



Pulsonix Design System

V6.1 Update Notes

Vertrieb für Deutschland, Österreich und Schweiz:

PESCHGES VARIOMETER GmbH

D-52078 Aachen, Zieglerstr. 11

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

www.peschges-eda.de

E-mail: vertrieb@peschges-eda.de

Contents

CHAPTER 1. VERSION 6.1 UPDATE SUPPLEMENT	5
Installing the New Version of Pulsonix	5
Licensing.....	5
Windows 7 Support	5
Part Browser.....	5
Technology File Changes.....	6
Net Names Page.....	6
Spacing Rules Dialog.....	7
Define Pad Style Exceptions for Top/Bottom Side	8
CAM Plot - PDF Output.....	8
Defining Paper Size in PDF Plot.....	8
Print across Multiple Pages in PDF Plot.....	8
Auto Weld in PCB	9
Reset Button on Bitmap Properties	9
Shift-Select to select an Attribute Owner	9
Drill id in Properties.....	10
Drill Size Description.....	10
Drill letters allocated to each slot.....	11
In-Layer Stack Preview Option	11
Moving Connectors and Doc Symbols on Component Grid.....	11
Joining Busses using Insert Bus in SCM.....	11
New Report Maker Commands	11
Choosing Gates in Insert Schematic Component / Connector.....	13
Next Symbol in Insert Schematic Connection	14
Start Schematic Connection on Testpoint Part.....	14
Change Shape Type on multiple-selected items.....	15
Changes to Defaults option.....	15
Applying Predefined Signals	17
Spokes only on enclosed pads	17
Move Track Segment Improvement.....	18
Auto Rename By Page in Schematics	18
Find Bar in Schematics	19
Definable Drawing Order for Doc Shapes.....	19
Check Drill Holes against boards in DRC.....	19
Coincident Drill Check	20
Switching off Ctrl-Drag To Duplicate.....	20
Display of Diameter for selected Arcs/Circles.....	20
Frame Select for Error Markers	21
Tooltip for Track Length Limits in High Speed Option	21
Support for P-CAD Master Designer Schematics Import	22
Library Changes in V6.1	23
Options previously released in V6.0 but not documented.....	24
Advanced Parts Searching.....	24
Reload Part using Pad Origin.....	24
Support for PADS 2007 format on Import.....	25
Support for Eagle V5.x format on Import.....	25

Chapter 1. Version 6.1 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 6.1 on top of your existing installation or along side if you prefer; however, you do not need to uninstall the old version first.

Licensing

If you are using a version of Pulsonix earlier than Version 6.0, you will require a new license for Version 6.0. This will be supplied to you by email under the contract conditions of your current maintenance agreement. If you are using V6.0, you will not require a new license.

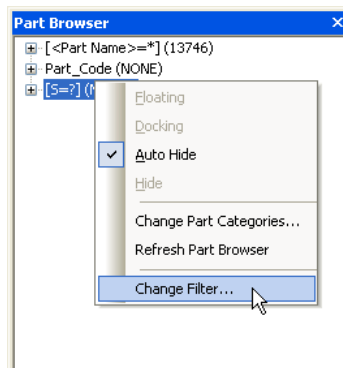
For existing users upgrading from a previous (non-V6.0) version, it is recommended that you save the new license and overwrite the existing one. When requested during installation, simply click the **No Change In Licensing** check box on the licensing page of the installation wizard. The **License Manager** can be used to add new licenses and make changes to network licensing after the installation has been completed.

Windows 7 Support

Pulsonix 6.1 and later will be fully supported under this operating system. No earlier Pulsonix versions can or will be supported.

Part Browser

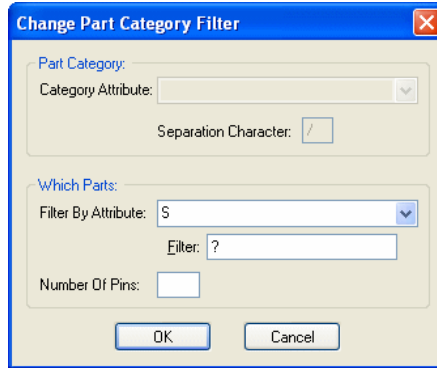
On the context menu of the **Parts Browser** there is now a new option **Change Filter**.



6 Pulsonix Version 6.1 Update Notes

Use this to change the filter used on the currently selected category in the browser tree. Only the current category will be refreshed to apply the filter to its parts. This is quicker than changing the filter using **Change Part Categories** which will rebuild all categories.

When selected, it opens the **Change Part Category Filter** dialog.



Technology File Changes

Net Names Page

Within the Technology file, on the **Net Names** page, you can multiple-select rows and apply the net class in the current cell to all selected rows. This also works for the **Guard Spaces**.

You can drag down the list if the net names required are adjacent, or you can use Ctrl select or Shift select to bring net names in and out of the selection.

Once selected, right click on the required **Net Class name** and select **Apply to selected cells** to apply it to your selection.

	Name	Net Class	Guard Space
Y	CLOCK	Signal	0.00000
Y	GND	Ground	0.00000
Y	HS1	HS	0.00000
Y	HS2	Signal	0.00000
Y	PPGND	Ground	0.00000
Y	RESET	Signal	0.00000
Y	\$4	Signal	0.00000
Y	\$7	Signal	0.00000
Y	\$12	Signal	0.00000
Y	\$13	Signal	0.00000
Y	\$14	Signal	0.00000

A context menu is open over the 'HS' cell in the 'Net Class' column of the 'HS1' row. The menu options are: 'Apply to entire Column', 'Apply to empty cells', and 'Apply to selected cells'. The 'Apply to selected cells' option is highlighted.

