Pulsonix Design System

V8.0 Update Notes

Contents

Chapter 1. Version 8.0 Update Supplement

Installing the New Version of Pulsonix

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

To install Pulsonix, insert the CD or double-click on the download executable and wait for a short time. The *Autorun* facility will start the installation procedure. Follow the on-screen commands from the install wizard. You can install Pulsonix 8.0 on top of your existing installation or alongside if you prefer; however, you do not need to uninstall the old version first.

Licensing

Version 8.0 requires a new license. If you are using a version of Pulsonix earlier than Version 8.x, you will require a new license, this would have been supplied to you under the terms of your maintenance contract.

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

New License Configuration File

To ensure that access to licensing settings is not prevented by issues with access permissions and user login, these settings are stored in a configuration file called **License.ini**. This file is created and maintained automatically by the installer and the applications, in a folder called **C:\PulsonixConfiguration**. If you prefer this file to be written to a different hard drive, you can create a **PulsonixConfiguration** folder on any hard drive up to drive letter **H** before running the installer.

This scheme was also available in build 7.6.5224, but will only happen if you create a folder called PulsonixConfiguration yourself, otherwise it will continue to use the registry as before. The V8.0 software will always use a file.

Windows 8 Support

As of October 2012, Microsoft have released Windows 8 to the market. Pulsonix 7.6 and Pulsonix 8.0 will both work under this operating system and can be fully supported under both 32 and 64 bit variants. To run V7.5 under Windows 8, please install the patch 4917 or later to ensure you are using a supported version. No earlier Pulsonix versions (pre-7.5 or earlier) can or will be supported.

Note: Beta or pre-release versions of Windows 8 cannot be supported.

Windows XP Support

Microsoft ended mainstream support for Windows XP in April 2009. In recognition of the widespread use of Windows XP and the relatively low take-up of Windows Vista, Pulsonix has continued to be supported on Windows XP.

In line with our Support Policy, and bearing in mind that Windows 8 is now available and Windows 7 has been shipping for over two years, this will be the last major release of Pulsonix to be supported on Windows XP. More information on support for different versions of Windows can be found on our web site under Support, O/S Life Cycles.

Features Added in V7.6

In the following feature descriptions, you will find some features documented that were included in a late release of 7.6. They have been documented here to highlight them for you.

Nudge Feature

Following the initial release of Pulsonix V8.0, a new **Nudge** feature has been added.

This allows you to move selected items by one grid step using the **Left**, **Right**, **Up** and **Down** keys (or 10 grid steps – using <Ctrl>-cursor keys). Online rule checking will still apply where appropriate.

This feature is enabled using the **Enable Nudge** feature on the context menu for selected items.

Interaction Options

From the Options dialog and Interaction tab, the **Enable Nudge** button also reflects the status of the option but allows you to preset it.
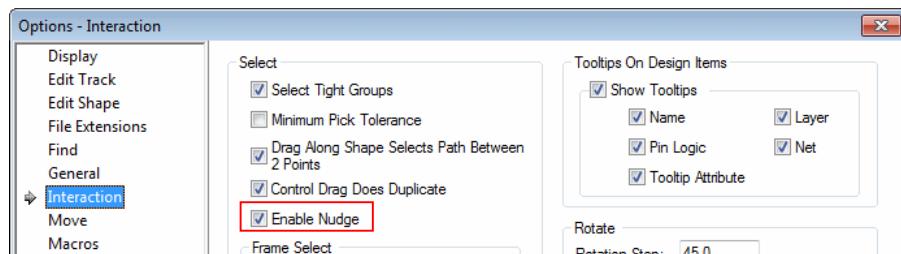
Report Maker – New commands

New commands have been added to support changes to the Differential Pairs option.

From within the **List of Diff Pair Gaps** command, you can now report **Allow Broadside Coupling** and **Allow Edge Coupling**, all to be used within **List of Differential Pairs**. For each Differential Pair gap you can output the **Layer Side**, **Layer Name**, **Area Name** and **Minimum Gap**.
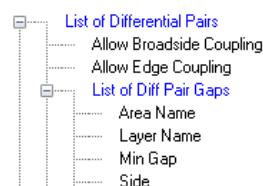
The reporting of **Allow Broadside Coupling** and **Allow Edge Coupling**, will show true or false as the status depending on the check box status in the Differential Pairs dialog.

Change to Type Offset option

When typing **relative values** in the **Type Offset** dialog, you can check **Remember previous value** so that the dialog retains the previous value and it can be used repeatedly. Otherwise the value returns to 0 each time.
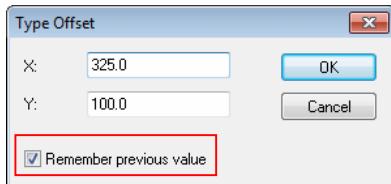
Arranging Columns in Attribute Editor

The columns can now be moved in the **Attribute Editor** and the new order saved. If a column can be moved, the appropriate **Move Right/Move Left** options are available on the context menu when you right click over the column name. Movement is achieved using this menu and not by dragging them.

Once a column has been moved an option is available to save the order for use the next time you select this option.

This feature is also available on the **Parts Attribute** page of the **Parts Editor** in the **Library Manager**. If the columns are moved you are prompted to save the order when the page loses focus.

Copy Tags check box

It is possible to copy and paste values to and from an external spreadsheet application such as Microsoft Excel perhaps to take advantage of the advanced features of a dedicated spreadsheet application. By default, the Attribute Editor adds embedded tags (<$UNDEF$> and <VIS>) in to the copied text to convey additional meaning for the undefined and visible attribute states and these will be visible when pasted in to an external spreadsheet. Text containing these tags can be pasted back into the Attribute Editor and will be interpreted appropriately. If you want to copy just the text without the tags you can uncheck the **Copy Tags** option, but if you paste the copied data back into the Attribute Editor any 'visible' and 'undefined' information is lost. Note that pasting a completely empty cell value from an external spreadsheet into an Attribute Editor cell will set that attribute to a 'defined but empty' state.
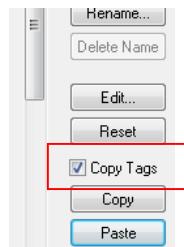
Changes to the Library Manager

Modeless Library Manager

There is a new option which allows the **Library Manager** dialog to become modeless during use. The option switch is located in the **Options** dialog under **General**.
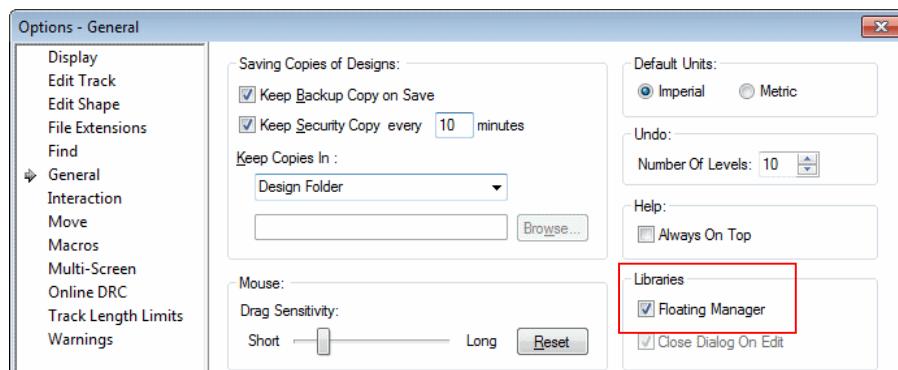
Once selected, items in the library and designs can be edited and the **Library Manager** will remain displayed.
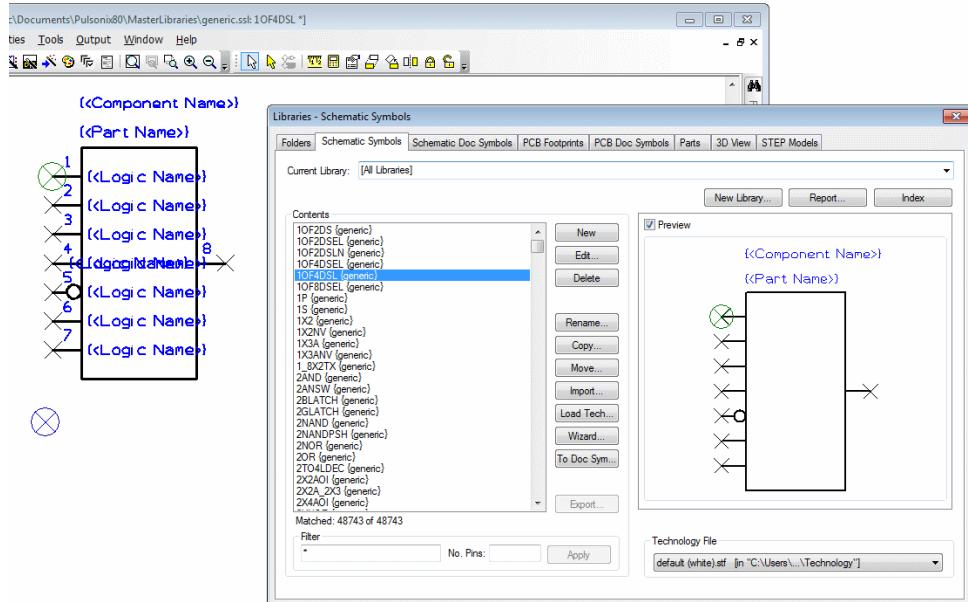
Ability to drop designs onto the Library Manager

Designs can now be dropped onto the **Library Manager**, this will invoke a dialog which allows you to save the Parts and Footprints/symbols into a specified library.

The top part of the dialog shows the three types of library, and a drop down list where you can choose the target library to which the relevant library items should be written. The check box alongside each of the library types controls whether or not that type of library item will be written. In the drop down list of libraries, there is also an entry [New Library] which if selected will prompt you to create a new library before copying starts.
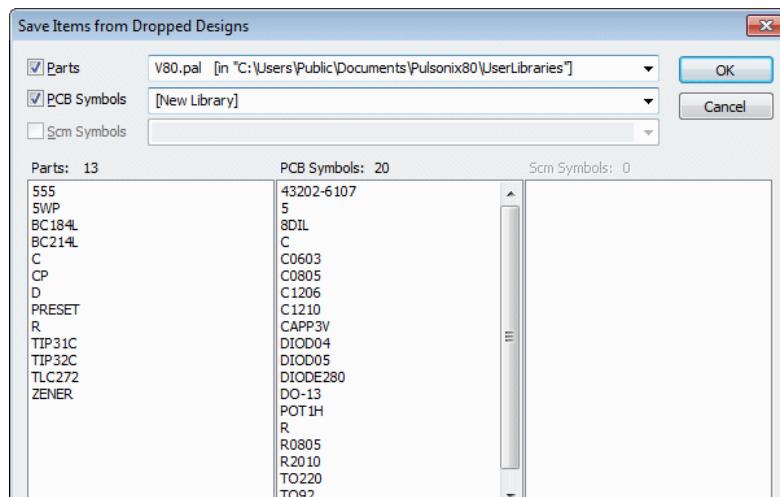
Underneath this are three lists, one for each type of library item. In the **Parts** list will be shown all the Parts that were selected in the design. The **PCB** and **SCM Symbol** lists will show all the symbols that are referenced by the components. Only one of the two Symbol lists will be filled as a design only contains one type of symbol.

Selecting items in the lists

If you click on any of the parts in the left-hand list, the PCB and Schematic symbols which are referenced by the parts you have selected will be shown selected. Conversely, clicking on any of the symbols will show you all the parts that reference that symbol.

This feature can be used to 'explore' the relationship between the Parts and Symbols you are going to save. This will not affect what is actually written to the target libraries, the 'writing' feature is only controlled by the check boxes at the top of the dialog.

Ability to drop libraries onto the application to create a library

Libraries can now be dropped onto Pulsonix, this will make a copy of them into the current library folder.

If, for example, you have a library in the MasterLibraries folder and want a copy of it in your own library folder, drop the library onto Pulsonix and a copy will be saved into the current Library folder selected in the Library Manager. In the example below, the folder c:\users\public\documents\pulsonix\userlibraries will be used.
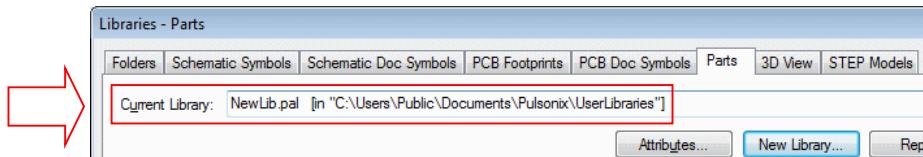
Save Component To Library from the Design

You can now save Parts and Symbols selected from a design into a library by using the **Save Items To Library** context menu. It can also be used to make a set of libraries containing some or all of the components and symbols in a PCB or Schematic design file. This can be useful for example if you have an archived design for which the libraries are no longer available.

With a component selected in the design (Schematics or PCB), right click and select the option from the menu.
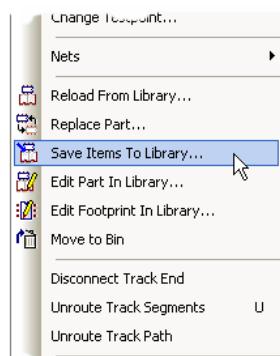
Once this option is selected, a dialog is displayed:
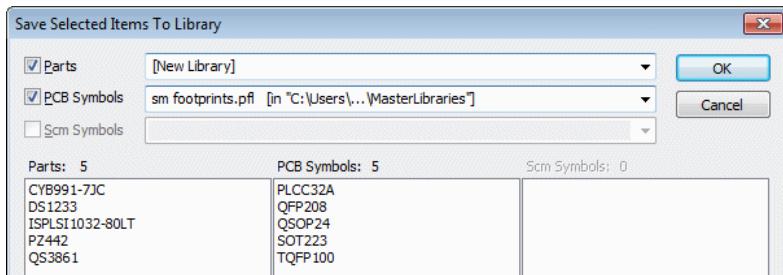
Check boxes for Parts plus the appropriate PCB or Schematic Symbol libraries will be selected. If you only want one or the other library item, uncheck the option.

Choose the library to save to item into using the drop down list box. This will display all the libraries available to you. If you wish to save it into a new library not in the list, select the [**New Library**] option from this drop down list.
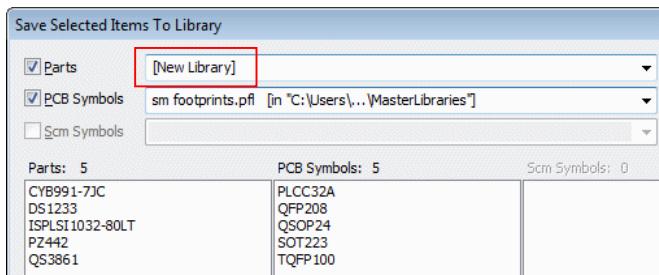
This will prompt you to create a new library before copying starts.
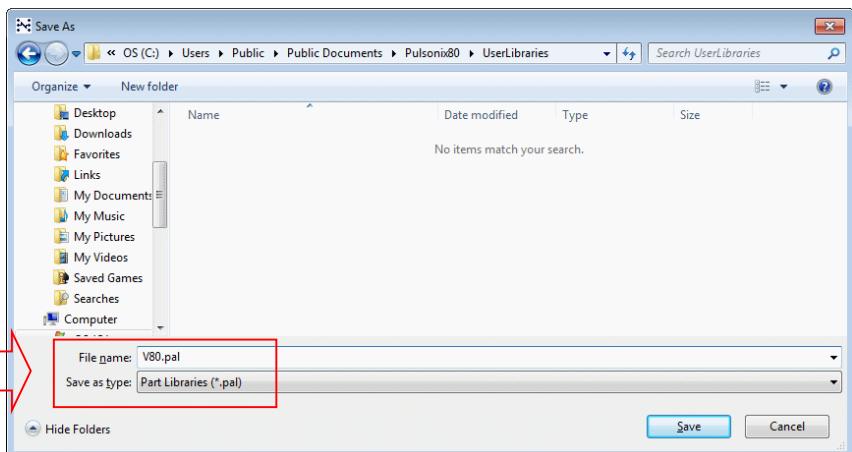
Underneath this are three lists, one for each type of library item. In the Parts list will be shown all the parts that were selected in the design. The PCB and SCM symbol lists will show all the symbols that are referenced by the components. Only one of the two symbol lists will be filled as a design only contains one type of symbol.

By selecting a PCB Symbol, in this example, you can view the relationship between the Parts using the selected footprint.

Additional Wildcard string matching

%P

The **%p** command has been added to wildcard strings which allows you to match SI unit prefixes for the International system of units.

%p - means an SI unit prefix (e.g. k). %P will also match upper case (in particular 'K'). Micro is 'u'.

For example, the %p wildcard could be used for a Validation check of component values, e.g. %i%p%i could be used for validation of a resistor value 2k2.

%^

The %^ command has been added to negate a match.

