

Pulsonix Design System V10.0 Update Notes

Copyright Notice

Copyright © WestDev Ltd. 2000-2018 Pulsonix is a Trademark of WestDev Ltd. All rights reserved. E&OE

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

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

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

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

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

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

All other trademarks are acknowledged to their respective owners.

Pulsonix, a division of WestDev Ltd.

Printed in the UK Issue date: 10/08/18 iss 4

Pulsonix

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

| Support Phone | +44 (0)1684 296 570 |
|---------------|---------------------|
| Sales Phone | +44 (0)1684 296 551 |
| Fax | +44 (0)1684 296 515 |
| Email | info@pulsonix.com |
| Web | www.pulsonix.com |

Contents

| CONTENTS | 3 |
|--|----|
| VERSION 10.0 UPDATE SUPPLEMENT | 6 |
| Installing the New Version of Pulsonix | 6 |
| Licensing | 6 |
| Version 10.0 Update Notes | 6 |
| New In Version 9.1 - Not Previously Documented | 7 |
| Page Link Attribute on Blocks and Block Ports | 7 |
| Save configuration Files - Speed Increase When Saving Over Network | 7 |
| Intelligent Gerber Import option - Add unmatched drill holes as pads | |
| Component Search Engine | 8 |
| Product Version/Build Requirements | 8 |
| Registering with Component Search Engine | 8 |
| Downloading the Library Loader | 9 |
| Using the Pulsonix Component Search Engine | |
| Updating Imported Parts to Company Standards | |
| Component Search Engine and Pulsonix in Dual Screen Mode | |
| Closing the Library Loader Application | |
| New IN VERSION 10.0 - NOT PREVIOUSLY DOCUMENTED | 14 |
| Accel Netlist Import – Part Mapping Dialog | 14 |
| Netlist Output to SiMetrix Spice Simulator | 14 |
| Current Selection Mask Shown on the Status Bar | 15 |
| STEP Output – Settings Menu Change | 16 |
| STEP Output – Define Silkscreen Layer | 16 |
| Fillet and Mitre Corners for Differential Pairs and Bus Route | 16 |
| Differential Pairs - Fillets | 16 |
| Bus Route – Fillets | |
| ODB++ Output includes Profile Cutouts | 19 |
| CAM Plots – Auto Generate Plot Ordered By Variants | 20 |
| Report Maker | 20 |
| Rotation of Attribute Command | 20 |
| Indent & Unindent Commands | |
| New In Version 10.0 | |
| Technology Dialog Changes | 23 |
| Enhanced Technology Dialog | 23 |
| Technology Dialog Tree Structure | 24 |
| New Style Naming pages for the Technology Dialog | 24 |
| Spacing Check of SMD Pads & Micro-vias | 27 |
| Pad Exceptions By Layer Class & Inner Electrical layers | |
| Acid Trap Rules | |
| Enclosure Spacing Rules | |
| Select Net Names by Attribute | |
| Variant Manager now a grid Dialog | |
| New Adjacent Nets Rule | |
| Define Pad Shapes now on Pad Styles dialog | |
| Design Settings now in Technology Dialog | |
| Grids now in Technology Dialog | |

| Locked Grid Origins | 34 |
|---|----------------|
| Apply Origin To All Grids | 34 |
| 3D STEP Preview Changes | 35 |
| Gradient Background (switchable between Solid and Gradient) | 35 |
| Improved Visualisation of the STEP Preview | 36 |
| Print option in STEP Preview | 36 |
| Toolbar with Icons of Pre-defined 3D Views | 36 |
| Perspective and Orthographic Views | 37 |
| Rotation of Design in STEP Preview now on Right Hand Mouse button | 37 |
| Selection of items in STEP Preview on Left Hand Mouse button | 37 |
| Move Component on Grid in STEP Preview | 38 |
| Automatically Synchronise 2D PCB from 3D | 39 |
| Working in both 2D and 3D Environments | 39 |
| Enclosure Spacing Rules | 40 |
| Adding Mechanical Enclosures | 40 |
| Transparency Setting for Enclosures | 41 |
| Colour Setting for Enclosures | 42 |
| Clash Detection within the 3D Environment | 42 |
| Using the Clash Detection Option | |
| Colouring for non-coloured STEP Models | 43 |
| Properties of Selected design items shown on Status Bar | 44 |
| Speed Increase of STEP file Generation through Multithreading | 44 |
| STEP Model to Footprint Alignment | 44 |
| 3D Package Built-in Attribute | 45 |
| Selection Mask | 45 |
| Undated User Interface | 47 |
| AK Granhics Support | 47 |
| simplified Default Toolhars | 47 |
| More Readability for Read Only Edit Boyes | 47 |
| Design Attributes in Variants | |
| Interactive Copper Pour Healing | 40 40 |
| Changes to Manual Routing Modes | - 7 |
| Segment Modes | 50 |
| Edit Mitre Mode | 50 |
| Design Rules Checks Dialog Changes | |
| New DRC Checks - Acid Trans and Adjacent Nets | |
| Clear Warnings Ontion | 52 |
| Automatically Open DRC Errors Bar Switch | |
| Attribute Editor Changes | 52 |
| Footprint Name in Edit Attribute Dialog | 52 |
| Hide Inbuilt Columns | |
| Reload All – Do Not Reload Doc Symbols | 53 |
| New Commands in Ontions Dialog | 55 |
| Middle Mouse Wheel/Button Usage Ontions | 53 |
| New Component Pads Selection | 55 |
| Create PCB Group for SCM Block | |
| Ioin Open Shapes | |
| Save Doc Symbols To Appropriate Library | 55 56 |
| Changes to Frror Markers | 50 |
| Find will Find Areas with no name | 57 |
| Find Bar Changes to Select All Find Items | 57 57 |
| Dynamic Starting for DCB Rus Routing | 57 57 |
| New Template Ontion _ Clear On Edit | 57 59 |
| Library Managar Changes | |
| Liorary manager Changes | |

| Part Pin Mismatch Report 5 STEP File Names in Part Details 5 Clear Filter 6 Custom Indexing for Footprints 6 Manual list Refresh in Library Manager 6 'Follow Item' on a Part option 6 Find Symbol 6 Find Footprint 6 Find Footprint 6 Find Footprint 6 Find Footprint 6 Rename in Library Manager – Replace Symbol References in Parts. 6 Swap Symbols Option 6 Create New Libraries directly from the Library Manager Folder tab 6 Part Description Using Substitute Attributes. 6 CAM Plots – Drill Output available for Board Outlines & Cutouts. 6 CAM Plot – Plot String Attribute 6 Report IPC-2581 - Omit BOM Data 6 Toggle True Width For Layers 6 Report Maker Changes 6 Access Other Tables in PDC Database 7 Mignoved Differential Pairs - Turn Backwards direction on Via Pattern 7 Improved Accuracy of Differential Pairs 7 Improved Differential Pairs Routing – Fillet Mode. 7 | Resizable Library Manager | 58 |
|--|--|----|
| STEP File Names in Part Details 5 Clear Filter. 6 Custom Indexing for Footprints. 6 Manual list Refresh in Library Manager. 6 Follow Item' on a Part option 6 Find Symbol 6 Find Footprint. 6 Find Footprint in 3D Packages 6 Rename in Library Manager – Replace Symbol References in Parts. 6 Swap Symbols Option 6 Create New Libraries directly from the Library Manager Folder tab 6 Part Description Using Substitute Attributes 6 CAM Plot – Drill Output available for Board Outlines & Cutouts. 6 CAM Plot – Drot String Attribute 6 Export IPC-2581 - Omit BOM Data 6 Toggle True Width For Layers 6 Adjacent Nets in Report Maker – List of Errors 7 Mifferential Pairs - Turn Backwards direction on Via Pattern 7 Improved Differential Pair Routing – Fillet Mode. 7 Improved Differential Pair Routing – Fillet Mode. 7 Improved Differential Pair Routing – Mirror Diff Pair routing from Angled Components 7 New Serpentine Skew Modes on Differential Pairs. 7 | Part Pin Mismatch Report | |
| Clear Filter 6 Custom Indexing for Footprints. 6 Manual list Refresh in Library Manager 6 'Follow Item' on a Part option. 6 Find Symbol 6 Find Symbol 6 Find Footprint 6 Find Footprint in 3D Packages 6 Rename in Library Manager – Replace Symbol References in Parts. 6 Swap Symbols Option 6 Create New Libraries directly from the Library Manager Folder tab 6 Part Description Using Substitute Attributes 6 CAM Plots – Drill Output available for Board Outlines & Cutouts. 6 CAM Plot – Plot String Attribute 6 Export IPC-2581 - Omit BOM Data 6 Access Other Tables in PDC Database 6 Adjacent Nets in Report Maker – List of Errors 7 High Speed Option Changes. 7 Improved Accuracy of Differential Pairs. 7 Improved Differential Pair Routing – Fillet Mode. 7 Improved Differential Pair Routing – Fillet Mode. 7 Improved Differential Pairs Outing – Fillet Mode. 7 Improved Differential Paire Routing – Fillet Mode. 7 | STEP File Names in Part Details | |
| Custom Indexing for Footprints | Clear Filter | 60 |
| Manual list Refresh in Library Manager 6 'Follow Item' on a Part option 6 Find Symbol 6 Find Footprint 6 Find Footprint in 3D Packages 6 Rename in Library Manager – Replace Symbol References in Parts. 6 Swap Symbols Option 6 Create New Libraries directly from the Library Manager Folder tab 6 Part Description Using Substitute Attributes 6 CAM Plots – Drill Output available for Board Outlines & Cutouts 6 CAM Plot – Plot String Attribute 6 Export IPC-2581 - Omit BOM Data 6 Access Other Tables in PDC Database 6 Adjacent Nets in Report Maker – List of Errors 7 High Speed Option Changes 7 Differential Pairs - Turn Backwards direction on Via Pattern 7 Improved Differential Pair Routing – Fillet Mode 7 Improved Differential Pair Routing – Fillet Mode 7 Improved Differential Pair Routing – Fillet Mode 7 New Serpentine Skew Modes on Differential Pairs 7 New Serpentine Skew Modes on Differential Pairs 7 Network License Server (NLS) Changes 7 | Custom Indexing for Footprints | 60 |
| 'Follow Item' on a Part option 6 Find Symbol 6 Find Footprint 6 Find Footprint in 3D Packages 6 Rename in Library Manager – Replace Symbol References in Parts. 6 Swap Symbols Option 6 Create New Libraries directly from the Library Manager Folder tab 6 Part Description Using Substitute Attributes 6 CAM Plots – Drill Output available for Board Outlines & Cutouts. 6 CAM Plot – Plot String Attribute 6 Export IPC-2581 - Omit BOM Data 6 Report Maker Changes 6 Access Other Tables in PDC Database 6 Adjacent Nets in Report Maker – List of Errors 7 High Speed Option Changes 7 Differential Pairs - Turn Backwards direction on Via Pattern 7 Improved Differential Pair Routing – Fillet Mode 7 Improved Differential Pair Routing – Mirror Diff Pair routing from Angled Components 7 Finish Track On Differential Paired Track (Spur) 7 New Serpentine Skew Modes on Differential Pairs 7 Move All Vias in Composite Span 7 Network License Server (NLS) Changes 7 | Manual list Refresh in Library Manager | 62 |
| Find Symbol 6 Find Footprint 6 Find Footprint in 3D Packages 6 Rename in Library Manager – Replace Symbol References in Parts 6 Rename in Library Manager – Replace Symbol References in Parts 6 Swap Symbols Option 6 Create New Libraries directly from the Library Manager Folder tab 6 Part Description Using Substitute Attributes 6 CAM Plots – Drill Output available for Board Outlines & Cutouts 6 CAM Plot – Plot String Attribute 6 Export IPC-2581 - Omit BOM Data 6 Toggle True Width For Layers 6 Report Maker Changes 6 Access Other Tables in PDC Database 6 Adjacent Nets in Report Maker – List of Errors 7 High Speed Option Changes 7 Differential Pairs - Turn Backwards direction on Via Pattern 7 Improved Differential Pair Routing – Fillet Mode 7 Improved Differential Pair Routing – Mirror Diff Pair routing from Angled Components 7 Finish Track On Differential Paired Track (Spur) 7 New Serpentine Skew Modes on Differential Pairs 7 Move All Vias in Composite Span 7 </td <td>'Follow Item' on a Part option</td> <td>62</td> | 'Follow Item' on a Part option | 62 |
| Find Footprint 6 Find Footprint in 3D Packages 6 Rename in Library Manager – Replace Symbol References in Parts. 6 Swap Symbols Option 6 Create New Libraries directly from the Library Manager Folder tab 6 Part Description Using Substitute Attributes 6 CAM Plots – Drill Output available for Board Outlines & Cutouts. 6 CAM Plot – Plot String Attribute 6 Export IPC-2581 - Omit BOM Data 6 Toggle True Width For Layers 6 Report Maker Changes 6 Access Other Tables in PDC Database 6 Adjacent Nets in Report Maker – List of Errors 7 High Speed Option Changes 7 Differential Pairs - Turn Backwards direction on Via Pattern 7 Improved Differential Pair Routing – Fillet Mode 7 Improved Differential Pair Routing – Mirror Diff Pair routing from Angled Components 7 Finish Track On Differential Paired Track (Spur) 7 New Serpentine Skew Modes on Differential Pairs 7 Move All Vias in Composite Span 7 Network License Server (NLS) Changes 7 Default Supplied Technology Files Changes < | Find Symbol | 62 |
| Find Footprint in 3D Packages 6 Rename in Library Manager – Replace Symbol References in Parts. 6 Swap Symbols Option 6 Create New Libraries directly from the Library Manager Folder tab 6 Part Description Using Substitute Attributes 6 CAM Plots – Drill Output available for Board Outlines & Cutouts. 6 CAM Plot – Plot String Attribute 6 Export IPC-2581 - Omit BOM Data 6 Toggle True Width For Layers 6 Report Maker Changes 6 Access Other Tables in PDC Database 6 Adjacent Nets in Report Maker – List of Errors 7 High Speed Option Changes 7 Differential Pairs - Turn Backwards direction on Via Pattern 7 Improved Accuracy of Differential Pairs 7 Improved Differential Pair Routing – Fillet Mode 7 Improved Differential Paired Track (Spur) 7 New Serpentine Skew Modes on Differential Pairs 7 Move All Vias in Composite Span 7 Network License Server (NLS) Changes 7 Unattended Installation 7 | Find Footprint | 63 |
| Rename in Library Manager – Replace Symbol References in Parts. 6 Swap Symbols Option 6 Create New Libraries directly from the Library Manager Folder tab 6 Part Description Using Substitute Attributes 6 CAM Plots – Drill Output available for Board Outlines & Cutouts. 6 CAM Plot – Plot String Attribute 6 Export IPC-2581 - Omit BOM Data 6 Toggle True Width For Layers 6 Report Maker Changes 6 Access Other Tables in PDC Database 6 Adjacent Nets in Report Maker – List of Errors 7 Differential Pairs - Turn Backwards direction on Via Pattern 7 Improved Accuracy of Differential Pairs 7 Improved Differential Pair Routing – Fillet Mode 7 Improved Differential Paire Track (Spur) 7 New Serpentine Skew Modes on Differential Pairs 7 Move All Vias in Composite Span 7 Network License Server (NLS) Changes 7 Default Supplied Technology Files Changes 7 Unattended Installation 7 | Find Footprint in 3D Packages | 64 |
| Swap Symbols Option 6 Create New Libraries directly from the Library Manager Folder tab 6 Part Description Using Substitute Attributes 6 CAM Plots – Drill Output available for Board Outlines & Cutouts 6 CAM Plot – Plot String Attribute 6 Export IPC-2581 - Omit BOM Data 6 Toggle True Width For Layers 6 Report Maker Changes 6 Access Other Tables in PDC Database 6 Adjacent Nets in Report Maker – List of Errors 7 Differential Pairs - Turn Backwards direction on Via Pattern 7 Improved Accuracy of Differential Pairs 7 Improved Differential Pair Routing – Fillet Mode 7 Improved Differential Pair Routing – Mirror Diff Pair routing from Angled Components 7 Finish Track On Differential Paired Track (Spur) 7 New Serpentine Skew Modes on Differential Pairs 7 Move All Vias in Composite Span 7 Network License Server (NLS) Changes 7 Default Supplied Technology Files Changes 7 Unattended Installation 7 | Rename in Library Manager – Replace Symbol References in Parts | 65 |
| Create New Libraries directly from the Library Manager Folder tab | Swap Symbols Option | 65 |
| Part Description Using Substitute Attributes 6 CAM Plots – Drill Output available for Board Outlines & Cutouts. 6 CAM Plot – Plot String Attribute 6 Export IPC-2581 - Omit BOM Data 6 Toggle True Width For Layers 6 Report Maker Changes 6 Access Other Tables in PDC Database 6 Adjacent Nets in Report Maker – List of Errors 7 High Speed Option Changes 7 Differential Pairs - Turn Backwards direction on Via Pattern 7 Improved Accuracy of Differential Pairs 7 Improved Differential Pair Routing – Fillet Mode 7 Improved Differential Paired Track (Spur) 7 New Serpentine Skew Modes on Differential Pairs 7 Nove All Vias in Composite Span 7 Network License Server (NLS) Changes 7 Default Supplied Technology Files Changes 7 Unattended Installation 7 | Create New Libraries directly from the Library Manager Folder tab | 66 |
| CAM Plots – Drill Output available for Board Outlines & Cutouts | Part Description Using Substitute Attributes | 67 |
| CAM Plot – Plot String Attribute 6 Export IPC-2581 - Omit BOM Data 6 Toggle True Width For Layers 6 Report Maker Changes 6 Access Other Tables in PDC Database 6 Adjacent Nets in Report Maker – List of Errors 7 High Speed Option Changes 7 Differential Pairs - Turn Backwards direction on Via Pattern 7 Improved Accuracy of Differential Pairs 7 Improved Differential Pair Routing – Fillet Mode 7 Improved Differential Paire Routing – Mirror Diff Pair routing from Angled Components 7 Finish Track On Differential Paired Track (Spur) 7 New Serpentine Skew Modes on Differential Pairs 7 Move All Vias in Composite Span 7 Network License Server (NLS) Changes 7 Default Supplied Technology Files Changes 7 Unattended Installation 7 | CAM Plots - Drill Output available for Board Outlines & Cutouts | 67 |
| Export IPC-2581 - Omit BOM Data 6 Toggle True Width For Layers 6 Report Maker Changes 6 Access Other Tables in PDC Database 6 Adjacent Nets in Report Maker – List of Errors 7 High Speed Option Changes 7 Differential Pairs - Turn Backwards direction on Via Pattern 7 Improved Accuracy of Differential Pairs 7 Improved Differential Pair Routing – Fillet Mode 7 Improved Differential Paire Routing – Mirror Diff Pair routing from Angled Components 7 Finish Track On Differential Paired Track (Spur) 7 New Serpentine Skew Modes on Differential Pairs 7 Move All Vias in Composite Span 7 Network License Server (NLS) Changes 7 Default Supplied Technology Files Changes 7 Unattended Installation 7 | CAM Plot – Plot String Attribute | |
| Togle True Width For Layers 6 Report Maker Changes 6 Access Other Tables in PDC Database 6 Adjacent Nets in Report Maker – List of Errors 7 High Speed Option Changes 7 Differential Pairs - Turn Backwards direction on Via Pattern 7 Improved Accuracy of Differential Pairs 7 Improved Differential Pair Routing – Fillet Mode 7 Improved Differential Paire Routing – Mirror Diff Pair routing from Angled Components 7 Finish Track On Differential Paired Track (Spur) 7 New Serpentine Skew Modes on Differential Pairs 7 Move All Vias in Composite Span 7 Network License Server (NLS) Changes 7 Default Supplied Technology Files Changes 7 Unattended Installation 7 | Export IPC-2581 - Omit BOM Data | 69 |
| Report Maker Changes 6 Access Other Tables in PDC Database 6 Adjacent Nets in Report Maker – List of Errors 7 High Speed Option Changes 7 Differential Pairs - Turn Backwards direction on Via Pattern 7 Improved Accuracy of Differential Pairs 7 Improved Differential Pair Routing – Fillet Mode 7 Improved Differential Pair Routing – Mirror Diff Pair routing from Angled Components 7 Finish Track On Differential Paired Track (Spur) 7 New Serpentine Skew Modes on Differential Pairs 7 Move All Vias in Composite Span 7 Network License Server (NLS) Changes 7 Default Supplied Technology Files Changes 7 Unattended Installation 7 | Toggle True Width For Layers | 69 |
| Access Other Tables in PDC Database 6 Adjacent Nets in Report Maker – List of Errors 7 High Speed Option Changes 7 Differential Pairs - Turn Backwards direction on Via Pattern 7 Improved Accuracy of Differential Pairs 7 Improved Differential Pair Routing – Fillet Mode 7 Improved Differential Pair Routing – Mirror Diff Pair routing from Angled Components 7 Finish Track On Differential Paired Track (Spur) 7 New Serpentine Skew Modes on Differential Pairs 7 Move All Vias in Composite Span 7 Network License Server (NLS) Changes 7 Default Supplied Technology Files Changes 7 Unattended Installation 7 | Report Maker Changes | 69 |
| Adjacent Nets in Report Maker – List of Errors 7 High Speed Option Changes 7 Differential Pairs - Turn Backwards direction on Via Pattern 7 Improved Accuracy of Differential Pairs 7 Improved Differential Pair Routing – Fillet Mode 7 Improved Differential Pair Routing – Mirror Diff Pair routing from Angled Components 7 Finish Track On Differential Paired Track (Spur) 7 New Serpentine Skew Modes on Differential Pairs 7 Move All Vias in Composite Span 7 Network License Server (NLS) Changes 7 Default Supplied Technology Files Changes 7 Unattended Installation 7 | Access Other Tables in PDC Database | 69 |
| High Speed Option Changes 7 Differential Pairs - Turn Backwards direction on Via Pattern 7 Improved Accuracy of Differential Pairs 7 Improved Differential Pair Routing – Fillet Mode 7 Improved Differential Pair Routing – Mirror Diff Pair routing from Angled Components 7 Finish Track On Differential Paired Track (Spur) 7 New Serpentine Skew Modes on Differential Pairs 7 Move All Vias in Composite Span 7 Network License Server (NLS) Changes 7 Default Supplied Technology Files Changes 7 Unattended Installation 7 | Adjacent Nets in Report Maker – List of Errors | 70 |
| Differential Pairs - Turn Backwards direction on Via Pattern 7 Improved Accuracy of Differential Pairs 7 Improved Differential Pair Routing – Fillet Mode 7 Improved Differential Pair Routing – Mirror Diff Pair routing from Angled Components 7 Finish Track On Differential Paired Track (Spur) 7 New Serpentine Skew Modes on Differential Pairs 7 Move All Vias in Composite Span 7 Network License Server (NLS) Changes 7 Default Supplied Technology Files Changes 7 Unattended Installation 7 | High Speed Option Changes | 71 |
| Improved Accuracy of Differential Pairs 7 Improved Differential Pair Routing – Fillet Mode. 7 Improved Differential Pair Routing – Mirror Diff Pair routing from Angled Components 7 Finish Track On Differential Paired Track (Spur) 7 New Serpentine Skew Modes on Differential Pairs 7 Move All Vias in Composite Span 7 Network License Server (NLS) Changes 7 Default Supplied Technology Files Changes 7 Unattended Installation 7 | Differential Pairs - Turn Backwards direction on Via Pattern | 71 |
| Improved Differential Pair Routing – Fillet Mode | Improved Accuracy of Differential Pairs | 71 |
| Improved Differential Pair Routing – Mirror Diff Pair routing from Angled Components 7 Finish Track On Differential Paired Track (Spur) 7 New Serpentine Skew Modes on Differential Pairs 7 Move All Vias in Composite Span 7 Network License Server (NLS) Changes 7 Default Supplied Technology Files Changes 7 Unattended Installation 7 | Improved Differential Pair Routing – Fillet Mode | 72 |
| Finish Track On Differential Paired Track (Spur) 7 New Serpentine Skew Modes on Differential Pairs 7 Move All Vias in Composite Span 7 Network License Server (NLS) Changes 7 Default Supplied Technology Files Changes 7 Unattended Installation 7 | Improved Differential Pair Routing – Mirror Diff Pair routing from Angled Components | 73 |
| New Serpentine Skew Modes on Differential Pairs 7 Move All Vias in Composite Span 7 Network License Server (NLS) Changes 7 Default Supplied Technology Files Changes 7 Unattended Installation 7 | Finish Track On Differential Paired Track (Spur) | 74 |
| Move All Vias in Composite Span 7 Network License Server (NLS) Changes 7 Default Supplied Technology Files Changes 7 Unattended Installation 7 | New Serpentine Skew Modes on Differential Pairs | 74 |
| Network License Server (NLS) Changes 7 Default Supplied Technology Files Changes 7 Unattended Installation 7 | Move All Vias in Composite Span | 75 |
| Default Supplied Technology Files Changes | Network License Server (NLS) Changes | 75 |
| Unattended Installation | Default Supplied Technology Files Changes | |
| | Unattended Installation | 76 |