The resultant selection will now display the new Net Class Name.

	Name	Net Class	Guard Space
Y	CLOCK	Signal	0.00000
Y	GND	Ground	0.00000
Y	HS1	HS	0.00000
Y	HS2	HS	0.00000
Y	PPGND	Ground	0.00000
Y	RESET	HS	0.00000
Y	\$4	Signal	0.00000

Spacing Rules Dialog

There is now a special **Check Spacing** rule level check box on the **Technology, Spacing Rules** dialog. This resolves all the spacing rules in the other three levels (**Design, Net Class** and **Match Net Class Pair**), to show you the actual spacing between two net classes.

In the example below, the **Check Spacing** has been selected between **Net Classes** of **Power** and **HS**. In the rules grid, it has highlighted Track to Track and Track to Pad as rules that will have different values. This allows you to check ‘what-if’ situations between different rules.

Net Class - Power	Net Class - HS						
	Track	Pad	Via	Testpoint	Mounting Hole	Copper	Text
Track	0.50000	0.75000	0.25000	0.25000	0.25000	0.25000	0.25000
Pad	0.25000	0.25000	0.25000	0.25000	0.25000	0.25000	0.25000

Rule Level:
 Design
 Net Class
 Match Net Class Pair
 Check Spacing

Net Class: Power
 Net Class: HS
 On Layers: All
 Within Area:

This is resolved from the following pages for **HS Net Class**, where the Track to Pad spacing rule is different:

Design	Net Class - HS						
	Track	Pad	Via	Testpoint	Mounting Hole	Copper	Text
Track	0.25000	0.75000	0.25000	0.25000	0.25000	0.25000	0.25000
Pad	0.25000	0.25000	0.25000	0.25000	0.25000	0.25000	0.25000
Via	0.25000	0.25000	0.25000	0.25000	0.25000	0.25000	0.25000

Rule Level:
 Design
 Net Class
 Match Net Class Pair
 Check Spacing

Minimum Spacing: 0.00000
 Net Class: HS
 Check between items on same net

And **Power Net Class**, where the Track to Track spacing rule is different:

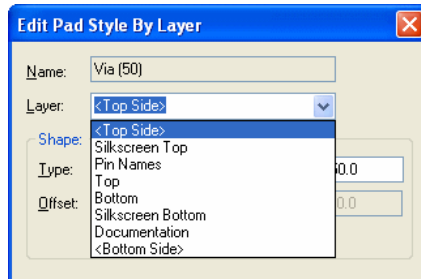
Design	Net Class - Power						
	Track	Pad	Via	Testpoint	Mounting Hole	Copper	Text
Track	0.50000	0.25000	0.25000	0.25000	0.25000	0.25000	0.25000
Pad	0.25000	0.25000	0.25000	0.25000	0.25000	0.25000	0.25000

Rule Level:
 Design
 Net Class
 Match Net Class Pair
 Check Spacing

Minimum Spacing: 0.00000
 Net Class: Power
 Check between items on same net

Define Pad Style Exceptions for Top/Bottom Side

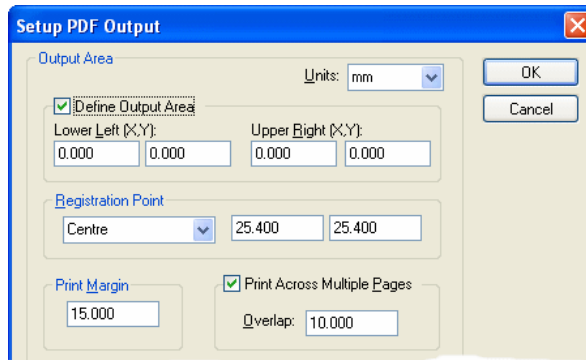
You can now define a pad style exception for the <Top Side> or <Bottom Side> (as well as the current per layer settings). This allows you to easily define a different pad shape for the top, inner & bottom sides. These are treated as the default shape for layers on that side but you can still create a specific exception for any single layer. This means instead of defining the same layer exceptions for Top Solder, Top Paste and Top Silkscreen for example, providing they are the same size, you can define a size for <Top Side> (or <Bottom Side>).



CAM Plot - PDF Output

Defining Paper Size in PDF Plot

In the **CAM Plot** dialog, you can now define a paper size if required. Select the **Define Output Area** check box to enable this. Type in values required for **Lower Left (X,Y)** and **Upper Right (X,Y)** to define your page size. So for example, an **A4** paper size, you might type **0,0** and **210, 297**, the **Units** would need to be set to **mm** for these values. This would produce a print on A4 paper produced in **Portrait** mode. For **Landscape** mode, type **297,210** for the **Upper Right (X, Y)** value.



Print across Multiple Pages in PDF Plot

You can also specify that you want to **Print Across Multiple Pages**. Select the check box to enable this option. This is only available once the **Define Output Area** check box has been selected. The **Overlap** defines how much overlap is reproduced on each page. This may be useful if you wish to paste printed pages together.

