



Pulsonix Design System

V4.6 Update Notes

Copyright Notice

Copyright © WestDev Ltd. 2000-2007

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

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

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

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

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

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

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

All other trademarks are acknowledged to their respective owners.

Pulsonix, a division of WestDev Ltd.

Printed in the UK Issue date: 05/09/07 iss 2

Pulsonix

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

Phone +44 (0)1684 773881

Fax +44 (0)1684 773664

Email info@pulsonix.com

Web www.pulsonix.com

Contents

CHAPTER 1. GETTING STARTED WITH V4.6.....	5
About these notes	5
Installing the New Version of Pulsonix.....	5
Licensing.....	5
System Files.....	5
New Additional Supplementary Files	5
CHAPTER 2. GENERAL OPTIONS.....	7
Add To Net.....	7
Area Properties.....	7
Auto Rename.....	8
Plotting Enhancements.....	8
Zoom and Pan Controls.....	9
Report Maker Changes.....	10
Changes to Find Browser.....	10
Hyperlinks.....	11
Drawing / Printing.....	12
Component Variants.....	12
Database Connection Extensions.....	16
CHAPTER 3. SCHEMATIC OPTIONS.....	17
Electrical Rules Check	17
New Pin Type.....	17
Block Instance Attributes.....	17
Save Schematic Design to Block File.....	18
Cross Probe Open Schematic Page.....	18
CHAPTER 4. PCB OPTIONS	19
Intelligent Gerber Import.....	19
Generate Component.....	19
Pad To Mounting Hole.....	19
Layers Bar.....	20
Design Rule Check.....	20
Copper Properties.....	21
DRC Error Browser.....	22
Thermal Pad Rules.....	22
Toggle Display Connection.....	23
Improvements to Interactive Track editing.....	23
Changes to Dimensions feature.....	23
ODB++ Output changes.....	25
Changes to CAM Plots option.....	26
Differential Pair Changes (High Speed option).....	26
CHAPTER 5. PULSONIX DATABASE CONNECTION (PDC)	31
New Cost Option.....	31
PDC Setup.....	32
PDC Checking.....	32
PDC Migration Tool.....	33
APPENDIX A. SUPPLEMENTARY FILE CHANGES.....	34
Changes to Format Files.....	34

Chapter 1. Getting Started With V4.6

About these notes

These update notes are provided for existing users as a supplement to the existing Pulsonix Users Guide. These notes are to highlight new features in version 4.6 and to briefly describe their use.

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

Installing the New Version of Pulsonix

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

As with any other installation of Pulsonix, insert the CD and wait for a short time. The *Autorun* facility will start the installation procedure. Follow the on-screen messages from the install wizard. You can install Pulsonix version 4.6 on top of your existing installation, you do not need to uninstall any old version first.

Licensing

If you are already using Version 4.5, you will find that Version 4.6 will run without any change to licensing. The only exception to this is if you are adding any more 'options' (such as the Database Connection), in which case a new license file will be supplied to you with the update or by email.

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

System Files

There are no new system files for this release.

New Additional Supplementary Files

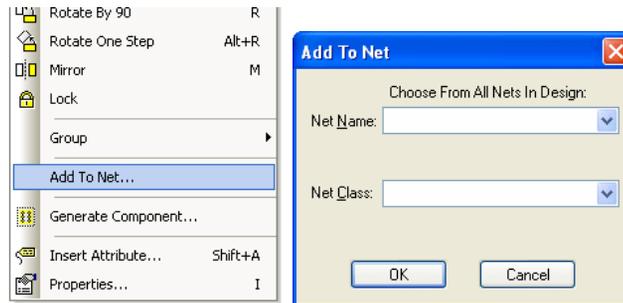
Details of all additional files supplied are supplied in Appendix A at the end of this document.

Chapter 2. General Options

Add To Net

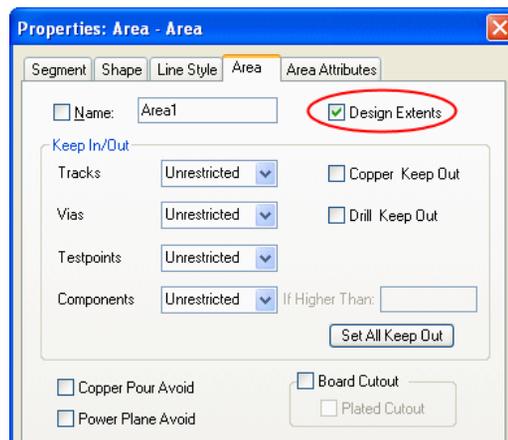
Add to Net can now be used on multiple items.

With multiple items selected, the Add to Net option is available on the shortcut menu.



Area Properties

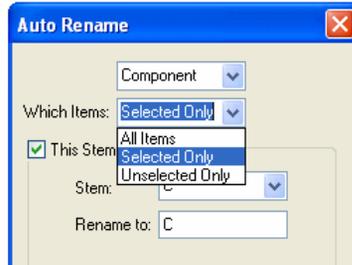
You can now designate an area as the design extents, when you do **View All**, this is the area shown. You can add multiple **Areas** with the **Design Extents** check box selected. You could add two areas with design extents checked. When **View All** is selected, these two areas would then both be in-view.



This feature is used for **View All** only and is not used within the CAM Plot option.

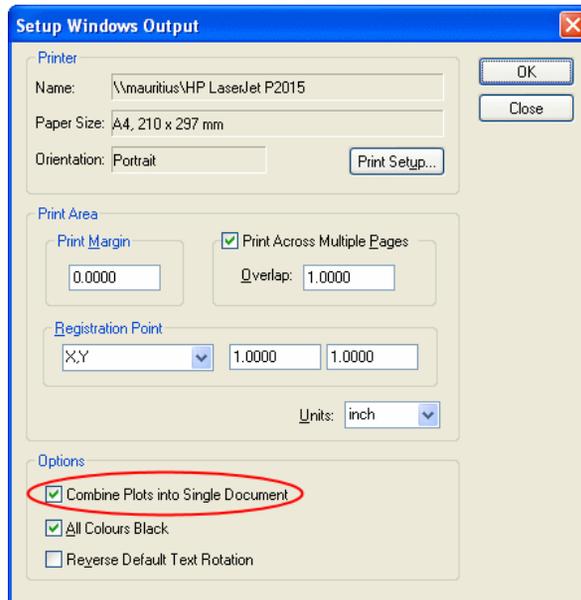
Auto Rename

Can now rename **Selected** items only (and **Unselected** items only) using the **Which Items:** drop down list on the **Auto Rename** dialog. The **Which Items:** list box is greyed out if no items are selected.