Version 10.0 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 10.0 on top of your existing V9.0 installation. If upgrading from V9.0 or V9.1, 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 10.0 requires a new license if a new user or upgrading from any older version of Pulsonix.

If you are using a version of Pulsonix earlier than Version 9.0 or 9.1, you will require a new license, this would have been supplied to you under the terms of your maintenance contract.

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

Version 10.0 Update Notes

Update Notes for Version 10.0 are available in the Pulsonix installation folder under \documents. Alternatively, select the direct link within the product itself: **Help menu | Online Manuals> Pulsonix V10.0 Updates Notes**

New In Version 9.1 - Not Previously Documented

The feature(s) below appeared in Version 9.1 but were not documented. They have been added here for your reference:

Page Link Attribute on Blocks and Block Ports

You can now use the page link attribute on **Blocks** and **Block Ports** to show the name of the page they link to.

A new option on **Design Settings** and **Naming** page enables you to **show** the **Page Number** instead of the **Page Name** on a page link.

| Prefix: | |
|---------------------------------------|---------------------------------|
| Separator: | Separator: |
| Newline After: 0 | |
| Show Current Page | Part Names |
| Show Page Number (instead of Name) | Display On Each Gate On Page |
| Page Link Attribute | |
| Show Page Number (instead of Name) | |

Page link on block showing Page Number





Save configuration Files - Speed Increase When Saving Over Network

A significant speed increase has been made for when saving your Configuration files to a network location. The speed is equivalent to that when saving the file locally now. This was added to a later version of Pulsonix 9.1

Intelligent Gerber Import option - Add unmatched drill holes as pads

| Intellige | ent Gerber In | nport - Input Files | |
|---|---|---|----------------|
| Tech | nnology File Name: | 4 Layer (White - Metric) | |
| Input | t Files | | |
| Gerl Gerl Gerl Drill Bill d | ber Ro ber Ro ber Ro ber Ro Ro Df Materials Ro | utingDemo1(Top Silk Screen).gbr Silkscreen Top utingDemo1(Top Electrical).gbr Top utingDemo1(Bottom Electrical).gbr Bottom utingDemo1(Bottom Silk Screen).gbr Silkscreen Bot utingDemo1(Through Hole).dr <through hole<br="">utingDemo1(Parts List CSV).csv</through> |) tom ⊳> |
| | <u>A</u> dd | Edit Deļete | |
| | nvert Pads to d unmatched o | Aas irill holes as pads OK Cano | cel |

With the **Add unmatched drill holes as pads** check box selected, the import option tries to match drill holes (in the drill file) with pads/flashes in the Gerber file. This enables data that has holes only but no pad, such as mounting holes, to be imported.

If the drill doesn't match a pad, it adds a pad for the drill hole even when it doesn't match a pad in the Gerber, otherwise a pad won't be added (and so no hole).

Component Search Engine

The **Component Search Engine** web site enables you to search Parts (along with Symbols and PCB Footprints) and download them to your Pulsonix application ready to use. Not only do they appear on the end of your cursor but are also automatically added to your library to use on another design.

Product Version/Build Requirements

You will need to be running Pulsonix 9.1 Build 6871 onwards to benefit from the **Component Search Engine** integration. This feature also runs will all version of Pulsonix 10.0 and later.

Registering with Component Search Engine

To use the Component Search Engine, you will first need to register an account.

Go here - http://pulsonix.componentsearchengine.com/ and click Register.



Downloading the Library Loader

Once you have registered, you are automatically logged in. You now need to download the **Library Loader** application. This is required as the interface between the Component Search Engine web site and the Pulsonix application.

Click on the Downloads button at the top of the Component Search Engine web page.



On the Downloads page, scroll down to the Library Loader section.

Pulsonix Downloads



This Excel based tool provides the following functions.

- · Find and download models from our library
- Fracture the models to your requriements
- Preview the symbols and footprints prior to export
 - Export to any of our supported CAD tools
- Request new models be created

Download EPW version 1.93



This helper application streamlines the process of loading our models into your CAD system.

Simply click this icon promotion from any website and once the Library loader is installed the model will appear on your cursor ready to place on your design. From a website to your design in seconds!

Download Library Loader version 2.12

Click the Download Library Loader button to start the download.

Once downloaded, run the LibraryLoaderSetup.msi program from your download folder.



Once the **Library Loader** has been successfully installed, open the **Library Loader** from your Windows **Start** menu.

You will be asked to either Register or Login. As you have already registered you can select the **Login** Tab and **Login** using your **User Name** and **Password**.

| Account Details | × | |
|-------------------------------|---|--|
| Register Login < | | |
| Email (User Name) Password | Login | |
| | Already have an account but forgot your password? | |

Once logged in, you now need to click the **Settings** button on the **Library Loader** dialog, this will open the **Pulsonix Settings**.

| Library Loader | _ | | \times | |
|---|--------|-------------|----------|--|
| Open EPW File Search for Parts Profile Feed | back I | <u>Help</u> | | |
| Downloads Folder C:\Users\Downloads | | Brows | e | |
| Your ECAD Tool Pulsonix | \sim | Setting | js | |
| You have downloaded these parts since 10:14 on 04/08/20 |)17 | | | |
| | | | ^ | |
| | | | | |
| | | | \sim | |
| User Name: tys@pulsonix.com | | | | |

From within the **Pulsonix Settings** dialog, you need to inform the Library Loader which version of Pulsonix you are using. If using 9.1 onwards, you will now be able to set the destination for where new Step Models, Schematic Symbols, PCB Footprints and Components (Parts) will be saved to.

| Pulsonix Settings | | |
|--------------------------|-------------------------------------|--|
| Version 9.1 or later | | |
| Libraries (9.1 or later) | | |
| Schematic Symbols | WestDev Internal.ssl 🗸 🗸 🖊 | |
| PCB Symbols | WestDev Internal.pfl V | |
| Components | WestDev Internal.pal | |
| STEP Model Folder | | |
| C:\Users\Public\Docu | ments\Pulsonix9.1\STEP Models 🛛 🗸 🦳 | |
| Show Library Import | : Instructions (9.0 or earlier) | |
| | OK Cancel | |

Note: you need to be using Pulsonix 9.1 (Build: 6869) onwards to have Parts automatically saved from the Component Search Engine into your Pulsonix Libraries.

Now you've set up the Library Loader you can now download Parts from the Pulsonix Component Search Engine website and they will load directly into your libraries and onto the end of your cursor ready for placement in a design.

Using the Pulsonix Component Search Engine

From the Pulsonix Component Search Engine website

(http://pulsonix.componentsearchengine.com/), enter the name of your Part or a description of it in the **Search** section. Once the search is complete you will see six columns presented including an ECAD Model, Datasheet and Pricing/Stock information for the Part from various suppliers.

Clicking the **Price/Stock** link will allow you to select and download the Part with pre-populated attributes from a specific supplier.

| | | | coilcraft Q | | | |
|-------|-------------------------|------|---|--------------|----------------|-------------------|
| | | | Components matching your search: 198506 | | | |
| Image | ECAD Model \downarrow | D.S. | Description | Manufacturer | Prices / Stock | |
| OF. | C 4 | PDF | COILCRAFT - XFL4020-152MEC - INDUCTOR, PWR, 1.5UH, 9.1A,20%,59MHZ | COILCRAFT | XFL4020-152MEC | $\langle \square$ |

Clicking the **Download ECAD Model** button will download the model from the website into your libraries and onto the end of your cursor ready for placement in the design. Attributes are automatically added to the Part depending on the section you download it from, therefore if we click **Download ECAD model** from Farnell, we would then also see the Farnell Part Number on the part in Pulsonix.

| Can | COILCRAFT - XFL4020-152MEC - | Visit Famell Product Page Download ECAD Model Datasheet INDUCTOR, 1.5UH, 9.1A,20%,59MHZ, REEL | Stock Farnell Part Number Manuf Part Number Pack Size Minimum Order Quantity Lead time ROHS Compliance | 0 2298252 XFL4020-152MEC 1 1000 56 Yes | From 1000 5000 10000 | To 4000 9000 | Cost (GBP) 0.733 0.696 0.661 |
|-----|------------------------------|---|---|--|---------------------------------|--|---|
| Cen | Fixed Inductors XFL402 | Visit Mouser Product Page Download ECAD Model Catasheet O AEC-Q200 1.5 uH 20 % 9.1 A | Stock Mouser Part Number Manuf Part Number Pack Size Minimum Order Quantity Lead time RoHS Compliance | 10715 994-XFL4020-152ME XFL4020-152ME 1 1 1 6 Yes | Fr EC 11 5 10 20 | om To 99 00 499 00 999 000 1999 000 2000+ | Cost (GBP) 1.68 1.34 0.811 0.746 0.695 |

Now the Part has been downloaded, you can investigate its attributes from the **Edit Part dialog** to see it has inherited the values from the online model, including the Supplier name and Suppler Part number.

| М | Pulsonix - [Part: C:\Users\ | Public\Documents\Pulsonix9.1\MasterLibraries\WestDev Internal.pal: XFL4020-152MEC] | - C | x c |
|------------|---|--|------------|------|
| : | <u>File E</u> dit <u>V</u> iew <u>I</u> nse | ert <u>S</u> etup <u>T</u> ools <u>W</u> indow <u>H</u> elp | | - 8× |
| : D | 🖻 🔒 🎒 🗠 🖂 | 🍋 ()) 🦇 🎕 🗟 💽 🐔 🛞 🌾 📄 🔍 🗟 🔍 🤤 🗦 🖨 😓 🖕 🦷 | | |
| | Attributes | | | |
| | Manually Name Parts | XFL4020-152MEC | | |
| ©∾ | Supplier_Name | Farnell / | | |
| $ \times $ | Farnell Part Number | 2298252 | \sim | 2 |
| | Manufacturer_Name | COILCRAFT | | |
| 5 | Manufacturer_Part_Nu | XFL4020-152MEC | 120-152MEC | - |
| 2 | Datasheet Link | https://componentsearchengine.com/Datasheets/2/XFL4020-152IMEC.pdf | | |

Updating Imported Parts to Company Standards

It is always worth remembering that imported parts may need to be updated to your company standards. For example, this may mean adding attributes to Parts or changing text, pin and line styles on Symbols and Footprints to be in-line with your other library items.