Auto Weld in PCB

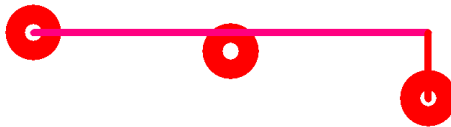
Auto Weld to vias and mounting holes

Selected tracks will now weld to Vias and Mounting Holes, previously, this was only available for Pads.

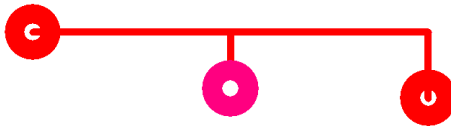
Auto Weld for tracks

If a **Track** crosses a **Pad** or **Via**, but is not directly over its centre, a short track spur will be added from the track to the pad or via.

In the example below, the track is selected and **Auto Weld Selection** selected from the context menu:

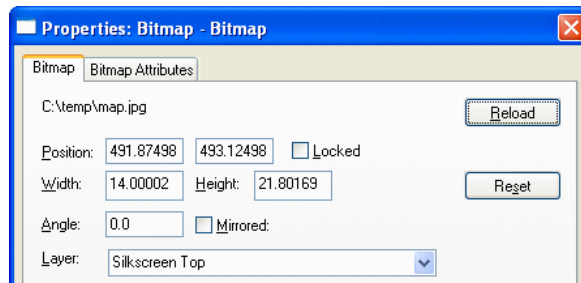


The track is welded to the pad in the centre. If the pad is then moved away, the new track spur can be seen.



Reset Button on Bitmap Properties

For a selected **Bitmap** in the design, from the **Properties** dialog, you can now select the **Reset** button to reset the bitmap back to its original size if this has been modified.



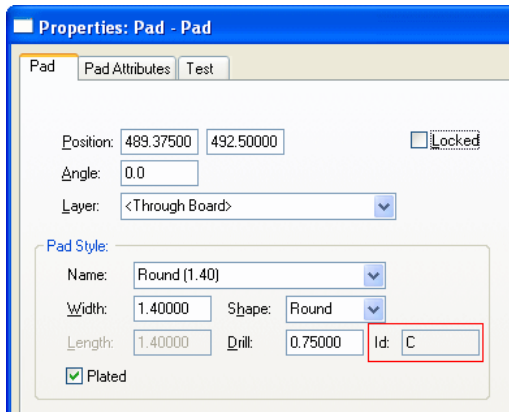
For multiple-selected bitmaps, regardless of whether they are the same or different, each bitmap will now be reloaded if the **Reload** button is selected.

Shift-Select to select an Attribute Owner

If you do a **Shift-Select** on an attribute, it will now select the attribute owner. Previously, this did not happen.

Drill id in Properties

You can now see the drill **Id** in **Properties** (for drilled pads, mounting holes & vias).

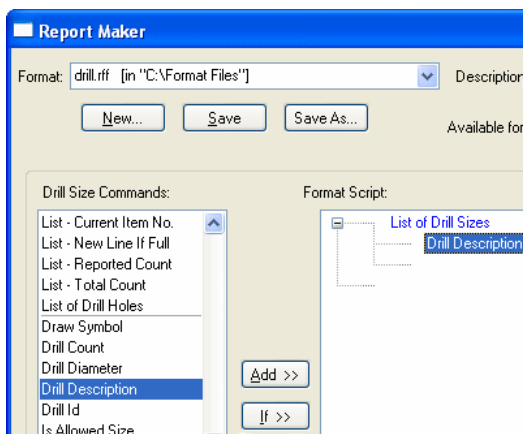


Drill Size Description

Each drill size can now have a **Description** string. These can be used to show drill tolerances or slot dimensions for example.

Diameter	Plated	Id	Symbol	Symbol Size	Symbol Filled	Count	Min Diam	Max Diam	Description
Special	Y	A	Round	1.0	<input checked="" type="checkbox"/>	2			Slotted pad 175x50
Special	Y	F	Round	10.0	<input checked="" type="checkbox"/>	1			Slotted pad 100x75
25.6	Y	B	Round	25.6	<input checked="" type="checkbox"/>	9			Tolerance +/- 0.01mm
29.5	Y	C	Asterisk	29.5	<input checked="" type="checkbox"/>	2			
31.5	Y	D	Round	31.5	<input checked="" type="checkbox"/>	8			
44.3	Y	E	Round	44.3	<input checked="" type="checkbox"/>	4			

The Description string can be output using **Report Maker** using the **Drill Description** command under **List of Drill Sizes**.



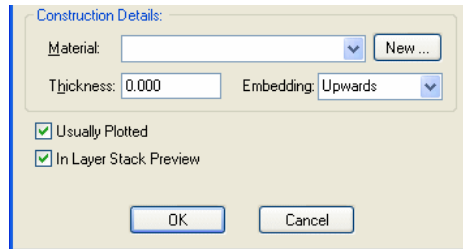
Drill letters allocated to each slot

A different drill letter is allocated to each different slot, a slot is different if it is on a different pad style. These are shown in the **Technology** under **Drill ID**. Previously, there was only one entry in a drill table for a slot, now there might be several.

Diameter	Plated	Id	Symbol	Symbol Size	Symbol Filled	Co
Special	Y	A	Round	1.0	<input checked="" type="checkbox"/>	
Special	Y	F	Round	10.0	<input checked="" type="checkbox"/>	
25.6	Y	B	Round	25.6	<input checked="" type="checkbox"/>	

In-Layer Stack Preview Option

When editing a **Layer** in the **Technology**, there is now a check box **In Layer Stack Preview**. This is used to specify whether each layer appears in the layer stack preview when viewed. You might use this to specify that none of your documentation layers appear in the Layer Stack Preview, for example, to make it less cluttered.



