instead you should using the **Edit** button and dialog.

Cells which are considered to be not relevant or read-only are greyed out.

Within the dialog, changing order of rows is restricted to reduce chance of invalid situations.

The number of electrical layers are shown at the bottom of the page.

Number of Electrical Layers: 6
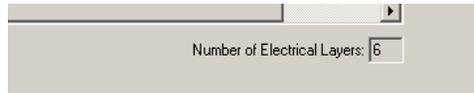
The **Edit Layer** dialog has been changed, it now supports associated layers. Restrictions are now imposed to what you can change based on layer type.

Edit Layer

Name: Resistor Material

Class: Resistor | New Class...

Type: Non-Electrical

Layer Association:

Associated With: <Resistor>

Side: Inner

Electrical Details:

Routing Bias: None

Power Plane Net:

Construction Details:

Material: | New ...

Thickness: 0.0 | Embedding: None

OK | Cancel
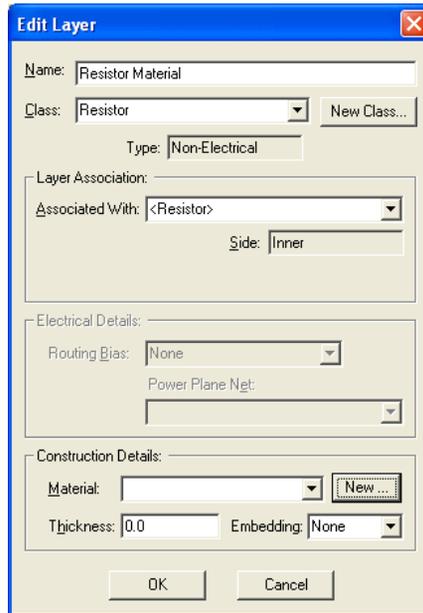
You can now set up an inner electrical layer to have associated non-electrical layers (using **Associated With:**). Non-electrical layers can be put on an inner layer. The inner layer set is the name of the inner electrical layer in angled brackets e.g. <inner 4>.

You can mark which layer sets you can place Components on by using a check box on the electrical layer – **Allow Components**. You can also mark whether the Component must be **mirrored** on that layer set.

Layer Association:

☑ Can Have Associated Layers

Side: Inner

☐ Allow Components ☐ Mirror Components
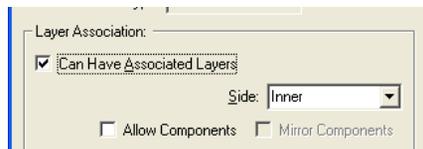
The ability has been added so you can select **Add Material** and **Add Layer Classes** directly from the edit layer dialog.

The **Layer Class** dialog has been changed for easier use. An **Essential For Manufacture** checkbox has been added for non-elect layer classes. Checking this box means that any items in the footprint on a layer using this class are essential for the manufacture of the Component. You will not be able to add this Component to a design that does not have the appropriate layers.
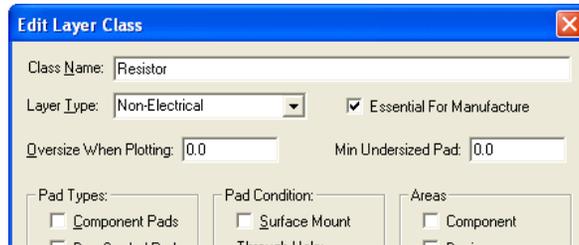
This is needed as you should not be able to add a embedded resistor that has its resistive material etch outline defined, on any layer in the design that does not also have an associated layer for the etch outline to live on.

The **Embedding** drop-down allows you to specify the direction of any embedded layers with the layer directly above or below it. This is used as reference for manufacturers and is not directly used in Pulsonix.
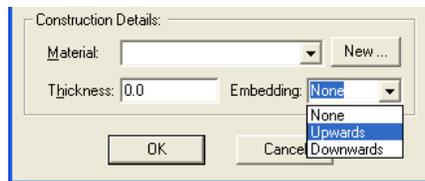
Footprints for Embedded Components

Embedded **footprints** can be defined in the footprint editor with pads and shapes etc. on inner layers. Footprints can be marked as **Embedded Component** when using the Symbol name dialog or when adding it to the library.
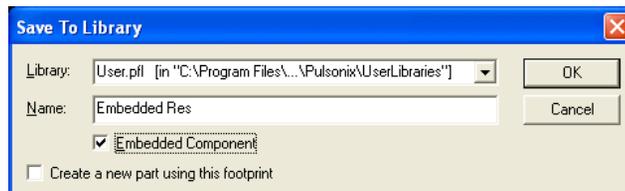
Insert Component

When adding a embedded Component to a design, it will try to place the device on the top most layer used in the footprint. An embedded Component in a PCB design will show its layer on the status bar. If the layers it was defined on don't exist in the design, it will be placed on the top most layer set that contains all the **essential** layers used in it.

Using Change Layer

You can select an embedded Component and use the **Change Layer** option to set its layer in the design. **Mirror** will not be available as a mirror is automatically performed when changing layers to a layer that is marked to **Mirror Components**.

You cannot change an embedded Component to a layer set that does not have an associated essential layer to place its critical items on. Copper and Pads are assumed to be essential.