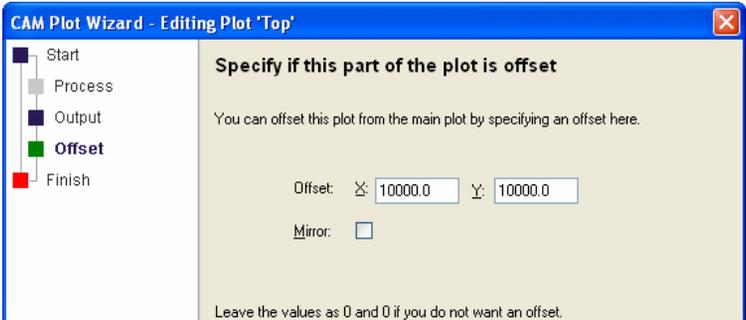


Plotting Enhancements

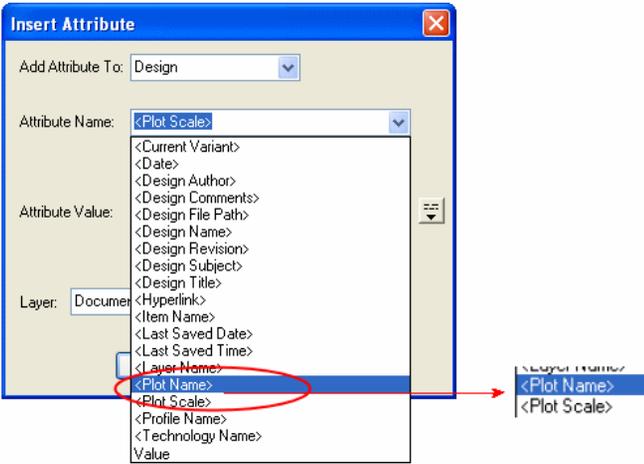
When outputting to Windows printers, you can choose to generate the output as single or multiple documents in the printer queue. This option is selected in the **Windows Setup Output** dialog located in the **CAM Plot** option and **Setup** tab.



You can offset additional layers on a plot, so you can create a composite of multiple layers shown side by side. The **Offset** page is available in the **CAM Plot Wizard**. You can define the offset and a **Mirror** flag.

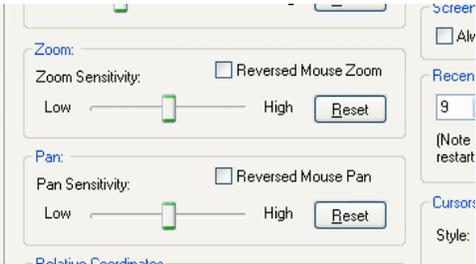


You can now display attributes <Plot Name> and <Plot Scale> to show the plot name and scale when the design is post processed. These two new attributes are added to the design as unfulfilled Attribute Positions that become fulfilled during plotting. They will then appear on the plots.



Zoom and Pan Controls

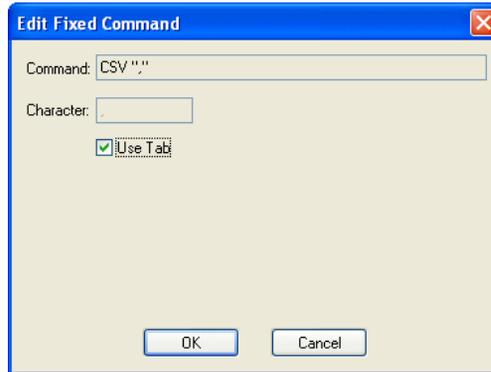
You can control the sensitivity of the Pan command (mouse wheel and shortcut keys), and the 'direction' in which Zoom and Pan commands operate. These are available on the **Options** dialog and **General** tab. Changing the slider will make the pan factor finer or coarser as preferred.



Report Maker Changes

General changes

You can now use a tab for the **CSV** command. The CSV command is edited and the **Use Tab** box checked.



You can now run a **List Of Nets** command for a doc symbol. This was specifically added to allow you to report all the net names on a star point. Also, you can now do **IF** on the **Type** field for a doc symbol. This is used to check if it is a **Star Point** for example.



New format files

Star Points – lists the positions of the **star points** in the design along with their nets.

Find Pad Style - (only in extra formats). Prompts for style name and lists items that use the style. Leave the style name blank for all styles.

Database Connection

Changes to the **Report Maker** have been made for the Pulsonix Database Connection. Please refer to the separate **Database Connection** documentation for details of these changes.

Changes to Find Browser

Symbols, Footprints and Parts

Find has new categories for Symbols, Footprints and Parts (depending on the currently open design type). Part is available as a category to make it easier to find (and select) all components of the same Part in the design.



Footprint appears on the list in PCB designs and (Schematic) **Symbols** on the list when in a Schematic design. **Parts** appear in both design types.

Filter by Attribute changes

Filtering by attribute has changed to include the **Does Not Exist** function. This is used to find items on which the specified attribute does not exist. This is different to existing and having a blank value.



Hyperlinks

You can now use attribute substitution in hyperlinks. You could use this for defining the folder path and the filename as two attributes which could then be substituted to one hyperlink. Use the substitution character defined in the **Options** dialog and **General** tab.

For example, design level attribute datasheets = 'c:\datasheets\' On a component, the hyperlink attribute could be datasheet=%%datasheets%%<Part_Name>%.pdf could produce c:\datasheets\AD8592.pdf

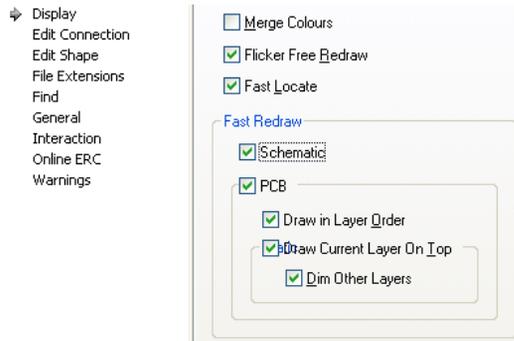
Double clicking and using **Execute** of the hyperlink still act the same.

You are now prompted with a choice if the item has more than one hyperlink (multiple hyperlinks can be defined using the **Technology** and **Attributes**).



Drawing / Printing

It is also now possible to switch Fast Graphics on/off separately for PCB and Schematics in the **Options** dialog and **Display**.



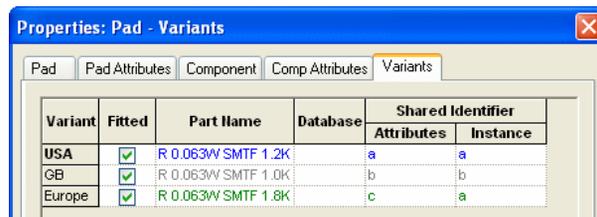
This will allow PCB to have Fast Graphics enabled as normal, but to be switched off in Schematics if the shape drawing order is not important. This still relies on the shapes having been created in the correct order in the symbol to give the correct appearance.

Component Variants

Variants has been significantly improved to now include Part variants in addition to the existing Fitted/Not-fitted status functionality.

The **Variant Manager** has not changed but the mechanism in which component variants are created has been.

When a component is selected, the **Properties** dialog now has a **Variants** tab. This tab is only available if the design contains variants and a variant is selected as 'current'.



How component variants can be used

There are four possible 'states' for component variants:

You can elect to have components fitted or not fitted in any particular variant.

- For each variant, components can have different Parts for the same Reference name, for example, R1 could be a 4308R-102 1K or a 4308R-102 2.2K.
- For each variant, the same component (R1) can have different footprints, for example, R1 Part 4308R-102 1K could have a RES300 or RES400 footprint. These could even be with a different number of pads.
- For each variant, the same component can have different Attribute values. For example, R1 with a value of 1K for the USA variant could be 1K1 for the GB variant.