Un-checking the **Usually Plotted** box on the same dialog no longer forces the layer to not be in the layer stack, this and the **In Layer Stack Preview** are separate.

Moving Connectors and Doc Symbols on Component Grid

Connector Pins, **Signal References**, **Star Points**, **Block Ports** and any **documentation symbol** that has at least one pin will now move on the **Component** grid. Previously, they moved on the **Working** grid.

Joining Busses using Insert Bus in SCM

When using **Insert Bus** and start or end on the end of an existing compatible bus, it will now join the busses to make one continuous one. Previously, it added a bus terminal between them.

New Report Maker Commands

New Commands

New commands have been added to expand the **Report Maker** capabilities:

Is In Component Bin - Used on components to report if the component is in the bin.

Minimum Pad Spacing – Used on PCB designs, reports minimum pad to *anything* spacing in spacing rules.

Minimum Track Spacing – Used on PCB designs, reports minimum track to *anything* spacing in spacing rules.

12 Pulsonix Version 6.1 Update Notes

Minimum Used Track Width – Used on PCB designs, reports minimum width on any track in the design.

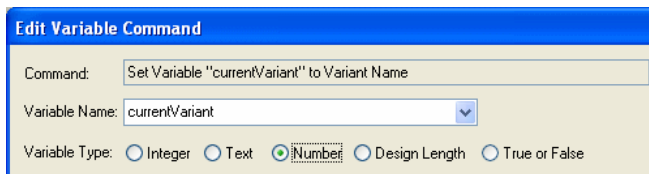
Is Slot – Used on a drill Size or on a drill. Reports true if the entry has a slot rather than a drill hole.

Slot Pad Style – Used on a drill size to report pad style name for a slot.

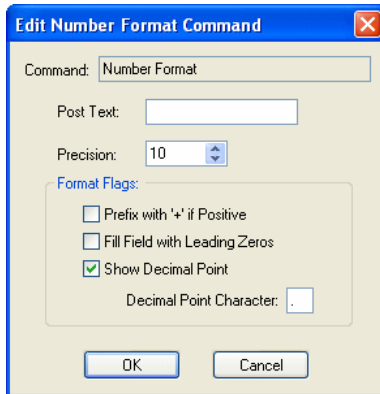
Drill Id – Not a new command but can now be reported as one of the sub-fields of a pad style.

Number Format command

There is a new general command to specify the precision and decimal point character to be used when outputting **variables** of type **Number**.

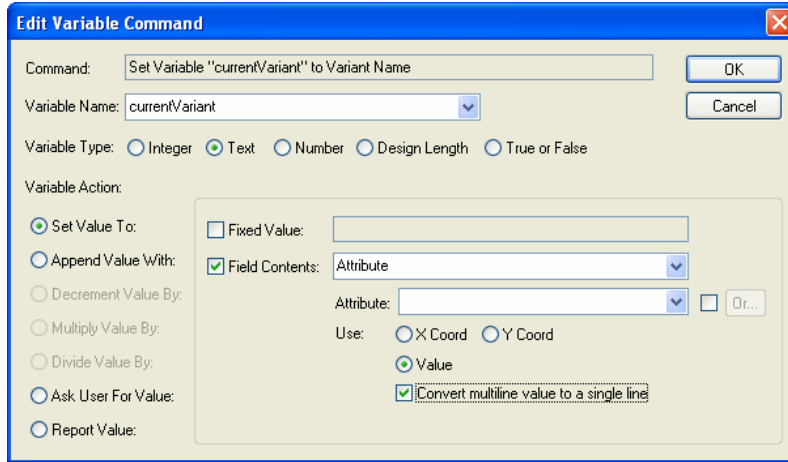


These can be placed anywhere in the script, and (like **Coordinate Units**) the last defined at that level will be used.



Attribute Command when using Variables

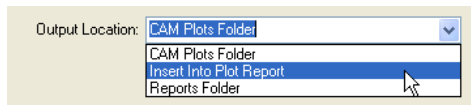
When editing an attribute command to report an attribute value, or editing a variable command that is setting a variable to an attribute value, you will see a new check box labelled **Convert multiline value to a single line**.



Check this if you want new line characters in the attribute value to be replaced with spaces before being output.

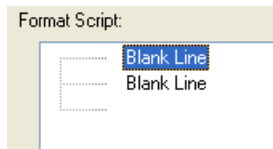
Running Report From Cam Plot option

If you include a user report as a **CAM Plot** and set the **Output Location** to **CAM Pots Folder** the report is written to this plots folder. Previously, any reports run in this option with the **Run Report** command were output to the **Reports Folder**. This means all reports will now be written to the same place as specified in the Output location entry.



Use of Ctrl+Drag in Report Maker dialog

If you use a select a command and then do a **Ctrl+Drag** on it, this will add a copy of the command. You can then move it to where it is required. This is a quicker method than finding it in the Commands list then adding it to the Format Scripts area.



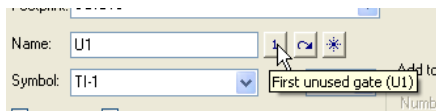
Choosing Gates in Insert Schematic Component / Connector

If you are adding a Component Part that has multiple gates or adding a connector pin, there are now three new buttons in the dialog to the right of the component name box.