This concludes the instructions for using the Pulsonix Component Search Engine.

Component Search Engine and Pulsonix in Dual Screen Mode

If you are using Pulsonix in Dual Screen mode, the Component Search Engine will automatically place the component symbol in the 'First' screen.

To change which environment you would like the Search Engine to place the symbol in, go to **Pulsonix**, **Options** dialog, **Multi-Screen** tab and click **Swap**.

| Enable Multi-Screen Support | |
|--------------------------------------|---|
| Windows for each type of design file | |
| Window for SCM and PCB designs | |
| Window#1 Window#2 | כ |

Closing the Library Loader Application

As the Library Loader will run in the background, you will need to close the background application if you wish to close it completely. To do this, select the **up arrow** on your **taskbar**, select the **Library Loader** icon and from the menu displayed, click **Exit**.



New In Version 10.0 - Not Previously Documented

The following features are new to Version 10.0 and have been added since the first release was created.

Accel Netlist Import – Part Mapping Dialog

When reading an **Accel Netlist** into Pulsonix PCB using the Accel Netlist Import facility from the **File** menu, the Part Mapping facility has been added. This enables the Part name to be mapped from one used in the netlist to one from your library.

Note: The import uses the *originalName* field in the *compInst* section in the Netlist for the part name.

| Import Accel/P-CAD Ascii Data | \times |
|-------------------------------------|----------|
| Use Part Mapping | |
| Pcb1.map | |
| Create Browse Edit | |
| Unplaced components to bin Show bin | |

One first run of the dialog, if no mapping files already exists, press the **Create** button. This will read the Parts in the Netlist and ask you for a name to save the mapping file to. You can edit the Parts with the dialog displayed.

| Orig. Part | New Part | Orig. Footprint | New Footprint |
|--------------------|--------------------|-----------------|---------------|
| C Polarised | C Polarised | | |
| 22FLZ-SM2-TB345678 | 22FLZ-SM2-TB345678 | | |
| R | R | | |

The **Browse** and **Edit** buttons enable you to choose an existing mapping file or edit an existing mapping file.

Note, the mapping file is an ASCII forms based file and can be edited if you understand the format and are careful.

Netlist Output to SiMetrix Spice Simulator

For users who have independently purchased SiMetrix Spice, there is now a specific output to write a netlist for the SiMetrix and SimPlis products. This is an extended version of the standard Spice netlist and is formatted for use with these simulators.

From the **Simulation** menu and **Set Netlist Spice Type**, use the **Spice Netlist Type** drop down option and select **SiMetrix**. If the **Run Simulator** check box is selected, it will populate the command line switches for you.

Current Selection Mask Shown on the Status Bar

The current **Selection Mask** being used is now shown on the **Status Bar**. Double click on it to choose a mask.

Once selected, it will display <None> if no mask being used (where the **Enable Selection Mask** switch is selected off in the main browser window), or <Default> if the default setting is being used or *User_Selection* which will be a named user-defined selection mask.

| Choose Selection Mask | × |
|---------------------------------------|---|
| Copper | |
| <none> <default></default></none> | |
| Copper | |

You can control whether it is displayed on the Status Bar by using the **Show Mask on Status Bar** switch on the main **Selection Mask bar**.

| Selection Masks | | × | |
|---------------------|---|---------------------------------|--|
| -jaj | 5 | Save New | |
| Enable Sele | ~ | Floating | |
| <default></default> | | Docking | |
| Area | | Tabbed Document | |
| Attribute | | Auto Hide | |
| Board | | Hide | |
| Breakout | | New Selection Mask | |
| | | Copy Selection Mask | |
| Constructio | | Reset Selection Mask | |
| Dimension | | Enable All | |
| Doc Symbo | | Disable All | |
| | | Import | |
| Additional | | Export | |
| Track Style: | X | Deselect All | |
| | | Select All Items Using Mask | |
| Attribute: | | Select Visible Items Using Mask | |
| | ą | View Selection | |
| Is: exists | ~ | Use Selection Mask Cursor | |
| [| ~ | Show Mask on Status Bar | |
| Override S | • | World View W | |
| Frame Select | | Component Rin R | |

STEP Output – Settings Menu Change

In the release version of Pulsonix 10.0, there was a change to the **Output STEP** dialog which meant that the settings switches were moved from the main front dialog to an inner **Settings** tab of the **3D Settings** dialog. This hadn't been previously documented.

| Output STEP | | × |
|----------------|--------------------------------------|-------------|
| STEP Filename: | c: \users \designs \step example.stp | Browse |
| Output Destina | ation | 3D Settings |
| | Output Preview Cancel | |

STEP Output – Define Silkscreen Layer

Previously, the STEP output was looking for the key word *Silkscreen* when outputting silkscreen data into the STEP file or STEP Preview window. This meant that if your silkscreen layer wasn't named 'silkscreen' or perhaps was been written in a different language, the silkscreen would not appear in the file. This has been replaced with the ability to define the **Silkscreen Layer Class** using the drop down list box displaying the **Layer Class** names in your **Technology**.

| 3D View Settings - Settings | × |
|--|---|
| Settings Colours Interaction Enclosures View | |
| Board Thickness: 1.580 | |
| Mounting Drill Holes No Components | |
| Advanced Settings (these are likely to increase generation time and file size) | |
| □ <u>P</u> ads □ <u>Vi</u> as □ Drill <u>H</u> oles | |
| ☐ <u>T</u> racks ☐ <u>C</u> opper ☐ <u>D</u> oc Shapes ☐ Te <u>x</u> t | |
| Silkscreen Layer Class: Silkscreen | |

Fillet and Mitre Corners for Differential Pairs and Bus Route

When using the Interactive High Speed option, you can now Fillet corners on a Differential Pair and Bus route.

Differential Pairs - Fillets

The ability to Fillet and Mitre corners in a Differential Pairs has been added.

Interactive Differential Pair Routing

During the routing of Differential Pairs, from the context menu, you can select the new Segment Mode of **45 Degree (Fillet)** to created filleted corners.



Existing Differential Pairs

With an existing Differential Pairs created, by selecting an existing mitred corner or the whole path (using Shift-select), you now have the ability to add Fillets to the corners.

From the context menu, two new commands exist, **Auto Mitre Selection** and **UnMitre Selection**. The **Auto Mitre Selection** option will use the settings from the **Technology** file, **Parameters**, **Auto Mitre** page.

| Curved Mitres | Maximum Mitre Size: | 50.0 |
|---------------|---------------------|------|
| Any Angle | Minimum Mitre Size: | 12.5 |
| | | |



Bus Route - Fillets

Interactive Bus routing

When interactively Bus routing in PCB, you can now switch to the new segment mode **45** Angled (Fillet) using the option from the context menu.



Existing Bus Routes

With an existing Bus route created, by selecting an existing mitred corner or the whole path (using Shift-select), you now have the ability to add Fillets to the corners.

From the context menu, two new commands exist, **Auto Mitre Selection** and **UnMitre Selection**. The **Auto Mitre Selection** option will use the settings from the **Technology** file, **Parameters**, **Auto Mitre** page.



ODB++ Output includes Profile Cutouts

In the ODB++ format, the board outline is defined and drawn as a 'Profile'. The new switch **Include cutouts in the Profile** will enable the output to also apply Cutouts to the board (Profile). Some of the later versions of the ODB++ viewers have the ability to read cutouts in Profiles.

| port ODB++ | | | | × |
|------------------------------|------------------|---------|-----------------------------------|--------|
| Compressed Compressed | | | | |
| c:\user\documents\h | igh speed desigr | i.tgz | Bro | wse |
|] <u>U</u> se CAM/Plot Layer | Combinations | 🗹 Inclu | ude Windows Plots | |
| Layer Classes | | | | |
| Silkscreen | Silkscreen | \sim | Output Positive Power Plane | |
| Solder Mask | Solder Mask | \sim | | |
| Solder Paste | Paste Mask | ~ | Neutral Net Names | |
| Plated Board Cutouts | <none></none> | ~ | Pin Names in Packages | |
| | | | All Closed Component DocShapes (C | AM350) |
| | | | Include Cutouts in the Profile | |

CAM Plots – Auto Generate Plot Ordered By Variants

Within the Schematic editor, in the **CAM Plots** option, there is a new switch in the **Auto Generate Plots** dialog to choose to generate plots and to **Order by Variant**.

| Auto Generate CAM/Plots | × |
|--|---|
| A default set of CAM/Plots will be gen current contents of your design and th | erated based on the OK |
| make in this dialog. Any existing CAM, deleted before the new set is created. | /Plots can also be Cancel |
| For each Page: | Windows ~ |
| Delete Existing Plots First | Generate For Each Variant Order By Variant |

Report Maker

Rotation of Attribute Command

You can now report the Rotation of an Attribute position.

Rotation will output the rotation of the displayed attribute. This is the real rotation value, not the adjusted value for readable text. This will be reported in the rotation units and precision defined in the Units dialog.

Rotation (Transformed) will output the rotation of the displayed attribute with any design transformation applied also.

| Edit Attribute Command | × |
|---------------------------------|---|
| Command: Attribute "Value" Name | |
| Attribute: Value | ✓ □ 0r |
| Use: X Coord Y Coord (X , | Y) Value Name Inherited Value Is Valid Name Position |
| New Column | Field Informatio Rotation Rotation (Transformed) |
| Column Width: 0 | Pre-Text: Validation |
| Column Justification: Centre | Post-Text: |

Indent & Unindent Commands

| mat: Net List | | Description: Net List Output | 0 | |
|--|-----------------------------|--|--|----------|
| New Save | Save As | Available for: PCB PCB Library Parts Libra Schematics SCM Library Papel | ry ∏⊻ault | |
| Pad Commands: | F | ormat Script: | | |
| Angle Units Attribute Bank Lines Cancel Report Coordinate Units Design Units Draw Line Draw Line Draw Line Draw Line Draw Line Draw Line Draw Line Control Line Finish Report Is Drawing Report Line Number Multi Line Text Name Range Format Number Format Run Report Selection Set Clip Area Set Clip Area | ▲dd >> If >> While >> | List of Nes Net Name List of Nes List of Nes List of Nis Lift of Nis | Edt Copy Delete Disable Up Down Indent Unindent Delete All Undo Redo | ⟨L ⟨Ţ |

From the main **Report Maker** dialog, there are two new buttons on the right side of the pane for **Indent** and **Unindent**.

Indent

Use this to add indentation to a set commands. Indentation can be used for a couple of reasons:

- To add clarity to a portion of commands in a script. It might be a long IF statement that you need to indent to view clearer.
- By Indenting a section of script, even commands outside of IF and ELSE statements, perhaps commands that follow the IF statements, such as End Of Line and Blank Line, you can then opt to Copy the commands or move them to another part of the script.

When the Indent button is pressed, after typing in a **Comment** into the dialog, you must select the portion of script required for indentation. This is an interactive command so drag the mouse up and down to select the portion of script required.

| Edit Fixed Command | × |
|---|---|
| Command: | |
| Comment: Declare variable and search for Attributes | |
| | |
| | |
| | |
| NOTE Alter constant OK selective lest constants to be indepted. | |
| NUTE: Arter pressing UN, select the last command to be indented | |
| OK Cancel | |

| i I Declare Variable, process Attributes If "Attribute: Part Number Value" is not blank Attribute: "Part Number" Value Set Variable "att" to Attribute: Part Number Value Replace Variable "att" text "Variable: newline" with "vt\n" Variable "att" Text "(textStyleRef "(Default)"))" |
|---|
| |

The comment is displayed in the script along with an indentation indicator.

This shows the indentation indicator:



The indent can then be 'closed' up to further aid clarity of your script. The Comment is still displayed in the script.



Unindent

Use this to remove the Indented status on a selected set of commands.

New In Version 10.0

The following documentation details features specifically for Version 10.0.

Technology Dialog Changes

Enhanced Technology Dialog

The Technology dialog has been further enhanced with the inclusion of **Colours**, **Units**, **Grids**, **Variants**, **Design Settings**, **Naming**, **Adjacent Nets**, **Advanced Router Parameters**, (base) **Router Parameters**, **Autoplace Parameters**, **Auto Gate/Pin Swap** and **Auto Mitre Parameters**.

| 🖭 Spacing Rules | | | | | | | | | |
|---------------------|--|--|--|--|--|--|--|--|--|
| | | | | | | | | | |
| 🛓 Layers | | | | | | | | | |
| | | | | | | | | | |
| Rules - DFM/DFT | | | | | | | | | |
| Rules - High Speed | | | | | | | | | |
| • Outputs | | | | | | | | | |
| 🖕 Naming | | | | | | | | | |
| Attribute Names | | | | | | | | | |
| Group Names | | | | | | | | | |
| Hatch Style Naming | | | | | | | | | |
| - Line Style Naming | | | | | | | | | |
| Pad Style Naming | | | | | | | | | |
| Text Style Naming | | | | | | | | | |
| Track Style Naming | | | | | | | | | |
| Colours | | | | | | | | | |
| 🚊 Grids | | | | | | | | | |
| Grid Setup | | | | | | | | | |
| Interactive Grid | | | | | | | | | |
| 🚊 Units | | | | | | | | | |
| Units | | | | | | | | | |
| - Variants | | | | | | | | | |
| Variants Manager | | | | | | | | | |

In addition to existing design facets, such as Colours and Units, version 10 also adds new functionality for auto-naming of style when required. Naming has been added under **Naming** for **Hatch Styles, Line Styles, Pad Styles, Text Styles and Track Styles**. See below for more details.

This means that many design Technology contents are now in one location.

Technology Dialog Tree Structure

You can now open and close all tree structures in the Technology dialog from the context menu.



New Style Naming pages for the Technology Dialog

Style Naming Dialog

As a switchable option, you can opt to have Pulsonix automatically create style names for you. This means that, for example, when creating new pad styles, you could use the auto-naming option to use the IPC naming convention.

Style Naming is available for all **Styles** within the Technology.

Each page works like any other Rules page within the Technology except that generally speaking, you'd only ever have one or two rules for each naming style.

| Line Pattern | Match Line Width | Line Width | Template String | |
|--------------|------------------|------------|--|--|
| Solid | | 0.00 | Line <pattern> <width></width></pattern> | |

The edit portion of the dialog is used to define specific naming items, this varies for each item style. The right side of the edit pane defines the **Rule** that will be created. Once the rule has been defined using keywords selected from the **Field:** drop down list, use the **Insert** button to add it to the Rule.

| Line Pattem: <any> > Match Width Less Than or Equal To: 0.00</any> | Rule: Line <width:><metric:> Field Keyword</metric:></width:> | ert |
|---|---|-----|
| | Field: Line Width ~ | |
| | Pre-text: | |

Buttons on the Navigation bar under **Replace All:** enable you to add a rule to the rules pane replacing any existing rules. A button on all pages (**Basic Set**), is used to set up a basic rule for you to use. This can be edited to your own style if required. On the **Pad Style Naming** page, an extra button (**IPC Set**) is used to define a more complex string for the IPC naming rule.

Basic Set & IPC Set Buttons

On each of the pages is a button named **Basic Set**. On the **Pad Style Naming** page, there is an additional button for the **IPC Set**. These buttons are under the heading **Replace All:** and are used to create you a basic style naming rule tailored for each of the particular styles. As it says, using this button will add you one style naming rule to the grid and will remove all other rules defined. The rule won't be applied until the **Apply** or **OK** buttons are used at the bottom of the dialog.

| | Settings | |
|---|---------------------------|--|
| | Replace All: Basic Set | |
| > | IPC Set | |

Note: When using the **IPC Set** button, this also sets up the **Settings** dialog as well as the rule definition. The IPC rule is a combination of both the rule and pad style/usage mappings.

Style Naming Settings Dialog

Each of the Style Naming dialogs has a **Settings** button. Each one is the same except for Pad Style Naming which has additional settings that can be used. The Settings dialog defines the **Units** and **Units Text** for the Style Naming. This is independent of the main Design Units dialog and is only used for Style Naming.

