



Pulsonix Design System

V7.6 Update Notes

2 Pulsonix Version 7.6 Update Notes

Copyright Notice

Copyright © WestDev Ltd. 2000-2013

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: 020113 iss 2

Pulsonix

20 Miller Court
Severn Drive
Tewkesbury Business Park
Tewkesbury
Glos, GL20 8DN
United Kingdom

Support Phone +44 (0)1684 296 570

Sales Phone +44 (0)1684 296 551

Fax +44 (0)1684 296 515

Email info@pulsonix.com

Web www.pulsonix.com

Contents

CONTENTS	3
CHAPTER 1. VERSION 7.6 UPDATE SUPPLEMENT	5
Installing the New Version of Pulsonix	5
Licensing	5
Windows 7 Support	5
Windows 8 Support	5
Report Maker – New commands	6
Symbol Origin Size.....	11
Changes to Grids.....	12
Move Cursor 10 Steps.....	13
View Actual Size	13
Calibrate Actual Screen Width	13
Defaults for Specific Attributes	15
Attribute Validation	17
Extension of Wildcard format.....	18
Changes to Component Variant Spreadsheet.....	19
Changes to search criteria in Find dialog.....	19
Scale Symbols.....	20
Additional Cursor Type – 45-Crosshair.....	20
Save Generated Block Symbol with Block Design (SCM).....	21
View Port Names in Block Symbol (SCM)	21
Unfinished Schematic Connections (SCM)	22
Measure Tool in Schematics (SCM).....	22
Changes to Construction Lines option (PCB).....	22
Tolerances in Dimensions (PCB).....	24
Changes to Radial Dimensions	25
STEP Export – No Components (PCB)	26
ODB++ Output - CAM350 Switch (PCB).....	26
Change to Optimise All Nets (PCB)	26
Optimise After Edit (PCB).....	27
Display Pad Numbers in Pads (PCB).....	27
Options - New Track Length Limits page (PCB).....	29
3+1 or 4+1 Resistor Bands for 3D Packages (PCB)	29
Reload Option - Keep local plane connect switch (PCB)	30
Changes to Apply Layout Pattern (PCB).....	30
Areas - Component Pad Keep In/Out (PCB)	31
Select Track Path option (PCB).....	31
Interactive High Speed (PCB) (Cost Option).....	32
Differential Pairs.....	32
Serpentine Routing	33
Eagle V6.3 Import.....	35
Allegro PCB Design Import (Cost Option).....	35
Library Changes.....	35

Chapter 1. Version 7.6 Update Supplement

Installing the New Version of Pulsonix

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

To install Pulsonix, insert the CD or double-click on the download executable and wait for a short time. The *Autorun* facility will start the installation procedure. Follow the on-screen commands from the install wizard. You can install Pulsonix 7.6 on top of your existing installation or along side if you prefer; however, you do not need to uninstall the old version first.

Licensing

Version 7.6 does not require a new license. If you are using a version of Pulsonix earlier than Version 7.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.

Windows 7 Support

As further confirmation, since the release of Pulsonix 6.1 (and for later versions) the release version of the Windows 7 operating system in both 32 and 64 bit variants is fully supported. No earlier Pulsonix versions can or will be supported.

Windows 8 Support

As of October 2012, Microsoft have released Windows 8 to the market. Pulsonix 7.5 and Pulsonix 7.6 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.

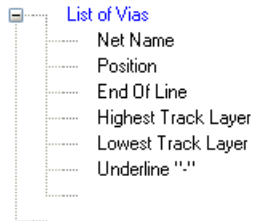
Beta or pre-release versions of Windows 8 cannot be supported.

Report Maker – New commands

Various **Report Maker** improvements have been added to Version 7.6:

List of Vias reports Highest and Lowest Track Layer

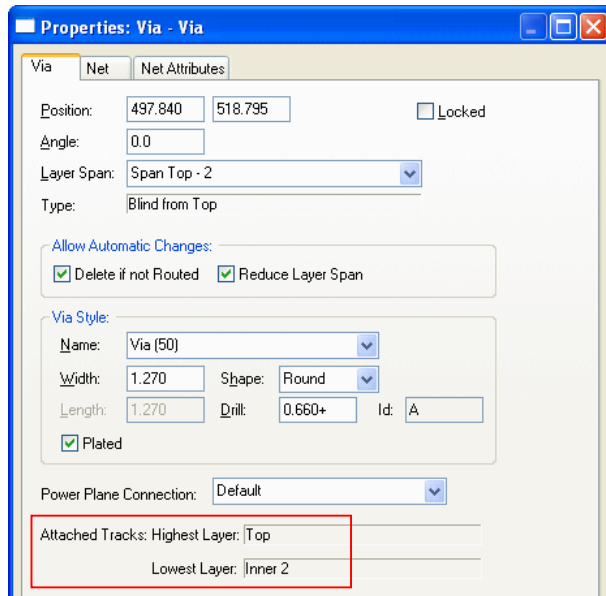
- From within the **List of Vias** command, you can now report the **Highest Track Layer** and **Lowest Track Layer** attached to a via.



A report output might look like this:

```
CLK (505.460,516.890)
Highest Via Layer: Top
Lowest Via Layer: Inner 2
-----
CLK (497.840,518.795)
Highest Via Layer: Top
Lowest Via Layer: Inner 2
-----
```

This reports the same information as in **Properties** of a selected **Via**.

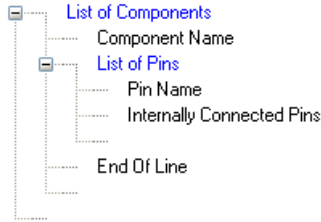


New command for reporting internally connected pins on a component

The ability to report a list of all internally connected pins in a component has been added. This command inspects all internally connected pins that are defined in the Parts library using the + separation character. This is mainly used for PCB design reports as this pin information is not available in the interface. The Pin Name would be that defined in the Part but only shows one name.

Note: this command does not report pins that are defined with a comma ',' as externally connected.

A small format example could look like this:

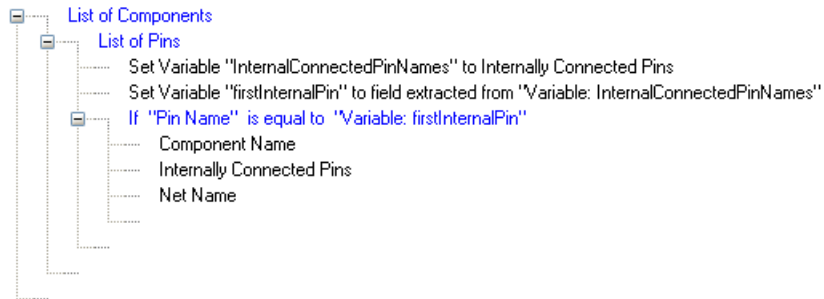


A report output might look like this if you run the above script to generate a list of all the pins on a device to show those internally connected:

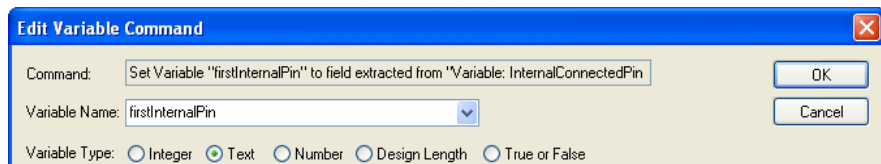
```

U2
1 1+2+5+6
2 1+2+5+6
3
4
5 1+2+5+6
6 1+2+5+6
  
```

If you want to refine the format file to only report pins on a device that are internally connected, the example below would accomplish this:

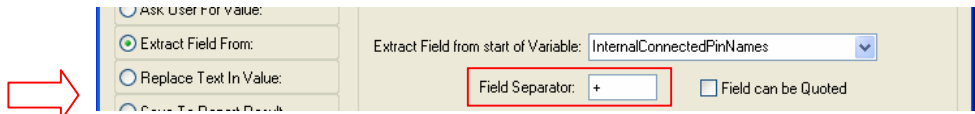


There are two variables used; the first places the Internally Connected Pins into a variable named InternallyConnectedPins. The second extracts the first pin name found and puts it into another variable named firstInternalPin.



8 Pulsonix Version 7.6 Update Notes

Within the firstInternalPin variable, the field separator + has been extracted from the InternallyConnectedPins list.



The script then tests the Pin Name (using the If statement) to see if it matches the first internal pin name, if it does then it reports the Component Name, the Internally Connected Pins and the Net Name.

