



# **Pulsonix Design System V9.0 Update Notes**

### Copyright Notice

Copyright © WestDev Ltd. 2000-2016

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: 16/12/16 iss 1

### **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](mailto:info@pulsonix.com)

Web [www.pulsonix.com](http://www.pulsonix.com)

# Contents

<b>CONTENTS .....</b>	<b>3</b>
<b>VERSION 9.0 UPDATE SUPPLEMENT .....</b>	<b>6</b>
Installing the New Version of Pulsonix.....	6
Licensing .....	6
<b>FEATURES CHANGED OR ADDED IN V8.5.....</b>	<b>7</b>
Import of Eagle Design & Library Files.....	7
New DRC Check – Copper Shapes.....	7
<b>NEW IN VERSION 9.0 .....</b>	<b>9</b>
Significant Changes to the Technology Dialog.....	9
Overview.....	9
Summary of changes.....	10
Rules and Attributes on ‘Net’ Technology dialogs .....	11
Spacings page separated into four dialog pages.....	12
Removal of the Design Rules Dialog .....	13
Removal of the DFM/DFT Dialog.....	14
Changes to Net Classes .....	15
New style of Net Styles dialog .....	16
Via Rules - Via in Pad Rule Layer/Area based.....	17
Using Pad Style Name for defining Thermal & Teardrop Rules.....	18
Import and Export of Rules .....	18
Attribute Editor – Export / Import CSV Tags.....	22
Net Type Field added Change Net Name dialog.....	22
Own Colour Column added to Net Names dialog.....	23
Enhancements to Spacing Rules Match Pair Level .....	23
New Categories of Net Items.....	24
Rules.....	25
What is a rule?.....	25
Creating and applying Rules.....	26
Wildcard Enhancements.....	28
New Wildcard Definitions.....	28
Wildcard Wizard .....	29
Signal Paths.....	30
What is a Signal Path? .....	30
Creating Signal Paths.....	31
Interactively Creating Signal Paths.....	34
Sub Nets.....	34
Overview.....	34
Creating Sub Nets.....	35
Connect In Pin Order .....	37
Interactively Creating Sub Nets.....	38
Interactive High Speed Option .....	39
Changes to Differential Pair Rules.....	39
Differential Paired Chains .....	41
Differential Pair Auto Mirror option.....	43
Drop Via option.....	44
Finish On Via option.....	45

## 4 Contents

---

Branch Points .....	45
New Serpentine Routing Rules.....	48
Dynamic Interactive Serpentine Routing.....	50
Finding Serpentes .....	51
New Serpentine Routing Functionality .....	52
Serpentine Properties.....	53
DRC Check for Serpentine Routing.....	54
Layer Change Length Rule.....	54
Extra Length Rule through Pin Package Attribute .....	55
Necked Length Rule .....	56
Track Length Rule.....	56
Track Length Factor Rule.....	57
Track Length Match Rule.....	57
Track Parallel Segments Rule.....	60
Track Length Limit Indicator .....	61
Changes to the Rules Spreadsheet.....	62
Part Browser Symbol Previews .....	64
Additional Part Browser Changes.....	67
Preview Scale.....	67
View Menu Changes.....	68
Changes to Net Class Override in Templates .....	69
Options For Importing From Other Systems dialog.....	70
Show Design Clearance when Moving a Component .....	71
Show Design Clearance on Breakouts option.....	72
Default Change Layer Preferences.....	72
Zoom at Cursor feature .....	72
Design Settings dialog Changes .....	73
Design Settings – Set Synchronise Design Name .....	74
Changes to Print dialog.....	74
Print to PDF .....	74
Print Scale.....	75
CAM Plot Changes.....	75
Change to Tab Order.....	75
Plot Unfitted Variants option in PCB.....	75
Show Group option.....	76
Plotting Drilled Out Pad Holes .....	76
Exclude Drill Holes from Plot .....	77
Gerber x2 Output Update .....	77
Toggle Fitted Status of Component Variants.....	77
Properties – Change Multiple Components Fitted Status .....	78
Current Variant shown on Status Bar.....	78
Fetch Component .....	78
Properties - Areas .....	79
Colour Dialog Changes.....	80
Highlights & Colours pages .....	80
Colours added for Branch Point (Vias) and Star Points.....	80
Connection Colours in Technology dialog .....	81
New Tabs for Differential Pairs, Signal Paths and Sub Nets .....	81
Selectively set the colour of a net .....	81
Set the colour of Drill Holes in PCB.....	82
Remove From Net for Multiple-selected Tracks/Connections or Components .....	83
Layer Classes now in alphanumeric name order.....	83
Pad Styles - 'Expand All' Exceptions .....	83
Changes to the Design Browser.....	84

Displayed Attributes on Insert Component dialog .....	85
Nets on Pins in Component Properties.....	85
Net Name in Pad option .....	86
Write STEP file from Preview window.....	86
Board Face Direction in STEP File Import.....	87
Specifying Footprints in the Schematic.....	87
Status Bar Improvements .....	87
Find Bar Improvements .....	88
Find Bar Changes .....	88
Report Maker Changes .....	89
Report Maker Vault Reports.....	90
Technology File Changes.....	91
Library Changes .....	91
Library Consolidation .....	91
LT Spice Libraries Added.....	92
<b>PULSONIX VAULT .....</b>	<b>93</b>
What is the Vault? .....	93
Do I need to use the Vault? .....	93
System Requirements .....	93
Deployment planning .....	94
Key Concepts of the Vault .....	94
What can be stored in the Vault? .....	94
What is an 'item'?.....	94
Vault Tools .....	95
The Vault Manager dialog.....	95
The Vault Admin Tool.....	96

# Version 9.0 Update Supplement

## Installing the New Version of Pulsonix

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

To install Pulsonix, insert the CD or double-click on the download executable and wait for a short time. The *Autorun* facility will start the installation procedure. Follow the on-screen commands from the install wizard. You can install Pulsonix 9.0 on top of your existing installation or alongside if you prefer; however, you do not need to uninstall the old version first.

## Licensing

Version 9.0 requires a new license. If you are using a version of Pulsonix earlier than Version 8.x, you will require a new license, this would have been supplied to you under the terms of your maintenance contract.

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

## Features Changed or Added in V8.5

In the following feature descriptions, you will find features documented that were included in a late release of 8.5. They have been documented here to highlight to you.

### Import of Eagle Design & Library Files

When using the **Import Eagle** option, you can now import the original Eagle 'binary' .XML file into Pulsonix without the need to go through the ULP file and intermediate format file process. This operation works for both Schematic/PCB designs and all library files.

The process to import Eagle files into Pulsonix will be exactly the same as before except there will be no intermediate file translation to perform beforehand.

---

*Note: For Eagle designs and libraries older than Eagle V6.0, you will still need to use the existing ULP files that are still supplied or save them in a later version in Eagle.*

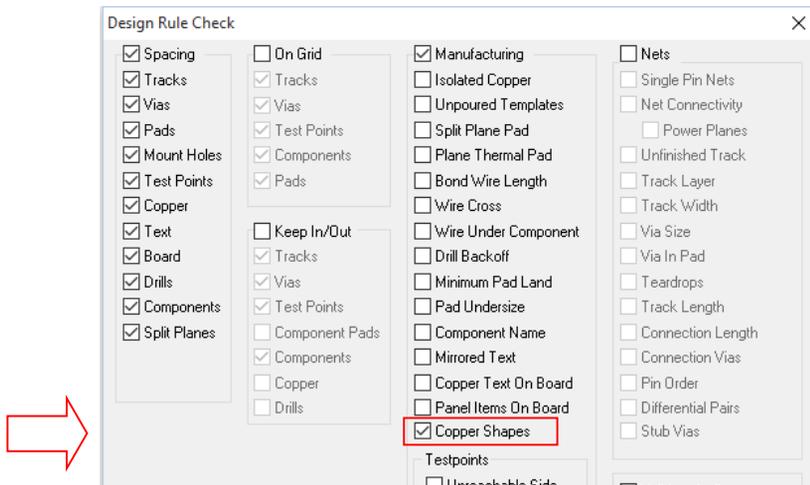
---

This feature was implemented at service patch 5904 of Pulsonix release 8.5

### New DRC Check – Copper Shapes

A new DRC check was introduced into the at service patch 5905 of Pulsonix release 8.5

**Copper Shapes** - Where shapes such as copper containing cutouts will have to be broken into multiple shapes during manufacturing, such as during Gerber plotting with hardware fill enabled, this check can be used to validate the resulting shapes.



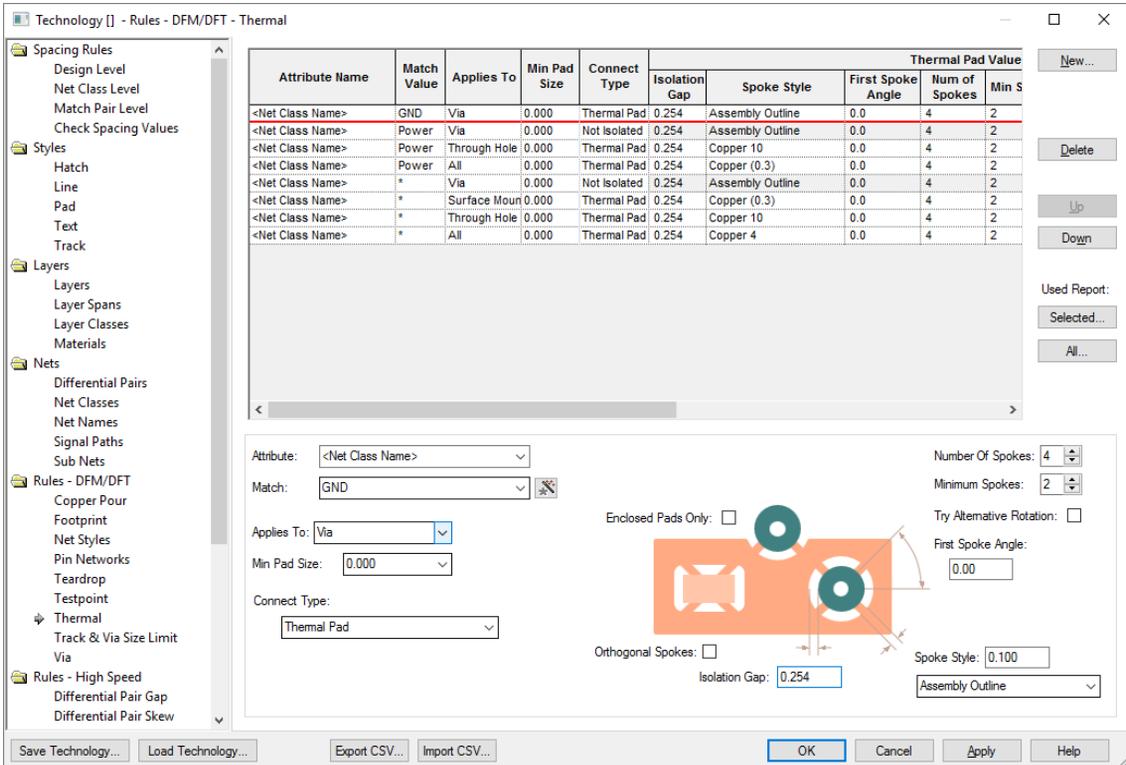


# New in Version 9.0

## Significant Changes to the Technology Dialog

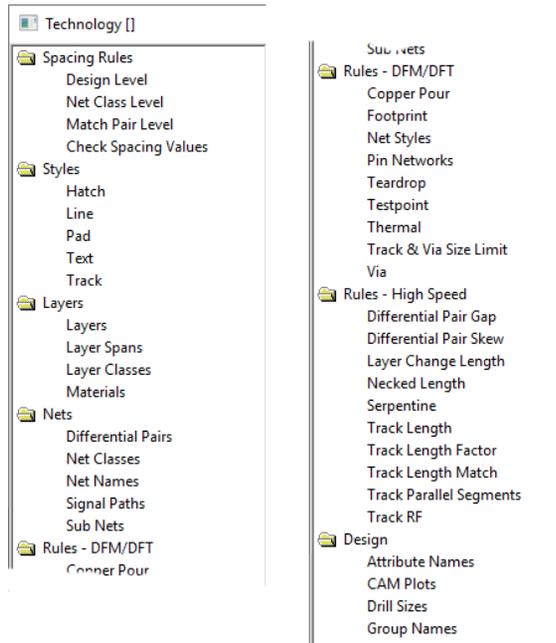
### Overview

The **Technology** dialog in Pulsonix 9.0 has been significantly modified to accommodate the new rules available and to introduce pictorial references for rules and values being entered and edited within the dialog.



## Summary of changes

### New page name order and categorisation



Within the Technology dialog, the structure of the menu items to the left side of the dialog has updated. The order has been changed to re-categorise menu items and accommodate new categories and rules.

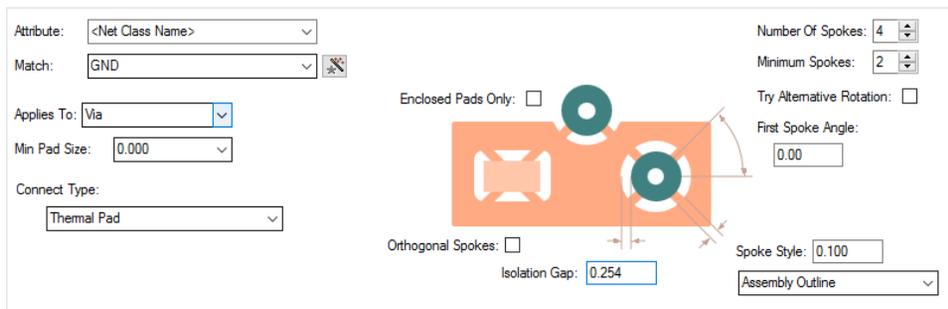
The Rules and Nets tabs where items within previous tabs have been moved out to their own page, for example, Thermal Rules. For designs created in Pulsonix V8.5 or earlier, when read in to V9.0, this information will automatically be migrated and populated in the new dialogs.

### Minor page renames

Some of the pages within the **Technology** dialog have been renamed to bring their names more in-line with their functionality. These are discussed as used further down in this chapter.

### Graphical indication of rules

To aid rules definition and to highlight the rule being edited, tabs now include graphical images of the rules. The picture below shows the **Thermal Rules** image for example:



## Removal of Edit button from dialogs

The **Edit** button has been removed from all **Technology** dialogs. All edit function is now done directly in the dialog itself without the use of a sub-dialog. Each line of the grid can be edited by selecting it.

## Use of the New button

Using the **New** button will create a new line item within the grid on the page selected. Previously, using New opened a new dialog from which to add and edit the item. This is now all done on the new style dialog with a summary in the grid itself. The grid can of course be edited directly once New has been pressed.

## Where Used Report buttons

Two new buttons on **Rules** dialogs will report the **Nets** that the **Selected** or **All** rules defined in the dialog have been used.

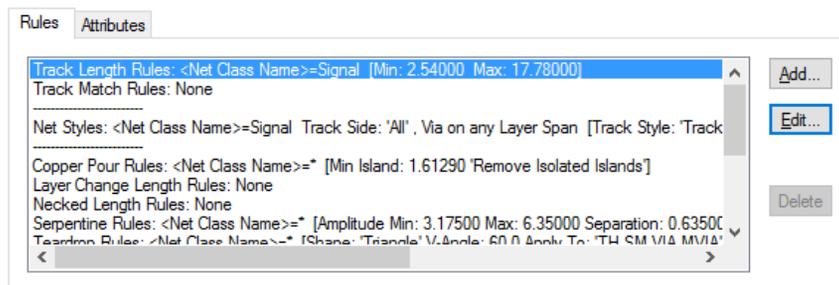
## Rules and Attributes on 'Net' Technology dialogs

This section is an introduction to Rules and Attributes on Nets. These are fully explained later in this chapter.

For each **Net** item page in the Technology that supports rules and attributes there is now a new tab for each rules type (**Rules** or **Attributes**).

### Rules

This page will display all the rules associated with the selected net. Not only does it display the rule, but rules can be added, edited and deleted here too.



**Add** – this button is used to add a new rule for the selected rule item. By default, a new rule is created for <Net Name> and matching the net name selected, this is ready to add rule values to. The default net name rule is can be changed to your own criteria if required. If you press Add where the rule exists, it will edit the rule ready for a further edit.

**Edit** – for a selected Net item, where the rule is defined, it can be edited in its appropriate rules page. This button is greyed out if the rule doesn't exist.

**Delete** – use this to delete a selected rule. However, special conditions apply if the rule applies to more than one net item, the rule cannot be deleted. For example, if the rule is specific to <Net Name>



## Removal of the Design Rules Dialog

Items within the **Design Rules** dialog have been separated out into their own entities in Version 9.0.

The image illustrates the migration of design rules from a single dialog to a structured tree view. On the left, the 'Technology [] - Rules - Design Rules' dialog is shown with several sections: Bond Wire Length, Die Pad Space, Minimum Pad Land, Drill Holes, Components, and Testpoints. On the right, the 'Technology [] - Spacing Rules - Design Level' dialog is shown with a tree view of rule categories and a table of design parameters.

**Technology [] - Rules - Design Rules**

**Bond Wire Length:**  
 Minimum: 0.00  
 Maximum: 0.00

**Die Pad Space:**  
 Minimum: 10.00  
 Hatched Copper  
 Check between hatch lines

**Minimum Pad Land:**  
 Pad Type: Through Hole  
 Radius Difference: 5.00  
 Radius Percentage  
 Absolute Area  
 Min Board To Board: 78.74

**Drill Holes:**  
**Drill to Drill Space:**  
 Minimum: 10.00  
 Allow Coincident Holes  
 Only if Same Size  
**Drill to Board Space:**  
 Minimum: 0.00

**Micro-via Drill Space:**  
 Minimum: 0.00  
 Stagger: 0.00  
 Allow Stacked Micro-Vias  
 Check suppressed pads as though pad is there.

**Components:**  
**Component to Component Space:**  
 Minimum: 0.00  
 Optimum: 0.00  
**Component to Board Space:**  
 Optimum: 0.00

**Testpoints:**  
**Testpoint to Component Space:**  
 Minimum: 0.00  
 Optimum: 0.00  
**Testpoint Land Size:**  
 Minimum: 0.00  
**Min Testpoint Count:**  
 Attribute:   
**Testpoint Grid:**  
 Step: X: 100.00, Y: 100.00  
 Origin: X: 0.00, Y: 0.00  
 Edit...

**Technology [] - Spacing Rules - Design Level**

**Design Level**  
 Net Class Level  
 Match Pair Level  
 Check Spacing Values

**Styles**  
 Hatch  
 Line  
 Pad  
 Text  
 Track

**Layers**  
 Layers  
 Layer Spans  
 Layer Classes  
 Materials

**Nets**  
 Differential Pairs  
 Net Classes  
 Net Names  
 Signal Paths  
 Sub Nets

**Rules - DFM/DFT**  
 Copper Pour  
 Footprint  
 Net Styles  
 Pin Networks  
**Testpoint**  
 Thermal  
 Track & Via Size Limit  
 Via

Design	Design				
	Track	Pad	Via	Testpoint	Mc
Track	0.25400	0.25400	0.25400	0.25400	0.2
Pad	0.25400	0.25400	0.25400	0.25400	0
Via	0.25400	0.25400	0.25400	0.25400	0
Testpoint	0.25400	0.25400	0.25400	0.25400	0
Mounting Hole	0.25400	0.25400	0.25400	0.25400	0.25
Copper	0.25400	0.25400	0.25400	0.25400	0.25
Text	0.25400	0.25400	0.25400	0.25400	0.25
Board	0.25400	0.25400	0.25400	0.25400	0.25

**Additional Design Level Spacings and Rules:**

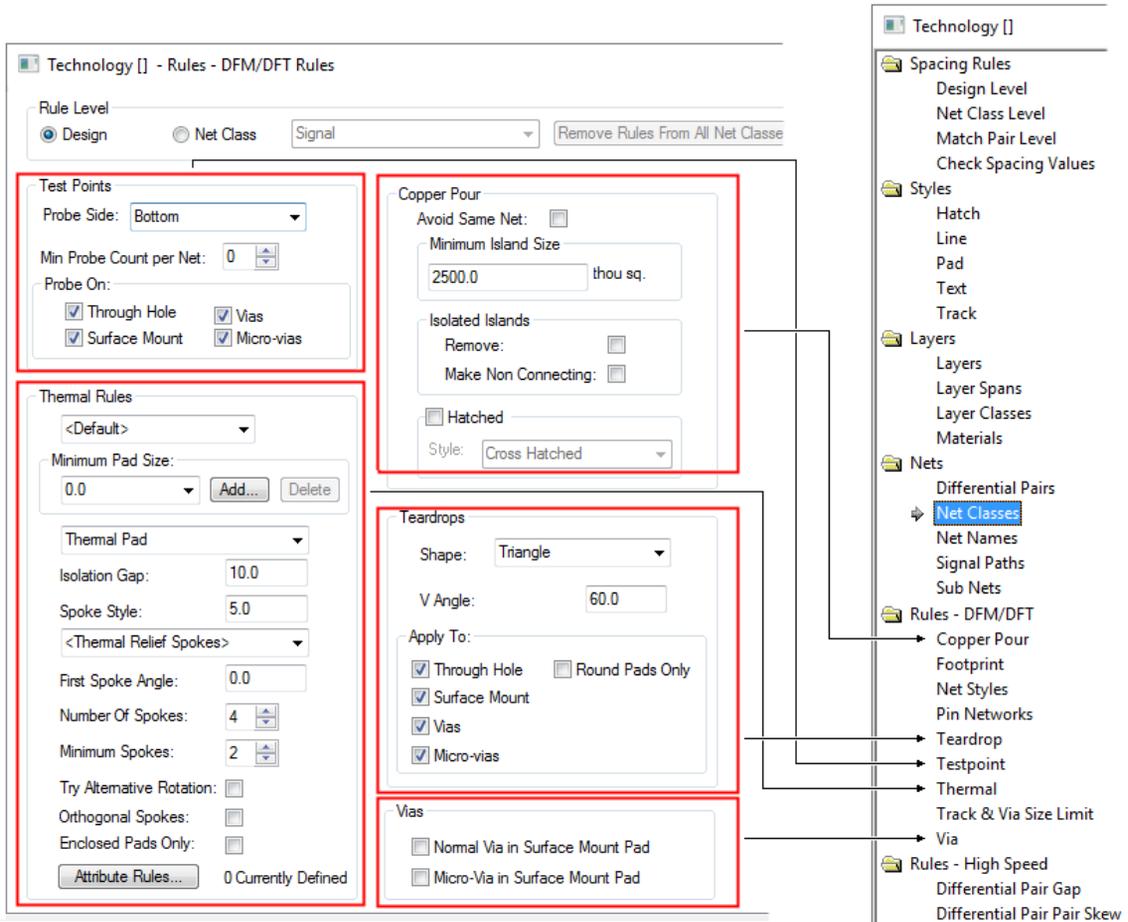
Board | Bond And Die Pad | Component | Copper | Drill | Pad

Use Board Centreline

Min Board To Board: 2.00000

### Removal of the DFM/DFT Dialog

Items within the **DFM/DFT** dialog have been separated out into their own entities in Version 9.0.