The standard Settings dialog looks like this:

| Style Naming Settings | | | | | | | | |
|--|--|--|--|--|--|--|--|--|
| Naming Units | | | | | | | | |
| Default to: Imperial | | | | | | | | |
| OMetric | | | | | | | | |
| Imperial | | | | | | | | |
| Unit: thou ~ | | | | | | | | |
| Unit Text: thou Trailing Zeros | | | | | | | | |
| Precision: 2 | | | | | | | | |
| Metric | | | | | | | | |
| Unit: mm 🗸 | | | | | | | | |
| Unit Text: mm 🗹 Trailing Zeros | | | | | | | | |
| Precision: | | | | | | | | |
| OK Cancel | | | | | | | | |

The **Pad Style Naming Settings** dialog also enables additional **Mappings** for **Shapes** and **Pad Usage**.

| Shape Name Mappings | | Naming Units |
|---------------------|-------|----------------------------------|
| Shape | Label | Default to: O Imperial |
| Annulus | с | Metric |
| Bullet | d | Terrendel |
| Chamfered Rectangle | r | |
| Diamond | u | Unit: thou ~ |
| Octagon | S | Unit Text: thou I Trailing Zeros |
| Oval | b | |
| Pointer | u | Precision: 1 |
| Rectangle | r | |
| Round | C | Decimal Character: |
| Rounded Rectangle | r | Metric |
| Square | S | Unit: mm |
| Target | u | |
| Triangle | u | Unit Text: mm 🔽 Trailing Zeros |
| | | Precision: 2 |
| Pad Use Mappings | | Decimal Character: |
| Pads Intended Use | Label | |
| Through Hole | | |
| Surface Mount | | |
| Via | V | |
| Micro-via | V | |
| Anv | | |

This is used where specific styles require mapping so that can be correctly defined, the IPC naming style for example.

Technology Styles Dialog

In addition to the Style Naming dialog for each of the styles, the actual **Style** page has been modified too to enable the naming.

Under each of the **Name:** fields for the style, there are now three additional radio buttons to define how the style will be **Named by:** There is a choice of **Typed** (manually named), **Rule** (using the new Style Naming rules for that style item) and **Template** (which is a hybrid, a combination of both Rule and Typed).

| | Na <u>m</u> e: | Used: | Shape: | | | Drill: | |
|---|------------------------|------------|-----------------|--------|--------|-----------|--------|
| | Mounting Hole | | Type: | Round | ~ | Shape: | Round |
| | Named by: Typed Rule |) Template | Width: | 3.8100 | | Width: | 1.1430 |
| / | Mounting Hole | | Length: | 2 9100 | | Length: | 0 0000 |
| | For Use By: | | Eongen | 3.0100 | | Longai. | 1 |
| | Through Hole Pads | | | | | Inner Dia | meter: |
| | Surface Mount Pads | | <u>O</u> ffset: | 0.0000 | 0.0000 | Offset: | 0.0000 |
| | Vias | | | | | Datation | nn |

Technology General/Defaults Dialog

From with the **Technology** dialog and **Design Settings**, **General/Defaults** page, there is an additional switch related to **Style Naming** called **Make named styles use naming rules**. When selected, this will force the addition of new named styles use the naming rules. Leaving it unchecked enables a **Typed** rule to be defined.

| Attributes |
|--|
| Substitution Char: % (when doubled) |
| Matching Styles |
| O By Name Only |
| By Name And Value |
| O By Value Only |
| Naming Rules |
| Make named styles use naming rules |
| Pad Exceptions |
| Mirror with Component |
| |
| |

Spacing Check of SMD Pads & Micro-vias

As with Pads, Mounting Holes and Vias, you can now set **Spacing** rules for **SMD Pads** and **Microvias**.

| | | | Design | | | | | | | | |
|-----------|---------------|-------|--------|---------|-------|-----------|-----------|------------------|--------|-------|--|
| | Design | Track | Pad | SMD Pad | Via | Micro-via | Testpoint | Mounting Hole | Copper | Text | |
| | Track | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | |
| | Pad | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | |
| \square | SMD Pad | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | |
| | Via | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | |
| \Box | Micro-via | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | |
| <i>V</i> | Testpoint | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | |
| | Mounting Hole | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | |
| | Copper | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | |
| | Text | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | |
| | Board | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | 0.254 | |

Pad Exceptions By Layer Class & Inner Electrical layers

You can now create **Pad Exceptions** by **Layer Class** and for **Inner Electrical** layers. Both options are available when using the **By Layer** option and **Exception Type:** field for a selected **Pad Style** in the **Technology** dialog.

| Name: | Shape: | |
|--|---------------------|---|
| Mounting Hole | Type: Round 🗸 | 7 |
| Named by: Typed Rule Template | Width: 3.810 | _ |
| Exception Type: | Length: 3.810 | |
| By Layer Class 🗸 🗸 | | |
| By Layer By Layer Class Inner Electrical | Offset: 0.000 0.000 | |
| Spacing Shape Micro-via Entry Pad Micro-via Stop Pad | | |

Acid Trap Rules

To facilitate additional functionality and checking, the **Design Spacings** page has been changed in the **Technology** to add spacing rules for **Acid Traps** under the **Tracks** tab. This checks for acid traps between Track segments and between Tracks and Pads.

| Additional | Design Lovel Spacin | an and Pulse | | | | | | |
|------------|-----------------------|----------------|--------|-------|-----|-------|------------|--|
| huulliona | i Desigit Level Spaci | iys and hules. | | | | | | |
| Board | Bond And Die Pad | Component | Copper | Drill | Pad | Track | Enclosures | |
| | | | | | | | | |
| Aci | d Traps | | | | | | | |
| Min | imum Allowed Angle: | 45.0 | | | | | | |
| Min | imum Allowed Gao: | 0.00 | | | | | | |
| I MILL | anum naoweu uap. | 0.00 | | | | | | |

An acid trap is considered to be an acute angle created between design items, usually tracks and pads (as demonstrated below).



As well as the detection of acute angles (less than the **Minimum Allowed Angle** defined), the angle also represents calculated angles based on tracks closer than the gap between the pad and the offending segment. This is defined as the **Minimum Allowed Gap**.



There is a new Acid Trap check box on the DRC dialog under the Manufacturing section.

| | | Copper Text On Board | Pin Order |
|----------|----------|----------------------|--------------------|
| | ✓ Drills | Panel Items On Board | Differential Pairs |
| | | Copper Shapes | Stub Vias |
| | | 🗹 Acid Traps | Parallel Track |
| <u> </u> | | Testpoints | Necked Track |

Enclosure Spacing Rules

To facilitate the changes within the **3D STEP Preview**, the **Design Spacings** page has been changed in the **Technology** to add spacing rules for **Enclosures**. *This is discussed below in more detail in the 3D STEP Changes section*.

| Addition | al Design Level Spacir | ngs and Rules | : | | | | |
|----------|--|-------------------------------------|--------|-------|-----|-------|------------|
| Board | Bond And Die Pad | Component | Copper | Drill | Pad | Track | Enclosures |
| F | Inclosure Space Enclosure to Board: Enclosure to Compon Enclosure to Enclosur | 0.500 ents: 1.000 re: : 1.000 | | | | | |

Select Net Names by Attribute

There is a new Select button on the Net Names page of the Technology dialog.



| | Name | Net Class | Туре | Guard Space | Check Same Net | Use Own Colour | Colour | Display Connection | Attr |
|---|----------|-----------|--------|-------------|-------------------|-------------------|--------|-----------------------|------|
| Y | +5V | | Power | 0.00 | | | | | |
| Y | A | | Signal | 0.00 | | | | | |
| Y | A1 | | Signal | 0.00 | | | | | |
| Y | A2 | | Signal | 0.00 | | | | | |
| Y | ADD1 | | Signal | 0.00 | | | | | |
| Y | ADD2 | | Signal | 0.00 | | | | | |
| Y | ADD3 | | Signal | 0.00 | | | | | |
| Y | ADD4 | | Signal | 0.00 | | | | | |
| Y | ADD5 | | Signal | 0.00 | | | | | |
| Y | ADD6 | | Signal | 0.00 | | | | | |
| Y | ADDR_DEC | | Signal | 0.00 | | | | | |
| Y | ALARM | | Signal | 0.00 | | | | | |

Once your Match: is chosen, pressing OK will make the matching selection in the Net Names grid.

You can then use **Apply to selected cells** on the context menu, to, for example, set the **colour** of all nets with a matching attribute.



Variant Manager now a grid Dialog

Along with the inclusion of the **Variants** option within the **Technology** dialog, this dialog has now been changed to be grid based similar in style to the other Technology pages.

| | Current | Name | Description | New. |
|---|---------------|-----------------------------|----------------------------------|--------------|
| | | <master design=""></master> | Entire Contents of Master Design | _ |
| Υ | | USA | 120V variant | <u>C</u> opy |
| Υ | | GB | 240V variant | |
| Y | | EU | 220V variant | |
| | | | | Delet |
| N | a <u>m</u> e: | EU | | |
| U | sed: | \checkmark | | |
| D | escription: | 220V variant | | |
| | | | | |
| | | | | |
| | | L | | |
| G | obal Setting | S | | |
| | | | Not-Fitted Components: | |
| V | Show Vari | iant on Status Bar | Display | |
| | | | | |

Initially in a design, variants will not be in use. Press **New** to create the first variant. This will automatically create the **<Master Design>** entry in the Variants List. This entry is always at the top of the list and cannot be deleted while other variants exist.

Each press of the **New** button will add a new variant to the list. Select a variant from the list and check the **Current** button to make the one the current variant.

New Variants Button

Note, the **Use Variants** button has been removed. Adding a **New** variant will enable variants for designs with no variants currently defined.

New Adjacent Nets Rule

There is a new rule under **Technology**, **Rules** – **DFM/DFT** called **Adjacent Nets**. This allows you to set up rules to control what nets are checked against each other, check a net against a board outline and check a net against all items not on a net; that is, nets that match the rule and are within an optional tolerance. You may use this to check for nets that are inside the **Spacing Rules** defined but where you require knowing the closest item to the net being checked.

Options on this dialog control how many warnings are generated and whether the set of nets is checked as a whole, or each net checked individually.

Warnings are listed separately in the Errors bar and cannot be locked.

| Check | k Nets | Nets Against N | | Check | | | Check On | | Adjacent Nets | | | |
|---------------------------|---------------------|--|-------------|---------|---------------|---------|--|----------|--------------------|---------------------|---|-----------------|
| Attribute Name | Match Value | Attribute Name | Match Value | Ag B | ainst oard | Side | Layer | Area | Individual Nets | Within Tolerance | Tolerance | Max Warnings |
| <net name=""></net> | * | <net name=""></net> | * | | | | | | | | <unrestricted< td=""><td>1</td></unrestricted<> | 1 |
| Check Nets: Attribute: | <net name=""></net> | | | Cheo | ck Aga | inst Bo | ard | | | | | |
| Match: | • | `````````````````````````````````````` | / 🔊 | | | | | | | | _ | |
| Against Nets | : | | | | - | - | | | | | _ | A1 |
| Attribute: | <net name=""></net> | `````````````````````````````````````` | / | | | | | | 1 | _ | — | D.(|
| Match: | | · · · · · · · · · · · · · · · · · · · | × 🕺 | | - | | | | | | | B1 |
| On Layers: | | | | Cheo | ck Eac | h Indiv | idual Net | | | | | B2 |
| Side: or or Laver: | <any></any> | ~ | | With | in Tole | rance: | <unres< td=""><td>tricted></td><td>]</td><td></td><td></td><td></td></unres<> | tricted> |] | | | |
| Within Areas: | | · · · · · · · · · · · · · · · · · · · | Ma | ximu | m Wan | nings F | or Each (| Check: | 1 ≑ | | | |

With the flexibility of Pulsonix, the rule can be defined very broadly, such as all nets to all nets, or it can be defined more specifically, such as Net Name XXX against Net YYYY on Layer X.

Additional rules can be also defined to refine the rule depending on your requirement.

The general rule without the extra check boxes selected will select use a basic adjacency rule.



Checking the Check Against Board check box enables checking between nets and the board outline.



Checking the **Check Each Individual Net** will check every net pair combination in the two sets, and not each set of nets as a whole.



Selecting the **Within Tolerance** will define a tolerance value, so that only near points less than that tolerance are reported.



Maximum Warnings For Each Net - this check box enables you to define the maximum number of warnings that each net can produce. Enter a value between 1 and 10. A large value may cover your design with warnings, meaning they are difficult to resolve. If there are many warnings, you should check your spacing rules.

Adjacent Nets Checks in DRC Dialog

A new check in **Nets** section of **DRC** dialog has been added for **Adjacency Nets**. This reports the closest points of adjacent nets with a warning marker. This check runs in its own thread.

Adjacent Nets in Report Maker – List of Errors

Report maker can detect the **Adjacent Nets** errors in a **List of Errors** command by using the **Is Warning** command. You can report the net names using the new **Item1Name** and **Item2Name** commands.

Define Pad Shapes now on Pad Styles dialog

As well as the **Define Pad Styles** option on the **Setup** menu, it has now been included as a button in the **Technology** dialog under **Pad Styles**.



Design Settings now in Technology Dialog

Design Settings Menu

Design Settings are now included as part of the **Technology** dialog. They are listed under their own menu item, **Design Settings**.

| Technology [] - Design Settings - Spacing Rules Styles Layers Nets Rules - DFM/DFT Rules - High Speed Outputs Naming Colours Grids Units | General/Defaults All Text Adjust To Readable Orientation Barring Character: (when doubled) System Font Proportional Width Digits CAM/Plot Transform about design extents STEP | Attributes Substitution Char: 1/2. (when doubled) Matching Styles By Name Only By Name And Value By Value Only Make named styles use naming rules Pad Exceptions |
|--|---|--|
| H. Variants | No Model Transformations | Mirror with Component |
| Design Settings Coordinate System General/Defaults Naming Placement Sites Synchronisation Parameters | | Defaults |

Defaults Pages

In the **Technology** dialog, **Design Settings** menu, **Defaults** pages are now available on the **Defaults** button which is on the **General/Defaults** page.





Grids Menu

Grids are now included as part of the Technology dialog. In the **Technology**, Grids menu, Current Interactive Grid is now replaced with Interactive Grid.

Grids Displayed Switches

There is a new switch in the Grids dialog, Only During Editing.

| Colour | Draw nth Step | Style ● Doţs ○ Lines ○ Crosses ○ X Crosses |
|-----------------|-------------------------|--|
| Gridg Displayed | Grid Only ng Editing | |

With **Grids Displayed** checked, when **Display Interactive Grids Only** is checked, the new **Only During Editing** switch decides what happens when in select mode. If **Only During Editing** is checked then when in **Select** mode will display all grids that are marked as **Displayable** (as it did in previous versions). If unchecked, select mode will display the interactive grid.

Locked Grid Origins

The ability to lock a grid to an origin of a particular type has been added. With the **Lock** check box selected, if the origin is moved, the grid will automatically update to be relative to it still.

| Different X & Y | |
|---|-----------------|
| Origin | |
| X 0.000 Board Centre | |
| Y 0.000 Coordinate Origin | \Box_{\prime} |
| Relative Origin | |
| Apply Origin Settings To All Grids | V |
| Displayable Colour Draw nth Step Style Onts | |

Apply Origin To All Grids

A new check box labelled **Apply Origin Settings To All Grids** has been added under **Grid Setup** dialog. All grids will take on the settings from a grid the check box is applied to, e.g. Working grid can use a **Locked Coordinate** that is then applied to all other grids.

| | | Relative Orig | n 🗌 Lock |
|--------|----------------------|-----------------|--|
| \Box | Apply Origin Setting | ıs To All Grids | |
| | Colour | Draw nth Step | Style Dots Lines Crosses X Crosses |

3D STEP Preview Changes



The **3D STEP** engine (STEP Preview), interaction, preview and environment within Pulsonix has been significantly reworked for Pulsonix 10.

Gradient Background (switchable between Solid and Gradient)

The background can now be switched from a single solid colour to a gradient background between two colours defined in the **3D View Settings** dialog under **Colours**. You can setup the colours prior to viewing your 3D Preview from the **Output** menu, **OUTPUT STEP** dialog and **3D Settings**, **Colours** tab, or from within the STEP Preview from **Setup** menu, **3D Settings**, **Colours** tab.

| Output STEP | | × |
|----------------|-----------------------------|-------------|
| STEP Filename: | \reference\designs\step.stp | Browse |
| Output Destina | Output Preview Cancel | 3D Settings |

Both the **Background** colour and **Gradient** colours can be defined. Without the Gradient check box enabled, just the Background colour chosen will be used as a solid colour. With the Gradient check box selected, the background will be graduated between the background colour and the Gradient colour from the top to the bottom of the screen.

| Settings Colour | Interaction Enclosures View | |
|-----------------|------------------------------------|------------------------------------|
| Board: | Dark Green 🗸 🗸 🗸 | Board (substrate) |
| Copper: | Dark Goldenrod | Electrical layers |
| Components: | Charcoal 🗸 🗸 🗸 | Default components |
| Holes: | Seashell 🗸 🗸 | Drill Holes |
| Non-Electrical: | Bisque V | Non-elec layers |
| | Lavender ~ Gradient ~ | Default Colour Secondary Colour |
| Enclosure | White V | Enclosure Colour |
| | Transparency Transparent Opaque | |
| | | |

Improved Visualisation of the STEP Preview

As a visualisation improvement generally for the 3D Preview, anti-alias rendering on models has been introduced. This provides smoother edges and a more realistic look to the design. In addition, the lighting of the design has been improved to add to the realism.

Print option in STEP Preview

The **Print** option has been removed from the **File** menu and a **Save As Image** option has been added instead.

Toolbar with Icons of Pre-defined 3D Views

Within the 3D STEP Preview, there is now a toolbar containing pre-defined 3D Views available for selection. Views have been provided (moving from left to right across the toolbar) for **Axonometric**, **Front**, **Back**, **Left**, **Right**, **Top** and **Bottom**.