When the script is run, the report looks like this, showing all the Internally Connected Pins on the device:

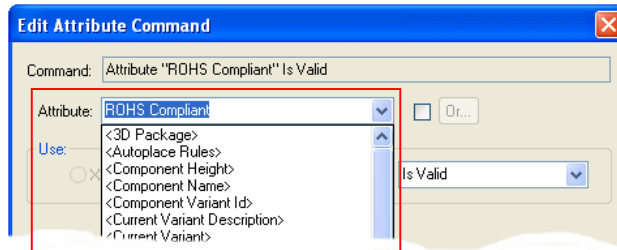
```
U2 1+2+5+6 Sig09
```

Changes to the Run Report command

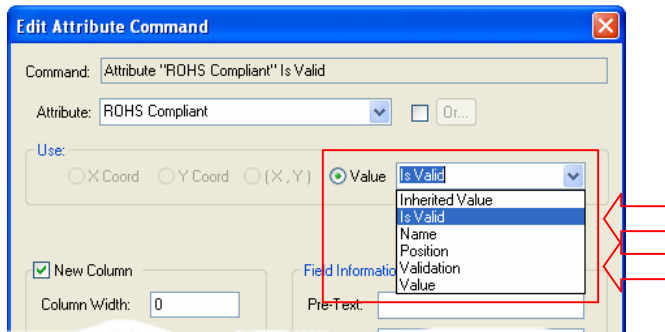
The **Run Report** command can now be placed anywhere in report script (previously it was only available as a top level command).

Changes to Attribute command and new Is Valid and Validation fields

A change to the **Attribute** command has been made. It now has a sub-fields drop down list instead of radio buttons.



New **Is Valid** and **Validation** fields have also been added to this dialog. This allows a report to be written to show where component values are not valid – see **Attribute Validation** below.



Use **Is Valid** to check if an attribute value is valid, and use **Validation** to report the validation string for an attribute.



This will report true or false in response to the **Is Valid** check. For example:

```

U2 false
U3 true
    
```

New Rotation (Transformed) command

A new **Rotation (Transformed)** command has been added. This differs from the **Rotate** command as it takes the Report Maker output rotation into consideration after using Transformation.

So for example, if a Component has been rotated in the design by 90 degrees, once transformed by 90 degrees in the report (because the whole design is transformed), using the new **Rotation (Transformed)** command, this will now report the Component as 180 degrees.



Running the above script reports the original Component Rotation first, followed by the Rotation after Transformation:

```

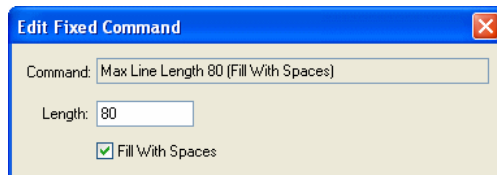
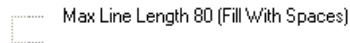
U2 90.0 180.0
U3 0.0 90.0
    
```

New command - Max Line Length & Fill With Spaces

New **Max Line Length** command that forces each line to be truncated at the supplied length.

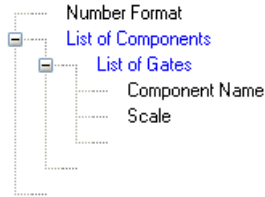
Within this new command there is an optional **Fill With Spaces** parameter command to ‘pad’ out the line with spaces if the Max Line Length has not been reached.

So for example, if a line is truncated to 80 characters using the new **Max Line Length** command (because the line length must be exactly but no more than 80 characters) but the line is only 70 characters long, then the **Fill With Spaces** command will add another 10 spaces to the line.



New command - Scale

A new **Scale** command for reporting the scale from **documentation symbols** and from **Schematic gate symbols** has been added. The format of the decimal number output can be defined by first using the **Number Format** command.

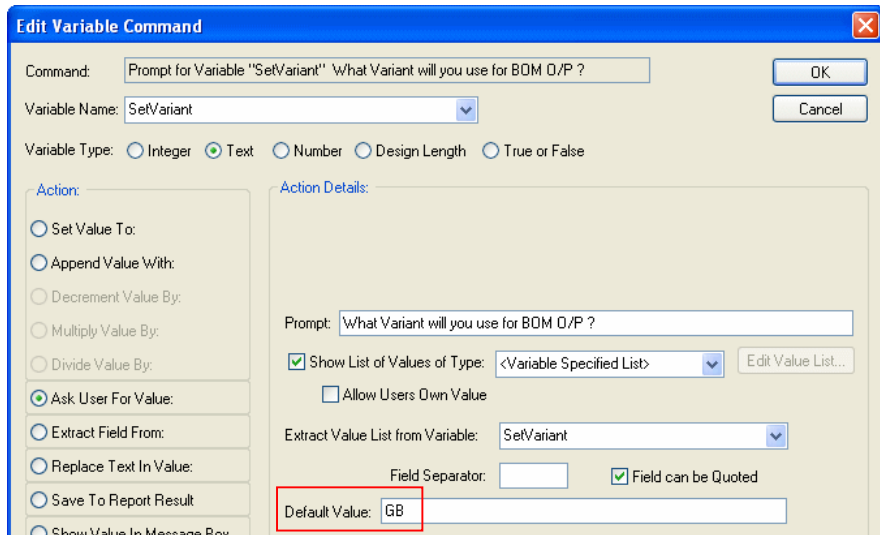


The script above would report something like this:

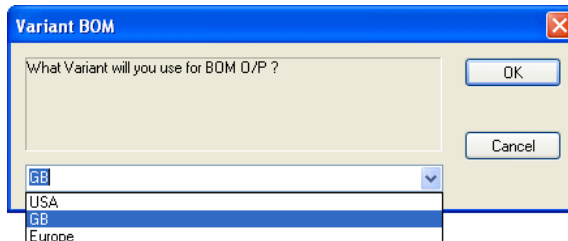
```
U1 Scale 1.000
U2 Scale 2.000
U3 Scale 1.000
```

Variable 'Ask User For Value' command now has a Default Value box

The Variable **Ask User For Value** command now has a **Default Value** box. This is presented as the default answer when the question is asked.

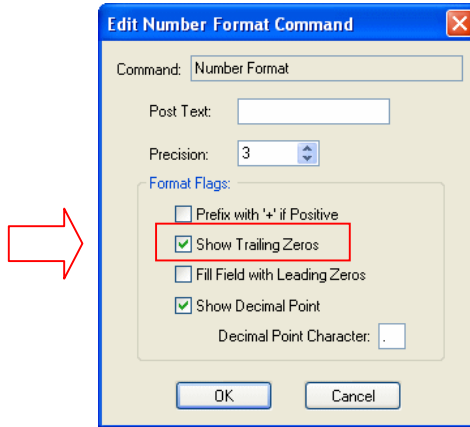


When run, the prompt is preset with a default response:



Number Format command now has a Show Trailing Zeros switch

The **Number Format** command now has a **Show Trailing Zeros** switch. For example, scales 2.0 and 1.23 are output (with precision 3) as 2.000 and 1.230 with it checked, and as 2 and 1.23 with it unchecked.



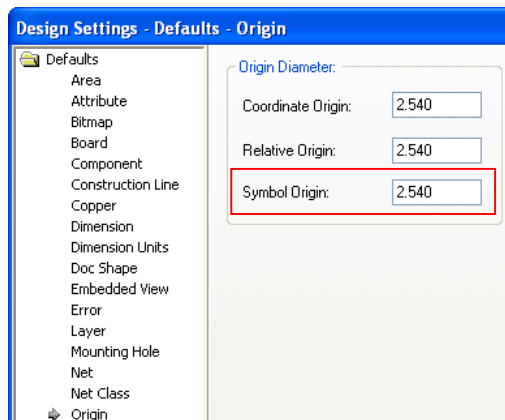
Problem fix to Line Length command

The **Line Length** command has been fixed and will now report the correct length of the current line.

Note: This fix may affect some users who use this command in their scripts.

Symbol Origin Size

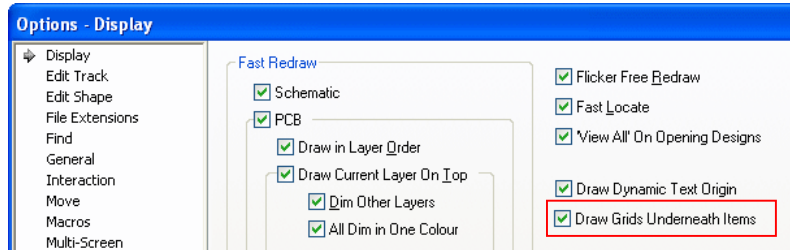
The ability to change the **Symbol Origin Default Diameter** in a PCB and Schematic design has been added so that you can set the size of Symbol Origins that are displayed on components. Changing the size in this dialog dynamically changes the size used in the design once the **Apply** or **OK** buttons have been pressed.



Changes to Grids

Draw Grids Underneath Items

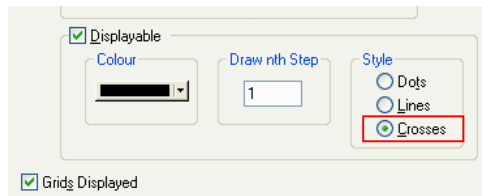
There is a new **Display** option on the **Options** dialog to **Draw Grids Underneath Items**. This is useful when displaying grid lines and working with Construction Lines.