You can now do negation using the %^ command. This means the following string matches everything that would not have matched.

For example, %^AB will match all strings except AB

Another example could be, 'A%^BC' would match any string starting with 'A', followed by anything except 'BC'. Negation can be reversed again, so 'A%^BC%^D' would match 'A' followed by anything not 'BC' followed by a 'D'.

New Item Position Attribute

The **<Item Position>** attribute is now available as a built-in **attribute**. This reports the X,Y coordinate of the attached item using the current design units.

For example, the **<Item Position> attribute** could be added to a **Text Callout** which is attached to a component. This would then display the component position.
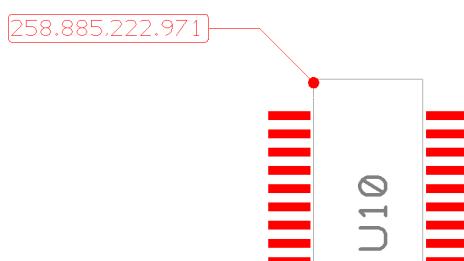
Additional Cursor Type – 45-Crosshair

A new cursor type of **45-Crosshair** was added to V7.6.5223. This is like the full screen Crosshair but at 45 degrees. This is available on the **Options** dialog under **General**.
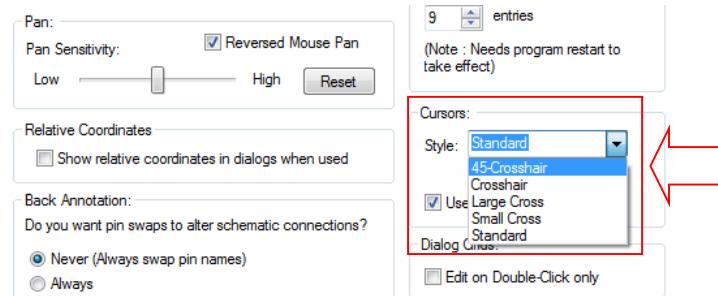
Changes to the Customise Dialog

The readability has been improved in the **Customise** dialog and the **Keyboard** page for use of longer command names. This new feature was also added to V7.6 5223

Hovering over long names, user defined **Toggle Layers** for example, will reveal the full name.
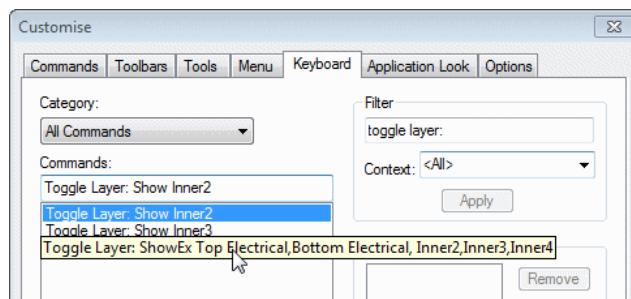
Changes to Drop-down Lists

Items that extend past the drop-down list width (for combo boxes) will now show a tooltip to show the full item text. Hover the mouse over the drop-down list box to reveal the name.
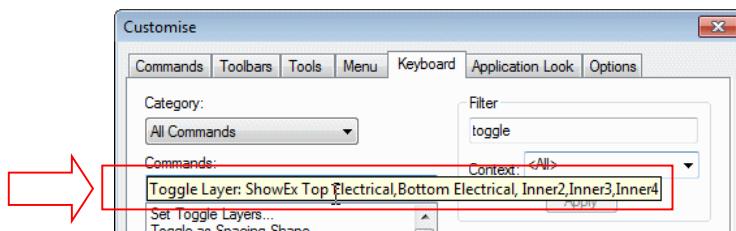
No Override option for Pin Attributes

In the **Part Editor**, you can now mark a **Part** (on the **Parts and Attributes** page) or **Part Pin Attribute** (on the **Pins** page) as **No Override.** This means the value cannot be overridden when the Part is used in a design. If the attribute already had a value in the design before the Part is reloaded, then that value is completely hidden but still available if reloading the Part with the Override switch set off.
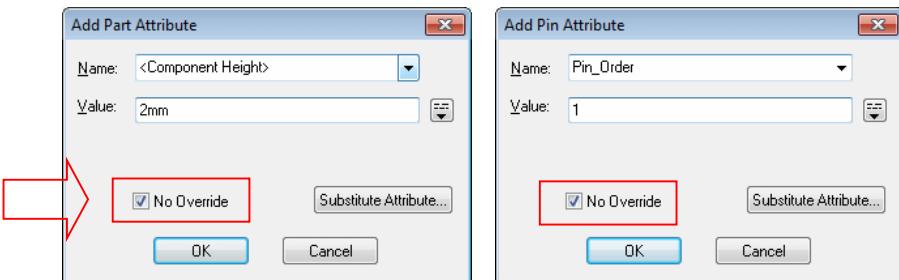
Once an attribute has been set with No Override, the grid will now be colour-coded on the Parts Editor Attributes page.

Change to Change Net Option

On the **Change Net** dialog, there is now an option to not **split subnets defined in lower level blocks**, as this can have unforeseen consequences. By default, the check box is unchecked to not split the nets in lower blocks. Use this for retaining the nets names in lower blocks where the upper level connectivity has been removed by renaming it.
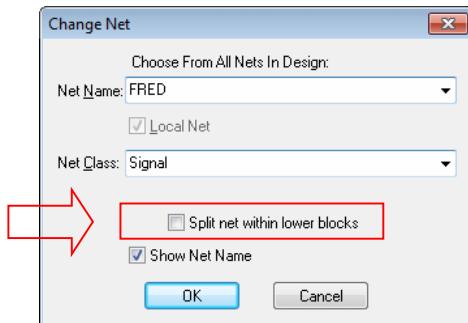
Rename a User-Defined Grid

From the **Grids** dialog, you can now rename user defined grids using the **Rename** button on the dialog.
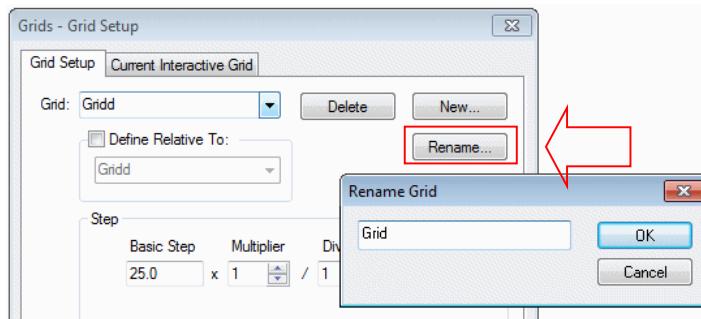
Next Style & Previous Style

New commands for **Next Style** and **Previous Style** are now available. These were also available in V7.6 5222. These are available whenever **Change Style** is available for a single item, but are not on the context menus as they are designed to be placed on a shortcut key through the **Customise** dialog and **Keyboard**.

Next Style changes the selected item to the next style in the style list that would be presented if **Change Style** were used. This list is usually sorted alphanumerically. For a **track** it will only change the style of the selected segments, and for a **Component Pad** in a SCM or PCB design it will only change its style if the pad is already using an **alternate pad style**. For a **Bus** it will change the style of all attached or same named busses. The same rules apply to the use of **Previous Style**.

Next Grid & Previous Grid

New commands for **Next Grid** and **Previous Grid** are now available. These were also available in V7.6 5222. These are not available on the context menus as they are designed to be placed on a shortcut key through the **Keyboard** tab on the **Customise** dialog.

Next Grid changes the current grid to the next grid in the list that would be presented if **Change Grid Step** were used. This list is usually sorted alphanumerically. This also applies to **Previous Grid**.
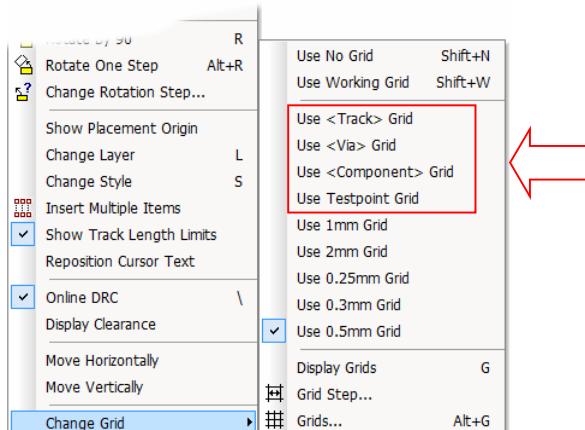
Next User Grid & Previous User Grid

New commands for **Next User Grid** and **Previous User Grid** are now available. These were also available in V7.6 5222. These operate in the same way as the **Next** and **Previous Grid** commands except they only change to grids that you have added.
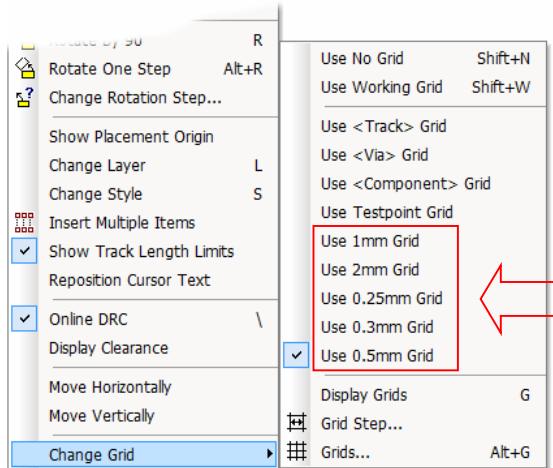
Change Style of Multiple Items

Change Style is now available on the context menu when multiple items are selected. This includes multiple items of the same type (e.g. multiple tracks) and supports some items of different types provided they can have the same type of style (e.g. text, or doc shapes, or pads and vias). You cannot change the style of multiple component pads in a PCB or Schematic design, or multiple Differentially Paired tracks.

If multiple types of items are selected, a warning dialog is given in case the user has accidently included a wrong item.

Change Layer of Multiple Items (PCB)

Change Layer, or **Change Layer Span** are now available on the context menu when multiple items are selected. This includes multiple items of the same type (e.g. multiple tracks) and supports some items of different types provided they can be put on the same type of layer (e.g. text and doc shapes). You cannot change layer on multiple Components, Documentation Symbols, Composite Vias or Differentially Paired tracks.

If multiple types of items are selected, a warning dialog is given in case the user has accidentally included a wrong item.

Manhatten Length in Nets Report

The standard **Nets Report** now shows **Manhattan lengths** for unrouted connections and routed tracks, as well as the actual lengths as previously shown.

```
Net: DRIVE    Net Class: Signal   Guard Space: 0.0
  Pins: 4    Testpoints: 0   Vias: 0   Tracks: 2   Connections: 1   Locked Cons: 0
  Unrouted Connection Length: 550.0   Track Length: 441.4+
  Unrouted Connection Length: 550.0   Track Length: 500.0    (Manhattan)
```

Changes to Replace Part (SCM)

From the **Replace Part** dialog, there is now an option to **Replace Gate Symbols With Default.** If left unchecked, this will match existing gate symbols rather than using the default for the new Part.
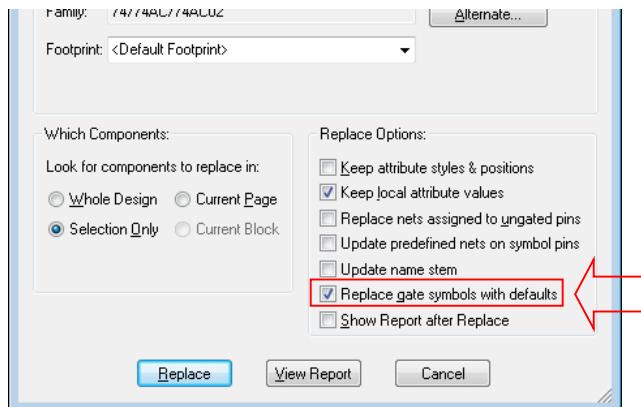
There is also an option on the **Replace Part** dialog to **Update** the **Name Stem**, if the Part being replaced has a different Name stem to that in the design.
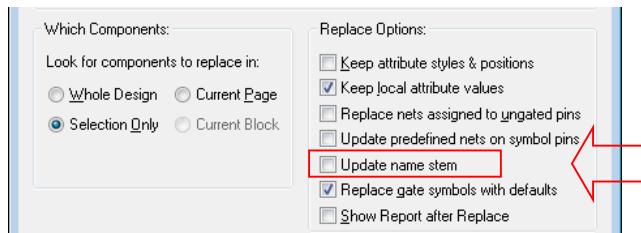
New Toggle True Width commands (PCB)

Three new commands have been added to toggle the true width of items, these can be assigned to shortcut keys if they are needed and are not presented on menus anywhere.

Toggle True Width Pads, Toggle True Width Shapes and Toggle True Width Tracks

Note if you also required the item to be hollow, you must also have the Hollow check box set in the Display tab of the Options dialog.

The above example shows the design before and after the **Toggle True Width** commands have been used.

CAM Plot – Allow Break Around Pads override (PCB)

From within the **CAM Plot** dialog, you can now disable the breaking of shapes around pads on a specific plot. This allows you to create a more complete silkscreen plot for example, without having to change the **Layer Class**. The breaking around text is unaffected.
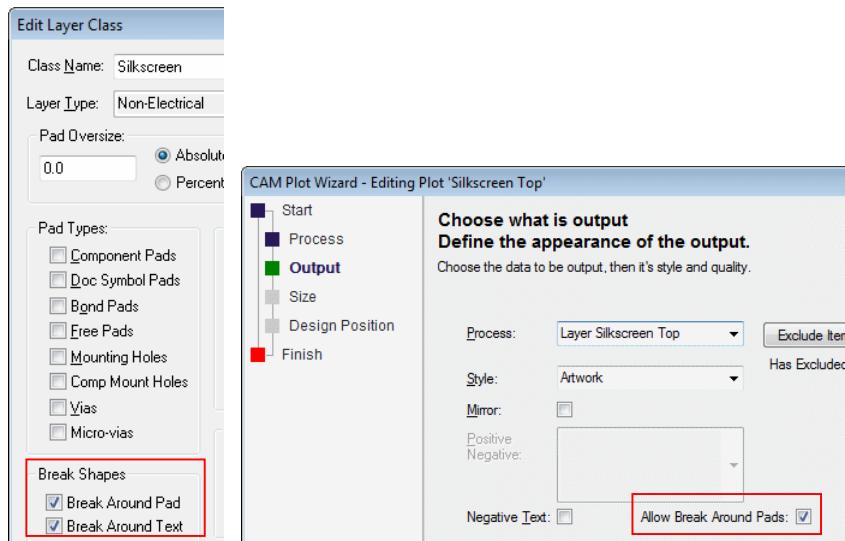
You may use this for example when a full silkscreen plot is required for production but has many broken shapes around the pads. The same silkscreen plot may also be required for an assembly or documentation purposes but the broken plot detail is not needed. To save creating a duplicate layer with a modified Layer Class, you can now simply select the **Allow Break Around Pads** check box on the **Output** page when **editing** the plot. The default switch setting would be 'checked' but this will be relevant to the Break Around Pads setting in the **Layer Class**.

Insert Track – Snap To Target (PCB)

An undocumented improvement was added to V7.6.5222, this has been further improved with the version 8.0 release. The feature allowed **Insert Track** or **Edit Track** to always be able to snap onto a position that is orthogonal to the start of the dynamic segments being added. This enables you to always be able to start a track in-line with an off-grid pad, or help when routing a **Differential Pair** (using the Interactive High Speed cost option) where the centre point between the pads is off-grid.
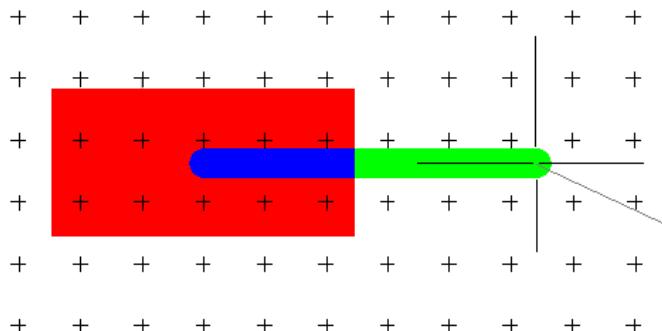
In version 8.0, this has now been extended to snap to a position orthogonal to the target item at the end of the dynamic connection or trailing track segment. This again will help to finish tracks to off-grid pads, and to join Differential Pairs or Bus routes.
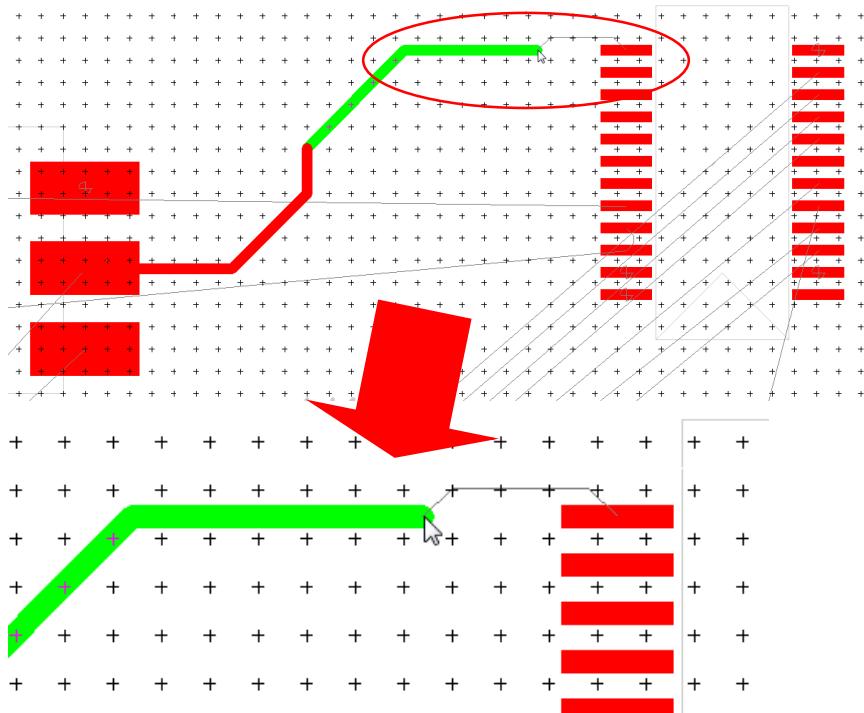
There is no interface change or menu options, this is an operational change to improve its use.