Component Vias

You can now add **Component Vias** to footprints. These can be added to any footprint, not just embedded Components. These can have the flag **No spacing check within footprint** set so that they can join two pads (for a embedded resistor via for example).
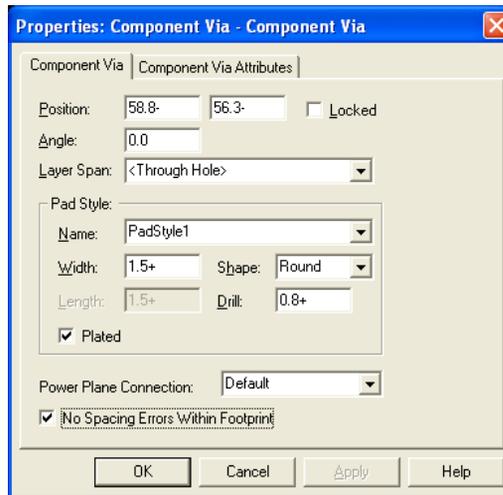
Properties dialog

Properties of an embedded Component shows if a Component is embedded and allows you to edit its **Layer** instead of its mirror flag. You cannot edit mixed embedded and normal Components using a multiple selection, you would have to edit these individually.

Layers Report

The **Layers Report** will report a reference layer for the inner layer side if there is one. It also reports if a layer class is essential for manufacture.

Report Maker

New fields have been added to the **Report Maker** to support embedded Components. *See above in the General Options sections and Report Maker for more details.*

Chapter 6. Spice Simulator Changes

Spice Update

For users who have the Pulsonix Spice A/D mixed mode simulator option, there have been a number of significant changes.

Spice Engine Update

Speed Increase

The Spice 'engine' itself has been updated to be significantly faster (between 50-150% simulation speed improvement depending on the circuit) and more accurate with convergence improvements.

CircuitSim90 Benchmark Netlists

Of the industry standard CircuitSim90 benchmark netlists (57 in all), Pulsonix Spice now runs all 57 to the point at which they converge and produce an accurate result.

Simulation Menu Changes

The simulation menu has been modified and updated to accommodate new functionality and to improve the usage of existing functionality.

Version 3.1 Simulation menu
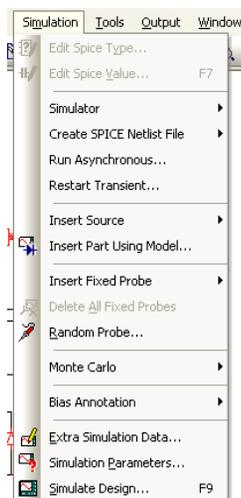
Version 4.0 Simulation menu

Monte Carlo Menu

The **Monte Carlo> Run** option has been removed. To support this there is an interface change, it is now just one of 6 sweep and step modes. This option is also much faster. (for .OP it can be up to 100 times faster!!).

The new **Plot Histogram** option now uses the **Performance Analysis** dialog in the simulator.
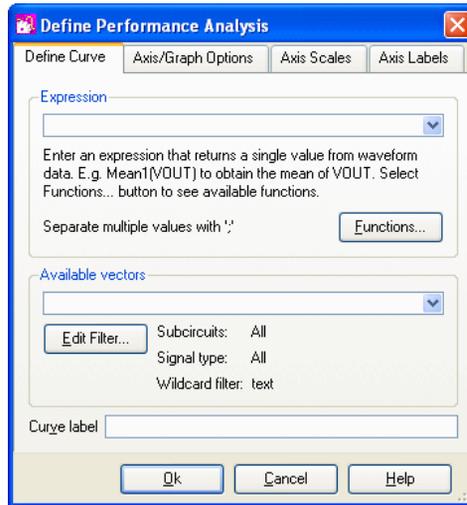
Monte Carlo Analysis

Tolerance Implementation has changed. Tolerances are now specified using expressions containing a distribution function. The old methods is still valid and can be used. This now an extension of the analysis line in netlist.

Step Parameter

Step Parameter has been removed from the Schematic **Simulation** menu and is now done using **multi-step analysis**.

Bias Annotation

On the **Bias Annotation>** menu, the **Insert Bias Current Marker** has been added to annotate the schematic with the results of a DC Operating Point analysis. This displays the current on the pin, that the markers are attached to. The values shown on the markers are updated each time you simulate. This has also been added to the **Spice** toolbar and **Misc.** component bar. To support this, a new **Bias Current Marker** Part and symbol has been created.

Run Asynchronous

There is a new option to run simulation in background – **Run Asynchronous**. Resulting data can be read into simulator in the future to analyse results.

Restart Transient

New **Restart Transient** option is used to continue transient simulation to run to a later stop time.

New Simulator Sub-menu Commands

The Simulator sub-menu has been changed to now allow the control of the simulator from within Pulsonix. Commands have been added to **Abort**, **Pause** and **Resume** the simulator.
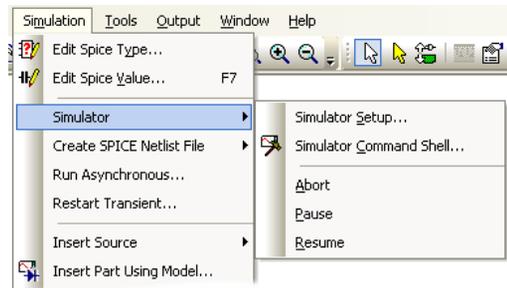
New Fixed Probes

There are a number of new fixed probes available on the **Fixed Probe** sub-menu:

- Inline Current Probe.

- Keep Inline Current.

- Bus Probe

- Bode Plot Probe

Also, the **Keep Voltage** and **Keep Current** probes have been moved into the **Fixed Probe** sub-menu.

New Specific Source Devices

Each source type pair (Voltage and Current) now has its own dialog. The old Sources is renamed to **Universal**.

- AC Voltage Source & AC Current Source

- DC Voltage Source & DC Current Source

- PWL Voltage Source & PWL Current Source

- Voltage Waveform Generator & Current Waveform Generator

A new menu item, **Insert Source** has been added to the **Simulation** menu.

Random Probe Changes

Probe Arbitrary Expression has been renamed to **Add Curve...**

New Random Probes have been added for **Bus Probe** and **Performance Analysis**.