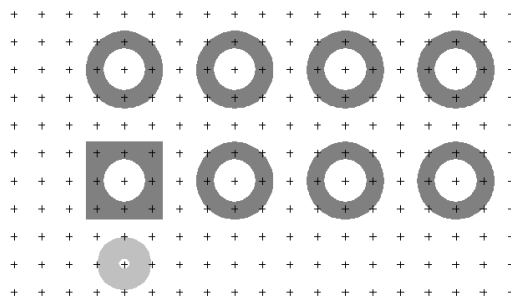
Grid Crosses

The ability to change a grid to use small 'crosses' instead of dots or lines has been added. These are easier to see than dots, but not as *visually crowded* as lines. The size of the cross is related to the size of the grid.

Use the **Crosses** radio button on the **Grids** dialog to select this option.



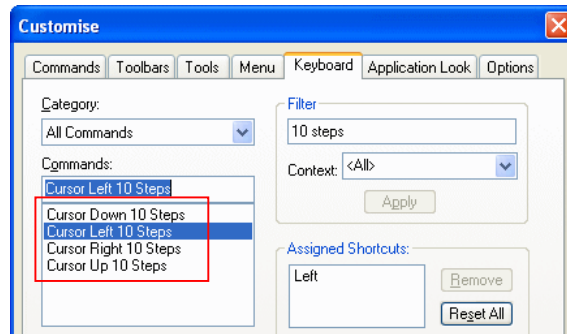
When used in the design, grid crosses look like this:



Move Cursor 10 Steps

Four new commands have been added to move the cursor 10 grid steps. These can be assigned to one of the four arrow keys for example on the extended keyboard.

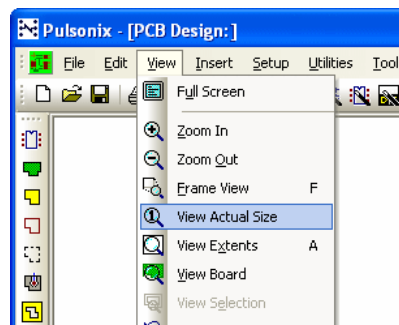
In the **Customise** dialog, choose '10 steps' as the **Filter** to see the four choices; **Cursor Down 10 Steps**, **Cursor Up 10 Steps** etc.



View Actual Size

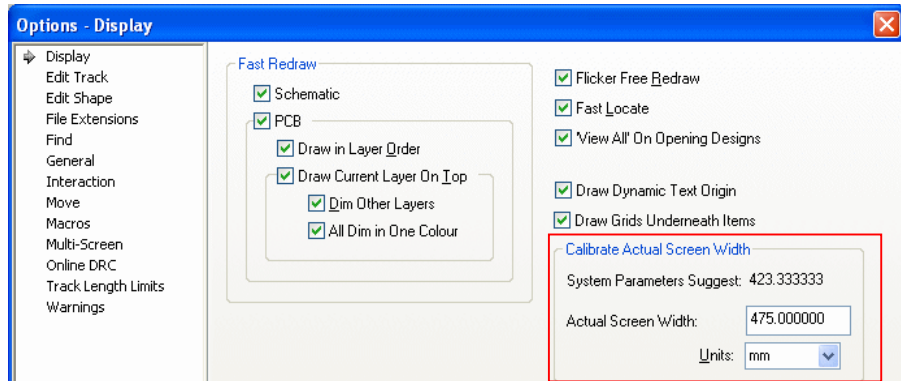
View Actual Size

From the **View** menu, select **View Actual Size** to view the design scale at 1:1 so that items on the screen are actual size. This is useful for taking a realistic perspective of your PCB designs.



Calibrate Actual Screen Width

The **View Actual Size** command on the View menu uses the Actual Screen Width to accurately display designs. By default, the screen width is calculated from system parameters and this value is shown in the **Options** dialog. If this value is not correct you can type in the screen width.



Clearing the **Actual Screen Width**: value will return it to the calculated system value.

Note that this is the width of the display area, not the diagonal, as is often quoted by manufacturers; and it may include a distance outside the visible screen, if your display does not have the correct horizontal adjustment.

You can choose the units used to display these values in this dialog.

Calibrating your screen

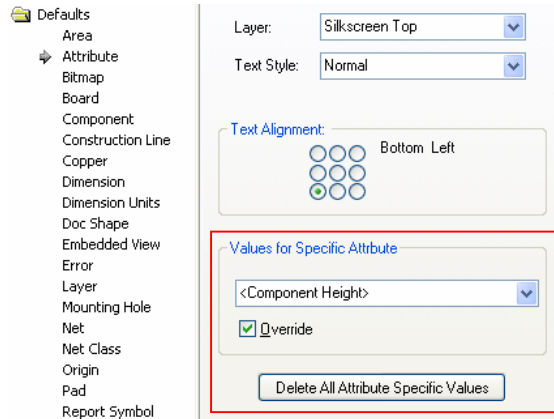
This is straight forward and needs only to be done once.

► To calibrate your screen

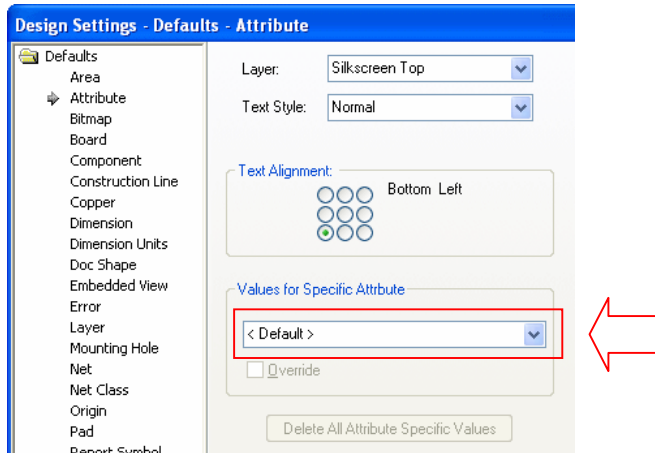
1. Within the design (Schematic or PCB), draw a shape to a known width across the screen. Draw the size so it will fit within the design area at a scale of 1:1. Make it 100mm for example (this size will fit most modern screens widths).
2. From the **View** menu, select **View Actual Size**.
3. Measure the shape width with a rule. Be careful not to scratch the screen. If the size is what you drew, then no calibration is required and you can finish. Some screens are correct, some aren't.
4. If it is not the correct size, carry on below:
5. Now physically measure the width of the screen with a rule. Measure the illuminated portion of the screen. Make a note of this value.
6. From the **Tools** menu, **Options** dialog, select **Display**.
7. Under **Calibrate Actual Screen Width**, check the value of the **System Parameters Suggest** again your 'measured' value. If they are not the same, type your value into the **Actual Screen Width** entry.
8. Press **OK** to close the **Options** dialog.
9. In the design, from the **View** menu, select **View Actual Size** again.
10. Physically measure the shape width in Pulsonix to verify it is now the shape width as drawn and shown in the **Properties** dialog.

Defaults for Specific Attributes

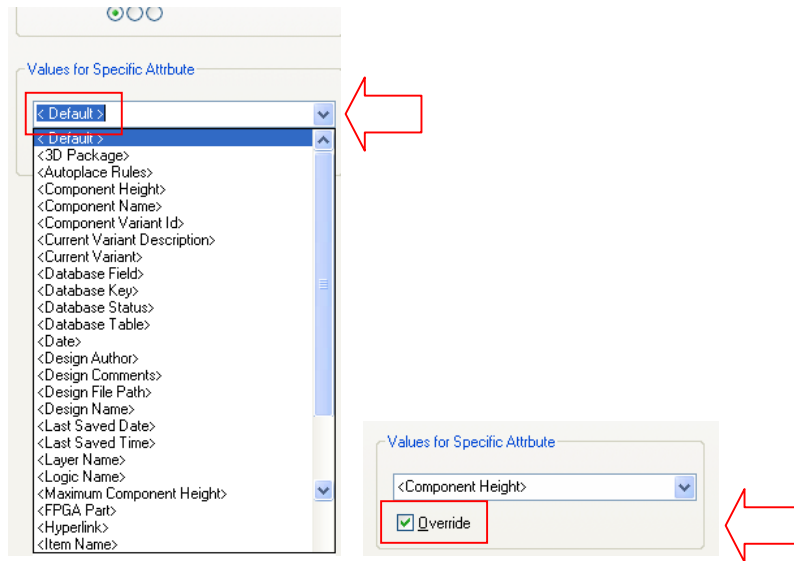
You can now define defaults for specific Attributes. You would use this to create a specific set of parameters such as the **Layer**, **Text Style** or **Text Alignment** for each Attribute.



You can also define default values for a specific **Attribute Name**. The dropdown list under **Values for Specific Attribute** is normally set to **<Default >**, which means that the above values are used for all Attributes which do not have specific values defined.



To define values for a specific Attribute, select the name in the drop down list and check the **Override** box. You can now change the values to that required.