For the perspectives used in **Axonometric** view, please see below under *Perspective and Orthographic Views*.
Perspective and Orthographic Views

For the **Axonometric** view, you can switch between **Perspective** and **Orthographic Views** depending on your preference. This can be changed from the **Setup** menu in the 3D environment and **View** tab or from the PCB design, **Output** menu, **STEP**, **3D Settings**, **View** tab.

| 3D View Settings - View | × |
|---|---|
| Settings Colours Interaction Enclosures View | |
| View Projection Type Perspective Orthographic | |

Note: when the perspective is changed, this is only applied when you next run the STEP Preview, it is not changed for this Preview session. A warning to say this is issued when the perspective is changed.

Rotation of Design in STEP Preview now on Right Hand Mouse button

Rotation of the design in the STEP Preview is now done using the **right-hand mouse button with the Ctrl button** also pressed. Previously, this was done using the left-hand mouse button. The movement of this command to the right-hand mouse frees up the ability to select items using the left-hand mouse (*this is discussed below in more detail*).

Selection of items in STEP Preview on Left Hand Mouse button

Using the **left-hand mouse button**, you can select both components and enclosures when they have been checked as selectable from the **Setup** menu in the 3D environment and **Interaction** tab or from the PCB design, **Output** menu, **STEP**, **3D Settings**, **Interaction** tab.

| J view settings - interaction | > |
|--|---|
| Settings Colours Interaction Enclosures View | |
| Selectable: | |
| Components | |
| | |
| | |
| Allowing Moving of 3D Objects | |
| | |
| Automatically Synchronize 2D | |
| Automatically Synchronize 2D Use Component Grid | |
| Automatically Synchronize 2D Use Component Grid Use Clash Detection | |
| Automatically Synchronize 2D Use Component Grid Use Clash Detection Board To Enclosure | |
| Automatically Synchronize 2D Use Component Grid Use Clash Detection Board To Enclosure Component To Component | |
| Automatically Synchronize 2D Use Component Grid Use Clash Detection Board To Enclosure Component To Component Component To Enclosure | |
| Automatically Synchronize 2D Use Component Grid Use Cash Detection Board To Enclosure Component To Component Component To Enclosure Enclosure To Enclosure | |

With this option enabled, if the mouse is moved over the either a Component or enclosure within the STEP Preview a wire frame selection colour will be added to the item.



If you use a single click of the mouse over the model, the model movement axes are displayed indicating the movement options available to you. This is displayed if the Component or Enclosure is marked as **Selectable** on the **Settings, Interaction** dialog.

Components will be able to move in an X and Y direction, whereas enclosures can be moved in X, Y, Z and Rotational axes.



Move Component on Grid in STEP Preview

There are two settings for moving Components in the 3D STEP Preview; on the **Component Grid** and without. The default mode is for the movement to be on the **Component Grid** (as defined in the PCB design). It is recommended not to change this setting.

| | so new settings interaction | |
|-----|--|---|
| | Settings Colours Interaction Enclosures View | V |
| | Calentable | |
| | Selectable. | |
| | | |
| | | |
| | | |
| | | |
| ĸ | Allowing Moving of 3D Objects | |
| \ | Automatically Synchronize 2D | |
| -) | Use Component Grid | |
| | Ulas Clask Datastian | |
| 1 | Use Clash Detection | |
| | Board To Enclosure | |
| | Component To Component | |
| | Component To Enclosure | |
| | | |
| | | |

Moving Components in the 3D Preview will use the Component Grid defined in the PCB. This is dynamic, meaning that if the grid is changed in the PCB (with the 3D Preview still enabled in another window), the grid step will change in the 3D Preview to match. Likewise, if you change the PCB grid to use another grid, such as the <Working> grid instead of the <Component> grid, this will now also be reflected in the 3D Preview and be used.

As an application note, you may find it easier to move Components within the Top view perspective rather than a 3D view.

Automatically Synchronise 2D PCB from 3D

You can opt to have Component and the Enclosure positions that are changed in the 3D Preview automatically made back in the 2D PCB view.

To do this, select the **Automatically Synchronise 2D** check box on the **Interaction** tab of the **3D Settings** dialog. By default, this will be enabled.

| Settin | gs Colours | Interaction | Enclosures | View | |
|--------|--------------|---------------|------------|------|-------|
| Sele | ctable: | | | | |
| | Components | | | |] |
| | nclosure | | | | |
| | | | | | |
| | llowing Movi | ng of 3D Obje | ects | | |
| | _ | | | | |

Working in both 2D and 3D Environments

If you intend working in both the 2D PCB environment and 3D Preview at the same time, and moving components around you should be aware of the following:

- If Components are moved in the PCB, then the 3D Preview must be regenerated (close it, then regenerate using the **Output** menu and **STEP Preview**).

- If Components are moved in the 3D Preview, then provided the **Automatically Synchronise 2D** check box on the **Interaction** tab of the **3D Settings** dialog is checked, then the component move will be automatically reflected in the PCB without doing anything else.

If adding Enclosure STEP models within the STEP Preview environment, these are also back annotated to the PCB design.

Enclosure Spacing Rules

New **Design Spacing Rules** are available for **Enclosures**. These are defined in the PCB design and transferred to the STEP Preview when the dialog is opened.

Spacing rules available for **Enclosure to Board**, **Enclosure to Components** and **Enclosure to Enclosure** will be used when the design is transferred into the **3D STEP Preview** and used for clash reporting.

| Board | Bond And Die Pad | Component | Copper | Drill | Pad | Track | Enclosures |
|-------|-----------------------|-------------|--------|-------|-----|-------|------------|
| | | | | | | | |
| Enc | closure Space | | | | | | |
| | Enclosure to Board: | 0.500 | | | | | |
| | Enclosure to Compon | ents: 1.000 | | | | | |
| | Enclosure to Enclosur | e: : 1.000 | | | | | |

See below under Clash Detection within the 3D Environment for how the Spacings are used.

Adding Mechanical Enclosures

Mechanical enclosures in STEP format can be included along with the Board and Components (using STEP models). There is a new **Enclosures** page in the **3D View Settings** dialog of the **Setup** menu when within the STEP Preview to add Enclosures or other STEP models that are not defined as Components to the STEP Preview. You may also add more than one enclosure to the design for use in the Preview. This is also available from the PCB design, **Output** menu, **STEP**, **3D Settings**, **Enclosures** tab.

| 3D View Settings - Enclosures | × |
|---|-----------------------|
| Settings Colours Interaction Enclosures | 1ew |
| Enclosures: base scaled.step | New Edit Delete |
| | |

On selection of the **New** button, the **Position STEP Model** dialog is displayed. This will allow you to add a new or additional STEP models:

| Position STEP Model X |
|---|
| Additional STEP Model STEP Model : base scaled.step |
| |
| |
| |
| |
| |
| $\sum_{i=1}^{Z} \sum_{i=1}^{Y}$ |
| Offsets X: 0.0 • Y: 0.0 • Z: 13.0 • |
| Increment: 0.100 |
| Rotations X: 0.000 	Y: 0.000 	Z: 0.000 	V |
| Colours OK Cancel |

If you select a model from the list, the Edit button will allow you to edit its position.

The **Transparency** of these STEP models is taken from the **3D** Settings, Colours dialog. Colours used are available on the Colours button which are replicated from the **3D** Settings, Colours dialog.

Transparency Setting for Enclosures

With the introduction of Enclosures and additional STEP models that are not Component related, you can set the **Enclosure Transparency** so that when overlaid on a board with Components, you can see the contents inside the enclosure. This is set on the **3D View Settings** dialog under **Colours**.

| ettings Colours | Interaction E | Enclosures | View | | |
|-----------------|----------------|------------|--------|--------------------|--|
| Board: | Dark Green | | ~ | Board (substrate) | |
| Copper: | Dark Goldenrod | | ~ | Electrical layers | |
| Components: | Charcoal | | ~ | Default components | |
| Holes: | Seashell | | ~ | Drill Holes | |
| Non-Electrical: | Bisque | | ~ | Non-elec layers | |
| Background | | | | | |
| | Lavender | | ~ | Default Colour | |
| | Gradient | | ~ | Secondary Colour | |
| Enclosure | | | | | |
| | White | | ~ | Enclosure Colour | |
| | Transparency | r | | | |
| | Transparent | (| Opaque | | |
| | | | | Reset | |

Colour Setting for Enclosures

Where enclosures are used and they have no colour defined of their own (this is normal as some are defined as neutral), there is a default colour setting that will be used by Pulsonix. On the **3D View Settings** dialog and **Colours**, there is an **Enclosure** colour that can be set.

| ettings Colours | Interaction Enclosures View | |
|-----------------|--------------------------------------|--|
| Board: | Dark Green Soard (substrate) | |
| Copper: | Dark Goldenrod | |
| Components: | Charcoal Default components | |
| Holes: | Seashell 🗸 🗸 Drill Holes | |
| Non-Electrical: | Bisque Von-elec layers | |
| Background | | |
| | Lavender 🛛 🗸 Default Colour | |
| | Gradient | |
| | Secondary Colour | |
| Enclosure | | |
| | White Colour | |
| | | |

Clash Detection within the 3D Environment

To use the **Clash Detection** feature within the STEP Preview, you must first enable it, then enable the options you wish to detect. These options are available in **3D View Settings** dialog and **Interaction** under **Use Clash Detection**.

If using the STEP Preview as simply a preview of the board, you may wish to switch off both the **Selectable** setting of **Components** and **Enclosures** and the **Use Clash Detection** options.

| 3D View Settings - Interaction | × |
|---|---|
| Settings Colours Interaction Enclosures View | |
| Selectable: | |
| Components Enclosure | |
| Automatically Synchronize 2D | |
| Use Component Grid | |
| Use Clash Detection | |
| Board To Enclosure Component To Component Component To Enclosure Enclosure To Enclosure | |
| Use Spacings | |

See below under Using The Clash Detection Option for information about using this option.

Using the Clash Detection Option

Once the **Clash Detection** option has been enabled (in **3D Settings, Interaction, Use Clash Detection**, see above), you can detect clashes using one of two options:

- Interactively move Components or Enclosures
- Or you can use the Detect Clashes option from the Tools menu

Once clashes are detected, they are highlighted in red.

To un-highlight clashes, select the Remove Clash Highlights from the Tools menu.

Colouring for non-coloured STEP Models

Where STEP models used for Components have not had their colour defined in the model, which some don't, Pulsonix will add a colour for you. Choose the **Components** colour under the **3D Settings** and **Colours** tab.

| Settings Colours | Interaction Enclosures View | |
|------------------|-----------------------------|--------------------|
| Board: | Dark Green 🛛 🗸 🗸 | Board (substrate) |
| Copper: | Dark Goldenrod | Electrical layers |
| Components: | Charcoal 🗸 🗸 | Default components |
| Holes: | Seashell 🛛 🗸 | Drill Holes |

Properties of Selected design items shown on Status Bar

For selected Components in the design, basic **Properties** (**Component Name** and **Part Name**) will be displayed on the **Status Bar**. It also shows you that a Component has been selected.



Speed Increase of STEP file Generation through Multithreading

With the introduction of multi-threading capabilities within the STEP file generation process, with drill holes selected, there is a significant speed increase. On large files tested, a speed increase of between 80-95% has been recorded. Obviously, the content of the design and the entities being exported will make a difference to the speed.

STEP Model to Footprint Alignment

The STEP option in the **Design Settings** dialog, **General/Default** page has changed from where it showed **STEP** and **No Model Transformations** to this below:

| Technology [] - Design Setting | s - General/Defaults | |
|--|--|---|
| Spacing Rules Styles Layers Nets Rules - DFM/DFT Rules - High Speed Outputs | All Text Adjust To Readable Orientation Barring Character: (when doubled) System Font Proportional Width Digits CAM/Plot | Attributes Substitution Char: % (when doubled) Matching Styles |
| Colours | Transform about design extents | Naming Rules |
| Grids Units Variants Design Settings Coordinate System | STEP Model Alignment Snap by: © Centre O Origin | Make named styles use naming rules Pad Exceptions Mirror with Component |
| ···· <mark>·General/Defaults</mark> ···· Naming ···· Placement Sites | Use Adjustment Attributes | Defaults |

Checking the 9.1 No Model Transformations option had two effects:

- Ignored the <STEP Offsets> and <STEP Rotations> attributes attached to Parts or Footprints when aligning a STEP model to a footprint.
- Stopped the automatic alignment of a STEP model to a Footprint by using bounding box centres (or Model Placement Areas)

In Pulsonix 10, the Snap by: values will have the following effect:

- **Centres** - will align the Footprint and the model using bounding box centres in the same way as in Pulsonix 9.1 without the **No Model Transformations** check box selected.

 Origins - will align the origin of the Footprint with the (0,0) in the model without reference to bounding boxes or areas in the same way as in Pulsonix 9.1 with the No Model Transformations check box selected.

The **Use Adjustment Attributes** option will effectively do the same as above in 9.1 and ignore the <STEP Offsets> and <STEP Rotations> attributes, regards of the **Snap by:** value.

3D Package Built-in Attribute

The <3D Package> attribute is used to override the normal method of finding a 3D package based on the footprint name. There has been a subtle change made for V10 is to detect an empty value for this attribute indicating that there should be no 3D component displayed on the 3D view. This allows you to suppress any attempt to display a 3D image for particular components such as edge connectors when they aren't required.

Note: when the 3D Viewer is used under normal circumstances, unless specifically required, the <3D Package> attribute will not be used on a Component or Footprint as it is automatically mapped from the footprint name.

Selection Mask

The new **Selection Mask** toolbar allows you to choose design items that are selectable within the design and save the mask for subsequent reuse. This is available in both the Schematic and PCB design editors.

This new option is available from the **View** menu and from the custom menu enabled by right clicking on any of the Pulsonix framework (enabled the same way as the **Design Browser**, **Part Browser**, **World View** etc.). It can also be enabled by right clicking on any of the existing open Browser windows and selecting it from the context menu.

| Selection Masks | 👻 🕂 🗙 | | |
|--|----------|---|----|
| | Save New | | |
| Enable Select Mask | | | |
| <default></default> | ~ | | |
| Area Attribute Htmap Board Bond Pad Breakout Component Connection Construction Line Copper Dimension Doc Shape | | Wire Additional Mask Parameters: Line Style: Attribute: | ** |
| Doc Symbol | | <net name=""></net> | ~ |
| | | Is: matching | ~ |
| Usunction Mounting Hole | | GND | ~ |
| Pad Teardrop | | Frame Select: | |
| Template Testpoint Text Track | | If Inside Frame | ~ |
| Via Wire | | Uo Not Pick Locked Items | |

Using the browser dialog

You can use the Selection Mask on a task by task basis or you may prefer to create a selection mask and save it for subsequent use.

You can create a list of selection masks that can be defined and saved. You might create a number of selection masks for all Track and Pads for example.

To activate the Selection Mask and start to use it, select the **Enable Select Masking** check box. Without this selected, nothing happens. Initially, the <Default> selection mask is shown. This is the selection of check boxes in the list as currently set. If you change the selections and press the **Save** button, this then becomes the new <Default> selection. However, you may wish to be more specific about your choices and to reuse them.

You can save multiple named selections by pressing the **New** button. When this button is pressed, the **Create New Selection Mask** dialog is displayed. When the **Create** button is pressed, the selection mask is saved. Using the **Copy Current Selection Mask** will enable you to make the selections, then save them. You could also use this to make slight modifications to an existing selection mask and save it to another named selection.

| Create New Selection Mask | | | | | | |
|---------------------------|-----------------------------|--------|--|--|--|--|
| Name: | Compoent & Pads | Create | | | | |
| | Copy Current Selection Mask | Cancel | | | | |

The drop down list on the Selection Mask dialog displays all previously saved Selection Masks



Check box options on the Selection Mask Dialog

There are additional filters to enable refinement of the selection. With **Pads** selected, selecting the **Additional Mask Parameters** enables the Pad Style drop down list from where you can choose a pad style. This can a specific pad style or can be a wildcard name. The Wildcard wizard helps you create the name.





Updated User Interface

4K Graphics Support

New icons have been created to enable 4K monitors to be supported. The new icons are scalable and will be automatically resized when Pulsonix detects the use of monitors that require the new icons.

As part of the 4K update, the use of graphics within Technology dialogs and other dialogs/wizards has also been resolved.

Simplified Default Toolbars

As part of the improved user experience, the default toolbars have been simplified by removing nonessential icons. If you have migrated from an older version of Pulsonix (9.1, 9.0 or previous), you will need to reset the toolbar icons using the **Customise** option (from the **Tools** menu), **Toolbars** tab and **Reset All** button.

You will also see that menus have been decluttered with the removal of non-essential icons. These icons are still available for use within a customised toolbar if required.

More Readability for Read Only Edit Boxes

Non-editable and read only edit boxes now have a set disabled colour to differentiate from edit boxes that can be changed and those that can't. This makes their editable status more obvious when using high resolution monitors.