Show Dynamic Cons Only (PCB)

When moving items attached to Nets and when editing Tracks, Differential Pair or Bus Routes, there is a new option available to hide all unrouted connections that are not dynamic during the interactive operation.

With this option selected, it will only show the dynamic connections if they were already visible. It is useful in the early stages of layout when there are a lot of connections on the screen, enabling you to just view the connections attached to the component or tracks you are moving or editing. Switch the option on using the Options dialog, **Move** and **Edit Track** pages or from the context menu during move or track edit.
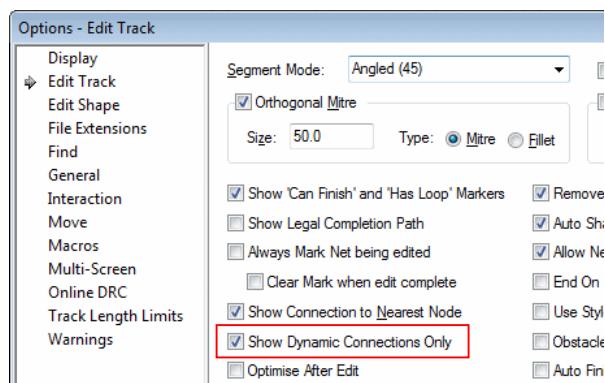
Apply Vias to Tracks and Copper (PCB)

You can now add additional **Vias** to **Tracks** or **Copper** shapes. It allows you to add vias along, inside or around a Track or Copper shape You can select the item then apply the **Vias** to it (using the menu option), or enter the **Apply Via To Selection** from the **Utilities** menu from where you are presented with a dialog.
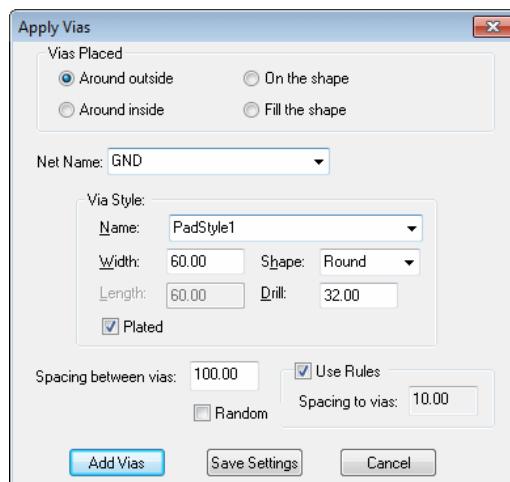
Using the dialog

Choose where to place the vias. **Around outside** means placing vias in all possible legal positions around the outside of the selected shape and on the net given. This is ideal for shielding a track or other item. **On the shape** means along an open shape, such as a track, or along the edge of a closed shape. This could be used to stapling an item to a power plane. **Around inside** means inside the edge of a closed shape. **Fill the shape** means cover the interior of a closed shape with vias.

Select or enter the **Net Name** to which the vias will be added. Leaving it blank creates a default net. If you select an item already on a net, and you are adding vias on the shape, then the net is already determined and you will not need to select a net.

Specify the Via Style to use (this is initialised to the default for the selected net).

Define the **spacing between vias**, this defaults to the Via spacing, but can be any positive value.

When placing around the shape the **Spacing to vias** is the spacing between the selected item and the vias, this should be at least the Via to item spacing defined in the **Spacing Rules**, but can be larger.

If you are placing vias to fill a closed shape, you can choose to **Stagger** them to give a closely packed array.

The **Random** option will randomly distribute the vias, so there will be no regular pattern.

Press **Add Vias** to add the vias. **Save Settings** will save the current settings, and use them to prime the dialog, where appropriate, next time it is used. **Cancel** exits the dialog without doing anything.
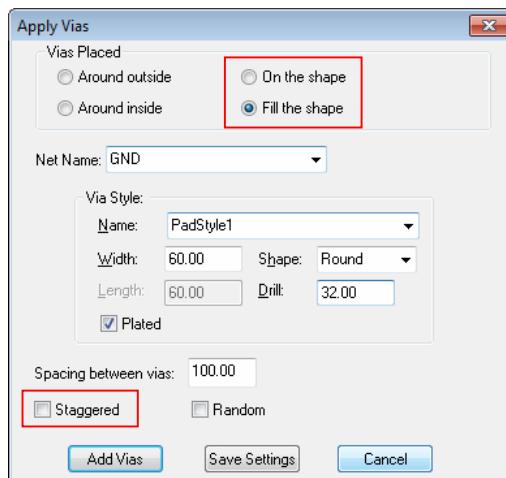
The **Staggered** pattern may look like this:
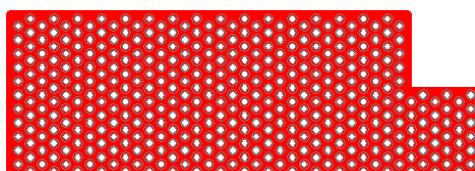
The **Random** option will randomly distribute the vias, so there will be no regular pattern.
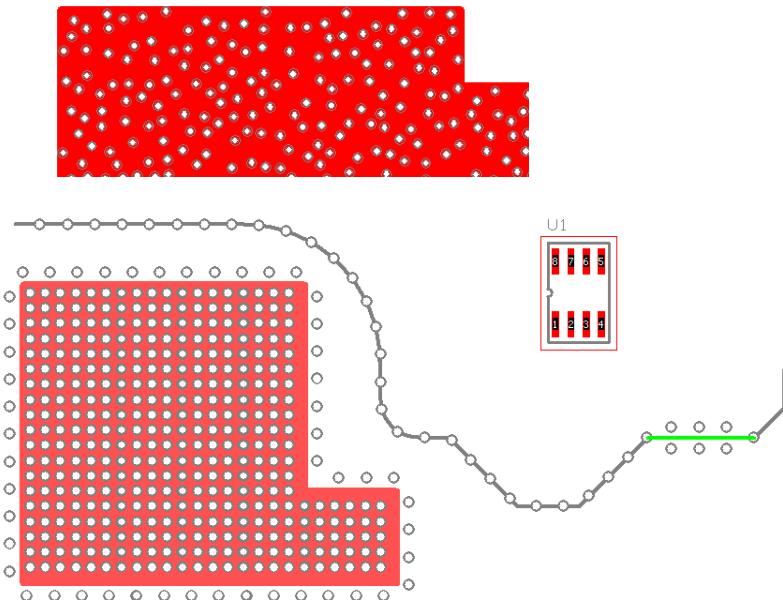
Using the **Around Inside** option, vias could be placed inside a board outline, like the example below:
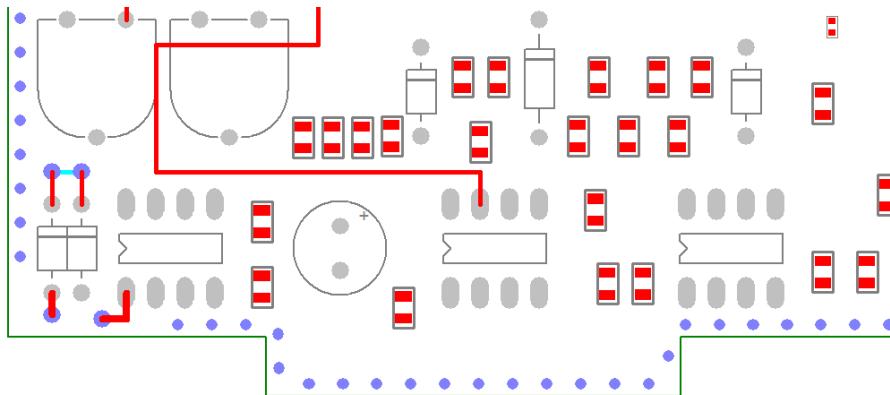
Removing Applied Vias

Vias added using this tool can be removed using the **Remove Applied Vias** from the **Utilities** menu.

You can remove all applied vias on the selected net or all nets by selecting one of the vias when using **Remove Applied Vias**. Selecting a track or copper shape will only remove such vias within the vicinity of the shape.

Create Breakout Pattern on Footprints (PCB)

When editing a footprint in the PCB editor, there is now an option to add a breakout pattern to the footprint.

Within the **Footprint Editor**, from the **Tools** menu, there is the **Create Breakout Pattern** option. This displays a dialog from which to make your breakout selection.
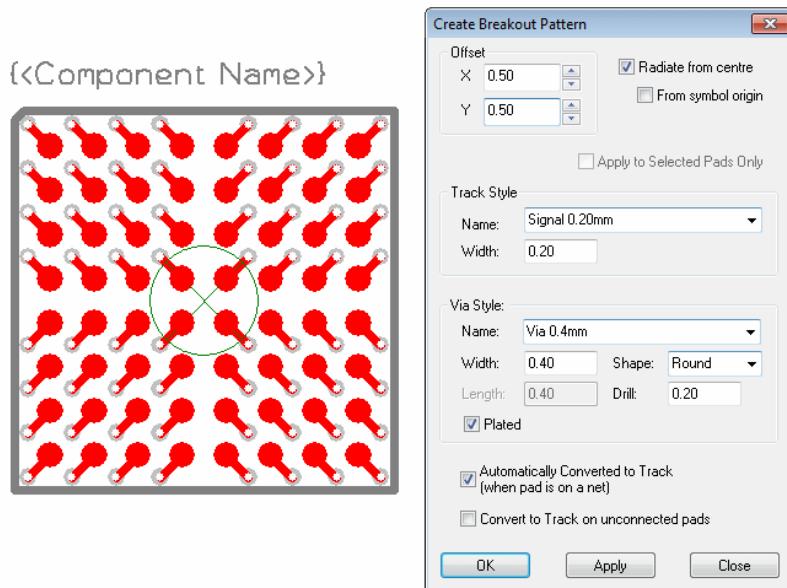
Using the Create Breakout Pattern dialog

Define the X and Y offset of the breakout relative to each surface mount pad. You can reverse the direction by specifying positive or negative values.

Radiate from centre causes the direction of the breakout to be determined from the position of the pad relative to the centre. The centre is defined as the centre of the bounds of the pads on that side, or with the **From Symbol Origin** checked, the Symbol Origin. The X, Y offsets defined are adjusted accordingly.

Apply to Selected Pads Only, when checked, will add breakouts only to the selected pads.

Select the **Track Style** to use for the breakout. Then select the **Via Style**, only **pad styles** defined as suitable for Vias are shown. Usually you would select a style already defined in the technology.

Automatically convert to Track (when pad is on a net) can be turned off to stop the breakout from ever "automatically" being converted to a track.

Convert to Track on unconnected Pads can be switched on to allow breakouts attached to pads that are not on a net in the PCB design to "automatically" be converted to tracks.

Press the **Apply** button to generate the pattern without leaving the dialog.

The pattern generated will replace any existing breakout patterns on the surface mount pads.

New check box on IDF Export dialog (PCB)

There is a new check box, **Reverse Rotation of Mirrored Components,** on the IDF export dialog.

This allows you to choose the export rules depending on how your third-party software package interprets the orientation of mirrored components. When this box is checked, the rotation of any mirrored component is reversed (angle = 360 - angle) on output to the IDB file. The default setting is for the check box to be selected, this is how the option has worked previously.
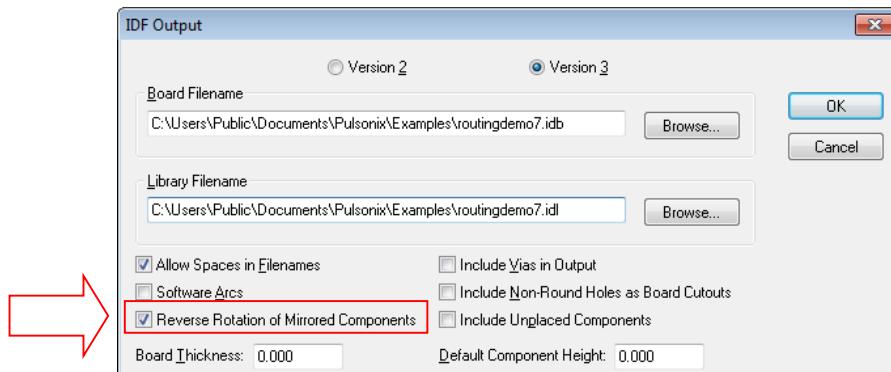
Footprint Rules (PCB)

Overview

Footprint Rules provide a mechanism for automatically selecting PCB footprints where the Part has multiple footprint alternatives based on defined rules. This may be desirable for a variety of reasons, for example, to provide support for different soldering techniques or technologies. The Footprint Rules allow you to define the footprint type and rotation of the footprint selected for use based on the soldering or manufacturing technology being used.

A Footprint Rule can be thought of as having two facets; the first facet comprises of the Layer and/or Area name, this identifies the side (or area) of the board to which the rule applies. The second facet comprises of the rule to which a footprint must match in order to be valid for use. This information consists of a 'Flow Direction' which is used when specifying rules for soldering technologies. It can also consist of attributes for more general criteria. These are made up of an attribute name (with wildcard values) that are used to match against attributes defined on the footprints from the library. The use of wildcard characters when specifying Footprint Rules allows a single rule to cover multiple combinations for example, multiple Areas or wildcard footprint names (i.e. *Wave* or *Reflow*).

How To Use Footprint Rules

Footprint Rules are added to the technology dialog using the **Technology** option from the **Setup** menu and **Footprint Rules**.

	Layer	Area	Flow Direction	Footprint Criteria Attributes		
				Name	Value	Essential
1	*Top*	Special Reqs	<None>	process	special	☑
2	*Top*		Left to Right	process	*Reflow*	☑
3	*Bottom*	SM	Top To Bottom	<Multiple Criteria>		
4	*Bottom*		Top To Bottom	<Footprint Name>	*Wave*	☑
5	*Bottom*		Top To Bottom			☐

New…
Edit…
Delete
Up
Down

☑ Use Rules to Auto-Select Footprints   ☑ Auto-Rotate Footprints when applying Rules
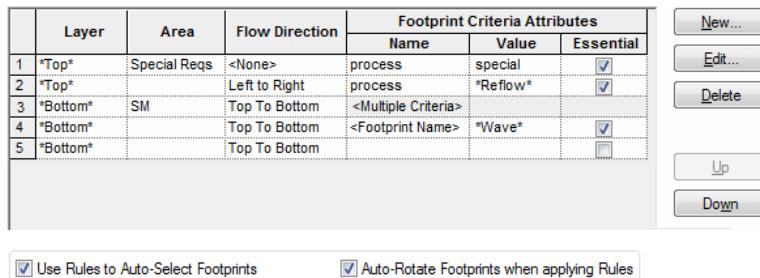
For more details on each of the columns shown for Footprint Rules, see the **Edit Footprint Rules** below.

For more details on the additional check boxes on the Footprint Rules dialog see below.

If a rule has a single criteria attribute, its details will be shown in the **Name**, **Value** and **Essential** columns under **Footprint Criteria Attributes**. However, if the rule has multiple criteria attributes, **<Multiple Criteria>** will be shown in the **Name** column and the criteria attributes can only be viewed or edited by using the **Edit** button.

Assumptions

Pulsonix makes the following assumption: When creating the Footprint in the Footprint Editor, the default solder flow direction is defined as being from left to right (the board will move from right to left [Board Transport Direction] which is the direct opposite of the solder flow direction). The rotation of the footprint created will be classed as 0 degrees at this point.