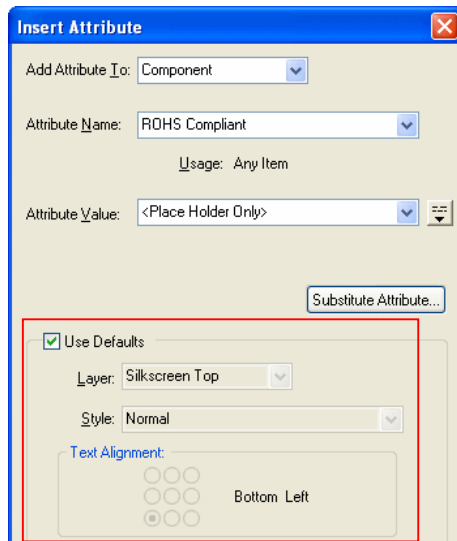


To remove specific values for an Attribute, select the name in the dropdown list, then uncheck the **Override** box.

You can remove all Attribute specific values by pressing the **Delete All Attribute Specific Values** button. If this button is greyed out, you do not have any such values defined.

Insert Attribute and Insert Attribute Position dialogs

The **Insert Attribute** and **Insert Attribute Position** dialogs now show all the values you will get and you can choose to use the defaults, or change them to the required value.



Attribute Validation

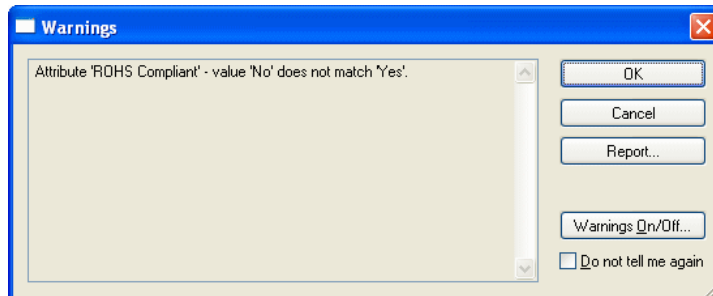
You can define a wildcard validation string along with an **Attribute** name on the **Technology** dialog. This is used to allow you to define specific values or contents for attributes up-front, and then for Pulsonix to check and verify them.

For example, you could define the **ROHS Compliant Attribute** with a **validation** string of **Yes**. If this is then changed to **No** or another string, it will be flagged by the **Design Rules Check** option.

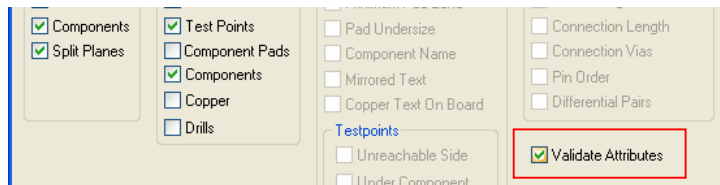
	Name	Usage	Context	Show Name	Show Value	Use as ToolTip	Use as Hyperlink	Back Annotate	Validation
<input checked="" type="checkbox"/>	<Component Height>	Any Item	PCB Design Only	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
<input checked="" type="checkbox"/>	<Hyperlink>	Any Item	All Designs	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	
<input type="checkbox"/>	<Spice Device>	Any Item	SCM Design Only	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
<input type="checkbox"/>	<Spice Node Template>	Pad	SCM Design Only	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
<input type="checkbox"/>	<Spice Parameters>	Any Item	SCM Design Only	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
<input type="checkbox"/>	<Spice Pin Order>	Pad	SCM Design Only	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
<input type="checkbox"/>	<Spice Tolerance>	Any Item	SCM Design Only	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
<input type="checkbox"/>	<Spice Value>	Any Item	SCM Design Only	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
<input checked="" type="checkbox"/>	Category	Any Item	All Designs	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
<input checked="" type="checkbox"/>	Rating	Any Item	All Designs	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
<input checked="" type="checkbox"/>	ROHS Compliant	Any Item	All Designs	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	Yes
<input type="checkbox"/>	Vendor	Any Item	All Designs	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	

Checking Validation strings

Attribute Validation is checked by **Design Rules Check**, the **Attribute Editor** and **Attribute Properties**. If you attempt to change the value to one not matching the Validation characters, you are presented with a warning dialog:



The **Design Rules Check** option has a new check box for checking **validation attributes**.



Extension of Wildcard format

The wildcard format has been extended for this and any other use of wildcards. It now includes lists of alternative substrings (or cases) and a number of special formats - %i – integer string, %f – floating point number, %a – alphabetic string, %s – a string of alphabetic or integer characters – see below for full list. *Other examples and full descriptions are available in the Online Help.*

For example, you could add a validation string of `%{ %iK| %iK%i% }` If the company standard only allows a value of 4K7 or 4K but not 4.7K, then this validation string will allow both valid values to be entered and check.

Wildcard characters supported:

* - means match any number of characters (including none).

? - means match exactly one character.

| - means match either string (the string before or the string after).

%d - means a single digit (e.g. 3).

%i - means an integer of any length (e.g. 12345).

%f - means a floating point number of any length (e.g. 123.456).

%c - means a single alphabetic character of either case (e.g. b), %C will only match an upper case character.

%a - means an alphabetic string of any length or case (e.g. abcdef), %A will only match upper case.

%s - means a string of a mixture of alphabetic and integer characters of any length or case (e.g. 1a2b3c), %S will only match upper case.

%u - means the name of a length unit - in long or short form (e.g. mm).

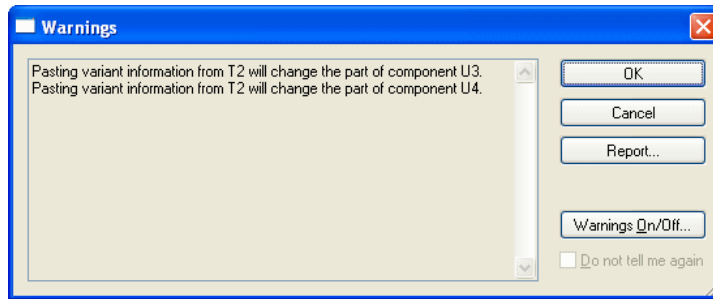
`%{string1|string2|string3%}` - means match any of these strings as part of a longer wildcard string. You can add a blank alternative at the start of the list - `%{|string2%}` which matches *blank* or *string2*.

Report Maker output of attribute Validation

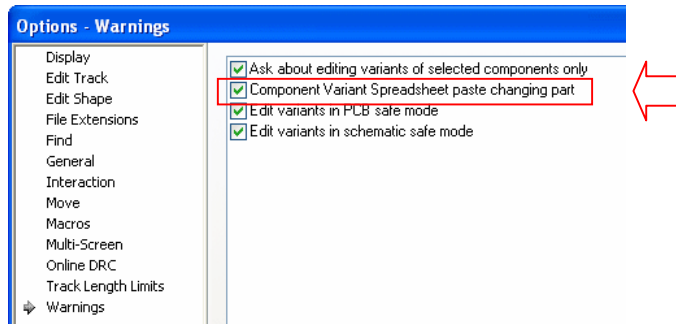
The **Report Maker** now has two new fields for an attribute, **Is Valid** and **Validation**. Use **Is Valid** to check if an attribute value is valid, and use **Validation** to report the validation string for an attribute. The Validation string is checked at run time for the report.

Changes to Component Variant Spreadsheet

Using **Paste** to copy variant information from one component to another, it will now give you a warning to show you which components will have their Parts changed, this allows you to cancel the operation. A report can also be produced using the **Report** button if you require an audit trail of changes made.

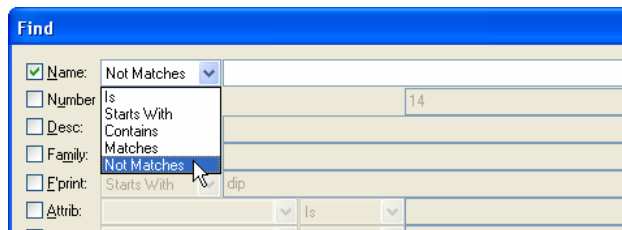


This warning can be switched off using the **Component Variant Spreadsheet** warning on the **Warnings** option tab of the **Options** dialog on the **Setup** menu.