14 Pulsonix Version 6.1 Update Notes

These are to help you easily choose between using free gates/pins on existing components, or adding new components. If you hover the cursor over the buttons, a tooltip will be shown describing their use and showing what component name will be used if they are pressed.



First Unused Gate - Use this button to set the dialog up to add the first unused gate on any component using this part in the design. The button will be disabled if there are no free gates in the design.



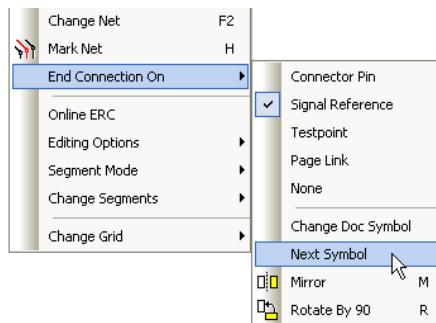
Next Component name - Use this button to set the dialog up to add the next component using this part with free gates in the design.



First Free Component Name - Use this button to always set the component name to the first free name that does not yet exist in the design.

Next Symbol in Insert Schematic Connection

Can now use the **Next Symbol** command when editing a schematic connection that **Ends On** a block port, signal reference or page link symbol. This will switch between different symbols displayed on the end of the connection. Although this command appears on the context menu, for general and more practical use it would usually be assigned to a shortcut key.



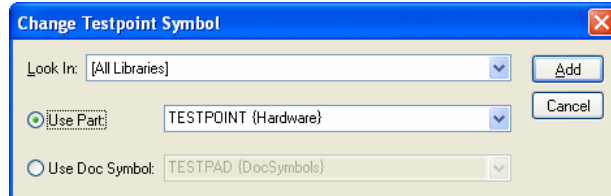
The different types of documentation symbol (signal reference, block port, star point etc) will now have their own add dialog library name and filters remembered.

Start Schematic Connection on Testpoint Part

You can now start or end a connection on a **Testpoint Part**.

Previously the option **Start/End on Testpoint** always entered a dialog to add a test point doc symbol. There was no way to change that symbol as you could only change the name with **Change Testpoint**.

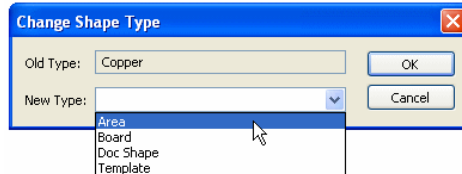
Now **Start/End on Testpoint** enters the same **Change Testpoint Symbol** dialog that is used in **Insert Testpoint**.



This allows you can choose to add Parts or doc symbols. When selected you will now have **Change Name** and **Change Testpoint Symbol** available on the **Start/End On** context sub-menu the same as **Insert Testpoint**.

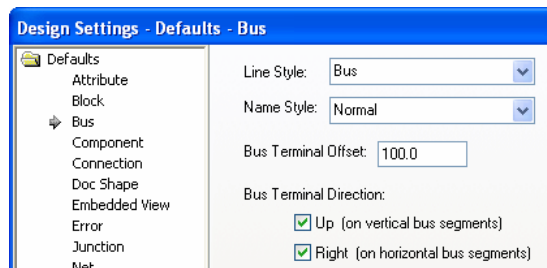
Change Shape Type on multiple-selected items

You can use **Change Shape Type** on multiple items of the same type. This enables you to change multiple areas of documentation shapes to copper for example.



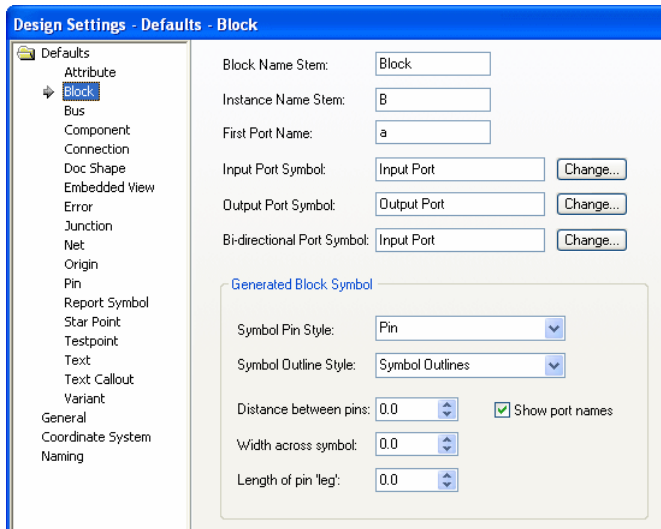
Changes to Defaults option

Bus – You can now set up a default **Name Style** for a bus name.