Solder Flow Direction

Board Transport Direction
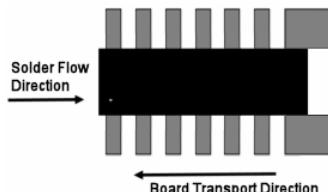
Footprint Rules Order

Footprint Rules are applied to the components in a PCB design in the order they are defined in this table. Successive rules will be compared until a match is found taking into consideration its Layer name and Area name (an area has to be defined as a **Footprint Rules override area**).

Use the **Up** and **Down** buttons to change the order of the Footprint Rules entries.

It is important to ensure that rules are in the required order and generally the more specific entries are nearer the top of the table and the general definitions using wildcard names and blank fields are at the bottom of the table. This is especially important when using Footprint Rules within a named Area.

☑ Use Rules to Auto-Select Footprints   ☑ Auto-Rotate Footprints when applying Rules

Use Rules to Auto-Select Footprints

When this option is checked, the Footprint Rules will automatically select the most appropriate footprint for any component being added to a PCB design, for example, when using the **Translate To PCB** or **Insert Component** options.

This option must also be enabled to allow automatic switching of footprints when mirroring or rotating components, although there are additional options in **Interactive Options** to disable this behaviour.

If this option is not checked, Footprint Rules can still be applied to components after they have been added to a design by using the **Auto Footprints** feature.

Auto-Rotate Footprints when applying Rules

This option is applicable when the Auto-Footprints function is used and 'Flow Direction' is being used in the **Footprint Rules**. Normally, when using 'Flow Direction' to support soldering techniques for example, this option would be left unchecked so Footprint Rules will select the footprint appropriate for a component's existing orientation in the design, in the same way as when a component is interactively mirrored or rotated (options permitting).
Checking this, allows the **Auto-Footprints** option to apply a different approach and automatically rotate a component to accommodate a valid footprint. Thus if the component's existing footprint is valid for its layer (and area, if specified) in the PCB design, but not its orientation, it will be rotated so that it is valid rather than selecting an alternative footprint. Where the component's existing footprint is completely invalid, a suitable alternative will be selected in precedence order rotating the component if necessary.

For specific details about attributes which effect Footprint Rules (for example Orientation), see the Footprint Rules Details page.

Not Allowing Automatic Changes when using Footprint Rules for a Component

Individual components can be locked so that its footprint does not allow automatic changes by the Footprint Rules option. This can be done using the **Allow Automatic Changes** switch on the **Component Properties** dialog.

Design Rules Check

There is a new check box available for checking the **Footprint Rules** on the **DRC** dialog.

Footprint Rules Dialog

The Edit Footprint Rules dialog is displayed when the **New** or **Edit** button is selected in the **Footprint Rules** page of the Technology dialog. The values that can be entered here correspond to the columns in the main Footprint Rules dialog.
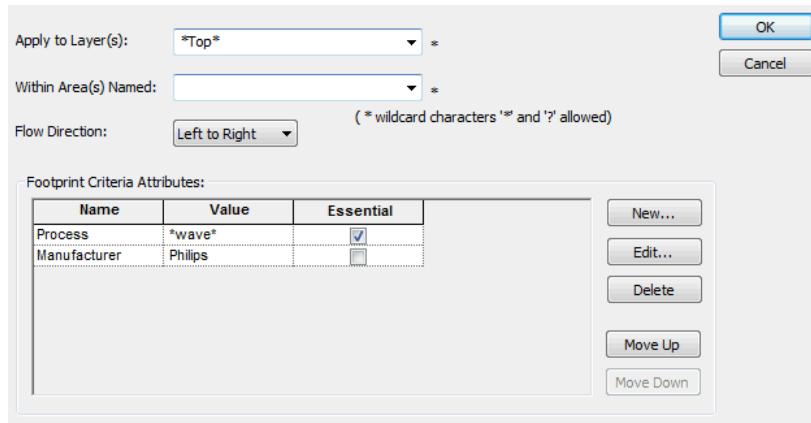
Apply to Layer(s)

Enter the name of the **Layer** that identifies the side of the board to which the rule will apply. The drop down list offers the names of the current relevant layers in the design, but this is not exclusive and any name may be entered. Wildcard characters may be used to specify a name that will match multiple layers. For example, using *Top* will match all layers whose name contains the string of characters *Top* including the special **Top Side** layer on which components are added by default.

Within Area(s) Named

Enter an area name if the Footprint Rule is only to be used within design areas whose names match the entered string. Only areas that are marked as **Use in Footprint Rules** in the **Override Within Area** section of the **Properties** dialog will be considered. The drop down list offers the names of the current relevant Areas in the design, but this is not exclusive and any name may be entered.

The name can be left blank if areas are not relevant to this entry, or you can use wildcard characters to specify a name that will match multiple areas.
Note, using the wildcard * will apply the Footprint Rule within any override area that is named.

It is important to ensure the Footprint Rule is moved up the priority order in the **Footprint Rules** table so the system prioritises it over others.

Flow Direction

If the rule is to support a particular solder technology and is required to select footprints correctly orientated for a specific flow direction, the appropriate selection from the drop down list should be made. If flow direction is not relevant **<None>** should be chosen from the list. For example, use <None> as the flow direction if swapping components for ones of a smaller size where rotation is not a major consideration.

For example, if the Flow Direction of the top side is **Left To Right**. For the bottom side of the board a Flow Direction of **Right To Left** would be used as the board is now *mirrored* in relation to the top side.

The Flow Direction specified is used in conjunction with the **<Orientations>** attribute added to a footprint in the library, as described on the help page for **Footprint Rules**.
Also see Footprint Rules Details for example of using attributes with this option.

Footprint Criteria Attributes

The **Name**, **Value** and **Essential** columns that appear under this heading specify the attributes that define the footprint selection criteria for the rule. If it is left blank then all footprints in the design will be matched by the rule (subject to the Flow Direction setting described above). There is no limit to the number of attributes that may be specified but rules are prioritise based on their order in the table.

To add an entry to the list of criteria attributes use the **New** button. Existing entries may be edited directly in the table by selecting the cell and typing the text or by using the **Edit** button. Use the **Delete** button to remove an entry.

Essential

If the **Essential** option is checked, then in order to successfully match the rule, the footprint must contain the attribute and its value must match the value specified in the rule. Both must be a match.

If **Essential** is not checked, the footprint will match if:

- it has the attribute and the attribute value matches the one in the rule, or
- it does not have the attribute

Using Footprint Rules

Footprint Rules Quick Overview

This is a quick summary of the steps required to use the Footprint Rules feature:

- Create or Edit the Footprint for the process required
- Define Attribute(s) on the Footprint (for example, the <Orientations> attribute and a "Process" attribute to identify the soldering technology and legal rotation) if required
- Define the Footprint Rules in your Technology file
- Add new Components to the design and have Pulsonix automatically select the correct footprint and orientation based on the Footprint Rules
- When interactively rotating Components or Mirroring to the other side of the board have the correct footprint and orientation automatically selected based on the Footprint Rules, or use the right click context menu option **Auto Footprint** to switch to the correct Footprint Rules based footprint at a point you choose
- Run 'What if' analysis on the Footprint Rules and report potential changes using the **Auto Footprints** feature on the **Tools** menu
- Apply the rules to the design using the **Auto Footprints** feature on the **Tools** menu
- Run the Design Rules Check to check the validity of the rules defined

Creating or Editing Footprints for Footprint Rules

If you wish to use Footprint Rules to change the size or shape of a footprint or its pads the set of footprints must be available in the Footprint library in order for the Part to find them. For example, different footprints may be required depending on the rotation and thieving pattern needed.

Below is an example of Solder Thieves that have been added to the footprint in the Footprint Editor on pins 4 and 5. *The style of thieving will depend on your manufacturing process.*
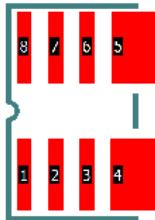
Note that when designing multiple footprints to represent the different arrangement of thieving pads all other aspects of the footprint, including its orientation, should remain constant across all the footprints, only the position and size of the thieving pads should vary.

Naming or Renaming Footprints for Footprint Rules

The Footprint Rules dialog can be utilised with the footprint name or additional attributes. For example, a Part could have alternative footprints for Wave and Reflow use. In this instance, you might have your footprint name reflective of these processes, e.g. SOIC8-Wave and SOIC8-Reflow. The Footprint Rules could then be defined to use <Footprint Name> and values of *Wave* and *Reflow*.

Attributes for Footprint Rules

To apply the Footprint Rules to a design and if the <Footprint Name> and values method is not being used, the **Footprint** must include the appropriate additional attributes. These can be added directly by editing the footprint in **Footprint Editor** or by using the **Attributes** button on the **PCB Footprint** tab of the **Library Manager**.
Attributes must be added to each footprint in the Footprint Library and not the Part itself.

Using Attributes

The Footprint Criteria Attributes defined within a Footprint Rule are used to identify an appropriate footprint by matching against the attribute(s) on the footprint. Any attribute attached to a footprint, including the system defined **<Footprint Name>**, can be referenced in a Footprint Rule.

For example, using the attribute method, if the **Name** column is set to **<Process>** (Process being the Attribute Name) and the **Value** is set to Reflow, the system will search for any footprint which has the **<Process>** attribute defined and contains the word **Reflow**. The appropriate rule will then be applied to that footprint.

<Orientations> Attribute For Flow Direction Alignment

When using Footprint Rules for defining Flow Direction alignment (the rotation), the attribute **<Orientations>** should be added to the footprint. This attribute is a system attribute and is available to you without adding it to your Technology.

<Orientations> Attribute Rule

The <Orientations> attribute value should specify the footprint rotation(s) for which it is valid. This is relative to the default system flow direction of Left to Right.
The <Orientations> attribute value can be any or all of the values 0, 90, 180 and 270. Multiple values are comma separated with or without a space. For example, 90,270 would be valid. The actual

footprint orientation used will match the defined Flow Direction rule in the design against the matched <Orientations> values on the footprint.

For example, the illustration below shows the footprint as created in the Footprint Editor at zero (0) degrees rotation. It has the attribute values of *Reflow* and *90* assigned to it for Process and <Orientations> respectively. This means, the legal rotation for the footprint will be 90 degrees to its current default position.
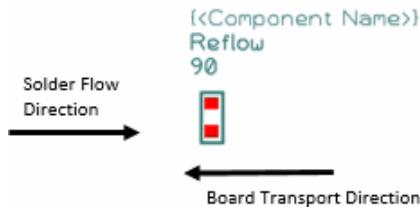
The Footprint Rules dialog could look like the example below. The rules dialog does not need to have the <Orientations> attribute defined here, it will be automatically matched using the Flow Direction defined in the rule.

Layer	Area	Flow Direction	Footprint Criteria Attributes		
			Name	Value	Essential
<Top Side>		Left to Right	Process	*Reflow*	✓

The illustration below now shows how the Process value rule has been matched and the <Orientations> attribute has been applied as a 90 degree rotation to the component to match the **Footprint Rule**.
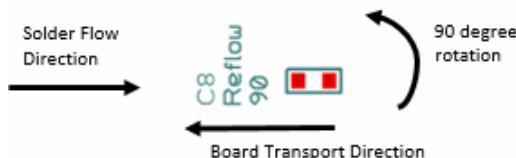
Footprint Rules for Solder Flow Direction

You can use the Footprint Rules for changing footprints to match your soldering process and Flow Direction. A Part definition may contain multiple footprints which are used to represent different thieving pad arrangements for the various orientations of the footprint pads.

For example, the illustration below shows the footprint in the PCB design, the footprint is at the desired orientation as the board is being designed with the solder Flow Direction Left To Right. However, the footprint is incorrect as it does not have any thieving pads on the footprint for this process.
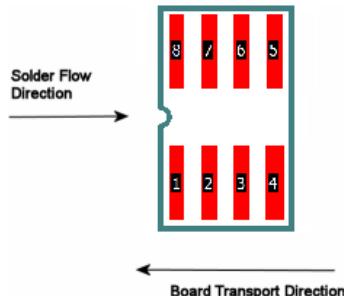
For our footprint to change to the one based on the Footprint Rules, the rules need to be applied. This is shown below; the Footprint Rules dialog has been defined to use footprints which match *Reflow* in their Process Attribute Value. The new footprint has been created with the correct rotation already applied for this process.
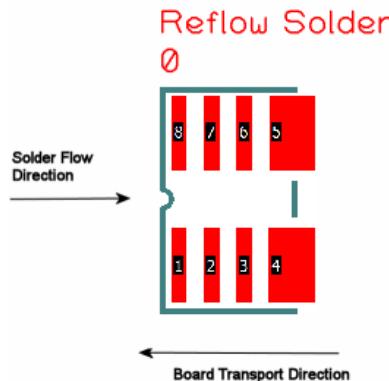
The illustration above now shows how the footprint has changed to the footprint with added thieves on pads 4 and 5. As the attribute for the footprint has been set to 0, and the Flow Direction in the design is the same as the default Flow Direction used when creating the footprint, it has not been rotated, it has already been created at the required rotation.

Footprints without the <Orientations> Attribute

Any footprint without a valid **<Orientations>** attribute or without the attribute at all, will be considered to match any Flow Direction in a rule without any additional rotation being applied.

Footprint Rules for size changes

Footprint Rules can also be used to change one or multiple footprint size within a design. This may be useful where one size of footprint is swapped out for another of a smaller size, for example, where the medium or large IPC standard footprint is swapped out for the small standard. This could be done for a whole Layer or Side or for an Area for example.

In the example below, we can swap out all footprints which have an alternative defined in their Part definition and which have a footprint name containing 0805-med. Wildcards (*) have also been used before and after the text string in the Value field to allow a wide enough search but refined enough to be specific. The Flow Direction has been set to <None> meaning the matching footprints can be orientated any way.

This scenario could also be used for swapping particular footprints in an **Area** by entering an Area name in the **Within Area(s) Named:** box.

Auto Footprint Tool

The **Auto Footprints** option allows the defined set of **Footprint Rules** to be applied to the components in a PCB design. This is used to identify, report and, if required, update to, the appropriate footprints as directed by the rules.

The Auto Footprints option is available from the **Tools** menu, **Auto Footprints**. The dialog is split into two tabs; **Run Now** and **Footprint Rules**.

Run Now Tab

The **Run Now** tab is used to set the options that will be applied when the Footprint Rules are run from the **Auto Footprints** option.
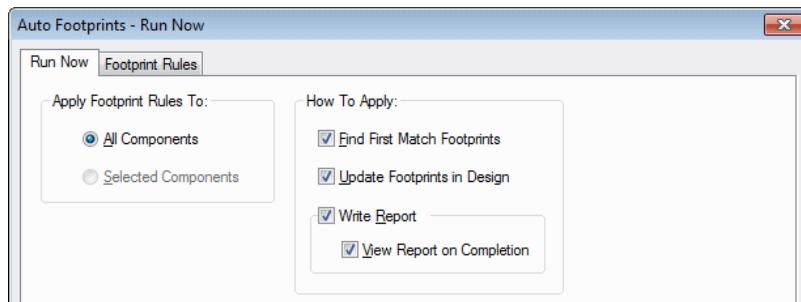
Apply Footprint Rules To

Choose between **All Components** in the design or just the currently **Selected Components** in the design.

An individual component may be locked against having its footprint auto-selected by Footprint Rules by deselecting the **Allow Automatic Changes** option in the **Component Property** dialog

*Note, the Footprint Rules may be applied to an individual component in a design at any time by showing the Context menu and selecting the **Auto Footprint** option.*

How To Apply

Even if a component's current footprint is valid according to the rules, by checking the **Find Best Match Footprints** option, the most appropriate one can be identified by applying the rules to footprints in precedence order from the Footprint Rules table.

The **Update Footprints in Design** enables you to apply the Footprint Rules when the **Run** button is clicked. With this option left unchecked and with the **Write Report** option checked, you can run a 'what if' analysis on the design and view the report to see what effect the Footprint Rules would have if they were applied.

The **Write Report** option enables a report to be written showing the results of running the Footprint Rules on your design when the **Run** button is clicked. Use the **View Report on Completion** check box to view the report after the **Run** button is selected.

Select the **View Report on Completion** option to show the generated report on screen immediately after the option is run.

Report Content

The report will detail the effect of applying the Footprint Rules to your PCB design.

If the **Update Footprints in Design** option is not checked the report will show what will happen if it is checked.

When it is checked, the report title will change to **Auto Footprints - Update** rather than **Auto Footprints - Check** and the additional line **UPDATING Footprints to comply with rules** will also be shown. The reported footprint changes will be applied to your design and the number of footprints changed will be shown.