Changes to search criteria in Find dialog

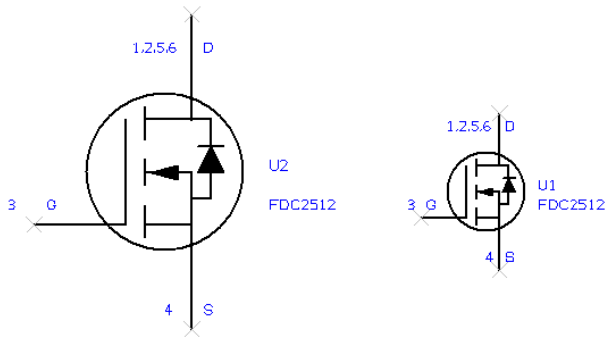
The Library searcher (**Find** option on **Insert Component** option) now has a **Not Matches** criteria option. Where **Not Matches** are specified, this takes any text that does not match, using wildcards if required



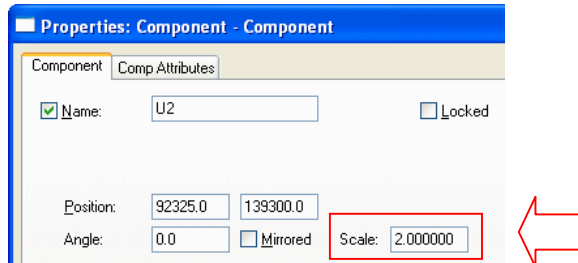
Scale Symbols

You can now scale **Documentation Symbols** and **Schematic Gate Symbols**, as well as selections of **Shapes** and **Text**, using the new **Scale** command. **Documentation Symbols** in both Schematic and PCB editors can be scaled but PCB footprints cannot be scaled.

So for example, you can scale a symbol and have two scales of the same symbol in the design.



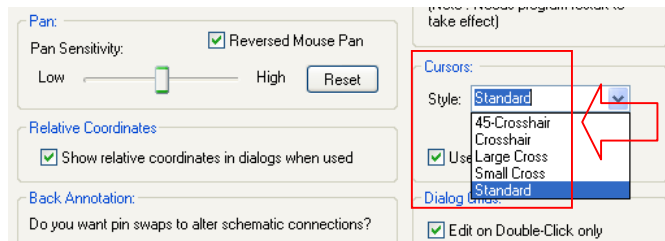
Styles are not scaled, so text remains the same height and lines remain the same width. Symbols retain their scale factor relative to the original symbol. The symbol scale can be seen or changed through the **Properties** dialog.



The Scale command will scale the selection including symbols, by the given scale factor.

Additional Cursor Type – 45-Crosshair

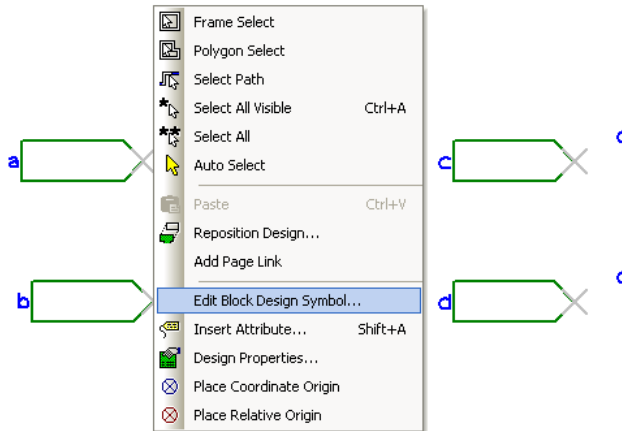
You can now use a new cursor type of **45-Crosshair**. This is like the full screen Crosshair but at 45 degrees. This is available on the **Options** dialog under **General**.



Save Generated Block Symbol with Block Design (SCM)

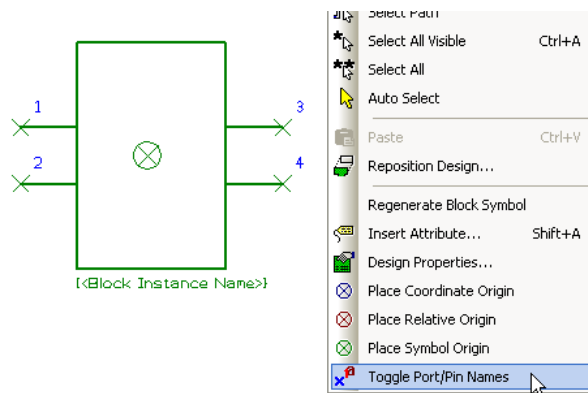
A **block symbol** can now be saved with the block design. This symbol will be used when the block is added to a design. Alternatively, the library of block symbols can still be used if required.

You can edit the Symbol while editing the block by selecting **Edit Block Design Symbol** from the context menu with nothing selected.



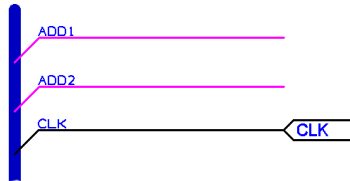
View Port Names in Block Symbol (SCM)

A new **Toggle Port/Pin Names** command allows you to see the port names whilst editing a **Block Symbol** from a design.



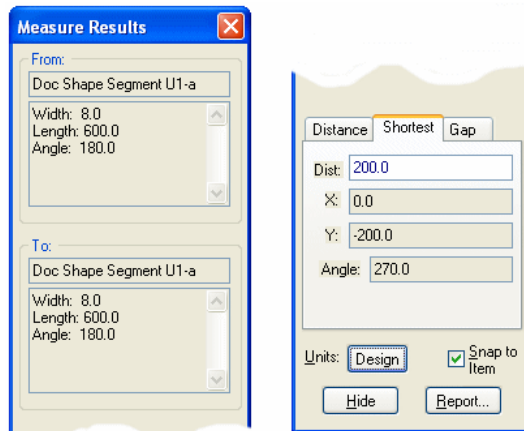
Unfinished Schematic Connections (SCM)

Schematic **Connections** are now highlighted as **unfinished** if they have one end on a bus and the other end dangling (even if they have a net name displayed). This means they are more easily identified as requiring action for completing the design.



Measure Tool in Schematics (SCM)

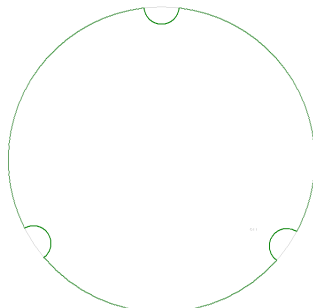
The **Measure Tool** is now available when editing **Schematic Designs**, **Schematic Symbols** and **Schematic Doc Symbols**.



Changes to Construction Lines option (PCB)

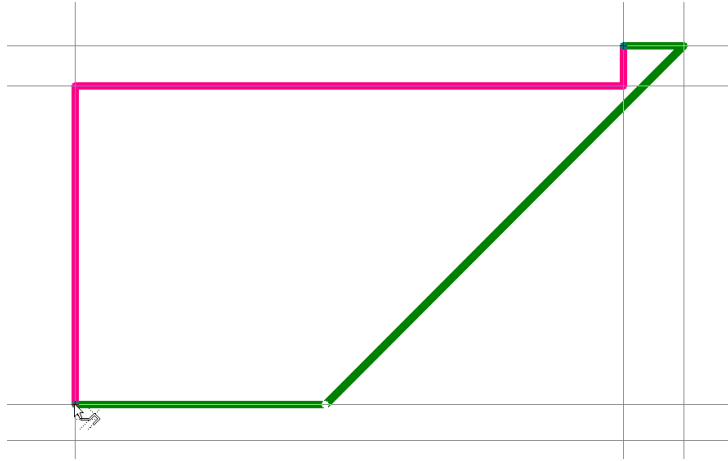
'Along A Segment' on circles and pad edges

When adding construction lines, you can now use **Along A Segment** on circles and on the edges of pads (including user defined pads). Adding a segment on a circle will draw you a Circle construction Line.



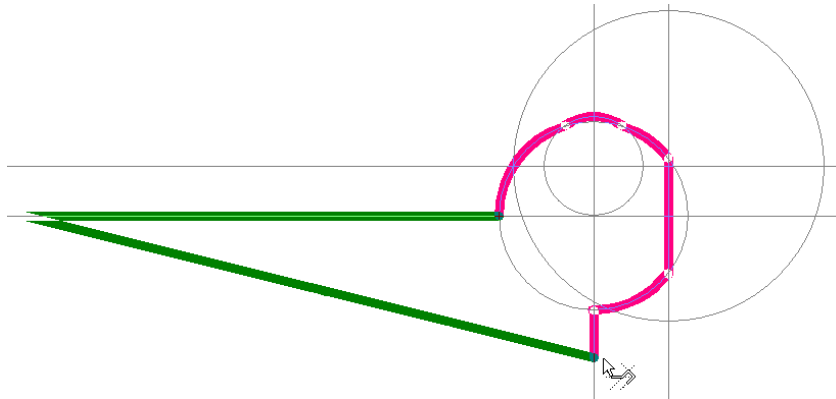
Remove last segment when following construction lines

When adding a track or shape and following construction lines, you can now use the **Backspace** key to remove the last segment added. This moves the cursor back to the end of the previous segment.