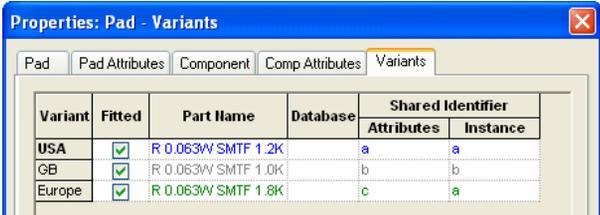
These variants can be used in any combination for each component.

Component Properties - Variants

You have made the design 'variant enabled' using the process above; you must now choose which components will be not fitted in each variant.

For a selected component, the **Properties** dialog allows you to edit its variant properties. It shows which variants the current component is fitted in and if there are Part variants. A fundamental rule of variants is that the bare board will be constant across all variants, so all pads, tracks and copper will always appear.

Parts variants allows you to have more than one component instance with the same name but different values or even a different Part or footprint in each variant. The rule is that only one component instance of a given component name can be fitted in each variant. Obviously, it can be confusing to have more than one component with the same name. In this dialog there is an additional Shared Identifier which enables you to refer to a specific instance.



Each named variant is shown for the component (USA, GB, Europe).

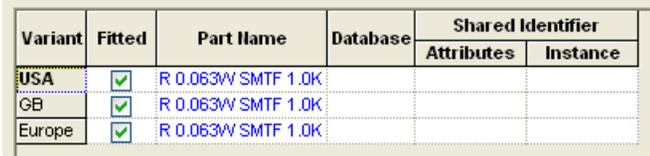
Colour coding for Properties dialog

There are a number of possible colour states to help you use the Variant Properties dialog:

The name of the **Variant** currently active in the design is shown in **bold** type. In the below example for the USA variant:



The row(s) for the component instance(s) represented by the symbol or footprint currently selected in the design are shown highlighted in blue text.



Instances that share the same symbol or footprint currently selected in the design but have different attributes or Parts are shown in green text.

Variant	Fitted	Part Name	Database	Shared Identifier	
				Attributes	Instance
USA	<input checked="" type="checkbox"/>	R 0.063W SMTF 1.2K		a	a
GB	<input checked="" type="checkbox"/>	R 0.063W SMTF 1.0K		b	b
Europe	<input checked="" type="checkbox"/>	R 0.063W SMTF 1.8K		c	a

Instances that use a different symbol or footprint to the one currently selected in the design are shown in grey.

Variant	Fitted	Part Name	Database	Shared Identifier	
				Attributes	Instance
USA	<input checked="" type="checkbox"/>	R 0.063W SMTF 1.0K		a	a
GB	<input checked="" type="checkbox"/>	R 0.063W SMTF 1.0K		b	b
Europe	<input checked="" type="checkbox"/>	R 0.063W SMTF 1.0K		c	c

Fitted Variants

Firstly, you must define the design Variants as in previous releases (using the **Variants Manager** from the **Setup** menu). Once you have done this, then the **Component - Variants** tab is available when **Properties** is performed on a component. This tab lists the Design **Variants** and shows if the current component is fitted in that variant (components are fitted in all variants by default). Use the checkboxes in the **Fitted** column to set the fitted status. The highlighted row shows the current variant.

Part Variants

A more advanced form of variants is to use a different Part for the same component in some variants. Various scenarios are possible for the same component:

- Different Parts
- Different Attributes
- Different Footprints

You may want a completely different part with a different symbol or footprint; this can be positioned and connected independently.

Alternatively, you may want the component to have different attributes, but still be represented by the same physical symbol or footprint.

To do this, you need to assign a **Shared Identifier** (Variant Id), this can be any name. There are two columns under the **Shared Identifier** heading; **Attributes** and **Instance**.

Variant	Fitted	Part Name	Database	Shared Identifier	
				Attributes	Instance
USA	<input checked="" type="checkbox"/>	R 0.063W SMTF 1.2K		a	a
GB	<input checked="" type="checkbox"/>	R 0.063W SMTF 1.0K		b	b
Europe	<input checked="" type="checkbox"/>	R 0.063W SMTF 1.8K		c	a

You can share **Attributes** by entering the same identifier in the first column, and you can share **Instances** (footprints) by entering the same identifier in the second column. You cannot share attributes, without also sharing instances. Instances can be shared provided the Parts they use are compatible. Parts are compatible if they use the same footprint.

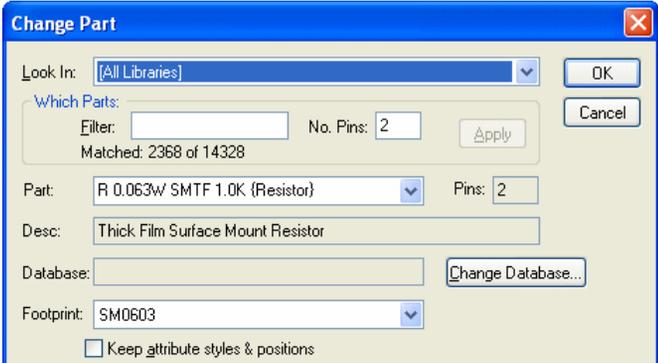
Variant	Fitted	Part Name	Shared Identifier	
			Attributes	Instance
USA	<input checked="" type="checkbox"/>	R 0.063W SMTF 1.2K	a	a
GB	<input checked="" type="checkbox"/>	R 0.063W SMTF 1.0K	b	b
Europe	<input checked="" type="checkbox"/>	R 0.063W SMTF 1.8K	c	a

In the above example there is a fitted instance in all three variants. Variant *GB*, uses identifier **b** which uses a completely different part to that used in *USA* and *Europe*, it will therefore be positioned separately. The components in *USA* and *Europe* have Attribute identifiers **a** and **c**, so can have different attributes, but share the same symbol instances, because they have the same Instance identifier **a**. Their positioning and connectivity are therefore identical.

To change the part in a specific variant, right click and select **Change Part** on the appropriate **Part Name**, or simply double click in the Part cell.

Variant	Fitted	Part Name	Shared Identifier	
			Attributes	Instance
USA	<input checked="" type="checkbox"/>	R 0.063W SMTF 1.2K	a	a
GB	<input checked="" type="checkbox"/>	R 0.063W SMTF 1.0K	b	b
Europe	<input checked="" type="checkbox"/>	R 0.063W SMT	Change Part...	

The Change Part dialog allows you to change the Part and/or footprint.



Component Variants Attributes

As an aid to identifying the component instances displayed in the design you may find it useful to add a **<Component Variant Id>** attribute position to be displayed alongside their symbol or footprint. This can be added to the symbol or footprint definition. This will display the appropriate 'Attributes Shared Identifier' for the component instance the symbol or footprint represents in the current variant (not the Variant Name).

You can also assign a **<Fitted In>** attribute to a symbol or footprint definition. This will display all variants the component instances represented by the symbol or footprint are fitted in. The variants in which the symbol or footprint represent a different component instance (because this has a different Part or Attribute) are shown at the end enclosed in brackets, for example, *first_variant,(second_variant)*

Current Variant Attribute

A <Current Variant> attribute can be added to a design, this will display the name of the 'Current' variant selected in the design.

Database Connection Extensions

These additional features are available as part of the Pulsonix Database Connection cost option.

Using the Database with Part Variants

When using the Database Connection, there is an additional level of integration between database Parts and Pulsonix Parts. The additional information is held as attributes, therefore sharing the same Part should also share the same attributes.