The following rules have been taken out of the **DFM Rule** pages, and added as separate rules under the **Rules – DFM/DFT** heading. The DFM rule page has been removed:

**Copper Pour** - Control over pouring style, islands, avoid same net, hatched state.

**Teardrop** - Shape and Angle, for diff pad types.

**Testpoint** - Probe side, min count per item, pad types.

**Thermal** - Isolation gap, spoke parameters, optionally defined for min pad sizes & diff types of pads.

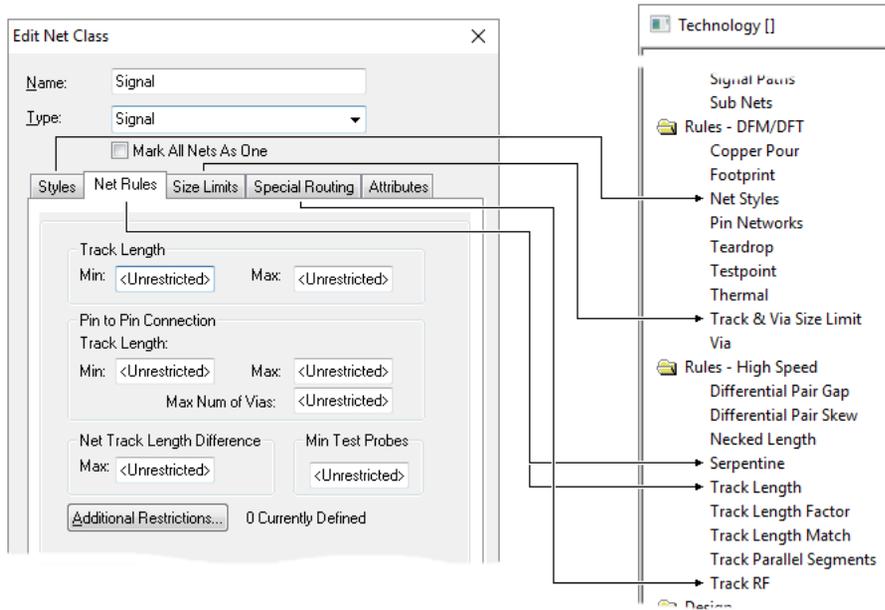
**Track & Via Size Limit** - Min/Max track width, Min/Max via diameter, optionally defined on layers and within areas.

**Via Rules** - allow via and/or micro via in surface mounted pad.

## Changes to Net Classes

The **Net Class** dialog has been significantly changed for V9.0.

The picture below shows how rules have been taken out of the **Net Class** pages and added as separate rules under new **Rules** sections.



## Existing Users

---

*Note: Net Classes, Nets, Rules and Attributes are fully explained further in this chapter.*

---

For existing users with designs that will be read into V9.0, all sections used under the **Net Class** definition will be 'converted' into the new format pages. All data and rules will remain but it will now be in a slightly different location.

You can still work in the same way as before using **Net Classes** and attaching rules to these Net Classes. However, you can now also attach rules onto **Nets** and other net-based items (such as **Signal Paths**). This is an alternative way of working and can provide more flexibility. Working with Net Classes as you have been is still perfectly acceptable though.

You can also use user-defined **Attributes** attached to **Nets** which by use of their **Match Name** will apply the defined **Rule** to the net item.

## New Users

---

*Note: Net Classes, Nets, Rules and Attributes are fully explained further in this chapter.*

---

As a new user or if working with a new design, you can choose to work with **Net Classes** or **Nets**, or a combination of both. Rules can be defined and attached to all net-based items. Using Net-based rules rather than Net Class-based rules will provide you a lower level of rules definition. You can also

## 16 Pulsonix Version 9.0 Update Notes

use user-defined **Attributes** attached to **Nets** which by use of their **Match Name** will apply the defined **Rule** to the net.

The **Net Class** page within **Technology** has been altered in functionality to reflect the overall changes in version 9.0.

	Name	Net Class	Type	Guard Space	Check Same Net	Track Length Rule				Track Length Match Rule		
						Attribute	Match	Minimum	Maximum	Attribute	Match	Max Difference
	8D		Signal	0.00000	<input type="checkbox"/>					<Net Name>	%d%c	3.81000
	9A		Signal	0.00000	<input type="checkbox"/>					<Net Name>	%d%c	3.81000
	9B		Signal	0.00000	<input type="checkbox"/>					<Net Name>	%d%c	3.81000
	9C		Signal	0.00000	<input type="checkbox"/>					<Net Name>	%d%c	3.81000
	9D		Signal	0.00000	<input type="checkbox"/>					<Net Name>	%d%c	3.81000
Y	A	Signal	Signal	0.00000	<input type="checkbox"/>	<Net Class Name>	Signal	2.54000	17.78000			
Y	ADD1	Sig2	Signal	0.00000	<input type="checkbox"/>	<Net Class Name>	Sig2	19.05000	27.94000			
Y	ADD2	Sig2	Signal	0.00000	<input type="checkbox"/>	<Net Class Name>	Sig2	19.05000	27.94000			
Y	ADD3	Sig2	Signal	0.00000	<input type="checkbox"/>	<Net Class Name>	Sig2	19.05000	27.94000			
Y	ADD4	Sig2	Signal	0.00000	<input type="checkbox"/>	<Net Class Name>	Sig2	19.05000	27.94000			
Y	ADD5	Sig2	Signal	0.00000	<input type="checkbox"/>	<Net Class Name>	Sig2	19.05000	27.94000			
Y	ADD6	Sig2	Signal	0.00000	<input type="checkbox"/>	<Net Class Name>	Sig2	19.05000	27.94000			
	ADDR_D	Signal	Signal	0.00000	<input type="checkbox"/>	<Net Class Name>	Signal	2.54000	17.78000			
	ALARM		Signal	0.00000	<input type="checkbox"/>							
	AXIS2		Signal	0.00000	<input type="checkbox"/>							
	AXIS3		Signal	0.00000	<input type="checkbox"/>							
Y	B	Signal	Signal	0.00000	<input type="checkbox"/>	<Net Class Name>	Signal	2.54000	17.78000			
	bob	Signal	Signal	0.00000	<input type="checkbox"/>	<Net Class Name>	Signal	2.54000	17.78000			
	COMPUT		Signal	0.00000	<input type="checkbox"/>							
Y	DIFF1	Diff	Signal	0.00000	<input type="checkbox"/>	<Net Class Name>	Diff	33.02000	44.45000	<Net Class Na	Diff	3.81000
Y	DIFF2	Diff	Signal	0.00000	<input type="checkbox"/>	<Net Class Name>	Diff	33.02000	44.45000	<Net Class Na	Diff	3.81000
Y	DIFF3	Diff	Signal	0.00000	<input type="checkbox"/>	<Net Class Name>	Diff	33.02000	44.45000	<Net Class Na	Diff	3.81000

New...

Copy...

Delete

Delete Unused

Only Show Used Entries
   
 Show Default Names

Name:

Used:

Net Class:

Type:

Check between items on same net:

Guard:

Rules    Attributes

Track Length Rules: None

Track Match Rules: None

Net Styles: None

Copper Pour Rules: <Net Class Name>=" [Min Island: 1.61290 'Remove Isolated Islands]

Layer Change Length Rules: None

Necked Length Rules: None

Serpentine Rules: <Net Class Name>=" [Amplitude Min: 3.17500 Max: 6.35000 Separation: 0.63500]

Tapering Rules: <Net Class Name>=" [Shape: Triangle 'V' Angle: 60.0 Angle To: TH SM Via MVA]

*Note: If you do not have the High Speed option, the Rules and extra rules columns are provided in the grid will not be displayed.*

When using the **Copy** button, it will copy attributes as well as creating a Net Class with a name incremented from the selected Net Class name. All explicit rules will be copied (i.e. rules that directly use the selected Net Class name) plus also copying the **Spacing Rules Net Class** level values.

*Note: Match Pair Spacing values are not copied.*

### New style of Net Styles dialog

As well as the general changes made to all of the **Technology** dialogs and the presentation of information on the page, **Net Styles** is the same as it was previously with the exception of the ability to assign a Net Style to different other types of Net items or attributes in addition to **Net Classes**. For example, you can add a **Net Style rule** to Net based items or to an **Attribute** on items.

Attribute Name	Match Value	Net Type	Area	Track Styles			Via Styles				
				Track Side	Track Layer	Def. Track	Alt. Track	Fat/Neck Min Len	Via Span	Via Style	Via Protected
ChangeStyle	HS				Top	Track 8	Track 8	<Default>		Via 30	
ChangeStyle	HS				Bottom	Track 4	Track 4	<Default>		Via 30	
<Net Name>	Gnd					Track 15	Track 15	<Default>		Via 40	

New...

Delete

Up

Down

Used Report:

Selected...

All...

Attribute:  For Nets of Type:

Match:   Within Areas:

Define Default Track Styles

For Tracks:

On Side:  or On Layer:

Default Track Style:

Name:  Width:

Alternate Track Style:

Name:  Width:

Fatten/Neck Min Length:

Define Via Defaults

For Vias with Layer Span:

Vias Not Allowed

Define Via Protection:  Delete if not Routed  Reduce Span

Define Default Via Style

Name:  Width:  Shape:  Length:  Drill:

Plated

### Via Rules - Via in Pad Rule Layer/Area based

From the **Technology** page and **Via Rules**, you can now assign a Via in Pad rule to **Layers** and **Areas**.

Attribute:

Match:

Micro-Via in Surface Mount Pad

On Layers

Side:

Layer:

Normal Via in Surface Mount Pad



Within Areas:

### Using Pad Style Name for defining Thermal & Teardrop Rules

For new rules pages containing **Thermal** and **Teardrop Rules**, in addition to be able to define Net Items and Attributes, you can also specify that the rule is associated with Pad or Via Syles

Attribute Name	Match Value	Applies To	Min Pad Size	Connect Type		
					Isolation Gap	Spoke
<Net Name>	GND	Surface Mount	0.000	Thermal Pad	12.000	Spoke 12
<Net Name>	A1	All	0.000	Thermal Pad	9.843-	LineStyle
<Pad Style Name>	Via*	Through Hole	0.000	Thermal Pad	9.843-	LineStyle
<Pad Style Name>	R_*	All	0.000	Thermal Pad	9.843-	LineStyle
<Net Name>	*	All	0.000	Thermal Pad	9.843-	LineStyle

You can achieve this by matching with the special attribute <pad style name>.

This means that if you prefer to have a matching pad style to every different pad style in the design, then you can do this. You can of course, also 'group' pad styles together using wildcards to create sensible collections of pads that will be plotted with the same thermal properties.

### Import and Export of Rules

Within the **Technology** dialog, the **Export CSV** and **Import CSV** options are available on dialogs for **DFM/DFT rules** and **High Speed rules**.



These operate for each page and each set of rules. Each set of rules has its own column and row formatting to accommodate rules and functionality. You cannot combine CSV files for different sets of rules in the same design, you can however use the CSV file containing rules on different designs.

#### Exporting to CSV format

When the **Export CSV** button is pressed, the following dialog is displayed, this example shows the dialog for the Track Parallel Segment Rules:

Export Rules CSV Data

CSV Format:

Field separation character:  ,  Use tab      Units for rule values:   Use Design Units

Decimal point character:

Include Table Title:

Map Rule Table Columns:

Rule Column Name	CSV Column Name
Check Segments On Attribute Name	Check Segments On Attribute Name
Check Segments On Match Value	Check Segments On Match Value
Check Segments On Side	Check Segments On Side
Check Segments On Layer	Check Segments On Layer
Check Segments On Area	Check Segments On Area
Against Parallel Segments On Attribute Name	Against Parallel Segments On Attribute Name
Against Parallel Segments On Match Value	Against Parallel Segments On Match Value
Parallel Track Segments Between Adjacent Layers	Parallel Track Segments Between Adjacent Layers
Parallel Track Segments Min Gap Between	Parallel Track Segments Min Gap Between
Parallel Track Segments Max Parallel Length	Parallel Track Segments Max Parallel Length

OK      Cancel

Headers for each type of CSV file are the same but **Table Columns** change for each one.

**CSV Format** – the contents under this header allow you to format the CSV file so that regional variations on the data format can be used.

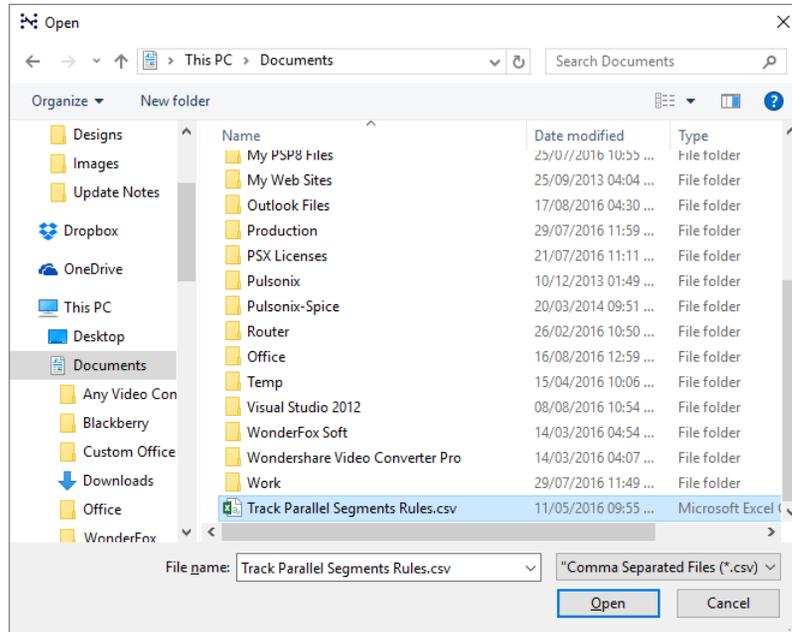
**Include Table Title** – this is used if you wish to include a title as the first line in the CSV file. You might wish to do this to identify the rule being exported or the design name for example. By default it will display the name of the rule but you can type over this to enter your own title.

**Map Rule Table Columns** – this allows you to map the rule name being exported into the CSV file using either default column names provided or your own typed column names. These names appear in the CSV file, it may be that you wish to use shorter or abbreviated names or names in your local language.

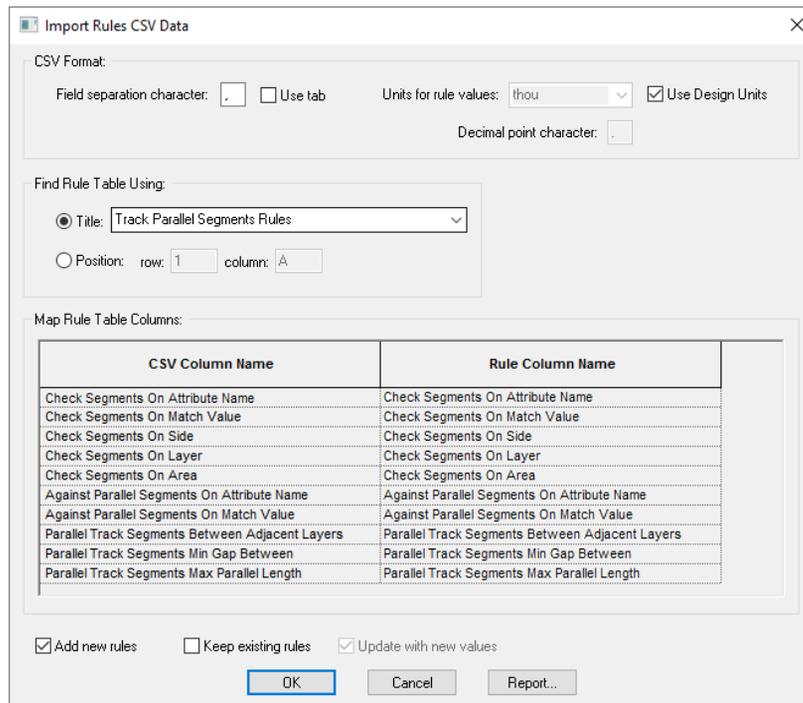
### Importing CSV format files

When the **Import CSV** file button is pressed, you are presented with a standard **Open** dialog from which to choose the file for import:

## 20 Pulsonix Version 9.0 Update Notes



Once the file has been read, you are presented with an **Import CSV** dialog. From here, you can inform the program on how the CSV file has been **formatted**, whether it uses a **Title** header or not and the ability to map the incoming **Columns** names against the ones expected in the **Technology** dialog.



There are three check boxes which when used in conjunction with each other will allow you to customise the way rules in the CSV file are imported and what happens in the event of new and exist rules found.

**Add new rules** – allows you to add new rules from the CSV file. Using a combination of the check boxes below will determine what happens if the rule already exists.

**Keep existing rules** – If a rule is imported and it already exists you can keep it (as is) or by selecting the check box below, will allow you to update the rule value.

**Update with new values** – use this check box if you wish to update an existing rule with new values from the CSV file. This option is only available if the **Keep existing rules** check box is selected.

## Report

The **Report** button on the import dialog will check and report the contents of the CSV file to be imported. The summary presents you with the number of rows found, this acts as verification and any errors found. Errors will include name or syntax errors within the file.

```
Import CSV Rules
-----
Report Written : 17/08/2016 04:33:20 PM
Design Path   : C:\Documents\HS Diff Pairs.pcb
Design Title  :
Created       : 25/10/2015 02:46:34 PM
Last Saved    : 14/01/2016 05:31:55 PM
Editing Time  : 588 min
Import File   : C:\Documents\Track Parallel Segments Rules.csv
Import Summary
-----
Rows Found : 6
Errors Found: 0
```

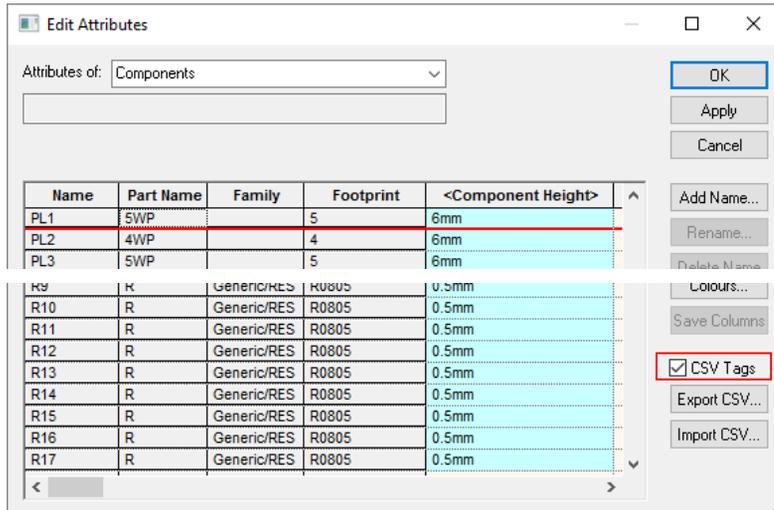
## Format of the CSV File

CSV files can be exported to provide you with a 'base' file formatted in the expected Pulsonix format ready for modification and subsequent import. You may also create CSV files from Excel for example using the appropriate format. Again, exporting an empty rules page in CSV format first will give you a blank data template to work with.

When editing values in the CSV file, the names used must match names as they appear on the dialog itself. For example, a layer side name of Top, Bottom, Inner or Outer is acceptable but Solder or Component would not be.

## Attribute Editor – Export / Import CSV Tags

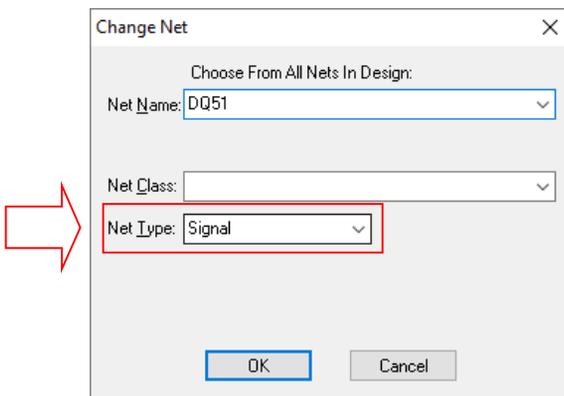
There is a new option on the **Edit Attributes** dialog named **CSV Tags**. This check box will say whether tags are exported and imported when handling CSV files.



CSV Tags are used to show the 'state' of each attribute cell.

## Net Type Field added Change Net Name dialog

From within the **Change Net Names** dialog, you can now add a Net Type by making a selection from the drop down list box.



## Own Colour Column added to Net Names dialog

From within the **Technology** dialog, you can now change the Net Name's **own colour** using the column provided. The settings for **Own Colour** and **Display Connections** will be reflected in the **Colours** dialog and **Nets** page too.