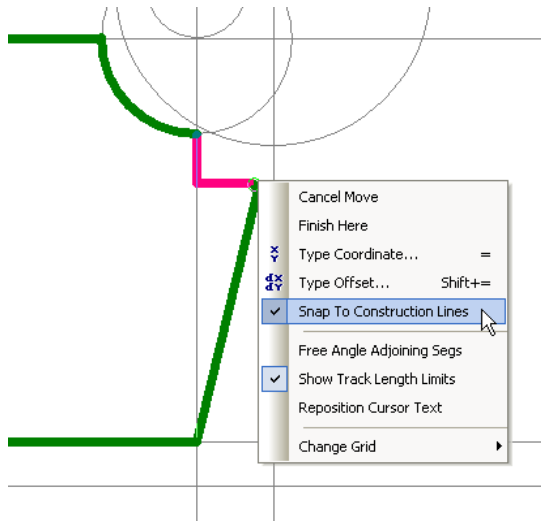
Disengaging the Follow Construction Lines feature

When adding a track or shape and following construction lines, you can uncheck the new **Auto Switch Construction Lines** option from the context menu to stop the shape from automatically switching to follow another construction line or circle. In this mode clicking on another construction line that intersects the one being followed will switch to the picked line. This is useful when you have a “nest” of lines and circles and want to follow just one path through it.



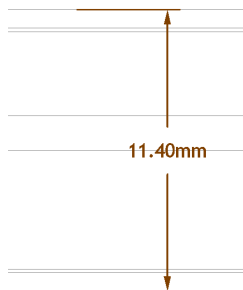
Snap to construction lines

When moving a shape corner or track corner, you can now snap it to a construction line. This option is available on the context menu.



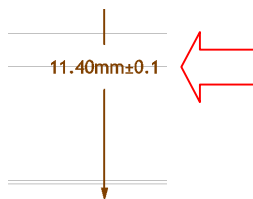
Dimensions attached to construction lines

Dimensions can now be attached to Construction Lines.

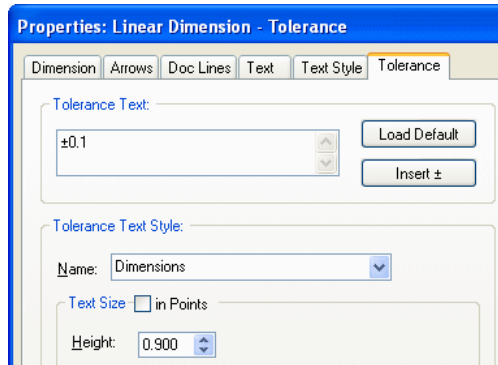


Tolerances in Dimensions (PCB)

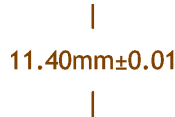
When adding **Dimensions** you can now automatically add **tolerance text** to the end of a selected dimension.



From within the **Properties** dialog of a **Dimension**, there is a new tab to define the **Tolerance**.



The tolerance text is a multi-line text item with its own text style, automatically attached to the end of the dimension text such that both text items are aligned through their vertical centres. The multi-line tolerance has been provided to create either unilateral or bilateral tolerances, or to be used for any other purpose you like.



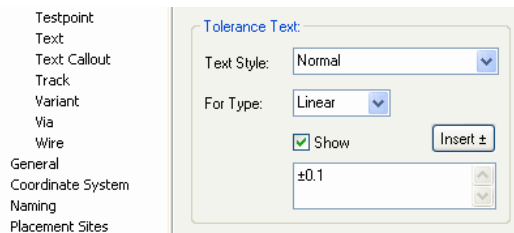
A Unilateral Dimension example



A Bilateral Dimension example

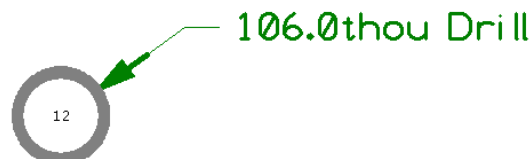
As well as being able to type the tolerance text, you can use the **Insert ±** button to insert the bilateral ± character at the start of the text, and use the **Load Default** button to set the text to the default for that dimension type.

Default **Tolerances** can be set up in the **Design Settings** dialog, **Defaults** and **Dimensions**. These can be defined for each of the three types of dimension, linear, radial and angular. A default tolerance text style can also be set up.



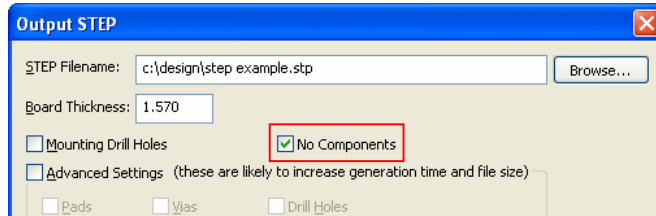
Changes to Radial Dimensions

You can now add a **Radial Dimension** to show a pads drill hole dimension. Use the **Next Snap Point** command from the context menu when adding the Dimension to change from a round pad edge to its drill hole edge.



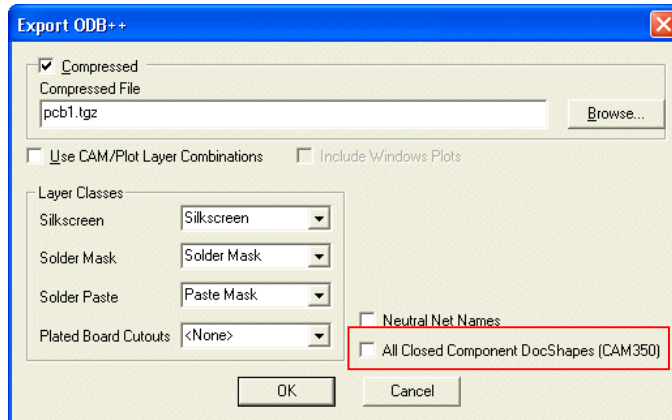
STEP Export – No Components (PCB)

There is a new check box which disables the outputting of components to output the STEP file. Select the **No Components** check box if you only want the board shape.



ODB++ Output - CAM350 Switch (PCB)

In general, only one closed documentation shape is output to the comp_+_top and comp_+_bot plots. CAM350 allows more than one shape for the component shape and some of its processes require this. Check the **All Closed Component DocShapes (CAM350)** option if you require this.



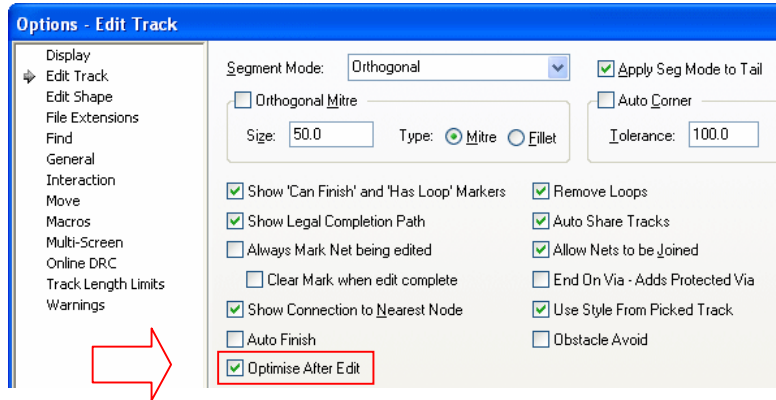
It should be noted that more than one viewer will reject the ODB++ design if there are more than one shape specified for each footprint definition. This should **ONLY** be checked if you are sure that your process can handle this.

Change to Optimise All Nets (PCB)

During normal use, Pulsonix now keeps track of which nets have been changed. This change means that when you use **Optimise All Nets** it will only optimise those that need it. For large nets, this means a significant speed up. There is no interface change for this.

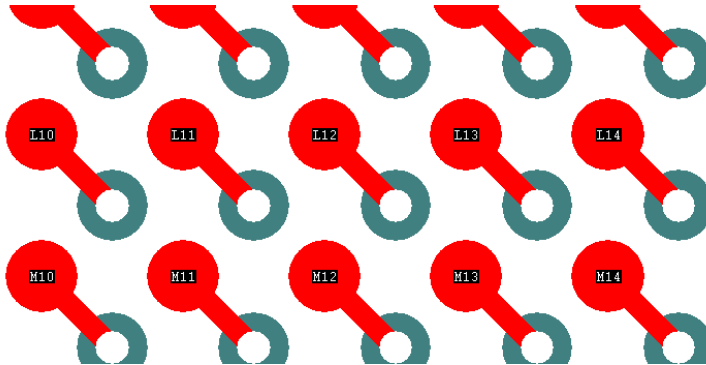
Optimise After Edit (PCB)

There is a new option (**Optimise After Edit**) on the **Options** dialog under **Edit Track** to force an optimise of the net being edited after the track edit has been completed. This affects **Insert Track**, **Edit Track**, **Sketch Track**, **Complete as Track** and the **Delete Corner** options.