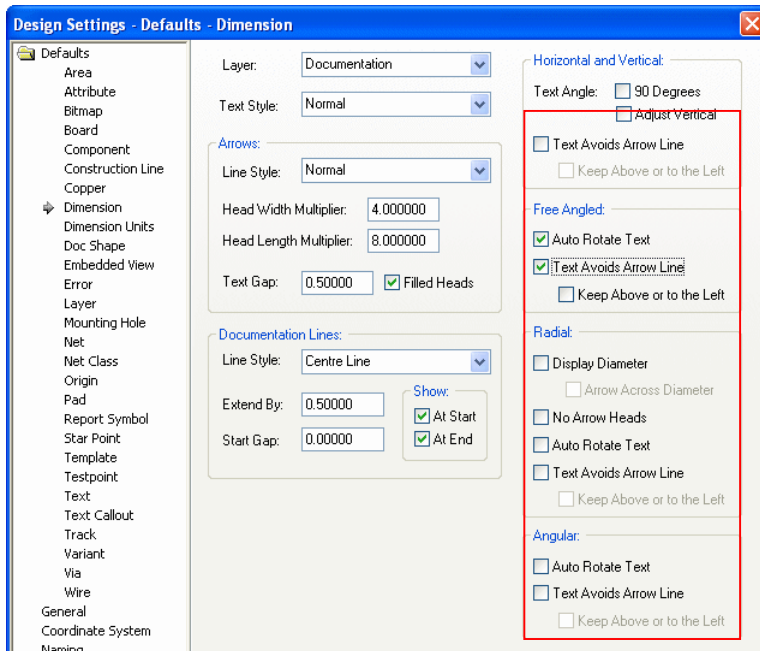
Net Class – The Net Class default has been removed from footprint defaults.

Block – **Block** defaults now include a **Generated Block Symbol** section.



These are the defaults to be used for **Cut To Block** and **Regenerate Block Symbol** instead of using the registry values last saved from the **Insert Block Instance - Specify** dialog. **Insert Block** initially now uses these new defaults, though you can still use the **Specify** button to change the block symbol appearance for that session.

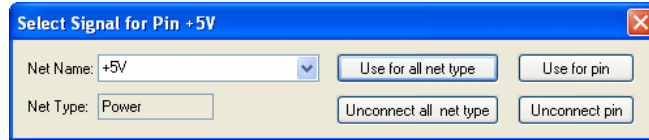
Dimensions – This page has been updated to offer text **auto rotate** and **avoid arrow line** switches for each dimension type.



There is also now an alternative mode for **Avoid Arrow Line** to always place text **Above or to the Left**. This supports the DIN standard for dimensions.

Applying Predefined Signals

When adding components or doc symbols in Schematics which require you to **Select Signal for Pin**, a component that has pins with predefined pin types for example, when the dialog is presented there are now additional buttons to make it more clear what the **all** buttons do. Previously, the buttons applied to all the remaining pins no matter their type, but now they only apply to the remaining pins of the same type.



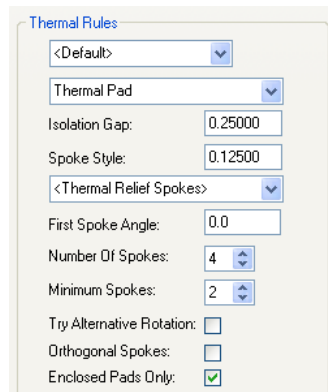
Select a net from the list and press the **Use for pin** button to apply it to the pin, or press the **Unconnect pin** button to leave the pin unconnected.

Alternatively, use either of the two **all net type** buttons to save time if you know that all the pins of the same type on the symbol or component being added are to receive the same net. This net will then be remembered by the design as the default name in this dialog for this net type in the future.

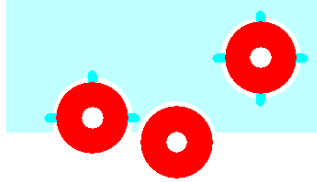
- Press the **Use for all net type** button to apply the chosen net to all of the remaining pins on the symbol or component that have the same predefined net type as shown in the dialog.
- Press the **Unconnect all net type** button to leave unconnected all of the remaining pins on the symbol or components that have the same predefined net type as shown in the dialog.

Spokes only on enclosed pads

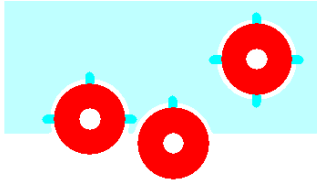
You can restrict which pads receive spokes to only those where the pad origin is contained within the template. To enable this, use the **Enclosed Pads Only** check box under **Thermal Rules** on the **DFMDFT Rules** page of the **Technology**.



Pouring with this switched selected shows the effect on the middle pad. Because the pad origin is outside the template area, no spokes are added:



Without the switch selected, a spoke is added to connect the pad to the template.

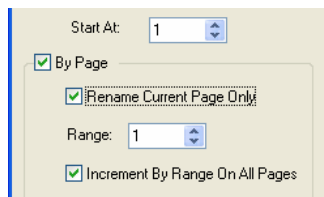


Move Track Segment Improvement

It is easier now to place a single track/copper segment at the minimum distance from another object. With **Online DRC** enabled and **Track Pushing** Off, move generally tries to move the selection to its new position and if it is in error it bounces back to its last known legal position. With the grid switched off (**Use No Grid** enabled), this makes it very difficult to place a segment at the minimum distance from a pad as you have to move very slowly. Moving a single segment has been changed to bounce back to the minimum distance from the obstacle, so with the grid off, if you move past the object it is placed as close as it can to it. This change makes it easier to push a track segment tight to a pad in order to get another track through a gap.