	Name	Net Class	Type	Guard Space	Check Same Net	Use Own Colour	Colour	Display Connection	Tra ^	
									Attribute	Match
	COMPUTER		Signal	0.00000	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>		
Y	DIFF1	Diff	Signal	0.00000	<input type="checkbox"/>	<input type="checkbox"/>	Green	<input type="checkbox"/>	<Net Class Na Diff	
Y	DIFF2	Diff	Signal	0.00000	<input type="checkbox"/>	<input type="checkbox"/>	Green	<input type="checkbox"/>	<Net Class Na Diff	
Y	DRIVE	Sig2	Signal	0.00000	<input type="checkbox"/>	<input checked="" type="checkbox"/>	Blue	<input checked="" type="checkbox"/>	<Net Class Na Sig2	
Y	E1	Power	Power	0.00000	<input type="checkbox"/>	<input type="checkbox"/>	Orange	<input type="checkbox"/>		
Y	E12	Power	Power	0.00000	<input type="checkbox"/>	<input type="checkbox"/>	Orange	<input type="checkbox"/>		
Y	FAT	Signal	Signal	0.00000	<input type="checkbox"/>	<input type="checkbox"/>	Orange	<input type="checkbox"/>	<Net Class Na Signal	
Y	FB01	Signal	Signal	0.00000	<input type="checkbox"/>	<input type="checkbox"/>	Orange	<input type="checkbox"/>	<Net Class Na Signal	
Y	FB02	Signal	Signal	0.00000	<input type="checkbox"/>	<input type="checkbox"/>	Orange	<input type="checkbox"/>	<Net Class Na Signal	
Y	FB03	Signal	Signal	0.00000	<input type="checkbox"/>	<input type="checkbox"/>	Orange	<input type="checkbox"/>	<Net Class Na Signal	
Y	FB04	Signal	Signal	0.00000	<input type="checkbox"/>	<input type="checkbox"/>	Orange	<input type="checkbox"/>	<Net Class Na Signal	
Y	Gnd	Power	Power	0.00000	<input type="checkbox"/>	<input type="checkbox"/>	Blue	<input type="checkbox"/>		
Y	Gnd1	Power	Power	0.00000	<input type="checkbox"/>	<input type="checkbox"/>	Blue	<input type="checkbox"/>		
Y	High_speed	Sig2	Signal	0.00000	<input type="checkbox"/>	<input checked="" type="checkbox"/>	Blue	<input checked="" type="checkbox"/>	<Net Class Na Sig2	
Y	HS	HS	Signal	0.00000	<input type="checkbox"/>	<input type="checkbox"/>	Orange	<input type="checkbox"/>	<Net Class Na HS	
	HS1	HS1	Signal	0.00000	<input type="checkbox"/>	<input type="checkbox"/>	Orange	<input type="checkbox"/>	<Net Class Na HS1	
	HS2	HS	Signal	0.00000	<input type="checkbox"/>	<input type="checkbox"/>	Orange	<input type="checkbox"/>	<Net Class Na HS	
Y	HS3	HS3	Signal	0.00000	<input type="checkbox"/>	<input type="checkbox"/>	Orange	<input type="checkbox"/>	<Net Class Na HS	
Y	HS4	HS	Signal	0.00000	<input type="checkbox"/>	<input type="checkbox"/>	Orange	<input type="checkbox"/>	<Net Class Na HS	
Y	HSE1	HSE	Signal	0.00000	<input type="checkbox"/>	<input type="checkbox"/>	Orange	<input type="checkbox"/>	<Net Class Na HSE	
Y	HSE2	HSE	Signal	0.00000	<input type="checkbox"/>	<input type="checkbox"/>	Orange	<input type="checkbox"/>	<Net Class Na HSE	

Name:   Own Colour:   Force Display Connections

Used:

Net Class:

Type:

Check between items on same net:

Guard:

Rules Attributes

Copper Pour Rules: <Net Class Name>\* [Min Island: 1.61290 Remove Isolat  
 Net Styles: <Net Class Name>=Sig2 Track Side: All' Via on any Layer Span  
 Teardrop Rules: <Net Class Name>\* [Shape: Triangle V-Angle: 60.0 Apply  
 Testpoint Rules: <Net Class Name>=Sig2 [Probe Side: Bottom Min Probes: 1  
 Thermal Rules: <Net Class Name>\* Applies To: All' Min Pad Size: 0.00000 [r  
 Track Size Limit Rules: None  
 Via Rules: <Net Class Name>\* All Layer Side []

Lower Channel Length Rules: None

## Enhancements to Spacing Rules Match Pair Level

As well as **Net Classes**, the ability to match **Net Names** and **Net Attributes** has been added to the **Match Pair** level rules.

Technology [] - Spacing Rules - Match Pair Level

- Spacing Rules
  - Design Level
  - Net Class Level
  - Match Pair Level
  - Check Spacing Values
- Styles
  - Hatch
  - Line

Item 1		Item 2		Side	Layer	Area
Attribute Name	Match Value	Attribute Name	Match Value			
<Net Class Name>	Power	<Net Class Name>	*	All		
<Net Name>	ADD%[0-9%]	<Net Name>	*	All		
NetTypeRule1	HSE	<Net Name>	*	All		
AreaMatchRule	BGA*	<Net Name>	*	All		BGA-6-6
LayerMatchRule	Diff*	<Net Name>	*	All	Inner 2	

For all **Net Attributes** that match the rule value, the **Spacing** rule will be applied. As with all the rules, rule priority is taken from the top of the list working down, any non-specific rule matching a value of \* for example should be at the bottom of the list.

**Copy** – this button is different to the **New** button in that it copies all the grid values.

Controls on this dialog enable you to produce a **Where Used** report for the Match Pair Spacing rules.

### New Categories of Net Items

There are new **Technology** pages in the **Nets** section for defining facets that are properties of a net or multiple nets, Signal Paths, Sub-Nets, Differential Pairs and Chains of Differential Pairs. Each of these new items can have attributes assigned and rules added to them. *The use of Net Attributes is discussed later on.*

#### Why and when to use each category

To summarise each category, here is what they are used for in Pulsonix:

**Nets** – Nets still have a **Net Name** with a **Type** (see special note below about Type) and (optionally) a **Guard Space**. They can now also have a set of attributes containing rules added, for example, a **Track Length Rule**. They can also have an optional **Net Class** associated with them.

**Net Class** – Net Classes have a **Net Class Name**, a **Type** and a flag to **Mark All Nets As One**. It also contains Net Class based **Track Length Rules**.

All information has been separated out to its own rule that can be applied to Nets Names. Nets do not require a Net Class now, all rules can be attached to a single net or multiple nets. If you prefer to work with Net Class Names, these can still be used and the new rules attached to them as well, for existing designs for example where you retain Net Class Names for simplicity.

**Net Styles** – This existing dialog now becomes more significant with the reduction in functionality for the Net Class dialog (moving from Net Class-based to Net-based rules). All default styles (Tracks, Vias etc.) associated with a Net Class or Net Attribute are defined in the **Net Styles** dialog. This dialog is also used to define how tracking will behave on different **Layers**, within **Areas** and for **Layer Spans**.

**Signal Paths** – Signal Paths now represent another level of net definition. The path is a named item containing an ordered list of pads that represent a signal path. You might use this within High-Speed designs for example where constraints are required. This could be where the overall track length of the signal path would require a specific defined length rule(s).

**Sub Net** – These define part of a net which may require special considerations. These are defined in the **Sub Nets** dialog using an **attribute name** and **value**. Pins on the same net with attributes that match it are deemed to be in the same sub net. One definition can define sub nets in multiple nets. You may use this for example, when creating branch lengths or a specific daisy chain order in a High Speed design.

**Differential Pairs** – The Differential Pairs dialog is now only used to define the Differential Pin Pairs and Differential Pair Chains (see below). All other Differential Pair specific rules such as **Pair Gap** and **Pair Length** are now contained in their own dialog under the **Rules – High Speed** section.

**Differential Pair Chain** – A Differential Pair Chain is two or more (existing) Differential Pairs added to a named list to create an extended list. This is used for associating multiple Differential Pairs so that lengths or net rules can be defined for the overall 'path'. This path may be split with a terminating component for example and will contain different net names. As with Differential Pairs, a design can contain multiple Differential Pair Chains.

---

*Type – although not a net as such, a net Type now becomes more significant with the changes to nets in V9.0. A net always has a Type, this is assigned when the net is introduced to the design. It can be one of three states; Power, Ground or Signal. Type is used for some net-based options such as Optimise, Design Rules Checks, ERC and Autoplace*

---

## Rules

### What is a rule?

A rule is a collection of specific conditions and characteristics that can be assigned to a net. Whereas previously these rules were bundled to be set on Net Classes, they are now individual facets that can be assigned to any net or sets of nets. Rules are attribute driven. Some ‘rules’ are system ‘attributes’ such as <Net Name>, <Net Class> etc. but user defined attributes can be added as well.

User defined attributes would be created when system attributes do not provide enough range for rules coverage. For example, a Track Length rule might be applied to multiple net names that do not have a common name format, like CLK, RST, DQ1, ADD3 etc.

### What characteristic might a rule have?

Rules can be standard items such as Copper Pour, Thermal connectivity, Teardrops and Net Styles etc. More advanced rules might include facets such as Track Length, Track Length Match, Serpentine and Track RF features.

### What can a rule be assigned to?

Rules can be assigned to any Net, Signal Path, Sub-Net, Differential Pair and Differential Pair Chain. This means one (or multiple) rules can be applied to multiple net categories. For example, a number of Thermal rules can be defined and applied to all signal or power style nets. Likewise, a Track Length Match rule can be applied to multiple sets of Differential Pairs to ensure they are all within length difference of each other.

Rules may also still be assigned to a Net Class if required but with the new rules structure are less likely to be used this way.

### Key headers used in the rules dialog

There are some essential headers that are used within the rules dialogs and are highlighted below:

Attribute Name	Match Value	Total Track Length		Max Vias	For Nets and Subnets Apply Rule To	
		Minimum	Maximum		Total Track Length	Pin To Pin Track Length
TrackLengthRule	1000-1500	20.00000	28.00000	2	<input type="checkbox"/>	<input checked="" type="checkbox"/>

**Attribute Name** – this is the name of attribute that will define the rule, for example, this could be one of the system ‘attributes’ such as <Net Name> or <Net Class Name>, or it could be your own user-defined attribute name, such as TrackLengthRule

**Match Value** – once an Attribute Name has been defined, you must give it the name of something to match. This could be a unique Net Name, CLK for example, or a range of Net Names to apply the rule to, such as ADD0 to ADD9. Using Ranges to match the attribute means a single rule can be applied to multiple instances.

## Creating and applying Rules

Rules can be created and applied using two methods:

### Method 1

Create your rules first in the Rules sections (in the Technology dialog under DFM/DFT and High Speed) and apply them to the net(s) required.

As an example, we'll show the addition of a **Track Length Rule** but the principle applies to all rules. Create the rule by adding an **Attribute Name** and **Match Value**, plus your rule **values**:

Attribute Name	Match Value	Total Track Length		Max Vias	For Nets and Subnets Apply Rule To	
		Minimum	Maximum		Total Track Length	Pin To Pin Track Length
<Net Name>	ADD*	15.00000	81.00000	2	<input type="checkbox"/>	<input checked="" type="checkbox"/>
<Net Name>	ALARM	33.00000	44.00000	2	<input type="checkbox"/>	<input checked="" type="checkbox"/>

On the **Nets** page, if the new attribute matches the rule, the attribute name and value will be automatically adopted. If the rule doesn't match automatically, apply the rule by typing the rule name in the **Attribute** field (or selecting it from the drop down list which will be populated from the list of available rules).

The example below is specific to the **Track Length** rule as this is directly shown in the **Nets** dialog:

Name	Net Class	Type	Guard Space	Check Same Net	Track Length Rule			
					Attribute	Match	Minimum	Maximum
Y A	Signal	Signal	0.00000	<input type="checkbox"/>				
Y ADD1	Sig2	Signal	0.00000	<input type="checkbox"/>	<Net Name>	ADD*	15.00000	81.00000
Y ADD2	Sig2	Signal	0.00000	<input type="checkbox"/>	<Net Name>	ADD*	15.00000	81.00000
Y ADD3	Sig2	Signal	0.00000	<input type="checkbox"/>	<Net Name>	ADD*	15.00000	81.00000
Y ADD4	Sig2	Signal	0.00000	<input type="checkbox"/>	<Net Name>	ADD*	15.00000	81.00000
Y ADD5	Sig2	Signal	0.00000	<input type="checkbox"/>	<Net Name>	ADD*	15.00000	81.00000
Y ADD6	Sig2	Signal	0.00000	<input type="checkbox"/>	<Net Name>	ADD*	15.00000	81.00000
AR_D	Signal	Signal	0.00000	<input type="checkbox"/>				
ALARM	Signal	Signal	0.00000	<input type="checkbox"/>	<Net Name>	ALARM	33.00000	44.00000
AXIS2		Signal	0.00000	<input type="checkbox"/>				
AXIS3		Signal	0.00000	<input type="checkbox"/>				
Y B	Signal	Signal	0.00000	<input type="checkbox"/>				
hsh	Signal	Signal	0.00000	<input type="checkbox"/>				

All rules added to a net-based item are displayed in the relevant dialog in the **Rules** and **Attributes** tabs:

Name: HS

Used:

Net Class: HS

Type: Signal

Check between items on same net:

Guard: 0.00000

Rules Attributes

Track Length Rules: <Net Class Name>=HS [Min: 5.08000 Max: 81.28000]

Track Match Rules: <Net Class Name>=HS [Max Diff: 12.70000]

Net Styles: <Net Class Name>=HS Track Side: 'All', Via on any Layer Span [Tr

Layer Change Length Rules: None

Copper Pour Rules: <Net Class Name>=" [Min Island: 1.61290 Remove Isolate

Necked Length Rules: None

Serpentine Rules: <Net Class Name>=HS [Amplitude Min: 3.17500 Max: 6.3500

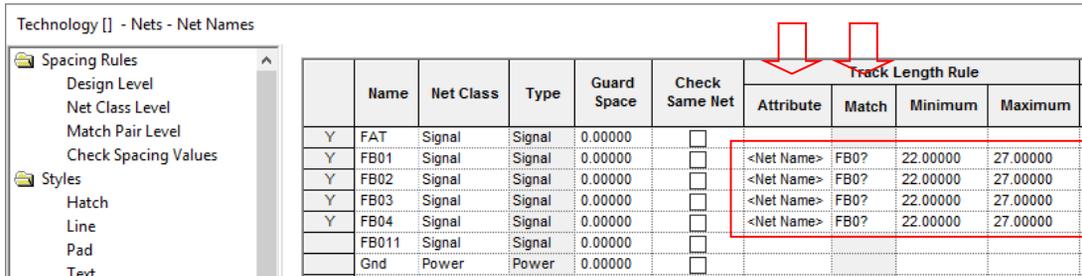
Teardrop Rules: <Net Class Name>=" [Shape: 'Triangle', Angle: 60.0, Ankle To

### Method 2 – Track Length Rule & Track Length Match Rule

When using the **Track Length Rule & Track Length Match Rules**, an alternative is to write the rule directly into the dialog that requires it (Net Name, Net Classes, Signal Paths and Sub Nets). Once

the attribute (Rule) is written, it then becomes a rule within its own right and appears in the relevant Rules page where it is available for use on other nets.

To do this: write the rule into the **Attribute** cell along with the **Match** and **Values** to be used. In the example below, the <Net Name> system attribute has been selected. The typed Match will be FB0? And values of 22.0 and 27.0 for Min and Max Track Lengths respectively. This then matches the Net names FB01-04 but not FB011 (? will only match one character).



### Using Attribute Rules

Once Attribute Rules have been defined, they can be matched on a net item by Rule Value or by Rule Name depending on which scheme best matches your requirements.

**Match the rule value** - any item with the rule attached and matching the value. For example Attribute Name=TrackLength, Match Value=2.0. Where a net then contains an attribute of TrackLength and a Matched Value of 2.0, the rule will be applied.

**Match the rule name** - any item with the rule attached, with no value (%# must be used as the value to show a field with no value), but match by rule (attribute) name.

### Rule Matching Examples

Wildcards are available for rules matching. New wildcards have been introduced at Version 9.0, these are shown in the section below (*see Wildcard Enhancements*).

Attribute Name (Rule)	Match Value	Rule Values	Description
<Net Class>	Signal	xx	A specific Net Class name of Signal has the rule applied (where xx is your rule value)
<Net Name>	DQS*	xx	Any net starting with DQS
<Net Name>	DQS%[1:7%]	xx	Any net between DQS1 to DQS7
<Differential Pair Name>	% {DQS* DSM*% }	xx	Any Diff Pair starting with either DQS or DSM will have the rule applied. The strings are separated by a pipe (   ) character.
TrackLength	2.0	xx	Any net that has the attribute with the name TrackLength matching a value of 2.0 will have the rule applied.
TrackLength	Blank (will be written as %#)	xx	Any net with the attribute name TrackLength with no value will have the rule applied.

## Wildcard Enhancements

The changes made to the use of wildcards within all dialogs will enhance the new functionality where specific selections are required.

### New Wildcard Definitions

%# - will match an empty string.

{string1|string2|string3%} - means match any of these strings as part of a longer wildcard string. The strings are separated by a pipe (|) character. You can add a blank alternative at the start of the list - %{|string2%} which matches *blank* or *string2*.

[string1:string2%] - means match a range defined by the strings, which are either a pair of numbers or a pair of letters which match values between that pair.

### Theoretical Examples

{DQS\*|DSM\*%} would match DQS *anything* and DSM *anything*

[93:101.73%] would match 93 or 100 or 97.8732 or 101.73

[c:f%] would match c or d or e or f

### Practical Examples

In V9.0, a Net doesn't require a Net Class name, the %# wildcard could be used to specifically include all nets with no net class name. For example, when specifying a Track Length Rule.

Attribute Name	Match Value	Min Track Length	Max Track Length	Min Pin to Pin	Max Pin to Pin	Max Vias
<Net Class Name>	%#	15.00000	55.00000	33.02000	44.45000	2
<Net Class Name>	HS	25.40000	76.20000	5.08000	81.28000	2
<Net Class Name>	UC4					

Where wildcards are used to match rules against multiple names, the following can be applied:

In the **Track Length Match** rule, define the rule as <Signal Path> where it matches the value DQ%[24-28%] and DQ%[14-17%], both with a value for the rule.

Attribute Name	Match Value	Max Length Difference
<Signal Path Name>	DQ%[24:28%]	1.0
<Signal Path Name>	DQ%[14:17%]	1.2

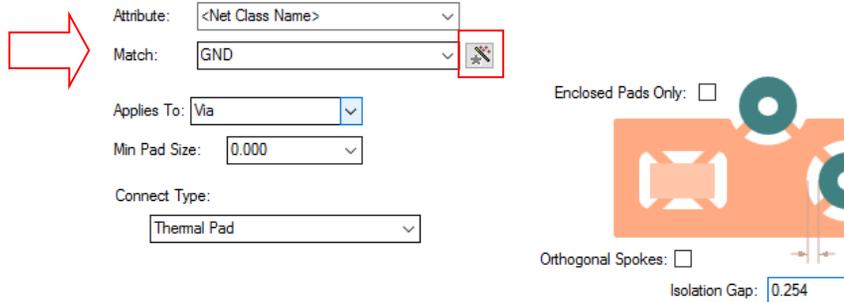
This rule then matches the **Signal Paths** defined for DQ14-17 and DQ24-28:

Name	Pin Count	Track Length Rule			Track Length Match Rule		
		Match	Minimum	Maximum	Attribute	Match	Max Differe
DQ14	4				<Signal Path Name>	DQ%[14:17%]	1.2
DQ15	4				<Signal Path Name>	DQ%[14:17%]	1.2
DQ16	4				<Signal Path Name>	DQ%[14:17%]	1.2
DQ17	4				<Signal Path Name>	DQ%[14:17%]	1.2
DQ24	4				<Signal Path Name>	DQ%[24:28%]	1.0
DQ25	4				<Signal Path Name>	DQ%[24:28%]	1.0
DQ26	4				<Signal Path Name>	DQ%[24:28%]	1.0
DQ27	4				<Signal Path Name>	DQ%[24:28%]	1.0
DQ28	4				<Signal Path Name>	DQ%[24:28%]	1.0

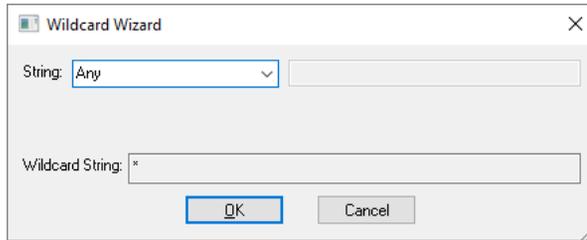
## Wildcard Wizard

As part of the introduction of new wildcard definitions, a new **Wildcard Wizard** has been introduced. This has been added to simplify some, often, complex wildcard strings to be available in more non-programming terms.

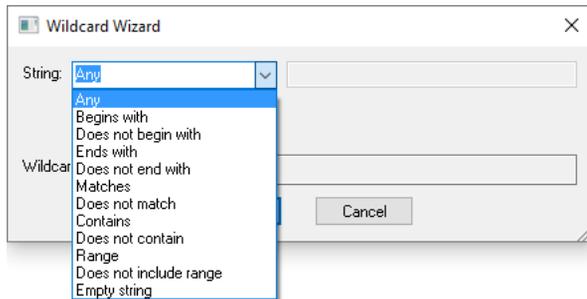
On dialogs where wildcards are accepted, there is now a button to access the wizard.



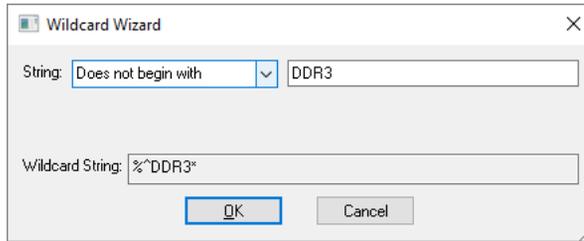
Pressing the button opens up the **Wildcard Wizard** dialog:



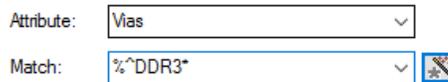
Wildcard String expressions are defined from the drop down list:



On selection of a **String**, you can type in the value to match. The **Wildcard String** is shown in the Wizard dialog.