The **Fourier Probe** has been added - **Probe Voltage (Quick Fourier), Probe Voltage (Custom Fourier)** and **Add Fourier Plot.**

Simulation Parameter Dialog Changes

Monte Carlo and Multi-Step Analysis has been added to all modes on the **Simulation** dialog.
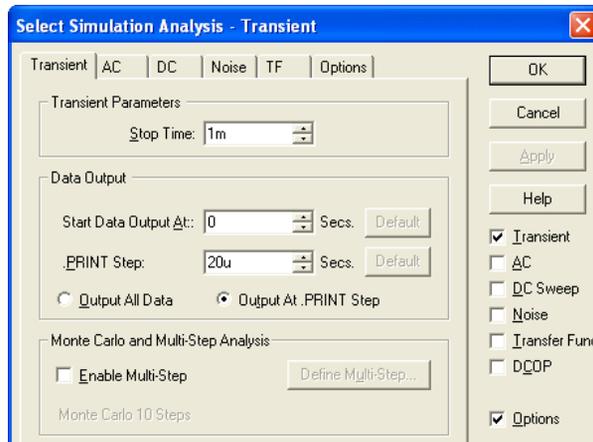
Sweep Modes have been added to **AC**, **DC**, **Noise** and **TF** modes. This is enabled through the **Define Multi-Step** button once the **Enable Multi-Step** check box has been selected.
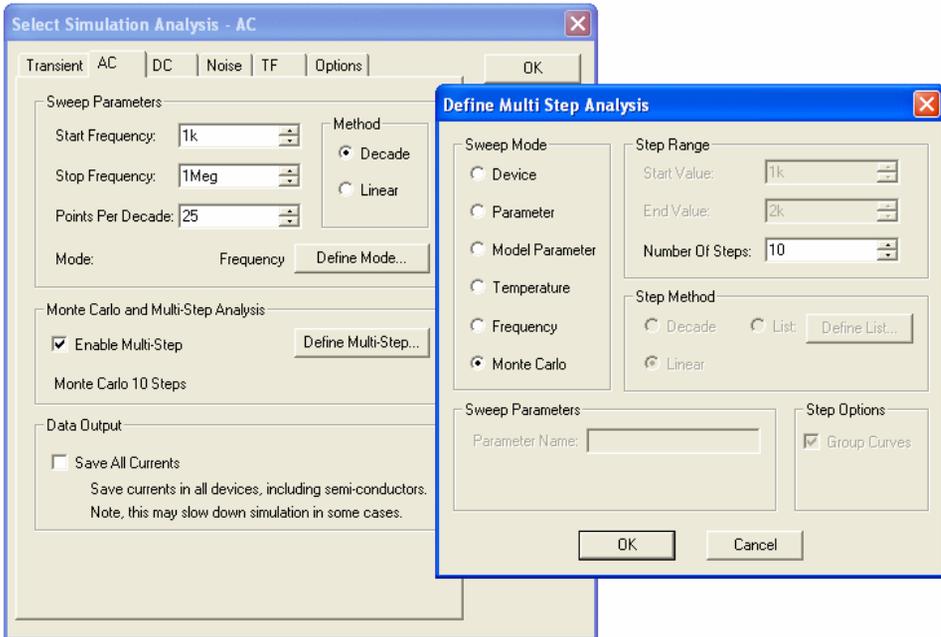
On the **Transient Analysis** dialog, **Min Time Step** and **Fast Start** are available using the **Advanced Options** button.
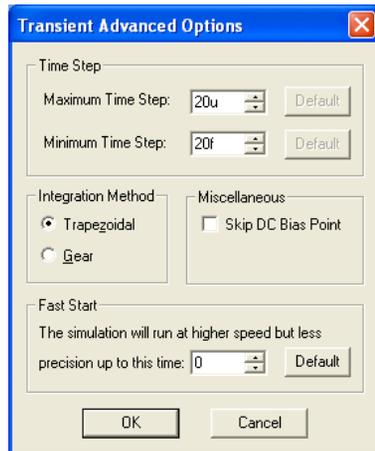
On the **AC** page, **Save All Currents** has been added to **Data Output**.
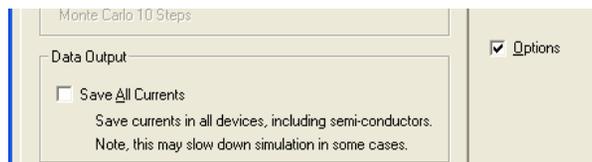
On the **DC** page, **Decade Vs Linear Sweep** method has been added.
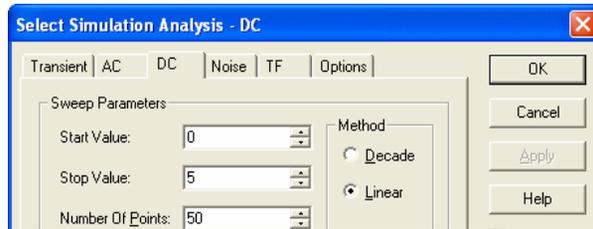
On the **Options** dialog - **Initial Condition Force**
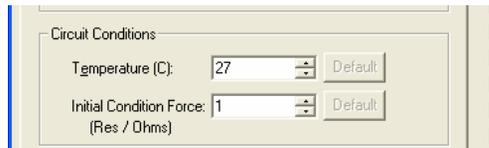
Under **Circuit Conditions, List File Output** a check box to **Expand Sub-circuits** has been added along with the **Parameters** output.

Under **Monte Carlo,** you can now select **Set Seed** using the check box.

