

Pulsonix Design System V11.0 Update Notes

Copyright Notice

Copyright © WestDev Ltd. 2000-2021 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 and Windows NT 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: 11/05/21 iss 4

Pulsonix

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

Support Phone	+44 (0)1684 296 570
Sales Phone	+44 (0)1684 296 551
Email	sales@pulsonix.com
Web	www.pulsonix.com

Contents

CONTENTS	3
VERSION 11.0 UPDATE SUPPLEMENT	7
Installing the New Version of Pulsonix	7
Licensing	7
Version 11.0 Update Notes	7
Installation	
New In Version 11.0	8
Report Maker – List of Files	8
Part Edit – Force Display of user attributes	
3D View Settings (STEP) Change	
Customize – Fixed Icons	
New Direct-X Graphics Engine	
International Language and Extended Character Support	
Inspector Bar (Dynamic Properties)	
Changes to Document Properties Dialog	
Vertical Text Alignment for Multi-line Text	12
Option for Track & Vias styles defined in SCM to be Translated to PCB	13
Options Dialog Changes	16
Option for Spacing Rule Grids	16
Thread Count Definition	17
New Options page for Pan & Zoom functionality	17
New Auto-Pan feature	
Probe Synchronised Design option added to Cross-Probe	
Removal of Fast Locate Option	
Edit Track Option for using Layer Bias	
Optimise On Clear Template Option	
Clear All Templates - Optimise now Multi-thread	
Optimise Settings for Large Nets	
Large Net Warning on Optimise All Nets	
Optimise All Nets Multi-Threading	
Net Optimise for Pad Centres within a Shape	
Dynamic Align of items	
Select from popup list	
Part Editor - Dynamic Attributes for Part Description field	
Save Items to Library dialog changes	25
Save Missing Component Items to Library	
Part Wizard changes	
Add Selected Footprint/Symbol	
Pin Type cell new option to Apply to Undefined	
Vault Library Export to Specific Folder	
Attribute Properties – View Name and Value for Protected status	
Group Names Displayed in Component Bin	
Toggle Bin Sort Order in Component Bin Dauble Clieb colortion in Schotien Mark Den	
Double-Click selection in Selection Mask Bar Override Readable Orientation of Text	
Split Net Highlights	
Shape Information/Editing Bar	

Scale Option	
Change Part - Same Footprint check box	34
Find Bar Changes	34
Multi-Select in Find Bar	34
Enhanced Selection	35
Fetch Component from Find Bar	
Dimming All Non-found Items in Colour	
Search for Non-Connecting Copper	
Close Other Pages (SCM)	
Allow Implied Junctions Pin Override Switch (SCM)	
Copy Net Names on Sig Refs with Ctrl-Drag (SCM)	
Insert Attribute Position (SCM)	
New Colour Category for Bus & Connections (SCM)	
No Connect Pin Highlight Colour (SCM)	
No Connect Pin context menu option to Show No Connect	
Display Pin Type Attribute (SCM)	
Changes to Insert Signal Reference dialog (SCM)	
New Signal Reference Type - Net labels (SCM)	
New Doc Symbol Type – Net Label	
Inserting Net Labels into your design	
Insert Connection - Start/End on a Net Label	
Next/Previous Doc Symbol Command	
Mounting Hole Symbols in Schematic (SCM)	
Copy Net Names for Signal References (SCM).	
Text Formatting option for Spice Netlist Export (SCM)	
Weld multiple Components to a Bus segment (SCM).	47
New ERC Check - Nets Only On Ungated Pins (SCM)	
First Free Component Name in Properties (SCM)	
New PCB Wizard	
Changes to Database Check and Update Options	
Technology Changes	
Pad Styles – new Usage Types	
Drill Removes Pad Warning now shows Pad Style	
Units shown in Technology	
Additional Cell Status Indicators	
Updated Design Settings Pages	
Changes to Grids dialog	
Group Name available in Component Place Rules	
Component Placement Rule - Default Mirror State	
Pad Styles - New Naming Rule for Non-round Holes	
Micro-Via to Buried Via Stagger Spacing	
User Defined Pad Shape Improvements	
Technology Layers - Reflect Layers	58
Thermal Rules – Rotate with Pad	
Thermal Rules for additional Pad Types	
Copper Neck Width Rules (Power Dissipation)	
Creepage Rules	61
Design Settings - new Synchronise Design options	63
Import IPC-2581 Layer Stackup into Layers dialog	64
Layers Import/Export CSV into Layers dialog	66
Option to use a spacing shape even when pad is suppressed	67
Load Technology - Matching Styles on Reload	68
Rotated Pad Styles for 'Long' Pads	
Pad Properties Layer Override	68

Lock Pad Details in Footprint Editor	
Define Mirrored Footprints in Footprint Editor	
Area Colour in PCB Doc Symbol Editor	
Background Dimming on Mark Net	
Copper Pour Multi-Threading	
Optimise after Clear All Templates Multi-threading	71
Change Style DRC Checks Multi-Threading	71
Change Style Performance Update	
Multi-Threaded Design Rule Checking	72
New Design Rule Checks	
Same Net Via To SMD	
Silkscreen Overlap	74
Time To Process shown in DRC Report	
Changes to Online DRC	
Display Clearances for Multiple Items	
Changes to Show Design Rule Clearance	
Continuous Online DRC switch added to context menu	
'End Track On Via' Clearances	
Continuous DRC and Via Errors	
Allow Checking of Multiple Items	
Dimensions Changes	
Hide Arrows on a Directional Dimension	
Linear and Radial Dimensions – Show both Metric and Imperial Units	
STEP 3D Changes	
Minimise STEP File Size Option in STEP Output.	
Improved Trihedron Axes Indicator for STEP Preview & Models	79
Added ability to 'Align' STEP items	
Added ability to 'Orient STEP items	
View and Alignment options added to 'Position STEP Model' dialog	
Import Mounting Holes and Vias into design from STEP Model	83
Import STEP Board Placement Sites	
3D Package Viewer Retirement	84
Line Select Mode	
Changes to Frame Select and Polygon Select Modes	
Changes to Double Click to Edit Mitre	
Insert PCB Track more responsive	
Restricted Movement Segment Mode – Snap To Angle Step	
Select Track Paths by selecting Components	
Copy/Paste into new Shape Type	
Layer Control Added To Change Shape Type	
Pad Auto Necking	
Auto Necking	
Add Item to Net in Net Properties Dialog for Copper and Templates	
Add Copper & Template Shapes to Net on Insert	
Non-Connecting Copper Highlight Colour.	
Reversed View Status Saved	
Attributes on Wires	
Layers Bar – Components Filter	
Toggle Layers Changes	
Alias Assignments	
New Toggle Layers Bar	
New Toggle Plane Command	
Measure Bar showing Spacing Rule	
Error Bar - Tooltip available on Error Rule	

Placement Vector - Include Power/Ground Net	95
Keep Existing Rotation in Swap Component Positions	95
Spread-Out Option Changes	96
Auto Insert Testpoints – Include Unreachable Testpoints	96
Create Breakout Pattern (BGA Fanout) in PCB	97
CAM Plot Changes	
Multi-Threading of Gerber Plots	98
Excellon Setup Min Drill Diameter	98
Optional Formatting information for Excellon output	99
Output file names template – Use Default button	99
Output to SVG Device	100
Output to ZIP File	
Panel Editor – Reset Layout Command	101
Track Impedance Calculator	
Design Calculators – Values taken from design	102
Interactive High Speed Option - New Rules & Changes	104
Back Drilling	104
Track Impedance Rules	112
Remove Serpentine Controls Option	
Rules Spreadsheet Multi-threading	113
Create Differential Pairs From Rules	
Create Differential Pair using Context Menu	
Define Differential Pair Rules using Net and Net Class Attribute	
Create Differential Pair Chain	
New 'Start Pairing' Mode for Differential Pairs	
Differential Pair Mirror Mode	
Legal Completion Path for Differential Pairs	
Change Style of Differential Pairs	
Shift to add corner in Differential Pair Mirror Mode	
New Via Pattern option - Auto Turn	125
Use copy of existing Differential Pair Via Pattern	
Editing Differential Pairs into Areas	
Differential Pair Properties	
Differential Pair Track Limits Display	
Chip-On-Board Option Changes	
Change Chip Body Layer	
Change Layer of Chip Body during Move	
Change Layer of Chip Bond Pads	
Report Maker Changes	
Scripting Changes	132
Changes to Import Alien Design Files and STEP Model Positions	134
Loading Partial Colour Files	134
New PCB Design Dialog – Set Design Units	
Changes to Default Supplied Files	135

Version 11.0 Update Supplement

Installing the New Version of Pulsonix

It is always 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, double-click on the download executable and wait for a short time. You may have been supplied a CD, in which case insert it and wait for it to run. The *Autorun* facility will start the installation procedure. Follow the on-screen commands from the install wizard. You can install Pulsonix 11.0 over your existing V10.5 installation. If upgrading from V10.5, you can install it alongside the older version if you prefer. In any case, you do not need to uninstall the old version first unless you wish to remove it from your hard drive.

Licensing

Version 11.0 requires a new license if you are a new user or upgrading from any older version of Pulsonix earlier than V10.0or V10.5. The new license 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 in the same location as the current one but make a backup copy first or rename it. 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.

Version 11.0 Update Notes

These Version 11.0 Update Notes are available in the Pulsonix product installation folder under \documents. Alternatively, select the direct link within the product itself: **Help** menu | **Online Manuals> Pulsonix V11.0 Updates Notes**. You can also find them on the Pulsonix web site.

The current and all previous Update Notes are available on the Pulsonix web site under **Documentation**.

Installation

The installation has been changed so that all Pulsonix 'documents' (Master Libraries, Technology files etc.) are now located under user\documents\Pulsonix11.0 rather than being placed in public documents\PulsonixXX

New In Version 11.0

Report Maker – List of Files

When creating a report using **Report Maker**, a new list command option **List of Files** has been added which when the report is run, will list all the files within a chosen folder.

The destination can be edited by double clicking the command in the format script section. A folder path can be added by browsing a folder using the browse button, or being picked up by a report variable or by typing in a path (this benefits from relative paths, so folders within the design folder can be selected by just typing in that folder's name). Whilst within the List of Files indentation in format script, another new command **File Name** is available which reports the name of the file that is found.

Part Edit – Force Display of user attributes

When editing a Part in the **Part Editor**, the **Force Attribute Displayed** dialog, available from the **Edit** menu, now includes two lists for you to define what attributes you want to force displayed when adding the Part to a schematic or PCB design. If not already in the symbol, the attribute positions will be added underneath each other in the order from the list.

3D View Settings (STEP) Change

To avoid graphical issues (dithering, also known as Z-fighting) when drawing two or more items at the precise same location, components are added to the STEP data using a tiny Z-offset to render them very slightly above the surface of the board. The same is true of additional design data that might be enabled (such as Pads).

If for some reason you need this tiny offset to be removed so that items are rendered at precisely the same Z value then two new check boxes on the **Output** tab of the **3D Settings** dialog allow you to do this:

	3D View S					
	Settings	Colours	Interaction	Enclosures	View	Output
× 1	🗹 Minii	mise STEF	^o file size			
Place items directly on board surfa						
/	Only when generating STEP output file					

The default setting for **Place items** check box is unchecked. On selection of this box, the next check box becomes available. This second check box, **Only when generating STEP output file**, controls whether the decision to place items directly on the board is done in the on-screen Preview or only when writing data to a STEP file. Note that if you check this box there is a performance overhead as the 3D data has to be regenerated before being written to the STEP file so that items are positioned as required in the correct position on the Z axis.

Customize – Fixed Icons

The default icons for the **SVG** and **STEP** commands on the **Output** menu bar were incorrect and have been updated to use the correct icon to represent the command. To view this change, you will have to reset the toolbar in the **Customise** dialog in order to get the fixed icons.

New Direct-X Graphics Engine

Pulsonix 11 now supports Direct-X graphics. From the **Options** dialog, **Display** page, there is a new slider for **Hardware Acceleration** to enable the new graphics engine. For large designs, this may take a moment to rebuild the graphics.

This slider will allow you to choose the level of hardware acceleration you wish to be applied when rendering graphics in a design window and balance the competing needs of initial graphics generation time, pan and zoom performance and amount of memory used.

The slider provides 5 possible settings; the first, leftmost position, indicates minimal hardware acceleration with purely GDI drawing and is the equivalent of the old **Enhanced Graphics** setting being unchecked. The **Enhanced Graphics** button has been removed from the dialog.

The second position is the equivalent of **Enhanced Graphics** being enabled and, like in 10.5, provides anti-aliased circle drawing but no performance improvement.

Positions three to five provide new Version 11.0 settings and indicate varying degrees of hardware acceleration utilising the graphics card GPU. The higher the position, the more improved the pan and zoom performance will be, but at the cost of a greater initial graphics generation time and increased memory usage. Moving the slider to the **High** position will utilise all the new graphics features and use any memory available and required.

The Flicker Free Redraw option has also been moved from the Display page.

Options Display		— 🗆 X
Design Backups Display Edit Shape Edit Shape Edit Track File Extensions Find	Fast Redraw ✓ Schematic ✓ PCB ✓ Draw in Layer <u>O</u> rder ✓ Draw Current Layer On <u>T</u> op	Hardware Acceleration Low High Reset
General In-Place Names Interaction Macros Move	Dim Other Layers All Dim in One Colour Decluttering	View All'On Opening Designs Draw Dynamic Text Origin Draw Grids Underneath Items

International Language and Extended Character Support

Pulsonix version 11 has been switched to use the full Unicode instruction set. This means full international language and extended character support. This will support special characters like Ω , \neq , \leq , \geq , \pm , \emptyset for example and also Chinese and Korean languages. Support is through all design editors, libraries, import filters and exports such as Gerber, Netlists, Parts Lists and all 3rd party products.

Inspector Bar (Dynamic Properties)

The new Inspector Bar acts as a dynamic Properties windows, however, it is different.

Accessed by right clicking on the Pulsonix framework and selecting **Inspector** off the list of bars available, or access it from the **View** menu and selecting **Inspector Bar**.

Generally speaking, the Inspector Bar would be left either docked open or floating so that it is accessible instantly.

With an item selected, the Inspector Bar will display Properties for that item, for example, a selected Component in PCB:

Component: IC4 (ATMEGA16U2-MU) Lock Locked Transform Position 19.939 34.798 Angle 90.0 Mirrored Layer <top side=""> Layer <top side=""> Explore Net Component Name Part ATMEGA16U2-MU Change Postprint QFN50P500X500X100-33N Pin Names Pin Logic Names Auto Footprints Name Value Visible Source</top></top>	Insp
Lock ^ Locked _ Transform ^ Position 19.939 34.798 Angle 90.0 Mirrored _ Layer ^ Layer Component Layer _ Explore Net _ Name IC4 Pat _ Antimetes _ Footprint _ QFN50P500X500X100.33N _ Pin Names _ Pin Logic Names _ Auto Footprints _ Value Visible Source <tompenent< td=""> _</tompenent<>	Selec
Locked □ Transform Position 19.939 34.798 Angle 90.0 Mirrored □ Layer Component Component ATMEGA16U2-MU Change Part □ ATMEGA16U2-MU Change Footprint □ QFN50P500X500X100-33N Pin Names □ Pin Logic Names □ Auto Footprints □ Component Attributes Name Value Visible Source Name Value Visible Source Part Name Value Visible Source Name Value Visible Source Part Name Value Visible Source Part Name Value Visible Source Part Par	Conn
Locked □ Transform Position 19.939 34.798 Angle 90.0 Mirrored □ Layer Component Component ATMEGA16U2-MU Change Part □ ATMEGA16U2-MU Change Footprint □ QFN50P500X500X100-33N Pin Names □ Pin Logic Names □ Auto Footprints □ Component Attributes Name Value Visible Source Name Value Visible Source Part Name Value Visible Source Name Value Visible Source Part Name Value Visible Source Part Name Value Visible Source Part Par	
Transform Position 19.939 34.798 Angle 90.0 Mimored Layer Layer Component Name IC4 Pat ATMEGA16U2-MU Change Postprint QFN50P500X500X100-33N Pin Names Pin Logic Names Auto Footprints ✓ Component Attributes	Lock
Position 19.939 34.798 Angle 90.0 Mirrored	Locked
Angle 90.0 Mirrored □ Layer ▲ Layer <	Net
Mirrored Layer Layer Corponent Name IC4 Part ATMEGA16U2-MU Change Description 8-bit Microcontrollers - MCU AVR USB 8K FLAS Pat Family IC4 Pat Family QFN50P500X500X100-33N Pin Names Pin Logic Names Auto Footprints Component Attributes Name Value Visible Source < STEP Filename> ATMEGA16U2-MU.s	Source
Layer <top side=""> Layer <top side=""> Explore Net <</top></top>	Name
Layer <	Sub Net Or
Layer <top side=""> Explore Net Component Name IC4 Part ATMEGA16U2-MU Description &bit Microcontrollers - MCU AVR USB 8K FLAS Part Family </top>	Net Class
Explore Net Explore Net Component Name IC4 Part ATMEGA16U2-MU Change Description B-bit Microcontrollers - MCU AVR USB 8K FLAS Part Family Alternate Footprint QFN50P500X500X100-33N Pin Names Pin Logic Names Attributes Component Attributes Name Value Visible Source STEP Filename> ATMEGA16U2-MU.s Part	Net Type
Component • Name IC4 Part ATMEGA16U2-MU Description 8-bit Microcontrollers - MCU AVR USB 8K FLAS Part Family Atemate Footprint QFN50P500X500X100-33N Pin Names Pin Logic Names Auto Footprints Component Attributes STEP Filename> ATMEGA16U2-MU.s	Own Colour
Component • Name IC4 Part ATMEGA16U2-MU Change Description 8-bit Microcontrollers - MCU AVR USB 8K FLAS Part Family Alternate Footprint QFN50P500X500X100-33N Pin Names Pin Logic Names Component Attributes Name Value Visible SOURCE	Explore Net
Part ATMEGA16U2-MU Change Description Bibit Microcontrollers - MCU AVR USB 8K FLAS Part Family Alternate Footprint QFN50P500X500X100-33N Pin Names Pin Logic Names Auto Footprints Attributes	Net
Description 8-bit Microcontrollers - MCU AVR USB 8K FLAS Part Family Alternate Footprint QFN50P500X500X100-33N Pin Names Pin Logic Names Auto Footprints Alternate Component Attributes Alternate STEP Filename> ATIMEGA16U2-MU.s Part Part	Name
Part Family Alternate Footprint QFN50P500X500X100-33N Pin Names Pin Logic Names Auto Footprints Component Attributes Name Value Visible Source STEP Filename> ATMEGA16U2-MU.s Part	Class
Footprint QFN50P500X500X100-33N Pin Names Pin Logic Names Auto Footprints Component Attributes Name Value Visible Source <step filename=""> ATIMEGA16U2-MU.s Part</step>	Nodes
Pin Names Pin Logic Names Auto Footprints Component Attributes Name Value Visible Source STEP Filename> ATIMEGA16U2-MU.s Part	
Pin Logic Names Attributes	
Name Value Visible Source <step filename=""> ATMEGA16U2-MU.s Part</step>	Component
Name Value Visible Source A <step filename=""> ATMEGA16U2-MU.s Part A</step>	Name
Name Value Visible Source A <step filename=""> ATMEGA16U2-MU.s Part Part</step>	Part
<step filename=""> ATMEGA16U2-MU.s Part</step>	Nets
Allied_Number Part	
Datasheet Link http://ww1.microchi Part Height 1 Part Y	

Context Menu Options

By right clicking on the Inspector Bar, you can access additional commands:

Change Group Order

Select **Change Group Order** to rearrange the ordering of the 'groups' or sets of properties displayed in the Inspector Bar.

Use the Up and Down arrows to move the selected group up or down in the Inspector Bar list.

Groups can be shown or hidden in the Inspector Bar using the check boxes next to the group or by using **Show All/Hide All**.

Change Group Order	>	×
Selection	Up	
☐ Layer ☑ Net	Down	
Explore Net Component Line Style	Show All	
Text Style Pad Style	V Hide All	
ОК	Cancel	

Explore Net

The group **Explore Net** enables you to see the nodes on a selected net and the nets on the component (if the node is from a component). This functionality also works in reverse, so for a selected component, the Inspector Bar will show all nets on that component.

Selecting a node or net from one of the lists in the group will update the other list (Net or Component or vice-versa), allowing the connected nets and nodes to be traversed (or explored).

Explore Net		•
Net		
Name	+5V	
Class	Power	
Nodes	C7.1 C8.2 C12.2 J9.5	^
	J10.1 LCD-PIN1.2 R2.1	v
Component		
Name	J9	
Part	22-27-2061	
Nets	\$16 \$17 \$18 \$33 +5V Gnd	
		v

Single Clicking to Navigate Nets & Nodes

Single click will 'navigate' net and nodes, using a double click will highlight net.

Find Options

This offers the same finder functionality as the **Find Bar**, with the Find Bar options being used here as well. These options can be viewed and changed from the Inspector Bar by right-clicking anywhere on the bar and hovering over the **Find Options** context menu (see below).

	Floating
	Docking
	Tabbed Document
~	Auto Hide
	Hide
	Select Found Item
~	Highlight Found Item
~	Brighten Found Item
	Dim Items In Colour
~	Flash Found Item
	Centre View on Found Item
~	Add Found Item to Favourites
	Close All Groups
	Open All Groups
	Change Group Order

Changes to Document Properties Dialog

Change to Summary Page

A new item on the **Summary** page of the **Design Properties** dialog has been added. This shows you the current full path name of the design file.

	Document Properties Summary				
	Summary Statistic	cs Password			
	Application:	Pulsonix			
	Author:	Pulsonix			
	Keywords:				
	Comments:	~	1		
		~			
	Title:	10L-SM-A			
	Subject:	Routing Example	1		
_)	Path Name:	C:\Users\Downloads\10L-SM-AUG.pcb			
V	Technology File:				
	Profile Name:	A3-Landscape			

Vertical Text Alignment for Multi-line Text

There is a new switch in **Design Settings, General**, under **All Text** to set **Multi-Line Text Vertical Align**. When selected, this causes all multi-line text to vertical align using the height of all the text, not just the first line. It will change the vertical alignment of all multi-line text in your design so care must be taken if applying this after already having added multi-line text, especially on electrical layers.

All Text	Attributes
Adjust To Readable Orientation	Substitution Char: (when %
Multi-Line Text Vertical Align	doubled)
Barring Character:	Matching Styles
(when doubled)	By Name Only
	O By Name And Value
System Font	
	O By Value Only
Descention of Minkle Distant	

The two images below show the effect of this switch. To clarify where the alignment is taken from, horizontal and vertical construction lines have been added to the text to show the movement.

The left image is before switch is activated (and as current product), the right-hand image is after:

Multiline text	Multiline text spread across different lines showing alignment
spread across different lines showing alignment	

Option for Track & Vias styles defined in SCM to be Translated to PCB

A new option in the **Design Settings** option under **Synchronisation** now allows **Net Styles** in PCB to be created from the **Track & Via Size Rules** in the Schematic. Use the **Apply Schematic Track & Via Size Rules to Net Styles** switch to enable this feature.

Grand Specing Rules Styles Styles Layers Rules - DFM/DFT Rules - High Speed Outputs Nets	PCB in Safe Mode Allow PCB Only single pin nets Synchronised Design Name Name: HSE6612 Back Annotation
Naming Colours Grids Outris	Enabled Clear History Synchronise with Schematic Analy All Data Sciente
	Apply All Rules Strictly Apply Footprint Changes Apply Net Class Changes Apply Net Clours Apply Net Colours Apply Schematic Track & Via Size Rules to Net Styles
 Defaults Naming Placement Sites Synchronisation Parameters 	Appy Schematic Track & Via Size Rules to Net Styles Ignore Attribute White Space Allow Update of Schematic to match PCB

This means, where you need to specifically define Track and/or Via Styles in the Schematic, they can now be passed through into your PCB where they will be used in Net Styles. Both the **Translate To PCB** and **Synchronise Designs** options will use this feature once the switch is set.

If **Translating to PCB** to create the initial design, your PCB Technology used must already have this switch set in order to pass across the settings. If it isn't set, then once set, use the **Synchronise Design** option to update the design with these settings.

The **Track & Via Size Rules** in the Schematic should be defined in accordance to your requirement. These can be generic such as all **Net Names** in the design, or more refined such as specifically named nets such as DQS*. Other parameters can be selected from the drop-down list as well as **Layers**, **Sides** and **Within Areas**.

Attribute:	<net name=""></net>	
Match:	<differential name="" pair=""> <net class="" name=""></net></differential>	*
On Layers	< <u>Net Name></u> <net type=""> <signal name="" path=""></signal></net>	
Side		~
Layer	:	~
Within Area	IS:	~

Design Settings Synchronisation

Once the **Apply Schematic Track & Via Size Rules to Net Styles** switch has been checked in the **PCB Design Settings Synchronise** dialog, you can then define the **Track and Via** styles in your **Schematic** design.

From in the **Schematic** design, use the **Technology** dialog and **DFM/DFT Rules** – **Track & Via Size Limit** dialog to add rules that will be passed into your PCB design.

Attribute:	<net name=""> ~</net>	Allow Tracks
Match:	GND* 🗸 🔊	Track Width
On Layers		Min: 5.5 Max: 9.9
Sid	e: <any> ~</any>	
Laye	r: 🔍 🗸	→
		Via Diameter
Within Are	as: V	Min: 35.3 Max: 44.5

Translate to PCB

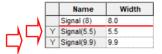
During **Translate to PCB**, with this switch set in your **PCB Technology file**, this setting will be initially applied to the new design and your **Schematic Track & Via Size Limit Rules** will be defined in the PCB

If the **Track & Via Size Limit Rules** are applied to the PCB, it will add new **Track** styles and a new **Pad Style** to the **Technology**. Two new Track Styles will be created based on the Min and Max Track width defined. The Pad style created will use the Min and Max Via Dimeter to create a pad of the Max diameter size with a drill using the Min Diameter value.

A new Via Pad style is added to the Pad Styles dialog:

Na <u>m</u> e: Used: 🗹	Shape:	Drill:
Rnd44.5h35.3	Type: Round	✓ Shape: Round ✓
Named by: OTyped Rule OTemplate	<u>W</u> idth: 44.5	Width: 35.3
<use><shape><width><length:_><drilled:h><n by:<="" for="" td="" use=""><td>Length: 44.5</td><td>Length: 0.0</td></n></drilled:h></length:_></width></shape></use>	Length: 44.5	Length: 0.0
☐ Through Hole Pads ☑ Vias ☑ Surface Mount Pads ☑ Micro-vias ☑ Through Mounting Holes ☑ Surface Mounting Holes	<u>O</u> ffset: 0.0 0.0	Inner Diameter: 0.0 Offset: 0.0 0.0 Rotation: 0.0

New Track styles are added to the Track Styles dialog:



Matching settings in the **Net Styles** dialog for the nets requiring them will also be created using the new (or matched) Track and Via Styles.

Attribute:	<net nan<="" th=""><th>ne> ~</th><th>F</th><th>or Nets of T</th><th>ype: <any></any></th><th>~</th><th></th><th></th><th></th><th></th></net>	ne> ~	F	or Nets of T	ype: <any></any>	~				
Match:	GND*	~	*	Within A	reas:		\sim			
Define [For Tr On §	acks:	<any></any>		~	Define Via De For Vias with <u>I</u>	ayer Span:	<any></any>			~
	<u>-</u> ayer: ılt Track St	yle:		~			☑ Delete if not	Routed	Reduce S	pan
N <u>a</u> m <u>W</u> idt		nal(5.5)		~	Name: Width:	Rnd44.5h35		Round	~	
Alterna Na <u>m</u> Widt		Style: nal(9.9)		~	Length:	44.5	<u>o</u> nape. Drill:	35.3		
· ·		.ength: <default></default>			Plated					

Options Dialog Changes

Option for Spacing Rule Grids

A new option, **Spacing Rules - Enable linking of types** has been added to **Design Settings**, **General**. When this is checked, some rows and columns on the Spacing grids will disappear (provided the spacings are the same), and the values will be linked together. This setting will be saved with your design so it is retained.

All Text	Attributes				
Adjust To Readable Orientation	Substitution Char: (when % doubled)				
(when doubled)	Matching Styles				
System Font	By Name Only				
Proportional Width Digits	O By Name And Value				
CAM/Plot	O By Value Only				
Transform about design extents	Naming Rules				
STEP Model Alignment	Make named styles use naming rules				
Snap by:	Pad Exceptions				
 Centre 	Mirror with Component				
Origin	Net Optimise				
Use Adjustment Attributes	Pads/Vias connect by centre				
	Spacing Rules:				
	Enable linking of types				

You can show the hidden values by unchecking the appropriate options which appear under the spacing grid.