Variant	Fitted	Part Name	Database	Shared Identifier	
				Attributes	Instance
USA	<input checked="" type="checkbox"/>	1KAB10E		values1	values1
GB	<input checked="" type="checkbox"/>	2KBB40		values2	values2
EU	<input checked="" type="checkbox"/>	1KAB10E		values3	values1

To be able to use different database parts as part variants the component must be at least set up to have a different **Attributes Shared Identifier** in each variant. This is because the database information is stored as attributes on the component and needs to be different for each variant. In this case, changing the database record for one instance will change it for all of its shared instances also.

If the component is already using different symbols or footprints and is set up with a different **Instance Shared Identifier**, using database parts will be no different to using parts directly from the Pulsonix Library as there will be a different symbol for each variant to which to attach the database attributes.

Additional Database Column

This displays the database key field information for each variant and is useful for seeing if different database Part variants are being used even if the Pulsonix Library part is the same across variants.

If the Part Name is changed for a variant, unless is chosen via the Database Browser method (shown below), any existing database information will be lost and it will revert to being a 'local' component. For a 'local' component, the database column will be blank.

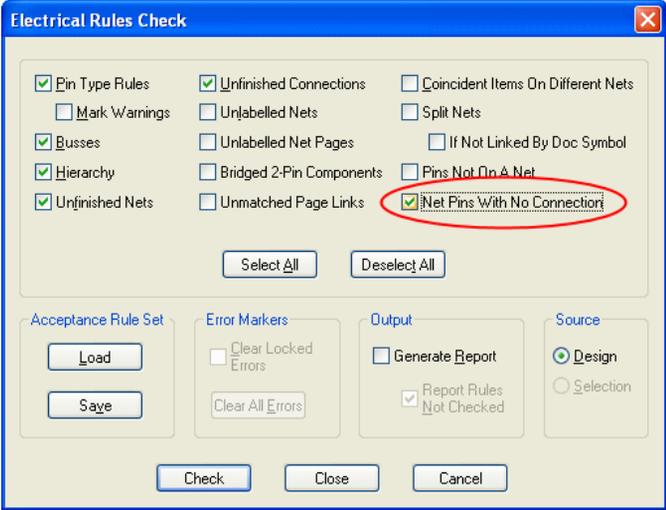
Field	Operator	Value	Value2
<input checked="" type="checkbox"/> PARTNAME	=	R 0.063W SM	

PARTNAME	PART NUMBER	VALUE	TOLERANCE	POWER	VOLTAGE	MFR	CATEGORY	IN_STOCK
R 0.063W SMTF 1.2K	R 0.063W SMTF 1.2K	1.2K		0.063W			Resistor/SM Track Film	1

Chapter 3. Schematic Options

Electrical Rules Check

This check will flag a pin which has been Net Named and does not have a connection attached to it.



New Pin Type

A new **Pin Type** of **PCB Connect** is available in the **Technology file** and **Pin Types** page.

Pin Type of **Pcb Connect** defines an ungated pin as connected in the PCB, it is not an error for the ungated pin to not be connected in the Schematic. This would typically be used on a pin which is connected through the internal connectivity of the footprint, and is of no concern to the Schematic designer. If the pin is connected in the Schematic, then it must be connected to the same net in the PCB.

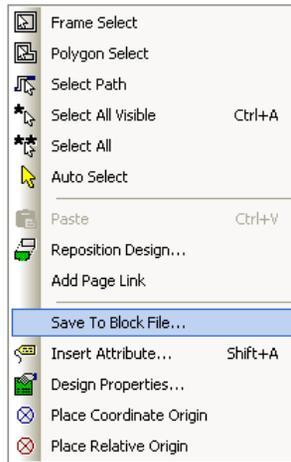
Block Instance Attributes

An attribute instance can be satisfied by any of the enclosing block instances.

So if a block instance has an attribute of `Block_Name=Processor` and a component in the block has an unsatisfied attribute position (including substituted attributes) for `Block_Name`, then it will be satisfied by the block instance and `Block_Name` will show as `Processor`.

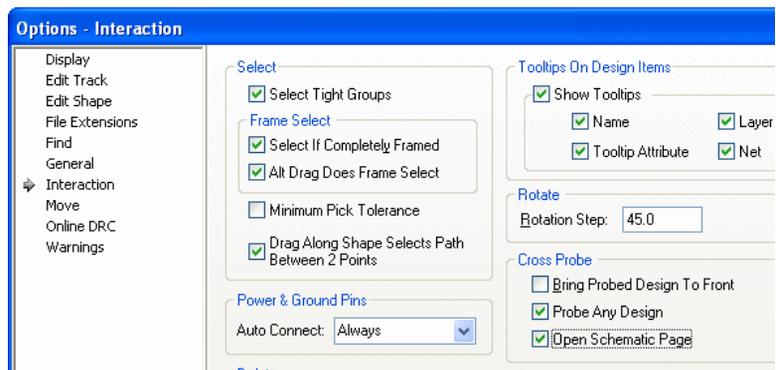
Save Schematic Design to Block File

Save to Block File is now available on the context menu with no Schematic block selected. This saves the whole design to a block file. This is used mainly for imported designs. Where the design contains multiple sheets, these will also be saved to the block file.



Cross Probe Open Schematic Page

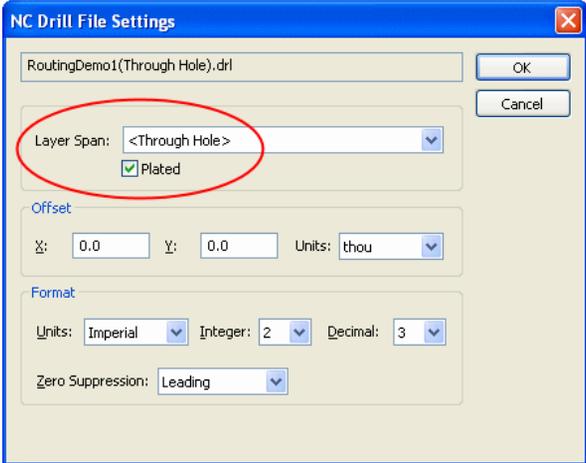
There is a new option to cause cross probe to open a Schematic page if the page is currently closed. This is available in the **Options** dialog and **Interaction** page under **Cross Probe**.



Chapter 4. PCB Options

Intelligent Gerber Import

The **Intelligent Gerber Import** option now allows you to specify the layer span and plated state of holes loaded from a drill file.

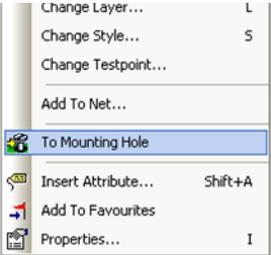


Generate Component

You can now include mounting holes in the component generation. If mounting holes are included in the selection, they are retained as mounting holes and the pad number in the Component Generation dialog excludes them in the total count.

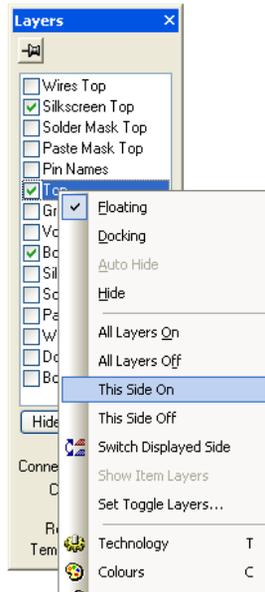
Pad To Mounting Hole

The Pad to Mounting Hole command is now available in PCB designs as well as Footprints. This is available on the shortcut menu for a selected free pad.