In the example dialog below, it can be clearly seem that entries such as **Name**:, **Position**: and **Angle**: can be edited, whereas the **Part Name**, **Description** and **Part Family** can't.

| Component Nets on Pins Comp Attributes Vauit Name: U2 Locked Postion: 487.680 523.240 Allow Automatic Changes: Angle: 0.0 Mimored Layer: <top side=""> Part: 1658694-6 Altom 30AU Description: CONN HEADER VERT 10POS .100 30AU </top> | | Properties: Component - Component - | | × |
|---|-----------|--|-----------|---|
| Image: U2 Locked Position: 487.680 523.240 Allow Automatic Changes: Angle: 0.0 Mirrored Layer: <top side=""> Part: 1658694-6 Phange Description: CONN HEADER VERT 10POS .100 30AU Alternate Part Family: Alternate Alternate</top> | λ. | Component Nets on Pins Comp Attributes Vault | | |
| Position: 487.680 523.240 Allow Automatic Changes: Angle: 0.0 Mirrored Layer: <top side=""> Part: 1658694-6 Change Description: CONN HEADER VERT 10POS .100 30AU Altemate Part Family: Altemate Altemate Eootprint: CON10_2X5_SHM ✓</top> | \Box | ⊠ <u>N</u> ame: U2 □ <u>L</u> ocked | | |
| Layer: <top side=""> Part: 1658694-6 Change Description: CONN HEADER VERT 10POS .100 30AU Atemate Part Family: Atemate Eootprint: CON10_2X5_SHM</top> | | Position: 487.680 523.240 Allow Automatic Changes: Angle: 0.0 Mirrored | - | |
| Part: 1658694-6 Change Description: CONN HEADER VERT 10POS .100 30AU Atemate Part Family: Atemate Eootprint: CON10_2X5_SHM V | , | La <u>v</u> er: <top side=""></top> | | |
| Description: CONN HEADER VERT 10POS100 30AU Part Family: | | Part: 1658694-6 | Change | |
| Part Family: | | Description: CONN HEADER VERT 10POS .100 30AU | | |
| CON10_2X5_SHM | N | Part Family: | Alternate | ə |
| | \square | Eootprint: CON10_2X5_SHM ~ |] | |

Design Attributes in Variants

Design level Variant Attributes can now have different values in each variant (as well as a fitted state). These are accessed through the **Edit** menu and **Design Properties**.

| Properties: PCB Design - Variant Attributes — | | | | | | | | |
|---|------------|----|------------------------------|-----------------|------------|----|--|--|
| Attributes Vault Variant Attributes / | | | | Associated Part | ts Journal | | | |
| | | | <master design="">*</master> | USA | GB | EU | | |
| | Global Tit | le | Master | USA | GB | EU | | |

The Design Attribute must first exist, this is defined on the Attributes page of Design Properties.

| Properties: PCB Design - Attributes | | | | | | | | Х |
|-------------------------------------|-----------|--------------------|------------------|---------|--|--------------------|----------------|---|
| Attributes | Vault | Variant Attributes | Associated Parts | Journal | | | | |
| Glob | al Title= | | | | | <u>N</u> ev Edi | w it | |

Each Variant instance for the attribute, in our example above, **Global Title**, can be edited to display a different value.

| tributes | Vault | Variant Att | ributes , | Associated | Parts Jo | umal | | | | |
|--|---|--|-----------|------------|-----------|------|-------|------------|-------|------|
| | | | | | | | | | | |
| | | <master de<="" th=""><th>sign>*</th><th>USA</th><th></th><th>GB</th><th></th><th>EU</th><th></th><th></th></master> | sign>* | USA | | GB | | EU | | |
| Global 1 | Fitle M | aster | | USA | GB | | EU | | _ | |
| | | | | | | | | | | |
| | | | | | | | | | | |
| | | | | | | | | | | |
| | | | | | | | | | | |
| | | | | | | | | | | |
| | | | | | | | | | | |
| | | | | | | | | | | |
| | | | | | | | | | | |
| | | | | | | | | | | |
| | | | | | | | | | | |
| | | | | | | | | | | |
| | | | | | | | | | | |
| Attribute: | Global | Title | | | | | | | | |
| Attribute: /ariant: | Global | Title er Design> | | | | | | | | |
| Attribute: Variant: Value: | Global <master< td=""><td>Title er Design></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td>Ť</td></master<> | Title er Design> | | | | | | | | Ť |
| Attribute: /ariant: <u>/</u> alue: | Global <master Master</master | Title er Design> | | | | | | | | ▼ |
| ^A ttribute: /ariant: <u>/</u> alue: | Global <master< td=""><td>Title er Design> r</td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td>••••</td></master<> | Title er Design> r | | | | | | | | •••• |
| Attribute: /ariant: <u>/</u> alue: | Global <master Master</master | Title er Design> | | | | | | | | |
| Attribute: Variant: <u>/</u> alue: | Global <master< td=""><td>Title er Design> r</td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td>••••</td></master<> | Title er Design> r | | | | | | | | •••• |
| Attribute: Variant: <u>/</u> alue: | Global <master< td=""><td>Title er Design> r</td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td>•</td></master<> | Title er Design> r | | | | | | | | • |
| Attribute: Variant: <u>V</u> alue: | Global <master< td=""><td>Title er Design> r</td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></master<> | Title er Design> r | | | | | | | | |
| Attribute : Variant : <u>V</u> alue : | Global <master< td=""><td>Title er Design> r</td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td>₹</td></master<> | Title er Design> r | | | | | | | | ₹ |
| Attribute : Variant : <u>V</u> alue : | Global <master Master</master | Title er Design> r | | Reset | To Master | | Subst | itute Attr | ibute | |
| Attribute: /ariant: <u>/</u> alue: | Global Master Master | <u>Title</u> er Design> r | | Reset " | To Master | | Subst | itute Attr | ibute | |

For each attribute it can also have a different Fitted status

Interactive Copper Pour Healing

The ability to 'heal' copper pour areas dynamically after edit has been added. The healing process will only heal poured regions when damage has occurred due to the edit. This may be caused by a track being moved into it or a pad being dropped in an area of poured copper.



This feature can be switched on or off using the **Interaction** dialog in **Options**. By default, it will be enabled.

| | Act Poured <u>P</u> our Un Add | Differentially Paired Tracks and Bus Routes |
|------------------------|--|---|
| | ☐ Hide Template When Poured ☑ Highlight Isolated Copper | Whilst Editing: Viceop Correctly Paired |
| | Clear On Edit | Route Selected Nets |
| $\left \right\rangle$ | Repour Affected Templates | Display Strategy Dialog |
| / ` | | |

Changes to Manual Routing Modes

Segment Modes

The existing **Segment Mode** menu items used for manual routing and Differential Pair routing have been adjusted to make them more obvious. All modes now displayed on the menu were already available previously but have now been expanded into individual 'modes' with a more verbose description.

The previous **Orthogonal Mitre** and **Curved Mitre** sub-options for the **Orthogonal Mode** have been replaced with new segment modes **Orthogonal (Mitre**) and **Orthogonal (Fillet).**

Also, the **Angled** segment mode has been renamed to **45 Angled**, along with a new mode called **45 Angled** (**Fillet**) that adds a fillet to every 45 degree corner.

Fillet modes are different to Curved in that they provide a smooth transition between angled segments.



These modes are also available for use during **Differential Pair** routing if you have the Interactive High Speed option on your license.

Edit Mitre Mode

| | Options - Edit Track | | |
|---|--|--|---|
| | Design Backups Display Edit Shape Edit Track File Extensions | Segment Mode: Pull Tight Interactive Mitre/Fillet Size: 50.00 Any Angle | Apply Seg Mode to Tail Auto Comer <u>T</u> olerance: 100.00 |
| Ý | Find General | Show 'Can Finish' and 'Has Loop' Markers 🛛 🖓 Rem | ove Loops |

There is a new **Any Angle** check box in **Interactive Mitre/Fillet** section of the **Edit Track** and **Edit Shape** options pages.

Checking this box to allow the **Edit Mitre** option (often assigned to Double-click on a corner) to mitre or fillet between straight lines at any angle between them.



Leave it unchecked to only allow mitres between lines orthogonal and perpendicular to each other. This does not affect the Mitre Corner or Auto Mitre options.

Design Rules Checks Dialog Changes

New DRC Checks - Acid Traps and Adjacent Nets

| Design Rule Check | ¢ | | × |
|-------------------|----------------|----------------------|--------------------|
| Spacing | 🗌 On Grid | Manufacturing | ✓ Nets |
| 🗹 Tracks | 🗹 Tracks | Isolated Copper | Single Pin Nets |
| 🗹 Vias | Vias | Unpoured Templates | Net Connectivity |
| Pads | 🗹 Test Points | Split Plane Pad | Power Planes |
| Mount Holes | Components | Plane Thermal Pad | Unfinished Track |
| 🗹 Test Points | 🗹 Pads | Bond Wire Length | Track Layer |
| Copper | | Wire Cross | Track Width |
| 🗹 Text | Keep In/Out | Wire Under Component | 🗌 Via Size |
| 🗹 Board | 🗹 Tracks | Drill Backoff | 🗌 Via In Pad |
| 🗹 Drills | 🗸 Vias | Minimum Pad Land | Teardrops |
| Components | 🗹 Test Points | Pad Undersize | Track Length |
| 🗹 Split Planes | Component Pads | Component Name | Connection Length |
| | 🗹 Components | Mirrored Text | Connection Vias |
| | Copper | Copper Text On Board | Pin Order |
| | Drills | Panel Items On Board | Differential Pairs |
| | | Copper Shapes | Stub Vias |
| | | Acid Traps | Parallel Track |
| | | Testpoints | Necked Track |
| | | Unreachable Side | Serpentine |
| | | Under Component | Adjacent Nets |
| | | Centre to Centre | |

Two new checks have been introduced for Acid Traps and Adjacent Nets.

Both options are described in more detail under function headings, but in summary:

Acid Traps is a check defined in the **Design Spacing Rule** dialog under the **Tracks** tab. This rule defines both the distance of a track to a pad and checks for acute angles.

A new check in the **Nets** section of **DRC** dialog has been added for **Adjacency Nets**. This reports the closest points of adjacent nets with a warning marker. The adjacency rules are defined in the **Technology** dialog under **Rules – DFM/DFT**.

Clear Warnings Option

As some of the new options such as **Adjacent Nets**, report warnings and not errors, the DRC dialog has a check box to enable you to clear existing warnings (**Clear Warnings**) when DRC is run.

| Acceptance Rule Set | Error Markers | Output | Source |
|---------------------|--------------------------|-------------------------------|--------------------------------------|
| Load | Clear Locked Errors | Generate <u>R</u> eport | • <u>D</u> esign |
| Sa <u>v</u> e | Clear All <u>E</u> rrors | Mot Checked ✓ Show Errors Bar | ○ <u>S</u> election ○ <u>W</u> indow |

Automatically Open DRC Errors Bar Switch

An extra button has been added to the DRC dialog to automatically open the DRC Errors Bar.

| Acceptance Rule Set | Error Markers | Output | Source |
|---------------------|-----------------------------|----------------------------|---------------------|
| Load | <u>C</u> lear Locked Errors | Generate <u>R</u> eport | Design |
| | Clear <u>W</u> arnings | ✓ Report Hules Not Checked | ◯ <u>S</u> election |
| Sa <u>v</u> e | Clear All <u>E</u> rrors | Show Errors Bar | ○ <u>₩</u> indow |

Attribute Editor Changes

Footprint Name in Edit Attribute Dialog

The **Attribute Editor** has been made more consistent across all areas where it is used; in the Schematic and PCB design editors and within the Library Manager on the Parts page. All column fields such as **Part Name**, **Description**, **Family**, and **Footprint** Name have been made available where they were previously missing.

| 🔳 Edit Af | ttributes | | | | | | | | | × |
|--------------|----------------------------|---|--------------|------------------|-------------|-------------------------|------|---|-------------------|-----------------|
| Attributes o | of: Components ce = 106 | | | | | × | | | Ok App Cano | (ily cel |
| Name | Part Name | Description | Family | Footprint | Capacitance | Category | Code | ^ | Add At Renar | trib me |
| AC | 22-27-2021 | KK Header+Lock 2.5mm Pitch | CONN/22-27-2 | MOLEX 22-27-2021 | | Connector/Header | | | Delete | lama |
| BELL | 22-27-2021 | KK Header+Lock 2.5mm Pitch | CONN/22-27-2 | MOLEX 22-27-2021 | | Connector/Header | | | Delete I | vanie |
| C1 | SR305C106KAA | Ceramic Radial Lead Epoxy MLCC X7R AVX RoHS | C/SR30 | SR30 | 106 | | | | | _ |
| C2 | SR301C105KAA | Ceramic Radial Lead Epoxy MLCC X7R AVX RoHS | C/SR30 | SR30 | 105 | | | | Edit | |
| C3 | SR212A220KAR | Ceramic Radial Lead Epoxy MLCC COG AVX RoHS | C/SR21 | SR21 | 220 | | | | Pag | at |
| C4 | SR212A220KAR | Ceramic Radial Lead Epoxy MLCC COG AVX RoHS | C/SR21 | SR21 | 220 | | | | 1165 | ει |
| C5 | 489D106X0010C1 | CAP 489D 16V TANT CASE C | C/489D-C | BEAD 0.6mm | | Capacitor/Radial Lead/S | | | | - |
| | 1 | | | | | | | | V/II ODII | L age |

Hide Inbuilt Columns

You can now quickly hide the inbuilt columns (except the Name column). The **Hide Column** option is available from the context menu on the column header. To show columns again, choose the field required from using the **Column Visib** button on the controls panel.

| | Attributes | | | | | |
|-------------------------|---|---------------------------|--------------|--------|--|---|
| Attribute | es of: Compo | nents | | | | |
| | | | | | | |
| Name | Part Na | - | p | Family | Footprint | <5 |
| Name PL1 | Part Na 3428-6502 | Add Attrib | | Family | Footprint 3MHDR-20 | <s1 2520-6002</s1 |
| Name PL1 U1 | Part Na 3428-6502 TS80C31X | Add Attrib | | Family | Footprint 3MHDR-20 PLCC-44N | <s 2520-6002 picc-44p1</s |
| Name PL1 U1 U2 | Part Na 3428-6502 TS80C31X 1658694-6 | Add Attrib Sort Column | 05 .100 30AU | Family | Footprint 3MHDR-20 PLCC-44N CON10_2X5_SHM | <pre><\$ 2520-6002 plcc-44p1 c-5102321</pre> |

In addition, the Part Description and other standard fields are now on all Part based grids.

Reload All – Do Not Reload Doc Symbols

You can now choose not to **Reload Doc Symbols** when doing **Reload All**. This is useful as Doc Symbols are not relevant to the Synchronise Designs process.

| Reload From Library/Vault | × |
|--|--|
| Only reload if different version in the library/vault | Reload From: |
| Reload control | |
| Keep attribute styles & positions | Keep bond pad position/style overrides |
| Keep local attribute values | Keep pad position/style overrides |
| Replace footprint with default | Do not reload footprint |
| Replace pin networks | |
| Re-apply name stem from part | |
| Match position of the first pad | Keep local plane connect |
| | Reload Doc Symbols |
| | |

New Commands in Options Dialog

Middle Mouse Wheel/Button Usage Options

On the Interaction page of the Options dialog, there is a new check box Rolling Wheel Does Zoom In/Out.

With this option checked (the default state), the existing zoom in/out behaviour will be used. With it unchecked, the wheel does **Pan Vertical** and **Pan Horizontal** if **Shift** is pressed. If **Ctrl** is pressed it will do a **Zoom**.

| General | Zoom Schattry. |
|-----------------------|---------------------------------------|
| In-Place Names | Low High <u>R</u> eset |
| Interaction Macros | Zoom at Cursor also re-centres window |
| Move | Pan: |
| Opline DPC | Pan Sensitivity: Reversed Mouse Pan |
| Perceive Net Namer | Laure Hinte Devel |
| Select | Low High |
| Synchronisation | Middle Mouse: |
| Tooltips | Rolling Wheel Does Zoom In/Out |
| Track Length Limits | |
| Warnings | |
| I | |

Also on this dialog is a new option to suppress the **Pan** facility of middle mouse button when the check box is unchecked. The default state is for the check box to be selected to do a Pan using the middle mouse button.

New Component Pads Selection

Within the **Options** dialog and **Select** page, there is now an extra option enabled when using **Select Whole Components**. Check the new **Select Component Pads** option to allow selecting of visible Component pads. Uncheck it to have the whole Component selected when picking pads, subsequent **shift-select** on the pad will change the selection to just that pad.

| Options - Select | | |
|--|---|--------------|
| Design Backups Display Edit Shape Edit Track File Extensions Find General In-Place Names Interaction Macros Move | Select Select Tight Groups Minimum Pick Tolerance Enable Nudge Do Not Pick Locked Items Select Whole Components Exceptions: Select Component Attributes Select Component Pads | Frame Select |
| Multi-Screen Online DRC Resolve Net Names <mark>Select</mark> | Drag | Double Click |

Create PCB Group for SCM Block

You can now mark a **Block** instance (using **Block Properties**) to have a group in the PCB.