A new mode is available **TF** - **Transfer Function**.
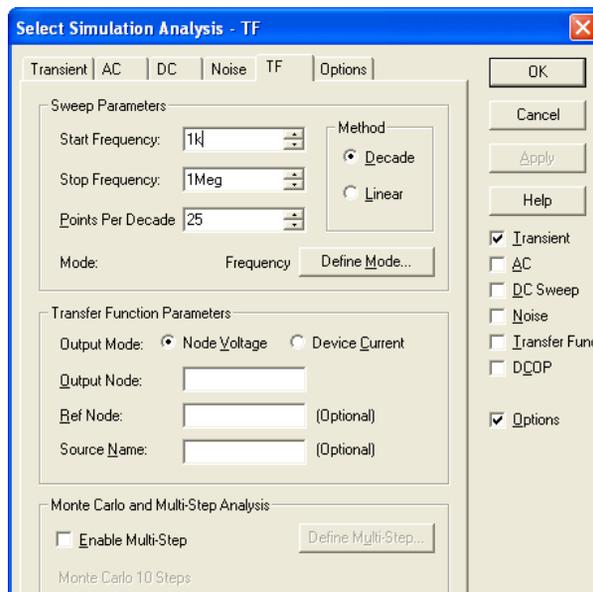
New Analysis Sweep Modes

.AC and .NOISE analysis can now sweep more than just frequency.

Transfer Function analysis. Calculates the gain from every source on the circuit to a nominated output.

Monte Carlo is now a sweep mode.

Second level application to provide **stepped** analysis.

Interactive parameter stepping is replaced by **Multi Step Analysis.**

Other Interface Changes

Select Model dialog

Using the **Select Model** dialog (from the **F7** Component selection), you can preview the model details, (to run this option you need the simulator running).
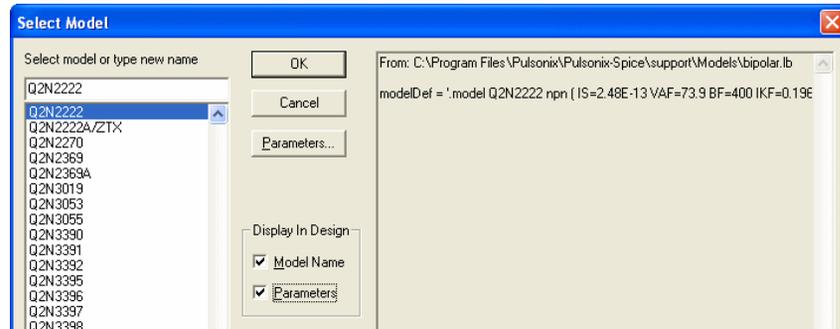
Add Part Using Model

Now has ability to preview either Model or Part details (needs simulator running).
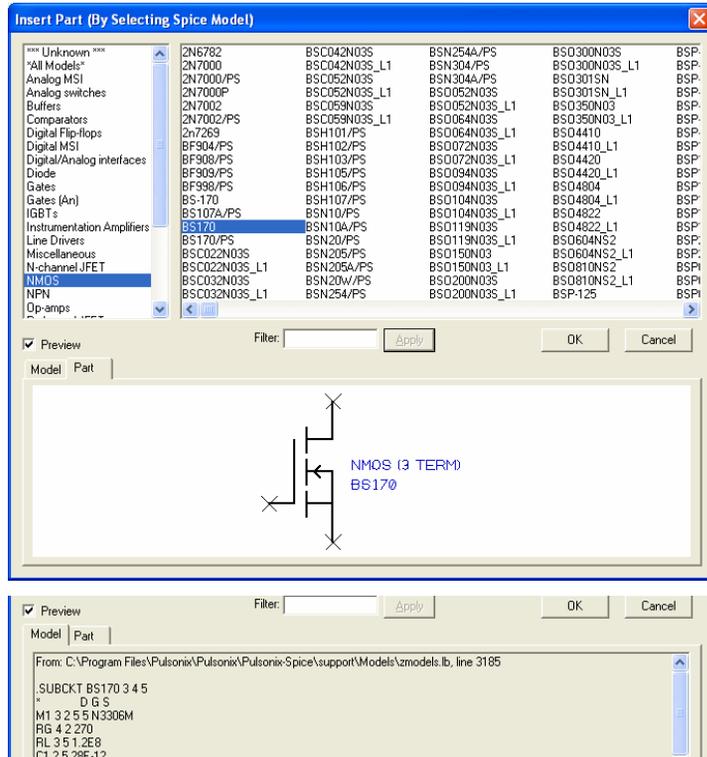
Potentiometer changes

When adding or editing (using F7) Potentiometer Vales, you are presented with a dialog to set the wiper step value and switch to run the simulator if the wiper position changes.
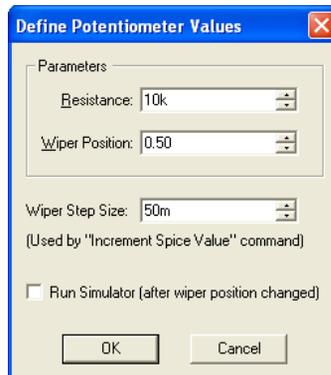
A new command has been added for **Increment Spice Value** and **Decrement Spice Value** used on passive devices and potentiometer. Shortcut key commands have been added using Shift-PageUp and Shift-PageDown

New/Changed Toolbar Popups

There are new toolbar popups named **Magnetics** and **Analog Behavioural.** The **Functions** toolbar popup is now named **Digital Generic.**