Design Mounting Track Pad SMD Pad Via Micro-via Testpoint Copper Text Hole Design Track 0.25 0.25 0.25 0.25 0.25 0.254 0.254 0.127 0.178 Pad 0.25 0.25 0.25 0.25 0.254 0.254 0.127 0.178 0.25 SMD Pad 0.25 0.25 0.25 0.25 0.25 0.254 0.254 0.127 0.178 Via 0.25 0.25 0.25 0.25 0.25 0.254 0.254 0.127 0.178 Micro-via 0.254 0.178 0.25 0.25 0.25 0.25 0.25 0.254 0.127 0.254 0.254 0.254 0.254 0.254 0.254 0.254 0.254 0.254 Testpoint 0.254 0.254 0.254 0.254 0.254 0.254 0.254 0.254 0.254 Mounting Hole Copper 0.127 0.127 0.127 0.127 0.127 0.254 0.254 0.178 0.178 Text 0.178 0.178 0.178 0 178 0.178 0.254 0.254 0.178 0.178 0.25 0.25 0.25 1.27 1.27 0.25 0.25 Board 0.25 0.25

Before linking, the Spacing grid looks like this:

With Enable linking of types checked, the Spacing grid now looks like this:

Design								
Design	Track	Pad	Via	Testpoint	Mounting Hole	Copper	Text	
Track	0.25	0.25	0.25	0.254	0.254	0.127	0.178	-
ad	0.25	0.25	0.25	0.254	0.254	0.127	0.178	
/ia	0.25	0.25	0.25	0.254	0.254	0.127	0.178	
estpoint	0.254	0.254	0.254	0.254	0.254	0.254	0.254	
Iounting Hole	0.254	0.254	0.254	0.254	0.254	0.254	0.254	
Copper	0.127	0.127	0.127	0.254	0.254	0.178	0.178	
ext	0.178	0.178	0.178	0.254	0.254	0.178	0.178	
Board	0.25	0.25	0.25	1.27	1.27	0.25	0.25	
			🗹 Link Pad -					

Thread Count Definition

A new option, **Thread Count** has been added to **Options, General**. The maximum value of the slider is automatically presented based on the number of logical processors your PC has.

Snow ∪esign browser Default to No Technology		Spacing Rules
Allegro Extract File Path	Browse	
Multi-Threading		
Thread Count		

If **Enable Threads** is checked, multiple threads will be used to perform task that utilise it, including but not exclusive to: CAM Plot Gerber Export, Copper pour, Design Rule Checks and Filling the Rules spreadsheet.

The Thread Count slider defines the number of threads used when performing the above tasks.

New Options page for Pan & Zoom functionality

The **Pan** and **Zoom** settings in **Options**, **Interaction** had been moved to a new page **Options**, **Pan & Zoom**. This page has been added to declutter the **Interaction** page and add new functionality (namely, **Auto Pan**, see below).

Mouse:	Middle Mouse:
Drag Sensitivity:	Rolling Wheel Does Zoom In/Out
Short Long Reset	Middle Button Does Pan
Zoom:	Pan:
Zoom Sensitivity: Reversed Mouse Zoom	Pan Sensitivity: 🔽 Reversed Mouse Pan
Low High Reset	Low High <u>R</u> eset
Zoom at Cursor also re-centres window	Auto Pan:
 Zoom at Cursor also re-centres window 	Delay:
	Speed:

New Auto-Pan feature

Auto-Pan has been added to Pulsonix. It has been added to the **Options**, **Pan & Zoom** page. This allows you to automatically scroll (pan) your design by moving the mouse to any edge of the design area (the target region). When the mouse gets to within 12.7mm (1/2 inch) from the edge of the design area, Auto-Pan is activated.

The **Delay** and **Speed** can be defined for this option.

Delay is the time it takes for the Auto-Pan option to 'react' when the cursor is moved into the target region.

Speed is the speed of the pan once Auto-Pan is activated. If the slider is to the left most side, the speed will be slow, to the right side, it will be much faster.

The **Reset** button is used to reset the Delay and Speed sliders back to their default factory settings (these have been predefined to be good general working settings).

Middle Mouse:
Rolling Wheel Does Zoom In/Out
Middle Button Does Pan
Pan:
Pan Sensitivity: Reversed Mouse Pan
Low High <u>R</u> eset
- 🗸 Auto Pan:
Delay: Reset
Speed:

Probe Synchronised Design option added to Cross-Probe

A new option, **Probe Synchronised Design** has been added to the **Options** dialog, **Interaction** page under **Cross Probe**.

Сору	Cross Probe
Copy Single Cutout As Shape	Bring Probed Design To Front
	Probe Any Design
Rotate	Open Schematic Page
Rotation Step: 45.0	Select Using Find Options
Power & Ground Pins	Probe Synchronised Design
Auto Connect: Always 🗸 🗸	
Undo	Cursor Text
- I lada Daa (7aaa)	04-++ from 0 250.0

Selecting this option enables cross probing to take into account synchronised designs which are assigned in the **Design Settings Synchronisation** dialog. When selected, cross probe will check if the nominated synchronised document exists, and if so, selects the probed component in that design. If the synchronised file is not found or this option is disabled, then cross probing will run as usual.

Technology [] – Design Settings – S	ynchronisation
	PCB in Safe Mode Allow PCB Only single pin nets Synchronised Design Name Name: Design 3345
	Back Annotation

Removal of Fast Locate Option

The **Fast Locate** option on the **Options** dialog and **Display** page has been removed (shown below as the empty red box). This setting was used to enable and disable the enhanced picking but is no longer required. It will always be set on now by default but unavailable for changing.

Options Display		— 🗆	×
Design Backups Display Edit Shape Edit Track File Extensions Find General In-Place Names Interaction Macros	Fast Redraw ✓ Schematic ✓ PCB ✓ Draw in Layer Order ✓ Draw Current Layer On <u>T</u> op ✓ Dim Other Layers ✓ All Dim in One Colour	Hardware Acceleration Low High Preset View All' On Opening Designs Draw Dynamic Text Origin Draw Grids Underneath Items	

Edit Track Option for using Layer Bias

A new option has been added to Options, Edit Track called Next Layer - Use Bias

Options Edit Track		
Design Backups Display	Segment Mode: 45 Angled	ΥĽ
- Edit Shape	Interactive Mitre/Fillet	——————————————————————————————————————
Edit Track File Extensions	Size: 1.27 Any Angle	
Find	Show 'Can Finish' and 'Has Loop' Markers	Remove
General		
- In-Place Names	Show Legal Completion Path	🗹 Auto Sha
- Interaction	Always Mark Net being edited	Allow Net
Macros Move	Clear Mark when edit complete	End On \
Multi-Screen	Show Connection to Nearest Node	Use Style
Pan & Zoom	Show Dynamic Connections Only	Obstacle
Online DRC	Optimise After Edit	Auto Finis
···· Resolve Net Names ···· Select	Next Layer - Use Bias	└ Change I
Synchronication		D17 0 1 D

When editing a track, you now have the option to utilise the **Layer Bias** defined in your **Technology Layers**. This means when using the **Next Layer** and **Previous Layer** commands, it will skip layers with a bias of **Power Plane**, **No Tracks** or **Minimum Tracks**.

With this option left unchecked, the Next/Previous Layer commands will use all electrical layers of any bias.

Optimise On Clear Template Option

On the **Options**, **Interaction** dialog, **Optimise On Clear** has been split out from the **Optimise On Delete** option to make it clearer that an optimise will be performed when you clear a template.

There is a new **Optimise On Clear** check box under the **Templates** section to enable/disable this.

In-Place Names	Auto Connect: Always 🗸 🗸	Onac
Macros Move Multi-Screen	Undo	Join Ope Jc
Pan & Zoom Online DRC Resolve Net Names Select	Delete ☐ Unextended Delete - Deletes Segments ☑ Optimise After <u>D</u> elete	Au
Synchronisation Tooltips Track Length Limits Warnings	Optimise Only Signal Nets Templates When Adding Templates	Different While
	Act Poured Deour On Add	Route S
	Hide Template When Poured	
	✓ Highlight Isolated Copper	
	Clear On Edit	
	Repour Affected Templates Optimise On Clear	

Clear All Templates – Optimise now Multi-thread

Multiple threads can now be use when optimising nets after clearing all templates. This significantly speeds up the processing time for designs with large nets.

The selection to **Enable Threads** for **Copper Pour** has been added to the **Options** dialog, **General** page. This setting is used by all Multi-threading technology in Pulsonix.

Optimise Settings for Large Nets

To reduce the impact of large nets on how **Optimise** runs, there is a new **Large Net** setting on the **Move** page of the **Options** dialog. This is additional to the setting that controls how optimise works on placing components (Continuous, Never, On Drop etc.).

This new option instructs Pulsonix to skip nets that have more than the specified number of nodes because they are 'expensive' to optimise. The main **Optimise Nets** command will still optimise all nets regardless of complexity, this change will improve responsiveness during interactive editing for large nets.

Note: a node is any connection point - pad, via, star point, test point, junction or copper.

Auto Weld	Component Pushing
☐ Auto Weld On Drop ✓ Allow Weld To Split Tracks ✓ Show Weld Spots	Push Mode: Never Push Direction: Both
Snap To Construction Lines Within Set Distance: 15.000 Within Grid Steps: 2	Show Force Vector
Optimise: Continuous Full Optimisation On Drop Large Net: 40 nodes	Dynamic Align Screen Snap Tolerance: 10.000 Align With: Detents Original Position Pads Vias
Show Placement Origin	Auto Rotate Bond Pads

Large Net Warning on Optimise All Nets

An additional **Optimising large net** warning has been added when optimising all nets. The warning is shown at the very beginning when **Optimise** is run. It lists all large nets that will be optimised. This gives you the option to not optimise these large nets.

Warnings		×
Large net(s) +5V, GND, \$5, \$27, \$33, \$36 are about to be optimised. Press OK to optimise these nets.	^	ОК
		Cancel
		Report
		Warnings <u>O</u> n/Off
	~	Do not tell me again
		//

A large net is identified in **Options**, Move under the Large Net setting (see above).

Optimise All Nets Multi-Threading

The ability to utilise multi-threading when **Optimising All Nets** has been added. This change speeds up the overall processing time of this function.

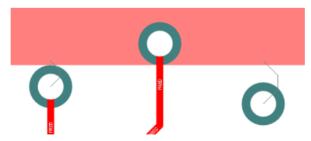
The selection to **Enable Threads** for use during **Optimise All** has been added to the **Options** dialog, **General** page. This setting is used by all Multi-threading technology in Pulsonix.

Net Optimise for Pad Centres within a Shape

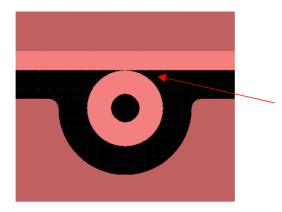
There is a new option in **Design Settings**, **General**, called **Net Optimise**. This forces a connection (and therefore it will not be properly connected) unless the centre of a pad or via is within the shape of the other item. This prevents pads which just touch the edge of a shape from being considered connected and thus avoid thin bridges which may or may not be connected after manufacture. This setting is saved with the design so that it isn't lost.

All Text	Attributes
Adjust To Readable Orientation	Substitution Char: (when % doubled)
(when doubled)	Matching Styles
System Font	By Name Only
Proportional Width Digits	O By Name And Value
CAM/Plot	O By Value Only
Transform about design extents	Naming Rules
STEP Model Alignment	Make named styles use naming rules
Snap by:	Pad Exceptions
 Centre 	Mirror with Component
Origin	Net Optimize
	Net Optimise
Use Adjustment Attributes	Pads/Vias connect by centre
	Spacing Rules:
	Enable linking of types

In the example below, three clear scenarios are shown where this option is enabled and **Optimise** has been run; one where the pad is touching and a connection exists, the middle one where the pad is completely enclosed and no connection is required, and the third pad where it is clear of the shape and shows a connection.



When **DRC** is run, this forms part of the **Net Connectivity** check (which checks for unrouted connections).



The pad touching here is considered a 'thin bridge' which may cause manufacturing issues.

Dynamic Align of items

There is now an option to dynamically align items being moved to static items in a Schematic or PCB design. From the **Options** dialog, **Move** page, there are options to set for refinement.

This option is compatible with; Components, Symbols, Pads, Vias and Text.

Optimise:	Ivever	\sim	
	Full Optimisation On Drop		
	c Align Snap Tolerance: 10 h: Ø Extents Ø Original I Ø Pads Ø Vias	Position	 ✓ Auto Rota ✓ Auto Rota ✓ Show Dya ✓ Always M

When an item is moved, with this option enabled, 'alignment' lines are displayed:



Screen Snap Tolerance – This is the value for the tolerance at which an item will snap to another item during move. The tolerance will remain the same screen distance regardless of the current zoom level. This value will be set in the current design units.

Align With

Extents - Any item will snap to another item's extents during move. Extents is the total circumference of a shape.

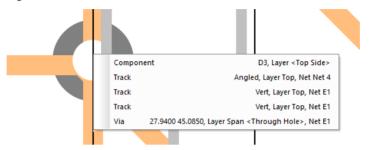
Original Position - Any item will snap to its Original Position during move.

Pads - Pads, Components/Symbols and Vias will snap to another item's Pad during move.

Vias - Pads, Components/Symbols and Vias will snap to Vias during move.

Select from popup list

Holding down the **Alt** key and **selecting overlapping items**, a context menu will list all items at that position, allowing you to select a specific item based on its description rather than the item. This is an alternative to selecting the first item, then pressing the $\langle N \rangle$ key to get the new selected item within the picking tolerance.



This feature can be disabled in the Select section of the Options dialog and Select page.

Using this option within a mode

As well as using this popup a select mode within the design, you can also use it within other modes to obtain the correct item for selection. For example, use it within the **Insert Dimension** or **Measure** tools.

Part Editor - Dynamic Attributes for Part Description field

For a **Part** in the **Part Editor**, the **Part Description** field can be created using an Attribute substitution. For example, the Part Description may be made up like this: %% Value%% %% Tolerance%% %% Watts%%

	Part Name:	SR215A103KAR	
	Description:	%%Value%% %%Tolerance%% %%Watts%%	
_/	Part Family:	SR/SR215A	
	Name Stem:	C	FPGA
	Pin Count:	2 Change	
	Footprints:	SR21 ~	Choose
	Spice Type:		Edit Spice

Once the Component is added to the **design** and the **Attribute Values** are satisfied, the Part **Description** field will be populated and viewed or reported using the relevant options, for example, **Report Maker**, **Properties**, **Attribute Editor** etc. It is viewed as a populated state in the design and can only be edited from within the Part Editor.

📙 Edit A	Attributes						
Attributes	of: Compone	nts	 ×				
Name	Part Name	Description	Footprint		Tolerance	Value	Watts
Name R1	Part Name R			Category Generic/RE			Watts 0.125W

In Properties, the Part Description gets resolved like this to show the populated values:

Properties: Component: R2 Component		×	
Component Nets on Pins Comp Attributes Vault			
✓ Name: 12 ★ □ Locked			
Position: 109850.0 126275.0			
Angle: 0.0 <u>Mirrored</u> Scale: 1.000000			
Part: R		Change	
Description: 22K 1% 0.125W			
Part Family: Generic/RES		<u>A</u> ltemate.	
Eastaviate Specify Footprint			

Save Items to Library dialog changes

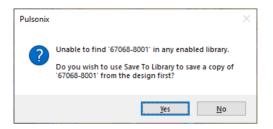
On the existing **Save Items to Library** dialog (available for a selected component in the design or by dropping a design onto the Library Manager), the list boxes have been changed to be **check list boxes**. This allows for you to specify which items should be added to the chosen libraries.

Additional Select All and Deselect All buttons have also been added to aid the selection of all items.

Save Selected Items	To Library						×
<u> </u>	User					\sim	ОК
PCB Footprint	user					\sim	Cancel
PCB Doc Symbol	[New Library]					\sim	
Parts 2		PCB Footprint	2	PCE	3 Doc Symbol	0	
☑ 555 ☑ D		ØBDIL DIOD04					

Save Missing Component Items to Library

From within a design (Schematic or PCB), when using the **Edit Part**, **Edit Footprint** or **Edit Symbol in Library** options, if the selected Component's Part definition, Footprint or Symbol does not currently exist in any library, you will have the option to use the **Save Items To Library** dialog to save it to a library first and then edit it once it's been saved.



The individual library item is displayed in the Save Items To Library dialog:

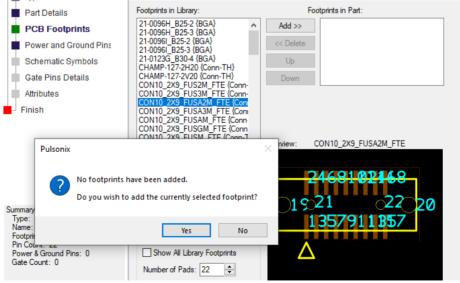
Save Selected Items	To Library					×
✓ Parts	User				~	ОК
PCB Footprint	user				~	Cancel
PCB Doc Symbol	[New Library]				\sim	
Parts 1		PCB Footprint	1	PCB Doc Symbol	0	
67068-8001		67068-8001				

Part Wizard changes

Add Selected Footprint/Symbol

With the **New Part Wizard**, when clicking **Next'** on the **PCB Footprints** or **Schematic Symbols** page, if no Footprints/Gates have been added but one is selected in the **Footprints/Symbols in Library** list, a message is displayed giving you the option to add the one currently selected.

The message is also displayed on the symbol page if you haven't added enough gates, allowing you to add the selected one.



Pin Type cell new option to Apply to Undefined

On the **Power and Ground Pins** and **Gate Pins Details** pages, once details have been defined, you can now right click on the **Pin Type** cell and use the context menu to apply that pin type to all undefined pin types using the **Apply To All Undefined** option.

Pi	in details for each gate						
G	Gate a - BLOCK2 Gate Swap Group: 0					0	
	Symbol Pin	Pin Name	Logic Name	Pin	Туре	Pin Swa	p
	1 1 Bi-Directional In						
	2	2		<un< td=""><th>Apply To</th><td>All Undefi</td><td>ned</td></un<>	Apply To	All Undefi	ned
		,					

Vault Library Export to Specific Folder

There is an enhanced to allow a specific **Vault folder** to be specified as an alternative to always exporting all library data.

Use **Export From: Vault Folder** to specify a particular Vault folder from which to export library data, or leave it blank to export ALL library data contained in the Vault.

The **Browse** button will display a dialog to allow you to browse the Vault folders to find the appropriate one.

Check **Include Sub Folders** if you wish library data to be exported from the selected folder and all of the sub folders it contains.

Export From:	
Vault Folder:	Browse
Include Sub Folders	
Export Now	

The VaultExport command is available for a Pulsonix Command File.

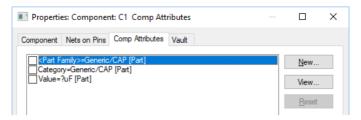
Optional parameters allow for the override of the options specified from within Pulsonix. By running Pulsonix from the command line with a Pulsonix Command File as a parameter, the Vault Library Export facility can be utilised from the Windows Scheduler to create a regular reoccurring task. The full recommended command line is shown below:

Pulsonix.exe -hidden -commandfile <path to command file>

This feature was back-fitted to 10.5

Attribute Properties – View Name and Value for Protected status

A change to the **Properties** dialog and **Attributes** page now means you can select a protected Attribute and Value and use the **View** button to view it (instead of Editing it). Previously, it wasn't possible to view it because it was protected and not 'editable'. The **Edit** button changes state between **Edit** and **View** depending on the attribute status.

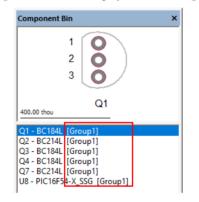


The feature enables you to copy and paste the Attribute name or Value using standard Windows commands from the dialog presented.

View At	tribute	×
<u>N</u> ame:	<part family=""></part>	
<u>V</u> alue:	Generic/CAP	

Group Names Displayed in Component Bin

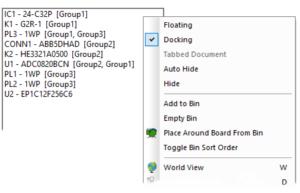
Group names are now displayed in the Component Bin.



Toggle Bin Sort Order in Component Bin

When using the **Component Bin**, a new option, **Toggle Bin Sort Order**, has been added to the context menu. The current sort order uses Component Name, this new toggle allows you to also sort by group name. By default, the sort order will use Component Name.

When sorting by group name, if an item is in multiple groups then its first group will be used. If multiple items are in the same group, they will be sorted by their Component Names. All items that are not in a group will be sorted by their Component Names and will appear after all the items in groups.



Double-Click selection in Selection Mask Bar

When in the **Selection Masks bar**, you can now double-click on a mask category check box which will deselect all currently selected categories and then only select the category on which you have chosen.

Selection Masks	•	-Þ X
	Save	New
Imable Select Mask		
<default></default>		\sim
Area Attribute Bitmap Board Board Component Component Connection Construction Line Copper Dimension Doc Shape Doc Symbol Embedded View Croor		
Mounting Hole		

In the above example where three different categories are selected, if say, Board is double-clicked, only Board will be selected and all other categories deselected.

Override Readable Orientation of Text

The **Ignore Readable Orientation** check box has been added to **Text Properties** and **Attribute Properties** dialogs. Selecting this switch disables the effect of the **Adjust To Readable Orientation** switch (from **Design Settings** – **General** dialog) for that specific text.

ropente	s: Compone	ent Attribute	– Attribute					Х
Attribute	Text Style	Component	Nets on Pins	Comp Attributes	Variants	Vault		
Name:	<component< td=""><td>t Name></td><td></td><td></td><td></td><td></td><td></td><td></td></component<>	t Name>						
Usage:	Part							
<u>V</u> alue:	C14							 •
<u>P</u> osition <u>A</u> ngle:	: 43.091 0.0		80 <u>L</u> oc rored	ked				
-	0.0			ked			~	

Split Net Highlights

There is a new check box on the **Split Net** dialog, **Keep Items Highlighted**. This means that when a net is split, if a resistor component is dropped on to the net for example, then no net highlight will take place. The net highlight is used to indicate the renamed portion of the net. This mode is off by default and can be selected to on for use in the option if and when required.

Split Net "\$5990"	×			
The net has been split into two nets and the highlighted net will take the new name supplied below. The unhighlighted net will remain as the original net.				
New Net Name: CLOCK				
O Remove From Net				
Rename <u>O</u> ther Half				
<u>Keep Items Highlighted</u>				
OK Cancel				

Shape Information/Editing Bar

A new Shape Information Bar has been added to Pulsonix. This appears as a dockable window accessible from the **View** menu and displays detailed information about the currently selected shape in a grid format that also updates on the fly. It is available to use when editing PCB and Schematic designs, symbols and technology files. The text in the grid describes the shape details that may be edited allowing precise modifications to be made.

Shape Information X					
-Mai Points V Rel Auto V					
Item: Board					
Layer: <through bo<="" th=""><th>ard></th><th></th><th></th></through>	ard>				
Locked Closed	Filled				
Туре	X	Y	^		
Start Point	63.2860	38.8620			
Clockwise To	65.7860	41.3620			
With Centre At	65.7860	38.8620			
Clockwise To	68.2860	38.8620			
With Centre At	65.7860	38.8620			
Line To	131.5720	38.8620			
Anti Clockwise To	133.0960	40.3860			
With Centre At	131.5720	40.3860			
Line To	133.0960	53.0860			
Line To	132.5880	53.5940			
Line To	127.2540	53.5940			
	127.25/0				

Most check boxes relate to existing Properties and are self-explanatory.

The **Auto** button will cause the Shape Bar to automatically switch to the most appropriate mode to display a selected shape, i.e. if a circle is selected it will be displayed in **Circle** mode. Polygon shapes will be shown in the most recently used of the possible modes.

Shapes drop down list

The shape information can be displayed in various formats dependent on the current mode selected from the dropdown list. When a different mode is selected the grid columns will reconfigure to display the chosen format. In all modes, shape points are shown as pairs of coordinates in adjacent X and Y columns in the grid. The various modes are described below:

Points

The shape is shown as an initial Start Point and a series of Lines or Clockwise/Anti-clockwise arcs to given points. For arc segments, the arc centre point is shown on the next row. The points will be shown in absolute or relative coordinates depending on the **Rel** setting.

Point Offsets

The shape is shown as an initial Start Point and a series of Lines or Clockwise/Anti-clockwise arcs to given points expressed as offsets from the previous point. For arc segments the next row shows the centre point for the arc as an offset from the previous point. Only the Start Point will be shown in absolute or relative coordinates depending on the **Rel** setting.

Segments

The shape is shown as a series of segments starting from the given point with the given length and arc angle. The end point of the segment is given by the start point of the next segment. For the final segment of an open shape there is an additional row showing its end point. The direction of arcs is given by the sign of the angle with a negative value indicating an anti-clockwise arc. The points will be shown in absolute or relative coordinates depending on the **Rel** setting.

Segment Offsets

The shape is shown as a series of segments, similar to Segment mode, but with their start points expressed as offsets from the previous point. Only the start point of the initial segment will be shown in absolute or relative coordinates depending on the **Rel** setting.

Circle

If the selected shape is a circle it is shown as a Centre Point and a Radius. For any other type of shape, the display will be blank therefore this mode is best used with the **Auto** option enabled.

Rectangle

If the selected shape is a rectangle it is shown as a Start Point and a Size; X = width and Y = height. For any other type of shape, the display will be blank therefore this mode is best used with the **Auto** option enabled.

Note, only orthogonal rectangles are recognised by the Shape Information Bar.

Context Menu Options

When a shape is selected, you can **right click** on a row in the grid of the shape bar and click **Find Segment**.

Shape Information			×
-µ Points ∨	Rel 🗌 Auto 🗸		
Item: Board			
Layer: <through boa<="" td=""><td>ırd></td><td></td><td></td></through>	ırd>		
Locked Closed	Filled		
Туре	X	Y	^
Start Point	63.2860	38.8620	
Clockwise To	65.7°en1	#1 2620	
With Centre At	65.7	Find Segme	ent
Clockwise To	68.2	Select Row	
With Centre At	65.7		
Line To	131.5	Insert Row	
Anti Clockwise To	133.0	Delete Row	
With Centre At	131.57zu	40.3860	
Line To	133.0960	53.0860	

This feature shares the **Find Bar** options (**Options, Find, Action On Found Item**) for **Highlight** and **Flashing** and will find the chosen segment in the design. If you have the **Highlight it** and **Flash it** boxes checked, then the chosen segment will be found, highlighted and/or flashed.

Find a segment using the grid

Right click the mouse over the row in the grid you with to find in the design. From the context menu select the **Find Segment** item. The segment will be located by altering the view to include it and then either selecting it, highlighting it, brighten it (dim all other items) or flashing it. The find action depends on the current **Find Options**. Find Segment is not available when using Circle or Rectangle mode.

Insert a new segment into a shape

Right click the mouse over the grid at the point you wish to insert a new segment. From the context menu select the **Insert Row** item. A new segment will be insert in to the shape at that point. The exact position of the inserted segment will depend on the current Shape Bar mode; in a Segments mode, the new segment is inserted from the centre of the existing segment to its end point. In a Points mode, the new segment is inserted after the end of the existing segment to the centre of the next segment (if there is one). In either case, the end point is gridded using a step size relative to the **Working Grid**. Insert Row is not available when using Circle or Rectangle mode.

Delete a segment from a shape

Right click the mouse over the grid row containing the segment you wish to delete. From the context menu select the **Delete Row** item. The way the segment is removed depends on the current Shape Bar mode; in a Segments mode, the corner at the start of the segment is removed. In a Points mode, the corner at the end of the segment is removed. It is not possible to delete segments from a shape if doing so would cause it to become invalid. Delete Row is not available when using Circle or Rectangle mode.

Scale Option

The Scale feature has been updated to allow for scaling on-the-fly of Shapes, Text, Schematic Gates and Bitmaps.

An item is selected in the design and the Scale option selected from the context menu.

Scale	×
Scale By: 1.000000	OK
	Cancel

When scaling text, the ability to cycle through styles of the same width is available on the context menu using the **Next Style** option.

The Scale option can also be used as a 'mode'. Select the **Scale** option first from the **Utilities** menu, there a number of modes you can use:

	Cancel Scale	
B	Exit This Mode	
	Finish Here	
₽	Scale	
	Type Offset	Shift+=

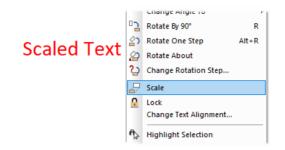
You can select an item and dynamically rescale it by moving the cursor. The actual scale is shown on the **Status** bar as the scale is changed.

Scale R1 Scale: 1.734312	rid: Working 25.0	Abs 116891.9+	125503.0-	thou	:

You can manually type in a Scale or Offset as a 'one shot' function to rescale an item.

Scaling Text, Layer Stack Previews, Drill Tables and Inserted Reports

Scaling text and non-symbol design items works slightly different in that when you scale these, it will run through each text style and next the next biggest one available. This is activated by selecting text and choosing Scale from the right-hand menu.



For example, each time the mouse is moved away from the text, the text is changed to the next size:



Change Part - Same Footprint check box

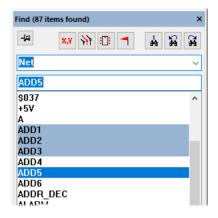
From the Change Part dialog in the filter section, you can now select the **Same Footprint** check box. This will allow you to filter the Parts to only contain ones with the same footprint as the current Part. If you are changing the Part of a Variant, then warnings will be displayed if the footprint or pin mapping of the newly chosen Part are different to the current part.