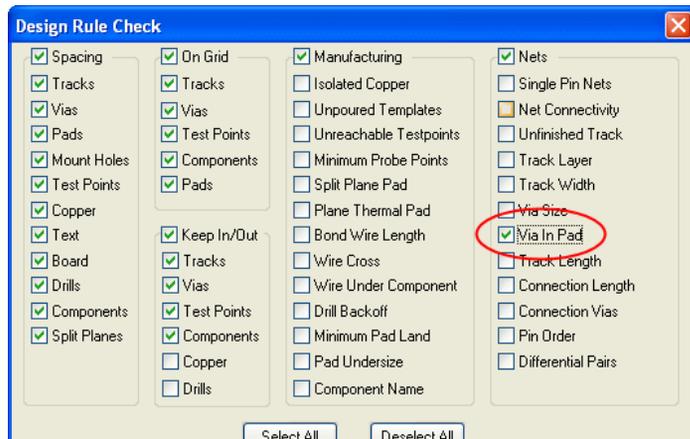
Layers Bar

You can now toggle the side of the selected layer on or off using the **This Side On** or **Off** option from the shortcut menu. All layers on the same side as the selected layer will be switched on or off.



Design Rule Check

You can now check for vias in SMD pads on the same net using the **Via in Pad** option under **Nets**. This gives you a Via in Pad error if the via touches the pad (VP).



The **locked status** of an error is now shown in the report description.

Drill to Drill check for coincident holes

You now have an option when performing drill to drill checks for flagging an error for coincident drill holes that are the same size. In Version 4.5, these were not flagged. This option is available on the **Spacings** dialog in the Technology. Select the option **Allow Coincident Holes** in the **Drill to Drill Space** box.



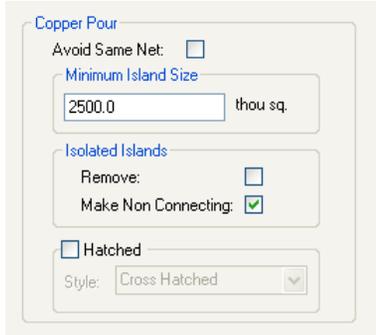
Copper Properties

You can make copper **Non Connecting**, which means when on a net it can be isolated without showing an unconnected net error and optimise does not create a connection to it. The Net Completion report will not report this as an error (even though the copper is connected to other items by virtue of its net name).

Each shape can be made non-connecting using the **Properties** dialog, you can also use this dialog to uncheck a particular shape.



In particular this status can be set on all isolated poured copper through the **Technology** and **DFM/DFT** rules.

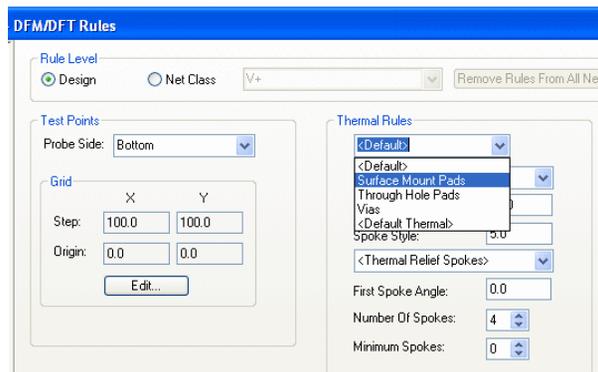


DRC Error Browser

When you lock an error from the context menu of the errors bar, the locked error tree is now expanded as well as the current tree that you have been going through. This leaves the next error selected so that you can work down the list locking errors without having to reopen the branch each time.

Thermal Pad Rules

You can add design level **thermal pad rules** specific to Surface Mount pads, Through Holes pads and Vias. In the **Technology** dialog, the **DFM/DFT** page has been reorganised accordingly.

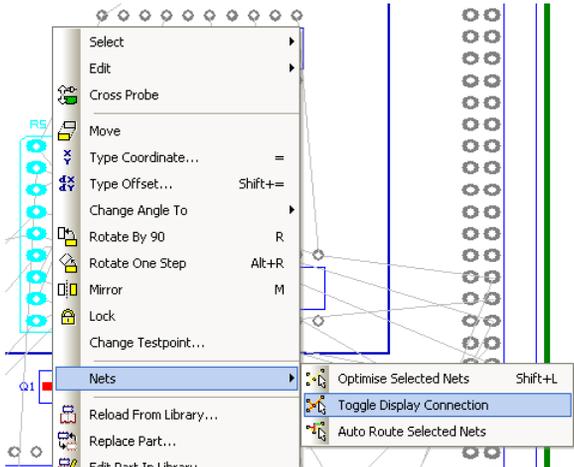


To determine how a thermal pad will be rendered, the following sequence is applied:

1. If the pad Properties defines *Isolated* or *Not Isolated* then no thermal pad is created.
2. If the pad is contained within a Copper Template, then the template may override the **Net Class** otherwise the **Net Class** is taken from the **Net**.
3. If the **Net Class** has rules defined, check for rules defined for a specific pad attribute which defines a sub net.
4. Check if the **Net Class** defines a rule for the specific pad type (through-hole, surface mount or via).
5. If the **Net Class** has a **Default** rule defined, use that.
6. Check if the design level defines a rule for the specific pad type (through-hole, surface mount or via).
7. If no other rules apply, use the **Default** design level rules.
8. If the Pad Properties - Power Plane Connection is set to Thermal Pad and no Thermal Pad rules can be determined using the above, then the **Default Thermal** rule, defined at the design level, is used.

Toggle Display Connection

The **Toggle Display Connection** command will switch the displayed connections to just those currently selected. This is available on the shortcut menu under **Nets**>



This option works for selected connections, nets and selected components. It can be added to a shortcut key using the **Customise** dialog.

Improvements to Interactive Track editing

The Edit PCB track feature using the Alt key to temporarily switch of continuous online DRC whilst editing, has now been added to the other interactive options such as Move, Edit Arc, Mitre etc.

This allows you to keep the continuous online DRC mode on, but occasionally put an item down in error, and then resolve it later.

Changes to Dimensions feature

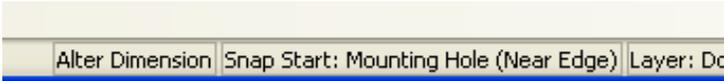
Select Next and **Select Previous** are now available after picking an item to **Snap** a dimension to. This allows you to cycle through the items close to the pick point in order to snap to the correct one.

There is now an extra 'phase' to stretching an existing dimension. If you have **Snap to Item** on and you drop the end of the dimension over more than one item, the cursor will change to include a question mark and you will enter a phase where you can use the context menu to perform **Select Next/Previous** to change the item.

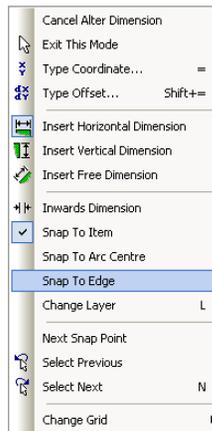


A left click or <Esc> key will keep the dimension as is, and re-enter select mode.