When **OK** is pressed, the wildcard string is shown as the **Match** value back in the host dialog.



## Signal Paths

### What is a Signal Path?

Signal Paths represent another level of net definition. The path is a named item containing an ordered list of pads that represent a signal path. For example, you can use this for High-Speed designs where constraints on specific signals are required or for defining portions of a net where specific rules should be applied, such as Thermal or Copper Pour Rules.

The difference between a rule on a Net and the same rule on a Signal Path is that the signal path can be just a portion of the net (or multiple nets), whereas, a net rule is applied to a whole net.

The **Signal Path** dialog is used from within the **Technology** and provides an interactive mode and modeless dialog to create signal paths.

### Using the Signal Paths dialog

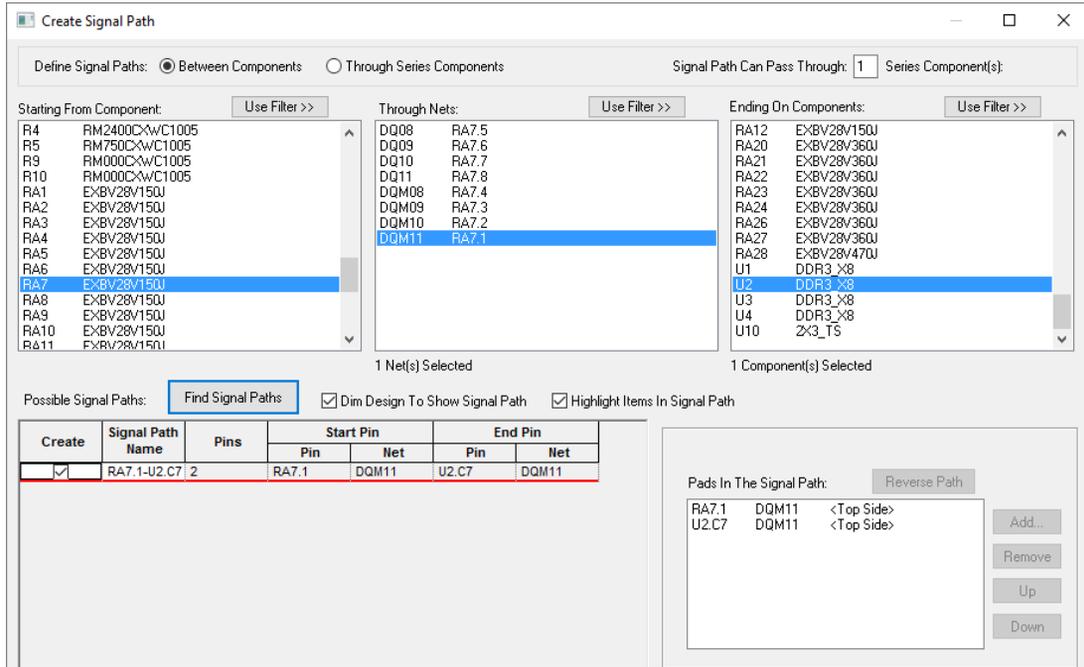
Signal paths are added in the Signal Paths page within the **Technology** dialog. From this page, **Track Length** and **Track Match Rules** can be applied if required or you can use the Signal Path name within other rules, such as **Net Styles**, **Serpentine Rules** or **Thermal Rules** etc.

Name	Pin Count	Start Pin		End Pin		Use Own Colour	Colour	Track Length Rule			Track Attribute	New...	
		Pin	Net	Pin	Net			Attribute	Match	Minimum			Maximum
RA7.1-U2.C7	2	RA7.1	DQM11	U2.C7	DQM11	<input type="checkbox"/>		TrackLength	2.0	1.0	2.0		



There are two dialogs to this function; the main dialog for managing the rules and the secondary dialog to create the signal path. The dialog above is the management dialog.

The dialog below (the signal path chooser) is for signal path creation and is activated from the Signal Paths dialog. It enables you to search for signal paths between two chosen Components, or signal paths that pass through a selected series Component. These items can be selected in the lists at the top of the dialog, or interactively in the design. A signal path doesn't necessarily have to be a direct path through pins and can be 'split' through a series component.



## Creating Signal Paths

The **Signal Path** selection dialog can be invoked using the **New** button from the **Signal Paths** dialog or from the context menu when a component or pads on a component(s) are selected. This dialog will provide you with **Start** and **End** Component pins and connecting nets (**Through Nets**).

A Signal Path doesn't have to be on the same net (it can be though) but when split, can be through a component. You can choose the radio button **Through Series Component** to refine the component pin selection and choose to how many series components the **Signal Path Can Pass Through**.

Once your signal path has been chosen, press the **Find Signal Path** button to add it to the **Possible Signal Paths** list. This will display your choice in the grid. As you select a row in the grid, the pads, connections and tracks in the signal path are displayed in the design.

If satisfied with the signal path displayed, press the **Create** button. If this is your only selection at this point, press **OK** or **Apply** to add the selection to main Signal Paths dialog. If not satisfied with the selection, leave the Create button unchecked and make a new selection in the chooser.

There are additional check boxes on this dialog to assist with the display and selection of items chosen. The **Dim Design To Show Signal Path** check box is used to dim all other items so you can

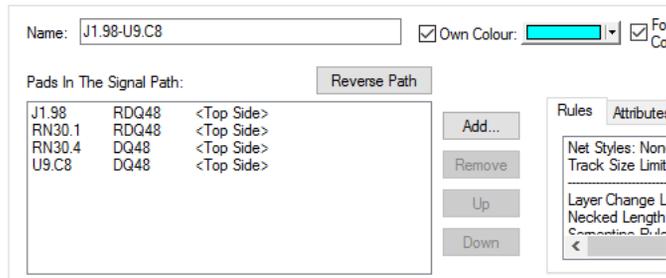
## 32 Pulsonix Version 9.0 Update Notes

easily see the signal path. Likewise, use the **Highlight Items In Signal Path** to make the selected items stand out. This can be used in conjunction with the **Dim Design** button.

Signal paths that are already defined in the main **Signal Paths** dialog will be indicated and changes disabled.

Press **Apply** or **OK** to add the signal paths as entries in the design. Each component pad in the path will be remembered and along with its order with the path.

Once a Signal Path has been created, the main dialog presents you with edit controls for the current signal path in the grid.

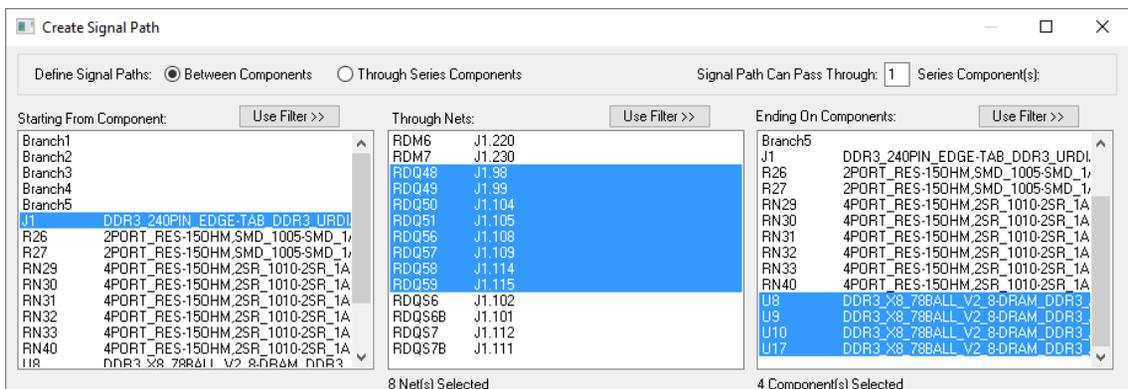


Use these to indicate you want to create the path, apply a name for the signal path and allocate a track length and length match rule to it. If the path is to pass through more than two pins within a particular net, the extra pins can be added into the correct place in a pad list.

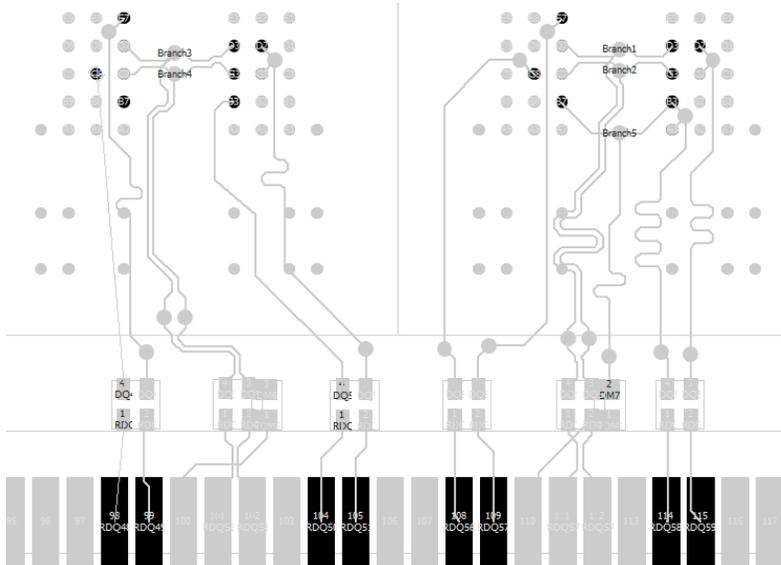
### Adding Multiple Signal Paths in one hit

Multiple signal paths can be created at the same time.

In the example below, Connector J1 has been chosen as the **Start Component**. As it happens, this has all the nets required exiting from it. You must then choose the nets required for the paths in **Through Nets**. Multiple nets can be chosen at one time by dragging down the list or using standard Shift/Ctrl combinations. Likewise, you can make multiple selections in the **Ending On Components** list. Choose the components required.



With this selection made, with the design dimmed, the proposed signal paths are displayed. You can use zoom and pan to navigate the design at this point.



When the **Find Signal Paths** button is pressed, the signal paths are added to the list of **Possible Signal Paths** if paths are available.

Possible Signal Paths:   Dim Design To Show Signal Path  Highlight Items In Signal Path

Create	Signal Path Name	Pins	Start Pin		End Pin	
			Pin	Net	Pin	Net
<input type="checkbox"/>	J1.98-U9.C8	4	J1.98	RDQ48	U9.C8	DQ48
<input type="checkbox"/>	J1.98-U10.C	4	J1.98	RDQ48	U10.C2	DQ48
<input type="checkbox"/>	J1.99-U9.E7	4	J1.99	RDQ49	U9.E7	DQ49
<input type="checkbox"/>	J1.99-U10.E3	4	J1.99	RDQ49	U10.E3	DQ49
<input type="checkbox"/>	J1.104-U9.B	4	J1.104	RDQ50	U9.B3	DQ50
<input type="checkbox"/>	J1.105-U9.D	4	J1.105	RDQ51	U9.D2	DQ51
<input type="checkbox"/>	J1.105-U10.	4	J1.105	RDQ51	U10.C8	DQ51
<input type="checkbox"/>	J1.108-U8.C	4	J1.108	RDQ56	U8.C8	DQ56
<input type="checkbox"/>	J1.108-U17.	4	J1.108	RDQ56	U17.C2	DQ56
<input type="checkbox"/>	J1.109-U8.E7	4	J1.109	RDQ57	U8.E7	DQ57
<input type="checkbox"/>	J1.109-U17.F	4	J1.109	RDQ57	U17.F3	DQ57

Pads In The Signal Path:

J1.98	RDQ48	<Top Side>
RN30.1	RDQ48	<Top Side>
RN30.4	DQ48	<Top Side>
U9.C8	DQ48	<Top Side>

To create the signal paths required, press the **Create** button. If all are required as would be in our example above, use **Create** on one cell and right click. Choose **Apply to entire Column** to select all columns.

Possible Signal Paths:   Dim Design To Show Signal Path  Highlight

Create	Signal Path Name	Pins	Start Pin		End Pin	
			Pin	Net	Pin	Net
<input checked="" type="checkbox"/>	J1.98-U9.C8	4	J1.98	RDQ48	RN30.1	RDQ48
<input type="checkbox"/>	J1.98-U10.C	4	J1.98	RDQ48	RN30.2	RDQ49
<input type="checkbox"/>	J1.99-U9.E7	4	J1.99	RDQ49	RN32.1	RDQ50
<input type="checkbox"/>	J1.99-U10.E3	4	J1.99	RDQ49	RN32.2	RDQ51
<input type="checkbox"/>	J1.104-U9.B	4	J1.104	RDQ50	RN29.1	RDQ56
<input type="checkbox"/>	J1.105-U9.D	4	J1.105	RDQ51	RN29.2	RDQ57
<input type="checkbox"/>	J1.105-U10.	4	J1.105	RDQ51	RN31.1	RDQ58
<input type="checkbox"/>	J1.108-U8.C	4	J1.108	RDQ56	RN31.2	RDQ59
<input type="checkbox"/>	J1.108-U17.	4	J1.108	RDQ56		
<input type="checkbox"/>	J1.109-U8.E7	4	J1.109	RDQ57		
<input type="checkbox"/>	J1.109-U17.F	4	J1.109	RDQ57		

Once created, the main dialog displays the Signal Paths:

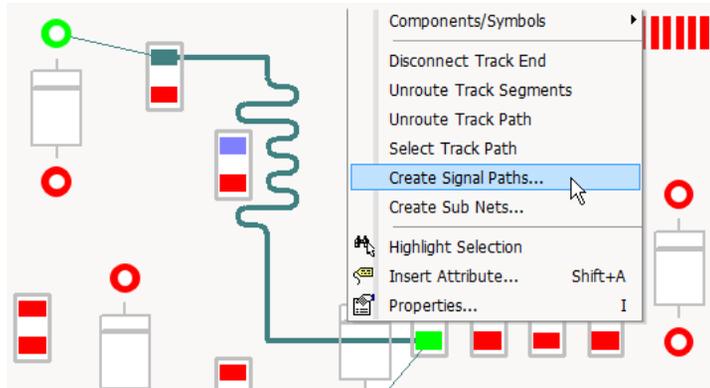
## 34 Pulsonix Version 9.0 Update Notes

Name	Pin Count	Start Pin		End Pin		Use Own Colour	Colour	Display Connection	Track Length Rule			
		Pin	Net	Pin	Net				Attribute	Match	Minimum	M
J1.102-RN33.2	2	J1.10	RDQ56	RN33.	RDQ56	<input type="checkbox"/>		<input type="checkbox"/>	<Signal Path N	J1*	12.000	13
J1.104-RN32.1	2	J1.10	RDQ50	RN32.	RDQ50	<input type="checkbox"/>		<input type="checkbox"/>	<Signal Path N	J1*	12.000	13
J1.105-RN32.2	2	J1.10	RDQ51	RN32.	RDQ51	<input type="checkbox"/>		<input type="checkbox"/>	<Signal Path N	J1*	12.000	13
J1.108-RN29.1	2	J1.10	RDQ56	RN29.	RDQ56	<input type="checkbox"/>		<input type="checkbox"/>	<Signal Path N	J1*	12.000	13
J1.109-RN29.2	2	J1.10	RDQ57	RN29.	RDQ57	<input type="checkbox"/>		<input type="checkbox"/>	<Signal Path N	J1*	12.000	13
J1.114-RN31.1	2	J1.11	RDQ58	RN31.	RDQ58	<input type="checkbox"/>		<input type="checkbox"/>	<Signal Path N	J1*	12.000	13
J1.115-RN31.2	2	J1.11	RDQ59	RN31.	RDQ59	<input type="checkbox"/>		<input type="checkbox"/>	<Signal Path N	J1*	12.000	13
J1.98-RN30.1	2	J1.98	RDQ48	RN30.	RDQ48	<input type="checkbox"/>		<input type="checkbox"/>	<Signal Path N	J1*	12.000	13
J1.99-RN30.2	2	J1.99	RDQ49	RN30.	RDQ49	<input type="checkbox"/>		<input type="checkbox"/>	<Signal Path N	J1*	12.000	13

From here, you can add **Track Length** and **Track Length Match** rules.

### Interactively Creating Signal Paths

In select mode, you can select two pads on the same component and use the context menu option **Create Signal Paths** to create signal paths through the selected component, forcing it to be treated as a series component.



Once selected, the **Create Signal Path** dialog is available with the signal path preselected from where you can use the **Find Signal Paths** button to select the signal path. This entry will then be added to the **Signal Paths** within the **Technology**.

### Sub Nets

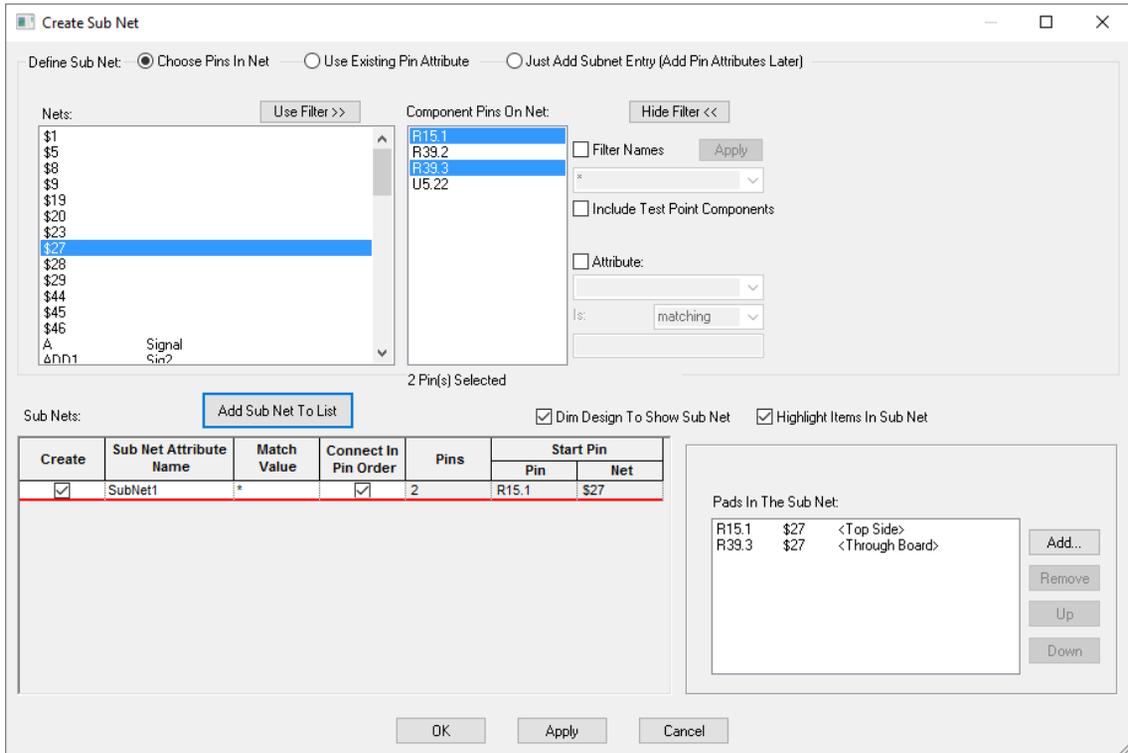
#### Overview

Sub Nets define part of a net which may require special considerations. For example, it allows you to create rules for portions of a net such as a specific pin order (Daisy Chain), or to use attributes on a net to define rules. It also allows you to use portions of a net to use a specific style (i.e. track thickness) and for copying 'channels' of functionality that have net branches and require matching.

Within the **Technology**, Sub Nets are defined in the **Sub Nets** dialog using a pin **attribute name** and **value**. Pins in the same net with attributes that match it are deemed to be in the same sub net. One definition can define sub nets in multiple nets. You may use this for example, when creating branch lengths or a specific daisy chain order in a High Speed design.

## Creating Sub Nets

A new tool provides an interactive mode and modeless dialog to create sub nets. It can be invoked from the **New** button in the **Sub Nets** page within the **Technology** dialog. It can also be accessed from the context menu when a component pin(s) or net is selected.



You can define sub nets by selecting a list of **Pins In a Net**, or you can choose an **Existing Pin Attribute** name if pads already contain their sub net attributes.

Once the Sub Net has been created, you are returned to the **Sub Nets** dialog with the new sub net shown in the **Attribute Name** list:

Attribute Name	Match Value	Connect In Pin Order	Use Own Colour	Colour	Display Connection	Track Length Rule			
						Attribute	Match	Minimum	Maximum
HS1	*	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	<Sub Net Name>	HS1	6.35000	12.70000
Pin_Order	*	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>	<Sub Net Name>	Pin_Order	19.05000	27.94000

---

Attribute:   Own Colour:   Force Display Connections

Match:   Connect In Pin Order (attribute value order)



Nets Containing Sub Net:

DRIVE
-------

Sub Net Pads In The Net:

PL2.1	0	<Through Board>
Q6.3	1	<Through Board>
Q5.3	2	<Through Board>
Q8.3	2	<Through Board>

Rules  Attributes

Net Styles: None
Track Size Limit Rules: None
Layer Change Length Rules: None
Necked Length Rules: None
Segmentation Rules: None

Once a sub net has been created that contains a list of ordered pads, you can use the **Add** and **Remove** buttons or **Edit Values** to adjust the sequence and make edits.

**Add** will allow you to pick multiple pins from any net. **Remove** will only work if the pad has a local attribute value (can't remove a part attribute value).

The **Edit Value** button is used to change the value on the attribute, for setting or changing a pin order for example. You will get an error if the value provided does not match the sub net attribute match string.

### Creating template sub net names with no attributes

When creating sub nets there is a radio button in the **Create Sub Nets** dialog called **Just Add Sub net Entry**. Choosing this will hide the selection lists and create a new sub net row in the grid each time the **Add Sub Net To List** button is pressed. This allows you to create sub nets and pin attribute names, but assign the attributes to the pins at a later stage.

If creating sub nets in a Technology file there are no pins, so pressing the **New** button directly adds a blank sub net to the Technology page.

### Renaming Sub Net Attribute Names

When renaming sub net attribute names, it should be noted that if the attribute name has already been defined for a sub net, that attribute name will still reside on the pin and will not be renamed. Effectively, you are creating a new Attribute name by renaming the existing one.