Auto Rename By Page in Schematics

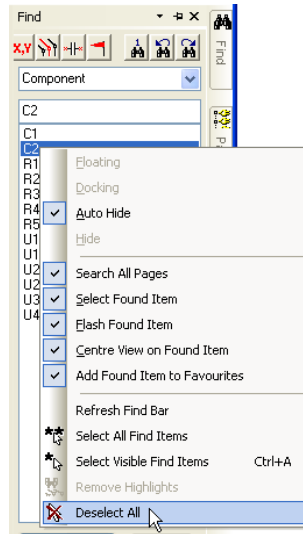
There is a new option in **Auto Rename** in Schematics under the rename **By Page** section, this is called **Increment By Range On All Pages**.



Checking this will ensure that the start number will be used on the first page and name numbers will be incremented by the range for every page, even if there are no items to rename on the page. Leaving this unchecked will work like it did previously and only increment by the range for each page the items are on.

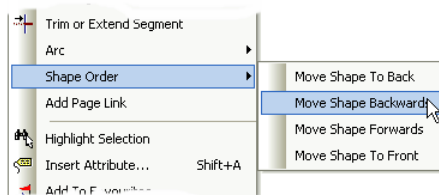
Find Bar in Schematics

The behaviour of **Deselect All** from the context menu has changed for Schematic designs. If using a schematic design and **Search All Pages** is checked, all items on all pages of the design will be deselected. If **Search All Pages** is unchecked, only the items on the current page will be deselected.



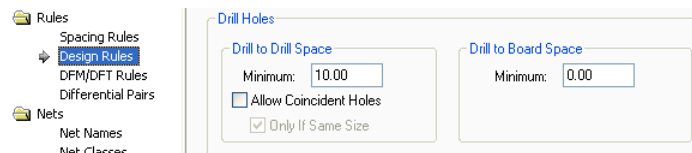
Definable Drawing Order for Doc Shapes

Doc Shapes in Schematics or Schematic Symbols may now have a defined order. Imported symbols will now show shapes in the order they are created. The order of shapes may be modified using the **Move Shape To Front**, **Move Shape To Back**, **Move Shape Forwards** and **Move Shape Backwards** options available on the context menu or as commands for a selected shape or symbol.



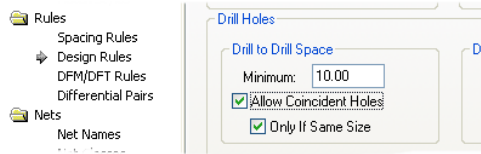
Check Drill Holes against boards in DRC

You can check drill holes against boards in the DRC dialog using the **Board** and **Drill** checks under the **Spacings** category. The **Drill to Board Space** distance is defined on the **Design Rules** page of the **Technology**.



Coincident Drill Check

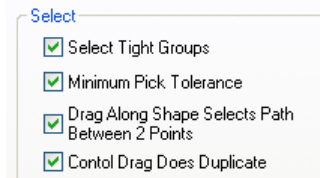
There is now an option in the **Technology** and **Design Rules** to allow coincident drill holes and to allow them if they are different sizes.



If **Allow Coincident Holes** is checked, you can decide if the two drill holes must be the same size by checking **Only If Same Size**.

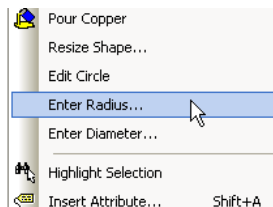
Switching off Ctrl-Drag To Duplicate

There is a new interactive option, **Control Drag Does Duplicate**, on the **Interaction** page of **Options** dialog. This allows users to switch off this option in PCB for example.

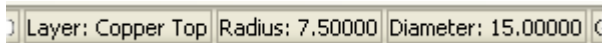


Display of Diameter for selected Arcs/Circles

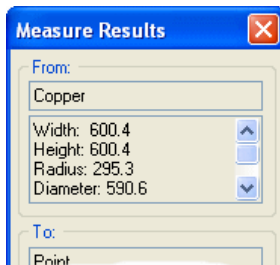
With a circle selected you can now directly use **Enter Radius** or **Enter Diameter** from the select mode context menu.



The **Edit Circle** and **Edit Arc** options now show the **Radius** and **Diameter** on the **status** bar.

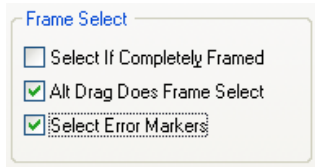


The **Measure** tool now shows the **Radius** and **Diameter** for circles and arcs.



Frame Select for Error Markers

On the **Interaction** page of **Options**, there is a new check box option to allow **Frame Select** to **select error markers**. Use this to include or exclude error markers when using Frame Select. You may want to enable it to actually select all error markers in a region for example to remove them in one go.



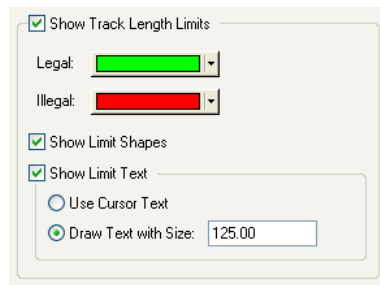
Tooltip for Track Length Limits in High Speed Option

For users who have purchased the **Interactive High Speed** option, there is a new alternative option for viewing the Track Length text. By default, all existing users will be switched over to this new method but the old option is still available if required.

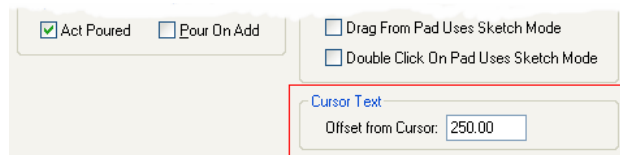
You can choose to **Show Limit Text** which will display the estimated track length and the minimum and maximum limits which currently apply, this text is drawn next to the cursor and updates as you move. You can choose between two methods of displaying this text.