The type of item snapped to is now shown on the status bar as you are adding the dimension.

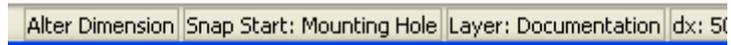


A new command on the context menu while adding dimensions, **Snap To Edge** enables you to snap the dimension to an edge of a line or pad, the default is the **near** edge. This is available for linear and radial dimensions.



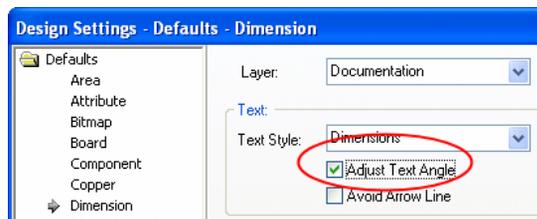
A new command on the context menu, **Next Snap Point** enables you to change the snapped end of a dimension between the near and far edges of an object and its centre. If the item is a pad, mounting hole or via with a drill hole, you will be able to set the snap point to either edge of the drill hole and, if the drill position is offset, to the drill centre.

The type of snap point is indicated on the status bar in round brackets.

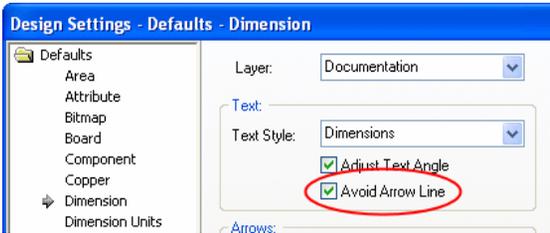


A new command, **Add Dimension**, replaces **Add Free Dimension** on the PCB toolbar. This adds a dimension of the same type as last added. The default for new users will be **Horizontal** dimension.

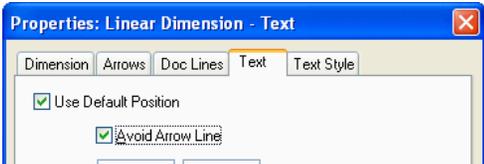
There is a new dimension default of **Adjust Text Angle** available on the **Design Settings** dialog and **Dimension**. Check this to make any new vertical dimensions added have a text angle of 90 degrees. Leave it unchecked to keep all vertical text horizontal.



There is a new dimension option **Avoid Arrow Line** available on the **Design Settings** dialog and **Dimension**.



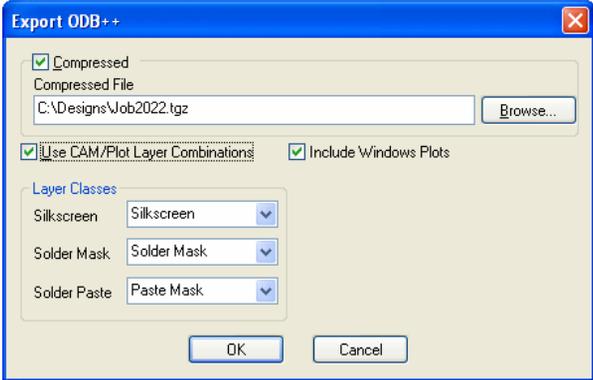
This appears as a default value for new dimensions and can be changed locally on a dimension using **Properties**. If checked the default dimension text is placed just to the far side of the dimension by the text gap, thus avoiding the line between the arrows. Leave it unchecked to have the default text placed between the arrows, splitting the arrow line as used to.



If when adding a horizontal or vertical dimension you use <Shift> select to snap on a design shape, you will now get the extreme point of the shape closest to the pick point. This enables you to easily pick the left hand side of a board outline without picking the exact left most point.

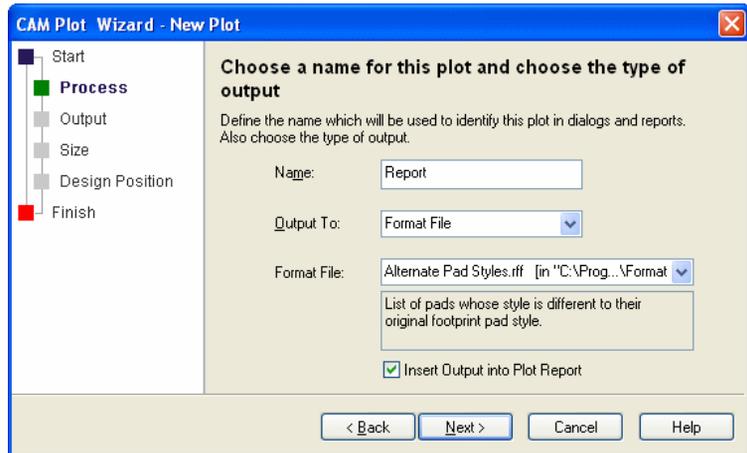
ODB++ Output changes

You now need to specify which **Layer Classes** are used for the silkscreen, solder resist and paste types. You can now output a positive plot if required (but not if using the CAM Plots option). If there are buried tracks you can now get composite positive and negative plots.



Changes to CAM Plots option

There is a new switch when defining a format file plot to **Insert Output into Plot Report**. Check this to write the output of the format file into the plot report. Leave it unchecked to write to a separate file (as it did previously).



Differential Pair Changes (High Speed option)

Gridding the 'Gap'

Differential Pairs are now defined by the path between them, rather than defining one track and cloning the other. When editing paired tracks the centre-line of the gap between the tracks is edited and placed on-grid, and the tracks are cloned either side at the required distance.

This is a more consistent method of working and means the track ends are more likely to be the same perpendicular distance from their pads.

Existing designs with differential pairs will be automatically converted to create the centre-line gap shape between the paired tracks. The only difference you will experience if editing these is if you alter the actual differential pair track it will 'jump' so that the centre line gap is on-grid. This allows you to edit existing differential pair tracks to make them equi-distance from their pads.

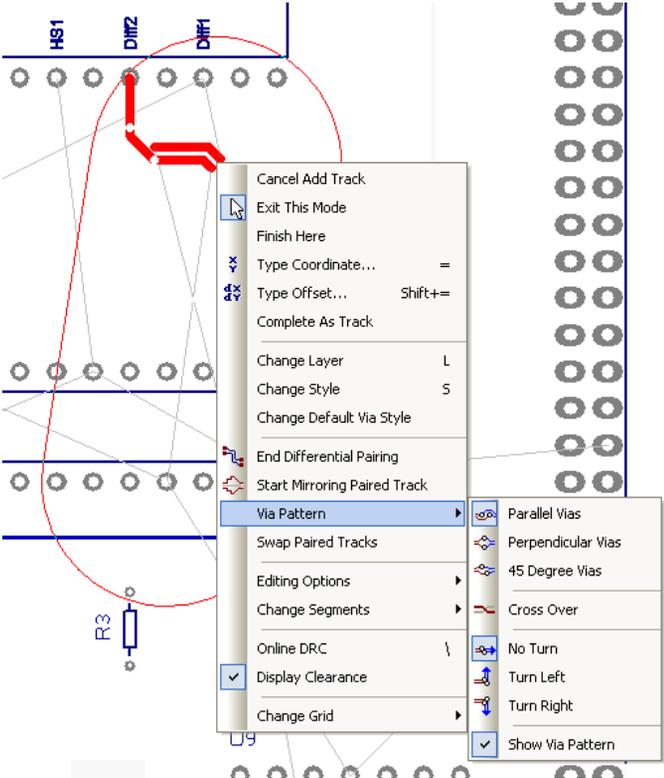
Changing Layers

Whilst adding paired tracks you will now be able to change layer using the usual **Change Layer** option or shortcut key <L>. When a layer is changed the track pairing will be automatically stopped and a via pair pattern will added prior to track pairing continuing on the other layer.