If you wish to change the attribute name, this must be done in the Attributes page of the Technology. Once the name has been changed, you will then need to select it using the Sub Nets dialog and Attribute Name.

In the example below, the attribute name pppppp has been renamed to Pin\_Order. In order to use this new name, Pin\_Order must be selected from the drop down list.

Attribute Name	Match Value	Connect In Pin Order	Use Own Colour
HS1	*	<input type="checkbox"/>	<input type="checkbox"/>
pppppp	*	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Bob			
HS1			
Pin_Order			

### Connect In Pin Order

Choosing the **Connect In Pin Order** button on this dialog will allow you to define the daisy chain order for the routing sequence in your design. This is defined by the numerical value on the pin and is displayed in the **Sub Net Pads In The Net** box.

Attribute Name	Match Value	Connect In Pin Order	Use Own Colour	Colour	Display Connection	Track Length Rule			
						Attribute	Match	Minimum	Maximum
HS1	*	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	<Sub Net Name>	HS1	6.35000	12.70000
Pin_Order	*	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>	<Sub Net Name>	Pin_Order	19.05000	27.94000

Attribute: Pin\_Order     Own Colour:      Force Display Connections

Match: \*     Connect In Pin Order (attribute value order)

Nets Containing Sub Net: 

DRIVE
-------

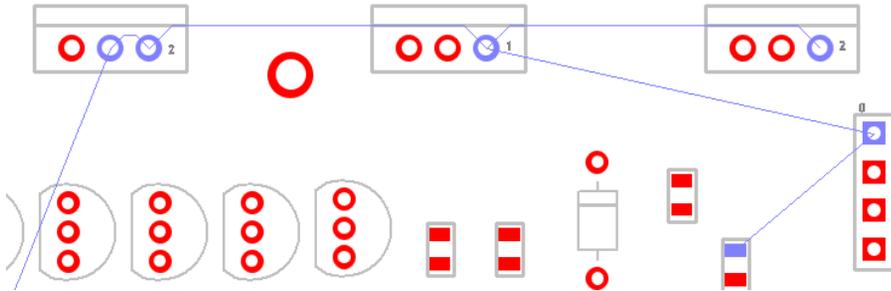
Sub Net Pads In The Net:

PL2.1	0	<Through Board>
Q6.3	1	<Through Board>
Q5.3	2	<Through Board>
Q8.3	2	<Through Board>

Rules    Attributes

Net Styles: None  
Track Size Limit Rules: None  
Layer Change Length Rules: None  
Necked Length Rules: None  
Sometime Edge Maps

The resultant design will connect in this specific order and **Optimise Nets** will also adhere to this order:



### Existing Pulsonix V8.5 Designs

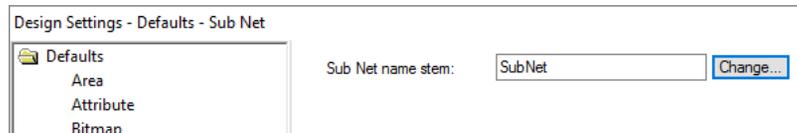
If you have already used **pin attributes** to define sub nets in a previous version of Pulsonix, i.e. V8.5 or older, then these would now be converted into sub nets.

### New Designs

If starting a new design in V9.0, you could use Sub Nets to define specific Daisy Chain orders on a net.

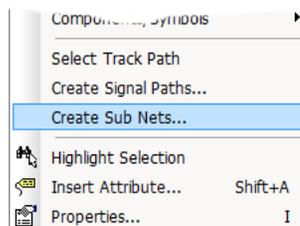
### Designs Settings and Default Sub Net names

Default Sub Net names can be predefined using the **Sub Net** entry on the **Design Settings** dialog. When used, the name is incremented each time it is used. You can edit and rename this name if required.



### Interactively Creating Sub Nets

In select mode, you can select multiple Component pads and Pad branches and use the context menu option **Create Sub Nets** to create sub nets containing those pins.



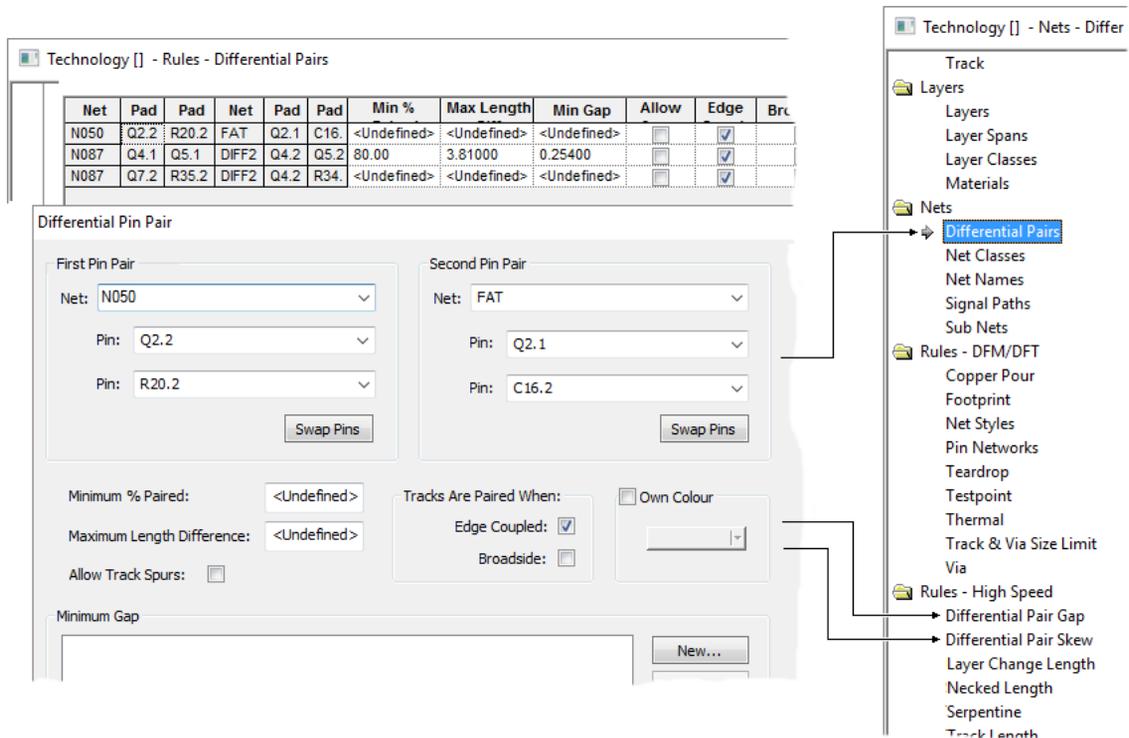
Once selected, the **Create Sub Net** dialog is available with the sub net preselected from where you can add this sub net entry to the **Sub Net** rules within the **Technology**.

## Interactive High Speed Option

### Changes to Differential Pair Rules

The **Differential Pair Rules** dialog has been separated into new pages for Differential Pair definitions and Rules, the rules are now on two pages.

Rules have been taken out of the Differential Pair page and added as separate rules (Differential Pair Gap and Differential Pair Skew), each associated with a rule that can be placed on a differential pair. The actual Differential Pair definitions have their own page. Differential Pair Chains can also be defined on this page (*see more below*).



**Nets - Differential Pairs** – allows **Own Colour**, **Allow Track Spurs**, **Edge Coupled** and **Broadside Coupled**. This dialog also allows you to swap pins to ensure continuity and to **Swap Pin Pairs**. **Rules** and **Attributes** are also displayed for the selected Differential Pair. Using the buttons, you can **Add**, **Edit** or **Remove** Rules and Attributes.

## 40 Pulsonix Version 9.0 Update Notes

Diff Pair Name: J1.112-RN40.2 J1.111-RN40.1 (1)  Own Colour:   

First Pin Pair: Net: RDQS7

Second Pin Pair: Net: RDQS7B

Tracks Are Paired When:  
Edge Coupled:   
Broadside:   
Allow Track Spurs:

Pin: J1.112 Pin: RN40.2  
Pin: J1.111 Pin: RN40.1

Rules Attributes  
Differential Pair Skew Rules: <Differential Pair Name>=J1.112-f Add...  
Differential Pair Gap Rules: None Edit...  
Track Length Rules: None  
Track Match Rules: None Delete

**Rules – High Speed - Differential Pair Gap** – from this dialog, define the **Minimum Gap** between tracks, optionally defined on **Layers** and **Within Areas**. The check box allows you to use the defined Track to Track spacing rule from the **Spacing Rules** dialog or to uncheck the box and override it with a specific typed spacing value.

Attribute: <Differential Pair Name>  Allow Tracks

Match: Q4.1-Q5.1 Q4.2-Q5.2

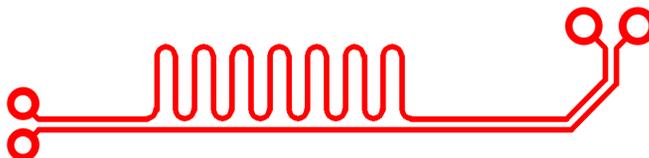
On Layers  
Side: <Any>  
Name:   
Within Areas:   
Minimum Gap:  Use Spacing Rule  
0.25400

**Rules – High Speed - Differential Pair Skew** – this defines the **Minimum Percentage** pairing of tracks. An additional new rule defines the **Maximum Length Difference** between the Differential Pair tracks. This defines the 'skew' between the two tracks in the pair.

Attribute: <Differential Pair Name> Maximum Length Difference: 3.81000

Match: Q4.1-Q5.1 Q4.2-Q5.2 Minimum % Paired: 80

This feature allows you to add skew to one track of the **Differential Pair**.



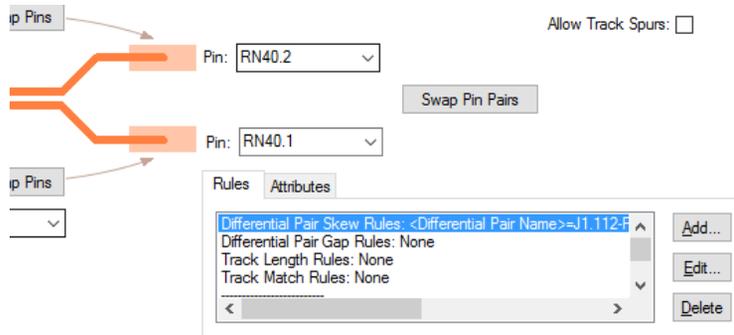
### Differential Pair Names

Differential pairs can also now have a unique name. The default name will be constructed from the four pin names, but can be changed to a name of your own choice.

Name	Chain Link Name	First Pin Pair			Second Pin Pair			Use Own Colour
		Net	Start Pin	End Pin	Net	Start Pin	End Pin	
Diff1-b		DIFF1	Q7.2	R35.2	DIFF2	Q4.2	R34.2	<input type="checkbox"/>
N050		N050	Q2.2	R20.2	FAT	Q2.1	C16.2	<input type="checkbox"/>
Diff Chain1			Q4.1	R36.2		Q4.2	R37.2	<input type="checkbox"/>
	Diff1	DIFF1	Q4.1	Q5.1	DIFF2	Q4.2	Q5.2	<input type="checkbox"/>
	Diff2	DIFF1	R36.1	Q5.1	DIFF2	R37.1	Q5.2	<input type="checkbox"/>
	Diff3	Diff3	R38.1	R36.2	Diff4	R39.1	R37.2	<input type="checkbox"/>

### Differential Pair Attributes

Differential pairs can now have attributes assigned to them for specifying which rules they use. The rules can be added using the **Add** button on the **Differential Pair Rules** dialog or by creating the rule to be assigned on the relevant **Rules** page.



### Differential Paired Chains

Differential Pairs can now also be placed into 'chains' to create extended net paths without the nets requiring the same net name. A Differential Pair may, for example, terminate on a resistor but require the overall length of the tracks to include the other side of the net. Differential Pair Chains can be more than two pairs of Differential Pairs if required.



From the **Differential Pairs** dialog you can use the **New Chain** button to define a **Differential Pair Chain**. If your design only contains one pair of defined differential pairs, then selecting this button will display a warning that more than one pair is required.

## 42 Pulsonix Version 9.0 Update Notes

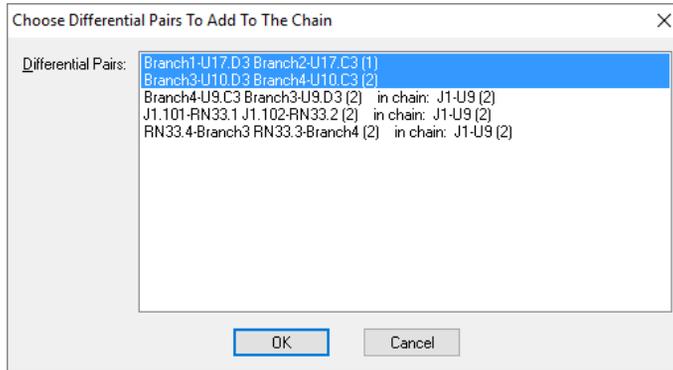
Name	Chain Link Name	First Pin Pair			Second Pin Pair			Use Own Colour
		Net	Start Pin	End Pin	Net	Start Pin	End Pin	
Diff1-b		DIFF1	Q7.2	R35.2	DIFF2	Q4.2	R34.2	<input type="checkbox"/>
N050		N050	Q2.2	R20.2	FAT	Q2.1	C16.2	<input type="checkbox"/>
Diff Chain1			Q4.1	R36.2		Q4.2	R37.2	
	Diff1	DIFF1	Q4.1	Q5.1	DIFF2	Q4.2	Q5.2	<input type="checkbox"/>
	Diff2	DIFF1	R36.1	Q5.1	DIFF2	R37.1	Q5.2	<input type="checkbox"/>
	Diff3	Diff3	R38.1	R36.2	Diff4	R39.1	R37.2	<input type="checkbox"/>

New...

New Chain...

Delete

The process is to choose a Differential Pair to make a chain from, then add additional Differential Pairs to create a chain. After pressing the **New Chain** button, the **Choose Differential Pairs** dialog is displayed:



From this, choose the pairs of pins to add to the chain, this can consist of more than two pairs. Drag the mouse to select more than one or use the standard **Ctrl** or **Shift** keys to make multiple selections.

With a new Chain defined, the lower portion of the Differential Pairs dialog now changes to a **Chain** image to allow you to define the chain pair.

Name	Chain Link Name	First Pin Pair			Second Pin Pair			Use Own Colour	Colour	Edge Coupled	Broadside	Allow Spurs	Attrib
		Net	Start Pin	End Pin	Net	Start Pin	End Pin						
J1-U8 (1)			J1.112	U8.D3		J1.111	U8.C3	<input type="checkbox"/>		<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
	J1.112-RN40.2 J1.111-RN40.1 (1)	RDQS7	J1.112	RN40.2	RDQS7B	J1.111	RN40.1	<input type="checkbox"/>		<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
	RN40.3-Branch2 RN40.4-Branch1 (1)	DQS7	RN40.3	Branch2	DQS7B	RN40.4	Branch1	<input type="checkbox"/>		<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
	Branch1-U8.D3 Branch2-U8.C3 (1)	DQS7B	Branch1	U8.D3	DQS7	Branch2	U8.C3	<input type="checkbox"/>		<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
J1-U9 (2)			J1.101	U9.C3		J1.102	U9.D3	<input type="checkbox"/>		<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
	J1.101-RN33.1 J1.102-RN33.2 (2)	RDQS6B	J1.101	RN33.1	RDQS6	J1.102	RN33.2	<input type="checkbox"/>		<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
	RN33.4-Branch3 RN33.3-Branch4 (2)	DQS6B	RN33.4	Branch3	DQS6	RN33.3	Branch4	<input type="checkbox"/>		<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
	Branch4-U9.C3 Branch3-U9.D3 (2)	DQS6	Branch4	U9.C3	DQS6B	Branch3	U9.D3	<input type="checkbox"/>		<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
J1-U10 (2)			J1.101	U10.D3		J1.102	U10.C3	<input type="checkbox"/>		<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
	J1.101-RN33.1 J1.102-RN33.2 (2)	RDQS6B	J1.101	RN33.1	RDQS6	J1.102	RN33.2	<input type="checkbox"/>		<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
	RN33.4-Branch3 RN33.3-Branch4 (2)	DQS6B	RN33.4	Branch3	DQS6	RN33.3	Branch4	<input type="checkbox"/>		<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
	Branch3-U10.D3 Branch4-U10.C3 (2)	DQS6B	Branch3	U10.D3	DQS6	Branch4	U10.C3	<input type="checkbox"/>		<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
J1-U17 (1)			J1.112	U17.D3		J1.111	U17.C3	<input type="checkbox"/>		<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	

New...

New Chain...

Delete

Diff Pair Chain: J1-U8 (1)

Pin: J1.112 Pin: U8.D3

Pin: J1.111 Pin: U8.C3

Differential Pairs In The Chain:

- J1.112-RN40.2 J1.111-RN40.1 (1)
- RN40.3-Branch2 RN40.4-Branch1 (1)
- Branch1-U8.D3 Branch2-U8.C3 (1)

Add... Remove Up Down

Rules Attributes

Differential Pair Skew Rules: None

Track Length Rules: None

Track Match Rules: Length Match=1(U8.9,10,17) [Max Diff: 1.000]

Net Styles: None

Add... Edit... Delete

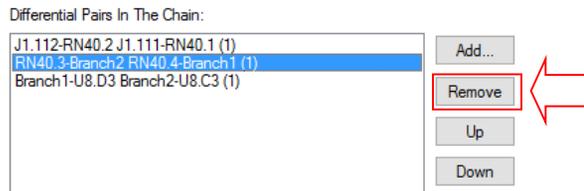
Selecting a Differential Pairs will toggle the image in this dialog between a standard Differential Pair and a Chain.

Additional buttons on this dialog allow you to manage selected Chains. Use the **Add** and **Remove** buttons to select more Differential Pairs to add to the chain or to remove exiting pairs. The **Up** and **Down** buttons allow you to change the order in which the Pairs appear in the chain. The chain name is derived from the first and last pair it finds in the list. It may also be that you add pairs to the chain out of sequence and these buttons allow you to re-sequence them. The order in the list is also used in the **Rules Spreadsheet**.

### Deleting Differential Pair Chains

**Differential Pair Chains** cannot be deleted without first removing the **Differential Pairs** within them. This allows you to remove some pairs and not others from the chain.

Using the **Remove** button, each of the Differential Pairs must be removed.



Once this has been completed and there are no more Differential Pairs shown in the chain, the **Delete** button on the dialog is then available to delete the **Chain** name.

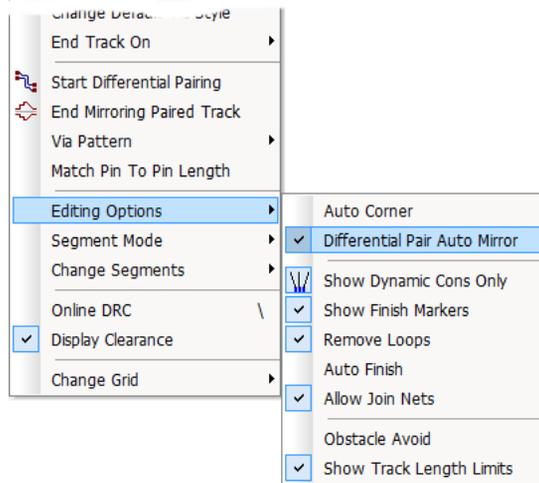
Name	Chain Link Name	First Pin Pair			Second Pin Pair			Use Own Colour	Colc
		Net	Start Pin	End Pin	Net	Start Pin	End Pin		
Branch1-U1		DQS7B	Branch1	U17.D3	DQS7	Branch2	U17.C3	<input type="checkbox"/>	
Branch3-U1		DQS6B	Branch3	U10.D3	DQS6	Branch4	U10.C3	<input type="checkbox"/>	
J1-U8 (1)			J1.112	U8.D3		J1.111	U8.C3	<input type="checkbox"/>	
	J1.112-RN40.2 J1.111-RN40.1 (1)	RDQS7	J1.112	RN40.2	RDQS7B	J1.111	RN40.1	<input type="checkbox"/>	
	RN40.3-Branch2 RN40.4-Branch1 (1)	DQS7	RN40.3	Branch2	DQS7B	RN40.4	Branch1	<input type="checkbox"/>	
	Branch1-U8.D3 Branch2-U8.C3 (1)	DQS7B	Branch1	U8.D3	DQS7	Branch2	U8.C3	<input type="checkbox"/>	

### Differential Pair Auto Mirror option

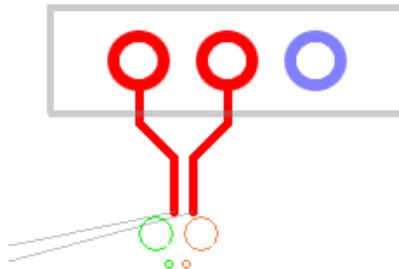
There is a new option to always start mirroring when adding a PCB track from a differential pair pad. This option is available on the context menu when in **Add Track**, and in the **Editing Options** sub menu on the context menu when editing the track.

## 44 Pulsonix Version 9.0 Update Notes

---



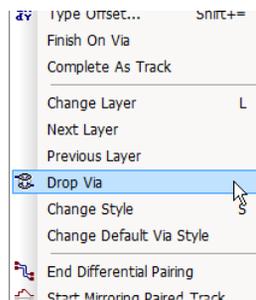
With the option enabled, when you then start routing, the differential pair will be immediately mirrored.



### Drop Via option

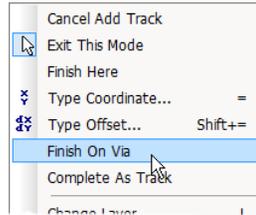
When adding a PCB track (or differential paired track), there is a new option on the context menu to **Drop Via**. This operates the same as **Change Layer**; adding a via at the end of the track but for this option, not changing the layer of the new track. A via is added along the track as you edit, thus enabling the continuation of the track after the via is dropped.