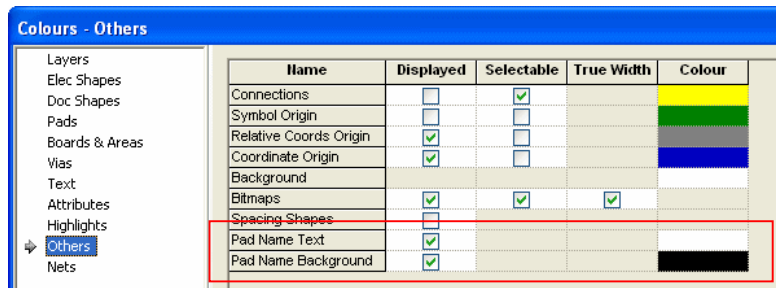


Display Pad Numbers in Pads (PCB)

The ability to display pad numbers within the pads has been added to version 7.6.



The display of pad names is controlled from the **Colours** dialog, **Highlights**



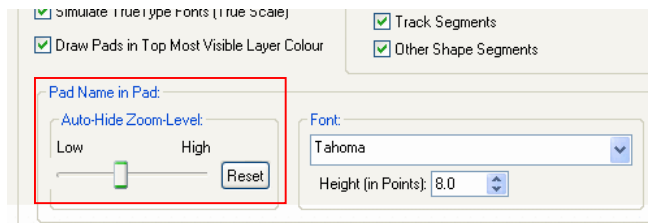
From here, you can switch the **Pad Name Text** and **Pad Name Background** on and off and change the colour. Enabling the foreground but not the background will display the pad names with no highlighted background.

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

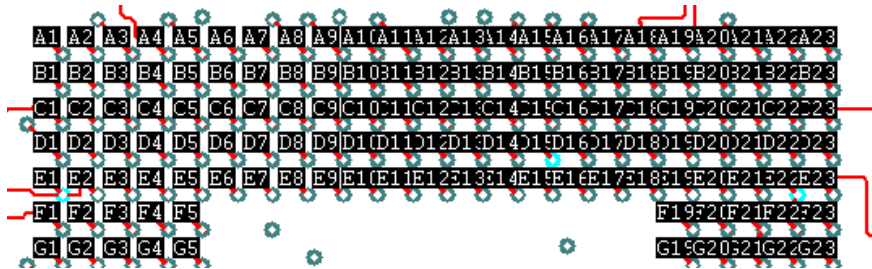
Options dialog

Auto-Hide Zoom Level

From the **Options** dialog and **Display**, you have a slider control for how far you can zoom in and out and the point that the text disappears. This allows you to zoom in close and for the text not to swamp the design.



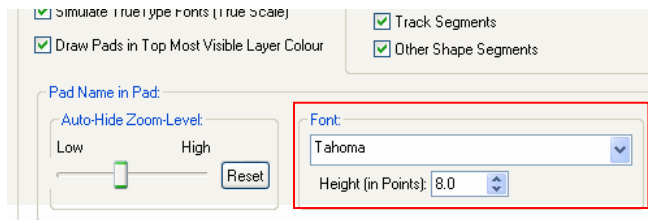
Setting the slider to the Low position avoids situations like this below where the pads can't be seen under the pad names:



Use the **Reset** button to reset the slider to its default position.

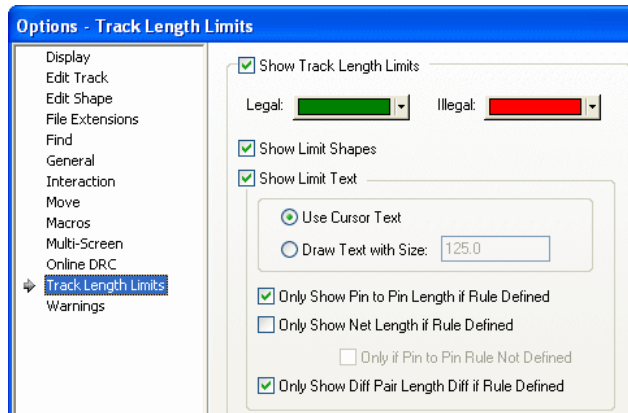
Font Size for Pad Names in Pad

The font style and size used to display the **Pad Names in Pads** can be adjusted from the **Options** dialog also 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 itself can be changed if you prefer a different style of font.



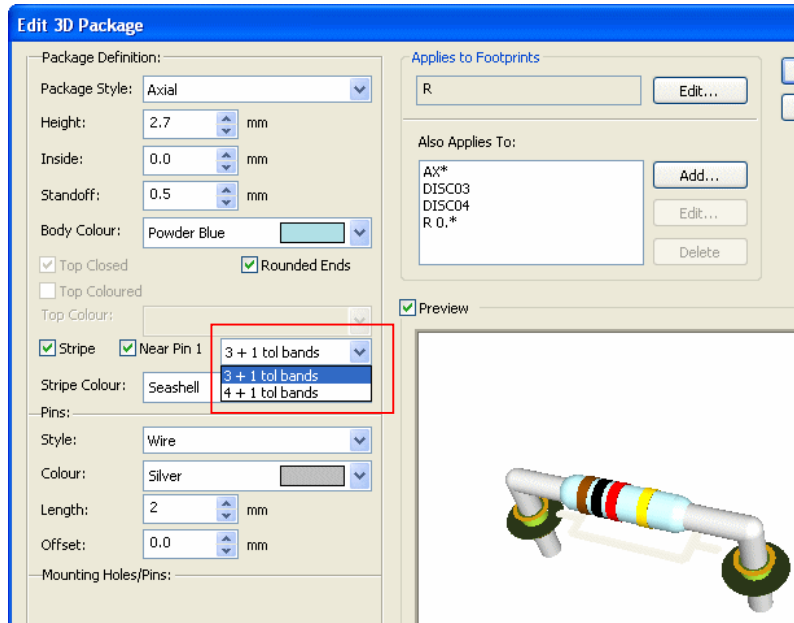
Options - New Track Length Limits page (PCB)

The **Track Length Limits** selection has been moved from the **Edit Track** page onto its own options page. This is mainly used for the Interactive High Speed option.



3+1 or 4+1 Resistor Bands for 3D Packages (PCB)

You can now have 3+1 or 4+1 resistor bands when creating 3D packages.

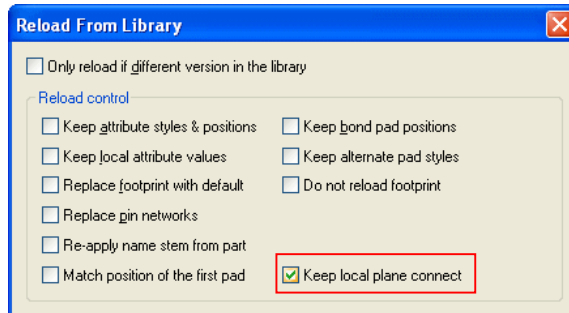


With the **Stripe**: box checked, Axial and Can components can be given a stripe on the component body. CAN shapes will get a stripe up the side of the body near to pin 1 or pin 2 depending on the pin 1 checkbox. Axial components get a single stripe near one end or the other, except for Resistors and Inductors which will get the correct set of coloured stripes depending on the value of each component.

Using the drop down list, you can choose to use 3 or 4 bands for the value (plus the tolerance band).

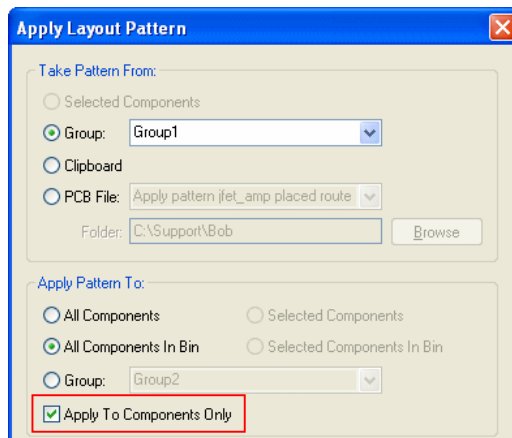
Reload Option - Keep local plane connect switch (PCB)

On the **Reload Part** dialogs, you now have option (**Keep local plane connect**) to keep the local plane connect value defined on the pad in the design. (Previously this was tied in with the **Keep attribute values** option).



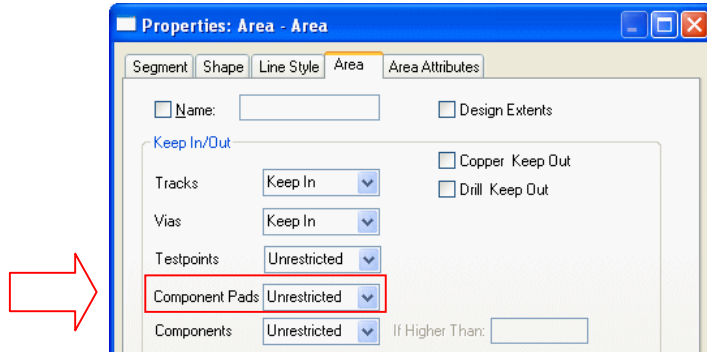
Changes to Apply Layout Pattern (PCB)

The **Apply Layout Pattern** feature now has an option to **Apply To Components Only**. When selected, this does not copy any tracks or additional design information with it, only Components.