Change Part		×
Look In: Generic Which Parts: Eilter: Matched: 27 of 410	No. Pins: 2 Same Footprint	OK Cancel

Find Bar Changes

Multi-Select in Find Bar

When you select several items in the **Find Bar** list using the **Ctrl key** they are all selected, but now they are **kept highlighted** in the list so you can see all the selections not just the current one.



Enhanced Selection

When using the **Find** bar, some list entries are used by more than one item in the design, for example **Group** and **Style** names. In this case, on selection in the Find bar, only the first item matching the search criteria will be found.

You can now hold the **Shift** key down when selecting these list entries to find all items matching the search criteria, not just the first one, for example, all items in the selected group.

Using the **Ctrl** and **Shift** keys together you can find multiple whole items, for example, multiple **Groups**.

This feature was back-fitted to 10.5

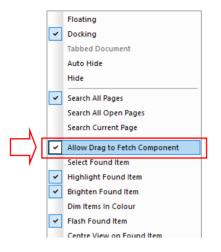
Fetch Component from Find Bar

Within the **Options** dialog and **Find** page, there is a new find option is available, **Finding Components – Allow Drag to Fetch Component**. This is set on by default. This allows you to drag a component from the find bar to "Fetch" it from elsewhere in the design, or from the bin.

In a Schematic design you will only be able to fetch a component that is on the current page.

Action On Found Item	Finding Nets ☑ Include Default Nets		
 ✓ Highlight It ✓ Brighten It (Dim all other items) ✓ Rash It Centre View On It 	Finding Error Markers Include Locked Errors Only List Errors In The Design		
Allow Find Under Sliding Bar	Finding Drill Size	/ \	
Flashing Items Flash For: 1 seconds	Finding Components	t	
Schematics Search	Favourites List		

When displaying the **Component** category in the **Find Bar**, the **Allow Drag to Fetch Component** option is available on the context menu.



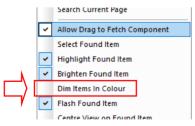
Dimming All Non-found Items in Colour

A check box has been added to the **Options** dialog and **Find** page to **Dim items in colour**. This will allow the Find bar to dim all the non-found items in colour, rather than all in grey.

A slider has also been added to allow the user to dim by an inputted amount.

	MIOW Drag to Fetch Component	
Schematics Search	Favourites List Maximum Items In List: 40	
O Current Page	Dim items in colour	
Variants ☑ Search In All Variants	Dim Bright Reset	

The **Dim items in colour** option is also available on the context menu on the Find bar, use it to toggle this mode.



Search for Non-Connecting Copper

You can now search for non-connecting copper in the PCB editor using the **<Non Connecting Copper>** attribute filter in the **Find Bar**. By setting the search to **Is: Exists**, all Non-connecting Copper by net name will be listed.

<no net=""> E1</no>	
Hide Filter << Apply	
Attribute:	
Layer: Top Electrical	

Close Other Pages (SCM)

A new command to **Close All Pages** has been added to the **File** menu and to the **Workbook** tab as a new command by right clicking on the tab. When applied, this works on all currently open Schematic pages except the current one.

🖬 Announcer 📴 Annou	incer.	Page1 * 📴 Appouncer	Page2 *
		Save Ctrl+S	Grid: Work
	1	Close	ond. tron
	₽ ₽	Close Page	
		Close Other Pages	

Allow Implied Junctions Pin Override Switch (SCM)

For a selected Pin in a Schematic design, on the **Pin Properties** dialog, there is a new switch for toggling **Allow Implied Junctions**. This enables you to set local Implied Junctions on individual pins and overrides the main **Allow Implied Junctions** setting (set in **Design Settings**, **General** page).

Proj	perties: Pin: U7	.2 Pin						\times
Pin	Pin Attributes	Component	Nets on Pins	Comp Attributes	Net	Net Attributes		
□ <u>N</u> a	ame: 2		Logic Name					
	sical Details erride				A	Now Interactive	Repositio	n
	Position:	10575.0	7900.0		🗹 A	Now Implied Jun	ctions	
Г	Andler	0.0						

Copy Net Names on Sig Refs with Ctrl-Drag (SCM)

A change has been made when using **Ctrl-drag** (to duplicate) a **Signal Reference** where the symbol has a user-defined Net Name. Using Ctrl-drag now copies the **Net Name** also. If you use Ctrl-drag on a Sig Ref symbol that has a default system Net Name allocated, only the symbol will be copied.

By default, this mode will be on. If you wish to disable it, use the **Options** dialog and **Select** page and deselect **Control Drag Sig Ref to Copy Net Name** under **Drag** options.

	Select	Frame Select
	Select Tight Groups	Select If Completely Framed
	Minimum Pick Tolerance	Alt Drag Does Frame Select
	☑ Enable Nudge	Select Error Markers
	Do Not Pick Locked Items	Select Construction Lines
	Select Whole Components	
	Exceptions: Select Component Attributes	
	Select Component Pads	
	Select From Popup List	
	Drag	Double Click
	☑ Drag Along Shape Selects Path Between 2 Points	Suppress Properties On Double Click
	Drag Unconnected Pin Starts New Connection	Double Click on Attribute to Edit Value
	Drag From Pin Uses Sketch Mode	Double Click On Pin Uses Sketch Mode
_/	Control Drag Does Duplicate	
_)	Control Drag Sig Ref To Copy Net Name	
-		

You can **override** this option locally by using **Ctrl-Shift-drag**. If the option is enabled then ctrl-shift-dragging will not copy the net name, and if it is not enabled control shift dragging will copy the net name.

This feature was back-fitted to 10.5

Insert Attribute Position (SCM)

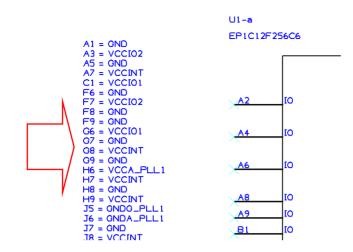
For a Component in a Schematic, there is a new **Attribute Position** attribute available called **<Ungated Pin Nets>**. This is available for a selected Component in the design using the **Insert Attribute Position** option.

When used, this will resolve to a list of Pin = Net for all nets assigned to ungated pins, thus allowing you to easily display the connectivity on the component that would otherwise be 'hidden' because the pins are not present on the symbol.

Insert Gate Attribute Position				
Attribute <u>Name:</u> <ungated nets="" pin=""></ungated>	~			
<u>U</u> sage: Part				
Z Hea Defaulte				

Previously, the only way to see ungated pin nets on the drawing was to insert a user report such as the provided **Ungated Pins Report** that makes a table of Parts, Components and their ungated pin nets that you can place on the drawing. With this new attribute value, you can display a simple list of those pins and nets right alongside the Component itself.

Inserted into the design it looks like this:



New Colour Category for Bus & Connections (SCM)

A new colour category for **Bus & Connections** has been added to **Schematic Colours** dialog. This brings together colours from **Others** and **Shapes**.

⊕ Spacing Rules ⊕ Styles	Name	Displayed	Selectable	True Width	Colour
Nets	Bus	Z	\leq	\checkmark	
Rules - DFM/DFT	Connections		\square		
	Connect Guides	\square	\square		
Rules - High Speed					
- Outputs					
- Naming					
- Colours					
Attributes					
-Bus & Connections					
Connect Points					
Differential Pairs					

No Connect Pin Highlight Colour (SCM)

There is a new setting in the Colours dialog and Highlights page for No Connect Pin.

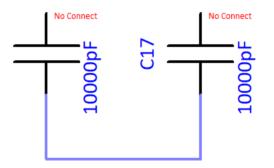
· Spacing Rules · Styles	Name	Displayed	Colour
Nets	Attached Callouts		
	Branch Point		
Rules - DFM/DFT	Differential Pair Path		
Rules - High Speed	Highlight		
Outputs	Highlight 'Fail'		
Naming	Highlight 'Pass'		
Colours	Highlight 'Unchecked'		
Attributes	Highlight 'Warning'		
Bus & Connections	Locked Connection Segments		
Connect Points	Marked Net		
	Marked Net Pins		
Differential Pairs	No Connect Pin		
Highlights	Not Fitted		
Nets	Selection		

When enabled from the **Pin Properties** dialog, this causes the name of the pin type to be substituted into the Net Name attribute for No Connect Pins. Once enabled, a Net Name attribute position will be added to a pin if you make it No Connect and it will be drawn in the highlight colour.

This will also work on Component Pins and Block Ports.

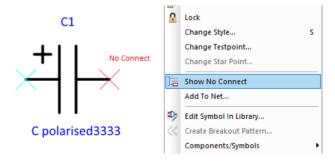
	Properties: Pin: C18.2 Pin	- D X
	Pin Pin Attributes Component Nets on Pins	Comp Attributes Net Net Attributes
	Name: 2 Logic Name:	
	Physical Details Override	Allow Interactive Reposition
	Position: 13700.0 11425.0	Allow Implied Junctions
K	Angle: 90.0	_
	Pin Type: No Connect	✓
/	Pin Style	

Displayed like this:



No Connect Pin context menu option to Show No Connect

Within the **Colours** dialog, **Highlights** and **No Connect Pin** enabled, selection of a pin with a type of **No Connect** and right clicking, shows you the option **Show No Connect**. This replaces **Show Net Name** and will add a **Net Attribute Position** with the text **No Connect** value.



Display Pin Type Attribute (SCM)

You can now display (in the design) the **Pin Type** on a pin using the **<Pin Type> attribute**. In either the symbol within the Symbol Editor or on the Component Pin in the design, use **Attribute Position** for a **selected pin** and choose **<Pin Type>**.

In the symbol, it will show as <Undefined> and in the design will be populated with the Pin Type if defined (locally in the design or in the Part definition).

Insert Pin Attribute Position				
Attribute <u>N</u> ame: <pin type=""></pin>	~			
<u>U</u> sage: Pad				
V Hea Dafaulte				

Changes to Insert Signal Reference dialog (SCM)

The **Insert Signal Reference** dialog has been changed to now include all the Sig Ref symbol types available. This means the **Insert Bus Reference** and **Insert Page Link** options are no longer on the **Insert** menu. They can now both be added using the **Insert Signal Reference** option, (the new **Insert Net Label** feature will also be included in this dialog, see below).

Insert Signal Reference	×
Look In: [All Libraries]	Add
Which Symbols:	Cancel
Eilter: * No. Pins: 1 Apply	Lancel
Туре: 📶 🗸	
Matche All	
Matche All Bus Reference Symbol: Symbol: VSS (Page Link Signal Reference Signal Reference	

On use, you can either scroll through all the names and decide which symbol you require or you can use the **Type:** drop down list to refine the selection. For Type, you now have **All**, **Bus Reference**, **Net Label**, **Page Link** and **Signal Reference** available.

New Signal Reference Type - Net labels (SCM)

New Doc Symbol Type – Net Label

A new symbol type called **Net Labels** has been added. These are used to 'hang' on a net to explicitly show the net name on your Schematic design. Because it uses a special doc symbol, it means you can customise how it looks and where it is attached to. The symbol's net name is dynamic and will update as the name changes. On creation, the symbol will contain a pin and an attached callout with the net name attribute stored in the callout.

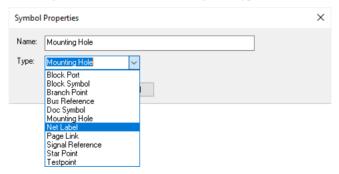
These can be created using the **New Doc Symbol** option from the **File** menu and **New** option by choosing the **Net Label** option or from the **Library Manager** and **SCM Doc Sym** page.

	New Doc Symbol X			
	Symbol Type			
	O Block Port			
	O Block Symbol			
	O <u>B</u> ranch Point			
	◯ B <u>u</u> s Reference			
Documentation Symbol				
	○ Mounting Hole ○ Net Label			
	○ Page <u>L</u> ink			
	◯ Signal <u>R</u> eference			
	○ <u>S</u> tar Point			
	○ <u>I</u> estpoint			
	OK Cancel			
	FRED_CLOCK			

Once you've created the new symbol, use

Changing Existing Doc Symbols (Signal References) to Net Labels

To change an existing doc symbol to a **Net Label**, edit the Symbol in library and use the **Symbol Properties** dialog from the **Edit** menu and change the Type to Net Label.



This can also be done when using **Save To documentation Symbol Library**. The symbol will then need to be reloaded into the design.

Save To Documentation Symbol Library				
Library:	user	~		
Name:	Mounting Hole			
Туре:	Mounting Hole Mounting Hole Block Port Block Symbol Branch Point Bus Reference Doc Symbol Mounting Hole Net Label Page Link Signal Reference Star Point Testpoint	Cancel		

If you don't do this, the **Next/Previous Symbol** option (see below) will still operate but will work on the current type. It may mean that it will spin through types that aren't relevant. Changing and reloading will make the Next/Previous symbol option more relevant and focused.

New Insert Net Label command

There is also a new **Insert Net Label** command available if required that can be added to your toolbar using the **Customise** dialog.

Inserting Net Labels into your design

You can add net labels by using the **Insert Sig Ref** and selecting the **Type** as **All** or **Net Label**. Net Labels are used to 'hang' on a net to explicitly show the net name. Because it uses a special doc symbol, it means you can customise how it looks and where it is attached to. The symbol's net name is dynamic and will update as the name changes.

The **Type** drop down box on the **Insert Sig Ref** dialog enables the Sig Ref choice to be filtered. This will filter on **All**, **Bus Reference**, **Net label**, **Page Link**, and **Signal Reference** Doc Symbols.

Insert Signal Reference	×
Look In: All Libraries	Add Cancel

In all other **Insert Doc Symbol** dialogs, the **Type** drop down box will be read only and will display the type of the symbol you are inserting.

Once on the end of your cursor, drop the Net Label onto the net requiring it.

FRED_CLOCK	1

Insert Connection - Start/End on a Net Label

When inserting or editing a **Schematic Connection**, you can choose to **Start/End Connection On** a **Net Label** using the option from the context menu.

	Cancel Insert Connection	on		
	Finish Here			
¥	Type Coordinate	=		
dX dY	Type Offset	Shift+=		
	Change Style	s		
	Change Net	F2		
P	Mark Net	н		
	End Connection On	•		Connector Pin
~	Online ERC			Signal Reference
	Editing Options	•		Net Label
	Segment Mode	•		Testpoint
	Change Segments			Branch Point
	Show Connection to N	et		Page Link
	Change Grid	•	~	None

Next/Previous Doc Symbol Command

When a single **Signal Reference**, **Net Label**, **Bus Ref** or **Page Link symbol** is selected in the design, you can now select **Next Symbol** and **Previous Symbol** options from the context menu.

P	۲ÏĬ	Mark Net
		Remove From Net
E	ም	Reload From Library
è	ም	Save Items To Library
		Next Symbol
		Previous Symbol
×	Dy	Edit Symbol In Library
	Î	Move to Bin
	3	Create Breakout Pattern
Gnc		Disconnect Con End
		Add Page Link
	ħ.	Highlight Selection

This will replace the currently used symbol with the next/previous symbol from your SCM Doc Sym Library. This will use Symbols of the same type.

This means it will choose the next Net Label or next Signal Ref Symbol for example as relevant to the currently selected symbol type.

Current Library: [All Libraries]					
Contents Block PortAA {DocSymbols} BOX {DocSymbols} BRANCH POINT {DocSymbols} BUS PAGES BIDIRECTIONAL {DocSymbols} BUS PAGES BIDIRECTIONAL {DocSymbols} C-Border-Landscape {DocSymbols} Comparator Block {DocSymbols} D-Border-Landscape {DocSymbols} DANGER-STATIC-ELEC {DocSymbols} DIFF PAIR {DocSymbols} DIFF PAIR {DocSymbols} DRAFT {DocSymbols}	^	<u>N</u> ew <u>E</u> dit <u>D</u> elete <u>R</u> ename <u>C</u> opy			
DRAWING HÉADER {DocSymbols} EARTH {DocSymbols} FOR INFORMATION ONLY {DocSymbols} FROM {DocSymbols} FULL ADDER {DocSymbols}		<u>M</u> ove <u>I</u> mport Load Tech			
GND (JocSymbols) GND TRIANGLE (DocSymbols) GROUND (DocSymbols) HALF ADDER {DocSymbols} INPUT LINK {DocSymbols} INPUT PORT {DocSymbols} INPUT SIGNAL {DocSymbols} OUTPUT {DocSymbols} OUTPUT LINK {DocSymbols} OUTPUT LINK {DocSymbols} OUTPUT SIGNAL {DocSymbols} OUTPUT SIGNAL {DocSymbols} POINTER {DocSymbols} POINTER BOTH {DocSymbols} POINTER BOTH {DocSymbols}		To Symbol			

Mounting Hole Symbols in Schematic (SCM)

You can now add a Doc Symbol to a Schematic design to represent a Mounting Hole in the PCB.

When creating a **New SCM Doc Symbol**, the choice now includes **Mounting Hole**. When saving the new Doc Symbol, you can choose **Mounting Hole** as the **Type**:

New Doc Symbol	×	
Symbol Type Block <u>P</u> ort Block Symbol Branch Point Bus Reference Documentation Symbol <u>Mounting Hole</u> Net Label Page Link		Save To Documentation Symbol Library
O Signal <u>R</u> eference <u>S</u> tar Point Iestpoint OK Cancel		Library: DocSymbols Name: Mounting Hole Type: Mounting Hole OK Cancel

Adding The Mounting Hole Doc Symbol

A new option on the **Insert** menu, **Mounting Hole**, enables this new type of doc symbol to be added to a Schematic design. Choose the Mounting Hole required from a list of items from your Doc Symbol library.

Insert Doc Symbol	×
Look In: [All Libraries]	<u>A</u> dd Cancel
Symbol: Mounting Hole {DocSymbols} V Pins: 1	
90.0 thou	

These symbols can be added to an existing net to enable their net connectivity in your PCB design.

Mounting Hole Doc Symbols In The Design

Once added to your Schematic design, the Mounting Hole Doc Symbol can be changed using **Properties**.

Properties:	📧 Properties: Mounting Hole: – Mounting Hole – 🛛 🗙						
Mounting Hole	Mounting Hole Attributes	Pin	Pin Attributes	Vault	Net	Net Attri	outes
						Locked	
Position: 1	13050.0 126675.0						
<u>A</u> ngle: 0	.0 <u>M</u> irrored	<u>S</u> cale:	1.000000				_
Symbol: Mo	ounting Hole					Change	
Drawn on: T	his Page Only						~

Following successful **Translation to PCB**, Mounting Holes are added to the **PCB** using the **Mounting Holes** setting in the **Design Settings**, **Defaults** page. This setting will select the layer for the Mounting Hole and Pad Style used, the same as if using the **Insert Mounting Hole** option.

Copy Net Names for Signal References (SCM)

Ctrl-drag, used to duplicate a Signal Reference Symbol, now also copies its Net Name.

This is particularly useful for a **Net Label Sig Ref** for example, where a Net Name should also be duplicated. It will only duplicate Net Labels where a user defined Net Name has been used, it will not duplicate auto-generated Net Names.

To disable this feature for use on another net, simply copy the Sig Ref symbol but press **Ctrl-Shift-drag** instead.



This feature was back-fitted to 10.5

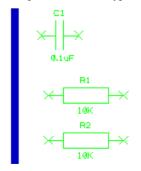
Text Formatting option for Spice Netlist Export (SCM)

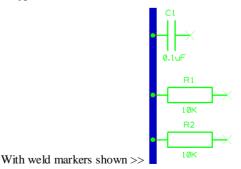
When exporting a Spice netlist through the **Simulation** menu, on the **Set Spice Netlist Type** page, you can now choose how the Spice netlist handles international characters using the ASCII, Unicode and UTF8 settings. A setting can be chosen using the selections on the dropdown list. By default, this will be set to ASCII.

Set Netlist Spice Ty	Set Netlist Spice Type X					
Spice Netlist Type:	LT spice 🗸					
Text Format:	ASCII ~					
Run Simulator	ASCII ASCII/UTF8 Unicode UTF8	Browse				
	ine cuitches:	D10436				

Weld multiple Components to a Bus segment (SCM)

The auto weld feature to drag the pins of a **Component** over a **Bus** segment has been extended to allow dragging of multiple Components over a Bus segment (like a set of resistors for example). This can be Components of a like type or dissimilar type.





This feature was back-fitted to 10.5.

New ERC Check - Nets Only On Ungated Pins (SCM)

A new option in **Electrical Rules Check** to allow you to find which **nets** are only on **Ungated Pins**. If you have Components in your Schematic design that use Ungated Pins and these are not connected outside of this device, this check will identify that single pin nets have not been created when translating to the PCB.

Electrical Rules Check		×
🗹 Pin Type Rules	Unfinished Connections	Coincident Items On Different Nets
Mark Warnings	Unlabelled Nets	Split Nets
✓ <u>B</u> usses	Unlabelled Net Pages	🗌 If Not Linked By Doc Symbol
✓ <u>H</u> ierarchy	Bridged 2-Pin Components	Pins Not On A Net
🗹 Un <u>f</u> inished Nets	Unmatched Page Links	Net Pins With No Connection
Validate Attributes	Common Pins	✓ Nets Only On Ungated Pins

First Free Component Name in Properties (SCM)

In Schematics, on the **Component Properties** dialog, there is now a button which will give you the first free component name that does not yet exist in the design.

E Properties: Component Attribute Component							×
Attribute	Text Style	Component	Nets on Pins	Comp Attributes	Vault		
			*	Locked			

New PCB Wizard

A new wizard is available on the New dialog and Wizards for creating a new PCB design.

The **New PCB Wizard** allows you to specify the **Technology** file you want to use (or use the default technology) and the default units.

PCB Wizard - Technology	PCB Wizard - Technology					
Start	Specify your required PCB Technology					
Technology						
Layers						
Board Profile	● Use Default Technology					
Finish	⊖Use a Technology File:					
	4 Layer (White - Metric)					
	Units:					
	● Imperial: thou ✓					
	◯ <u>M</u> etric: mm ∨					
	Precision: 1					

Then you can either use the **Layers** from the chosen Technology, or define your own using the type and bias required.

PCB Wizard - Layers		
Start Technology Layers	Specify which layers you require	
Board Profile ■ Finish	Define Layers: Electrical Layers Single Sided Board 2 Layer Board 4 Layer Board 6 Layer Board 8 Layer Board 10 Layer Board 12 Layer Board 14 Layer Board	Paste Mask for SMT Assembly Top Side Top Side Bottom Side Bottom Side Solder Mask Silkscreen Top Side Top Side Bottom Side Top Side Bottom Side Top Side Top Side Top Side Bottom Side Solder Mask
	Power Plane Layers	Auto-Route Bias
	Layer 2 Using Net: <none> ~</none>	First Routing Direction:
	Layer 3 Using Net: <none> <</none>	OY Allow Boutes:
	Layer 4 Using Net: <none> <</none>	⊡On Top Side
	Layer 5 Using Net: <none></none>	 ✓ On Inner Layers ✓ On Bottom Side

Finally, you can either define a **board** size, use a **Profile**, or copy a board from another file. Your selections will then create the appropriate PCB design once the design name is chosen on the **Finish** page.

PCB Wizard - Board Profile	
Start Technology	What kind of board would you like to create?
Layers	Define Board Size
Board Profile	Circle Width: 3000.0 thou Height: 2000.0 thou
- Finish	◯ Use this Profile:
	[None] V Browse
	Copy Board From Another File: Board Preview:

Changes to Database Check and Update Options

If you have the **Pulsonix Database Connection (PDC)** option, from the **Tools** menu and **Check/Update Against Database** option in the Schematic editor, you can now choose to update the selected Components on the **Current Page** or **All Pages**, (All Pages means components selected on different pages).

	Database Check		×
	Choose Items to Check All Components Selected Components (Current Page) Selected Components (All Pages)	Update Design from Database Allow Update of Design to Match Database Update Component Attributes Update Local Component to Database	<u>C</u> heck Update C <u>a</u> ncel
,	Choose What to Check Components using Local Parts Attributes Different To Database	Reporting Results Highlight Components Highlight Components using Local Parts Highlight Mismatched Components	Default Reset

Technology Changes

Pad Styles – new Usage Types

Pad Style usage on the Pad Styles dialog in the Technology has been expanded to include use for Through Mounting Holes and Surface Mounting Holes.

	Name: Used:	-Shape:	Drill:	
	Slotted Oval (50 x 70)	Type: Chamfered Rectangle ~	Shape: Oval 🗸	
	Named by: Typed Rule Templ Slotted Oval (50 x 70) For Use By:	te <u>W</u> idth: 1.2700 Length: 1.7780	Width: 0.4572 Length: 1.1430	
\Box	 ✓ Through Hole Pads ✓ Vias ✓ Surface Mount Pads ✓ Micro-via ✓ Through Mounting Holes ✓ Surface Mounting Holes 	Comer Radius: 0.0000 Offset: 0.0000 0.0000	Inner Diameter: 0.0000 Offset: 0.0000 0.0000 Rotation: 0.0 0.0 Plated Through: ✓	0.9000 mm

When adding Mounting Holes, if you swap from though-hole to surface mount and then change the Pad Style, if you have a long list of styles, this will aid distinguishing and refining the list to only use pads that are approved for this function.

This functionality was back-fitted to version 10.5.

Drill Removes Pad Warning now shows Pad Style

When adding or editing **Pad Styles**, when you press the **OK** or **Apply** buttons on the dialog to exit it, if there are a number of pads that have a drill size larger than the pad size, the warning dialog presented will now also show you the Pad Style name so it can be easily identified. Previously, the Pad Style wasn't shown.

Pulsonix	×	
?	Warning: Drill will totally remove pad. Continue with this drill value <mark>? Round (0 x 0.5)</mark>	
	Yes <u>N</u> o	1

Units shown in Technology

You can now temporarily switch the units on the **Technology** dialog using the Units button at the bottom of the page. The button will display the units currently being used on this page of the Technology. When toggled, a small asterisk * will appear to show you that the units currently being used are different to the design units. The setting of this button does not affect the design units.

	Differential Pair Gap Rules: <di< th=""></di<>
	<
mm	ОК

Additional Cell Status Indicators

The small triangle on the top left of the first cell in each row now indicates more states, Orange and Grey states have been added;

Red is an Error (as before). For example, on a Retangle Pad Shape where the Width defined exceeds the Length.

Y VIA 500	коипа	0.50000		0.200	00
Rect	Rectangle	3.24800	10.76200	0.000	00 🔽
Na <u>m</u> e:	<u>U</u> sed:	Shape:			Drill:
Rect		Type: Rect	angle	\sim	Shape: Round
Named by: Typed	○ Rule ○ Template	Width: 10.76	5200		Width: 0.0000
Rect		Length: 3.248	300		Length: 0.000(
For Use By:					lana Diamatan

Blue is a Warning (as before). For example, a Drill Hole larger than the pad on a Pad Style.

-	Mounting Hole	Round	3.81000	
	Ovel11	Oval	0.63500	

Orange indicates a new row

I		v1∠un8upu	Kouna	1.2000
	Υ	via.2547	 Round	0.4699
		c60h30	Round	0.6000

Grey is a modified row. This has been extended to all pages that have a **Used** or **Enabled** first column, e.g. **Styles**, **Layers** etc.

Y	Via 500	Round	0.50000
	Rect	Rectangle	0.76200

For **Rules** pages only, the Green triangle in the top right corner of the cell indicates the rule contains a note (as before).

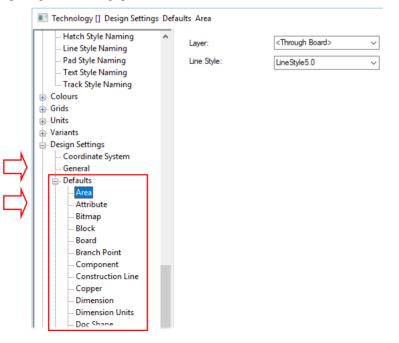
				Differential Pair Nets		Match	Within
E	Enable	Attribute Name	te Name Match Value	Net 1 Match	Net 2 Match	Item Type	Name N
Г	\checkmark	<net name=""></net>	ż	*TX	*RX		
		<net name=""></net>	*	*P	*N		

Updated Design Settings Pages

The **Design Settings Defaults** pages have been moved from a separate property sheet (accessed from General/Defaults in Technology) to a sub-category of **Defaults** off the **Design Settings** tab in the **Technology**.

The new *switch Units* button that has been added to the Technology dialog also works for all compatible **Defaults** pages, for example Branch Points.

The **Edit Default**" button that appeared on the **General/Defaults** page has been removed as it is no longer required and the page has been renamed to **General**.



Changes to Grids dialog

There are three main changes to Grids;

- Grid presentation in a table
- Removal of in-built grids
- New Tools Grid page

Grid Presentation

Design **Grids** are now presented in a 'grid' and operate in much the same way as other gridded pages in the Technology.