The **Use Cursor Text** option displays the limit text similar to a tooltip always on top of your design keeping it more legible especially in dense areas of the design. You can alter the distance the text box is from the cursor using the **Interaction** options. You can also use the **Reposition Cursor Text** command whilst cursor text is being displayed to change its position relative to the cursor.

The **Draw Text** option simply draws the limit text in the design window. You can specify the height of this text in the current design units. This is the actual height on the screen and is not related to the current drawing scale of the design.



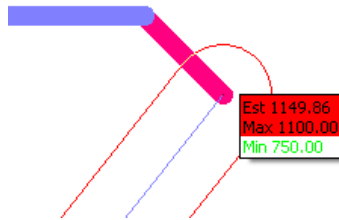
There is also an option on the **Options** dialog and **Interaction**, **Offset from Cursor**.



Some interactive operations may display information text next to the cursor. For example, displaying **Track Length Limits** when editing tracks. Use **Offset from Cursor** to define how far this text is from the cursor. You can also use the **Reposition Cursor Text** command whilst cursor text is displayed to change its position relative to the cursor.

22 Pulsonix Version 6.1 Update Notes

The tooltip is setup to show a default tooltip background (white) with the limits text in the legal colour (green) and for illegal text to be shown using reverse-highlight with the illegal colour as background (red).



Support for P-CAD Master Designer Schematics Import

As a cost option on the PSX-IMPORT option, Pulsonix will now support the P-CAD Master Designer Schematics import of Schematic designs and Libraries.

Library Changes in V6.1

In Pulsonix 6.1, new Parts libraries have been added and some existing libraries modified, the changes comprise the following:

- Connectors have been added to create new library files:

AMP	Hirose	Molex
Assman	JAE	NorComp
CWR	JST	Samtec
Glenair	MiscConn	Sullins
FCI		

- Microcontrollers have been added to create new library files:

Analog Devices – ADSP	Luminary Cortex	Philips – P89
Altera – Cyclone	Maxim – MP	Philips – XA
Atmel – 8051	Maxim - DS8x	Power – PC
Atmel – ARM	Maxim 8051	Renesas R8C
Atmel – AVR	MC68HC	SonyCX
Atmel – AT91	Microchip – PIC	ST-ARM7
Atmel ATSAM7	MSP – 430	TMS – 320
ColdFire	NXP LPC32x0	TMS – 470
Fairchild – ACEX	PICAXE	ToshibaTMP86
Infinion TC116x	Philips – ARM	Zilog
Intel - 8051	Philips – 80C51	

- Additional Parts and associated Schematic symbols and footprints have been added to create new library files of more general Parts.

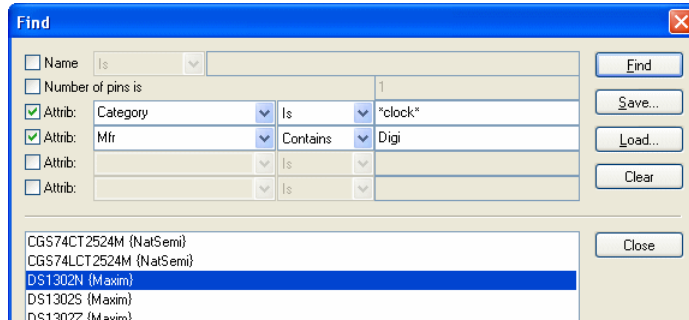
Additions for libraries from:

Altera	Maxim	ST
Atmel	Microchip	TI
Cypress	NatSemi	Xilinx
Lattice	Quicklogic	

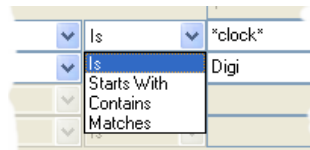
Options previously released in V6.0 but not documented

Advanced Parts Searching

From the **Insert Component** option, the **Find** dialog can be used to create more advanced searching based on additional criteria such as Attributes. These can be combined using the match criteria to focus the search.



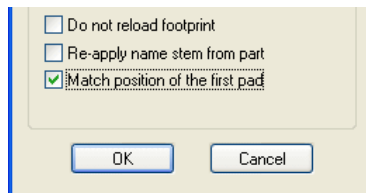
For all the values apart from the number of pins, there is a drop-down list where you can choose how the values of the selected field should be matched:



- Is - matches exactly
- Starts - has the specified text at the beginning of the value
- Contains - has the specified text somewhere in the value
- Matches - You can choose to match with the part name, number of pads, and up to four attributes.

Reload Part using Pad Origin

When using **Reload Part**, for Footprint changes, you can specific to match the position of the first pad. This can be used when a surface mounted pad symbol origin is swapped from pin one to the component centre for example.



When the **Match position of the first pad** is checked, the position of the reloaded component is determined by matching the position of the first pin on the footprint, rather than the footprint origin.

Support for PADS 2007 format on Import

Pulsonix now supports the import of Mentor PADS 2007 ASCII format.

Support for Eagle V5.x format on Import

Pulsonix now supports the import of Eagle V5.x ASCII format for designs and libraries. Use the Eagle ULP files supplied with Pulsonix to convert Eagle files into an intermediate format that can be read by Pulsonix.