The main body of the report is divided into three section as shown below:

```
Components matched to Footprint Rules
-------------------------------------
Component: U1. Rules found match, Retain footprint SOIC8_BOT45
Component: U2. Rules found match, Replace footprint SOIC8_BOT with SOIC8_BOT45

Components NOT matched to Footprint Rules
-----------------------------------------
Component: U3. Rules did not find match, Retain footprint SOIC8

Components excluded from Footprint Rules
----------------------------------------
Component: U4. Rules ignored, Retain footprint SOIC8_BOT
```

The **Components matched to Footprint Rules** section shows those components for which the Footprint Rules identified a valid footprint while the **Components NOT matched to Footprint Rules** section shows those for which a valid footprint failed to be identified. The **Components excluded from Footprint Rules** section shows those components which are **locked against being changed by the Footprint Rules**.

A line is shown under the appropriate section, for each component in the design (or selection), detailing if and how its footprint is affected.

Footprint Rules Tab

For convenience, the **Footprint Rules** tab from the **Technology** dialog is duplicated on this dialog.

Running the Rules

Once the desired options have been set, use the **Run** button to apply the Footprint Rules.

To save changes to the options without applying the rules use the **Apply** button.

Design Rules Checking

The Footprint Rules check will enable the DRC option to check that component footprints are valid according to the defined **Footprint Rules**.

Library Manager – Edit Attributes available on PCB Footprints Tab (PCB)

The **Attributes** option is now available in the **Library Manager** for **PCB Footprints**. This has been added to enable easier additional and editing of footprint attributes required for the new **Footprint Rules** option.
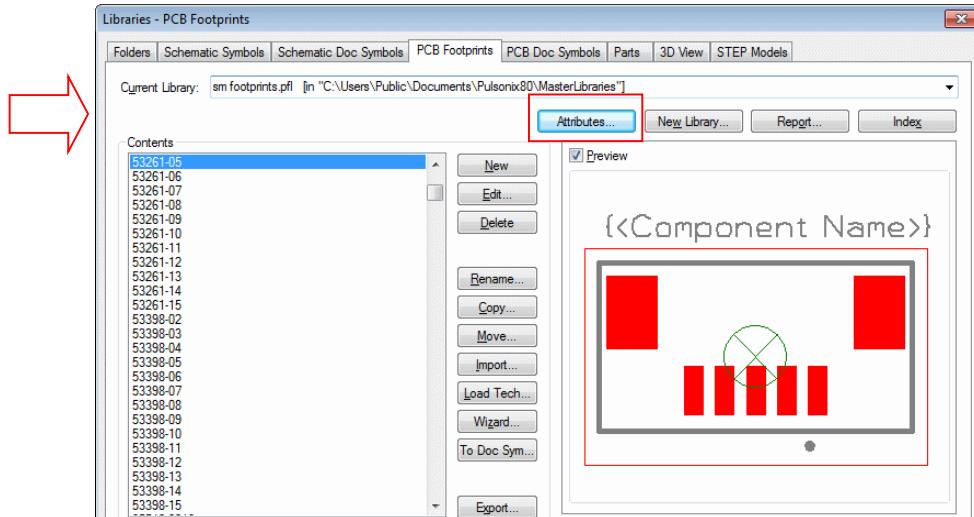
Additional Design Rules Check (PCB)

Net Styles checked under Via Keep Out check

Net Styles which define **No Vias** within **Named Areas** are now checked as part of the **DRC** and **Via Keep Out** check. There is no interface change, just a change to check and report the Via Keep Out rule.
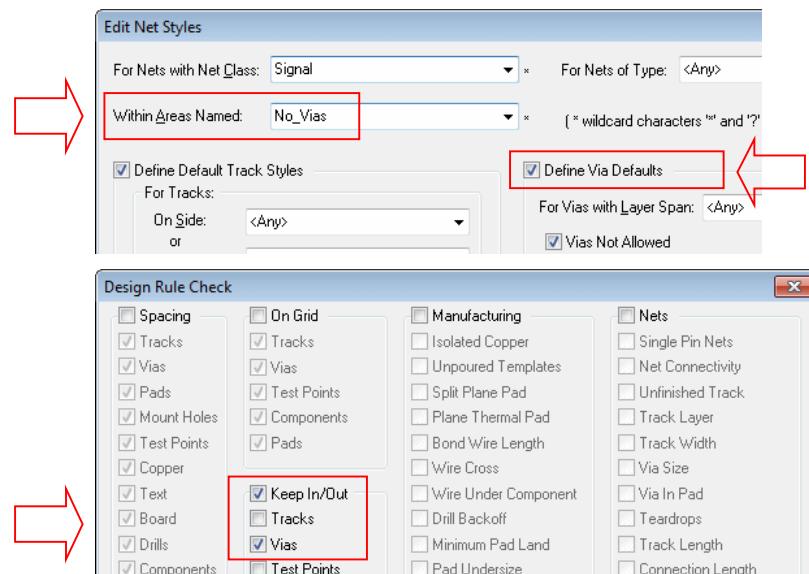
The report looks like this:

```
DRC Errors Report
-----------------
Via to Area Error (V-A) Between (3370.00 2845.00) and
(3300.00 2845.00). Layer 'Top'. Required 0.00 Actual 0.00.
Via to Area Error (V-A) Between (3190.00 2915.00) and
(3270.00 2915.00). Layer 'Top'. Required 0.00 Actual 0.00.
Total:
      2 Via to Area Error (V-A)
Number of errors found : 2
```

Drill hole check of a suppressed pad

Drill Check – this also checks that the drill hole of a suppressed pad does not break a track of the same net. There is no user interface change but the Drill Check option has been enhanced to include this.

Display Pad Numbers in Pads (PCB)

The ability to display **pad numbers within the pads** was added to version 7.6.5223, this is different from the <Pin Name> attribute in that is displays all the pin names in all the components in the design regardless of whether they have the <Pin Name> attribute or not. The <Pin Name> attribute would be used instead of this feature if you decided that some Pin Names needed to be displayed and some not. It also enables you to display Pin Names at a given position, layer and font size.
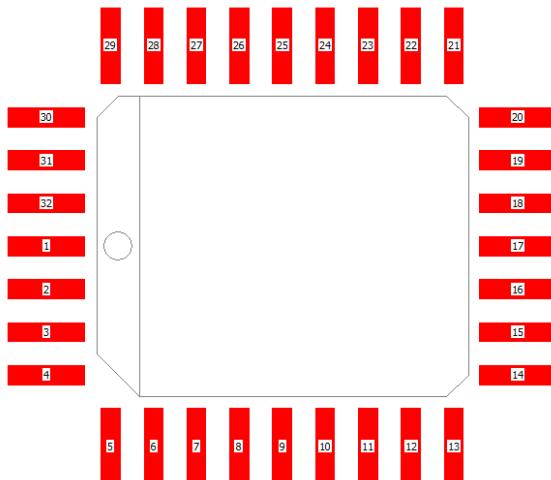
The display of pad names is controlled from the **Colours** dialog, **Highlights**
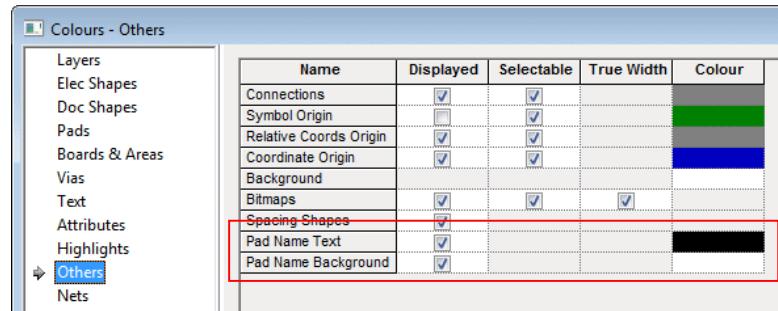
From here, you can switch the **Pad Name Text** and **Pad Name Background** on and off and change the colour. Enabling the foreground but not the background will display the pad names with no highlighted background.

Save/Load Colours has also been updated to allow these settings and colours to be used.

Options dialog

Auto-Hide Zoom Level

From the **Display** tab on the **Options** dialog, you have a slider control to define the zoom level at which the pad name will disappear. This allows you to zoom in close and for the text not to swamp the design. This allows the pad names to be switched off at a point before they swamp the design when zooming out.
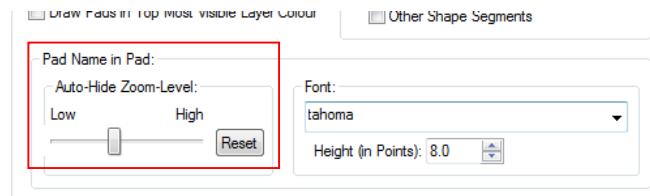
Setting the slider to a **higher** position avoids situations like this below where the pads can't been seen under the pad names:
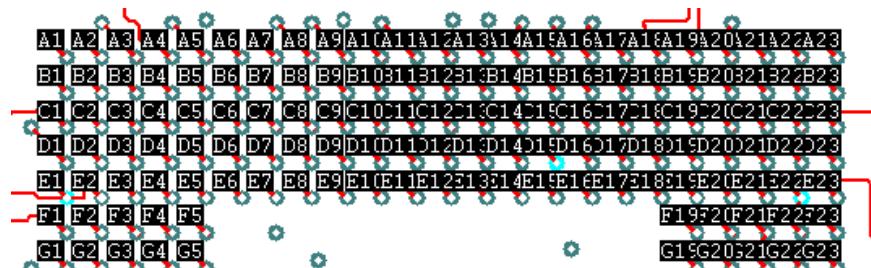
Use the **Reset** button to reset the slider to its default position.

Font Size for Pad Names in Pad

The font style and size used to display the **Pad Names** in **Pads** can be adjusted from the **Options** dialog to grow or shrink the font to your requirements. Some systems have been found to have very small display of fonts. This allows you to adjust the font bigger to make it visible. The font face itself can be changed if you prefer a different style of font.
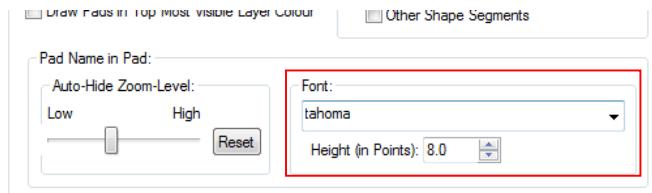
Intelligent Gerber Import reading Parameterised Macros (PCB)

The **Intelligent Gerber Import** option now reads parameterised macros. For example, Macros that may define user-defined pad shapes with parameters will now read in. There is no interface change for this update.

Changes to Toggle Layer (PCB)

Toggle Layer visibility switches (PCB)

The **Set Toggle Layers** feature, accessed via the **Layers Bar** context menu, has been enhanced to allow you to define commands to always explicitly **Show** or **Hide** layers rather than always toggling their visibility between the two. These commands can then be placed on shortcut keys to speed up switching layer visibility whilst editing the PCB design. This feature was also added to V7.6.5223.

If you want a **Toggle Layer** command to set a layer to be specifically visible or invisible rather than toggling between the two there are additional keywords that can be used as a prefix to the layer(s) specification.
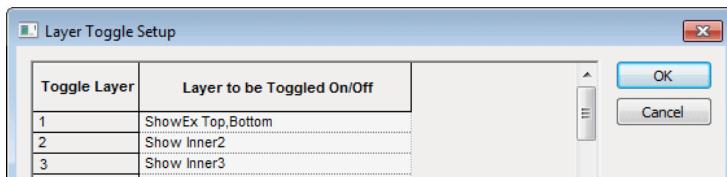
These keywords and their effect are described below:

Show - the specified layer(s) will always be made visible.

Hide - the specified layer(s) will always be made invisible.

ShowEx - the specified layer(s) will always be made visible and all other layers will be made invisible.

HideEx - the specified layer(s) will always be made invisible and all other layers will be made visible.

Toggle - is provided for completeness and has the same effect as no prefix keyword. The specified layer(s) will toggle between being visible and invisible.

For example, defining Toggle Layer 3 as "ShowEx Electrical Top, Electrical Bottom" will allow the top and bottom electrical layers to be shown exclusively with all other layers hidden.

Increased number of Layer Toggles available (PCB)

The number of **Layer Toggle** commands available has been increased from 16 to 100. These are available in the **Customise** dialog and **Keyboard**. This increase was added to V7.6.5222
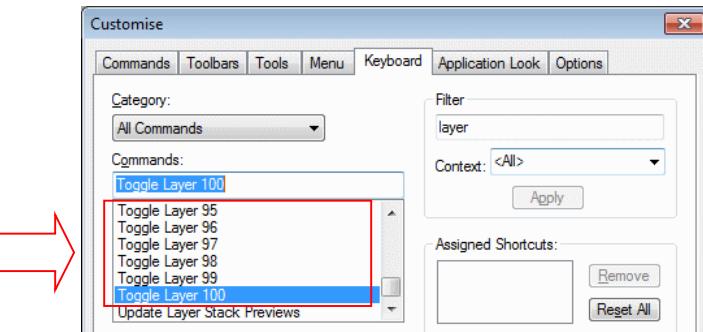
Layers Bar – Display of Layer Colours (PCB)

The **Colour** of each layer is now displayed on the bar alongside the layer name on the **Layers bar**. You cannot change the colour here, only view which colour is used for a particular layer. Use the **Colours** dialog to change the colour.
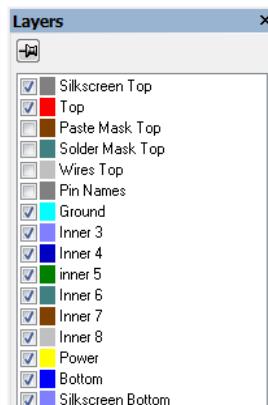
3D Package Editor – User-Defined Shapes (PCB)

As an addition to the existing built-in shapes, the ability to create your own 3D packages will allow you to customise existing packages with extra graphic elements as well as create fully custom packages from scratch.

From within the **Library Manager** and **3D Packages** tab, enter the name of the package (which, like other 3D packages, be used to match the PCB Symbol name of your components) and choose the library in which the new package definition should be saved. Click **OK** to open the **Edit 3D Package** dialog.

Selecting the new **Package Style: User** will turn off most of the controls on the package editing dialog, and instead make available a new **Edit** button that takes you to a separate dialog.

There is also an option to initialise a User package from the elements that are auto-generated for one of the standard packages. This will depend on the PCB symbol that is matched or selected based on symbol name. You could use this for example to take an auto-generated DIP package based on a 14-pin DIL footprint, switch that to a User package to gain access to the individual elements which you could then manipulate.