	Name	New
1	/ Component	
	Panel PCB Instance	
۲	/ Testpoint	
1	/ Text	
1		Delete
<u>``</u>		
	/ Working	
Name: Panel PCB Instance Relative to Grid: <not relative=""></not>	Used:	Step Polar Grid Basic Step Multiplier Divisor Step 2.54000 x 1 = 2.54000
Displayable		Different X & Y
	th Step Style	Origin
<u> </u> ▼1	Dots Lines	X 0.00000 Board Centre Lock
	◯ <u>C</u> rosses ◯ <u>X</u> Crosses	Y 0.00000 Coordinate Origin Lock
Global Grid Settings		Relative Origin Lock
Display: No Grids	\sim	Apply Origin Settings To All Grids

No In-built Grids

There are now no in-built grids, and you can rename and delete any unused grids.

Previously, built-in grids were shown in brackets, like <Working> for example. When your designs or Technology files are read into V11.0, the grid names will be preserved but you can now rename them or delete them (if unused). Like any table that has a value that is used shown with a **Y** next to its name, the name cannot be removed until unused. You can also remove all grid names from the list although this isn't advised. If all grids are removed, you will only be able to work gridless.

If you have allocated **shortcut keys** for existing commands **Next Grid** and **Next User Grid**, these will spin through all grids now. Any grids not required can be removed, for example, existing 'system' grids (in brackets such as <Working>).

		Snow riacement Origin				
		Change Layer	L			
		Change Style	s	~	Use No Grid	Shift+N
		Insert Multiple Items		_	Use Tools Grid	Shift+W
	~	Show Track Length Limits				
		Reposition Cursor Text			Use 25 thou Grid	
		Use Dynamic Align			Use Placement Grid	
			_		Use 0.1mm Grid	
	*	Online DRC	۷.		Use 50 thou Grid	
	~	Continuous Online DRC			Use 0.125mm Grid	
Name	~	Display Clearance			Use 0.500mm Grid	
0.1mm					Ose 0.500mm Gru	
0.125mm		Move Horizontally			Display Gride	Ctrl+G
0.500mm		Move Vertically			Display Grids	Ctri+G
25 thou					Grid Step	G
50 thou		Change Grid	_	#	Grids	Alt+G
Placement		change onu	'	+++	onom	

New Tools Grid page

Under **Technology** and **Grids**, there is a new **Tools Grid** page. This is used to define the 'default' grids used by non-interactive items.

Tools Grid:	Working ~	Step:	0.625
☑ Use Different G	rid for Item Types:		
Component:	Component ~	Step:	2.500
Testpoint:	Testpoint ~	Step:	2.500
Text:	Text ~	Step:	0.625
Track:	Track ~	Step:	0.625
Via:	Via ~	Step:	2.500
Panel PCB	Panel PCB Instance \lor	Step:	2.500

When the Use Different Grid for Item Types is selected, these grid settings are used:

Component: auto place, arrange

Test Point: auto place, auto insert TP, testability report, auto routers

Track: auto routers

Via: apply vias, auto routers

PCB Panel and Text, these are currently not used.

Group Name available in Component Place Rules

You can now use **<Group Name>** as the rule attribute in the **Component Place** rules dialog in the Technology. This name is available on the **Attribute** drop down list.

	Attribute:	<component name=""></component>	/	Default Side:	
\neg	Match:	<component name=""> <footprint name=""> <group name=""></group></footprint></component>	*	Restriction: Keep Out	\sim
`	Side	Category Category Description Title Value	~		
	Within Area	IS:	~		

Component Placement Rule - Default Mirror State

There are new ways to define the default mirror state when adding a Component.

Footprint Editor – Mirror Status

In the **Footprint Editor**, using the **Edit** menu and **Symbol Properties**, you can define a footprint to be added **Mirrored** or **Not Mirrored**. This is used when adding a Component to a PCB design.

The normal state is **Mirror Undefined** - its mirror state is determined elsewhere. **Not Mirrored** means the footprint will not be mirrored, **Mirrored** means it will be mirrored.

Symbo	Properties	Х
Name:	21-0073E_25×25 Embedded Component Mirror Undefined Designed Mirrored Mirror Undefined OK CanceNot Mirrored	

You can define a footprint as **Defined Mirrored**, which means it is designed as it would appear when mirrored. See section below under *Define Mirrored Footprints*.

Technology Component Placement Rule

You can create a Component Placement Rule to default the side (i.e. the mirror state) based on Component Attributes.

Attribute:	Component Name>	~	Default Side: 🗹
Match:		× 🕺	Restriction: Keep Out ~
On Layers	[
Side:	Bottom	~	
Within Areas:		~	

By defining a rule based on a specific **Side** and selecting the **Default Side** check box, you can force the placement side when adding Components to your PCB design.

Pad Styles – New Naming Rule for Non-round Holes

You can now add drill length for non-round holes. A new **Field** of **Drill Length** is available for selection. A **Template** value of **<dlen>** is used when inserted.

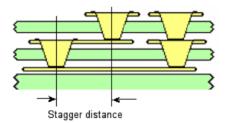
Shape <u>Type</u> : <any> ~</any>	Template: <dlen></dlen>	
For Use by:	Field Keyword	
Through Hole Pads Vias	<dlen></dlen>	Insert
Surface Mount Pads Micro-vias		
Through Mouting Holes	Field: Drill Length ~	
Surface Mounting Holes	Pre-text:	1
Drilled status:		
Drilled Plated Through		

Micro-Via to Buried Via Stagger Spacing

Within the **Technology Spacing** rules and **Drill** tab of design level spacings, the **Stagger** option to has been renamed to **Stagger to Micro-via**.

A new spacing rule has been added to the **Drill** tab named **Stagger to Buried Via**. This allows for checking spacings between micro vias and buried vias on adjacent layer spans.

Each of these rules will be the minimum centre to centre distance between micro-vias on adjacent layer spans or the minimum centre to centre distance between a micro-via and a buried via on adjacent layer spans respectively.



This feature was back-fitted to 10.5

User Defined Pad Shape Improvements

Define Pad Shape - Multiple Shapes Allowed

In **Define Pad Shape**, you can add multiple shapes for on a **Non-Electrical layer**. For example, to define multiple glue spots. This can be done by adding the extra shapes and changing their layer to an appropriate one.

\bigcirc	

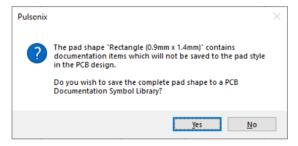
Define Pad Shape now contain Text, Dimensions and Callouts

When using the **Define Pad Shape editor** (when editing a pad style from a PCB Design) you can now add **Text**, **Dimensions** and **Callouts**. These are in addition to being able to add Construction Lines. **Properties**, **Technology Styles**, **Defaults** and **Colours** are available to support these documentation items.

Saving Documentation items for reuse

In version 11, you can now save all of these items within the Pad Shape so that they can be recalled for use on future edits. Effectively, for special Pad Shapes that have this additional information you will create a set of Pad Shapes in your library. Previously, Construction Lines we transient and lost when you edited the Pad Shape editor. **To save these Documentation items, you must save the Pad to a PCB Doc Symbol in a library.** If you choose not to save the Pad to a Doc Symbol then the additional information will be lost. Not saving the Pad Shape will still mean it retains any user defined shapes as before.

When saving back to a style in the PCB design, if the Pad Shape is in a library or if the Pad Shape contains documentation items, you are asked if you want to save to a Doc Symbol library.



Choosing Yes will display the Save To Documentation Symbol Library dialog:

Save To Documentation Symbol Library					
Library:	user	~			
<u>N</u> ame:	Rectangle (0.9mm x 1.4mm)				
	OK Cancel				
	Caliba				

This can also be accessed from the File menu using the Save To Library option.

Choosing **No** will exit the Pad Shape editor, save the pad shape defined but will not save any of the additional documentation information, this will be lost.

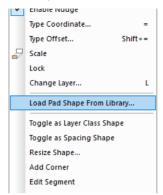
User defined pad shapes can now be added to the **Vault** as well as your **PCB Documentation Symbol Library**.

Saving a Pad Shape as a **Doc Symbol** will be saved as a new symbol type, **Pad Shape**. These can be edited the same as any other Doc Symbol and accessed from the **PCB Doc Symbol Library Manager**.

Load Pad Shape From Library

When editing PCB pad style, from in the **Footprint Editor** for example, if you wish to use a Pad Shape that has already been saved as a Doc Symbol, then for a selected pad, you can use **Load Pad**

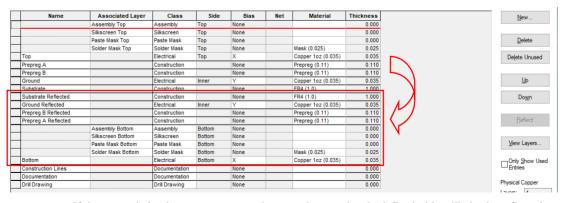
Shape From Library from the context menu. If the pad style name is already in a library or vault you will be asked if you want to load it when first editing the style.



Technology Layers - Reflect Layers

A new **Reflect** button to the **Layers** page in the **Technology** has been added. This allows you to create the 'other' half of a layer structure by reflecting the current layers. For example, you can create the top 7 layers of a 14-layer board, and using Reflect, then create the bottom 7 layers automatically.

Name	Associated Laver	Class	Side	Bias	Net	Material	Thickness
	Assembly Top	Assembly	Тор	None			0.000
	Silkscreen Top	Silkscreen	Тор	None			0.000
	Paste Mask Top	Paste Mask	Тор	None			0.000
	Solder Mask Top	Solder Mask	Тор	None		Mask (0.025)	0.025
Тор		Electrical	Тор	X		Copper 1oz (0.035)	0.035
Prepreg A		Construction		None		Prepreg (0.11)	0.110
Prepreg B		Construction		None		Prepreg (0.11)	0.110
Ground		Electrical	Inner	Y		Copper 1oz (0.035)	0.035
Substrate		Construction		None		FR4 (1.0)	1.000
Construction Lines		Documentation		None			0.000
Documentation		Documentation		None			0.000
Drill Drawing		Drill Drawing		None			0.000



After using **Reflect**:

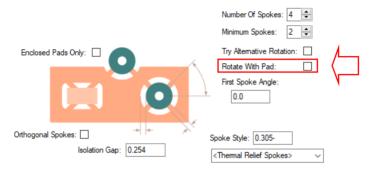
If, in your existing layer structure, you have a substrate already defined, this will also be reflected. Any additional unwanted and unused layers can be deleted as required.

The Reflect button will be disabled if you add or delete any layers from the grid and are yet to click apply (this could cause issues with the layer order). The button will enable once the changes are applied to the design.

If you use Reflect and you have (for example) a Silkscreen Top layer and you already have a Silkscreen Bottom layer that is used in the design, no changes will be applied to that bottom layer. If the Silkscreen Bottom layer is not used in the design, then the layer data from Silkscreen Top will be copied to it (for example, Silkscreen Top is reflected and that layer replaces the current Silkscreen Bottom layer).

Thermal Rules - Rotate with Pad

Within the **Thermal Rules** page of the **Technology** dialog, there is an option to rotate the thermal pattern to the angle of the pad (**Rotate With Pad**).



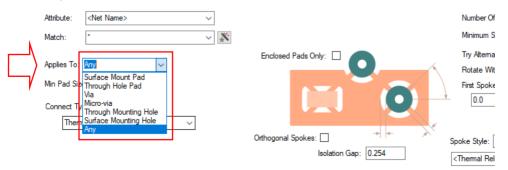
If a pad is at 25 degrees for example, and the first spoke angle is 0 degrees, the first spoke would actually be a 0 degrees relative to the pad, but 25 degrees relative to the board.



This feature was back-fitted to 10.5

Thermal Rules for additional Pad Types

You can now define explicit **Thermal rules** for **Micro-vias**, **Through Mounting Holes** and **Surface Mounting Holes**. To use this correctly, you should define the rules in the correct order required in order for them to take precedence. For example, Micro-via before Via (as via also applies to Microvias).



Copper Neck Width Rules (Power Dissipation)

The **Copper Neck Width Rules** dialog within the **Technology** is used to define the minimum necked width of a piece of copper which forms a connection on a net (this is not a slither check). The minimum necked width is the minimum width for the body of copper between two pins connected by that copper. The rule can be checked using a **Design Rule Check** and the rule under **Nets**, **Copper Neck Width**.

This rule could be used where **power dissipation** is important and you require a certain minimum gap between pads to be adhered to. It would be normal to check for specific nets to be adhering to this rule, such as power and ground.

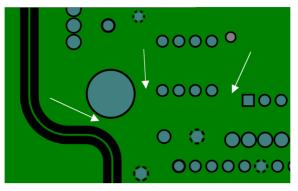
This rule works on all poured copper but is **not** used for implied powerplanes (**Layers** defined in your **Technology** with a **bias** of **Powerplane**).

Enable	Attribute Name	Match Value	Side	Layer	Minimum Width		New
	<net name=""></net>	GND*			0.100		
						1	Delete
							Up
							Down
						١	Where Used:
							Selected
							All
						L	
Attribute	e: <net name=""></net>		\sim				
Match:	GND*		94.4				
	GND		~ 🕺				
On Lay	yers				0	0	
9	Side: <any></any>			\sim			
La	ayer:			~	1		
					Minimum Neck W	/idth:	
					0.100		

Minimum Neck Width

The **Minimum Neck Width** is the minimum width of copper between two pads connected by the copper. The width is the total of each piece of copper between cutouts in the path between the two pads. This will be positive closed copper shapes and doesn't include tracks or thermal spokes.

When the rule defines Pads, it means all pad types including, Component and free Pads, Mounting Holes, Vias and Doc Symbol pads provided they exist on the same layer as the check.



The illustration below highlights the 'gaps' that are available for power dissipation to take place.

Creepage Rules

A new page on the **Technology** dialog under **DFM/DFT**, named **Creepage Rules**, enables you to define rules for Creepage.

	Check	Check Nets Against Ne				Check O	n	Creepage			
Enable	Attribute Name	Match Value	Attribute Name	Match Value	Side	Layer	Area	Max Outer Distance	Max Inner Distance	Flow Around	Max Steps
	<net name=""></net>	GND	<net name=""></net>	ż				0.5	0.3		1
	<net name=""></net>	HV*	<net name=""></net>	ż				0.5	0.3		1
	Check Nets: Attribute: <net Match: HV*</net 	Name>	~ ~	Flow Arou	ınd Edge						
	_	N				-			+		
	Attribute: <net< td=""><td>Name></td><td>~</td><td></td><td></td><td></td><td></td><td></td><td>+</td><td></td><td></td></net<>	Name>	~						+		
	Match: *		~ *	:		-	_				
	On Layers:									_	
	Side: <any< td=""><td>></td><td>\sim</td><td>Maximum Ou</td><td>ton Distan</td><td>0.5</td><td></td><td></td><td></td><td></td><td></td></any<>	>	\sim	Maximum Ou	ton Distan	0.5					
	or		~	Maximum Ou	iter Distar	ice: 0.5					
	Layer:		Ŷ	Maximum Inn	ner Distan	ce: 0.3					
	Within Areas:		\sim	Maximum Ste	eps:	1					

The purpose of the check is to report where the total surface gaps could allow creepage between two nets, either directly or through third party copper. You can also check creepage around board edges or cutouts. You can limit this check to a specific layer or named area.

Creepage is caused by current flowing through pollutants on the surface of a board, this is typically increased with higher voltages. The creepage check finds gaps between critical nets between which current could flow. Third party copper can conduct current, meaning that the creepage can accumulate in several steps or jumps between the two critical nets. As this flow is across the surface of the board, the distance can effectively be increased by cutting a slot in the board, the creepage distance then follows the lip of the board edge. Current can also flow around the edge of the board in the third dimension, so changing layers. Although typically less of a problem, creepage can occur across inner board layers. There are therefore two values for the maximum creepage distance - Outer and Inner, which are then combined to give the final result.

Using this Rule

It is normal to define a specific net or a few nets that are critical to your design when considering creepage. These nets would be defined to be minimum distances from other nets in the design, possibly all others.

To achieve this, as with all other rules, you define the critical net (**Check Net**) and define the net which it is checked against (**Against Nets**).

Nets can be checked on a Side, Layers, Layer sets or within an Area.

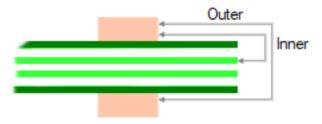
-Check Nets	3:	
Attribute:	<net name=""> ~</net>	Flow Around Edge
Match:	HV* ~	
Against Net	s:	
Attribute:	<net name=""> ~</net>	1
Match:	• • *	
On Layers:		
or		Maximum Outer Distance: 0.5
Layer:	~	Maximum Inner Distance: 0.3
Within Areas	s: 🗸 🗸	Maximum Steps: 1 🖨

Maximum Outer Distance

The maximum total distance for which creepage would cause a problem on the outer surfaces of the board. This includes the distance around a board edge. This distance around the board edge can include changing layers, as well as around the lip of the edge. A value of 0 means that no creepage distance is checked on an outer layer.

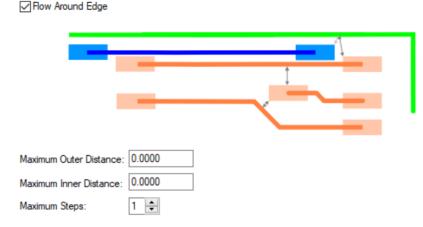
Maximum Inner Distance

The maximum total distance creepage would cause a problem on inner layers of the board. This value is typically less than the Maximum Outer Distance. This does not include any layer changes as these can only be done on the outer edges of the board. A value of 0 means that no creepage distance is checked on an inner layer.



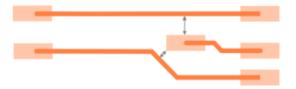
Flow Around Edge

Check this box to allow layer changes by flowing around the edge of the board in the third dimension (the board edge includes board cutouts). This does not prevent flow around the lip of the board edge in the layer plane.



Max Steps

The maximum number of steps between copper. One step means a single gap between the two nominated nets. Two steps means that an intermediate piece of copper is traversed.



Combining Inner and Outer Distances

Each step will contribute to the total inner or outer distance. The inner and outer distances are combined in proportion to their respective maximums. If the total percentage does not exceed 100%, then there is an error. For example: Maximum Outer Distance = 2mm, Maximum Inner Distance = 1mm; actual total outer distance = 1.5mm (75% of maximum), actual total inner distance = 0.3mm (30% of total); total percentage = 105% so there is no error.

Design Settings – new Synchronise Design options

In the **PCB Technology** file under **Design Settings** and **Synchronise**, there is a new option switch in the list named **Apply Net Colours**. This will cause the **Own Colour** assignments for **Net Names**, **Differential Pairs**, **Signal Paths** and **Sub Nets** in the PCB to be synchronised with those defined in the Schematic. This may be useful for quickly identifying these items in the two designs.

✓ PCB in Safe Mode							
Allow PCB Only single pin nets							
Synchronised Design Name							
Name: HS2							
Back Annotation							
Enabled Clear History							
Synchronise with Schematic							
Apply All Rules Strictly							
Apply Footprint Changes							
Apply Net Class Changes							
Apply Net Colours							
Ignore Attribute White Space							
Allow Update of Schematic to match PCB							

Also, in this dialog, the Differential Pairs colour changes has been removed from strict synchronise option when the **Apply All Rules Strictly** switch is use.

	PCB in Safe Mode Allow PCB Only single pin nets
	Synchronised Design Name Name: HS2
	Back Annotation
_\	Synchronise with Schematic
	Apply All Rules Strictly
7	Apply Footprint Changes

Import IPC-2581 Layer Stackup into Layers dialog

An option to import **Layers**, **Layer Classes**, **Materials** and **Impedance rules** from an **IPC-2581 Layer Stackup** file produced by 3rd party products, such as Polar Instruments has been added.

This format is now available from the **Technology** page and **Layers**, and also from the **Import Design Data** option on the **File** menu.

Layers Technology Page

On the **Layers Technology** page the **Import CSV...** button has been replaced with **Import...** When selected, you can import **CSV files** (as before) and new **Layer Stackup** (*.**xml**) files, (the .xml format is used by IPC-2581). The **Files of Type** filter allows you to choose the type required.

~	All Import Files (*.csv;*.xml)	\sim
	All Import Files (*.csv;*.xml) Comma Separated Files (*.csv)	
	Comma Separated Files (*.csv)	
	IPC2581 Layer Stackup (*.xml)	
	All Files (*.*)	

Import IPC-2581 Layer Stackup dialog

On selection of a valid IPC-2581 file you are presented with a dialog from where you can choose an existing mapping file or create a layer mapping set.

🛛 Use Layer Mapping		
Use Mapping File		
No Selected Mapping File	Browse	
IPC-2581 Layer Stackup Layer	Pulsonix Layer	^
SOLDERMASK_TOP	Solder Mask Top	
L1	Тор	
DIELECTRIC_1		
L2	Ground	
DIELECTRIC_2		
L3	Power	
DIELECTRIC_3		
DIELECTRIC_4		
DIELECTRIC_5		
L4		
DIELECTRIC_6		
L5		
DIELECTRIC_7		
DIELECTRIC_8		
DIELECTRIC_9		
6		×

The imported file populates the **Layers** dialog:

Name	Associated Layer	Class	Side	Bias	Net	Material	Thickness
	Assembly Top	Assembly 🗸	Тор	None			0.000
	Silkscreen Top	Silkscreen	Тор	None			0.000
	Paste Mask Top	Paste Mask	Тор	None			0.000
	Solder Mask Top	Solder Mask	Тор	None	1	Mask (0.025)	0.025
Тор		Electrical	Тор	None		Copper Foil	0.036
Prepreg A		Construction		None		Prepreg (0.11)	0.110
Prepreg B		Construction		None		Prepreg (0.11)	0.110
DIELECTRIC_1		Prepreg	Inner	None		PrePreg 3113	0.060+
Ground		Electrical	Inner	None		FR4 Core Cu	0.053
Substrate		Construction		None		FR4 (1.0)	1.000
DIELECTRIC_2		Core	Inner	None		FR4 Core	0.075
Power		Electrical	Inner	Х	-	Copper 1oz (0.035)	0.035
Prepreg D		Construction		None		Prepreg (0.11)	0.110
Prepreg C		Construction		None		Prepreg (0.11)	0.110
DIELECTRIC_3		Prepreg	Inner	None		PrePreg 1080	0.069
DIELECTRIC_4		Prepreg	Inner	None		PrePreg 7628	0.184
DIELECTRIC_5		Prepreg	Inner	None		PrePreg 1080	0.069
L4		Electrical	Inner	None	-	FR4 Core Cu	0.035
DIELECTRIC_6		Core	Inner	None		FR4 Core	0.300
L5		Electrical	Inner	None		FR4 Core Cu	0.035
DIELECTRIC_7		Prepreg	Inner	None		PrePreg 1080	0.069
DIELECTRIC_8		Prepreg	Inner	None		PrePreg 7628	0.184
DIELECTRIC_9		Prepreg	Inner	None		PrePreg 1080	0.069
L6		Plane	Inner	None		FR4 Core Cu	0.035
DIELECTRIC_10		Core	Inner	None		FR4 Core	0.075
L7		Electrical	Inner	None		FR4 Core Cu	0.053
DIELECTRIC_11		Prepreg	Inner	None		PrePreg 3113	0.060+
Bottom		Electrical	Bottom	None		Copper Foil	0.036
	Solder Mask Bottom	Solder Mask	Bottom	None		Mask (0.025)	0.025
	Paste Mask Bottom	Paste Mask	Bottom	None			0.000

Layers Import/Export CSV into Layers dialog

There are new options on the **Layer Technology** dialog that enable you to **Export Layers** to a CSV file and **Import Layers** from a CSV file. This is intended to allow you to create layer stacks quickly or from an external resource.

The export column headings for a layer are Name, Class, Type, Side, Top Facing, Bias, Net, Material, Thickness, Embedding and Associated With.

CSV Format:		
Field separation character: 📜 🗌 Use tab	Decimal point character:	S
Include Table Title: Layers Page	~	
Map Table Columns:		
Layer Column Name	CSV Column Name	^
Layer Column Name	CSV Column Name	î
		^
Name	Name	^
Name Class	Name Class	^
Name Class Type	Name Class Type	^
Name Class Type Side	Name Class Type Side	^
Name Class Type Side Top Facing	Name Class Type Side Top Facing	^
Name Class Type Side Top Facing Bias	Name Class Type Side Top Facing Bias	^
Name Class Type Side Top Facing Bias Net	Name Class Type Side Top Facing Bias Net	^
Name Class Type Side Top Facing Bias Net Material	Name Class Type Side Top Facing Bias Net Material	

Import dialog

Similar to the Export dialog with controls for defining how the CSV file is formatted, any concessions for Row 1 and Column A and layer mappings available.

CSV Format:		
Field separation character: 💭 🔲 Use tab	Units for values: mm VUse Der Decimal point character:	sign Units
ind Table Using:		
Title: Layers Page		
OPosition: row: 1 column: A		
1ap Table Columns:		
1ap Table Columns:		
Aap Table Columns: CSV Column Name	Layer Column Name	ŕ
	Layer Column Name	
CSV Column Name		
CSV Column Name	Name	
CSV Column Name Name Class	Name Class	
CSV Column Name Name Class Type	Name Class Type	
CSV Column Name Name Class Type Side	Name Class Type Side	
CSV Column Name Name Class Type Side Top Facing	Name Class Type Side Top Facing	
CSV Column Name Class Type Side Top Facing Bias	Name Class Type Side Top Facing Bias	
CSV Column Name Class Type Side Top Facing Bias Net	Name Class Type Side Top Facing Bias Net	
CSV Column Name Class Type Side Top Facing Bias Net Material Thickness	Class Class Type Side Top Facing Bias Net Material Thickness	
CSV Column Name Class Type Side Top Facing Bias Net Material	Class Class Type Side Top Facing Bias Net Net Material	

Option to use a spacing shape even when pad is suppressed

There is a new option in the Technology dialog, Spacing Rules and Design Level - Pad tab named Use Suppressed pad spacing shape.

With this enabled, it means that even when pad is suppressed (on an inner layer for example) as defined in your Technology, Layers, it will still use a spacing shape on a pad if one is defined. All other pad suppression will still be adhered to though if defined.

		Component	Copper	Drill	Pad	Track	Enclosures
Minimu	um Pad Land						
Pad 1	Type: Through Ho	ole Pad 🛛 🗸					
● R	Radius Difference	0.1270					
OR	Radius Percentage						
OA	bsolute Area						

Load Technology - Matching Styles on Reload

Reporting on Reload

When loading a Technology file, a report will now be generated showing any styles in the design that have been changed as a result of the load.

Matching Styles on Reload

You can now view the current **style matching** method on the **Load Technology** dialog. This will be set to **By Name Only** by default but can be changed to **Use Design Settings'** using the check box. This will use the style matching method from the **Design Settings** dialog, **General** page and **Matching Styles** option.

💿 🖂 Serpentine Rules	~
Add if not already in design Match Styles: By Name And Value	Replace all rules in design with selected rules
🔽 Use Design Settings	
Load	Report Cancel

Rotated Pad Styles for 'Long' Pads

Long pad styles in Pulsonix are defined pointing up so the length lies along the Y axis. Some long pad styles in your existing designs are invisibly rotated by 90 degrees, so the length lies along the X axis. This situation normally happens when the design or library is imported from another system, or when using the **User-defined Pad Shape Editor** to define a rectangle or oval pad with that orientation. This has caused confusion as the rotation cannot be seen and there is no easy way to remove it.

A change has been made to show "Rotated" next to the pad width and length if it is rotated. Also, if the style is changed to a point shape (like round, square or octagon) the rotation is removed. This way a customer can change a rotated rectangle to a round and back to a rectangle to remove its hidden rotation.

Pad Properties Layer Override

Added a layer override check box has been added to **Pad Properties** when Component Pads are selected in the design. This is useful if, for example, you locally swap a through hole pad Component Pad for a surface mount pad but then also need this to be on a specific <Side> layer, such a Top Side.

	Properties: Pad: D3.1 Pad			_	
	Comp Attributes	Variants	Net	Net	Attributes
	Pad Pad Attributes	Back Drill	Test Cor	mponent	Nets on Pins
	<u>N</u> ame: 1	Logic Name: A			
	Physical Details				
	Override		_	Allow Interactive	e Reposition
	<u>P</u> osition: 31.750	41.9100		s Star Point	
	Angle: 90.0				
	✓ <u>L</u> ayer: <throu< th=""><th>gh Board></th><th>~</th><th></th><th></th></throu<>	gh Board>	~		
	Pad Style:				
· · · ·	Name: Round	1 (60)			
	MALL. 1 504	Character 1	D		

From an application perspective, a specific Part purchased is only available as through-hole and the board is mounted on a metal housing. If the connections of the high-voltage components were assembled as through-hole, this would produce a high-voltage flashover on the housing. Assembling the component as surface mounted on the top side, the board serves as insulation.

Lock Pad Details in Footprint Editor

Within the **Footprint** editor, for **Pad Properties**, there is now a **Lock Details** check box. This stops the physical details on a pad from being edited in the design.