Part Editor Changes

The Part editor has been changed to support the new devices (listed over). This is available on the **Edit Spice Type** dialog and **Edit Template**.
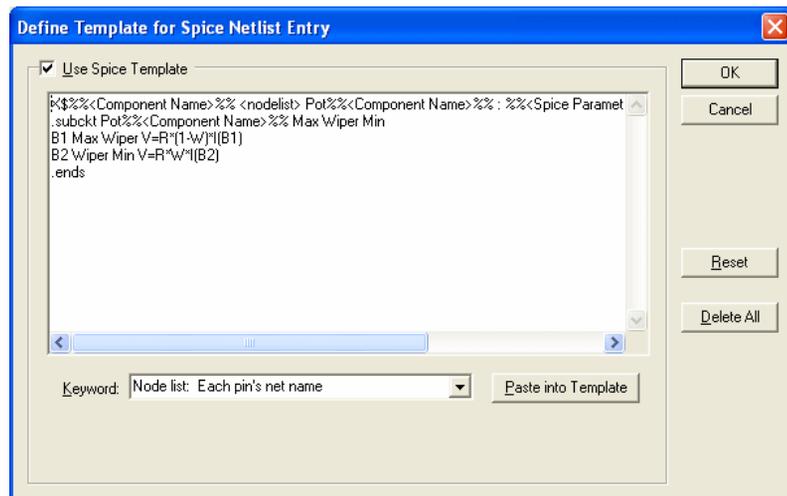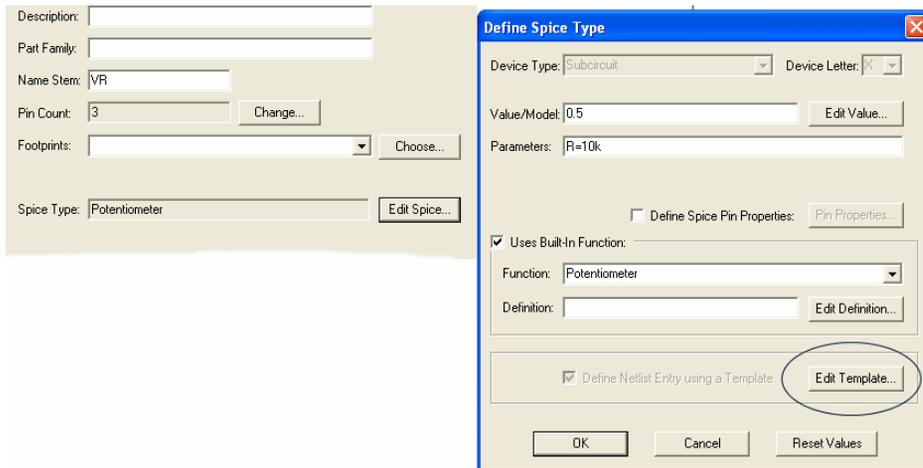
New template driven device **keywords** have been added to support new models:

```
<param["parameter name"]>
```

This extracts the value of the named parameter from the SpiceParameters or SpiceValue attributes.

```
<if["parameter name"]> ..... <else> ..... <endif>
```

This extracts the parameter value like above and if it is defined and not "0" or "false" then it writes out the template commands before the <endif> keyword. If not defined, it writes the template commands between <else> and <endif>. Note: the <else> is optional and you cannot nest if commands.

        <repeat[limit]> .....   <endrepeat>

This repeats the template commands between the keywords as many times as specified by the "limit". Limit is a parameter name to retrieve the number from or a direct integer. You must not nest repeat commands.

        <count>

This is used in a repeat loop to output the current loop count (starting from 1).

        <add[arg1][arg2]>

arg1 and arg2 are any number, or a parameter name to get the value from, or <count>. Adds two numbers and writes result to netlist.

Can also use 'sub', 'mul' and 'div'. If both numbers are integers, then the result is an integer apart from 'div'.

Arbitrary Source Improvements within Expressions

A conditional function IF() has now been added.

You can now put circuit variables (i.e. references to voltages and currents using V() and I()) into parameter expressions that are used in arbitrary source expressions. So you can do something like ".PARAM Vmult = { V(a)*V(b) }" then use Vmult in the defining expression for an arbitrary source.

DDT and SDT functions for differentiation and integration respectively. These work in AC analysis as well as transient.

Arbitrary source optimisation algorithm. Analyses arbitrary source expressions for common sub-expressions and factors. By default this is enabled only for expressions that use

.FUNC definitions. Can be switched on globally with an option setting. This feature improves the running speed of the Infineon power models, sometimes dramatically so. Compatible with the PSpice equivalent.

Floor function is supported. When used this returns an argument truncated to the next lowest integer.

.PARAM Command

The PARAM command can now be used in library models and sub-circuits. It defines a simulation variab**le for use in an expression. Expressions** may be used to define device parameters, to define model parameters, for arbitrary sources and to define variables themselves.

Support of .FUNC Function

> Pulsonix Spice now supports the .FUNC function. .FUNC defines a function that can be used in a model or device parameter expression, a parameter defined using .PARAM or in an arbitrary source expression.

Improved .KEEP

> Includes wildcards, making it possible to specify all the voltages or currents from a specified sub-circuit to be saved. Text filters using wildcard characters '*' and '?' may now be used with .KEEP

Changed/Updated Devices

Changed Devices

Capacitor	New dialog with IC voltage section. New params: DTEMP M
Ideal Inductor	New dialog with IC current section. New Params: M
Resistor	New Params: ACRES DTEMP L M SCALE W
Diode & JFET	New Params: DTEMP M
Mosfet	New Params: DTEMP

New Devices

Ideal DC Transformer	New device like Ideal Transformer, but non inductive so simpler dialog.
Infinite Capacitor	

New Parameterised Devices

> This presents you with the **Edit Device Parameters** dialog to edit the parameters.

> **Lossy Inductor**

> **Transmission Line (Lossless)**

> **Electrolytic Cap (Level 1-3)**

> **Electrolytic Cap (Level 4-5)**

> **Electrolytic Cap (ladder model)**

> **Transmission Line (Lossy RLC)**

Parameterised Opamp

Switch With Hysteresis

Peak To Peak Voltage Source

New "Saturable Transformer / Inductor" device

This replaces the **Saturable Inductor**, **Saturable Transformer** and **Saturable Transformer 2 secs** devices.