This is also available when adding differential pairs where it adds two vias with the appropriate track patterns to them, but not changing layer. This is useful when using DDR2 or DDR3 flyby routing to drop a via pair near each target pad pair, ready to be routed later.



## Finish On Via option

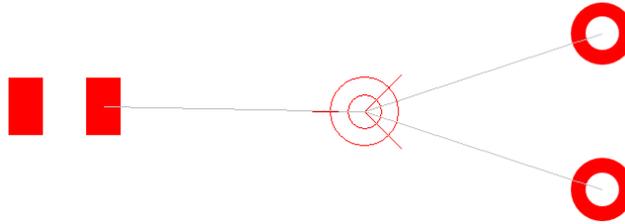
The **Finish On Via** option is now available when adding differential paired tracks. Use it to finish a paired section on vias, you may need to do this if creating a branch point for multiple-connected differential pairs. It was always available when adding a single PCB track, but is now available when adding differential paired tracks.



## Branch Points

Branch points have been added Pulsonix to enable intelligent splitting of nets. They are named items in the form of auto-generated doc symbols or user-defined doc symbols or vias (in PCB).

They can added to a net in a Schematic to indicate a point where the track in the PCB is to split to branch to two or more target pads. Alternatively they can be added to sections of a Net(s) for when you wish to create specific rules for given track segments e.g. fattening a track segment.

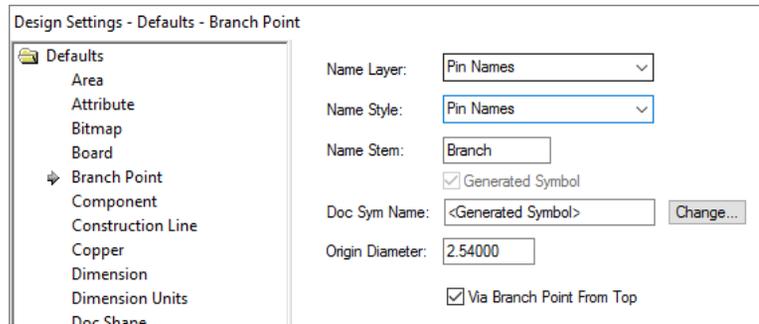


When added to a Schematic design they are automatically transferred to the PCB. Once added they can be used on **Signal Paths**, **Sub Nets** and **Differential Pairs**.

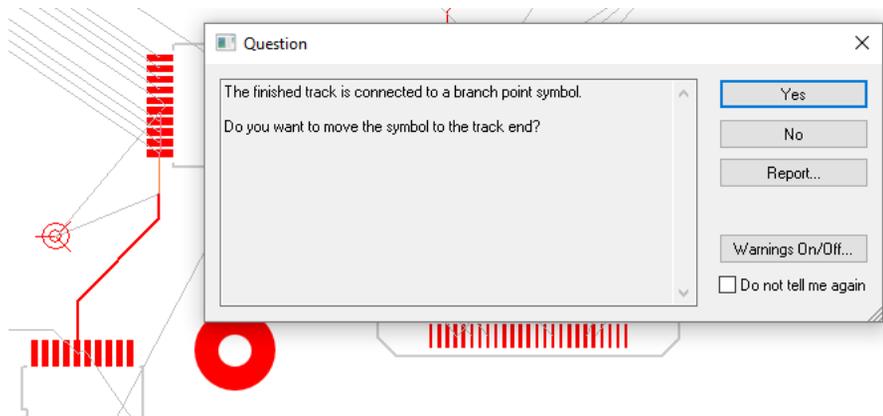
Branch points are added to the design using the **Add Branch Point** option or by using **Change Branch Point** on a selected **via**. When added, you are required to attach the branch point to a pad or connection.

Branch Points do not require a doc symbol for addition, you can specify that they are to be **Auto-Generated**, in which instance Pulsonix will add its own basic symbol suitable for use. The default branch point origin is shown as a target with three lines to make it stand out in a design.

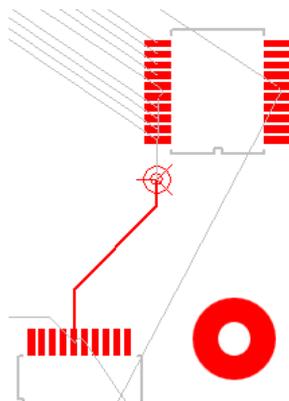
The default symbol or auto-Generated symbol is specified in the **Design Settings** dialog and **Branch Point** page. You might use your own **Branch Point Doc Symbol** to add extra detail for plotting or some other highlighting. When branch points are used in the design they are zero size items and are purely used for electrical connectivity purposes.



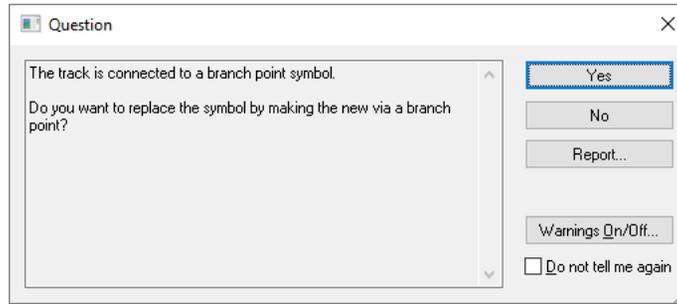
When adding a track (or a differential paired track) with a connection attached to the end that ends on a branch point, after finishing the track at a set position you will be asked if you want to move the branch point to the end of the track:



If **yes** is pressed, the branch point is then moved to the end of the track:



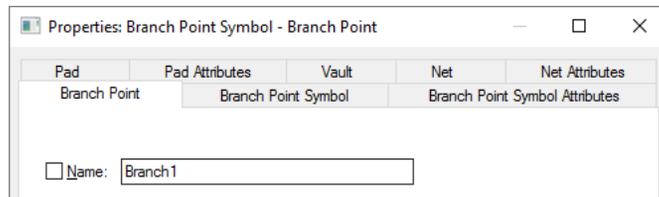
If the track ends on a via then you will be asked if you want to change the via to a branch point via, replacing the doc symbol branch point.



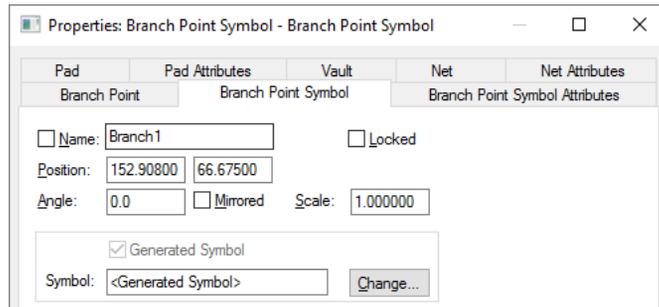
## Branch Point Properties

A branch point doc symbol or via has two new tabs in Properties to show the **Branch Point** name and **Branch Point Symbol** information.

The **Branch Point** tab allows you to display and change the name of the branch point.



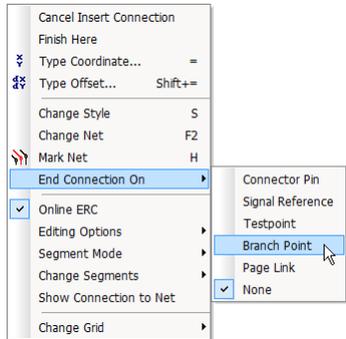
The **Branch Point Symbol** tab displays information about the position of the branch point and the symbol used.



For **auto-generated** branch points, the **Properties** dialog will display a **Symbol** name of **<Generated Symbol>** with the option to change it for one of your own from the library.

## Branch Points – Additional Features

In schematic you can start and end a connection directly on a new branch point from the context menu using the **Start/End Connection On** option.



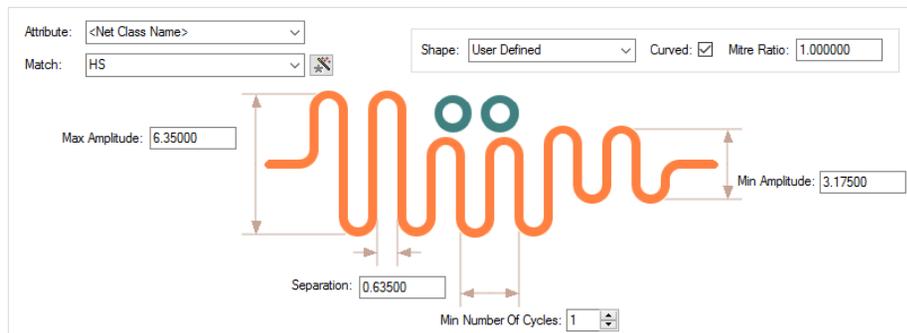
You can **cross probe** branch points between Schematic and PCB, and vice-versa.

You can use **Auto Rename** and **interactive rename** tool on branch points.

## New Serpentine Routing Rules

**Serpentine Rules** are now available on a separate dialog under the **Rules – High Speed** section. From this dialog, you can create rules for use with nets and attributes. The serpentine rules are visible as rule shapes are selected in the preview window.

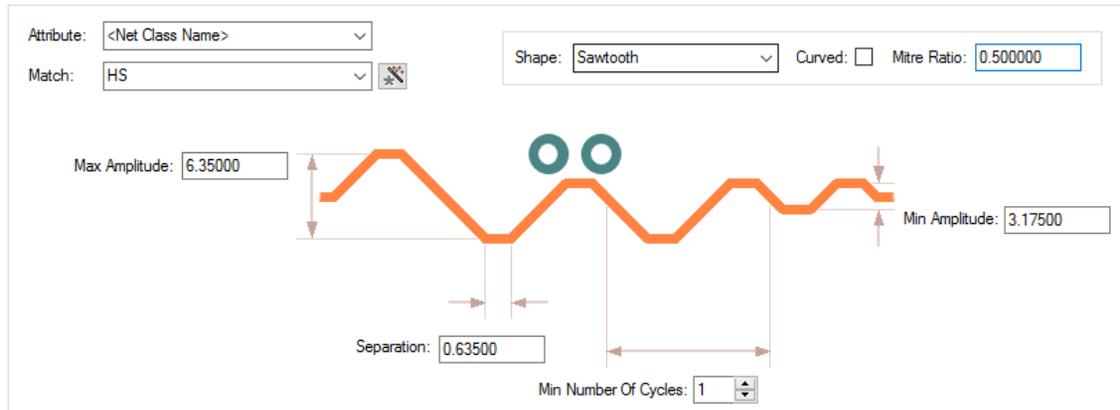
Attribute Name	Match Value	Min Amplitude	Max Amplitude	Separation	Min Cycles	Shape	Curved	Mitre Ratio
<Net Class Name>	HS	3.17500	6.35000	0.63500	1	User Defined	<input checked="" type="checkbox"/>	1.000000
<Net Class Name>	HS1	1.52400	3.04800	0.30480	1	User Defined	<input checked="" type="checkbox"/>	1.000000
<Net Class Name>	RF	1.52400	1.77800	0.30480	1	User Defined	<input checked="" type="checkbox"/>	1.000000
<Net Class Name>	*	3.17500	6.35000	0.63500	1	180 Degree C	<input checked="" type="checkbox"/>	1.000000



There have also been new rules introduced for **Sawtooth** and **Trombone** styles for use during serpentine routing.

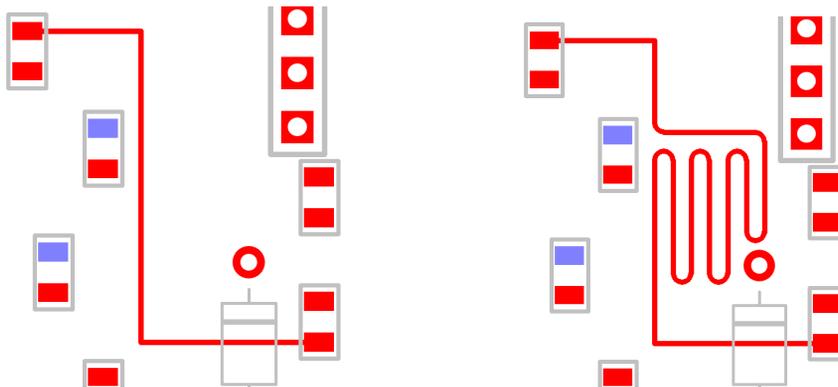
## Sawtooth

The sawtooth shape has been introduced. **Min** and **Max amplitude** as well as **separation** can be defined. Changing the **Mitre Ratio** will adjust the sawtooth shape allowing more 'flat' spots on the waveform.

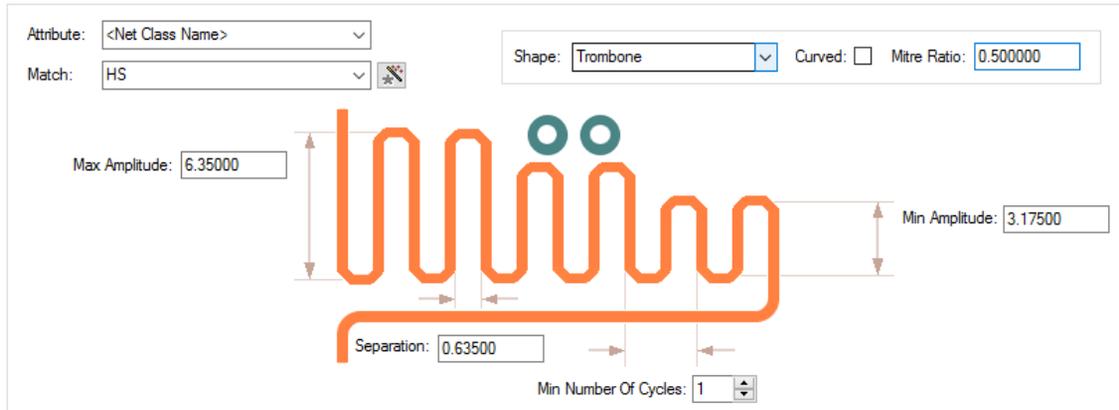


## Trombone

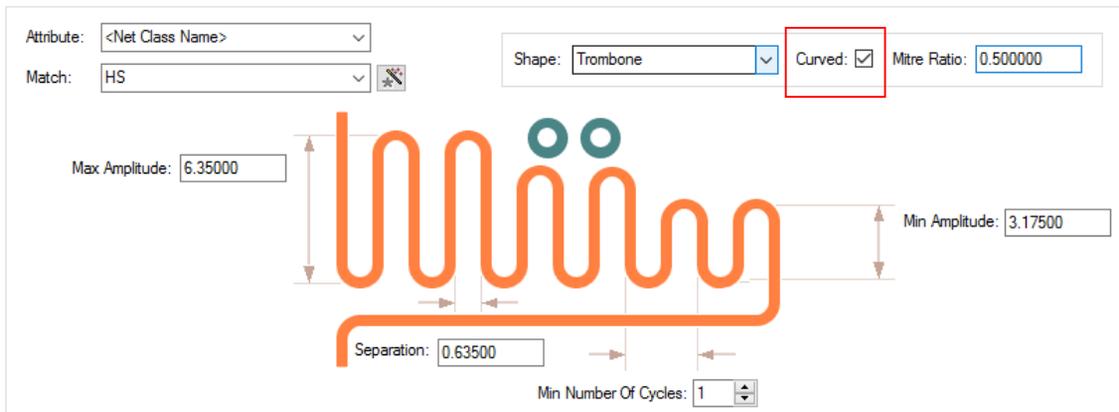
The **Trombone** style breaks into the track and edits it into the trombone shape. The pictures below show the track before serpentine routing and after:



Two styles are available for **Trombone** – **Octagonal** and **Curved**.



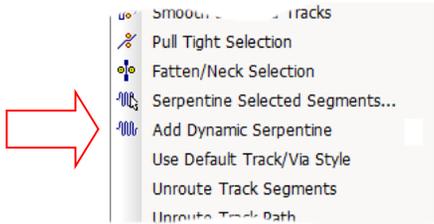
Check the **Curve** button to toggle between **Octagonal** to **Curved** shapes.



### Dynamic Interactive Serpentine Routing

Serpentines tracks can now be created and edited using the new interactive serpentine tool. The serpentine track can be added by selecting and dragging along a track and automatically finishing once the correct length is met. This can be done once a track has been added to the design to increase or decrease its length. Obstacles will be avoided with Online DRC enabled; the serpentine tool will increase/decrease its size around obstacles. Interactive Serpentine will work continuously around angled and curves.

To facilitate this, a new mode, **Add Dynamic Serpentine**, is available on the context menu for a selected track segment. This mode can also be assigned as a command to a shortcut key for fast deployment.

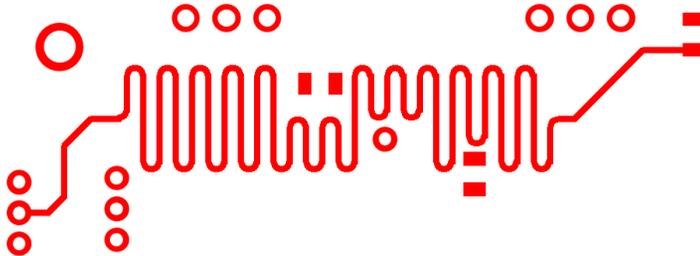


This can also be invoked from the **Utilities** menu, **Serpentine, Serpentine Mode** option.

You can now interactively add a serpentine by dragging along a track and automatically finishing when the correct length is reached. You can edit a serpentine to change its size, but still adhere to a length rule. With Online DRC switched on, the serpentine reduces its width in places to avoid obstacles, and pushes tracks out of the way if it can.

Once a track has had a serpentine applied to it, that serpentine is remembered by the system and can subsequently be edited with all rules and values available.

The dynamic serpentine mode can be applied to Tracks and Differential Pairs, as well as one track of a Differential Pair to create a 'skew' (this requires the **Differential Pair Skew Rule** to be defined in the Technology).

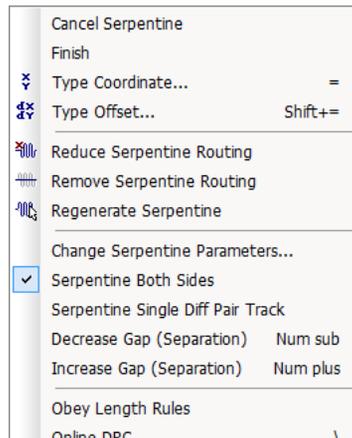


## Finding Serpentine

You can now use the **Find** option to locate serpentine routing. The reported serpentine will be the Net Name, this will be routing that has been created using the **Serpentine** option.

### New Serpentine Routing Functionality

There are new context menu items available when serpentine routing:



#### Existing Commands:

**Reduce** and **Remove Serpentine Routing** – existing commands to reduce the serpentine by one half cycle and to remove the selected serpentine routing.

**Serpentine Selected Segments** – with a track segment selected, this interactive command enables you to apply serpentine routing. It is only available on the context menu for a selected track segment.

#### New commands:

**Regenerate Serpentine** – use this command to regenerate the serpentine based on the **Serpentine Rule** defined. This will remove the current routing pattern and regenerate it.

**Change Serpentine Parameters** – once a serpentine has been added, the selected routing can be regenerated using local parameters defined in this dialog. This enables you to define the parameters or overwrite the rule already defined in the **Serpentine Rule** that the current serpentine routing is using.

**Serpentine Single Diff Pair Tracks** – with a Differential Pair selected, this command allows you to serpentine just one track of the pair. This command is only available if the Differential Pair matches the rule and value defined in **Differential Pair Skew Rules**. To activate this and for it to be available on the menu, you must be adding a serpentine to the Differential Pair. Whilst editing, right click and the following dialog is displayed. Select **Serpentine Single Diff Pair Tracks** to add a serpentine to one track. The single track using the serpentine is the one selected when the Differential Pair is selected.



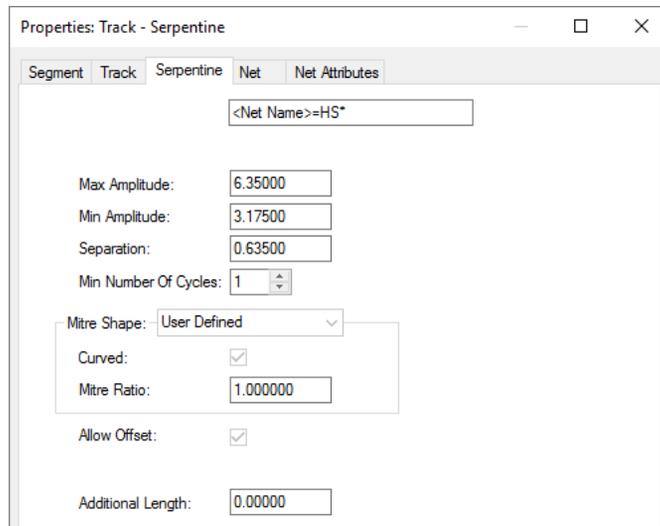
**Decrease Gap** – during interactive serpentine editing, use this command to decrease the gap between the serpentine tracks. This spaces the serpentine closer together apart and overrules the **Serpentine Rule** if defined.

**Increase Gap** – during interactive serpentine editing, use this command to increase the gap between the serpentine tracks. In other words, this spaces the serpentine wider apart and overrules the **Serpentine Rule** if defined.

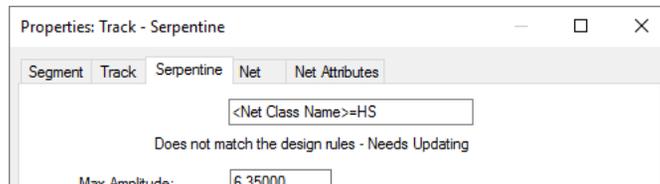
**Obey Length Rules** – with this option selected, the serpentine routing will follow the **Serpentine Rule** and **Track Rule** if defined.

## Serpentine Properties

When selecting a **track** that contains a **serpentine** and the serpentine itself is selected, the **Properties** dialog contains an extra tab to display the serpentine rule used and a report of the actual values of the serpentine.