Properties: Pad: 1 – Pad	— 🗆 X
Pad Pad Attributes Net	
☑ <u>N</u> ame: 1 □ Logic Name:	
Physical Details	└ Lock Details □ Locked
Position: 51.434+ 57.005+	Is Star Point
<u>Angle:</u> 90.0	

With this box checked, the pad on the footprint in the design is locked and its properties greyed out. The dialog is marked with **Override Locked** under **Physical Details**:

Properties:	Pad: U20.1	– Pad						×
Vari	ants		Vault	Net		1	Net Attributes	
Pad	Pad Attrib	outes	Test	Component	Nets or	n Pins	Comp Attrib	utes
<u> </u>			Logic N	ame: RA2				
Overrid	e Locked				Allo	w Interac	ctive Repositio	n
	Position:	47.69	05+ 94	.3569+	Is S	Star Point		
	<u>A</u> ngle:	90.0						
	Laver:	<botto< td=""><td>om Side></td><td></td><td></td><td></td><td></td><td></td></botto<>	om Side>					

Define Mirrored Footprints in Footprint Editor

Within the **Footprint Editor**, you can define a footprint as **Defined Mirrored**, which means it is designed as it would appear when mirrored.

Save To Library				
Library:	SM 🗸	OK		
<u>N</u> ame:	C0805	Cancel		
	Embedded Component Mirror Undefined 🤍 🗹 Designed M	lirrored		
🗌 Creat	e a new part using this footprint			

If the **Designed Mirrored** box is checked, the orientation of the footprint, as defined in the editor, is the mirrored state. It is likely that you will also want to set the **Mirrored** state as described above. The normal state is to leave this box unchecked.

Area Colour in PCB Doc Symbol Editor

Area colours can now be defined in the PCB Doc Symbol Editor.

Background Dimming on Mark Net

In the PCB editor, there is a new functionality that allows you to toggle the background dimming whilst using **Mark Net** (Selected Net) in **Latch Mode**. This toggle is listed in the context menu whilst within this mode.

When in Mark Net mode, right click and toggle Dim Design To Brighten Net mode.

Cancel Mark Net		
Mark Net		
Auto Select		
Dim Design To Brighten Marked Net		

Copper Pour Multi-Threading

The option, **Pour All Templates** can now use multiple threads. For large or complex designs, or designs with many templates, this significantly speeds up the processing.

The selection to **Enable Threads** for **Copper Pour** has been added to the **Options** dialog, **General** page. This setting is used by all Multi-threading technology in Pulsonix.

Optimise after Clear All Templates Multi-threading

Multiple threads can now be used when **Optimising** nets after **clearing all templates**. The **Optimisation of nets** is performed when Clear All Templates feature is run. **Optimise** is also controlled by **Optimise On Clear** switch on the **Interaction** page in **Options**. For designs with large nets, unchecking this option will significantly speed up the clearing of Templates.

General In-Place Names Interaction	Power & Ground Pins Auto Connect: Always	Offset from C
Macros Move Multi-Screen	Undo Undo Pan/Zoom	Join Open Shape Join Toler
Pan & Zoom Online DRC Resolve Net Names Select	RC Delete RC Unextended Delete - Deletes Segments Optimise After Delete	Auto Footprints
Synchronisation Tooltips Track Length Limits Warnings	Optimise Only Signal Nets Templates When Adding Templates	Differentially Pair Whilst Editing
	Act Poured Pour On Add Hide Template When Poured Highlight Isolated Copper	Route Selected
	Clear On Edit Repour Affected Templates Optimise On Clear	

The selection to **Enable Threads** for **Optimise** after **Clear All Templates** has been added to the **Options** dialog, **General** page. This setting is used by all Multi-threading technology in Pulsonix.

Change Style DRC Checks Multi-Threading

Design Rules Checking can now use multiple threads when after changing styles of many items.

The selection to **Enable Threads** for **Design Rules Checking** has been added to the **Options** dialog, **General** page. This setting is used by all Multi-threading technology in Pulsonix.

Change Style Performance Update

Changing a Style

Changing a Style is now classed as an **After Edit** DRC action. If this option is disabled in the **Options** dialog, **Online DRC** page, then DRC will not be performed after changing style.

Options Online DRC	
Design Backups Display Edit Shape Edit Track File Extensions	☑ Online DRC Checking ☑ Continuous (Avoids errors during some interactive operations) ☑ On Drop (Check changed data after interactive operations)
Find	Check Attached <u>T</u> racks & Wires On Drop
General In-Place Names	After Edit (Check item after having its properties edited)
Interaction	

Vias Attached On Different Layers

When multiple vias are selected that have different connectivity, the wording <Different> will be seen in the **Attached on layers** list box, this makes opening the **Via Properties** dialog much faster when a lot of vias are selected.

<u>W</u> idth: 1.5240	Shape: Rou	ind	~	
Length: 1.5240	<u>D</u> rill: 0.81	128	d: B	
✓ Plated				
Power Plane Connection: Attached Tracks: Highest Lowest I	Default Layer: <different Layer: <different< td=""><td></td><td>Attached on Layers: <different></different></td><td></td></different<></different 		Attached on Layers: <different></different>	

This feature was back-fitted to 10.5

Multi-Threaded Design Rule Checking

Design Rules Checking can now use multiple threads when doing Spacing checks.

The selection to **Enable Threads** for **Design Rules Checking** has been added to the **Options** dialog, **General** page. This setting is used by all Multi-threading technology in Pulsonix.

New Design Rule Checks

The **DRC** dialogs for PCB, Footprints and Panels have been rearranged for clarity with options under each category are now sorted in alphabetical order. The category **Testpoints** have been split from **Manufacturing** for additional clarity.

Same Net Via To SMD

Manufacturing 🗹 Nets 🗹 Spacing Testpoints 🗹 Board Acid Traps Adjacent Nets Centre to Centre Components Bond Wire Length Connection Length Min Points Per Net Copper Component Name Connection Vias Pad Size 🗹 Drills Copper Shapes Differential Pairs Under Component Mount Holes Copper Text On Board Necked Track Unreachable Side Pads Drill Backoff Net Connectivity Split Planes Footprint Rules Power Planes 🗌 On Grid 🗹 Test Points Isolated Copper Parallel Track Components 🗹 Text Minimum Pad Land Pin Order Pads Tracks Mirrored Text Same Net Via To SMD 🗹 Test Points Serpentine 🗹 Vias Pad Undersize 🗹 Tracks Panel Items On Board Single Pin Nets 🗹 Vias Plane Thermal Pad Stub Vias 🗹 Keep In/Out Silkscreen Overlap Teardrops Component Pads Split Plane Pad Track Layer Components Unpoured Templates Track Length Validate Attributes Copper Wire Cross Track Width

There is a new option in DRC to check the spacing between Vias and SMD pads on the same net.

The **Same Net Via To SMD Pad** spacing is defined in your **Technology** dialog under **Via** rules (**Rules – DFM/DFT**).

Attribute:	<net name=""> ~</net>		
Match:	* ~ *		
On Layers		Normal Via in Surface Mount Pad	Micro-Via in Surface Mount Pad
Side	e: <any> ~</any>	── ○ ~─	
Layer	· 🗸 🗸		
		Same Net Via To SMD Pad Min Spacing: <unrestricted< td=""><td></td></unrestricted<>	
Within Area	35:		

This can also be enabled in **Online DRC** by selecting the **Check Same net Via To SMD** check box in the **Options** dialog and **Online DRC** page. When using Online DRC, the clearances will also be displayed.

Options – Online DRC			
Design Backups Display Edit Shape Edit Track Elit Track File Extensions Find General		(Check cha	rs during some interactive operations) nged data after interactive operations) acks & Wires On Drop
In-Place Names	✓ After Edit	(Check item	after having its properties edited)
Interaction			
Macros	Add Error Ma	arker (to show	first error)
Move Multi-Screen	Continuous		On Drop Only
Pan & Zoom	Check Pour	ed Copper	✓ Check Via In Pad
···· Online DRC			Check Comp To Comp
Resolve Net Names			Check Same Net Via To SMD
Select			

Silkscreen Overlap

There is a new option in **DRC** to check for any pads that are overlapped by silkscreen items.

Design Rule Check	:			Х
Spacing	Manufacturing		Nets	
🗹 Tracks	Isolated Copper	Copper Text On Board	Single Pin Nets	
🗹 Vias	Unpoured Templates	Panel Items On Board	Net Connectivity	
🗹 Pads	Split Plane Pad	Copper Shapes	Power Planes	
Mount Holes	Plane Thermal Pad	Acid Traps	Unfinished Track	
🗹 Test Points	Bond Wire Length	🗹 Silkscreen Overlap	Track Layer	
Copper	Wire Cross	Testpoints	Track Width	
🖂 Tevt	Wire Under Componer	nt 🗌 Unreachable Side	Via Size	

Time To Process shown in DRC Report

The **DRC** report now shows the time taken to perform the check. You'll find this in the summary at the bottom of the report.

	Fotal:		
	7	Board to Component Error (B-Cm)	
	1	Board to Copper Error (B-C)	
	223	Board to Pad Error (B-P)	
	54	Board to SMD Pad Error (B-SM)	
	4	Component to Component Error (Cm-Cm)	
	47	SMD Pad to SMD Pad Error (SM-SM)	
(Checking took 0.657 Seconds		
י /ר	Number of errors found : 336		

This feature was back-fitted to 10.5

Changes to Online DRC

Display Clearances for Multiple Items

You can now see clearances when moving multiple items, for example, multiple Tracks as well as multiple Components.

The Online DRC dialog in Options has been changed to Allow Checking of Multiple Items).

Show Design Rule Clearance		
On All Items		
Refine Within Distance		
Within Set Distance: 30.000000		
O Within Grid Steps: 10		
Show Clearance On Breakouts		
Allow Checking of Multiple Items		

Changes to Show Design Rule Clearance

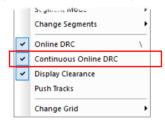
Within the **Online DRC** page of **Options**, display clearances section has been changed. Instead of three radio buttons for **On All Items**, **Within Distance** and **Within Grid Steps**, **On All Items** is now a check box. This allows for refining the clearances when moving a selection that has multiple possible clearances. When checked, the clearances for the selected item will be shown, but when another selected item comes within the defined distance or grid steps of an item it has a different clearance with, the clearance for that item will change.

The **Refine Within Distance** words will toggle between this and **Only Within Distance** depending on the **On All Items** check box selection.

Show Design Rule Cleara	ance		
☑ On All Items			
Refine Within Distance			
Within Set Distance:	30.000000		
◯ Within Grid Steps:	10		
Show Clearance On Breakouts			
Allow Checking of Multiple Items			

Continuous Online DRC switch added to context menu

A new switch has been added to the context menu to enable **Continuous Online DRC**. This enables you to suspend **Continuous** mode so that errors can be added (whilst using **On Drop** mode) without having to use the **Options** dialog.



'End Track On Via' Clearances

Continuous DRC and Via Errors

Via Spacing clearances are now shown for **Vias** added using the **End Track On Via** option when editing a Track. With a Via on the end of the track and **Continuous DRC** enabled (from the context menu), the clearance shown will toggle to the error colour when the via is in error.

The clearances for the via when using **End Track On Via** will only be shown if the **Allow checking of multiple items** option is enabled (it is no longer called **Allow checking of multiple components**).

If **Continuous Online DRC** is disabled, the clearances will use the error colour if either the via or the Track breaks the **Spacing Rules** defined in your Technology. If **Continuous Online DRC** is enabled, the Track will be 'blocked' from breaking any Spacing Rules, and the via will make clearances the error colour if it is breaking any Spacing rules.

l		Eaiting Options	•
		Segment Mode	×
		Change Segments	•
	~	Online DRC	X
	~	Continuous Online DRC	
	~	Display Clearance	
	~	Allow Checking of Multiple Items	
		Push Tracks	
		Change Grid	•

Allow Checking of Multiple Items

Allow Checking of Multiple Items has been added to the context menu when using the End Track On Via option. If this option is not checked, only the clearances for the Track will be seen. If it is checked, the clearances between the Track and the item, or the via and the item will be seen; whichever one is closest to being in violation.

If the **Clearances** are being displayed on **all items**, the clearance for the Track will be shown, and the clearances for the via will be shown when the via is within the distance defined in the '**Refine within distance**' section of **Options**, **Online DRC**.

	Eaiting Options	•
	Segment Mode	
	Change Segments	•
~	Online DRC	X
~	Continuous Online DRC	
~	Display Clearance	
~	Allow Checking of Multiple Items	
	Push Tracks	_
	Change Grid	•

This option works for End Track On Via, End Track On Testpoint Via and End Track On Testpoint Pad.

Dimensions Changes

Hide Arrows on a Directional Dimension

You can now hide the arrows on a Directional Orthogonal Dimension. A new check box, **Directional Type: Show Arrows**, on the **Defaults Dimension** page, allows show or hide arrows.

Layer: Silkscreen Top ~	Horizontal and Vertical:
Text Style: Names 🗸	Text Angle: 90 Degrees Adjust Vertical
Arrows:	Text Avoids Arrow Line
Line Style: Line (5) 🗸	Keep Above or to the Left
Head Width Multiplier: 4.000000	Directional Type: Show Arrows
Head Length Multiplier: 8.000000	Free Angled:

When you add a new directional dimension, the arrows will be shown depending on this setting.

Show Arrows checked

27.94mm	0.00mm
	T
Shows Arrows unchecked	
27.94mm	0.00mm ¶

If you select a directional dimension in the design, and use **Properties**, the **Dimension Arrows** page now includes a **Show Directional Dimension Arrows** check box that you can use to show or hide the arrows for the selected dimensions.

	Properties: Linear Dimension Arrows					
	Dimension Arrows Doc Lines Text Text Style Tolerance					
	Text Gap: 0.0000					
	Arrows Offset: 23.6220 Force Point Inwards					
	Arrow Head Size: Width Multiplier: 4.000000					
	Length Multiplier: 8.000000					
7	Arrow Line Style:					
	Shele: Line (5)					

Linear and Radial Dimensions – Show both Metric and Imperial Units

Linear and Radial dimensions can display both Metric and Imperial units for the same dimension.



Additional check boxes to Show Alternative Length Units and Show Alternative Radial Length Units have been added to the Defaults Dimension Units page.

When selected, the alternative units text is added after the normal dimension distance text, enclosed in the Prefix and Postfix text, for example **4141 thou (22.61 mm)**. Brackets and formatting can be added using **Prefix** and **Postfix** entries.

Angle Units:		
Degrees	⊖Radians	
Precision:	1	
Prefix:	Unit Text: deg	
Length Units:		Show Alternative Length Units:
O Imperial:	thou \sim	● Imperial: thou
Metric:	mm v	OMetric: mm ∨
Precision:	2	Precision: 1
Prefix:	Unit Text: mm	Prefix: Unit Text: thou
		Postfix:
Radial Length	Units:	Show Alternative Radial Length Units:
O Imperial:	thou \sim	● Imperial: thou ∨
Metric:	mm v	OMetric: mm ∨
Precision:	2	Precision: 1
Prefix:	Unit Text: mm	Prefix: Unit Text: thou
		Postfix:
Decimal Poin	t Character:	

If you select a linear or radial dimension in the design, and use **Properties**, the **Dimension** page now includes a **Show Alternative Units** check box that you can use to change the alternative units displayed for the selected dimensions.

There is a new section to accommodate this:

Properties: Linear Dimen	ision Dimension	— 🗆 X
Dimension Arrows Doc	c Lines Text Text Style Tolerand	ce
Dimension Type:	Units Units Units Units Units Metric: mm Vrecision: 3 Vrefix: Units Text: mm Decimal Point Character: .	Show Alternative Units Imperial: thou Metric: cm Precision: 1 Prefix: Units Text: thou Postfix:
Measurement: 27 940		4

STEP 3D Changes

Minimise STEP File Size Option in STEP Output

There is a new option to **Minimise STEP File Size** available on the new **Output** page of the **STEP 3D Settings**. This reduces the size of the generated STEP file by omitting non-essential parametric curve data. This is set on by default.

3D View 9	Settings –	Output					×
Settings	Colours	Interaction	Enclosures	View	Output	Layers	
🗹 Mini	mise STEF	^o file size					

Improved Trihedron Axes Indicator for STEP Preview & Models

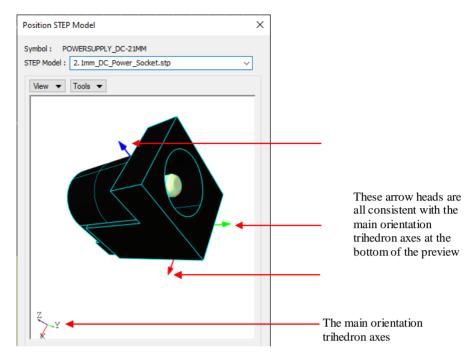
The old axis indicator 'nodules' have been replaced with more standard trihedron axes. This displays arrows designating the positive direction away from the origin on that axis, with the red, green, and blue colours matching the axes.



As well as the main 3D Preview window, the trihedron axes have also been added to **3D View Settings, Enclosures (New** and **Edit** buttons) and **Library – STEP Models**.

STEP Model Trihedron View

The model trihedron axes (in addition to the overall trihedron axes) is now also displayed in the **Footprint editor** and **3D Settings Enclosures**, **Position STEP Model**. The lines and spheres in previous releases have been replaced with colour-coordinated arrow heads, to represent the direction the model is pointing away from origin.



Added ability to 'Align' STEP items

In the **STEP Preview**, a new **Align>** mode can be selected from the **Tools** menu. You can select any two shapes, faces, edges or vertices (or any combination thereof) and the first selected item will instantly snap to meet the second. This allows you to select an item in the 3D Preview and align a sub-selection of that shape with a sub-selection of any other shape, such that they're inline on that axis.

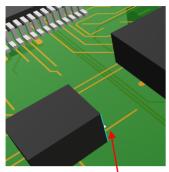
Four new modes are available to select from the Align> option: Snap To, Align by X, Align by Y and Align by Z

When any of these modes are selected, the Spacing threshold can now be declared prior to aligning on an axis or snapping to, meaning that the two points selected will be exactly that distance apart after the operation. A dialog is presented to choose the **Snap to Spacing** value. The dialog is pre-set with the default **Spacing Rule** value from the **PCB design**.

Snap To S	pacing
Amount:	0.000 🔺 mm
	ОК

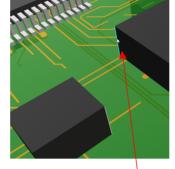
The **Snap To** option is an interactive mode where the first face is selected and the item snapped to the second selected object.

This will move the STEP model (along with the Component), into a new position chosen. This positional change will be reflected in the PCB design with a reposition of the Component.

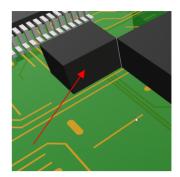


The images below show the process for alignment:

Select the first vertices



Select the second vertices

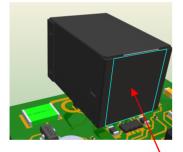


The first shape will move to align

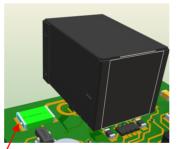
Added ability to 'Orient STEP items

In the **STEP Preview**, with **Orient** mode selected from the **Tools** menu, you can select any two faces of any 3D Items (including two of the same item), and the first face, along with the attached shape, will **rotate** around to face in the same direction as the second face.

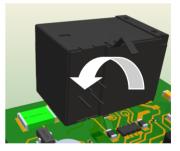
This will move the STEP model (along with the Component), into a new position chosen. This positional change will be reflected in the PCB design with a reposition of the Component.



Select the first vertices or face



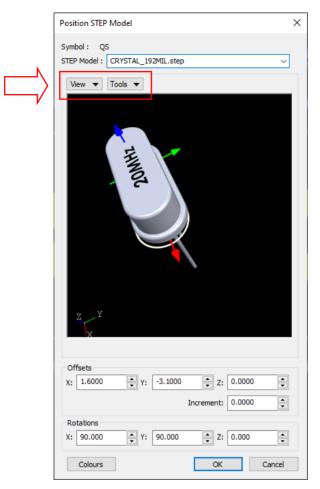
Select the second vertices, the edge of the capacitor in this example



The first shape will rotate to match the capacitor

View and Alignment options added to 'Position STEP Model' dialog

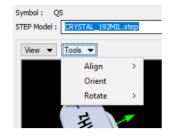
View orientation and Tools options have been added to Position STEP Model dialog on the Tools menu when editing a Footprint.



The drop-down **View** menu enables you to select one of the preset views available. These are the same as the views available on the existing **3D View Orientations** toolbar.

Symbol: QS	
STEP Model :	CRYSTAL_192MIL.step
View 🔻	Tools 🔻
Axon	ometric View
Front	t View
Back	View
Left \	/iew
Right	t View
Top \	/iew
Botto	om View

The drop-down **Tools** menu will enable you to interactively position and align your STEP model with the footprint. Options for **Align**, **Orient** and **Rotate** are available. These are same as described above for the main **STEP Preview** window.



Design Units now used

Another change on this dialog is that all units shown will now be current design units and not forced to mm (Metric) as they were previously.

Import Mounting Holes and Vias into design from STEP Model

The **Import STEP Board** dialog, available from the **File** menu and **Import Design Data on a STEP file**) has been extended to provide additional fields to provide max sizes of **Mounting Holes** and **Vias** seen in the STEP model being imported to the design data. When a circle is discovered that is smaller than the specified mounting hole diameter, the circular element will be removed from the shape and a mounting hole will be created in its place, or a via if it is small enough for that. It assumes that mounting holes are bigger than via holes so it does the tests that way round.

	Import from STEP	×
	✓ Import Board Board Face Direction	
	® XY O XZ O YZ	
	Max Mounting Hole Diameter: 0.0	
7	Max Via Diameter: 0.0	
	Import Placement Sites Attribute Name: Placement Site	

Import STEP Board Placement Sites

The **Import STEP Board** dialog, available from the **File** menu and **Import Design Data on a STEP file**) has been extended to provide additional options to allow the location of items in the STEP data to be turned into **Placement Site attribute positions** in the design. These will only be added if they are inside a board outline in the design. The STEP files have an object location catalog for the topmost elements, components like connectors, switches and other objects that the mechanical department preplaces on the board for placemen of those objects.

Additional controls allow you to choose the **Attribute Name** to use, the **Text Style**, and the **Top** and **Bottom Layers** for the attribute positions that are added to mark placement sites. The Top layer is used for STEP file locations where $Z \ge 0$, and the Bottom layer for Z < 0.

Choose from the Attribute Names listed in the drop-down list, or enter a new attribute name.

	Import from STEP			Х
	✓ Import Board Board Face Dire	ection O XZ	Оyz	
	Max Mounting Ho Max Via Diameter		0.0	
	Import Placemen	t Sites		
	Attribute Name:	Placement Site	~	
	Text Style:	Names	~	
	Layer Top:	Silkscreen Top	~	
Y	Layer Bottom:	Silkscreen Bott	om 🗸	
	🗹 Only Within Bo	ard		
	ОК	Car	ncel	

You can also use the **Only Within Board** check box to choose whether or not to keep any positions that are outside all board outlines.

3D Package Viewer Retirement

The old **3D** package viewer (not associated with viewing STEP models) is no longer available on the menu. This has been retired and removed along with **Packages** feature from the **Library Manager** and **3D Settings** options from the **Setup** menu. If you wish to continue using this feature, you can add it to your menus using the **Customise** option or use the **Run Command** option or assign the command to a shortcut key.

Line Select Mode

With nothing selected in the design, right click and the new **Line Select** selection mechanism will be available, if something is already selected it will be on the Select sub folder. Use this to define a straight line to select only items that cross that line.

Enable Select Mask	
Frame Select	
Line Select	
Polygon Select	
Select Path	
Select All Visible	Ctrl+A
Select All	
Auto Select	
Reverse Layer Order	Alt+P
Paste	Ctrl+V
Renosition Design	
	Frame Select Line Select Polygon Select Select Path Select All Visible Select All Auto Select Reverse Layer Order Paste

Once selected from the menu, draw the line across the items you wish to select.



The selection will then be made:

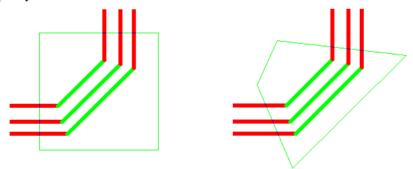


Like **Frame Select**, keep the **Ctrl** key pressed down to alter the selection. Use the **Shift** key to extend the selection to whole items.

Line Select is also available from the context menu in **Align** mode and **Latched** operations such as Move when nothing is initially selected.

Changes to Frame Select and Polygon Select Modes

Both **Frame Select** and **Polygon Select** modes will now act on individual track segments to make segment picking more versatile. This will operate with the option in **Options**, **Select** page, **Select if Completely Framed** is set to on or off.



Changes to Double Click to Edit Mitre

Double click on a mitre has been changed so that it will enter the **Edit Mitre** mode regardless of its mitre size. The **Edit Mitre** mode is also available on the **context** menu for a selected mitre.

For adjusting multiple mitred corners, such as parallel tracking, this change works in conjunction with the three selection modes described above (Line, Frame and Polygon).

In Version 10.5, it was changed to work on double-click if it was less than twice the default mitre size as defined in the Options dialog and Edit Track option. This change has been reversed in Version 11.

Insert PCB Track more responsive

Snapping to a line orthogonal with the target (item at end of connection, or point between the two connections if a Differential Pair) is now responsive and easier to snap to, retaining the pad centre to the orthogonal track.

You can now also avoid snapping to the Pad centre by holding the down the **Shift** key whilst moving the track.

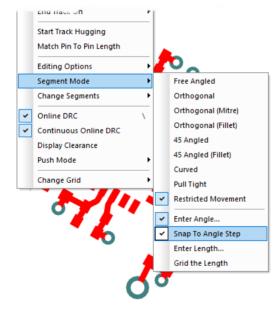
This feature was back-fitted to 10.5

Restricted Movement Segment Mode – Snap To Angle Step

When editing a Track or shape in **Restricted Movement Segment** mode and the restricted angle is not zero, you can now change how the angle is snapped to.

Once routing in **Restricted Movement Segment** mode, from the **Segment Mode** context menu, there is a new option **Snap To Angle Step**.

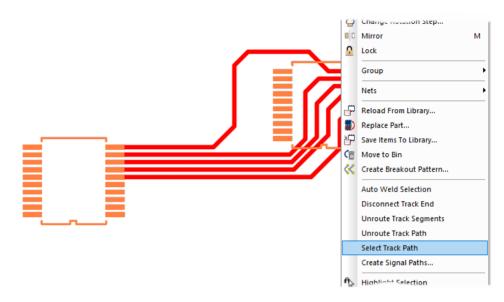
This option will only appear if the **Snap Angle** is a multiple of the restricted movement angle. Unselect this option to snap to any 45-degree step relative to the restricted movement angle (as it worked previously). Using this new mode, you can snap to any multiple of 30 degrees. Previously, it would only snap to the angle entered and any 45-degree step from that angle (30, 75, 120 ...).



This feature was back-fitted to 10.5.

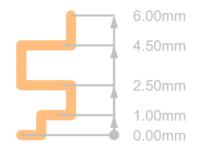
Select Track Paths by selecting Components

In **Select Mode**, if 2 or more components are selected, if you right click, the context menu displays **Select Track Path**. Selecting this, the current selection will be deselected and all track paths directly between the previously selected components will be selected.



Copy/Paste into new Shape Type

You can now **Copy** and **Paste** a single track as a generic Doc Shape. This feature can be accessed when using **Insert Doc Shape** and then using **Paste** before starting to draw a shape. The selected, and copied track shape will be pasted in as a doc shape.



Layer Control Added To Change Shape Type

You can now change the **Layer** in the **Change Shape Type** dialog using the **Layers** drop down list box. Previously, a Layer change was only possible using a different option or dialog

	Change Sha	аре Туре		×
	Old Type:	Template		OK
	New <u>T</u> ype:	Area	\sim	Cancel
\Box	Layer:	Тор	~	

Pad Auto Necking

A new rule page has been added to **Technology DFM/DFT rules** called **Pad Track Neck**. This specifically for Pad necking into and out of pads.

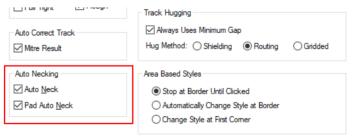
This rule allows you to define a **Pad Track Percentage** which is the percent that the necked track width will be in comparison to the pad width (for example, if the pad width is 50 and the percentage is 80% then the necked tracks width will be 40).

	Enable	Attribute Name	Ma	tch Value	Pad Tr	rack Percentage	Min Le	ngth
	\checkmark	<net name=""></net>	ż	8	80		100.0000	
Attribute:	<net< td=""><td>Name></td><td>~</td><td></td><td></td><td></td><td></td><td></td></net<>	Name>	~					
Match:	•		~ 🕺			>	Min Length:	100.0000
					-			
				Pad Track Percer	ntage:	80		
Global Pa	d Track I	Neck Rules:						
🗸 Alway	vs Create	Style with Correct Width						

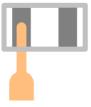
The **Min Length** option defines the length of the necked segment. This length is measured from the edge of the pad to the tip of the '**fat**' track. The completion shape also takes this new calculation of the length into account when creating a necked completion shape.