The vias are added on the via grid and the paired track is offset to ensure the gap centre-line is in-line with the mid-point of the two vias. An appropriate track pattern is added between the vias and the paired tracks to accommodate the track grid, track and via spacings and current segment mode.

There will be a choice of three different via patterns (**Perpendicular**, **Parallel** and at **45 Degrees**) and three different via pattern exit directions (turn **Left Turn**, **Right Turn** and

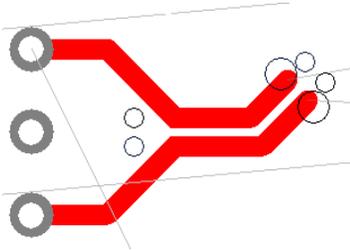
straight on, **No Turn**). Also you can choose to cross over the tracks within the pattern such that the two nets switch to the other side of the gap. All of these choices are presented on a **Via Pattern** > sub-menu off the context menu to define the pattern that will be added when the layer is changed next.



You can place any on these options on a shortcut key, but there are two additional commands provided to help.

Using **Differential Pair Via Pattern** cycles between the three via patterns (if appropriate) and **Differential Pair Via Exit Direction** cycles between the three exit directions (again if appropriate).

A representation of the via pattern that would be used if you changed layer is displayed at the end of the track pair being added. This shows both the actual via positions and start positions of the next track pair if the layer would be changed.



If online DRC is enabled and the vias would cause errors, the pattern is not shown and change layer option is not allowed at that point. A cross is drawn in the middle of one of the vias to show that the layer change will cross over the paired tracks. Showing the via pattern is optional and can be altered using the **Show Via Pattern** option on the **Via Pattern>** shortcut menu.

Whilst adding a track pair and after changing layer, the backspace key can be used to remove the via pattern and return to adding the track pair on the previous layer. This enables you to go back and change the via pattern if required.

Mirror Mode

Whilst adding paired tracks you will find an option **Start Mirroring Paired Track** on the shortcut menu. Use this to split the two tracks in order to go either side of an obstacle, or it can be used to separate the tracks to start or end them on equi-spaced differential pair pads.

Whilst in mirror mode one of the tracks will be edited (shown selected) and the other will mirror its movement about a line between them. Movement of the tracks will automatically be restricted so that they do not become closer together than the differential pair gap.

When the tracks are exactly the differential pair gap apart and left click is used to corner, mirror mode will be stopped and paired track editing re-started.

Whilst the mirrored tracks are apart you can use **End Mirroring Paired Track** from the shortcut menu to finish them where they are and continue editing a single track from one of the ends.

After using either method of ending the mirroring, you can use backspace to go back to adding the mirrored tracks.

Finishing the mirrored track being edited by clicking over the end pad will also finish the mirrored track if it is over the other differential pair end pad. Alternatively you can finish both tracks using the **Complete As Track** option.

Swapping Sides

When adding paired tracks, each track is placed on what seems like the correct side of the gap according to the current cursor position relative to the differential pair pads. If this is incorrect, the tracks can be swap with each other by using the **Swap Paired Tracks** option from the shortcut menu.

Completing Connections

This has been improved when adding paired tracks so that when **Complete As Track** is used, both dangling connections from the end of the tracks being added are routed. This is also the case when in **Mirror Mode**.

When starting a track from a differential pair pad and continuing through to add a paired track you can now use the **Complete Differential Pair Start** option from the shortcut menu to route the unrouted connection from the start of the paired track to the other differential pair start pad.

Other Changes

In select mode you can now use **Add Differential Pairing** on a selected unrouted connection between two sections of paired tracks.

Remove Differential Pairing will now leave both unpaired tracks in place (it used to remove one of them).

You can use **Unroute Track Segments** on a section of paired track to leave two connections where the paired segments were.

When double clicking on the junction at the end of a paired track, the paired track will be edited at its end allowing new segments to be added.

When adding a track pair with dangling unrouted connections going to the start of another track pair, adding a corner when the paired track ends are directly over the static paired track starts will finish the current track pair and join it to the static track pair.

Design rule check between any two tracks on different nets but in the same differential pair will use the differential pair gap instead of the track to track gap. This is true for design clearance, track hugging, measure tool etc. that uses the gap between two tracks.

There are no paired track *master* and *clone* now, so there is only one highlight colour for both paired tracks in the **Colours** dialog.

Chapter 5. Pulsonix Database Connection (PDC)

New Cost Option

Using an industry standard ODBC connection, you can now connect your corporate Parts database into Pulsonix using the Pulsonix Database Connection.

Powerful parametric searches can be made using SQL commands with a ‘human’ language interface. Simple button clicks and drop down list choices make the searches easy and scaleable.

In addition, the Report Maker has additional reporting commands to support this exciting new feature.

For further information, please refer to the additional supplementary Users Guide for this product supplied online or on the distribution CD.