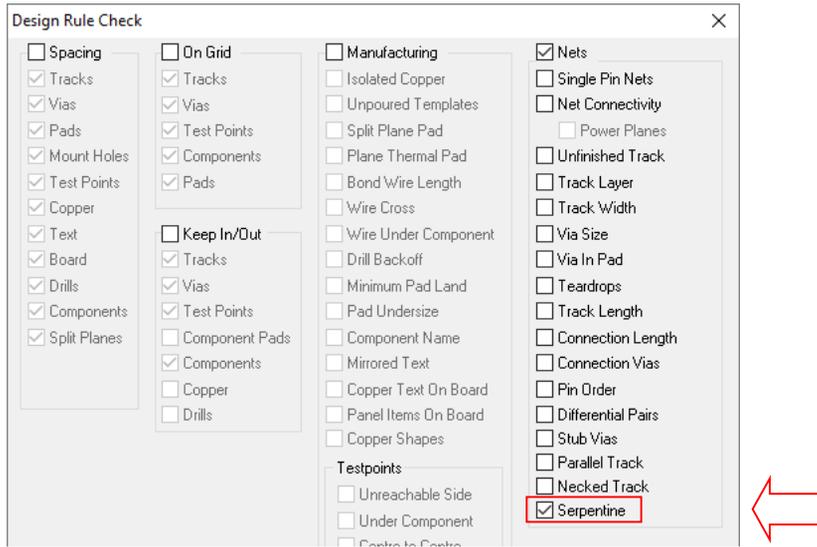


If the serpentine has been defined and the rule changed or the rule created manually using local serpentine parameters so now it doesn't match the Serpentine Rule defined in the Technology, the Serpentine Properties will report that the serpentine needs updating. Use the Regenerate Serpentine option from the context menu of the selected serpentine to regenerate it according to the serpentine rule.



## DRC Check for Serpentine Routing

There is a new **DRC** check (**Serpentine**) in the **Nets** section to show serpentes generated with out-of-date parameters according to the rules defined in **Technology** and **Serpentine**.



## Layer Change Length Rule

A new **High Speed** rule has been added - **Layer Change Length Rule**. Use this to add extra lengths to be added to net length calculations when a track changes layer through a via or component pin. The 'best fit' layer pair will be searched for when looking for extra length to be added for a layer change.

Using this rule allows you to add layer span thicknesses for addition and consideration to the overall net length where accurate length is critical.

From the **Technology** dialog and **Layer Change Length Rule** page, this rule is defined using an **Attribute Name** (such as Net Name, Net Class, Signal Path name etc.), a **Match Value** and the **pair of layers** to specify the rule for.

Attribute Name	Match Value	From Layer	To Layer	Applies To		Extra Track Length	
				Pads	Vias	Use Layer Thicknesses	Length
<Net Class Name>	HS	Top	Bottom	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1.60500

Attribute: <Net Class Name>

Match: HS

From Layer: Top

To Layer: Bottom

Applies To:  Pads  Vias

Use Layer Thicknesses

Extra Track Length: 1.60500

**Applies To:** - this enables you to specify whether the rule applies to only **Pads** or only **Vias** or both. You cannot have both check boxes not selected.

**Extra Track Length:** - this allows you to specify the rule. By using the layer span, you can define a thickness that gets added to the overall track length. This additional length can be derived using the layer **Material** thickness and layer setup from the **Layers** dialog or can be specified directly as a typed value by unchecking the **Use Layer Thickness** check box.

## Extra Length Rule through Pin Package Attribute

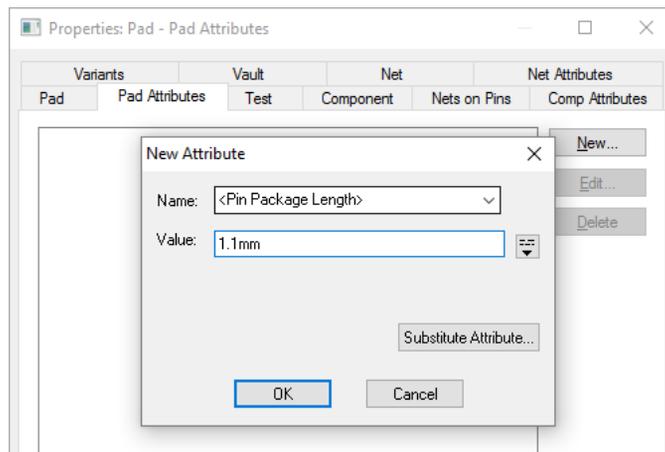
It is now possible to add extra length to the overall track length for **Pads** or to **define an internal length within a component** (the pin package length). This rule comes into effect particularly where bus clock speeds of 500Mhz or above are being used. It is used to add extra length to the net length calculations

There isn't a Rule page for this but it fits here in relation to the above **Layer Change Length Rule**.

The extra length is defined on an IC manufacturer's datasheet and defines a pins internal package length. The internal bond wire to the die introduces a delay to the signal. This delay information can usually be found in the IBIS 6 document for the device.

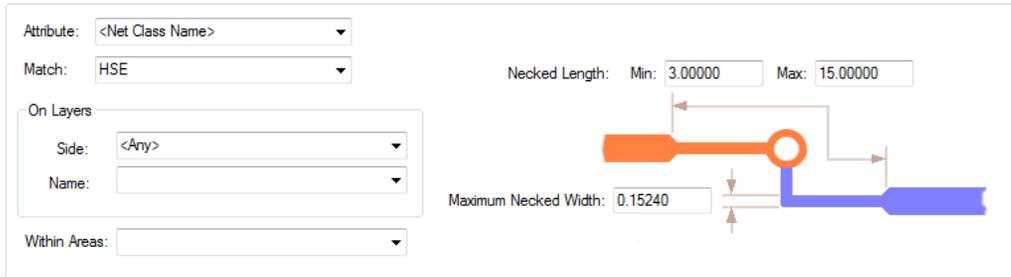
## Defining the extra length

Within the design, the **<Pin Package Length>** attribute can be added to any pads or in Parts if you wish to account for internal package lengths. It should be added using **Properties** of a **Pad**, **Pad Attributes** and adding the **<Pin Package Length>** attribute plus a length value. The **Value** should also have units defined otherwise the current design units will be used. This could be a problem if swapping design units dynamically and the wrong length being used. The value should be the length of the continuation of the net inside the package.



### Necked Length Rule

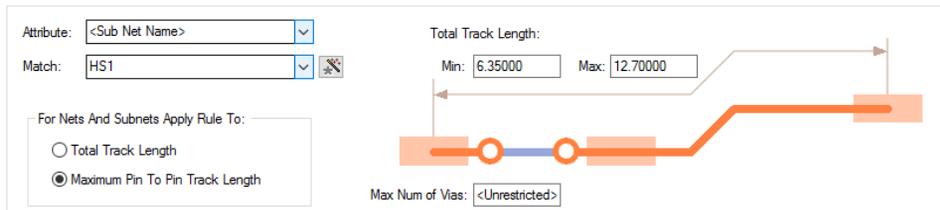
A new rule has been introduced to **Rules – High Speed** for defining track **Necked Length**. The **Min** and **Max Necked Length** rule will be checked when the track length falls below the **Maximum Necked Width**. Select **Necked Track** in the **Design Rules Check** dialog to check this rule.



### Track Length Rule

The **Track Length Rules** page enables you to define **Min** and **Max Track Lengths**. These can be applied to any Net-based items and as Attributes to the items. Pre-V9.0, this information was previously found under the **Net Class** dialog and **Net Rules** page.

Attribute Name	Match Value	Total Track Length		Max Vias	For Nets and Subnets Apply Rule To	
		Minimum	Maximum		Total Track Length	Pin To Pin Track Length
<Differential Pair Name>	Diff	33.02000	44.45000	2	<input type="checkbox"/>	<input checked="" type="checkbox"/>
Track_Length	HS	5.08000	81.28000	2	<input checked="" type="checkbox"/>	<input type="checkbox"/>
<Net Class Name>	HS1	63.50000	76.20000	<Unrestricted>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
<Net Class Name>	HSE	8.00000	14.00000	1	<input type="checkbox"/>	<input checked="" type="checkbox"/>
<Net Class Name>	PAIR	38.10000	45.72000	<Unrestricted>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
<Net Class Name>	Sig2	19.05000	27.94000	2	<input type="checkbox"/>	<input checked="" type="checkbox"/>
<Net Class Name>	Signal	2.54000	17.78000	<Unrestricted>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
<Sub Net Name>	HS1	6.35000	12.70000	<Unrestricted>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
<Sub Net Name>	Pin_Order	19.05000	27.94000	2	<input type="checkbox"/>	<input checked="" type="checkbox"/>



Within this rule there is a sub-category to define separate rules the **For Nets And Sub Nets Apply Rule To:** can be selected for the **Total Track Length** or for the **Maximum Pin To Pin Track Length**. You can have the same rule contain both of these sub-categories but they must be defined twice, once for each rule.

## Track Length Factor Rule

The **Track Length Factor Rule** allows you to adjust the reported length of a track dependant on its side, layer or area.

The **Track Length Factor** is defined as a multiplier of the actual track length. The default value will be 1.00. Making the value less than 1.00, i.e. 0.90, will mean the track will be reported as being 90% of its actual length. This adds delay or increases delay to the track if this is required.

Attribute Name	Match Value	Side	Layer	Area	Track Length Factor
<Net Name>	DQ*	Inner			0.900000

Attribute: <Net Name>

Match: DQ\*

On Layers

Side: Inner

Layer:

Within Areas:

Track Length Factor: 0.9000

## Track Length Match Rule

The **Track Length Match Rule** allows you to define the rule that will match the length of different nets.

Attribute Name	Match Value	Extra Delay Length	Max Length Difference
<Differential Pair Name>	DQ*	0.00000	3.81000
<Net Name>	HS*	0.00000	12.70000
<Net Class Name>	HSE	0.00000	2.00000
<Net Class Name>	PAIR	0.00000	6.35000

Attribute: <Net Name>

Match: HS\*

Max Length Difference: 12.70000

This is defined as the difference between the longest and shortest track using this rule. The first track added is both the longest and shortest at that point. If the next track using this rule added is shorter, the rule is now checked. If the shorter track is within tolerance, it is legal. If outside of this tolerance then it will be flagged as illegal.

The illustration below shows how the rule will work with two matched tracks. The inwards facing arrows indicate the Max Length Difference:



The next scenario is where a third track is added (that also has the same match rule) but this time it is shorter than the current shortest track. This now becomes the shortest track. As you can see, the Length Difference has increased.



If the third track added was longer than the current shortest track (the middle one) the middle track is still the shortest and the Length Difference hasn't changed:



If the third track added is longer than the current longest track, then the rule span now changes and potentially the rule has been violated (because of the increased length):



When adding tracks, you should be aware that by default, Pulsonix will treat the longest track as the matched target unless you override this rule using the **Matched Target** check box in the **Rules Spreadsheet** (see below). Therefore, adding a longer track can invalidate existing tracks using this same rule.

### Rules Spreadsheet Matched Target

The **Matched Target** check box on the **Rules Spreadsheet** allows you to override the current matched target (the 'longest' track) and select a different track that is more preferential to match the rule against.

Rules Spreadsheet

All Item Types Edit Columns... Colours... Options... Export CSV...

	Matched Target	Max Pair	Pair Skew	Length Rule	Min Length	Max Length	Match Rule	Max Length	Length	Len
	<input type="checkbox"/>		-0.859+				Length Match=J1[U8,9,10,17]	1.000	18.946+	2.55
	<input type="checkbox"/>			Length Match=RDQ48-59	14.000	15.500	Length Match=RDQ48-59	1.000	14.942-	1.14
	<input type="checkbox"/>			Length Match=RDQ48-59	14.000	15.500	Length Match=RDQ48-59	1.000	14.942-	1.14
	<input type="checkbox"/>			Length Match=RDQ48-59	14.000	15.500	Length Match=RDQ48-59	1.000	16.082+	1.14
	<input type="checkbox"/>			Length Match=RDQ48-59	14.000	15.500	Length Match=RDQ48-59	1.000	16.082+	1.14
	<input type="checkbox"/>			Length Match=RDQ48-59	14.000	15.500	Length Match=RDQ48-59	1.000	15.539-	A 1.14
	<input type="checkbox"/>			Length Match=RDQ48-59	14.000	15.500	Length Match=RDQ48-59	1.000	15.789-	A 1.14
	<input checked="" type="checkbox"/>			Length Match=RDQ48-5	14.000	15.500	Length Match=RDQ48-59	1.000	15.834-	1.14
	<input type="checkbox"/>			Length Match=RDQ48-59	14.000	15.500	Length Match=RDQ48-59	1.000	16.084-	1.14
	<input type="checkbox"/>			Length Match=RDQ48-59	14.000	15.500	Length Match=RDQ48-59	1.000	15.345-	1.14
	<input type="checkbox"/>			Length Match=RDQ48-59	14.000	15.500	Length Match=RDQ48-59	1.000	15.295-	1.14
	<input type="checkbox"/>			Length Match=RDQ48-59	14.000	15.500	Length Match=RDQ48-59	1.000	15.295-	1.14
	<input type="checkbox"/>						<Signal Path Name>="Branch"	0.500	16.364+	A 1.33

With this selected, all other matched tracks will display any length differences relative to this one.

### Extra Match

The **Track Length Match Rule** also has ability to add **extra match** strings to it. Use the Extra Match button to create extra rows for the current Match rule. Using this, enables you to directly define a rule for two net names for example without having to use complicated match strings.

Attribute Name	Match Value	Extra Delay Length	Max Length Difference
Length Match	RDQ48-59	0.000	1.000
<Signal Path Name>	"Branch"	0.000	0.500
Length Match	J1[U8,9,10,17]	0.000	1.000
<Net Name>	DM6	0.000	0.000
	DM7	0.000	

Attribute: <Net Name>

Match: DM6

Extra Match: DM7

Extra Delay Length: 0.000

Max Length Difference: 0.000

When the **Extra Match** rule is set to zero, this means simply ensure that the explicit nets are matched using the Max Length Difference. If, however, an explicit override rule is required, then the **Extra Delay Length** can be assigned (see below).

### Extra Delay Length

The **extra match** strings can also have an optional **Extra Delay Length** assigned to them. This is done using the extra match button but then using an explicit **Extra Delay Length** value. This means the extra match item(s) must be that length longer than the base match item to satisfy the rule. The value can also be negative meaning it must be shorter than the base item.

Attribute Name	Match Value	Extra Delay Length	Max Length Difference
Length Match	RDQ48-59	0.000	1.000
<Signal Path Name>	*Branch*	0.000	0.500
Length Match	J1[U8,9,10,17]	0.000	1.000
<Net Name>	DM6	0.000	1.100
	DM7	2.000	

Attribute: <Net Name>

Match: DM6

Extra Match: DM7

Extra Delay Length: 2.000

Max Length Difference: 1.100

### Track Length Match Rule – Sub Nets

An additional option is available for the **Track Length Match Rule**. It is present when the <Sub Net Name> attribute has been selected from the drop down and now displayed an additional check box for **Only Match Sub Nets Within the Same Net**.

This allows you to say only match a sub net with other sub nets in the **same net**. You would select this if having copied an identical section of circuit that has sub nets within it. Leave it unchecked to match the length of all matching sub nets in any net.

Attribute: <Sub Net Name>

Match: HS1

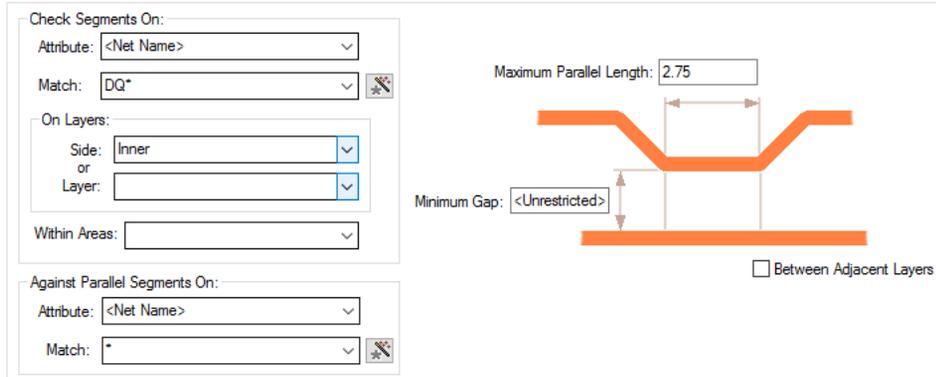
Only Match Sub Nets Within the Same Net

Max Length Difference: 3.81000

### Track Parallel Segments Rule

A new rule has been introduced to **Rules – High Speed** for defining **Track Parallel Segments**. This is used to define the maximum length of two parallel segments. You can also define the minimum gap between the parallel tracks. Both rules can be used on the same or adjacent layers.

Check Segments On					Against Parallel Segments On		Parallel Track Segments		
Attribute Name	Match Value	Side	Layer	Area	Attribute Name	Match Value	Between Adjacent	Min Gap Between	Max Parallel Length
<Net Name>	DQ*	Outer			<Net Name>	*	<input type="checkbox"/>	<Unrestricted>	3.00000
<Net Name>	DQ*	Inner			<Net Name>	*	<input type="checkbox"/>	<Unrestricted>	2.75000



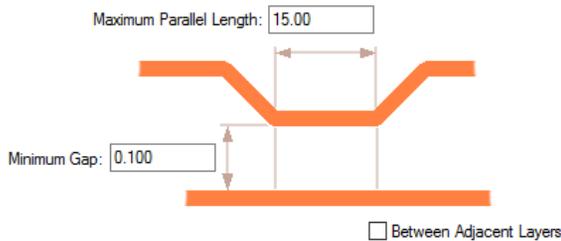
If the **Minimum Gap** is left undefined <Unrestricted>, it will use the default Track to Track spacing rule defined in the **Spacing Rules** dialog.

You can also define multiple rules between two sets of tracks with increasing minimum gaps.

Check Segments On					Against Parallel Segments On		Parallel Track Segments		
Attribute Name	Match Value	Side	Layer	Area	Attribute Name	Match Value	Between Adjacent	Min Gap Between	Max Parallel Length
<Net Name>	DQ48				<Net Name>	*	<input type="checkbox"/>	0.100	15.000
<Net Name>	DQ48				<Net Name>	*	<input type="checkbox"/>	0.150	17.000

### How the rule works

For the example below, the rule is: if the two tracks are within the minimum gap of 0.100mm, you cannot have a run of longer than 15.00mm



### Track Length Limit Indicator

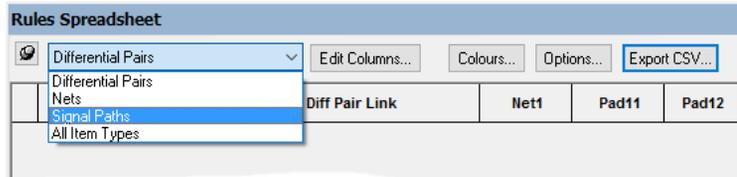
When the track length indicator is displayed, it now also shows the Min and Max limit for match length rules.



### Changes to the Rules Spreadsheet

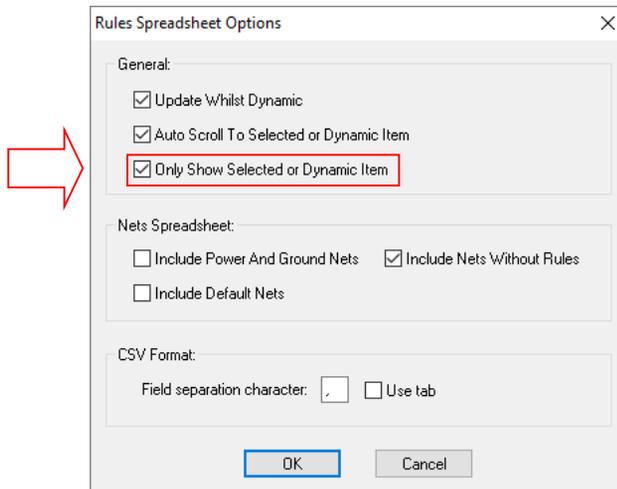
#### Inclusion of Signal Paths

The **Rules Spreadsheet** has been changed to include **Signal Paths** nets and their rules.



#### Only Show Selected or Dynamic Item

From the **Options** button on the **Rules Spreadsheet**, there is a new option to **Only Show Selected or Dynamic Item**.



With this checked, only currently selected or net items being edited in the design (Nets, Signal Paths, Differential Pairs, etc.) will be seen in the spreadsheet. The one shown will be the item selected plus all other items that are in a Match Length group with this item. This enables the Rule Spreadsheet to be decluttered for greater clarity.

---

*Note: be aware, with this option selected, it is easy to think that nothing is being shown in the Rules Spreadsheet, remember, you must have a net item selected or edited in the design!*

---

#### New fields added to Rules Spreadsheet

New fields have been added to the Rules Spreadsheet:

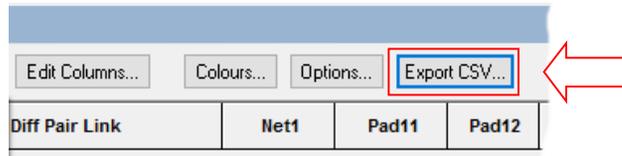
- **Track Length** and **Track Length Match** fields have been added to the **Differential Pair** and **Signal Path** spreadsheets.

- **Diff To Shortest** and **Diff To Longest** fields have been added to show the difference between the length of each row item and from the longest and shortest in its match rule set.
- **Matched Target** field has been added to set a net, signal path, sub net or Differential Pair as the length target for the match rule set.
- **Has Spurs** has been added for Differential Pairs, Signal Path and All Item Types spreadsheets.
- Previously, the **Complete** field was shown as false if a Differential Pair had a spur and allow spurs was false. Now in 9.0 **Complete** means there is a complete path between the pins.
- There is now a **Has Spurs** column to show if Signal Path or Differential Pair have spurs and it is highlighted in error (using the **Rules Spreadsheet Colours** for **Value is less than the minimum rule value** colour) if **Allow Spurs** on the Differential Pairs **Rules** page is not checked (false) but a spur exists on the Differential Pair.