The **Global Pad Track Neck Rules** at the bottom section includes the option to **Always Create Style With Correct Width**. With this option disabled, when the necked track is created and if the style is not already in the design, a message will display asking if you want to add the style. If the option is enabled, then the style will be added without warning.

For this rule to be operative, you must have the option from the **Options** dialog, **Edit Track** page and **Auto Neck** enabled. It can also be enabled in the editing options section of the context menu when editing a PCB track.



When operating, it will look like this when you click to make a corner:



If the check box to **Always Create Style With Correct Width** is enabled and a new Track Style needs to be added, a warning dialog is presented:

Pulsonix		×
?	Add new track style to the design? PadNeck1	
	Yes <u>N</u> o	

The new Track Style will be created at the percentage width to that of the pad it is exiting.

Auto Necking

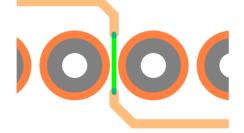
You can now neck to and from pads or through gaps that would normally cause a DRC error. It can be used when routing anywhere in the design and will neck down the current segment to avoid DRC errors. Once avoided, it then reverts to the regular style after passing the items that would cause the error, you can then continue routing.

This mode can be used by either clicking directly on the pad or using the completion shape. When using the completion shape for pad auto necking, it will show the necked segment at the end of the completion shape.

You can enable this option from the **Options** dialog, **Edit Track** and by selecting **Auto Neck**. It can also be enabled in the editing options section of the context menu when editing a PCB track.

Auto Correct Track	Track Hugging ☑ Always Uses Minimum Gap Hug Method: ○ Shielding
Auto Necking ☑ Auto <u>N</u> eck ☑ Pad Auto <u>N</u> eck	Area Based Styles Stop at Border Until Clicked Automatically Change Style at Border Change Style at First Corner

When editing a track near a pad a potential violation takes place, the necking will be performed:



Add Item to Net in Net Properties Dialog for Copper and Templates

You can now use Properties to add an item to a net, for example a Copper shape or Template.

Segment	Shape Line Style Copper Attributes Net Net Attribute	
<u>N</u> ame:	Choose From All Nets In Design: <no net=""></no>	
Net <u>C</u> lass:		
Net <u>T</u> ype:	Signal V	
Guard Spacing:	0.0000	

An item not on a net will show as <No Net>. Changing this to a net name will add it to the selected net. Clearing the net name will add it to a default net, but it will ask you first.

If multiple items are on different nets it will now show as <Different> (whereas, it used to be blank). By typing a single net name will merge the nets to one net (after asking about merge).

If multiple nets or multiple no nets, and the net name is cleared, after asking if you want default net names, you will be asked if you want to merge all nets into one default net, or to create multiple nets. (Note: In Schematics, you will not be asked and individual default nets will always be created).

Add Copper & Template Shapes to Net on Insert

Whilst adding **Copper** or **Template** shapes to your design, you can now add them to a net using the **Add To Net** option from the context menu option.

<u></u>		
0	Cancel Edit Polygon	
	Finish Here	
	Type Coordinate	=
	Type Offset	Shift+=
	Change Layer	L
	Change Style	s
	Add To Net	F2
	Snap To Item	
	Editing Options	+
	Segment Mode	•
	Change Segments	•
	Change Grid	•

This can be done either before you start the first point, or whilst adding the shape. If you have a similar item preselected the net will be taken from it (previously, this worked for templates), but it has been changed so that if you preselect a different type of item (a track for example) the net will be taken from that item and a warning will be issued to ask if you want to use that net.

Cancel Edit Polygon	
Finish Here	
Type Coordinate	=
Type Offset	Shift+=
Change Layer	L
Change Style	S
Change Net	F2
Snap To Item	
Editing Options	,
Segment Mode	,
Change Segments	,
Change Grid	,

If whilst adding the shape you wish to change the net name or remove it from the net, use the **Change Net** option from the context menu and edit the name or remove it in the dialog.

This feature was back-fitted to 10.5.

Non-Connecting Copper Highlight Colour

A new category for Non-Connecting Copper has been added in PCB Colours under Highlights.

The **Non Connecting** switch on **Technology**, **Copper Pour Rules** and **Copper Shape Properties**, causes isolated copper areas to be marked as Non Connecting. This means that it can remain isolated and does not form part of the connectivity, although it does still retain the same net name. You may require this to maximise copper on the board.

Name	Displayed	Colour
Attached Dimensions/Callouts		
Branch Point Via		
Bus Tracks		
Clearances		
Component Pad 1		
Differential Pair Path		
Differential Paired Tracks		
Highlight		
Highlight 'Fail'		
Highlight 'Pass'		
Highlight 'Unchecked'		
Highlight "Warning"		
Locked Track Segments		
Marked Net		
Non Connecting Copper		
Not Filled	\leq	
Online DRC		
Selection		

Reversed View Status Saved

When **Reversed View** is enabled in your design, this status is now saved when **Save** or **Save** As are used, this means the design will remain reversed when re-opened.

Attributes on Wires

You can now add attributes to **Wires** in the design using **Properties** or **Add Attribute**. These can be used to hold copper wire material or the wire diameter for example.

🔳 Pro	operties: Wire – V	Vire Attı	ributes			×
Wire	Wire Attributes	Net	Net Attributes			
	colour=Black Diameter=0.5mm ength=8mm Part Number=209-4 Supplier=RS				<u>N</u> ew <u>E</u> dit <u>D</u> elete	

These attributes can then be used in the Find bar to select all wires of the same diameter for example.

You can also use the **Report Maker** option to report wire links and their attributes in a bill of materials, along with the standard Wire attributes that already exist (such as Length and Insulated status).

This feature was back-fitted to 10.5

Layers Bar – Components Filter

From within the **Filter** section of the **Layers Bar**, you can now toggle the **Show** and **Pick** status of **Components**:

Layers		:	×
- [J]			
Paste Ma	sk Top ssk Top d sk Bottom sk Bottom n Bottom tom s (Bottom) tation	^	
		~	
Hide <<	S Areas:	ihow Pick	k A
	Components:		ין ר
ı Pir	Connections: Constructs: Copper: Pads: Routing: Templates: Name in Pads: Locked Items:	K K K K K K K K K K K K K K K K K K K	

Toggle Layers Changes

Alias Assignments

Layer Toggle commands can now have an **Alias** name assigned; this allows for more readable/shorthand descriptions of the command. These names can be defined in the **Layer Toggle Setup** dialog.

Layer	Toggle Setu	р				
Toggle Layer	Alias		Layer to be Toggled On/Off	^	Layer Toggle Builder: Layer Definition:	
1	Тор	S	iow Top Electrical, Show Silkscreen Top		Keyword: Show	_
2						
•					Layer Name:	La
4					~	
5 6						
,					Side: Top	

Layer Toggle Alisa names can be viewed in the new Toggle Layer Bar (see below).

New Toggle Layers Bar

A new dockable **Toggle Layers Bar** has been added. This allows you to activate any of your defined layer toggles on the fly, while in design.

Toggle Layers	×
- (#	
Toggle Top Toggle Bottom Show Electrical Top, Bottom	
Setup	

The **Setup** button will take you to the **Setup Toggle Layers** dialog from where you can define the Toggle Layers.

If you right click on the Toggle Layers Bar, a context menu is available:

	Floating	
~	Docking	
	Tabbed Document	
	Auto Hide	
	Hide	
	View Toggle Numbers	
	View Toggle Aliases	
	View Toggle Shortcuts	
	Refresh List	
	Set Toggle Layers	
	World View W	

Three active settings enable you to display additional information if defined in the Toggle Layers dialog on the Toggle Layers Bar: **View Toggle Numbers**, **View Toggle Aliases** and **View Toggle Shortcuts**. The **Set Toggle Layers** switch will take you to where these can be defined.

New Toggle Plane Command

A new command has been added to toggle the display of Powerplanes on and off.

If you have a plane layer called **Ground** then you would create a toggle **plane ground**, this will toggle the Powerplane display state of that layer in the same way that you get when clicking on the little thermal icon on the layer in the **Layers Bar**.

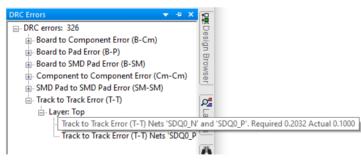
Measure Bar showing Spacing Rule

The Measure Bar now displays the rule that corresponds to the calculated spacing.

X: 0.0000
Y: -6.2865
Angle: 270.0
Spacing: 1.2700
Rule: Design Spacing Rules
Units: Design Snap to Item

Error Bar - Tooltip available on Error Rule

When an error marker is displayed in the **Errors Bar**, the rule violated can be displayed in a tooltip by hovering the mouse over the marker. This is available when the full error name cannot be displayed.



Placement Vector - Include Power/Ground Net

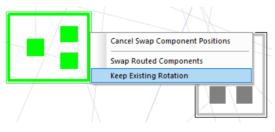
New check boxes in **Options** and **Move** allows you to select whether **Power** and/or **Ground Nets** are to be included in the calculation of the most optimal point for Component placement using the placement **Force Vector**. Power and Ground nets are defined by their **Net Type** (Power or Ground).

– Design Backups – Display – Edit Shape – Edit Track – File Extensions – Find	Auto Weld ☐ Auto Weld On Drop ☑ Allow Weld To Split Tracks ☑ Show Weld Spots	Component Pushing Push Mode: Never Push Direction: Both Spring Back
General In-Place Names Interaction Macros <mark>Move</mark>	Snap To Construction Lines O Within Set Distance: 0.0000 Within Grid Steps: 2	Show Force Vector Show As: O Long Arrow Compass
Multi-Screen Pan & Zoom Online DRC Resolve Net Names	Optimise:	Include Power Net Include Ground Net Dynamic Align

Unchecking either of these boxes will change the dynamics of the force Vector. Unless you specifically require these nets to be included on the calculation, we suggest you deselect these.

Keep Existing Rotation in Swap Component Positions

In the **Utilities** menu, **Swap Component Positions** tool, there is a new feature on the context menu available during use of this option. You can now **Keep Existing Rotation** if enabled, this will allow swapped components to keep their original rotations but still have their positions changed. If disabled, swap component positions will be completed as before (rotations will be changed to match swapped component).



Spread-Out Option Changes

The **Spread Out** utility, available from the **Utilities** menu, now has a companion **Spread Out Manually** dialog, which is accessible from the context menu. This allows for more specific changes to the dimensions and spread distance, plus refinements to spread parameters.

			Spread Out Manually X
_		1	Method of Spreading: Centre Equispace Spread Dut
	Cancel Spread Out		Spread Direction:
Ŷ	Type Coordinate =		 Linear Spread (Angle 45)
	Spread Out Manually		 Linear Spread (Angle 90) Radial Spread
×	Deselect All		
	Centre		Origin: X: 0.0
~	Equispace		Y: 0.0
\Leftrightarrow	Spread Out	V	
~	Linear Spread (Angle 45)		Distance: Angle: 360°/0° V
	Linear Spread (Angle 90)		
	Radial Spread		0.0
~	Snap To Item		
~	Snap To Arc Centre		Online DRC Position Item Centres
~	Snap To Line Middle Or End		Grid Items After Spreading
~	Online DRC \		Align Items
	Position Item Centres		Between Item Edges
	Grid Items After Spreading Shift+G		
	Align Items		<u>U</u> nits: Design <mark>⊡</mark> Snap to Item
	Between Item Edges		
~	Equispace Outer Items		<u>O</u> k <u>C</u> ancel

Auto Insert Testpoints - Include Unreachable Testpoints

From the **Tools** menu and **Auto Insert Testpoints** dialog, there is now an option to include previously placed unreachable testpoints.

Automatic Testpoint Insertion Number of Testpoints Testpoints Per Net: Add To Unconnected Pads	Selection Filter Net Class: Net Name:	 Select Deselect
Testpoint Type Make existing pads Testpoints where possible Madd Testpoints using Doc Symbol Symbol Star Point (25) {DocSymbols} Mattempt To Place Place Attempt To Route Place Around Board	Matched Unmatched Net DRIVE High_speed HS HS4	Required Actual 1 0 1 0 1 0 1 0 1 0
Delete Testpoints Report OK Cancel	Ignore Power and Ground Nets	Indude Unreachabl

Excluding these would mean that previously placed testpoints, which are unreachable according to the current rules, would not be included in the count, so additional testpoints would be required.

Create Breakout Pattern (BGA Fanout) in PCB

Within the PCB design, there is now a command available in the **Tools** menu called **Create Breakout Pattern.** This is used to create breakout (fanout) patterns within the design without the need to create a Breakout pattern in the footprint first. To use the option, select a component or a pad on the component and use the **Create Breakout Pattern** option on the context menu. A dialog is displayed from which to choose the pattern required:

Create Breakout Pattern	n X				
Offset ✓ Radiate from centre × 0.5 ✓ Y 0.5 ✓					
	Apply to Selected Pads Only				
Track Style					
<u>N</u> ame: Track (6)	~				
<u>W</u> idth: 0.15240					
Via Style:					
<u>N</u> ame: Via (18)	~				
<u>W</u> idth: 0.45720	S <u>h</u> ape: Round ~				
Length: 0.00000	<u>D</u> rill: 0.25400				
Plated					

The dialog will only create a pattern for components that have at least one pad that is on a net and does not already have a routed connection. Applying a pattern will not delete or edit any existing tracks that are attached to pads on the component (no changes will be made to these pads).

When applying a pattern, a DRC check will be performed. Selecting **No** will remove the inserted pattern.

The command for this option has changed from **Create footprint breakout pattern** to **Create breakout pattern**. You will need to reassign this if you have already assigned it to a shortcut key.

Another small change is that you are no longer able to use this option if the selected component in your PCB design already has breakouts.

CAM Plot Changes

Multi-Threading of Gerber Plots

Gerber Plots can now be completed using multiple threads (including Gerber verification plots) when running CAM Plots.

The selection to **Enable Threads** for **CAM Plots Gerber export** has been added to the **Options** dialog, **General** page. This setting is used by all Multi-threading technology in Pulsonix.

Excellon Setup Min Drill Diameter

In the **CAM Plot** dialog, **Excellon Setup** page, there is a new setting to define the **Min Tool Diameter**.

This is very useful for the output of **Pad Slots** that have special or intricate shapes. This option defines the smallest valid tool diameter that will be used for these shapes. These would be shapes that are not regular slots that can be milled using a standard drill. When used to route custom shapes, no attempt is made to shrink these shapes so the drill should be as small as possible and in accordance with what your manufacturer can mill.

Bight (X.Y): 254.000	Cancel
0.000	
Units Format:	
Units: millimetres Type: Absolute	
Omit Zeroes: None	
Decimal: 5	
	Units: millimetres Type: Absolute Omit Zeroes: None Integer: 3

Optional Formatting information for Excellon output

In the **CAM Plot** dialog, **Excellon Setup** page, you can include optional formatting information at the start of the Excellon file.

Drilling Area	Units: mm 🗸 🛛 OK
Lower Left (X,Y):	Upper <u>Bight (X.Y):</u> 254.000 254.000 254.000
Registration Point Auto Shift 🗸 🗸	0.000
Options:	Units Format:
	Units: millimetres
Format Type: Format 1 Format 2	Type: Absolute Omit Zeroes: None Integer: 3 Decimal: 5
Format 1 Format 2	Type: Absolute Omit Zeroes: None Integer: 3 Decimal: 5
Format 1 Format 2 Include FMAT format statem Include FILE_FORMAT co	Type: Absolute Omit Zeroes: None Integer: 3 Decimal: 5
Format 1 Format 2 Include FMAT format statem Include FILE_FORMAT co Include FILE_FORMAT co	Type: Absolute Omit Zeroes: None Integer: 3 Decimal: 5
Format 1 Format 2 Include FMAT format statem Include FILE_FORMAT co	Type: Absolute Omit Zeroes: None Integer: 3 Decimal: 5

Include FILE_FORMAT comments includes some comments, at the start of the file, of the form ;FILE_FORMAT=2:5 (where 2:5 is the integer:decimal places format), ,INC is you are using incremental, and ,LZ is you are using leading zeros. You also get a comment of the form ;TYPE=PLATED or ;TYPE=UNPLATED when appropriate.

Include Format after Units includes, LZ (for leading zeros) or TZ (for trailing zeros) followed by the number format shown as 00.00000 for 2.5 (e.g. **METRIC,LZ,00.00000**). This is only relevant if you are using leading or trailing zeros.

Output file names template - Use Default button

A new button, **Use Default**, has been added to the **Output File Names Template** section of the **Plot Settings** tab in the **CAM Plot** dialog. This returns the template to its default value which is:

Output Device:	Folder For Output Files:
Gerber V Setup	Design Folder \checkmark
New Delete	Plots
Save To File Load From File	Pack output files into a ZIP Browse
	Remove individual files after ZIP
Gerber Plotting area: 0.000, 0.000 to 297.000, 210.000 mm Registration Point: Auto Shift Hardware arcs: No Hardware fill: No Extended Fill:Yes Include portures: Yes Include format: Yes Include mode:Yes Include:Yes Includ	Output File Names Template: \$(DesignName)\$(PRE)-\$(Variant)\$(PRE)(\$(PlotName))\$(POST) Use Default Change Output File Name:

Output to SVG Device

Using the **Output** menu and **CAM Plot** dialog, you can now output an **SVG file**. The SVG file will be an image of the design using the standard rules selected for content and scale. This is available for both Schematic and PCB designs.

The SVG output is available on the list of 'devices' on the drop-down list:

Start Process	Choose a name for this plot and choose the type of output					
Output	Define the name which Also choose the type of	will be used to identify this plot in dialogs and reports. foutput.				
Design Position	Name:	SVG				
Finish	Plot Group: Plot String:	<u> </u>				
	Output To:	SVC Gerber HP-GL Penplot Windows PDF Excellon				
		PDF/A Format File ODB++ LPKF IPC-2581 GenCAD STEP				
		IDF DXF UniDAT				



									_
Name	Enabled	Group	Device	Process	Scale	Rotate	Mirror	Position	
Top Electrical			Gerber	Layer Top Electrical	1.000	Auto Rotate		Auto Shift	<
Bottom Electrical			Gerber	Layer Bottom Electrical	1.000	Auto Rotate		Auto Shift	<
Top Silk Screen			PDF	Layer Silkscreen Top	1.000	Auto Rotate		Auto Shift	<
Bottom Silk Screen			PDF	Layer Silkscreen Bottom	1.000	Auto Rotate		Auto Shift	<
<through hole=""></through>			Excellon	Layer Span <through hole=""></through>	1.000	Auto Rotate		Auto Shift	<
SVG	\checkmark		SVG	Layer Top Electrical	1.000	Auto Rotate		Auto Shift	<
				Layer Silkscreen Top					

Output to ZIP File

You can now pack all the generated output files (from the CAM Plot option) into a ZIP file.

From the CAM Plots dialog, on the Plot Settings page, there are three new check boxes:

Output Device:	Folder For Output Files:
Gerber ~ Setup	Design Folder V
New Delete Save To File Load From File	Pack output files into a ZIP Remove individual files after ZIP
Gerber Plotting area: 0,000,0.000 to 297,000, 210,000 mm Registration Point: Auto Shift Hardware Till: Yes Extended Fill: Yes Include apertures: Yes Include mode: Yes Include mode: Yes Include mode: Piss Include mode name (IN): No	Output File Names Template: \$(DesignName)\$(PRE)-\$(Variant)\$(PRE)(\$(PlotName))\$(POST) Change Output File Name:
Include image offset (0F): No Include image polarity (IP): No Include image polarity (IP): No Include layer/level polarity (IP): Yes Include X2 Aperture Function :Yes Include X2 Aperture Function :Yes Include X2 Are function :Yes Include X2 Are function :Yes Include X2 Part Command :Yes	Plot Report Image: Create A Report Image: Wiew When Run Image: Save Wth Output Files Image: A Report To Existing Image: Save Wth Output Files Image: A Report To Existing Image: Save To Reports Folder Image: A Report To Existing Image: Save To Reports Folder Image: A Report To Existing Image: Save To Reports Folder Image: A Report To Existing Image: Save To Reports Folder

Check **Pack output files into a ZIP** to create a ZIP of all the generated output files in the folder specified in the **Folder for output files** section.

Check **Remove individual files after ZIP** to delete all the individual output files created, leaving you with the ZIP file only.

In the **Plot Report** section, check **Include in ZIP** under the **Plot Report** section to also add the CAM Plot report to the ZIP file.

Panel Editor – Reset Layout Command

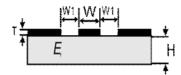
When in the **Panel Editor**, there is a new command **Reset Layout**... This is available on the context menu dialog and is used to re-apply **Panel Gap/Spacing** between PCBs.

Enter Pan	el Dimensions &	Gap Amour	nt X
Layout:	3	rows	OK
	4	≑ cols	Cancel
Gap: X:	10.000000	mm	
Y:	-170.020000	mm	

Track Impedance Calculator

Differential Pair (CPWG) impedance calculations have been added to the **Design Calculators**, the CPW and CPWG options also allow for calculation of the conductor space.

Coplanar Waveguide



Coplanar Waveguide with Ground



Edge Coupled Coplanar Waveguide with ground



Design Calculators - Values taken from design

The **Design Calculator** dialogs will now take more parameters from the selected track or via. The dialogs will indicate which values have come from the design by using **bold** text.

Heat Sink		Basic RLC	F		Convert
Scientific Tra	ck Width and Res	istance	Track Imped	ance	Via Resistance
Calculate: Required Tr	ack Width	\checkmark			Hide
Inputs: Track Layer:	External) Internal			
Track Thickness:	0	oz (per sq. f	foot) 🗸		
Current:	1.2	Amps			
Temperature Rise:	10	Celsius (C)	\sim		
Ambient Temperature:	10	Celsius (C)	\sim		
Track Length:	4.053	mm	\sim		
	-	Ohm-cm	Use Default		

New Material Parameters

New parameters for **Electrical Conductivity** and **Dielectric Constant** defined in the **Materials** dialog are used by calculators.

Also added is the ability to specify a **Material** for special use **Hole Plating** and **Micro-Via Hole Plating**. This is selected in the **Materials** dialog:

Name: Copper 1oz		
Used: 🗹		
<u>T</u> hickness:	0.0350	
Electrical Conductivity:		S/m
Dielectric Constant (Er):		
<u>S</u> pecial Use:	<none> 🗸 🗸</none>	
	< <u>None></u> Hole Plating Micro-Via Hole Plating	

Track Impedance Calculator – Calculating Impedance Track Geometry

In on-the-fly mode, you can use the context menu to choose the **Geometry** to use, or choose **Auto Detect Track Geometry** to try and choose the correct geometry for the track that has been picked (by looking at layers stack and if it is buried in a plane).

The Differential Pair check box will be checked (or not) depending on if the picked track is a paired track.

Relative Permittivity

Will be taken from the **Dielectric Constant** value on the **Material** used by the dielectric layer above or below the layer of the picked track (if it is defined).

Dielectric Thickness and Track To Plane Gap

These will be taken from the **thickness** of the **dielectric layer** above and below the layer of the picked track (if it is defined).

Track Spacing

When picking a Differential Paired track, the **Differential Pair Gap** will be used for the track spacing.

Conductor Space

When picking a track, the **Track to Copper Spacing** for that track and at that point will be used for the conductor space for Coplanar Waveguide calculations.

Track Thickness

Will be taken from the thickness of the layer of the picked track (if it is defined).

Track Impedance Calculator – Calculating Track Width

Differential Pairs

The Impedance value provided as an input will be the **Differential Impedance** when calculating the required track width for a Differential Pair. The last field in the result will be the **Characteristic Impedance**.

Take Impedance Values from design

The **Characteristic Impedance** will be taken from the **Track Impedance Rules** for the net of the picked track, or to get the Differential Impedance from a Differential Pair (or one of the nets if not on the pair),

Track Width & Resistance Calculator

Copper Resistivity

This will be calculated from the **Electrical Conductivity** value on the **Material** used by the layer of the picked track (if it is defined).

Via Resistance Calculator

Via Depth

This will be calculated from the **thickness** of the **layers** that the via passes through (if they are defined).

Plating Thickness

This will be taken from the **thickness** of the **Material** that has special use **Hole Plating** or **Micro-Via Hole Plating**, dependent on the type of via selected. (if it is defined).

Resistivity

This will be calculated from the **Electrical Conductivity** value on the **Material** that has special use **Hole Plating** or **Micro-Via Hole Plating**, dependent on the type of via selected. (if it is defined).

Interactive High Speed Option – New Rules & Changes

Back Drilling

The **Back Drilling** functionality is a feature of the **Pulsonix Interactive High Speed option**, it is only available with this license.

Back Drilling, is a process used to remove the unused stub of drill holes from a through-hole when creation high-speed PCBs. This technique ensures signal stubs are minimised (which can be a source of impedance discontinuities and signal reflections). With increased data rates this aids the reduction of signal distortion. This is alternative method to using blind and buried vias which are costly to manufacture.

The process of Back Drilling

In order to utilise Back Drilling, the summary below informs you of the general process:

- Identify a requirement for it, then:
- Add Back Drill Spans to accommodate Back Drilling in your Technology
- Create Back Drilling Rules in your Technology
- Route your design, vias and spans that match the Back Drilling rules will be created
- Export the Back Drilled Vias to the Excellon NC Drilling format using the CAM Plot option You can also export a user report created using the Report Maker.

In addition to the above, new features have also been added for Design Rules Checking, exporting to various other formats such as IPC-2581, GenCAD, ODB++ etc. and user defined reports using the new commands within the Report Maker.

Technology Back Drilling Spans

A **Back Drill Span** page has been added to the **Technology** under **Layers**. These spans define the layers of a via that need to be Back Drilled. The start layer of the span is either **Top** side or **Bottom** side, and the end layer will always be an inner layer. All layer spans have to have unique name; a **Layer Span** cannot have the same name as a **Back Drill Span**.

					Depth		New
		Name	Start Layer	Stop Layer	Use Layer Thicknesses	Depth	
	Y	Top - Inner BD	<top side=""></top>	<inner></inner>		2.4-	-
	Y	Bottom - Inner2 BD	<bottom side=""></bottom>	Inner2		3.7+	
							Delete
							Delete Unused
							Up
							Down
							Only Show Used Entries
							Auto Gen View Layers
	-						Type: Back Drill Span Bottom
Name:	Botto	om - Inner2 BD			Use Layer Thickr	nesses	туре, васк они эран вошон
Used:	\checkmark			Back Drill	Depth: 3.7+		
Start Laye	er: <	Bottom Side>	\sim				
Stop Laye	er: In	nner2	\sim				

The **Back Drill Depth** is the depth of the oversized drill hole after Back Drilling. This value takes into account the **Back Drill Stub Length** (taken from the **Back Drill Rules** page), and is measured from the top or bottom of the board to the inner electrical 'Stop' layer of the Back Drill span (this does not include the thickness of the stop layer). It is calculated from the layer thicknesses, or can be specified as an explicit value by unchecking this option and typing the value required. If you have specified layers thicknesses, you are likely to want to use these.

If the **Use layer thickness** box is unchecked, you can define your own thickness for the span. When the drill depth is <undefined>, the calculated drill depth is less than zero. The most likely cause of this is when the defined **Back Drill Stub Length** being too large.

The **View Layers** button will display a cross sectional view of the layer stack the same as the **Layers** dialog.

Auto Generating Back Drilling Spans

The **Auto Gen** button on this page enables you to automatically generate a list of Back Drill spans that are usable by your current design. These will be generated from the rules defined in the **Back Drill Rules**. The grid is filled with Back Drill spans that can be used in the design.

🖪 Auto	o Generate Ba	ck Drill Span	5		>
	Include	Name	Start Layer	Stop Layer	
1		Back Drill 1	<bottom side<="" td=""><td>Inner2</td><td></td></bottom>	Inner2	
2		Back Drill 2	<top side=""></top>	Inner2	
3		Back Drill 3	<bottom side<="" td=""><td><inner></inner></td><td></td></bottom>	<inner></inner>	

The check box for whether you **Include** then or not include them enables Back Drill Spans rules used to be overridden if not required.

Technology Back Drilling Rules

The **Back Drill Span** rules page in the **Technology Layers** dialog allows you to define the nets that will be Back Drilled (or not if excluded from the list). A net will match with the first relevant rule in the list, so the order of the rules is important. As with all rules, wildcard matches are permitted.

Enable	Attribute Name	Match Value	Back Drill	Comp. Pads	Mounting Holes
	<net name=""></net>	DQS*	\checkmark		
Attribut Match			clude:]Vias]Component Pa]Mounting Hole		
Drill C 5 0 / • F	Back Drill Rules: Diversize: Absolute Size Percent of Drill Size Length: ance: 0.01		Stub Length	1	

Use the **Include** check box options to select the **Vias**, **Component Pads** or **Mounting Holes** that will be Back Drilled. The **Vias** check box is always selected but Components and Mounting Holes can be optionally added if required.