One dialog now maps to different symbols depending on the number of primaries and secondaries have been defined.

Simulator Changes

- Support for HSPICE devices, support includes - Netlist Compatibility, Model Compatibility and Process Binning.

- parameters for level 3 Diode

- .PARAM library support

- Junction GMIN DCOP Convergence Method

- Netlist pre-processor provides conditional devices using .IF and repetition using .WHILE.

- Digital Simulator: Two new parameters for digital devices that define output resistance for positive and negative slopes.

- Digital Simulator: Wire Delay. An additional capacitance can be added to each output related to the number of inputs that it drives

- Subcircuit currents are now saved as actual data rather than aliases when it is efficient to do so. This reduces the size of the data file and improves plotting speed in situations where only top level data is stored.

- The 'M' multiplier as an instance parameter has been added to more devices including capacitors, diodes and JFETs.

New Resistor Parameters

- ACRES – different value for AC than DC

- Flicker Noise parameters are supported as part of Noise vectors.

AC Currents

Current in AC analysis is now available for virtually all devices including BJTs, JFETs, MOSFETs, diodes and all SPICE passive devices. Note however that AC current has to be explicitly enabled for semiconductor devices. This is because outputting current data is expensive in simulation time.

Simulator Devices

Lateral PNP. Use device name of LPNP. The lateral pnp connects the substrate capacitance to the base. Also, the substrate diode DC characteristics are now implemented. (ISS non-zero)

PSpice JFET parameters. All the PSpice JFET parameters are now implemented.

HSPICE level 3 diode implemented. This has a number of additional parameters notably for sidewall capacitance.

Full support for Infineon power device models, by virtue of the addition of .FUNC and the DDT function. Also arbitrary source optimisation algorithm improves running speed of these models, in some cases dramatically so.

Drag n Drop models

You can now drag models or library folders onto the **Command Shell** window to add new libraries to Pulsonix Spice.
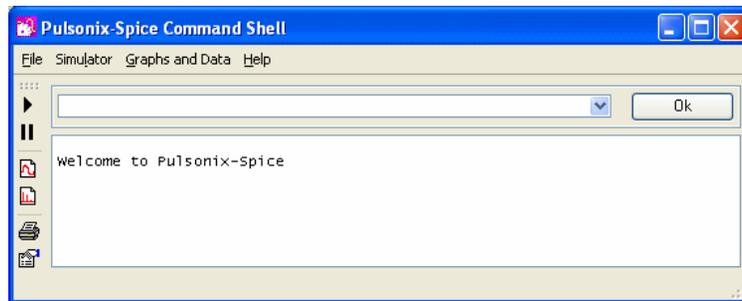
Dropping the folder from the Windows Explorer dialog onto the **Command Shell** window displays the **Select Library** dialog.
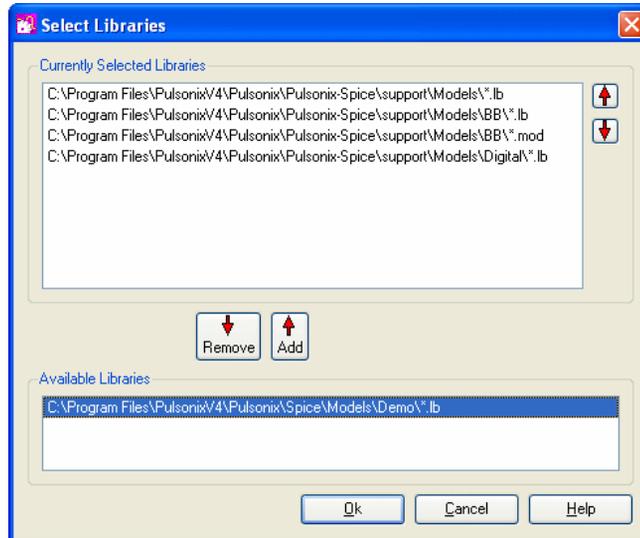
Much better Graph dialog editing

Below is a summary of the new features found within the new simulator interface:

All dialogs used can now be resized

Windows XP themes are now supported when running Windows XP

The command shell now has a small toolbar

Toolbars and graph legend panel are dockable. That is they can be moved to new locations or undocked completely from their parent window.

Graph Annotation

There is now a range of annotation objects that can be placed and freely moved on a graph in order to annotate it for documentation purposes. These include a curve marker, free text, boxed text and captions. You can also add legend into the graph for printing.

Cursor Enhancements

The measurement cursors are now cross hairs instead of the small cross used before. (The older style is still available if preferred). The cursor position and separation display is now displayed on the graph itself as an arrowed dimension. (The older method is still available and may be preferred on slow machines) The method of moving cursors has been improved. Previously you could only move the cursor along the horizontal axis while the cursor tracked its curve. This made it hard to position the cursor on a vertical edge. Now you can move the cursor along the vertical axis or alternatively you can pick it up and place it where you want it including on another curve. When released it will snap to the nearest point on the nearest curve.

New features in the Graph Windows

Graph windows now have fixed menus at top of program.

Graph windows will now scroll vertically to accommodate more grids and digital axes.

Update Curves Feature

This instructs Pulsonix Spice to re-plot the curves in a graph sheet using the latest simulation data. An option to add rather than replace the new curves is provided.

Define Curve Improvements

A Wildcard Filter is now available for vectors selection. In the Define Curve dialog box, a new method of filtering the available vectors has been added. This uses a wildcard specification that is related only to the full name of the vector. That is, it doesn't care whether it is a voltage, current or digital signal.

The history drop down box for expressions in the **Add Curve** dialog is now saved between sessions.

It is now possible to highlight curves within graphs.

Fixed Probe Improvements (.GRAPH)

This can now efficiently incrementally plot expressions that satisfy certain conditions. This was possible in the earlier release but was inefficient.