Note that a user-defined package will have no automatic relationship to the footprint to which it is then assigned, so for example the user package created for a 14-pin DIP will not automatically gain two extra pins if you assign it to a 16-pin footprint. All you will get is a 14-pin 3D image sat on a 16-pin footprint.
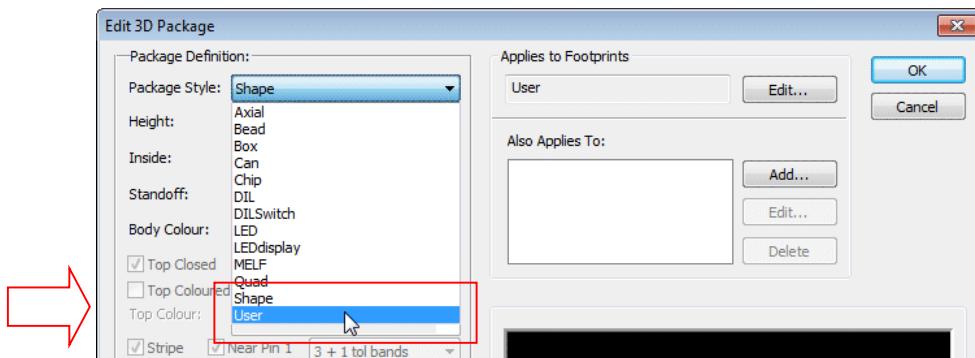
You will be asked to initialise the user defined package feature.
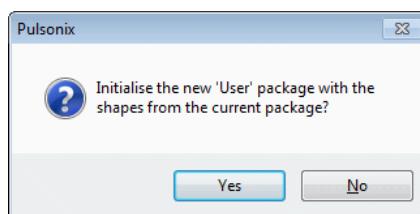
Click **Yes** to use the shapes from the current package and **No** to start a new shape.
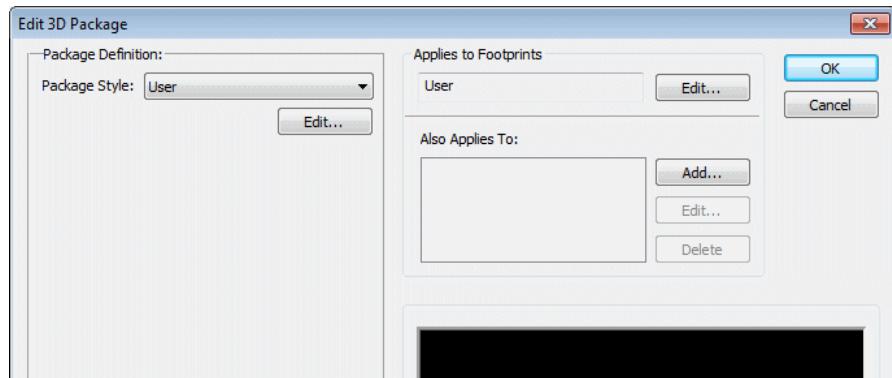
For user defined packages to be previewed and edited successfully, you must have a relationship between the 3D Package and the PCB Footprint(s) it applies to. The **Add** button allows you to do this.
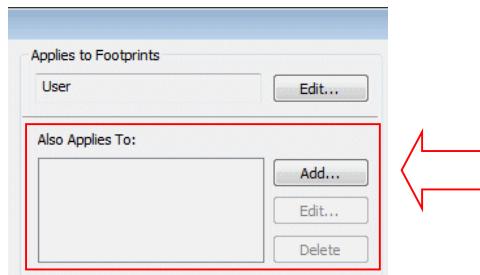
Clicking the **Edit** button starts the editor where you will be able to see a list of all the graphic elements of the package, edit their properties, and add or delete elements.
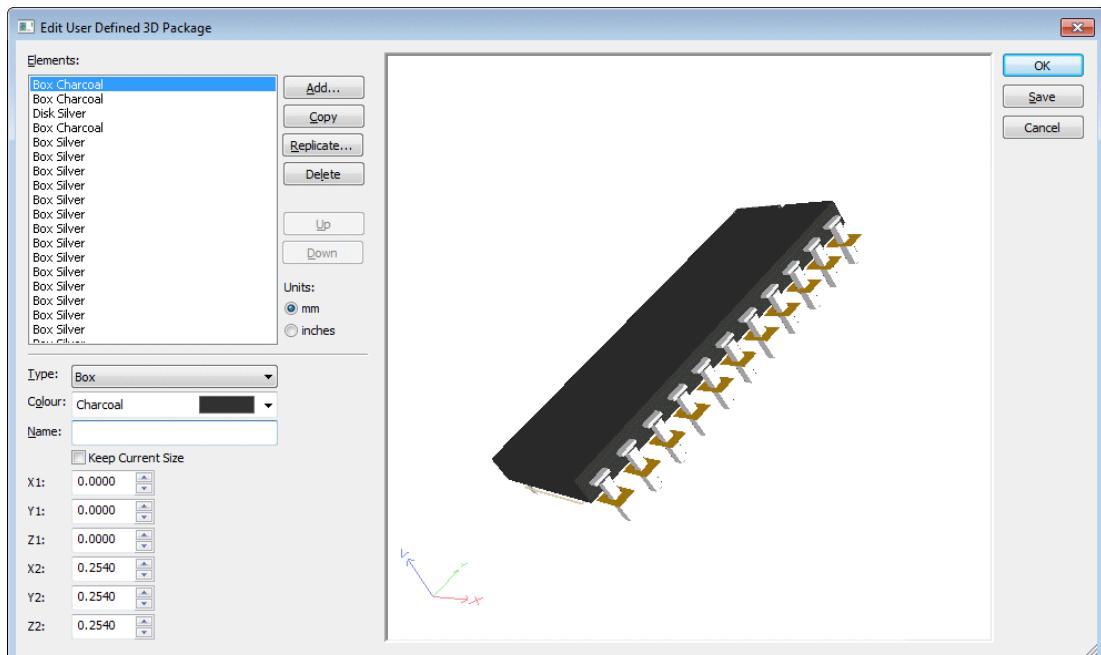
The following graphical primitives will be available to use by selecting the **Add** button and selecting **Type:**

Box	Disk
Pipe (horizontal cylinder)	Sphere
Stack (vertical cylinder)	Cone
Elbow	Prism
Hex	
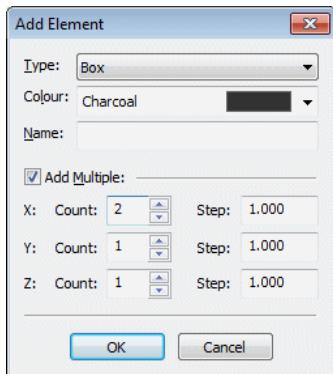
The **Colour** of each element can be specified, and the various position and size properties will also be available to be changed.
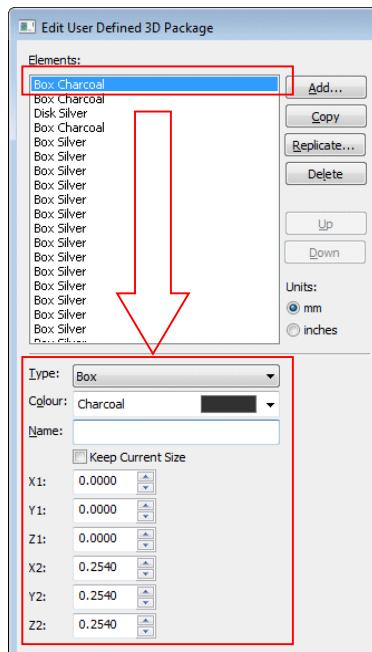
Each Element listed will be expanded in the lower section of the dialog. Selecting each element will reveal its characteristics.

The coordinates displayed on this section show the size and position of that element in the 3D package in the current design units. This is also shown in the Preview to the right. Changing these values will automatically change the element in the preview.

For the best results, it is advisable to edit an existing user package to shorten the creation process.

Coordinate values are taken from the PCB Footprint Symbol Origin. The two sets of values (X1, Y1, Z1 and X2, Y2 and Z2) represent the relative offsets in directions away from the Footprint Origin.

The **Replicate** button is used to copy and repeat existing elements. Multiple copies can be made in both the X, Y and Z directions. By using the X, Y and Z Offsets, you can create rows and columns of similar elements.
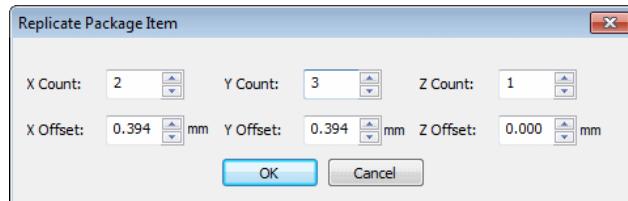
Interactive High Speed (PCB) (Cost Option)

Rules Spreadsheet

As part of the **Interactive High Speed Rules** option, the new **Rules Spreadsheet** dockable PCB bar displays information about Nets, their rules and where the rules are being violated. The display is updated dynamically as tracks are added and edited.

This option is available as the **Rules Spreadsheet Bar** on the **View** menu or from the context menu when right-clicking on the Pulsonix framework.
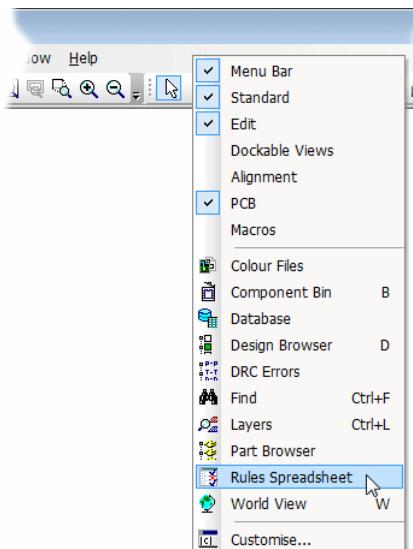
It is presented in the form of a spreadsheet in a dockable bar, you can switch between different information content.

Rules Spreadsheet

Net	Net Class	Bus Name	Sub-Net Attri	Pad1	Pad2	Min Length	Max Length	Length	Complete	Max Vias	Num Vias	Min Text Probes	Num Test P	Max Length	Length Diff
DIFF1	Diff							1281.54 Est.		2			0	150.00	
DIFF2	Diff							1880.92 Est.		2			0	150.00	
DRIVE	Sig2					1000.00	1500.00	3031.84 Est.		2		1	0		
			Pin_Order					0.00	✓	2					
				C12.2	PL2.1	750.00	1100.00	617.31 Est.		2	0				
				C12.2	Q5.3	750.00	1100.00	719.61 Est.		2	0				
				C12.2	Q6.3	750.00	1100.00	951.72 Est.		2	0				
				PL2.1	Q5.3	750.00	1100.00	370.71 Est.		2	0				
				PL2.1	Q6.3	750.00	1100.00	1701.27 Est.		2	0				
				Q5.3	Q6.3	750.00	1100.00	1330.56 Est.		2	0				
FAT	Signal					100.00	700.00	4136.39 Est.					0		
High_speed	Sig2					1000.00	1500.00	5095.79 Est.		2		1	0		

Differential Pairs and **Nets** are available for selection on the drop down list.
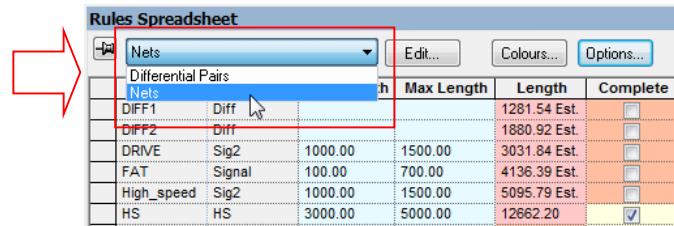
Choosing the spreadsheet type

Use the dropdown list in the dialog header to set the current spreadsheet type, or if the list is not visible select the required spreadsheet type from the shortcut menu. Choose between displaying a list of **Differential Pairs** or a list of **Nets**. Differential pairs have one row per item, whereas Nets can have additional rows to display sub nets (defined by pin attribute) or to display pin pairs in the net. The existence of these rows depends on which columns have been included. The additional rows can be shown or hidden by pressing the small triangle icon in the owner row above them.

Rules Spreadsheet Edit

The **Edit** button is used to define what information (columns) are shown in the rules spreadsheet grid.
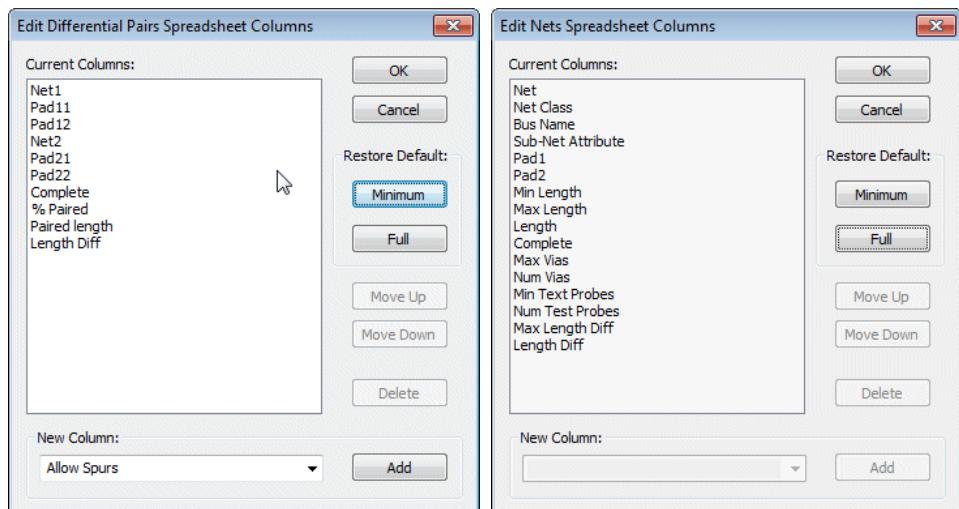
Using the **Minimum** button allows you to select a minimum number of fields to display. The **Full** button shows a more complete list. Both lists can be enhanced by selecting additional fields using the **New Columns** button and using the **Add** button to add them to the list. Once chosen, the order can be

moved using the **Move Up** and **Move Down** buttons. Use the **Delete** button to remove the field from the display list.

Rules Spreadsheet Options

The **Options** button is used to set up your preferences for the rules bar behaviour and to specify which items are not to be included. If the button is not visible, Use **Spreadsheet Options** from the shortcut menu. The options are retained in the system registry so that your choice is preserved for future use.
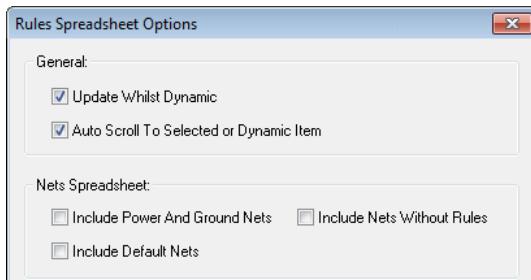
Update Whilst Dynamic - Generally information in the spreadsheet is updated as soon as items in the design are changed. For large nets on some designs this can take a while, especially whilst placing items and adding a track segments. Unchecking this box will stop the "continuous" updating that happens whilst moving items during interactive operations, and they will only be updated when the item is dropped or a corner is added. Leave the box checked if you want to see the track length changing "on the fly" as you move across the board, for example if you are using the spreadsheet instead of the head up track length display on the cursor.

Auto Scroll To Selected or Dynamic Item - Check this box to ensure the current item in the design is always visible in the spreadsheet by automatically performing a vertical scroll.

Include Power And Ground Nets - Check this box if you want to include nets that use a net class with type power or ground. These nets are often not required as net track length is not an issue with them. They usually have a lot of nodes and can take a long time when calculating values, like completion for example, as they need to check planes and poured copper.

Include Default Nets - Check this box if you want to include nets that don't have a user defined net name.

Include Nets Without Rules - Check this box if you want to include nets that don't use a net class containing track, via or probe rules. These nets are not normally required in the nets spreadsheet as they have no rules to check against.

Rules Spreadsheet Colours

Use the **Colours** button to change the colours of the spreadsheet cells, and to define how to highlight the row that represents the item that is currently selected or being edited in the design. If the button is not visible, Use **Spreadsheet Colours** from the shortcut menu. All colours in this dialog are retained in the system registry so that your choice of colours is preserved for future use.
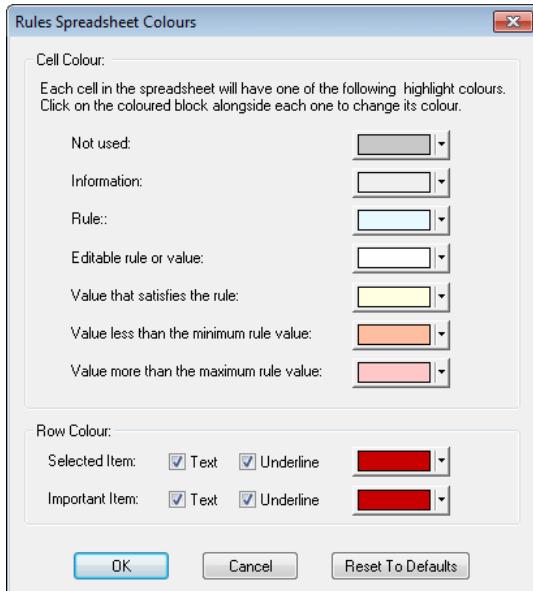
The top section shows a set of colour buttons showing the colours that will be used for each cell information type. Click on a colour block to change its colour.

Not Used - This colour is used for cells that are not needed, i.e. their column is not relevant for their row. For example the "Net Name" cell on a pin pair row.

Information - Used for cells that represent non editable information about the rows data item that is not directly associated with a rule. For example the two net names for a differential pair.

Rule - This colour is used for non-editable values that represent a rule limit. For example the maximum track length for a net. To change these values you must visit the appropriate dialog in the technology.

Editable rule or value - Used for values that are editable. These usually represent a rule limit that is directly editable in the spreadsheet (like the differential pair rules) or an information cell that you can alter to change what data its row represents (pin name cells in a pin pair row on a large net for example).

Value that satisfies the rule - This colour is used for values that are within their corresponding rule limits, and therefore represent a rule pass. For example a nets track length that is within the defined minimum and maximum rule values.

Value less than the minimum rule value - Used to highlight values that fall short of their corresponding minimum rule limit. For example a pin to pin track length that is less than the minimum pin to pin length defined for a net, or a net complete cell on a net that has tracks missing.

Value more than the maximum rule value - Used to highlight values that exceed their corresponding maximum rule limit. For example a pin to pin via count that is greater than the maximum number of vias allowed between pins on the net class.

The lower section shows how the selected or important items in the design have their rows highlighted in the spreadsheet. For each highlight type you choose a colour that will be shown as a block in the first column of the row, and choose whether the text in the row is displayed with this colour, and also if the row is underlined with the colour.

Selected Item - This row represents the single net or differential pair in the design that is selected, or a part of it that is being edited (a track being edited for example).

Important Item - These rows represents nets or differential pairs that have been marked as important in the spreadsheet. See above for information about highlighting rows for important items.

The **Reset To Defaults** button will return the colours and highlight check boxes to their original "factory" settings.