The equivalent group is added to the PCB and **Synchronise Design** will keep them in-line. The group has the same name as the Block, with an optional prefix (which is defined in the PCB, see below).

| 🔳 Prop | oerties: Block | clostance - Bloc | k | |
|--------|-----------------|------------------|---------------------------|---------|
| Block | Block Ports | Block Instance | Block Instance Attributes | Vault |
| 🗹 Na | ame: B1 | | | |
| 🗆 Blo | ock: Block | 1 | | Change |
| | | Generated Bloc | k | |
| De | escription: | | | |
| | | | | // |
| Car | n be Multiply I | nstanced | Create PCI | B Group |
| | cked Instance | • | | V |
| Loc | cked To Vault | | | |

Block Name Prefix

If required, set the Block Name Prefix for use in the PCB from the **Technology** dialog, **Design Settings – General/Defaults** and **Defaults** button. Choose **Block** and type the name of the **Block Group Name Prefix:**

When viewed as a group, the name will look like this, showing the **Prefix** first, followed by the **Block** Name.

| Group Name | | | | |
|------------|-----------------|--------------|--------|--|
| Name: | Block_Block1 | | \sim | |
| Context: | PCB Design Only | | \sim | |
| | | Add To Group | | |
| | ОК | Cancel | | |

Join Open Shapes

The **Join Open Shapes** option has been added to enable you to select multiple open shapes and then join them into a single shape. This is useful in a number of situations; where shapes have been imported from another system, DXF for example, and because of tolerances of the exported DXF, the shape ends do not meet. The second instance can occur when a design is imported from another system. As with the DXF import, this new feature allows you to join segments which have become dis-connected.

If the shapes to be joined form a complete loop with no crossing lines, the resultant shape will be closed.

This option is available for Doc shapes in Schematic, Symbols and PCB designs. It is also available on Copper in PCB designs.

Join Tolerance Value

To enable a more accurate joining of shapes and to allow for variations in data, there is a new **Join Tolerance** factor on the **Interactions** page of the **Options** dialog. This is used to determine if the shape ends are close enough to be joined.

| | Multi-Screen Online DRC Resolve Net Names Select | Pan Sensitivity: Reversed Mouse Pan Low High Reset | Cross Probe |
|----------------|---|---|--|
| | Synchronisation Tooltips | - Middle Mouse: ☑ Rolling Wheel Does Zoom In/Out | Open Schematic Page |
| | Track Length Limits Warnings | Middle Button Does Pan | Offset from Cursor: 6.350000 |
| $\Box \rangle$ | | Unextended Delete - Deletes Segments | Join Open Shapes Join Tolerance: 0.000500 |
| · · · | | Optimise Only Signal Nets | Auto Footprints |
| | | Templates | Automatically Apply Footprint Rules |

Note: there is already a tolerance built into the importer, in some instances, this is not 'large' enough to cope with the joining of segments. The tolerance is a very small value, you may need to adjust your design Units precision to 5, 6 or lower decimal places to be able to view and set the join tolerance value.

Save Doc Symbols To Appropriate Library

You can now save **Doc Symbols** to the appropriate library when doing **Save To Library** or when dropping an individual design on to the **Library Manager**. This includes **Parts**, **Footprints/SCM Symbols**, **PCB Doc Symbols/SCM Doc Symbols** depending which type of design you are currently editing.

| Save Selected Items To Library | | | | | × | |
|--------------------------------------|---------------|-----------------------------------|---|------------------|--------|--------|
| Parts | Fairchild | | | | ~ | ОК |
| PCB Footprint | user | | | | \sim | Cancel |
| PCB Doc Symbol | [New Library] | | | | \sim | |
| Parts 3 | | PCB Footprint | 3 | PCB Doc Symbol | 2 | |
| 1658694-6 LM317S TS80C31X2-MCB | | CON10_2X5_SHI PLCC-44N SMS8 | М | PLCC-44N SMS8 | | |

You can also save all library items from a design in one go by using **Save To Library** from the context menu with nothing selected.

| Select All | |
|---------------------|---------|
| Auto Select | |
| Reverse Layer Orde | r Alt+P |
| Paste | Ctrl+V |
| Reposition Design | |
| Ordered Template | Pour |
| Clear All Templates | |
| Save Items To Libra | ry |
| Reload From Librar | y |
| Fetch Component. | |
| Find XY | |

Changes to Error Markers

Error Markers that refer to a single point, such as Single Pin Net or Unfinished Track for example, now have an indicator drawn to show that point. Previously, no specific marker was drawn, this is now an X with a circle around it at the error origin along with the error marker letter.



Find will Find Areas with no name

In PCB, you can now use **Find Area** to find unnamed areas using the new **<No Name>** entry. This will list both unnamed Component Areas and 'free' Areas within the design.



Find Bar Changes to Select All Find Items

Within the **Find Bar**, small change have been made to **Select All Find Items** and **Select Visible Find Items**. Previously, they did not deselect anything before use, now they will deselect all before use unless the **Ctrl** key is held down during use.

Dynamic Starting for PCB Bus Routing

PCB Bus Routing will now add and move tracks from the start items to the moving bus start indicator **on-the-fly**, whereas in the previous release it was only when you clicked to start adding the bus segments.



As the images below show, once started, if the Bus routing start items move down, the start tracks dynamically move to reroute.

New Template Option – Clear On Edit

There is a new **Template** option – **Clear On Edit**. This is available on the **Options** dialog and **Interaction** tab. With this enabled, it will remove poured copper when editing the Template.

| | Uptimise Unly Signal Nets | Auto Footprints |
|-----------------|--|---|
| | Templates When Adding Templates | Automatically Ap |
| N | Act Poured <u>rour On Add</u> Hide Template When Poured Hidblight legisted Conner | Differentially Paired Tra Whilst Editing: [[|
| $ \rightarrow $ | Clear On Edit | Route Selected Nets - |

Library Manager Changes

Resizable Library Manager

All pages within the **Library Manager** can now be resized. As with all other resizable dialogs, use the corner grabber to resize the dialog.

| PartTech | ~ |
|---------------|--------------------------|
| Add To Design | Add To Comp Bin 1 Copies |
| | Close Apply Help |

Part Pin Mismatch Report

In the Library Manager Parts dialog a new report has been added to Check For Pin Mismatch.

This looks at the expected pin count of a Part and finds and checks against the pin counts on the Parts' Footprints and Schematic Symbols to look for a mismatch. This takes into account Connectors

typically having a single pin symbol for all its Parts. It will also inform you if it simply can't find the Symbol or Footprint the Part references.

| | VariableAskQuestion | \$(LibraryName) (\$(ReportTit) |
|----|--|--|
| | | Change Report File Name AllLibraries (Part Pin Count N |
| | Make/Modify User Report: Report Maker | Which Parts: |
| _/ | Check For Duplicate Items Check For Pin Mismatch Standard Report Rich Text Format Part Details | |

STEP File Names in Part Details

Within the **Library Manager** and **Parts** page, the Part **Details** tab now has a list of **STEP Files** associated with is as defined in the **Part Attributes**. It defaults to the STEP file attribute on a Part, however if it doesn't find one associated with the Part, it will search through the Parts' Footprints and will display the STEP file if it has one associated with it.

| | Attributes | New Library Report Index | |
|---|-------------------|---|---|
| 1 | Preview Detai | ils | |
| | Description: | Voltage Regulator |] |
| | Part Family: | LM/LM317 |] |
| | Name Stem: | U Pins: 4 | |
| | Footprints: | SMS8 |] |
| | Sy <u>m</u> bols: | 1 x Block4s | 1 |
| | <u>G</u> roup: | LM317S | |
| | <u>V</u> ault: | Part: LM317S (not from Vault) Footprint: SMS8 (not from Vault) Symbol: Block4s (not from Vault) | |
| • | Step Files: | dpak3p-841x1018h444.stp | |

Clear Filter

A new button to **Clear** the **Filter** entries in the **Library Manager** has been added. This works the same as clearing the Filter box and/or No Pins box and pressing apply but is much quicker.

| atched: 1 of 3750 | Refresh |
|-------------------|-----------------|
| LM317S* | No. Pins: Apply |
| | Clear |

Close Dialog On Edit

Custom Indexing for Footprints

Footprint Type

The **PCB Footprint Wizard** now gives you the option on the **Finish** page to attach the 'type of footprint as an attribute to the footprint.

| Your footprint will now be created. | |
|---|-------------|
| Once it has been created do you want to: | |
| Save the footprint to the library: | New Library |
| LCC | ~ |
| With the name: QFN1 | |
| Edit the footprint now | |
| Create a part using this footprint Save the part to the library: | New Library |
| Generic | ~ |
| With the name: QFN1 | |
| Edit the part now | |
| Save Type/Kind as Attribute | Value: |
| Footprint Type | QFN |

The footprint 'Type' or 'kind' is taken from the **PCB Footprint Wizard Type** page using the Type selected for creation:



When the Footprint created is subsequently edited, the **Footprint Type** attribute will be available in the **Design Properties** along with its value. This will be an editable value if you need to update its

Type. Likewise, if you wish to categorise any of your existing Footprint library footprints, this Attribute and Value can be added using the **Attribute Editor**.

| Properties: PCB Footprint - Attributes | | × |
|--|----------------|---|
| Attributes Vault Associated Parts | | |
| Footprint Type=QFN | <u>N</u> ew | |
| | <u>E</u> dit | |
| | <u>D</u> elete | |

Custom Footprint Indexes

You can now Index more than just the pin count. This is now similar to the Parts indexing where you can index any Attribute name available or a custom Attribute when typed. When the **Index** button is selected on the **PCB Footprints** page, this dialog is now displayed:

| <pre><hyperlink> <part family=""> Category -Check DateOrawing No- Portprint Type -Issue DateIssued By-</part></hyperlink></pre> | Cancel Choose the attributes which will be used for dassification and searching. |
|---|--|
|---|--|

Using the Footprint Type

With the Footprint Type now in the footprint definition, you can now use it in the Parts index. When indexing your Parts, chose Footprint Type or type Footprint Type and then re-Index the Parts. Once in the Parts index, it can be used for categorisation within the **Parts Browser** for example.

| Part Brows | er Cate | egories | | |
|------------|---------|--|--------|-----------------|
| Use N | ame | Branch Attribu | ıte | Split Character |
| Cat | egory | Category | \sim | 1 |
| | | <none> <part family=""> Category</part></none> | | |
| | | Footprint Type | | |
| | | Mfr Spice Category | | |

Manual list Refresh in Library Manager

A new **Refresh** button has been added to all pages of the **Library Manager** and including the **Folders** page. This allows you to manually refresh the contents lists. This would be used when manually adjusting the library environment that the Library Manager does automatically know about. For example, if you manually remove a library file that has previously been accessed because it is within the library search path. In this instance, the refresh option will re-search the library path.

| Matched: 1 of 3750 | Refresh | | Vault Revision: |
|--------------------|-----------|------|-----------------|
| LM317S* | No. Pins: | pply | PartTech |
| | C | lear | Add To |

'Follow Item' on a Part option

The ability to follow a Part to a new library has been added. With the **Follow Item** check box selected on the **Move** or **Copy Library Item** dialog, this will action the option, then when copying or moving the Part will open the Part on the new library and select it in the **Contents** list.

| Move Librar | ry Item (removes original data) | \times |
|--|---|----------|
| From: Library: Name: | AtmeLpal [in "C:\Users\\MasterLibraries"] AT91SAM7SE256-BGA_7 | |
| To: Library: Name: | Avago AT91SAM7SE256-BGA_7 | - |
| Keep Wil Move parts t Copy group | ithin Group to the specified library. will copy the whole group of parts to the new library. s No Yes to All Cancel I Follow Ite | m |

Find Symbol

The **Find Symbol** option has been added **Schematic Symbol Library** page. This new option allows searching of symbols that fit certain criteria or match a preselected criteria using a symbol referenced from the list.

This dialog can work in two modes, the first is from a pre-selected **Symbol** name off the symbol contents list on the Schematic Symbol Library page, the second is as a **Custom Search** using defined criteria.

| Find Symbol | × |
|-------------------------------------|----------------------|
| Symbol: CON3_1 | |
| Custom Search Symbol Layout | |
| 0 | |
| | |
| 0 | |
| ✓ Match Pins Count ✓ Match Rotation | |
| Exact Symbol Match | |
| Matching Origin | Current Library Only |
| Matching Pin Order | ✓] Generate Report |
| Match Symbol Shapes | Search OK Cancel |

The **Find Symbol** option can take parameters passed through on specific, **Symbol**: name selected from your library. As the name is pre-selected from the Contents list in your library, it is not editable here. Parameters are displayed on this dialog, extracted from the symbol, such as Pins on the Symbol Layout.

Alternatively, by selecting the **Custom Search** check box, you can define your own parameters.

The Find Symbol tool will generate a value for the Part you have selected in the Library Manager page, these values will represent the pin layout in relation to the Symbol selected. The values represent the number of pins and in which direction they are found in relation to the Symbol.

At this point you can perform two types of searches all with different results based on the search options and criteria. By default the tool will run a simple 'pin match' basic search.

Check boxes on the dialog allow various parameters to be defined, these are used to add extra filtering the search.

Find Footprint

A new **Find** tool has been added to the **PCB Footprint Library Manager** page. This allows searching of existing Footprint libraries and reports matched items.

Footprints that fit defined criteria or match a preselected Footprint name from the list.

| Find Footprint | × |
|--|---|
| Footprint: 124pin Edge Connector | |
| Custom Search Footprint Details Pads: Standard: 124 Copper Shapes: 0 Match Standard Pads Match Copper Pieces Styles: Rect (2.0 × 0.6) Match Pad Styles Match Any Style | |
| Perfect Pads Match (Pad count and positions) | □ Current Library Only 0 ☑ Generate Report Search OK Cancel |

You can make a search for a particular footprint in open libraries by pressing the **Find** button from within the **PCB Footprints** tab of the Library Manager with a footprint selected. Or you can search for a user defined number of pads and copper shapes by entering the find tool with no selection or by clicking the * button next to the name of the footprint.

The Find Footprint tool will generate pad values for the footprint you have selected in the Library Manager page, these values will represent the pad count, copper shape count and styles present in the Footprint selected.

At this point you can perform different types of searches all with different results based on the search options and criteria.

Find Footprint in 3D Packages

Within the **Library Manager** under **3D View** page, a new **Find** button has been added. This enables you to search for Footprints that are referenced by the 3D Packages.

| Find | × |
|--|--------|
| Footprint Name: c0805 3D Package: C04* | Search |
| | ОК |
| | Cancel |

Rename in Library Manager - Replace Symbol References in Parts

In the **Library Manager** under the **Rename** button for **SCM Symbols** and **PCB Footprints**, there is a new check box for the **Replace References** feature. This will find and then enables you to replace all references of the old symbol name with the new symbol name in your open Part libraries.

| Rename Library Item | ? | × |
|---|----------|----|
| Library: C:\Users\Public\Documents\Pulsonix\MasterLibraries\BGA.pfl | OK | |
| 0ld Name: 21-0073E_25X25 | Cance | :I |
| New Name: 21-0073E_25×25 | Replac | e |
| UPPERCASE Jowercase | in Parts | |

Swap Symbols Option

Within the Parts Editor page of the Library Manager there is a new Swap Symbol button.

This enables you to find and replace all references to an old symbol name or multiple symbol names and to replace it with a new name in your Parts library.

It also gives you the option to add an alternate footprint to Parts that contain an existing one (e.g. parts containing footprint x can have y added to their footprint list as an alternative).

| Swap Symb | ols In Parts: | | × |
|------------------------|---------------|---|---------------------|
| Old Name: New Name: | C0805 0805 | Search | OK Cancel |
| Libraries | | Footprints Symbols Add As Alten This Library (| native Dnly |
| | | Delete Old St Done Generate Re Load Report | ymbols When port |

Optionally choose to add the swapped Footprint or Symbol as an **Alternate** symbol in the Parts Library. Selecting this option will force it to check the footprints you are adding using a pin count match before adding them as alternatives in the found Parts.

Select the **This Library Only** option to only make swaps in the current selected Parts Library. If **All Libraries** has been selected in the Parts Library dialog, this check box will be greyed out as not available. With a single library selected in the Parts dialog, the search will search and swap (where applicable) in all libraries unless this option is selected.

Select the **Delete Old Symbols When Done** option to delete the old symbols once they have been found and replaced in the Parts library.

Select the **Generate Report** option to report any changes and to report the success of name replacements when the option is run.

Search

On selection of the **Search** button, when the search is complete, you will be presented with a list of Parts that use the Footprint or Symbol and the relevant Parts libraries. From the list, you can choose which items to replace before continuing. The search will be made on an entered Footprint or Symbol name defined as **Old Name** or by using the results of the **Load Report** option.

The **OK** button will start the process and replace all references to the previous Footprint or Symbol with the new names. If the new name list has more items than the old name list you have entered, the Swap Symbol option will replace all Parts with the single new Part name. This enables you to swap out the symbols to multiple old Parts with a single new Part.

Load Report

You can enter a comma separated list of Parts to use as your search and a comma separated list of parts to replace them with.

You can also use the **Load Report** button to open a 'Similar Symbols to (X)' report produced through the **Find Symbol** option on the PCB Footprints or Schematic Symbols tabs of the Library Manager. This will use the perfect match option to fill the old names list for you. When you have entered the Parts to search for, click the Search button. A progress will show the progress of the search. The search is **NOT** case sensitive, however it requires the Part name to be exact, including spaces, otherwise the name will not be found.

Be careful when using this tool as no check is made to see if the replacement Footprint or Symbol is actually available, unless using the option with footprints to add alternative footprints.