Database

Search: New Open... Save As... Synchronize

Look In: Capacitor Found 112 items Search

MFR: AVX = SUPPLIER: RS* =

Add Add to Bin Update Setup... Fit Cols Help

PARTNAME	PART_NUMBER	VALUE	TOLERANCE	POWER	VOLTAGE	MFR	CATEGORY	IN_STOCK	SUPPLIER
08055C473KATDA	08055C473KATDA	47000pf	10%		50	AVX	Capacitor/SM/Ceramic	1	RS COMPONE
08050G1032ZTDA	08050G1032ZTDA	100000pf	20%		50	AVX	Capacitor/SM/Ceramic	1	RS COMPONE
08050G1042ZTDA	08050G1042ZTDA	100000pf	20%		50	AVX	Capacitor/SM/Ceramic	1	RS COMPONE
08050G4732ZTDA	08050G4732ZTDA	47000pf	20%		50	AVX	Capacitor/SM/Ceramic	1	RS COMPONE
08055C472KATDA	08055C472KATDA	47000pf	10%		50	AVX	Capacitor/SM/Ceramic	1	RS COMPONE
08055C473KATDA	08055C473KATDA	47000pf	10%		50	AVX	Capacitor/SM/Ceramic	1	RS COMPONE
08050G1032ZTDA	08050G1032ZTDA	10000pf	20%		50	AVX	Capacitor/SM/Ceramic	1	RS COMPONE
08050G1042ZTDA	08050G1042ZTDA	100000pf	10%		50	AVX	Capacitor/SM/Ceramic	1	RS COMPONE
08050G4732ZTDA	08050G4732ZTDA	47000pf	20%		50	AVX	Capacitor/SM/Ceramic	1	RS COMPONE
08055C102KATDA	08055C102KATDA	10000pf	10%		50	AVX	Capacitor/SM/Ceramic	1	RS COMPONE
08055C103KATDA	08055C103KATDA	10000pf	10%		50	AVX	Capacitor/SM/Ceramic	1	RS COMPONE
08055C104KATDA	08055C104KATDA	100000pf	10%		50	AVX	Capacitor/SM/Ceramic	1	RS COMPONE
08055C222KATDA	08055C222KATDA	22000pf	10%		50	AVX	Capacitor/SM/Ceramic	1	RS COMPONE
08055C223KATDA	08055C223KATDA	22000pf	10%		50	AVX	Capacitor/SM/Ceramic	1	RS COMPONE
0603YC104KATDA	0603YC104KATDA	100000pf	10%		16	AVX	Capacitor/SM/Ceramic	1	RS COMPONE
0805YC334KATEA	0805YC334KATEA	330nf	10%		16	AVX	Capacitor/SM/Ceramic	1	RS COMPONE
0805YC474KATEA	0805YC474KATEA	470nf	10%		16	AVX	Capacitor/SM/Ceramic	1	RS COMPONE
0805ZC105KATEA	0805ZC105KATEA	1uf	10%		10	AVX	Capacitor/SM/Ceramic	1	RS COMPONE
08055C332KATDA	08055C332KATDA	3300pf	10%		50	AVX	Capacitor/SM/Ceramic	1	RS COMPONE
08055C333KATDA	08055C333KATDA	33000pf	10%		50	AVX	Capacitor/SM/Ceramic	1	RS COMPONE
08055C471KATDA	08055C471KATDA	470pf	10%		50	AVX	Capacitor/SM/Ceramic	1	RS COMPONE
06033C223KATDA	06033C223KATDA	22000pf	10%		25	AVX	Capacitor/SM/Ceramic	1	RS COMPONE
06033C473KATDA	06033C473KATDA	47000pf	10%		25	AVX	Capacitor/SM/Ceramic	1	RS COMPONE
06033G104ZATDA	06033G104ZATDA	100nf	20%		25	AVX	Capacitor/SM/Ceramic	1	RS COMPONE
06035C102KATDA	06035C102KATDA	10000pf	10%		50	AVX	Capacitor/SM/Ceramic	1	RS COMPONE
06035C103KATDA	06035C103KATDA	100000pf	10%		50	AVX	Capacitor/SM/Ceramic	1	RS COMPONE

C2
1000pf
08055C102KATDA

C1
2
1

32 Pulsonix Database Connection

PDC Setup

Using the setup dialog, mapping between your Parts database and the PDC is easily achieved.

The screenshot shows the 'Database Connection Setup' dialog box. It is divided into several sections:

- Database Connection Configuration File:** A text box containing 'C:\Program Files\Pulsonix\Database\Parts.dcc' with 'Open...', 'Save As...', 'OK', and 'Cancel' buttons.
- Connection To Database:** 'Connection Type' is set to 'Access' (with a 'Change...' button). 'Path Name' is 'C:\Program Files\Pulsonix\Database\Parts.mdb' (with a 'Browse...' button). There are fields for 'User Name' and 'Password' (with a 'Disconnect' button).
- Database Tables and Mappings:** Two radio buttons: 'Specify which tables are visible to searches' (unselected) and 'Define table properties and field mappings' (selected). Below are dropdowns for 'Table Name' (AVX), 'Key Field' (PART_NUMBER), and 'Partname Field' (PARTNAME). An 'Apply To All Tables' button is present.
- Table Mapping Table:** A table with columns: Field Name, Show, Map To, Load, Critical, and Hyperlink. The rows are: PARTNAME, PART_NUMBER, VALUE, TOLERANCE, POWER, VOLTAGE, MFR, CATEGORY, and IN_STOCK. Each row has a checked 'Show' box, a 'Map To' dropdown with the corresponding field name, a checked 'Load' box, and unchecked 'Critical' and 'Hyperlink' boxes.

PDC Checking

Checking between the design and the database is done using the **Database Check** dialog. Setup of checks is categorised into logical areas of functionality to make the setup as easy as possible but still giving you the flexibility.

The screenshot shows the 'Database Check' dialog box. It is organized into several sections:

- Choose Items to Check:** Radio buttons for 'All Components' (selected) and 'Selected Components'.
- Update Design from Database:** A 'Check' button and an 'Update' button. Below are checkboxes for 'Allow Update of Design to Match Database' (unchecked), 'Update Component Attributes' (unchecked), and 'Update Local Component to Database' (unchecked). A 'Cancel' button is also present.
- Choose What to Check:** Checkboxes for 'Components using Local Parts' (checked), 'Attributes Different To Database' (checked), 'Check Critical Attributes' (checked), 'Check Normal Attributes' (checked), and 'Associated Parts' (checked).
- Reporting Results:** Checkboxes for 'Highlight: Components' (checked), 'Highlight Components using Local Parts' (checked), 'Highlight Mismatched Components' (checked), 'Write Report' (checked), and 'View Report on Completion' (checked). Buttons for 'Default', 'Reset', 'Setup...', and 'Remove All' are on the right.
- Save/Load Settings:** A text box for settings, with 'Load...', 'Save...', and 'Clear' buttons below it.

PDC Migration Tool

The PDC is supplied with a migration tool to migrate Parts library data to and from your database. Before the database can be used with Pulsonix, you must 'synchronise' the Parts information from the database with Parts in Pulsonix. The database contains a Part reference and other attribute information, the Pulsonix Parts library contains physical references to Schematic Symbols and PCB Footprints.

Library To/From Database

Transfer in which direction

From Database to Pulsonix Library

From Pulsonix Library to SQL

From Pulsonix Library to CSV

What to do

Do the Transfer

Just Produce a Report

Overwrite/Update existing entries

View Report on Completion

Additional Field Names

Schematic Symbol:

PCB Footprint:

Choose Library Names:

- C:\Program Files\Pulsonix\MasterLibraries\3M.pal
- C:\Program Files\Pulsonix\MasterLibraries\Actel.pal
- C:\Program Files\Pulsonix\MasterLibraries\AD.pal
- C:\Program Files\Pulsonix\MasterLibraries\Altera.pal
- C:\Program Files\Pulsonix\MasterLibraries\AMP.pal
- C:\Program Files\Pulsonix\MasterLibraries\Atmel.pal
- C:\Program Files\Pulsonix\MasterLibraries\AVX.pal
- C:\Program Files\Pulsonix\MasterLibraries\Capacitor.pal
- C:\Program Files\Pulsonix\MasterLibraries\Dies.pal

All:

Where to place results

Create One Table Per Library

Prefix:

In: ...

All Parts Into This Table:

New...

OK

Cancel

Setup...

Appendix A. Supplementary File Changes

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

Changes to Format Files

New format file examples are supplied to demonstrate the addition of commands for the Database Connection in the **Report Maker**.

- **Parts List (Database)** A Parts list that reports Parts taken from the Pulsonix Database connection product.
- **Parts List (Database) CSV** A Parts list in CSV format that reports Parts taken from the Pulsonix Database connection product.
- **Schematic Acceptance (Database)** An acceptance report that specifically checks Parts against the Database.
- **PCB Acceptance (Database)** An acceptance report that specifically checks Parts against the Database.
- **Star Points** Lists the positions of the star points in the design, along with their nets
- **Find Pad Styles** (Only in extra formats). Prompts for style name and lists items that use the style. Leave the style name blank for all styles.