Hiding Dialog Buttons

You can maximise the space available for the spreadsheet grid by using **Hide Dialog Buttons** from the shortcut menu to remove the dialog controls above the spreadsheet and move the spreadsheet up.
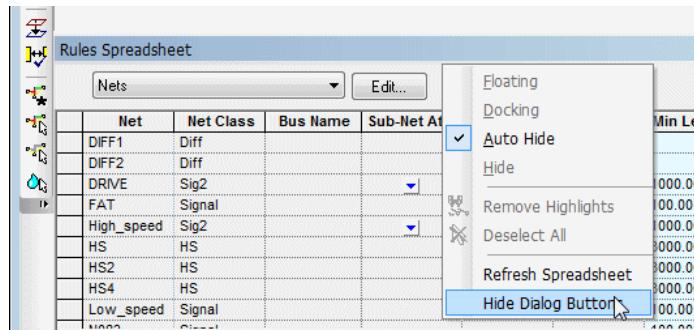

The **Show Dialog Buttons** option will reverse this. When hidden, you can still change the spreadsheet colours, options and columns by using commands on the shortcut menu. If the buttons are hidden, the title of the bar will include the spreadsheet type.
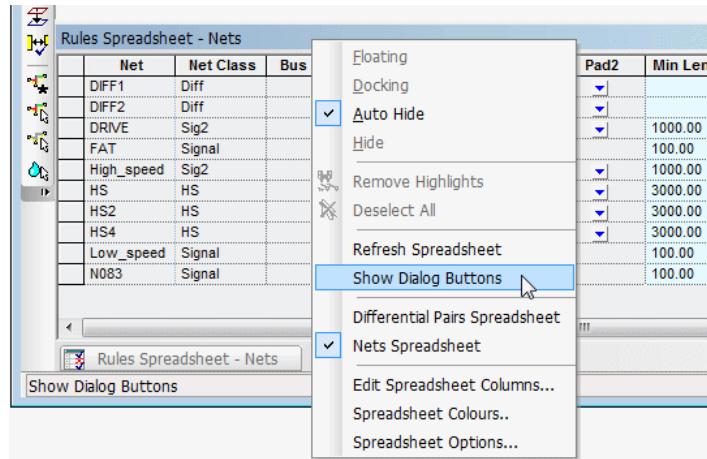
Highlighting important items

Using left click in any cell in the first column that represents a differential pair or net (i.e. not a sub row) will mark the item the row represents as "important" to you. The row will be highlighted to enable you to quickly see the information whilst editing the design. You can specify how the row is highlighted using the spreadsheet **Colours** dialog. Use **Remove Important Items** from the shortcut menu to clear the spreadsheet of any highlighting of important items. Use **Make Selection Important** from the shortcut menu to highlight all rows that have selected items in the design.
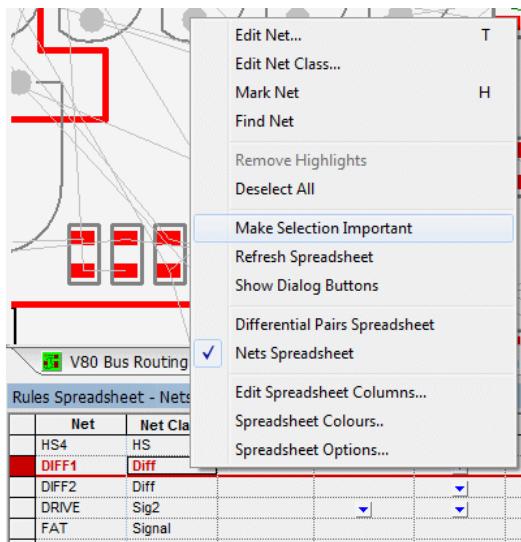
If a single differential pair or net is selected or is being edited, the row will be highlighted as the "selected" row. How the selected row is highlighted is defined in the spreadsheet **Colours** dialog. There is a spreadsheet option (see below) to ensure that the selected row is always scrolled to be visible.

Interactive Bus Routing

As part of the **Interactive High Speed** option, in a PCB design, you can add and edit multiple parallel tracks at the same time representing groups of signals such as Memory, Differential Pairs and Buses that need to be routed together using the **Bus Route** option. Like Differentially Paired tracks, the tracks added to a Bus route are kept together when moving or editing them; treated as if they are a single item.

The option can be used to quickly add a set of tracks on new nets to a design representing a channel that can be connected to existing nets at a later stage, or can be used to route together a group of signals that already exist in the design.

To aid this feature further, names of Busses in Schematics, and the Nets within them are synchronised to the PCB design so that when adding PCB Bus routes you can choose the signals required by using the Schematic Bus Names from the context menu.

Insert Bus Route has three phases:

- Select the items (Pads, Vias, Track ends, Connections (Nets)) that you wish to start from and select the option.

- Position the line of junctions at the start of the Bus route that represent the start point of each track in the set.

- Add segments to define the path of the Bus route.

▶ **To add Bus routes**

Use the **Bus Route** option from the **Insert** menu.

The corners of the centre line of the Bus route will be gridded.

When the mode is entered, **Insert Bus Route** is displayed on the **Status Bar**. Use the following steps to add a multi-track path:

1. First, if another track is pre-selected before entering the mode, the width and layer are taken from the selected track segment to use when adding the tracks in this session. These are shown on the Status Bar.
   The pre-selected track will remain selected until picking is done in the Insert Bus Route mode.

2. Alternatively, if a set of pads or vias are pre-selected these will form the set of nets to be routed together.

3. Before you select an item to start the track from, you can right click the mouse to use the shortcut menu to do the following:-

   Change Layer - Use this to set the layer that you wish the tracks to be on. This layer is used if the picked pads to start from reaches it, otherwise the layer the start pads are on will be used. The new layer is shown on the status bar. You can also use **Next Layer** and **Previous Layer** to step through the different layers.

   Change Style - Use to change the width of the tracks to a specified value. This value will be used even if the picked item is already on a net and has a track style associated with a net class. The new style is shown on the status bar.

   Use Default Style - Use to change the width of the track back to using the default style. The style on the status bar is cleared to indicate that the track style will be defaulted as follows:-

   For new nets the track style will be taken from the **Net Class** defined on the **Nets** page of the **Technology** dialog. If the track is started on an item that is already on a net, the track style will be taken from its net class.

   Change Number of Tracks - This option is available if you do not have any items selected and is used to specify how many tracks you want in a Bus route containing new nets. The number of tracks that will be added is displayed in the Status Bar at the bottom of the screen.
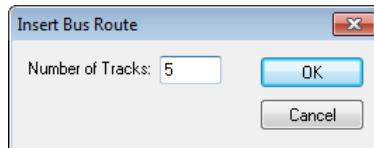
   Show Dynamic Cons Only – Use this option so that you can only see the connections from the end of the bus route to their target pads

   Route Connections At Bus Start - Switch this on if you want the dynamic unrouted connections at the start of the Bus routing to be converted to tracks as soon as you select the position to start the Bus route from. A simple routing algorithm will be used that does not allow layer changes or complicated patterns to be created. If the connections cannot be routed simply, they will be left unrouted for you to complete later.

   Segment Mode - use this sub-menu to change the track segment mode that will be used. This affects what direction the Bus route can be started. This will include 45 degree angles if the current track segment mode is not purely Orthogonal.

4. As soon as you enter the **Insert Bus Route** mode a modal cursor will be displayed for you to use. If you want to use existing nets in the Bus route, select the items to start on as follows:-

   Click on a component pad, via or dangling track end. Its selection state will be toggled.

Drag a frame to select a set of component pads, vias or dangling track ends. This set of items will define the Bus route unless the control key was held down when ending the frame, in which case you will be able to carry on selecting or deselecting items.

Click the mouse on an unrouted connection.

Do not select any items. A new Bus route will be started from the position you picked. The tracks will be placed on new nets and connected to junctions at their start and end points.
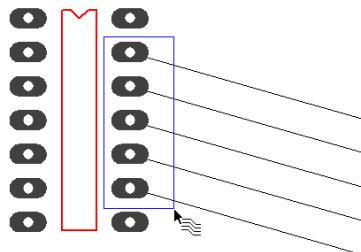
Use **Define Bus Route Nets** to choose nets, the closest items on these nets will be used. If it is difficult to pick items to define the nets you want in the bus route, use this option to choose the required nets from a list of net names in the design.
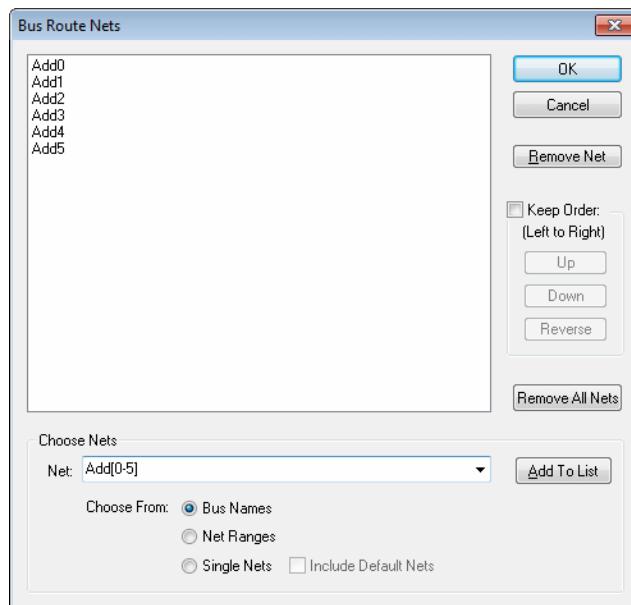
5.   Click in space to start the Bus routing. If a set of pads, vias or connections have been selected you will enter a positioning phase to pick the start point of the centreline of the Bus route. During this phase, dynamic dots of the correct track width and separation will be displayed to show you where the tracks will start and what direction the first track segments will be. This will include 45 degree angles if the current track segment mode is not Orthogonal. The mode can be changed using the shortcut menu.
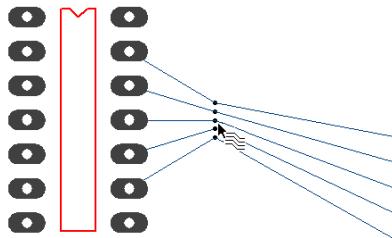
6.   You can use options on the context menu at this stage to alter the gaps between the tracks that will be added.

Define Gap Between Tracks - Use this to exactly set the gap between the edges of adjacent tracks to a typed value. This is useful if you are adding a Bus route to represent a differential pair where the gap has to be an exact value. Note: the specified gap will only be used if it is not less than the Track to Track spacing.
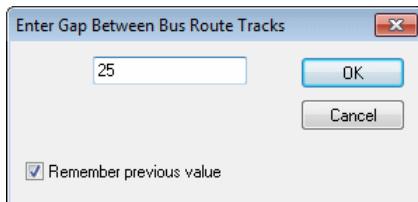
Use Minimum Gap Between Tracks - Use this to make the tracks in the Bus route as close as legally possible to each other. It will use the relevant Track to Track spacing between each pair of tracks.

Use Gaps Between Start Items - Use this to set the gaps between the tracks in the Bus route so that they are in-line with the start items. This is useful when using track routing to add memory routing.

Increase Bus route Gaps - This is available if you previously used the Minimum Gap. Each time you use it the tracks will be spread out by another grid step.

Decrease Bus route Gaps - This is available after using Push Tracks Further Apart to bring the tracks closer together by reducing their offset by a grid step. You can use it until the minimum gap is reached.

7.   Now left click to pick the position to start the Bus route from.

You now have a new set of parallel tracks dynamically displayed on the screen and are ready to add their track segments. If you had the **Route Connections At Bus Start** switch on (see above) the dynamic unrouted connections at the start of the Bus routing will be converted to tracks. If the connections cannot be routed simply, they will be left unrouted for you to complete later.

If you are adding an odd number of tracks in the Bus route you will be controlling the middle track of the set with the cursor, and it will be added on grid just like if you were adding a single track. If you are adding an even number of tracks, a dummy thin track will be added in the middle for you to control on grid. When you have finished adding the dummy track it will be removed. The rest of the tracks will be created either side of the middle track you are adding. This is similar in operation to the **Edit Track** mode except multiple tracks are added together.
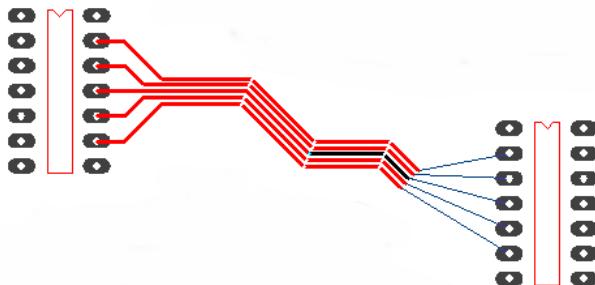
The first track segment added will be forced to use the selected start direction away from the start items. Once you have left clicked once to add the first corner, free movement can be used. The segment mode used can be changed from the context menu. Note that small tight corners created in the middle track path might give strange results for the outer tracks being added if they do not have not enough room to maintain their required gap.

Whilst adding the track segments you can change the gap between the tracks using the various options from the context menu described above (**Define Gap Between Tracks**, **Use Minimum Gap Between Tracks**, **Increase Bus Route Gaps** and **Decrease Bus Route Gaps**). Note that **Use Gaps Between Start Items** cannot be used at this point. When the gaps are changed, if you had the start connections routed they will be re-routed to accommodate the new track start positions.

8. Once the required path has been added, there are several ways of finishing as follows:

Use **Left Double Click** in space, or the **Finish Here** option from the context menu to finish the tracks at the cursor position, leaving the trailing connections as they appear on the screen.

Use the **Complete As Track** option from the context menu to finish the Bus route at the cursor position and attempt to convert the dynamic unrouted connections at the end of the Bus route to tracks, finishing on their end items. A simple routing algorithm will be used that does not allow layer changes or complicated patterns to be created. If the connections cannot be routed simply, they will be left unrouted for you to complete later.

Move the end of the Bus route over a previously created Bus route that contains at least as many tracks as you have in the dynamic Bus route. This is best done by approaching the existing Bus route at the same angle as its last segment. When the end of each dynamic track is over one of the tracks in the existing Bus route a finish marker dot will be shown (if the finish markers are displayed). At this point pressing left click should join each track in the dynamic Bus route to its corresponding track in the static one.

After finishing the tracks you will be ready to select a different set of pads to add the next Bus route from, or use the <**ESC**> key to exit the Bus route mode.

Editing Bus Routes

After creating a Bus route you can edit it again as a single item, just like Differential Pairs. Individual tracks within it can be edited as normal, but doing this makes it hard to maintain the gap between the tracks. The method of changing a Bus route's path is to remove track segments in the middle and add a new Bus route, starting from the dangling set of dangling tracks at one end and ending on the dangling set of tracks at the other end. Start the new Bus route by framing over the dangling ends of the existing Bus route. The gaps between the parallel tracks will be kept to match the existing Bus route.

Use the **Cut Track** interactive option to remove track segments from the Bus route as it will cut the tracks in a nice line ready for adding a new Bus route from.

This method can also be used to split a Bus route to go around an obstacle or to drop the outer tracks and continue adding a Bus route with less tracks. Just frame the correct number of route ends that you wish to continue with the next Bus route.

Bus Route Colours

There is a highlight colour for tracks in a Bus route, use the **Colours** dialog, **Highlights** tab and **Bus Tracks**.
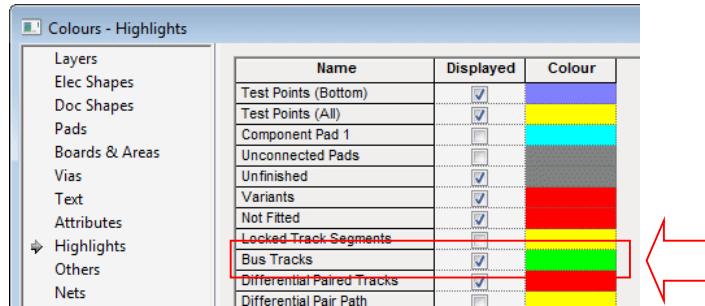
Removing Bus Routes

You can use **Remove Bus Route** from the shortcut menu to convert the parallel tracks back to ordinary tracks again.
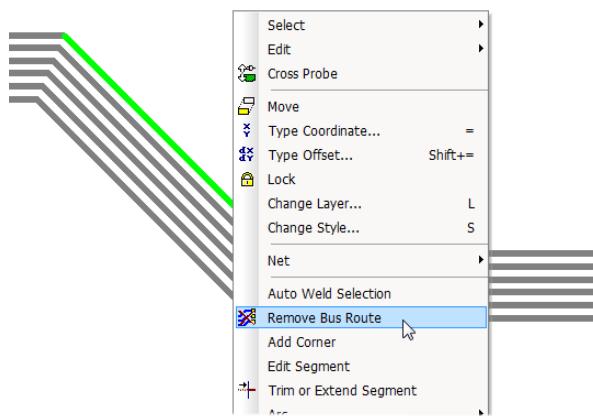

Cut Track

The **Cut Track** function (see below) has been added to make it easy to remove and add new parallel segments in a bus route.