Copy and Paste

Its now possible to move or copy a curve to a new graph sheet using a copy and paste curve feature.

Tabulated ASCII data, for example from a spreadsheet, may be pasted directly into the waveform viewer.

Axes

You can now switch between log and linear axes after the graph has been created.

You can now adjust the height of digital axes. See the **File** menu and **Options**, **General...** select **Graph**, **Probe**, **Data Analysis Grids** and digital axes may now each be reordered arbitrarily.

The zoom to fit y-axis feature now operates on only the displayed portion of the x-axis instead of the whole x-axis as before. So, for example, if you wish to view a small oscillation at the top of a pulse, you can adjust the x-axis limits appropriately then zoom to fit y-axis to achieve optimum magnification.

You can now re-order grids in the Y direction (separate digital and analogue).

Saving

You can now save a graph to a binary file and recover it later. Everything is saved including cursor positions, annotations and all the data for each curve.

There is also a facility to save the graph to a windows metafile for importing to other applications, such as Microsoft Word.

Simulation data may be exported in SPICE3 raw format. This is useful for interfacing to other applications including third party waveform viewers.

You can now read in SPICE3 raw files for importing data from other simulation tools. The data is written out to a .SXDAT file in an efficient format. This allows the handling of very large files. Many simulators can output data in this format.

Define Fourier plot

A Fourier spectrum of a signal can be obtained in a number of ways. You have a choice of using the default settings for the calculation of the Fourier spectrum or you can customise the settings for each plot. The following menus use the default settings:

Graph menu and Measure Plot Fourier of Curve

Graph menu and Measure Plot Fourier of Curve (Cursor span)

The **Define Fourier** option on the **Graphs and Data** menu displays the following dialog:
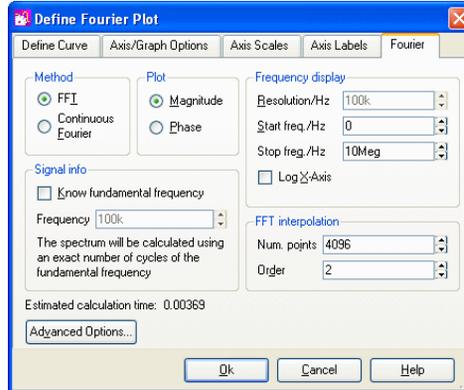
You will see a dialog similar to that shown for **Plotting arbitrary expressions** will be displayed but will include a **Fourier** tab. Click on the this tab to display the **Fourier** analysis options.

Fourier Analysis

A wide range of options are now available for Fourier analysis including:

- Specify time range to apply transform

- Specify fundamental frequency so that an exact number of cycles is used for calculation

- More windows

- Specify start and finish frequencies

- Specify log x-axis

- option to plot phase instead of magnitude

In addition there is now an alternative algorithm to the **FFT**. This is **Continuous Fourier**. This method is much slower but has the advantage of not suffering from aliasing and therefore is a better method when applied to signals with high frequency content such as narrow pulses.

New Models

New models have been added to the Pulsonix Spice product libraries.

- 390 new MOSFETs, Diodes and IGBTs have been added to the IR library

- 85 new Opamps have been added to the LTC library

- 140 new Transistors and MOSFETs have been added to the old Zetex library to make new Zmodel libraries.

New Spice libraries have been added for the following:

- New Infineon library containing 450+ Discrete MOSFET models

- Magnetic Core libraries containing 50+ models

New Components/Symbols have been added to support the new models supplied.

New Features for Version 4.0 Build 2569 Issued Sept 2006

Safe Operating Area (SOA) Testing

Overview

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

Defining SOA in Pulsonix

To use SOA testing, you must do two things:

1. Define the SOA limits for the models or devices you are using.

2. Enable and configure SOA testing.

Item 1. above is covered below in the section on .SETSOA and also the LIMIT parameter described under .MODEL.

Item 2. above is covered in the .OPTIONS section below.:

Both of these commands can be placed in the netlist by using the **Extra Simulation Data** option in Pulsonix.

Defining Simple Limit Tests

In the next release we will include fixed probes to perform some simple limit tests, but for now you can use the .SETSOA commands to define these tests to report the following:

1. Over and under voltage on a single net

2. Over and under current on a single device pin

3. Over and under differential voltage on a net pair

This can be done using:

.SETSOA Label="*your label*" expression=(*minimum limit, maximum limit*)

Where expression is "net name", "pin name" , or "net1 name – net 2 name" for the reports mentioned above.

For example:

.SETSOA Label=Watch1 "Q1_C" =(-1, 1)

Each of these commands has three parameters that specify:

1. The minimum limit. Use a large negative number (e.g. -1e100) if you don't wish to specify a minimum limit.

2. The maximum limit. Use a large positive number (e.g. 1e100) if you don't wish to specify a maximum limit.

3. A label.

Setting up SOA Testing

To enable SOA testing, you need to add a .OPTIONS line to your Extra Simulation Data as follows:

.OPTIONS SOAMODE=<Mode> SOAOUTPUT=<Output>

Where <Mode> is "Summary" or "Full". In Summary: output mode, only the first violation for each SOA device will be reported. In Full output mode, all violations are reported.

For <Output>, choose where you would like the results reported. Use "msg" to report to the Command Shell message window, or "list" to add results to the list file. Note that writing results to the message window is a time consuming operation and it is recommended that you should not select this if you are expecting a large number of violations.

Running Simulation

Run the Transient or DC Analysis Simulation in the normal way using the additional .SETSOA and .OPTIONS commands in the Extra Simulation Data. If there are any violations, the results will be reported in the location you specified.