Create New Libraries directly from the Library Manager Folder tab

There is an additional button on the **Library Manager Folders** dialog. The **New Library** button allows you to create all five library types at once with the same name (SCM Sym, SCM Doc Sym, Parts, PCB Footprints and PCB Doc Sym). These empty files can then be populated as required using the various methods.

| Files <u>F</u> ound: 174 | | |
|---|-----------------------|---|
| C:\Users\Public\Documents\Pulsonix9.1\MasterLibraries\3m.pal | Pulsonix Part Library | |
| C:\Users\Public\Documents\Pulsonix9.1\MasterLibraries\Actel.pal | 2m | |
| C:\Users\Public\Documents\Pulsonix9.1\MasterLibraries\AD.pal | | |
| C:\Users\Public\Documents\Pulsonix9.1\MasterLibraries\ADSP.pal | Enabled Enable All | |
| C:\Users\Public\Documents\Pulsonix9.1\MasterLibraries\Altera.pal | | |
| C:\Users\Public\Documents\Pulsonix9.1\MasterLibraries\Altera2.pal | Delete Refresh | |
| C:\Users\Public\Documents\Pulsonix9.1\MasterLibraries\AlteraCyclone.pal | | |
| C:\Users\Public\Documents\Pulsonix9.1\MasterLibraries\AMP.pal | Move New Library | |
| C:\Users\Public\Documents\Pulsonix9.1\MasterLibraries\Assmann.pal | Conv | _ |
| C:\Users\Public\Documents\Pulsonix9.1\MasterLibraries\Atmel.pal | | |

All five library types are saved at the same time using the **Save As** dialog:

| Profile | s 🚯 ADSP.pal | 07/02/2017 12:07 | Pulsor |
|-----------------------|---|------------------|-----------|
| Scripts | ADSP.palx | 07/02/2017 12:07 | PALX F |
| STEP N | ADSP.ssl | 30/03/2017 04:44 | SSL Fil ♥ |
| | ~ < | | > |
| File <u>n</u> ame: | | | ~ |
| Save as <u>t</u> ype: | All Libraries (*.all;*.pal;*.pfl;*.pdl;*.ssl;*.sdl;*) | | ~ |
| ∧ Hide Folders | | <u>S</u> ave Car | ncel |

Part Description Using Substitute Attributes

You can now create a 'dynamic' Description field for a Part using Substitute Attributes. This can be done in the **Library Manager** when editing a **Part** (using New or Edit).

The **Description** field can be created from Attributes that appear in the Part definition. The format of the name will be:

Syntax: Res_Value (%%Value%%) with the resultant value displayed as: Res_Value (10k)

The Parts Attributes field would look like this:

| | Attributes | |
|-----------|-----------------|------------------------|
| Part Name | Auto Name Parts | 4308R-102 |
| | Category | Resistor/Resistor Pack |
| | Value | 10K |
| | MFR | Тусо |

The Parts Details page will show an edited Description field like this:

| Part Name: | 4308R-102 1K | | |
|--------------|-----------------------|--------|--------|
| Description: | Res_Value (%%Value%%) | | |
| Part Family: | 4308R/4308R-102 | | |
| Name Stem: | R | | FPGA |
| Pin Count: | 8 Change | | |
| Footprints: | SIP8 | \sim | Choose |

When the Part is used in the design, the Component **Description** field is populated with substituted values.

| | Uariant Id: | | | |
|----|-------------|--|--------|------------------|
| | Position: | 2710.00 5740.00 Allow Automatic Changes: | | |
| | Angle: | 0.0 Mirrored | | |
| | Layer: | <top side=""></top> | | |
| N | Part: | 4308R-102 10K | | Change |
| > | Description | Res_Value (10K) | | |
| _/ | Part Family | 4308R/4308R-102 | | <u>A</u> ltemate |
| | | SIP8 | \sim | |

CAM Plots – Drill Output available for Board Outlines & Cutouts

In the **CAM Plots** option, for a **Drill** output within the **CAM Plot Wizard**, you can now add board outlines and area board cutouts to the drill output.

On the **Output** page, there is a new option to select **Boards:** From here, you can choose **None**, **All Boards**, **Cutouts Only** and **Outlines Only**.

| CAM Plot Wizard - New P | lot | | | × |
|-------------------------|----------------------|--|---------------|---|
| Start | Choose what | is output | | |
| Process | Define the ap | pearance of the output. | | |
| Output | Choose the data to I | be output, then it's style and quality. | | |
| Size | | | | |
| Design Position | | | | |
| - Finish | Process: | Layer Span <through hole=""> \sim</through> | Exclude Items | |
| | Style: | Drill | | |
| | Mirror: | | | |
| | Drill Type: | All ~ | | |
| | Hole Shape: | All ~ | | |
| | Boards: | None | | |
| | | All Boards Cutouts Only | | |
| | | None | | |
| | | Outlines Only | | |

All Boards - means all board outlines including Area Board Cutouts,

Outlines Only - does not include **Area Board Cutouts** or **Cutouts Only** (only **Area Board Cutouts**). Area Board Cutouts can be plated or non-plated, and these are selected using the **Drill Type** above.

Select None to prevent any Board outlines being output.

CAM Plot – Plot String Attribute

You can optionally provide a Plot String on a CAM Plot output. This is used to satisfy any Attribute Positions of type <Plot String> placed on your design. This could be used to define a reference number or other identifier on your output.

When adding an attribute position to the design, if you select <Plot String>, the value cannot be added here but is instead, added when the **Plot String** is satisfied in the CAM Plot dialog.

To define the Plot String, it can be added using the **CAM Plot Wizard** or directly in the **CAM Plot** dialog.

| CAM Plot Wizard - Editing | Plot 'Amplifier' | × |
|-----------------------------------|--|---|
| Start Process | Choose a name for this plot and choose the type of output | |
| Output Size Design Position | Define the name which will be used to identify this plot in dialogs and reports. Also choose the type of output. Name: Amplifier | |
| Finish | Plot Group: ~ Plot String: Amplifier Output To: PDF | |

| C | AM Plots | Plot | t Settings | Plot Pre | view | | | | | | | | | |
|---|----------|------|--------------|----------|-----------|----------|-------------|--------|------------|------------------------------|-------------|------------|-------------|---|
| | | | | | | | | | | | | | | |
| | Name | - 1 | Enabled | Device | Process | Scale | Rotate | Mirror | Position | Area | Step Repeat | Exclusions | Plot String | Ι |
| | Amplifie | r | \checkmark | PDF | Amplifier | Auto Fit | Auto Rotate | | Auto Shift | <design extents=""></design> | | Y | Amplifier | Ī |
| | PSU | | | PDF | PSU | Auto Fit | Auto Rotate | | Auto Shift | <design extents=""></design> | | Y | PSU | l |

Whatever value you type in the Plot String box will be added to the plot once output. This is plotted at the <Plot String> attribute position.

Export IPC-2581 - Omit BOM Data

When exporting the IPC-2581 netlist, there is now a new check box, **Omit BOM Data**. This is only displayed when choosing a **Fabrication** output which normally would contain BOM data but enables it to be removed for security reasons and protection of design IP.

| Export IPC-2581 | × |
|---|----------------|
| c:\users\reference\designs\High speed design.cvg | <u>B</u> rowse |
| Units Functional Mode FABRICATION VILLevel 1 V | |
| Layer Classes Silkscreen Neutral Net Names Silkscreen Omit BOM Data | |

This feature has also been back fitted to Pulsonix 9.1

Toggle True Width For Layers

Alan 01/06/18 - Yes

Added a command to toggle true width for all layers. This is available on the context menu of the Layers Bar.

Report Maker Changes

Access Other Tables in PDC Database

Report Maker - Database Value

Within the **Report Maker** option and the **List of Components** command, the **Database Value** command now has an extra ability. As well as reporting the database value from the current Component you can now use it to report a value from a related Component using the required Component's key value identifier.

This is done by specifying a variable name that at run time will contain the key value identifier for the alternative component. For example this identifier could be taken from an attribute on the Component being reported.

Use Alternative Database Identifier From Variable

Check the **Use Alternative Database Identifier From Variable** check box and then provide the **Variable** name from the drop down list.

| Edit Databa | ise Value Command X |
|-------------|--|
| Command: | Database Value '''' Value |
| 🗹 Use A | lternative Database Identifier From Variable: Var_Alterantives |
| Field: | ✓ □ 0r |
| Use: | ● Value Value ✓ |

This command has also been back fitted to Pulsonix 9.1

Adjacent Nets in Report Maker - List of Errors

Report maker can detect the **Adjacent Nets** errors in a **List of Errors** command by using the **Is Warning** command. You can report the net names using the new **Item Name** and Second **Item Name** commands.

This might be used like the example below:

| . ⊫ List of | Errors |
|-------------|----------------------|
| ⊫ | "Is Warning" |
| | Error Description |
| | Code |
| | Item Name |
| | Item Position |
| | Second Item Name |
| | Second Item Position |
| | End Of Line |
| | |

This would report a message like this:

```
Adjacent Nets (AN) Between (57.800 22.496) and (43.800 31.196). Layer

'Top'. Actual 16.483

Code: AN

Net1: PG0 (57.800,22.496)

Net2: PE5 (43.800,31.196)
```

Another example to report adjacent net rule errors might be like this:

| Eist of Errors | | |
|--------------------|--------|--------------------------|
| i⊟ If "Is Warning" | | |
| <u> </u> | n lf ' | 'Code'' is equal to "AN" |
| | | Error Description |
| | | Code |
| | | Item Name |
| | | Item Position |
| | | Second Item Name |
| | | Second Item Position |
| | | End Of Line |
| | | |

High Speed Option Changes

Differential Pairs - Turn Backwards direction on Via Pattern

When swapping layers with Differential Pairs, the **Via Pattern** selectable can now be **Turn Backwards**. This means, once the layer swap has been made, the pair will turn back on itself.



Improved Accuracy of Differential Pairs

Improved tightness of Differential Pairs on a Layer Swap

During Differential Pair editing, when applying a **Layer Swap**, a predefined pattern is added. In version 10, the accuracy and 'tightness' of the shape of the tracks around the via pattern when using a **Parallel** style has been significantly improved.



Acid trap removal using Minimum Allowed Gap Spacing Rule

During Differential Pair routing, when performing a **Layer Swap** using the **Parallel** pattern, the Acid Trap rule defined will be used. Using a combination of the **Minimum Allowed Angle** and the **Minimum Allowed Gap**, you can control the removal of acid traps.

These values are defined in the **Technology** dialog under **Spacings**, **Design Level** and the **Track** tab. Under the **Acid Traps** rule, you can change the values of the **Minimum Allowed Angle** and **Minimum Allowed Gap** to that required.



To utilize this fully, an angle entered for the **Minimum Allowed Angle** will control the acid trap removal to ensure that the track entry to the via will conform to this rule. If you require additional control, you can specify the **Minimum Allowed Gap** also. This will mean that if the angle is violated, the track will then be backed off by the gap defined. If the gap is set to zero, the minimum required to remove the violation will be applied.

Improved Differential Pair Routing - Fillet Mode

While routing Differential Pairs, you can now switch **Segment Mode** to use one of the two new additional routing modes: **Orthogonal (Fillet)** and **45** Angled (Fillet).



When using 45 Angled (Fillet), you can produce Differential Pair tracks with smooth filleted corners.


Fillets can also be applied to Differential Pairs on existing tracks that were routed previously by selecting the corner and using Edit Mitre from the context menu. The Mitre used (45 degree or Fillet) will depend on the setting using from the Tools menu, Auto Mitre, Auto Mitre All. Both Curved Mitres and Any Angle should be enabled.

| Auto Mitre All | | × |
|----------------|---------------------|--------|
| | | |
| Curved Mitres | Maximum Mitre Size: | 1.270 |
| Any Angle | Minimum Mitre Size: | 0.320 |
| L | | |
| Mitre | Apply Settings | Cancel |
| | | |

Improved Differential Pair Routing – Mirror Diff Pair routing from Angled Components

Differential Pairs can now be routed out of pads at any angle.



Do this by start mirroring a Differential Pair track from two pads that are at an odd angle (not a multiple of 45 degrees), you will be presented with a warning dialog asking if you want to enter **Restrictive Movement** mode to add segments that are perpendicular to the line through the start pads.

| Question | | × |
|---|---|-------------------------|
| The start items are on a 145.0 degree angle line. Do you want to use Restricted Movement segment mode to add segments perpendicular to this line? In this mode you will be able to add one segment at a time until you switch Restricted Movement off using the context menu. | ^ | Yes No Report |
| | ~ | Warnings <u>O</u> n/Off |

Choose **Yes** to do this. The restrictive movement angle will be automatically set and the track segments will be at the correct angle. If this is an operation you do often on non-45 degree components, then you can switch this message off using the check box on the dialog.

Using this mode

After cornering, the mode stays and you can turn in 45 degree steps relative to the start angle. When close together and you click, it will enter adding paired tracks still in the restrictive mode. Add a

segment or two and then use the context menu to switch **Restrictive Movement** off and now you can add the segments in your normal segment mode.

Choose **No** to the dialog to add the mirrored segments in a direction that is a multiple of 45 degrees closest to where you clicked to start mirroring. (This is how Version 9.1 and previous versions worked).

Note: **Restrictive Movement** mode always allows you to turn a multiple of 45 degrees now. In 9.1 this only worked if you set the angle to zero.

Finish Track On Differential Paired Track (Spur)

You can now finish a single track on a Differential Paired track segment provided the track is not an arc and is on the same net. To allow this happen, in the **Technology** under **Nets – Differential Pair**, the pair must have the **Allow Spurs** rule enabled.

With the single track over the Differential Paired track to connect to, on drop, it will unpair a small section of the segment and join the track to it. If you also have **Optimise After Edit** enabled in **Options**, **Edit Track**, the connection from the single track will be optimised automatically.



New Serpentine Skew Modes on Differential Pairs

Selection of single track on a differential Pair

A change to the way in which **Serpentine** operates on **Differential Pair** routing has been made. When a single track segment of a Differential Pair is selected, when using the **Dynamic Serpentine** mode, regardless of the position of the cursor relative to the 'other' Differential Pair track, the Serpentine will now only be applied to the selected segment. Previously, the position of the cursor when in this mode dictated which track was selected from either pair.

Serpentine Skew Modes on Differential Pairs

Serpentine mode has been improved for Differential Paired tracks. There is now biasing, selectable from the context menu, to enable you to bias the track upwards so that the bottom track is not affected, to centre the bias around both tracks of the pair and to bias the serpentine track to the bottom and to have the bottom paired track move to allow space for routing.



Biased Top

Biased Both

Biased Bottom

Move All Vias in Composite Span

When using the **High Speed** option and **Composite Vias**, there is a new option in **Options** under **Move** to **Always Move All Vias In Composite Spans.** This is only applied when a single via in a span is selected and move performed. This now defaults to a checked status to always be on so that the Composite Via moves as one unit. Uncheck it to get behaviour like version 9.1 where just the single selected via moves.

| Design Backups | Auto Weld | Component Pushing | | |
|-----------------------------|--|--|--|--|
| Edit Shape | Auto Weld On Drop | Push Mode: Never ~ | | |
| - Edit Track | Allow Weld To Split Tracks | | | |
| File Extensions Find | Show Weld Spots | Push Direction: Both ~ | | |
| General In-Place Names | Snap To Construction Lines | Spring Back | | |
| - Interaction | ◯ Within Set Distance: 0.00 | | | |
| Macros Move | Within Grid Steps: 2 | Show Force Vector | | |
| | Optimise: Continuous 🗸 | Show As: Long Arrow Compass | | |
| Resolve Net Names Select | Full Optimisation On Drop | | | |
| - Synchronisation | Show Placement Origin | Auto Rotate Bond Pads | | |
| Tooltips | | Auto Rotate Components (on polar grid) | | |
| Warnings | Rotate/Mirror About Centre | Show Dynamic Connections Only | | |
| | Show Invisible Attached Tracks/Connections | Aways Move All Vias In Composite Span | | |
| | Free Angle Adjoining Segs | L | | |

Network License Server (NLS) Changes

A complete rework of client-server interaction for network licensing has been undertaken for Pulsonix 10.

On the server side:

| NLSD | new application that does the actual licensing runs as service or application |
|------|--|
| NLSM | manager application that controls NLSD either on server or from another computer |
| NLSL | log viewer application used from NLSM as an easy way to view log files |

Client side:

No visible change for clients, with all changes being within the client application itself. The separate **PulsonixMonitor** application that used to run as a hidden program to maintain the heartbeat signals to the server is now obsolete. A separate thread inside the main Pulsonix application now manages heartbeats. Performance is better from the client side, allowing quicker application startup time.

The separate **Network Licensing User Guide** has full details about installation and management of NLS.

Default Supplied Technology Files Changes

All Schematic and PCB Technology files have been set to the **Basic Set** for **Style Naming Rules** for **Hatch**, **Line**, **Pin**, **Pad**, **Text**, **Connection** and **Track**.

All PCB Technology files have the 'Footprint Type' attribute added as Any Item, PCB Design Only.

Unattended Installation

The Pulsonix 10.0 installer has been modified to allow IT Managers of large networked installations, the ability to mirror an installation with ease across other clients. This uses the standard Microsoft network installation where an install set is pushed onto clients.

An additional user's guide named Pulsonix Unattended Installation.pdf is provided on the Pulsonix web site under documentation.