Cut Track Option

In PCB designs you can now use the **Cut Track** tool from the **Edit** menu to define two cut lines and remove any section of track path between the two cuts. This can be applied to multiple tracks at the same time, including **Differentially Paired** tracks and the new **Bus Route** option (see above).

▶ **To use cut track**

1. Select the **Cut Track** option from the **Edit** menu. You will notice the cursor change to the Cut Track modal cursor.

2. Before you select the tracks to cut you can right click the mouse to use the shortcut menu to **Change Layer** to the layer containing the tracks to be cut.

3. Select an electrical layer, or select [Through Board] to cut tracks any layer. The chosen layer will be displayed on the status bar at the bottom of the screen.

4. Use two left mouse clicks to define the start and end points of the first cut line, or drag the cursor to define the cut line. The position of the second point will be restricted to make the angle of the line a multiple of 45 degrees.
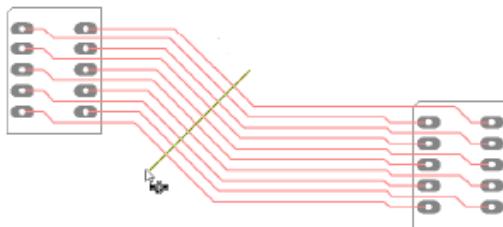

5. All track paths crossing it will be highlighted indicating they are available to be cut.

6. A second dynamic cut line will be displayed at the same angle and length as the first line for you to position with your cursor. You can use the **Rotate** option from the context menu to rotate this line in 45 degree steps to align the cut perpendicular to multiple track segments that are to be cut.
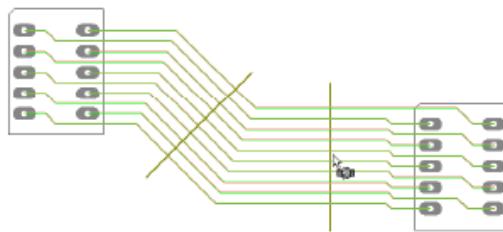
7. Drop the second cut line over highlighted tracks to cut them.

8. If you need a different length or different angle second cut line, it can be changed by moving its centre to where you want the second cut to start and dragging the cursor to define the length and angle of the line.

9. When both cut lines have been defined the highlighted tracks will have their segments between the lines removed.
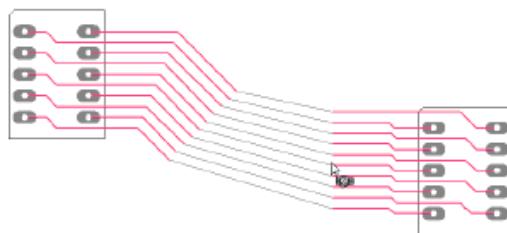
10. This will result in the tracks being split in two by having the section between the two cuts removed unless one of the cut lines is over the track start or end point.

Differential Pairs – Changes to Technology dialog

Within the **Technology** dialog, the **Differential Pairs** dialog has been rearranged for the introduction of Pair Gaps by Side/Layer/Areas.
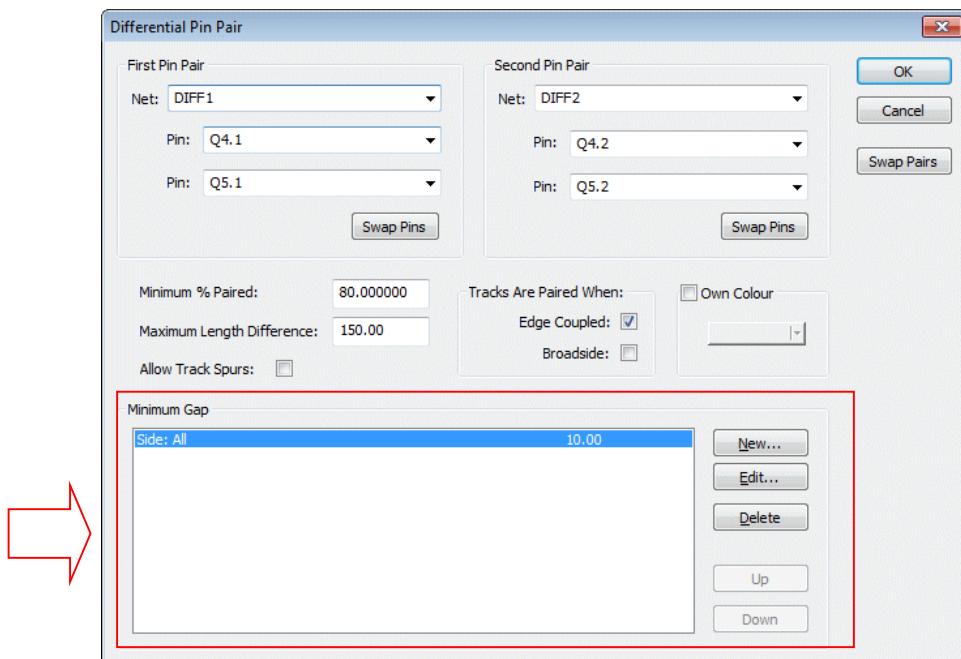
Differential Pairs - Define Pair Gaps By Side/Layer/Area

You can now define the **Differential Pair gap** by **Layer**, **Side** or **Named Area**. You can also define a <No Track> gap which causes an error if the Differential Pair appears in the layer or area. Interactively adding tracks to Differential Pairs will not allow changing to <No Track> layers unless needed to attach to surface mount pads.
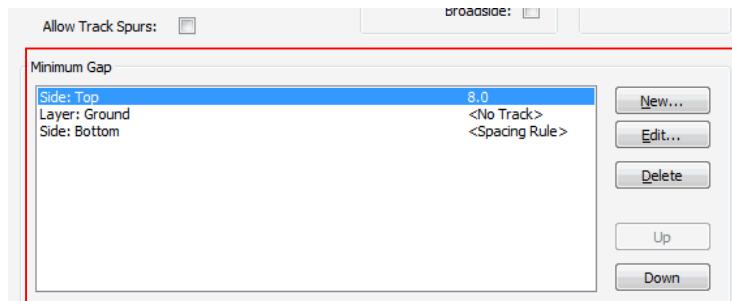
Both start and end points of a Differential Pair have to be inside the area for an area spacing rule to work. If one is outside, the default spacing rules apply and a Design Rules Check error will be created if the Differential Pair encroach on the area.

By pressing the **New** or **Edit** button, the **Edit differential Pair Gap** dialog is displayed:
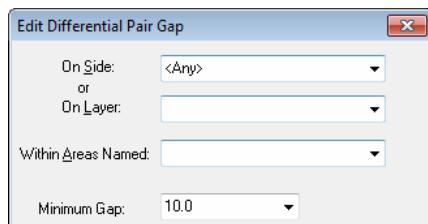
Rules can be defined by **Side** using the drop down list box:
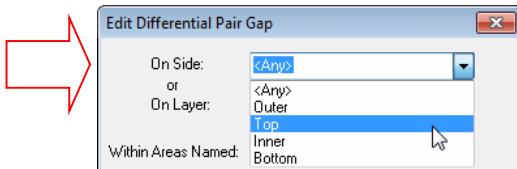
Or they can be defined by **Layer** using the drop down list box or wildcard:
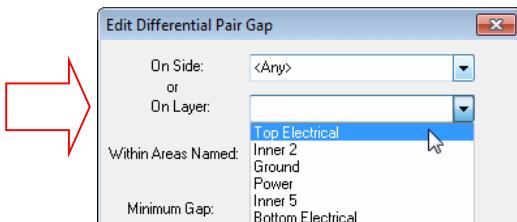
Rules can also be defined for use **Within Named Areas** or wildcard:
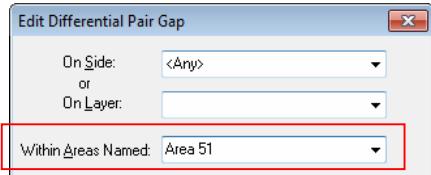
Differential Pairs - Own Colour switch

The Differential Pairs dialog now contains an **Own Colour** switch so that each set of Differential Pairs can be defined with their own colour. This highlights the whole Differential Pair path and not just the paired sections.
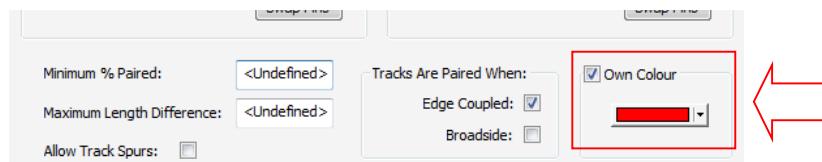
Differential Pairs - Change Layer While Mirroring Differential Pairs

You can now change layer whilst **mirroring Differential Paired** tracks. This makes it easier to add mirrored tracks out of surface mounted pads and when changing down to the layer required for the paired tracks. There is no interface change other than it now allows the **Change Layer** dialog to be used; this is a functional change.

Differential Pairs - Broadside Coupled Pairs

You can now set a **Differential Pair** to allow both **Edge Coupled** (side by side) and **Broadside** coupled tracks (on top of each other on different layers). In previous versions to V8.0, track pairing was using Edge Couple tracks, side by side with a specified gap. You can now enable Broadside coupling of tracks where the two tracks are positioned directly on top of each other on different (normally adjacent) layers. Usually you would use one or the other method, but we also allow mixing the track pairing methods.

A dialog change now allows you to define how tracks will be paired:
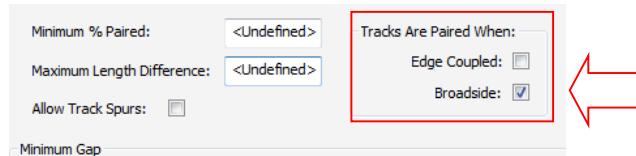
For **Broadside** only coupling, the gap can still be provided and will be used for the minimum separation of the tracks when they are on the same layer, for example when routing out of surface mounted pads but this will not contribute to the % paired.

Adding Broadside Differential Pair Tracks

When adding a Broadside Differential Pair track, using **Start Differential Pairing** will prompt for the **Broadside Clone Layer**. This is the layer that the 'copy' of the track being added will go on.

When using **Start Mirroring Paired Track** if both tracks are attached to surface mounted pads, mirrored tracks will be added, but you will not be able to start pairing yet and the defined gap will be maintained. If you start mirroring from through-hole pads or vias, you will be asked for the layer for

the clone and the mirrored track will change to that layer. Now the mirrored tracks will be allowed to come together and when this happens a click will start broadside pairing, one track on top of the other.

Changing Layers of Broadside Differential Pairs

While using Broadside Differential Pairs, you can change the layer for both the track being edited and its broadside clone at the same time.
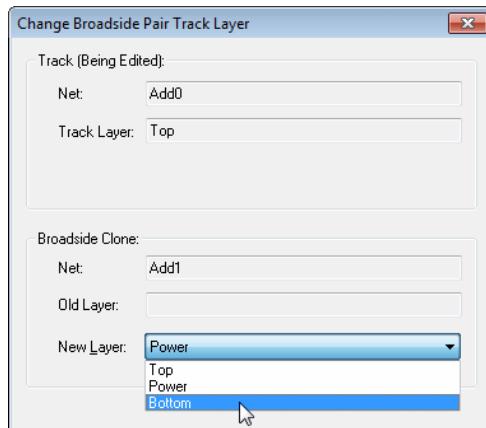
Note: This dialog is sometimes displayed when you can only change the clone layer, for example when starting differential pairing on a broadside diff pair.

Switching between Differential Pair types

When adding differential pair tracks that allow both coupling types, you can switch between them using **Switch To Broadside Coupling** or **Switch to Edge Coupling** from the context menu.
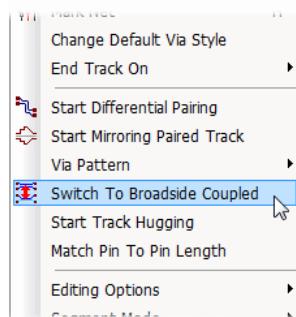
This can only be done when not actually adding paired tracks, so you have to un-pair, change coupling type and re-pair again.

When switching to Broadside coupling you will again be asked for the layer for the broadside clone track.
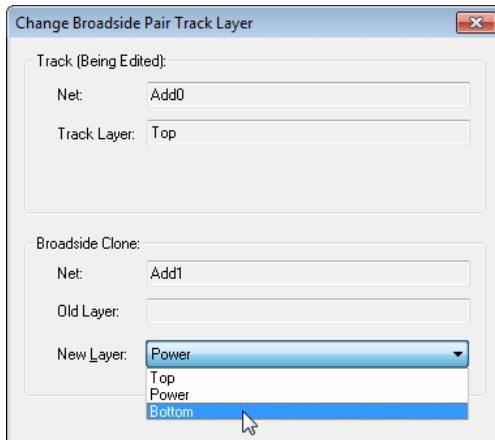
For Broadside coupling the paired length will be calculated where one track is completely over the other, but if both edge and broadside coupling is allowed, both types of paired sections will be added together to give the total figure.

Change to Serpentine Routing

You can now enter an amplitude less than half the cycle length. This allows better definition of PCI Express format serpentine tracks.
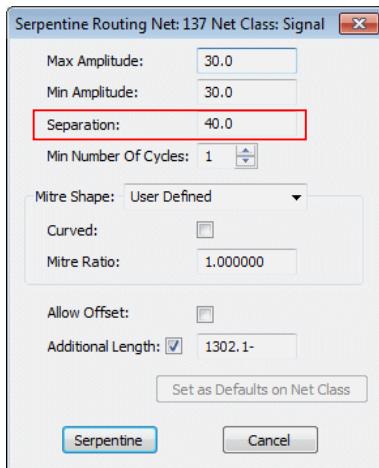
This allows this type of pattern to be produced:
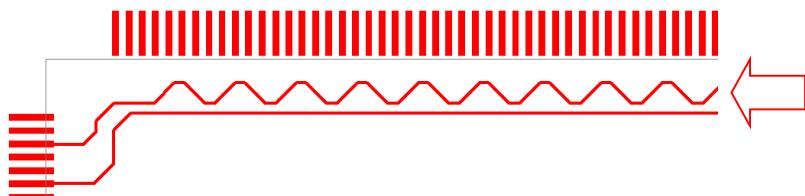
Cadence Allegro PCB Design Import (Cost Option)

As part of the **Advanced Import** cost option, you can import Cadence Allegro PCB designs. This feature was introduced into a late version of 7.6.
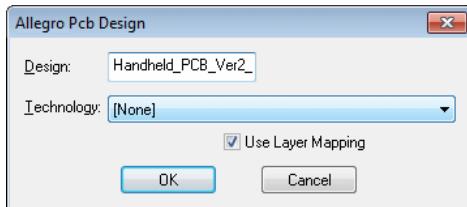
You can import Footprint libraries and Part libraries once the Parts have been saved to a PCB design, they cannot be imported directly. Please contact sales or your local Pulsonix distributor for more information.

Allegro is a trademark of Cadence Corp.

New Format Files for Report Maker

New **Format** files used for the **Report Maker** option have been added and will be installed as part of the V8.0 update or new product. These are detailed below:

Quick Rollup BOM Report

An example Quick Rollup BOM report file to produce a report detailing the following:

```
Part Name : Qty : Component Reference on single lines across the page width
```

Rollup Parts List Report

An example Rollup Parts List report file to produce a report showing:

```
Component Name : Part Name : Description : Footprint : Part Number
```

Report is in columns with Component Name column set to 5 references wide only

Rollup Parts List CSV Report

An example Rollup Parts List report file to produce a report in CSV format showing:

```
Component Name : Value : Part Name : Description : Footprint : Part Number by line
```

Variant Parts List Report

An example Variant Parts List report file to produce a report detailing the following:

```
Name : Value : Qty : Footprint : Part Number : Manufacturer : Description :
Comments for fitted and not-fitted parts on the current variant
```

New Example Designs

LED Display Example Designs

A new example design created within version 8. Two schematics, before and after re-numbering and four stages of PCB: LED Display

1: Translate to PCB, Auto - Component Groups to Area, LED Display

2: Initial Placement from Areas, Tight Groups, LED Display

3: Routing, Apply Vias, Teardrops, Auto Testpoint, Copper Pour, Auto Rename, Callouts, Layer Stack-up, Embedded View.

CharacterLeds Schematic Block

A schematic block used in the LED Display example design.

LED_Block Schematic Doc Symbol

A schematic block symbol used in the LED Display example design.

Library Changes

New Library Parts Added

New Library Parts have been added to the MasterLibraries to support the new LED Display examples. All parts are to a standard and include manufacturer and height attributes as well as multiple Parts where appropriate.

1N4148, 2N2222, 489D106X0010C1 (range), CD4017BE, KP-2012PBC, MKS4/0.01uF/63/10%/PCM7.5 (range), NE555N, RM-065-100K (range)

Revised Library Footprint

The standard footprint libraries have been updated to support the new LED Display example designs. For example, the R 0.500 footprint has been amended (the placement area has been reduced).