The **Global Back Drill Rules** at the bottom of the dialog let you define a **Drill Oversize** for the larger drill used for all Back Drilling in the design.

Drill Oversize will be the increase in the drill radius compared to the vias' defined **Drill Size**, or as a **Percentage** increase in radius compared to the vias' drill size.

You also define the remaining **Stub Length** required after Back Drilling has been performed. This is your safety margin to absolutely ensure the via stop pad required isn't drilled out.

The sizes and values for **Drill Oversize** and **Stub Length** will be supplied by your board manufacturer, it is important to define these very carefully to avoid errors.

Identifying Back Drilled Vias in your Design

Back Drilled vias have a circle drawn around them (with the same diameter as the oversized drill) and a curved line going through the pad.

The colour of the circle is dependent on whether it is a Top Back Drill (same colour as Top layer) or a Bottom Back Drill (same colour as Bottom layer), with the curved line being the same colour as the Stop layer of the Back Drill. The curved line will be horizontal for a Top Back Drill and vertical for a Bottom Back Drill. If the via has both a Top and Bottom Back Drill span, then both shapes are drawn together.



Back Drill Colours

The colours for the three Back Drill types can be set using a new tab in PCB Colours, called **Back Drill Spans** and by selecting the relevant **Back Drill Span** in the list.

Back Drill	Displayed	Colour
Top > Inner 3		
Top > Inner 4		
Bottom > Inner 6		
Bottom > Inner 7	\checkmark	

Update Back Drills

A new feature named **Update Back Drills** has been added to the **Utilities** menu. Use this option to check and assign the most appropriate Back Drill spans to the vias that are on Back Drilled nets and that currently don't have this status.

This option is also automatically run when the CAM Plot option is initially run.

Back Drilled vias can be checked for using new checks in the **DRC** option, see below.

Back Drill Spacings

Back Drills are taken into account when doing both Design Rules Checking and Online DRC.

A new Drill spacing has been added to the **Design Level Spacing Rules** on the **Drill** tab called **Back Drill to Copper Items**. This is the spacing between Back Drills and other items such as Pads, Vias, Copper etc.

The 'general' **Drill to Drill** spacing handles Back Drill to Back Drill spacing, and Drill to Board handles Back Drills to Board spacing.

Additiona	I Design Level Spacir	ngs and Rules:						
Board	Bond And Die Pad	Component	Copper	Drill	Pad	Track	Enclose	ures
	I to Drill Space Minimum: 0.254 Allow Coincident Ho Only If Same S			cro-via Dri Minimum: Stagger: ☑ Allow	0.000)	as	
Drill to Board Space Minimum: 0.000				ck Drill To 1inimum:	Copper 0.254			

Back Drilling DRC

The Design Rules Check dialog now has a Back Drill check box in the Spacing section.

This checks the spacing between Back Drills and other items using the **Back Drill To Copper Items** spacing defined in your Technology. Back Drill to Board and Back Drill to Drill is checked in the **Drills** Spacing.

🗹 Spacing		Manufacturing
	🗹 Back Drill	🖂 Acid Traps
	🗹 Board	Bond Wire Le
	🗹 Components	Component N
	🗹 Copper	🗹 Copper Shape
	🗹 Drills	🖂 Copper Text C
	🗹 Mount Holes	Drill Backoff

A new check has been added to the **DRC** under the **Manufacturing** section called **Incorrect Back Drills**. This check will report an error if a Via Back Drill span is currently incorrect (checked against the **Back Drill Rules**) and requires updating using the **Update Back Drills** option.

Design Rule Check							
Spacing	Manufacturing	Nets					
🗹 Back Drill	Acid Traps	Ad					
🗹 Board	Bond Wire Length	Co					
Components	Component Name	Co					
Copper	Copper Shapes	Cre					
🗹 Drills	Copper Text On Board	Dif					
🗹 Mount Holes	Drill Backoff	Ne					
🗹 Pads	Footprint Rules	Ne					
🗹 Split Planes	Incorrect Back Drills						
🗹 Test Points	Isolated Copper	Pa					
🗹 Text	Minimum Pad Land	Pir					

Running this will also check if there are any Back Drill spans that have a negative drill depth. This happens when the **Global Back Drill Stub Length** is greater than the depth of the span itself (as defined in the **Back Drill Rules** page).

Via & Component Pad Properties

The **Via & Component Pad Properties** dialogs now has a **Back Drill** tab which shows the used Back Drill spans. Checking and unchecking the option allows you to enable or disable Back Drilling for that specific via. The Back Drill spans cannot be changed inside Via/Pad Properties.

Properties: Via Back Drill						×	
Via	Back Drill	Branch Point Net	Net Attributes				
Back Drilling Enabled							
<u>T</u> op Span:		Top > Inner2 BD					
Bottom Span:							

Back Drilling Pin Depth Attribute for Component Pads

A new attribute on a Component pin called **Pin Depth** has been added. This allows you to define the depth of an individual pin so that you will know the maximum depth allowed for Back Drilling.

A new attribute has been added called **Part Pin Depths** that allows you to define all the pin depths for that specific Component. This means by setting this attribute on a Part, it will not need to be set on every pin of the footprint. This will override any value assigned using the **Pin Depth** attribute.

If a Back Drill rule defined includes Component Pads, neither of these attributes are used.

Layer Span Layers automatically included in Layer Stack Preview

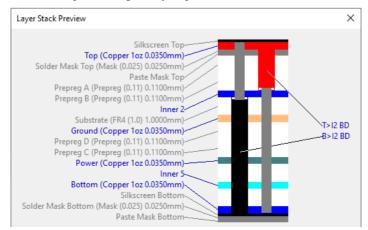
The **From Layer** or **To Layer** of a Layer Span, or the **Stop Layer** of a Back Drill span is now included in the **layer stack preview**. For the applicable layer where these are used, the **In Layer Stack Preview** check box on the **Technology Layers** page will be checked and disabled (greyed out).

	סטונטווו		Electrical	DULLUIII	T	Copper 102	0.0000	Used Entries
Y		Silkscreen Bottom	Silkscreen	Bottom	None		0.0000	
		Solder Mask Bottom	Solder Mask	Bottom	None	Mask (0.025)	0.0250	Physical Copper
		Paste Mask Bottom	Paste Mask	Bottom	None		0.0000	Layers: 6
Y		Pin Names (Bottom)	Non-Electrical	Bottom	None		0.0000	
	Board		Board		None		0.0000	Board Thickness
	Construction Lines		Documentation		None		0.0000	1.6300
	Documentation		Documentation		None		0.0000	
	Drill Drawing		Drill Drawing	<through hol<="" td=""><td>None</td><td></td><td>0.0000</td><td></td></through>	None		0.0000	
	me: Inner 5			Electrical				
Us	ed:			Routing	<u>B</u> ias: X	~		
Cla	ass: Electrical		✓ New Class		Power Plane N <u>e</u> t:			
	Type: Electric	al Physic	al Copper Layer			~		
E.	ayer Association: —			Construct	ion Details:			
0	Can Have <u>A</u> ssocia	-		Materia	:	~	Usually Plotte	
(Top Facing	<u>S</u> ide: Inn	er 🗸			New Material	🗸 In Layer Stad	x Preview
	Bottom Facing (M	irror Components)		Thic	ckness: 0.0000		Suppress Unconr	nected Lands:
1.5				1	10 KI	1	On Pads	

Inserted Layer Stack Previews

Both the Layer Stack Preview inserted into the design and the View Layers option (in Technology, Layers, Layer Spans and Back Drill Spans) now show Back Drills.

In the Technology, the **View Layers** button works the same on the Layers and Layer Spans pages (shows layers and regular layer spans) but on the Back Drill span page, it will show the layers along with the Back Drill spans, no regular layer spans are shown.



On the **Insert Layer Stack Preview** dialog, there is now an **Include In Preview** option. This defines which spans are included in the preview – **All spans**, **No Back Drill spans**, or **Only Back Drill spans**.

Insert Layer Stack P	review		×
Show Table On Laya	r: Silkscreen Top	~	OK
Include In Preview:	All Spans	\sim	Cancel
	All Spans		
	No Back Drill Spans Only Back Drill Spans		

This can also be changed later by changing the newly added include option in **Symbol Properties** for the selected **layer stack preview**.

📧 Propert	ies: Symbol — 🗆 🗡	<
Text Style	Line Style Symbol Attributes	
Position: Angle:	35.5600 131.8260 0.0 Mirrored Scale:	
Symbol:	Layer Stack Preview Change	
Layer:	Silkscreen Top 🗸	
Include:	All Spans All Spans No Back Drill Spans Only Back Drill Spans	

When changing the text box fill colour of a **Callout** on a **Layer Stack Preview**, the outline of the span will also be in the selected colour – this was changed to compliment the Back Drill span shape in layer stack preview.

CAM Plot Wizard

In the **CAM Plot Wizard**, you can now select **Back Drill spans** as a process, allowing for the creation of drill files and drill drawings.

The insert drill table dialog now allows for Back Drill span drill tables. For values to be correct the Back Drill span must be used by a via, and a drill oversize must be defined.

Pad Exceptions for Back Drilling

When creating a pad exception, Back Drill Top and Back Drill Bottom **Exception Types** can be used. This allows for a pad exception to be added on a via that will be Back Drilled, on layer top or bottom depending on the exception type.

The Exception Type allows you to select Back Drill Top or Bottom from the list:

Name:	Shape:
Via (18)	Type: Ro
Named by: Typed Rule Template	Width: 0.4
Exception Type:	Length: 0.4
By Layer 🗸 🗸	
By Layer By Layer Class Inner Electrical Spacing Shape Micro-via Entry Pad Micro-via Dop Pad	Offset: 0.0

Excellon Report Chanegs

The Back Drill **depth** and Back Drill **side** are both output in the Excellon report. The drill output will always be unplated.

```
Excellon Drill Output
Output File: C:\Designs\ExcellonPlots\Job66(Excellon).drl
```

	Plot Name:	Excellon
	Variant:	<master design=""></master>
/	Contains:	Bottom >Inner 2 BD Unplated
	Side:	Bottom
<u> </u>	Drill Depth:	0.0500
	Drill Type:	Unplated
	Scale:	1.000
	Mirrored:	No
	Rotated:	No
	Offset:	-41.9040 -65.1905mm
	Step & Repeat:	Not used

IPC-2581 and ODB++ Exports include Back Drills

Back Drills are now included in IPC-2581 and ODB++ exports. No extra selection is required, this is automatically done for you.

Back Drilling Commands in the Report Maker

New commands have been added so that Back Drilled Vias can be reported and/or added as a user report to the design to display manufacturing information about them.

For more detailed information, see section below Report Maker Changes.

Find Bar

You can use the **Find Bar** to locate any vias that currently have a Back Drill span. When searching for vias, check the **Layer Span** box at the bottom of the bar, and select a Back Drill span from the drop down list.

- []	x,y 汭 🖺 ٵ	1	C
Via			•
DRIVE			
<search enti<="" td=""><td>e design></td><td></td><td></td></search>	e design>		
DRIVE			
Hide Filter «	< Apply		
🔄 Filter Nam	es		
×			
Attribute:			
s: matching			`
			9
			1
🗹 Layer Spa			

Track Impedance Rules

A new rule in the **Technology** under **High Speed** rules, **Track Impedance**, enables you to assign **Single-Ended Track Impedance** and/or **Differential Track Impedance** (also called Edge-Coupled Impedance) to nets or Differential Pairs on defined layers, and optionally within named areas. The **Design Calculator** will search the rules for the impedance for a selected track.

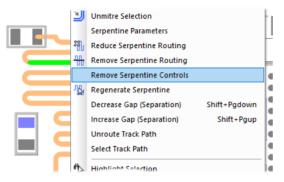
These rules will be set up when **Importing an IPC2581 Layer Stack** and used by the **Design Calculators** and **Track Impedance Calculator** (when calculating Track Width).

When using this dialog, the **Attribute** selection is made using the drop-down list where a net or attribute can be defined, this also include <Differential Pair Names>.

	Enable	Attribute Name	Match Value	Side	Layer	Area	Single Ended Impedance	Differenti Impedan		
		<differential name="" pair=""></differential>	DQS*				50.000000	90.000000		
)ifferential QS*	Pair Name> ~	**				Differential Imp	edance: 9	90.000000	Ohms
On Layers Side: Layer:	<any></any>		~			Er			Er	
Within Areas:			 ✓ Sing 	le Ended In	npedance:	50.000000	Ohms			

Remove Serpentine Controls Option

You can now remove the controls on a selected Serpentine track using the **Remove Serpentine Controls** option from the context menu. This reverts it to a normal track that can be edited in the usual way. Once this is done, it cannot be converted back to a controlled Serpentine (although Reduce and Remove Serpentine routing commands may still work as Pulsonix recognises serpentine patterns).



Rules Spreadsheet Multi-threading

The **Rules Spreadsheet** can now be filled using multiple threads when not using the **Only show** selected or **Dynamic item** options. This will speed up filling for designs with large nets or where the design contains many nets.

The selection to **Enable Threads** for the **Rules Spreadsheet** has been added to the **Options** dialog, **General** page. This setting is used by all Multi-threading technology in Pulsonix.

Create Differential Pairs From Rules

You can now create Differential Pairs in a Schematic or PCB design by setting up and then applying new **Differential Pair Nets** rules, defined in the **High Speed Rules** section of the **Technology** dialog. This enables you to create a rule or rules that capture the net names in your design to apply to nets that should become **Differential Pairs** without having to do this process manually.

Each rule defines a pair of net match strings that are used to find two nets that match the strings where the common part of the name is the same.

The **Attribute** value, Net Name for example, will be the **Match** criteria. You can set this to a specific match, such as SDQ0* or can be more generic, such as * for the whole name.

Once the attribute Match has been satisfied, the rule then uses the Differential Pair Nets rule to make the pairing. For example, rule *_P and *_N will match nets **SDQ0_P** and **SDQ0_N**. When two nets are matched, the rule defines what name the Differential Pair will take, whether an attribute should be added to use when allocating length rules, and some default parameters for the pair.

Γ				Differentia	I Pair Nets	Match	Within		Edge				Add Differentia	I Pair Attribute
	Enable	Attribute Name	Match Value	Net 1 Match	Net 2 Match	Item Type	Name Match	Template String	Coupled	Broadside	Allow Spurs	Include All	Attribute Name	Attribute Value
P		<net name=""></net>	DQS_*	*P	*N	Area	<none></none>	<netcommonname></netcommonname>	\checkmark					
ſ	\checkmark	<net name=""></net>	LCO_D*	*1	*2	Area	<none></none>	<netcommonname></netcommonname>	\checkmark					
ľ	\checkmark	<net name=""></net>	SDI_In*	*_1	*_2	Area	<none></none>	<netcommonname></netcommonname>						
Г		<net name=""></net>	SDQ0*	*_P	*_N	Area	<none></none>	<netcommonname></netcommonname>						

Attribute: <net name=""> ~</net>		
Match: SDI_In* 🗸 🔊		
Differential Pair Nets: Net 1 Match: *_1 v	Differential Pair Name: <netcommonname> Field: Common part of the net names v Insert Match Separator: v</netcommonname>	New Differential Pair Values:
Match Within: Item Type: Area V Name Match: None> V	Add Differential Pair Attribute:	Edge Coupled: Broadside: Allow Track Spurs: Include All:

Attribute

A rule defines two sets of nets. Each set is defined using a net **Attribute Name** and a **Match** value. The Match Value can be wildcard. In particular, you can match the inbuilt attributes **Net Class Name** or **Net Name**. When these checks are done, each rule is applied, so the order of these rules is important.

Match

The Match value, Net Name for example, will be the Match criteria use for the rule. You can set this to a specific match, such as SDQ0* or can be more generic, such as *. Only nets matching this value will be considered for a Differential Pair.

Differential Pair Nets

Once the **Attribute Match** has been satisfied, the rule then uses the Differential Pair Nets rule to make the pairing. For example, rule *_P and *_N will match nets SDQ0_P and SDQ0_N. When two nets are matched, the rule defines what name the Differential Pair will take, whether an attribute should be added to use when allocating length rules, and additional default parameters for the pair.

The Net 1 Match

This is used to match the net defining one 'half' of the Differential Pair net. It is matched along with the Attribute Name. The Match Value can be a wildcarded string. For example, if the Attribute name matches SDQ0, the net 1 match might be *_P so will match SDQ0_P.

The Net 2 Match

This is used to match the net defining the second 'half' of the Differential Pair net. It is also matched along with the Attribute Name. The Match Value can be a wildcarded string. For example, if the Attribute name matches nets SDQ0*, the net 2 match might be *_N so will match SDQ0_N.

Match Within (for a PCB design)

You can also choose to only match nets if they have pads within a certain area. The **Item Type** will show **Area** or <None> when used within a PCB design. The **Name Match** can be an area name selected from the drop-down list or it can be left blank so the rule is applied to the whole design. The Match Value can be wildcarded.

Differential Pair Name

This box shows you how the Differential Pair Name will be constructed if you let the system create one for you. This will be based on the Field and Match Separator fields below. This is an editable box

and can be constructed from the Fields selected form the drop-down list. At any point, you can also use the **Delete** key on your keyboard to removed fields if not required.

The **Field** drop-down has a list of possible contents from which to create the Differential Pair Name. Possible contents available are:

Common part of the net names - use this to name the Differential Pair with the common part of the two net names. If the Attribute Match string contains a stem before the wildcard * you can choose a separator to be used after that stem in the resultant name. If this is required, type the **Match Separator** or select it from the drop-down list. For example, if the Attribute Match string is SDQ* and *_P and *_N are used to match nets SDQ0_P and SDQ0_N, then the common part of the names will be SDQ0, and if the Match Separator is "_" it inserted after the stem to give a Differential Pair name of SDQ_0.

First net name

Second Net Name

Net Attribute - select the net attribute name from the list presented. The value will be extracted from the first net used to define the Differential Pair.

Component Attribute - select the component attribute name from the list presented. The value will be extracted from the first of the pins used to define the Differential Pair.

The **Insert** button takes the Field (and Match Separator if selected) and passes them into the Differential Pair Name to show you how it will be constructed in the rule when used. Multiple fields can be selected from the list and inserted. The insertion will depend on your cursor position in the Name field above.

Add Differential Pair Attribute

Use this to define an attribute to be added to the newly created Differential Pairs using this rule. For example, to assign an attribute to be used to allocate a Differential Pair Gap or Skew rule. Leave the fields blank if no attribute is to be added.

If required, select the **General Use** or **Net Attribute Name** from the drop-down list, and type the Value to be added. If some design items already have attributes using this attribute name their values can be selected from the Value drop-down list.

New Differential Pair Values

You can define how the tracks are paired. **Edge Coupled** are the usual Differential Pairs, the edges of the two tracks separated by the specified gap. **Broadside** are paired vertically, the tracks laid on top of each other on different layers. It is possible to allow both types of pairing on a Differential Pair.

Allow Track Spurs

Normally, the path between the pins in a Differential Pair should be without any spurs or branches to other pads, vias, etc. Checking the **Allow Track Spurs** option will allow spurs from the track path. These spurs must not have any further spurs or branches and should be terminated on a pad, via or testpoint. These will be checked as part of the **Differential Pair Design Rule Check** and in the Differential Pair report.

Include All

Normally pins are not included in the path between the pins at the ends of the Differential Pair, but sometimes this is required. Checking the **Include All** box will instruct **Optimise Nets** to connect to any extra pins on the net that are in this path and not assigned to other Differential Pairs, allowing you to route them into the path without a Differential Pair error.

Match Within Schematic

If setting the **Match Within** value from a Schematic, the drop-down list will also display additional settings for **Block** and **Block Instance**.

In this case, Block Instance Name is the name of an instance of a block in the design. Block Name is the name of a block used by the instance; several block instances may use the same named block. The nets may be across the design but the reason for the area is only pins defined in that area, block, or block instance will be considered when creating the Differential Pairs.

Attribute: <net na<="" th=""><th></th><th>1</th><th></th><th></th><th></th></net>		1			
Match:	~ *				
⊂ Differential Pair Net Net 1 Match: *_F Net 2 Match: *_1	P ~	 × × × 	Differential Pair Name: Field: Common part of the net names Match Separator:	✓ Insert✓	- New Diff∉ - Tracks
Match Within:					E
Item Type:	ea 🗸 🗸	*	Add Differential Pair Attribute:		
Are Name Match: Blo	ock	*	Attribute:	~	Allow
Blo	ock Instance		Value:	~	Inclu

Create Pairs From Rules Dialog

When the rules have been set up, use the new **Create Pairs From Rules** button on the **Technology**, **Differential Pairs** page. You will be presented with a dialog that informs you which new pairs will be added and which existing pairs will be renamed or have an attribute added due to the rules. Here you can choose to not add or rename a Differential Pair by unchecking it in the list, or can type their own name for the pair.

The New Chain dialog will not create a Differential Pair if the nets do not have enough nodes.

		First Pin Pai	r		Second Pin	<u>N</u> ew
Chain Link Name	Net	Start Pin	End Pin	Net	Start Pin	New Chair
1	DQS_P	Q4.1	Q5.1	DQS_N	Q4.2	
	SDQ0_N	U5.1	U6.2	SDQ0_P	U5.2	
	DQS N	U11.T4	Q5.2	DQS P	U11.T5	Delete
	Chain Link Name	Net DQS_P SDQ0_N	Net Start Pin DQS_P Q4.1 SDQ0_N U5.1	Net Start Pin End Pin DQS_P Q4.1 Q5.1 SDQ0_N U5.1 U6.2	Net Start Pin End Pin Net DQS_P Q4.1 Q5.1 DQS_N SDQ0_N U5.1 U6.2 SDQ0_P	Net Start Pin End Pin Net Start Pin DQS_P Q4.1 Q5.1 DQS_N Q4.2 SDQ0_N U5.1 U6.2 SDQ0_P U5.2

In the Technology dialog, if you select an existing Differential Pair Name or add a new one and it matches a rule that would give a different name or would add an attribute, a button will appear next to the **Name** field to use to apply the rule. This will change its name and/or add an attribute.

reate	Name	Net 1	Net 2	Add or Change Attri	bute	
	HMIB_12V0_AD0	HMIB 12V0 AD0 P	HMIB 12V0 AD0 N			
	HMIB_REFCLK1_C	HMIB_REFCLK1_C_P	HMIB_REFCLK1_C_N	1		
	HMIB_REFCLK2_C	HMIB_REFCLK2_C_P	HMIB_REFCLK2_C_N			
	HMIB_REFCLK3_C	HMIB_REFCLK3_C_P	HMIB_REFCLK3_C_N			
	HMIB_REFCLK4_C	HMIB_REFCLK4_C_P	HMIB_REFCLK4_C_N			
	HMIB_REFCLK5_C	HMIB_REFCLK5_C_P	HMIB_REFCLK5_C_N			
	HMIB_REFCLK6_C	HMIB_REFCLK6_C_P	HMIB_REFCLK6_C_N			
\square	HMIB_REFCLK7_C	HMIB_REFCLK7_C_P	HMIB_REFCLK7_C_N			
	HMIB REFCLK8 C	HMIB_REFCLK8_C_P	HMIB_REFCLK8_C_N			
\checkmark						
ential pai	HMIB REFCLK9 C rs that already exist, but will b Old Name	New Name	Net 1	Net 2	Add Attribute	
ential pai	HMIB REFCLK9 C	e renamed or have an attrib	oute added or changed: 65	Net 2	Add Attribute	
ential pai	HMIB REFCLK9 C Is that already exist, but will b OId Name DIFF100_HMIB_ETH1_A DIFF100_HMIB_ETH1_B	e renamed or have an attrib New Name HMIB_ETH1_A HMIB_ETH1_B	Nute added or changed: 65 Net 1 HMIB_ETH1_A_N HMIB_ETH1_B_N	HMIB_ETH1_A_P HMIB_ETH1_B_P	Add Attribute	
ential pai	HMIB REFCLK9 C rs that already exist, but will b Old Name DIFF100_HMIB_ETH1_A DIFF100_HMIB_ETH1_B DIFF100_HMIB_ETH1_C	e renamed or have an attrib New Name HMIB_ETH1_A HMIB_ETH1_B HMIB_ETH1_C	Mute added or changed: 65 Net 1 HMIB_ETH1_A_N HMIB_ETH1_B_N HMIB_ETH1_C_N	HMIB_ETH1_A_P HMIB_ETH1_B_P HMIB_ETH1_C_P	Add Attribute	
ential pai	HMIB REFCLK9 C Is that already exist, but will b OId Name DIFF100_HMIB_ETH1_A DIFF100_HMIB_ETH1_B	e renamed or have an attrib New Name HMIB_ETH1_A HMIB_ETH1_B HMIB_ETH1_C HMIB_ETH1_D	Nute added or changed: 65 Net 1 HMIB_ETH1_A_N HMIB_ETH1_B_N	HMIB_ETH1_A_P HMIB_ETH1_B_P	Add Attribute	
ential pai	HMIB REFCLK9 C rs that already exist, but will b OId Name DIFF100_HMIB_ETH1_A DIFF100_HMIB_ETH1_C DIFF100_HMIB_ETH1_D DIFF100_HMIB_M2_PER1	e renamed or have an attrib New Name HMIB_ETH1_A HMIB_ETH1_B HMIB_ETH1_C HMIB_ETH1_O HMIB_M2_PER1	Net 1 HMB_ETH1_A_N HMB_ETH1_B_N HMB_ETH1_C_N HMB_ETH1_C_N HMB_ETH1_D_N HMB_ETH1_D_N	HMB_ETH1_A_P HMB_ETH1_B_P HMB_ETH1_C_P HMB_ETH1_D_P HMB_M2_PER1_N	Add Attribute	
iname	HMIB REFCLK9 C Is that already exist, but will b OId Name DIFF100_HMIB_ETH1_A DIFF100_HMIB_ETH1_C DIFF100_HMIB_ETH1_C DIFF100_HMIB_TH1_C DIFF100_HMIB_M2_PER1 DIFF100_HMIB_M2_PER2	e renamed or have an attrib New Name HMIB_ETH1_A HMIB_ETH1_B HMIB_ETH1_C HMIB_ETH1_D HMIB_M2_PER1 HMIB_M2_PER2	Nute added or changed: 65 Net 1 HMB_ETH1_A_N HMB_ETH1_B_N HMB_ETH1_C_N HMB_ETH1_D_N	HMIB_ETH1_A_P HMIB_ETH1_B_P HMIB_ETH1_C_P HMIB_ETH1_D_P	Add Attribute	
ntial pai	HMIB REFCLK9 C rs that already exist, but will b OId Name DIFF100_HMIB_ETH1_A DIFF100_HMIB_ETH1_C DIFF100_HMIB_M2_PER1 DIFF100_HMIB_M2_PER2 DIFF100_HMIB_M2_PER3 DIFF100_HMIB_M2_PER3	e renamed or have an attrib New Name HMIB_ETH1_A HMIB_ETH1_B HMIB_ETH1_C HMIB_M2_PER1 HMIB_M2_PER2 HMIB_M2_PER3	Net 1 HMIB_ETH1_A_N HMIB_ETH1_B_N HMIB_ETH1_C_N HMIB_ETH1_D_N HMIB_ZPER1_P HMIB_VEPER2_P HMIB_VEPER3_P	HMIB_ETH1_A_P HMIB_ETH1_B_P HMIB_ETH1_C_P HMIB_ETH1_C_P HMIB_M2_PER1_N HMIB_M2_PER2_N HMIB_M2_PER3_N	Add Attribute	
ential pai	HMIB REFCLK9 C rs that already exist, but will b DIFF100_HMIB_ETH1_A DIFF100_HMIB_ETH1_B DIFF100_HMIB_ETH1_C DIFF100_HMIB_M2_PER1 DIFF100_HMIB_M2_PER3 DIFF100_HMIB_M2_PER3 DIFF100_HMIB_M2_PER3 DIFF100_HMIB_M2_PER3	e renamed or have an attrib New Name HMIB_ETH1_A HMIB_ETH1_B HMIB_ETH1_C HMIB_ETH1_D HMIB_M2_PER1 HMIB_M2_PER2 HMIB_M2_PER3 HMIB_M2_PET1	Net 1 HMB_ETH1_A_N HMB_ETH1_B_N HMB_ETH1_C_N HMB_ETH1_D_N HMB_ETH1_D_N HMB_RETH1_D_N HMB_M2_PER1_P HMB_M2_PER2_P HMB_M2_PER3_P HMB_M2_PET1_P HMB_M2_PER3_P HMB_M2_PET1_P	HMIB_ETH1_A_P HMIB_ETH1_B_P HMIB_ETH1_C_P HMIB_ETH1_O_P HMIB_M2_PER1_N HMIB_M2_PER2_N HMIB_M2_PER3_N HMIB_M2_PET1_N	Add Attribute	
ntial pai	HMIB REFCLK9 C rs that already exist, but will b OId Name DIFF100_HMIB_ETH1_A DIFF100_HMIB_ETH1_C DIFF100_HMIB_M2_PER1 DIFF100_HMIB_M2_PER2 DIFF100_HMIB_M2_PER3 DIFF100_HMIB_M2_PER3	e renamed or have an attrib New Name HMIB_ETH1_A HMIB_ETH1_B HMIB_ETH1_C HMIB_M2_PER1 HMIB_M2_PER2 HMIB_M2_PER3	Net 1 HMIB_ETH1_A_N HMIB_ETH1_B_N HMIB_ETH1_C_N HMIB_ETH1_D_N HMIB_ZPER1_P HMIB_VEPER2_P HMIB_VEPER3_P	HMIB_ETH1_A_P HMIB_ETH1_B_P HMIB_ETH1_C_P HMIB_ETH1_C_P HMIB_M2_PER1_N HMIB_M2_PER2_N HMIB_M2_PER3_N	Add Attribute	

The dialog is split into two sections; Differential Pairs that will be created and Differential Pairs that already exist, but will be renamed or have an attribute added or changed.