Full SOA Options Details

Option Name	Default value	Description
SOADERATING	1.0	Scales min and max values used in ".SETSOA" specification. This allows a de-rating policy to be globally applied to SOA limits.
SOAMODE	off	Controls the Safe Operating Area (SOA) test mode. See ".SETSOA" below for details on how to define a SOA test.

Can set to:

Off		SOA testing is not enabled. In this mode .SETSOA controls will be read in and any errors reported, but no SOA testing will be performed during the run.
Summary		SOA testing enabled and results given in summary form with only the first violation for each expression given being output.
Full		SOA testing enabled with full results given. Every violation will be reported in this mode.

Option Name	Default value	Description
SOAOUTPUT	list	Can be:
	msg	Results displayed in command shell message window, or console if run in "non-GUI" mode
	list	Results output to list file
	msg|list	Results output to both list file and command shell message window
	none	Results not output to either list file or message window.

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

Setting up SOA limits within a model

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

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

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

Advanced SOA Limit Testing

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

You would usually enter a .SETSOA control in the Pulsonix Extra Simulation Data option. See the Pulsonix Online Help for details.

.SETSOA Definition

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

[ALLOWUNUSED] [ALLOWWILD]

expr1=(min1, max1[, xwindow1])

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

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

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

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

instname	If specified the expression or expressions supplied in expr1 etc. are applied to the specified instance (e.g. Q23, M10, R56). In this case the expression may refer to node voltages and pin currents of the specified instance. See details under expr1, expr2...
modelname	If specified the expression or expressions supplied in expr1 etc. are applied to every instance belonging to modelname. In this case the expression may refer to node voltages and pin currents for each instance processed. See details under expr1, expr2...
expr1, expr2...	Expression to be evaluated and compared against minimum and maximum specs. This expression can access simulation results using access variables. The format and scope of these variables depends on whether MODEL, INST or neither is specified.
	If neither is specified, the expression can use the global access variables defined below:

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

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

Use the following values if MODEL or INST is specified. In each case (excepting the global access variable) the variable accesses a quantity for the instance being processed. With INST this will be the single instance specified by instname. With MODEL all instance belonging to the model specified by modelname will be processed.

Syntax	Function	Example
pinname	Current in pin	c - current in collector of transistor
Ipinname	Current in pin	Ic - current in collector of transistor
Vpinname	Voltage on pin	Vc - voltage on collector of transistor
n(pinname)	Voltage on pin	n(c) - voltage on collector of transistor
Vxy Where x =	Voltage between x	Vbc – voltage from base to collector

pin name 1, y= pin name 2. Both x and y must be single letters	and y.	
pow	Power in device	
param	Readback parameter	vdsat - 'vdsat' for MOSFET
#global_name	Global node voltage or pin current	#VOUT – voltage on net called VOUT #q23#c – current in collector of q23
paramname	Parameter defined using .PARAM	

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

min, max	Minimum and maximum values respectively. A violation message will be produced if the value of the associated expression is less than min or greater than max. Use '*' if the limit is to be ignored. E.g. (*, 15) will test a maximum value of 15 but the minimum value will not be tested. min and max values may be scaled using the SCADERATING .OPTIONS setting (see below for details). Currently, only constant values are accepted for min and max. Expressions are not permitted.
xwindow	Optional value specifies a minimum window that must be surpassed before limit violations are registered. For example if 10u is specified for xwindow for a transient analysis, then the limit must be exceeded continuously for at least 10uS before the violation is recorded.
ALLOWUNUSED	If INST or MODEL are specified, an error will result if no instances to be processed are found. If INST is specified the error will occur if instname doesn't exist. If MODEL is specified, the error will occur if there are no instances using modelname even if modelname itself is valid. This error will be inhibited if ALLOWUNUSED is specified
ALLOWWILD	If specified, wildcards can be used for modelname and instname. In this case Pulsonix Spice will search for all devices that match the wildcard specification. Use '*' to match any sequence of characters and '?' to match a single character.

New SOA Testing Script

GetSoaResults

No arguments

Return type: string

Returns the SOA results for the most recent simulation.

Return Value

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

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

Appendix A. Supplementary File Changes

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

Format Files

New format file examples are supplied on our web site and on the installation CD. These show examples of Report Maker features to guide you on creating your own. These are not installed with the product but can be accessed from the CD in the \Formats\Examples folder Installed format files are also available on the CD to enable you to restore default ones if they become overwritten, these are in \Formats\Installed

Standard Reports

New reports have been added for:

- **Differential Pairs Report**   A new report to support the new high speed cost option

- **Pin Network Report**   A new report to support the pin network feature

Changes have been made to:

- **Shortcut Keys Report**   Modified
- **Technology Report**   Modified
- **Critical Nets Report**   Modified

Updated Libraries

Existing Spice Libraries

The **spicemodels.lib** and **spiceexamples.lib** libraries have been removed and rationalised with the main **spice.lib** library. Only one Spice library is now supplied.

New Spice Models

New models have been added to the Pulsonix Spice product libraries.

- 390 new **MOSFETs**, **Diodes** and **IGBTs** have been added to the **IR** library
- 85 new **Opamps** have been added to the **LTC** library

- 140 new **Transistors** and **MOSFETs** have been added to the old **Zetex** library to make new **Zmodel** libraries..

New Spice libraries have been added for the following:

- New **Infineon** library containing 450+ Discrete **MOSFET** models

- **Magnetic Core** libraries containing 50+ models

New Components/Symbols have been added to support the new models supplied.

Standard Libraries

Some of the **MasterLibrary** files have been updated to fix problems with existing library items.

Design Examples

New PCB design files have been added to show examples of the new embedded component technology. These are basic examples to show the principle of layer construction, layer class and access to this technology with the Technology file.

- **EmbeddedComponentsExample1.pcb**
- **EmbeddedComponentsExample2.pcb**

New Spice examples are supplied in the \Examples\Spice folders.