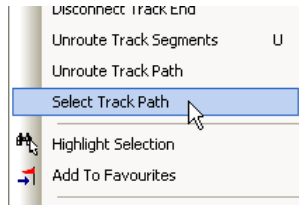
Areas - Component Pad Keep In/Out (PCB)

Within an Area properties, you can now make an area a **Component Pad Keep In/Out** area. This option does not apply to **Free Pads**, **Die Pads** or **Mounting Holes**.

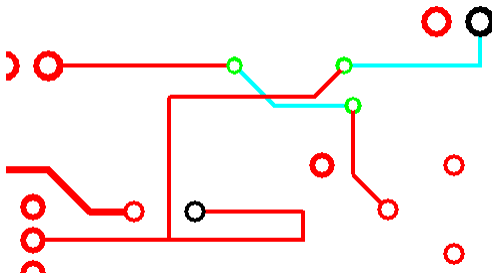


Select Track Path option (PCB)

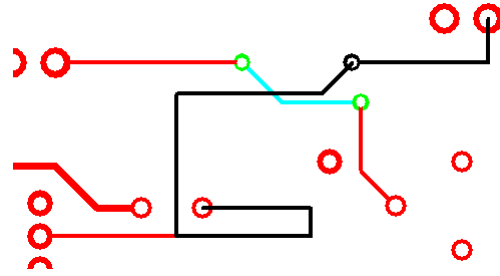
There is a new option available for PCB designs to select the tracks and vias in a track path. It is also available in the PCB Footprint editor. The **Select Track Path** option is available on the context menu when the following is selected:



- When a **track segment** or **via** are selected, it selects all tracks and vias on the whole track path.
- When **multiple track segments** are selected, it selects all tracks and vias on all selected track paths.
- With **2 pads selected on the same net**, this selects the tracks and vias in the minimum length path between the pads (if there is one). This path may pass through other pads, but they will not be selected.

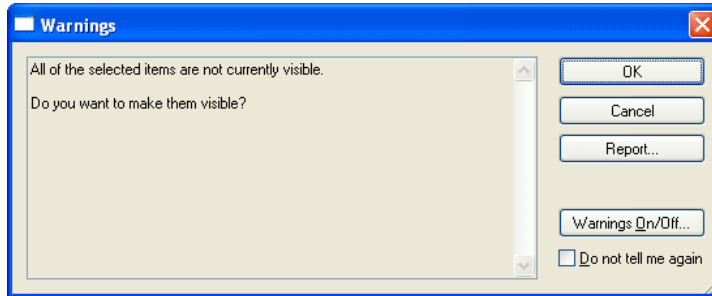


Two pads on the same net in the design are selected (using Ctrl-Select).

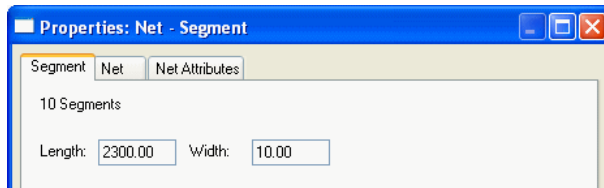


The **Select Track Path** option is selected from the context menu. The shortest track path is highlighted.

If some tracks or vias in the path are not visible, a warning dialog will be displayed that allows the user to make them all visible.



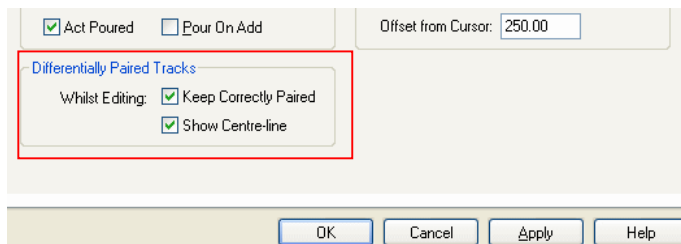
Properties has been changed to show the selected track segments length if you have tracks and other items selected in a single net. By using Select Track between 2 selected pads, in Properties you can see the minimum track length between the pads.



Interactive High Speed (PCB) (Cost Option)

Differential Pairs

Two new check boxes have been added to the **Options** dialog and **Interaction** for **Differentially Paired Tracks**:

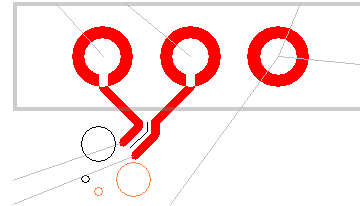


Keep Correctly Paired

When checked, the **Keep correctly Paired** option will not allow doubling-back and acute corners when adding or editing paired tracks. This avoids the tracks ending up curved or further apart than the required gap. Uncheck it to allow full movement (as in previous releases). It may be that allowing illegal pairing during an edit is the only way to get to the actual position required.



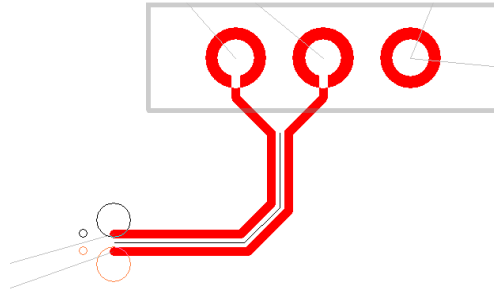
Without **Keep correctly Paired** checked. Tracks are allowed to be edited in illegally.



With **Keep correctly Paired** checked. Tracks are restrained to not allow an illegal path.

Show Centre-line

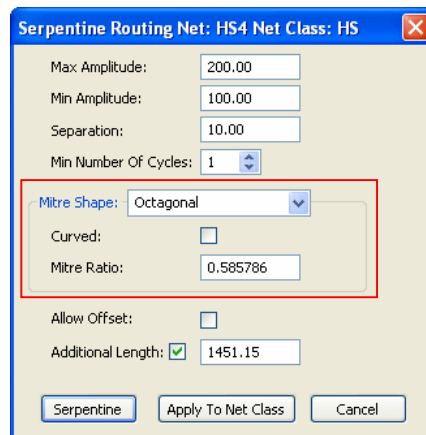
When checked, the **Show Centre-line** option is used to show the line between the paired tracks that you are actually editing. The centre line is the grid position that the differential paired tracks are following. This visual indication helps ensure the tracks are in the correct position and shows the selected dynamic segments.



The line is removed as soon as the edit, move or move corner is complete on the differential pairs.

Serpentine Routing

You can now add serpentine with mitred or curved corners. These can range from 180 degree curves to 90 degree corners.

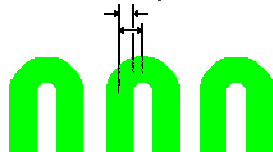


You can control the shape of the top of each loop. The mitre around each turn can be curved or straight, and the size of the mitre is defined by the **Mitre Ratio**, which is the proportion of the 90

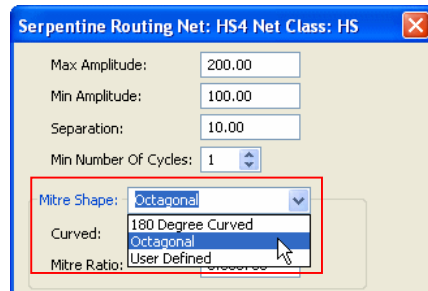
degree corner taken up by the mitre. A value of 1.0 gives a complete 180 degree curve (or *sawtooth* if straight) around the top of the loop; a value of 0.0 gives a squared off top to the loop; values in between give a loop of two curves or 45 degree angled lines with a flat top between. The default is **Curved** with a **Mitre Ratio** of 1.0, which results in 180 degree curve around the loop.

A **Mitre Ratio** of 0.585786 and straight mitre gives an octagonal shape around the loop (angled mitres and flat top with the same length).

Mitre Size defined as proportion
of half loop width



The **Mitre Shape** drop down list allows you to set the parameters for the most common shapes.



180 Degree Curved – this would be a standard serpentine with curved corners. This presets the **Mitre Ratio** to 1.0 and produces 180 degree curved corners. If the **Curved** button is unchecked, it will produce a Sawtooth serpentine around the top of the loop.

Octagonal – this produces a serpentine with the 45 degree mitre and flat ‘top’ lengths the same value.

User Defined – if the value is set to a value not matching the above cases.

Eagle V6.3 Import

The import of Eagle V6.3 designs and libraries is now supported. This feature was added to a later V7.5 issue with the addition of new ULP conversion files and new DLLs. These are now all included in the main V7.6 release.

Allegro PCB Design Import (Cost Option)

As part of the Advanced Import cost option, you can now import Cadence Allegro PCB designs. 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.

Library Changes

New Libraries

New libraries have been introduced for the **Coilcraft LPS4018 series**. This is new Parts and associated footprint.

Changes to Existing Libraries

Changes have been made to the **Coilcraft 1008CS** series Parts library to make values and Attribute names more consistent with other libraries.

New Footprints

New footprints have been added for the **0201** and **01005** surface mount footprints.

New footprints have been added for the **148-QFN** and **169-FBGA** footprints.