Differential Pairs that will be created

The number next to this header informs you of the number of potential new Differential Pairs that will be added to your design if all the rules available are applied. The main list is a summary of all the Differential Pair names and their paired nets. By default, the **Create** check box next to each name will be selected (so that all new Differential Pair Names required will be added). If the new name is blank then the name will be the default name generated from the four pins names. Uncheck this box if you do not wish to create a new pair in your design. If you do this, care should be taken to ensure you haven't missed a pair that you defined in the rules.

This list is applied when the **OK** button is pressed at the bottom of the dialog.

Differential Pair that already exist

This reports any matches between the rules defined and the Differential Paired nets already in the design. The number displayed next to the header is a match based on the rules defined. Using the Rename check boxes, you can select which existing differential Pairs to rename. Again, care should be observed if you uncheck any of these as it means they match rules defined but you are choosing to ignore them.

This list is applied when the **OK** button is pressed at the bottom of the dialog.

Summary boxes

Three summary boxes at the bottom of the dialog report an overall position to you based on the check boxes selected in the grids above, any rules that it fails to match based on the Differential Pair Rules defined and a count of the number of rules that were not matched.

Number of Differential Pairs already created that match the rules, shows the number of Differential Pairs which match your defined rules and that exist in your design. This is shown as a reference for you.

Number of Differential Pairs already created that do not match the rules, shows how many Differential Pairs that are currently in your design that do not match any rules. This is a very useful check to ensure that if you do want to cover all Differential Pairs using rules, then this should read zero. If the number 1 or more is shown then there are Pairs that are not covered using a rule.

The **Number of rules that were not matched** shows that Differential Pairs rules have been created but there are no matches in the design to satisfy this.

Use the **Report** button to report listing the Differential Pairs that already matched, did not match at all, and the rules that were not matched (with an explanation of why).

Reapplying Rules

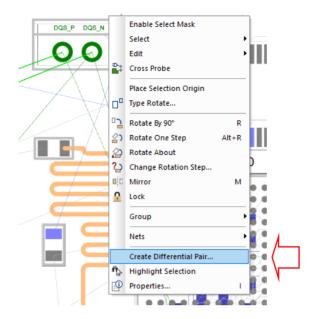
A new button on the **Differential Pairs Nets** dialog next to the **Diff Pair Name** enables you to **Rename to Apply Rule** to match any new rules defined in the **Differential Pair Net Rules** dialog.

Diff Pair Name: CLK_CORE Rename To M.	Edge Coupled:
	Colour: Connections Broadside:
Pin: C242.1	Pin: IC1.AF1 Include All:
	Swap Pin Pairs
Pin: C241.1 V	Pin: IC1.AG1 \lor
Swap Pins	Rules Attributes
Second Pin Pair: Net: CLK_COREN ~	Differential Pair Skew Rules: <differential name="" pair="">=* [Min A Differential Pair Gap Rules: <differential name="" pair="">=* Outer Differential Pair Gap Rules: <differential name="" pair="">=* Inner I Edit</differential></differential></differential>
	C Delete

Create Differential Pair using Context Menu

You can now select two nets in a PCB or Schematic design and from the context menu use **Create Differential Pair** to create the 'pair'. Once these nets are selected, it will open up the **Differential Pair Rules** dialog in your **Technology** with these nets pre-selected and ready to define.

If four pads are selected, they will be used for the pair, otherwise the pads will be automatically allocated. You will not see this command on the context menu if the nets already form a Differential Pair, or don't have at least two pins on each net.

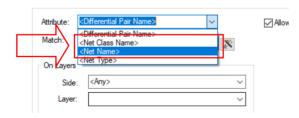


Define Differential Pair Rules using Net and Net Class Attribute

You can now define **Differential Pair Gap** and **Skew** rules with a **Net or Net Class attribute**. The rule is only used on a Differential Pairs where both (paired) nets match the attribute. For example, defined for a specific Net Class name.

Previously, it was only the Differential Pair name or Differential Pair attribute that could be used, now you can assign a Net attribute and it will check both nets for a match.

Note: Differential Pair Skew rules defined this way will not be used on Chains (which can have multiple nets), only on Differential Pairs.



This feature was back-fitted to 10.5

Create Differential Pair Chain

The **New Chain** dialog on the **Differential Pairs** dialog has changed to make it easier to choose the correct Differential Pairs for a chain.

Choose Differential Pairs To Add To The Chain		×
Differential Pairs in the Design:		Differential Pairs in the Chain:
Q7.2-Q6.2 Q7.1-Q6.1 SDQ0_	<u>A</u> dd >>	DQS_ Q5.1-U11.T5 Q5.2-U11.T4
	<< Dejete	
	Up	
	<u>D</u> own	
	<u>R</u> eset	
	Apply	
	OK	
	Cancel	
Net 1: N087 From: Q7.2 To: Q6.2]	Net 1: DQS_P From: Q5.1 To: U11.T5 Net 2: DQS_N From: Q5.2 To: U11.T4
Filter Differential Pairs:		New Chain Properties:
Attribute:		Chain Name: Chain1
	e-	
	×.	
Include Pairs In Chains		

It now features two lists; the left one contains a filtered list of Differential Pairs in the design to choose from, and the right list contains the chosen Differential Pairs that will represent the chain.

Selecting a Differential Pair name in ether list will display its nets and pins in an information panel below the list. You can filter the left list by attribute that can be a **Differential Pair attribute**, **Net attribute**, **Component attribute** or **Pin attribute**. The Net attribute will have to exist on either Net on the pair and the Component or Pin attribute will have to exist on any of the four pins in the pair.

You can also **Include Pairs In Chains** (or exclude by unchecking the option) all Differential Pairs that are already in a chain from the list, this helps refine the list.

The **Chain Name** text box allows you to set a name for the newly created Differential Pair chain when you exit this dialog. If this box is left blank, then a default unique chain name will be generated for you.

The **Apply** button allows you to quickly create a chain and start the creation of another chain. This is done by creating a chain using the currently input details. Checking on Apply will ensure if name is unique, if not enough Differential Pair have been added, etc. It then then resets the dialog and updates source list to allow you to start the creation of another Differential Pair chain without you needing to reopen the dialog using the **New Chain** button again.

Once you have the Differential Pairs selected, type a new Chain Name and press **OK** to return to the Differential Pairs dialog where the chain will now be presented:

				First Pin Pa	ir		Second Pin P	air
	Name	Chain Link Name	Net	Start Pin	End Pin	Net	Start Pin	End Pin
	Chain DQS			Q4.1	U11.T5		Q4.2	U11.T4
		DQS_	DQS_P	Q4.1	Q5.1	DQS_N	Q4.2	Q5.2
		Q5.1-U11.T5 Q5.2-U11.T4	DQS_P	Q5.1	U11.T5	DQS_N	Q5.2	U11.T4
1	Q7.2-Q6.2 Q7.1-Q6.1		N087	Q7.2	Q6.2	N083	Q7.1	Q6.1
	SDQ0_		SDQ0_N	U5.1	U6.2	SDQ0_P	U5.2	U6.1
DQS	airs In The Chain: 15 Q5.2-U11.T4		Edit Remove	Differer Net Sty	Attributes htial Pair Skew les: None			Add
			Up Down		Size Limit Rules "hance Length			► Edit > Delete

Once you have created the chain, selecting it in the grid still displays the chain controls, but the **Add...** button has been replaced with an **Edit...** button. Selecting it displays the New Chain dialog where you can edit the chain.

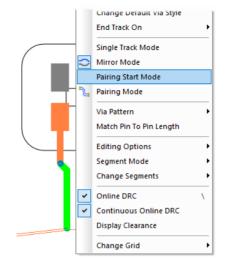
New <Component Pin Name> attribute

To make this chain easier to locate, a new pin attribute has been created **<Component Pin Name>**, for example **U12.6**. Use this if you need to find the pair that connects to that pin. This attribute is available throughout Pulsonix.

New 'Start Pairing' Mode for Differential Pairs

The ability to start routing a Differential Pairs using new **Pairing Start Mode** instead of using **Mirror Mode** has been added.

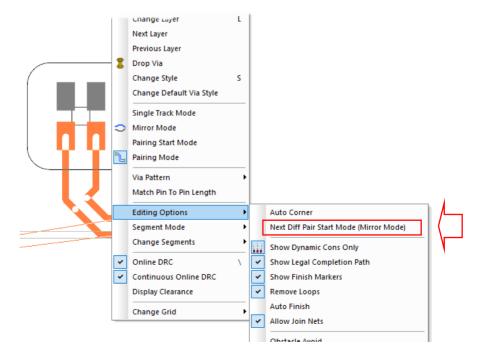
When you start editing a Differential Pair track, the mode used will be as defined in the **Options Edit Track** page.



The choice is Single Track Mode, Mirror Mode and the new Pairing Start Mode.

Paring Start mode is fundamentally the same as using the **Insert Bus Route** option in that it gathers the two tracks by single side routing to the two junctions representing the start of the paired section. Moving the cursor moves the junction pair to the required position, and left click starts the pairing. You can switch between the three modes using the context menu, which will show which mode you are in.

Whilst editing, you can check what auto start mode is currently set using the **Editing Options** sub menu from the context menu. The mode is shown in brackets.

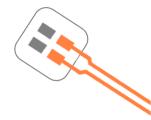


Choosing **Next Diff Pair Start Mode** cycles to the next mode from the one shown. This replaces the **Start Mirroring Paired Track** option in 10.5.

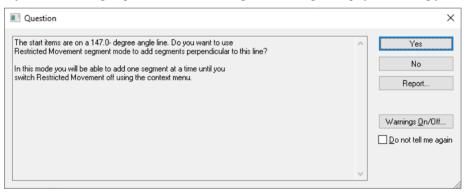
Whilst adding paired segments, you can end pairing by selecting one of the three modes to carry on editing. Using **Pairing Start mode** here allows you to automatically route the tracks around an obstacle and start pairing again.

Differential Pair Mirror Mode

When using **Mirror Mode** and mirroring paired tracks (Differential Pairs), the initial segments and the mirror line is now forced to be perpendicular to the line through the pads. Previously, the angle of this line was based on where you clicked to start the track. This mainly effects Differential Pairs that start on angled pads.



If you start on an angled pad, one that isn't 45-degrees, a message is displayed informing you of this:

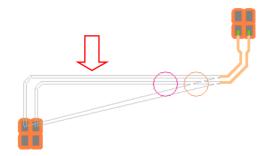


If you want to change the angle of the mirror line use the new **Rotate Mirror Angle** option from the context menu if it is available.

If you did not intend to start mirroring this time and only want to edit the single track, switch to **Single Track Mode** from the context menu.

Legal Completion Path for Differential Pairs

The **Show Legal Completion Paths** option now works when pairing tracks for a Differential Pair. The path shown is as if using pairing start mode from the target pads to the junctions at the end of the moving paired tracks. Using **Complete As Track** will convert these paths to tracks.



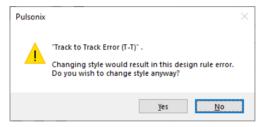
This mode also now works when in Mirror mode.

Change Style of Differential Pairs

You can now change the style of paired Differential Pair tracks if multiple tracks are selected. The style of both paired Tracks will be changed, along the entire paired section.

Do this by changing the style in the **Change Style** dialog, or in **Track Properties** (the style combo box is no longer disabled).

If you select a track style that would create a DRC error, a warning dialog is displayed:



Shift to add corner in Differential Pair Mirror Mode

When using **Mirror Mode** and mirroring paired tracks (Differential Pairs), adding a corner when the tracks are close together ends the mirroring mode and starts editing the pair. If you now hold the Shift when adding the corner, you will remain in Mirror Mode. This enables you to add multiple corners to design the track shape required before the pairing starts.



This feature was back-fitted to 10.5

New Via Pattern option - Auto Turn

There is a new **Via Pattern** option available - **Auto Turn**. When changing layers, use this to initially exit the vias straight, but as you move the cursor after vias are added, the pattern will change to exit left, right or backwards to best match the cursor position. Once you corner or backspace, the Auto Turn mode will be switched off. This is now the default action.

Select Via Pattern> and Auto Turn from the context menu:

	Change Derault VIa Style	1	
	Single Track Mode		
\sim	Mirror Mode		
	Pairing Start Mode		
۴.	Pairing Mode		
	Via Pattern	5	Parallel Vias
	Match Pin To Pin Length	===	Perpendicular Vias
	Editing Options	*	45 Degree Vias
	Segment Mode	×	Cross Over
	Change Segments	*	No Turn
~	Online DRC \	33	Turn Left
~	Continuous Online DRC	3	Turn Right
~	Display Clearance	-% (Turn Backwards
	Change Grid	2	Auto Turn
			Allow Off-Grid Vias
		~	Show Via Pattern

Use copy of existing Differential Pair Via Pattern

You can now choose to use a copy of an existing Differential Pair Via pattern when changing layer whilst adding paired tracks. You may have already created a pattern that you want to reuse, this can be copied and reused.

Before you start Differential Pair routing you have to choose the pattern to use. To do this first select all items in an existing pattern from the end of the paired tracks on one layer to the start of the paired tracks on the other layer. Using **Frame Select** with the **Select If completely Framed** option enabled will help ensure precise selection. Then, if needed, add any Ground vias that have been placed around the pattern (used for shielding) to this selection.

In the example below, selecting the red and yellow tracks plus the vias would be suitable:



In the example below, the selection is more extended to include one segment of the Differential Pair, this is no longer eligible for the Copy option to appear on the context menu.



If the pattern is as required, you can then use **Copy Diff completion Pair Pattern** from the context menu to save it to a special location on the clipboard.

	Group
	Nets •
	Auto Weld Selection
	Copy Diff Pair Via Pattern
	Auto Mitre Selection
	Unmitre Selection
	🔓 Smooth Selected Tracks
	Pull Tight Selection
•	• Fatten/Neck Selection
\sim	Use Default Track/Via Style
	Disconnect Track End
	Unroute Track Segments
	Unroute Track Path
-	Highlight Selection

The pattern must contain Tracks and Vias on two nets, each net selection must contain a track that starts from a paired track on one layer, and then a continuous path from it to a track that ends on a paired track on another layer. This path can contain several micro-vias. The pattern can change direction and cross over.

Then, when adding new paired tracks, you can now right click to use the **Via Pattern** sub-menu and choose **Paste Diff Pair Via Pattern**.

\sim	Mirror Mode	í.	
	Pairing Start Mode		and
Ŀ,	Pairing Mode		_
	Via Pattern	5	Parallel Vias
	Match Pin To Pin Length	==	Perpendicular Vias
	Editing Options		45 Degree Vias
	Segment Mode		Paste Diff Pair Via Pattern
	Change Segments	×	Cross Over
~	Online DRC	*	No Turn
 ~	Continuous Online DRC	33	Turn Left
	Display Clearance	Ŷ	Turn Right
	Change Grid	-‰	Turn Backwards
-		~	Auto Turn
			Allow Off-Grid Vias
		~	Show Via Pattern

If the copied via pattern is suitable for this pair you will see all its vias as you move the paired tracks. Change layer and the copied via pattern will be inserted in order for you to change layer. If the pattern is not suitable (wrong gap between tracks, wrong layer, etc) then change layer will not be available until you switch the via pattern.

The via pattern option **Paste Diff Pair Via Pattern** will be remembered and will be used in future whilst the pattern is available on the clipboard. Changing layer back to the start layer will use the same pattern rotated about 180 degrees.

You cannot use **Auto Turn** or **Drop Via** with this feature. When you use change layer it is fixed to the layer that the chosen pattern ends on.

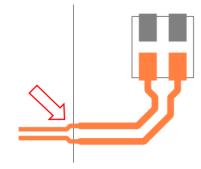
Editing Differential Pairs into Areas

When adding Differential Paired tracks and you cross an **Area** boundary that has different Track styles or Differential Pair gaps defined (in **Net Styles**), the pairing is automatically stopped on the near side and started again on the far side with two transition tracks added over the border joining the ends of the paired sections together. If it was a mistake entering the area, use the **Backspace** key to remove the area transition Tracks and revert to editing the original paired tracks.

	Attribute Name	Match Value	Net Type	Area	Track Styles				
Enable					Track Side	Track Layer	Def. Track	Alt. Track	Fat/Neck Min Len
\checkmark	<net name=""></net>	SDQ0*		DiffArea			Signal (4)	Signal (4)	<default></default>
	<net name=""></net>	SDQ0*					Track (6)	Signal (4)	<default></default>
	<net name=""></net>	ADD*					Track (6)	Track (8)	<default></default>
	<net name=""></net>	DQS_*		BGA			Signal (4)	Signal (4)	<default></default>
	<net name=""></net>	DQS_*					Track (8)	Track (6)	<default></default>
	<net name=""></net>	DQS_*			Inner		Track (8)	Track (8)	<default></default>

Attribute:	<net nan<="" th=""><th>ne> 、</th><th>For N</th><th>ets of T</th><th>ype: <any></any></th><th>~</th><th></th><th></th><th></th></net>	ne> 、	For N	ets of T	ype: <any></any>	~			
Match:	SDQ0*	· · · · · · · · · · · · · · · · · · ·	× 💉 V	Vithin A	reas: DiffArea		~		
Define [ck Styles		5	∠Define Via D	efaults —			
For Tr On S		<any></any>	~		For Vias with	<u>L</u> ayer Span:	<any></any>		~
or On <u>L</u>	_ayer:		~		Vias Not		Delete if not	Routed 5	Reduce Span
	lt Track St	yle: nal (4)	~			efault Via Styl		Houted	
N <u>a</u> m <u>W</u> idtl			~		Nam <u>e</u> :	Via (18)			~
Alterna	ate Track :	Style:			Widt <u>h</u> :	0.4572	<u>S</u> hape:	Round	~
Nam	je: Sigr	nal (4)	\sim		Length:	0.4572	Drill:	0.2540	
Widt	h: 0.10	016			✓ Plated				
Fatten/1	Neck Min l	ength: <default></default>							

As the Differential Pair enters the named **Area**, it changes to the styles defined in the **Net Styles** dialog.



This feature was back-fitted to 10.5

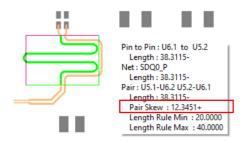
Differential Pair Properties

For completeness the **Differential Pair Properties** tab when selecting a Differential Pair track, now shows a list of Differential Pair Rules (like the **Differential Pair Technology** page).

Hauk Lengun. 20.2020	20.0133
Via Count: 0	0
Rules:	
Minimum % Paired: 80.000000	Minimum Gap: <a>Kultiple>
Max Length Difference: 4.0000	
Differential Pair Skew Rules: <differe Differential Pair Gap Rules: <differential <differential="" <none<br="" gap="" pair="" rules:="" skew="">Necked Length Rules: None Serpentine Rules: None Track Length Rules: None Track Length Factor Rules: None Track Length Factor Rules: None Track Length Match Rules: None Track Length Match Rules: None</differential></differe 	ntial Pair Name>=U5.1-U6.2 U5.2

Differential Pair Track Limits Display

The display for Differential Pair editing now always shows the **Skew Min** and **Max** limits as well as the other rule min and max limits. Previously, it only showed skew limits if it was the min or max limit, and if it was it did not show net length limits (for example).



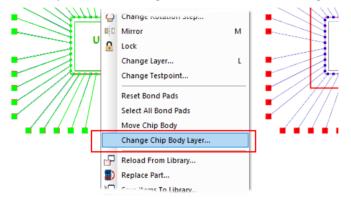
This feature was back-fitted to 10.5

Chip-On-Board Option Changes

The Chip-On-Board options below are only available if you have the Chip Toolkit license.

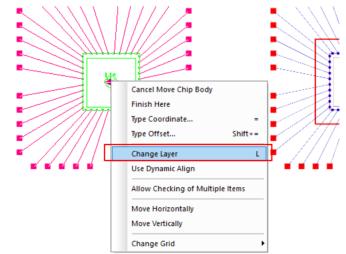
Change Chip Body Layer

With a Chip-On-Board component with bond pads selected, if you right click, there is a new option, **Change Chip Body Layer** available on the context menu. Use this to move the body of the chip down into a cavity, whilst leaving the bond pads on the top side. This avoids the need to move the chip body and then change layer whilst moving it. This option is only available if the design has a suitable layer to move the component to that allows normal components on it.



Change Layer of Chip Body during Move

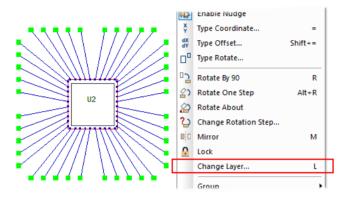
When moving a Chip-On-Board component, using the context menu option **Move Chip Body**, you can now use **Change Layer** from the context to move the component body into a cavity. The Bond Pads layer is not changed.



This feature was back-fitted to 10.5

Change Layer of Chip Bond Pads

In a PCB design for a **Chip-On-Board** component, with **All Bond Pads** selected (using the interactive mode, **Select All Bond Pads** from the context menu) and interactively editable, you can now use **Change Layer** from the context menu to move them to any electrical layer including layers that are not the same as the Die/Die Pad layer.



This feature was back-fitted to 10.5

Report Maker Changes

New top-level commands for Append Report & View Report

Two new commands have been added to the top level of the Report Maker - **Append Report** and **View Report**. Both commands can take the value **Always** or **Never**. They override the **Reports** dialog settings for these values and enable you to suppress a report if you wish to run it silent or if the report visibility isn't required (running a format file from CAM Plots for example or within a Script). If using the **Test** button, this command is ignored. It is available when running live reports.

Edit Fixed Cor	nmand	Х
Command:	View Report - Always	
View Report:	● Always O Never	

New commands to support Back Drilling

Additional commands have been added to **List of Vias** and **List of Pins** to support the new **Back Drilling** feature:

Is Back Drilled

Top Back Drill Span Name

Bottom Back Drill Span Name

Under List of Back Drill Spans, commands for Back Drill Oversize and Back Drill Stub Length have been added. Commands supported here too; Must Cut will return the last electrical layer to be drilled, and Must not cut will return the first electrical layer with a connection to the via after Back Drilling has been performed. Also supported, Back Drill Span Name, Start Layer, Start Layer Number, Stop Layer and Stop Layer Number.

List - New Line If Full	
List - Reported Count	i⊒ List of Back Drill Spans
List - Total Count	
List of ASCII File Lines	
List of Drill Sizes	
List of Vias	
Is Used	
Must Not Cut Layer	
Must Cut Layer	
Back Drill Span Name	
Span Depth	
Start Layer	
Start Layer Number	
Stop Layer	
Stop Layer Number	
Туре	
	v

Additional commands are available when using:

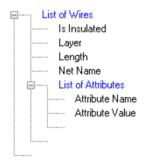
On a via - Is Back Drilled, Top Back Drill Span Name, Bottom Back Drill Span Name.

On a layer - Layer Depth.

On a Layer Span - Span Depth.

New command – List of Attributes available in List of Wires

With the introduction of **Attributes** for **Wires**, you can now extract the attributes and values using **List of Attributes** from within a **List of Wires**.



This command was back-fitted to 10.5

New command - Is Any Instance Fitted

A new command has been added **Is Any Instance Fitted** and can be used in a Schematic or PCB design report. Use within a **List of Variants** within a **List of Components**. This is best used in conjunction with the **If** command to test if any of the variant instances of the current Component are fitted in the current variant.

This varies from the **Is Fitted** command that is available to just check if the current Component instance is fitted in the current variant.

This command was back-fitted to 10.5

New command - Attribute In Part Library

A new command **Attribute In Part Library** and can be used in a Schematic or PCB design report. Use on a Part, Component, Component Group or Associated Part to report the value of a named attribute on the latest Part in the library (or Vault). This was required so that obsolete parts could be marked in the library (or Vault) and then a report created of which parts in a design are obsolete without having to reload all the parts. If the part is added to the design from the vault, it will get the value of the named attribute on the latest version of the part in the vault.

This feature was back-fitted to 10.5

Define the attribute name in an attribute command using a variable

Report Maker is now able to define the Attribute name in an attribute command using a Variable.

Edit If Command	×
Command: If "Attribute: Variable: attribName Value" is not blank	
If: Attribute V Attribute: attribName V Or.	
☑ Named By Variable Use: ○×Coord ○Y Coord	
Test the Value field length	
Is: not blank.	
Fixed Value: R*	
O Variable:	
Else:	
Prompt for Variable "attribName" ask user "Which Attribute ?" List of Components If "Attribute: Variable: attribName Value" is not blank Component Name Attribute "Variable: attribName" Value	

This feature was back-fitted to 10.5.

New command - Has Track Sturs

A new command **Has Track Spurs** is now available to be used in a PCB report within **List of Differential Pairs**, **List of Differential Pair Chains** and also in **List of Pin Pairs** when used within the previously mentioned two lists. This reports if the pair, chain or one side of the pair or chain have track stubs in the path from start node to end node. This can be used in a **Variable** to compare against **Allow Track Spurs** to report an error.

Scripting Changes

Scripting changes are only outlined below. More syntactical details are available in the Online Help under Scripting.

Folders object

The *Folders* object provides access to the various sets of folder information available in the application through the **Setup Folders** dialog.

This allows you to add and remove folders across all possible categories, potentially allowing a script to be written to provide a full "change workspace" capability by programmatically resetting all folders.

New property "DeviceString" in a CamPlot object

There is a new property **DeviceString** in a CamPlot object. This allows access to the printer name of a plot.

This feature was back-fitted to 10.5

New method "ActivateDesign" in the Application object

There is a new method **ActivateDesign** in the Application object. This allows an already open design to be activated (brought to the front).

This feature was back-fitted to 10.5

New Scripting Commands for Export of ODB++ and DXF

New scripting commands are available for export of ODB++ and DXF.

This is an extension to the existing command in Scripting, which previously only allowed you to output plots for the 'normal' output devices (Gerber etc.). It now allows any 'device' that can be used in a plot on the CAM Plot dialog; DXF, Excellon, GenCAD, Gerber, IDF, IPC-2581, LPKF, ODB++, PDF, Reports, Pen, STEP, SVG, and Windows.

New Scripting Command for LoadTechnology

A new LoadTechnology function is now available for a Document object. Calling this function with a full path to a technology file name will load that technology into the design. If you wish to load part of a Technology file, you can do this now but you will have to save a partial Technology file and load the whole file in order to achieve this.

This is a simple VBScript example of how you could use this new function:

Call the function with a full path name to the required Technology file and it will be loaded into the design. You can do this with a full or partial Technology. You can to use this for example to load just CAM/Plot settings by first saving the required section out as a partial technology file. Help has been updated to show the scope of this change.

Changes to Import Alien Design Files and STEP Model Positions

When importing Altium designs as ASCII files, Pulsonix now reads the STEP model name, offsets and rotations and places them into the three STEP attributes on the Footprint within Pulsonix.

Note, there have been some examples supplied that have no rational logic for model rotations and therefore have to be manually adjusted once in Pulsonix. However, these are rare and usually only one model in a design.

Loading Partial Colour Files

You can now load a cut-down version of the **Colours** file for both SCM and PCB. This should only be done if you know what you're doing. Further information and formatting details can be found on the Help page under the **Index** heading **Load Colours**

New PCB Design Dialog – Set Design Units

From the **File** menu and **New Designs** tab and **PCB Design**, you can now choose to overwrite the **Units** defined in the **Technology File** selected.

By selecting the **Set Units** check box, you can then select between **Imperial** and **Metric**, choose the **unit type** and set the unit **Precision**.

Designs	Technologies + Profiles Wizards
E PCB	Design
🗗 Sche	ematic Design
📴 Scm	Hierarchy Block
PCB	Footprint
PCB	Doc Symbol
📳 Sche	ematic Symbol
불 Sche	ematic Doc Symbol
📭 Part	
📳 Pane	el Design
Technology	y: 4 Layer Flex-Rigid (Black - Imperial) 🗸 🗸
Profile File:	[None]
🗹 Set Unil	ts: O Imperial: thou V Metric: mm V
	Precision: 3
	OK Cancel

Changes to Default Supplied Files

Technology Files

All Schematic and PCB Technology files have been updated to reflect the new rule sets in V11.0.

Libraries

The Schematic Doc Symbol library now includes example of new Doc Symbol types for **Net Label**, and **Mounting Hole**. Existing Doc Symbols have been redefined to give them the correct category used for searching within the **Insert Signal Reference** dialog.