Length Diff	Diff To Targ	Diff To Shor	Diff To Lon	Complete	Allow Spurs	Has Spurs	Edge Coupled
0.84169	-0.84169	0.84169	-0.84169	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>

### Export to CSV

You can now export the contents of the spreadsheet to a CSV file using the **Export CSV** button.



This will only export the nets (or Differential Pairs/Chains) columns used within the Rules Spreadsheet. If you need all rule columns exported, select them in the Edit Columns option to be displayed first. Alternatively, use the **Report Maker** option to write out the rules information.

### Errors shown in the Length Diff field

The rules bar now shows errors in the **Length Diff** field only on items too long or short relative to the matched target item. The default matched target item is the longest. The only exception to this is if the **Match Target** field has been checked to match against a track which is not the longest. See above under **Track Length Match Rule** section for more details about using this rule.

Paired length	Length	Length Diff	Diff To Target	Diff To Shortest	Diff To Longest	Matched Target
30.10843	35.02583	0.84169	-0.84169	0.84169	-0.84169	<input type="checkbox"/>

### All Item Types spreadsheet

When using the **All Item Types** spreadsheet, it now makes it clearer that the **Differential Pairs** and **Chains** have a row for each of the two signal lines by marking the second one with a (2).

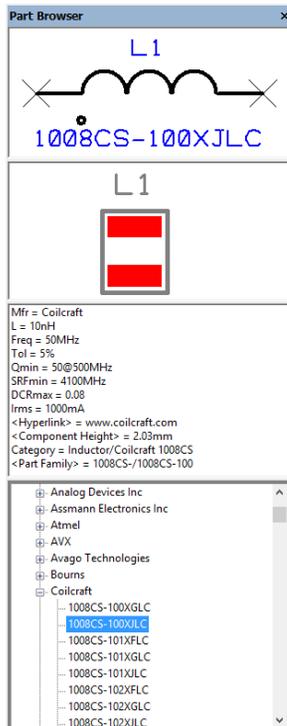
Rules Spreadsheet

All Item Types Edit Columns... Colours... Options... Export CSV...

Matched	Target	Type	Name	Sub Name	Net Name	Net Class	Pad1	Pad2	Ma
<input type="checkbox"/>		Pair Chain	J1-U8				J1.112	U8.D3	
<input type="checkbox"/>				J1.112-RN40	RDQS7		J1.112	RN40.2	
<input type="checkbox"/>				RN40.3-Bran	DQS7		RN40.3	Branch2	
<input type="checkbox"/>				Branch1-U8.	DQS7B		Branch1	U8.D3	
<input type="checkbox"/>			(2)				J1.111	U8.C3	
<input type="checkbox"/>				J1.112-RN40	RDQS7B		J1.111	RN40.1	
<input type="checkbox"/>				RN40.3-Bran	DQS7B		RN40.4	Branch1	
<input type="checkbox"/>				Branch1-U8.	DQS7		Branch2	U8.C3	
<input type="checkbox"/>		Pair Chain	J1-U9				J1.101	U9.C3	
<input type="checkbox"/>			(2)				J1.102	U9.D3	
<input type="checkbox"/>		Pair Chain	J1-U10				J1.101	U10.D3	
<input type="checkbox"/>			(2)				J1.102	U10.C3	

### Part Browser Symbol Previews

The **Part Browser** window has been updated to now include preview windows of the **Schematic** and **PCB** symbols used and the **Attributes** within the selected **Part**.

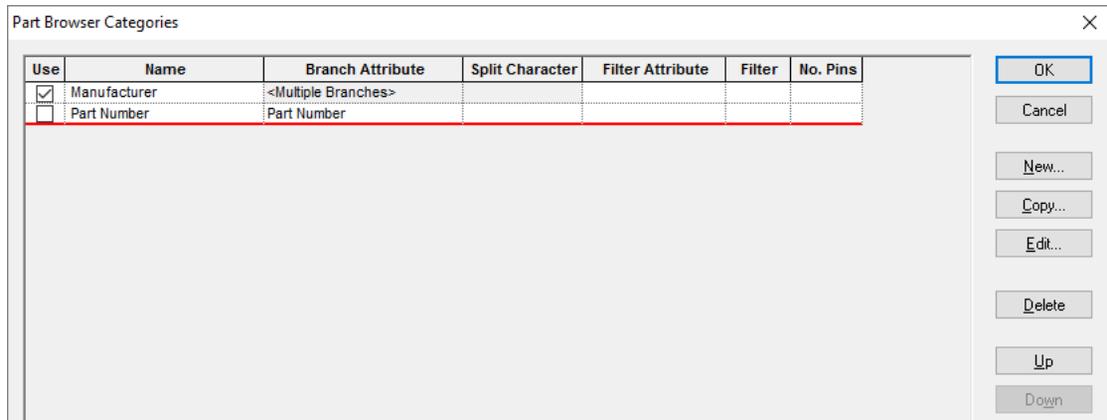


## Part Categories

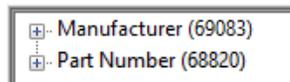
Changes have been made to the Part Categories selector. You can now have multiple Categories that can be saved for use over different designs or between Schematic and PCB for example. The list is saved until Delete is used.

The **Use** button enables you to set up multiple category fields and enable them when required.

**Name** entry has been added, this allows you to add your own name to identify the category search. Because you can now choose multiple categories, the previous use of an attribute name may not accurately now describe the search being performed. For example, you may choose categories based on Manufacturer information but using different Branch Attributes such as Manufacturer, MFR, Vendor etc. and another based purely on a single Part Number field.



Choosing both Manufacturer and Part Number as Category name in the above example will display the names in the browser.

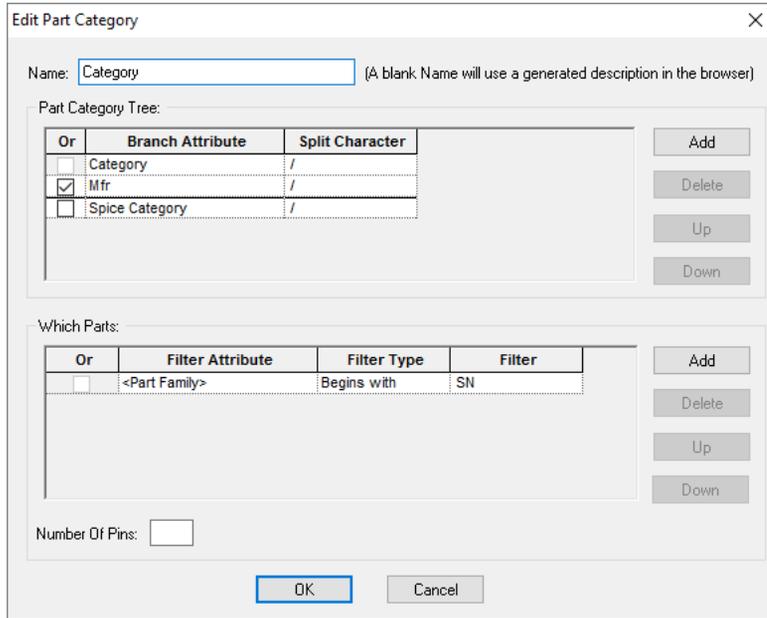


The **Copy** button will create a new category with selected category's branch attributes and filters.

Each category can have a user supplied name, which is shown at the top of the tree in the browser. If left blank it will be generated from the category attributes.

Categories can now have multiple branch attributes and multiple filter attributes, defined using **Edit** button.

Using the **New** or **Edit** buttons, you are presented with the **Edit Part Category** dialog:

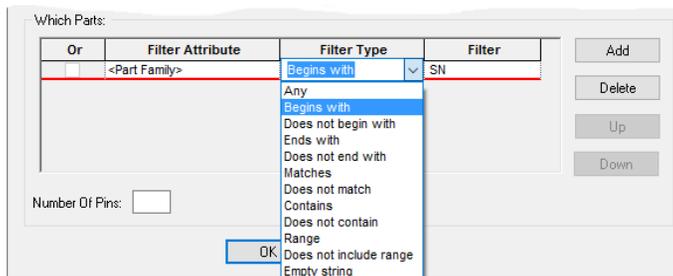


This now allows you to choose multiple **Branch Attributes** by adding them to the **Part Category Tree** list.

Can now choose which of the categories are currently used in the browser using **Use** column.

The **Which Parts:** enables a filtering of **Parts** found. You can add this to further refine Parts presented in the browser. **Filter Attribute** allows you to choose the attribute to create the filter on.

**Filter Type** is the standard selection of filters from which to make your selection refinement.



**Number Of Pins** is an additional filter. For example, you could make a category filter for just 2 pin components.

You can use an **Or** check box when defining filters to specify that one of a set of filters must be satisfied for the part to be included in the tree.

You can define the browser tree branches by using multiple attributes. Each attribute row will define a new branch in the browser tree (or multiple branches if a split character is used) unless the **Or** column is checked. If **Or** is checked the attribute will not be used as a new branch, but will be used instead of the previous branch if a part does not have the previous branch's attribute.

You can also define multiple attribute based filters that all must be satisfied for a part to be included in the browser tree. The grid contains a filter type column (like the part finder) to make it easier to define some of the common filter actions without having to understand the wildcard mechanism.

Substitution of attribute values will be performed when getting an attribute value for a category, but only if the attribute name is included in the library indexes (or in the vault user attributes table if the part is being read from the vault). If the substitute attribute does not exist one of two things will happen depending on the value of the new “Show Unsatisfied Category Substitution Attributes” check box in the Part Browser Categories dialog. If checked the attribute name enclosed in { } brackets is substituted, else a blank string is substituted. Default is unchecked.

## Additional Part Browser Changes

In addition to the inclusion of preview windows and the Attribute window, other changes have been made to the Parts Browser:

### Context menu changes

Within each of the Preview Windows for Schematic symbols and PCB footprints, the context menu has appropriate commands for **View Extents** and **Edit In Library** functions.

### Hyperlink usage in Attribute window

Attributes Window. Shows attributes on selected part, followed by Vault information if from vault. Can double click of line with web reference to follow hyper link to the website. Context menu allows you to hyperlink and copy the selected lines.

### Parts from the Vault

Within the Browser Window, names of Parts that come from a Vault will be colour coded to indicate if they are in the current vault, or were added to a library from the current vault, or were added to a library from a different vault.

### Replace and Reload

If you select a part in the Browser that is in the current design, **Replace** and **Reload** are available from the context menu.

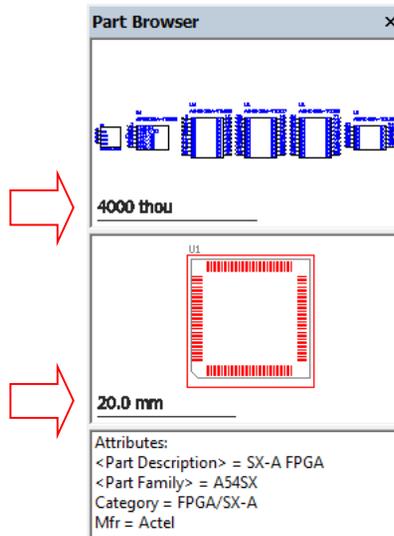
### Cursor key use

You can now use the up/down, left/right cursor keys to traverse the Part Browser tree.

## Preview Scale

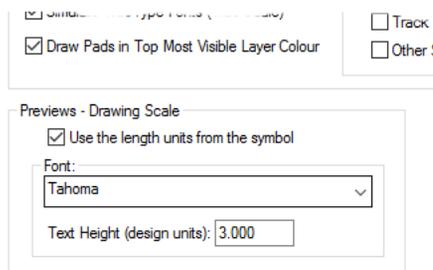
Symbol and Footprint previews when used in options such as Part Browser, Library Manager Part Manager, Component Bin, Insert Component, Insert Signal Reference etc. now show a scale. This is helpful when selecting symbols to ensure they are the correct size when used in the design.

The preview (optionally) shows a scale line and a relative size below the symbol.



For the scale indicator, you can set the font and text size in the **Options** dialog and **Display** tab. **Previews – Drawing Scale** is shown at the bottom of the dialog. The text font and Height can be changed as required.

If you wish to turn the scale off completely, change the **Text Height** to 0.



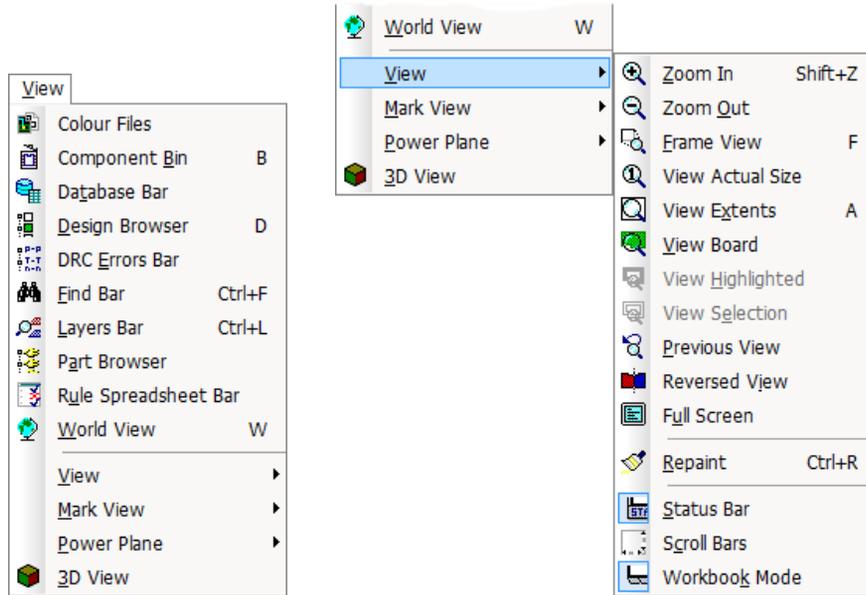
## View Menu Changes

### Personalised Menus and toolbars

Menus now default to 'unfolded', no longer hiding the less commonly used items. This only applies to new installations, existing installations will retain your existing settings. If you wish to change this, use the **Customise** dialog from the **Tools** menu. With this, change the settings on the **Options** page.

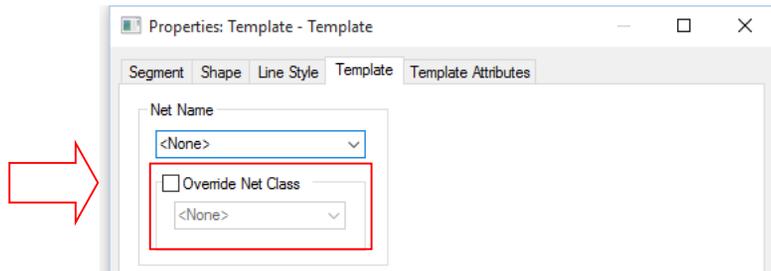
### Changes to View menu

The **View** menu has been re-organised to promote dockable bars to the top of the menu and relegate the various view/zoom commands to a subsidiary menu. With extensive use of mouse and shortcut keys for view and zoom commands, these become less significant on a menu itself.



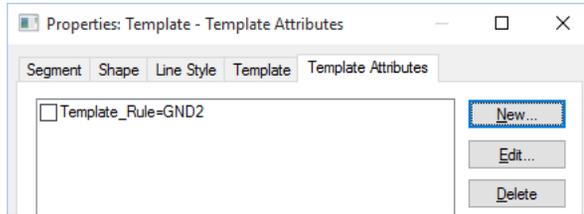
## Changes to Net Class Override in Templates

Previously, **Template Properties** contained an **Override Net Class** check box. This enabled special conditions to be defined for a new **Net Class** and applied to a **Template**. For example, different thermal relief properties.



In Version 9.0 this has been changed so that a Template can contain an attribute that can be used to define **Thermal Rules** (or **Copper Pour** rules). The **Override Net Class** button has been removed. If you have previously used the **Override Net Class** option, the rule defined will now appear as a **Template Attribute** called **<Net Class Name>** with the Net Class you defined.

If you wish to apply a new set of rules to the Template, create an attribute on the **Template Attribute** page. It can be any name provided it matched the one used in the Thermal Rules page. It is advised to use a meaningful name, such as **Template\_Rule**. The attribute name will be the one to match the rule to.

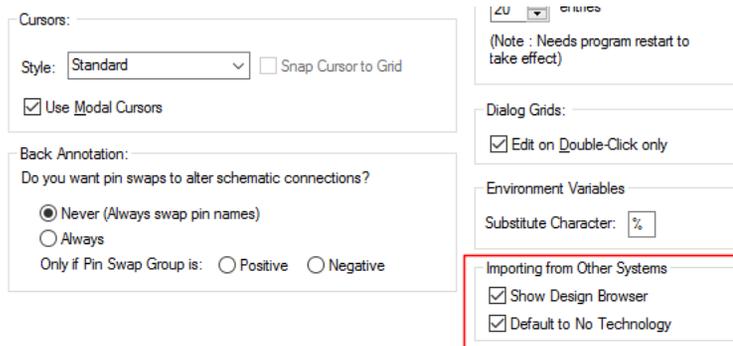


When editing **Thermal** rules, this translates into a set of defined rules:

Attribute Name	Match Value	Applies To	Min Pad Size	Connect Type	Thermal Pad Value				
					Isolation Gap	Spoke Style	First Spoke Angle	Num of Spokes	Min S
Vias	*	<Default>	0.00000	Thermal Pad	0.38100	<Thermal Relief Spokes>	0.0	7	<Unde
<Net Class Name>	GND	Via	0.00000	Thermal Pad	0.25400	LineStyle10.0	0.0	5	<Unde
Template_Rule	GND2	Via	0.00000	Thermal Pad	0.25400	LineStyle10.0	0.0	3	<Unde
Template_Rule	GND2	Surface Mount	0.00000	Thermal Pad	0.25400	LineStyle10.0	0.0	4	<Unde
Template_Rule	GND2	Through Hole	0.00000	Thermal Pad	0.25400	LineStyle10.0	0.0	6	<Unde
Template_Rule	GND2	<Default>	0.00000	Thermal Pad	0.25400	LineStyle10.0	0.0	4	<Unde
<Net Class Name>	HS1	<Default>	0.00000	Thermal Pad	0.38100	Style2	0.0	5	<Unde
<Net Class Name>	RF	<Default>	0.00000	Thermal Pad	0.38100	<Thermal Relief Spokes>	0.0	7	<Unde
<Net Class Name>	*	<Default>	0.00000	Thermal Pad	0.25400	LineStyle10.0	0.0	4	<Unde

## Options For Importing From Other Systems dialog

Two new options have been added for use when importing designs from other systems; **Open Design Browser** and **Default to No Technology**:



### Open Design Browser Option

A new option, **Open Design Browser**, has been added to the **Options** dialog, **General** page. This is used when importing Schematic Designs so that you can see all pages and hierarchy in the design and browse to the pages if required.

## Default to No Technology

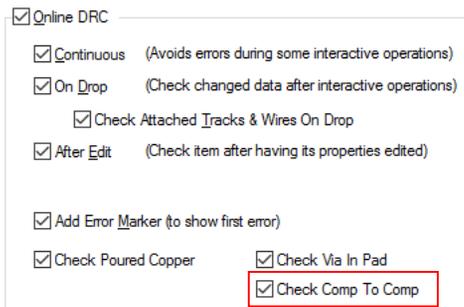
A new option, **Default to No Technology**, has been added to the **Options** dialog, **General** page. When checked, this will default the **Technology** drop down to **None** when you attempt to import a new design.



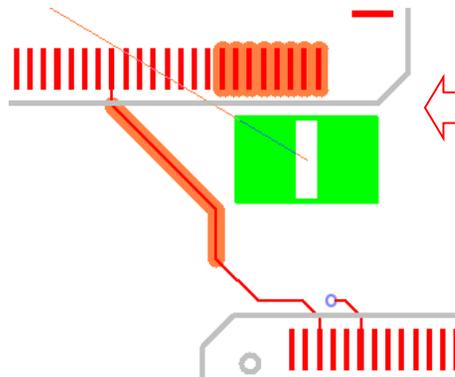
## Show Design Clearance when Moving a Component

When **moving** a **Component**, you can now display the design clearances. During **Component Move**, the context menu will have switches that can be toggled for **Online DRC** and **Display Clearances**. This will work the same as with track editing when the design clearance is activated.

This can be enabled in the **Options** dialog and **Online DRC**, using the **Check Comp to Comp** switch.



When enabled and moving components, it will display like the example below:



The component being dragged here shows **Track Spacing** rules with the **Display Clearances** option enabled.

## Show Design Clearance on Breakouts option

From within the **Options** dialog and **Online DRC**, you can now **Show Design Rule Clearances on Breakouts**.

Add Error Marker (to show first error)  
 Check Poured Copper     Check Via In Pad  
 Check Comp To Comp

Show Design Rule Clearance

On All Items  
 Within Set Distance: 12.70000  
 Within Grid Steps: 4

Show Clearance On Breakouts

## Default Change Layer Preferences

You can now specify the **default** layer to change to when editing tracks. This can be defined in the **Design Settings** dialog and **Track**.

Track Layer:

Change From Layer	To Layer
Top	Inner 2
Inner 2	Inner 5
Ground	Top
Power	Top
Inner 5	Bottom
Bottom	Inner 5

Using **Change Layer** or **Swap Layer** will default to the layer defined in the **Change From Layer** dialog.

Within the **Options** dialog and **Edit Track** page, there is also an option to decide if the default layer should always be used, or the previous layer (same as the previous release). The default setting is for this option to be enabled (as per the previous release).

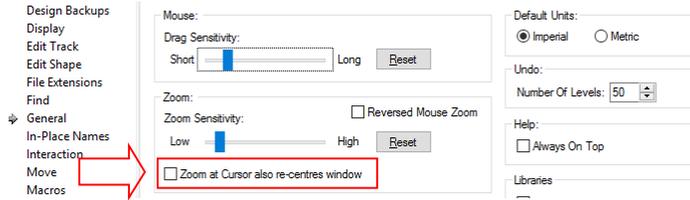
Multi-Screen	<input checked="" type="checkbox"/> Show Connection to Nearest Node	<input type="checkbox"/> Use Style From Picked Track
Online DRC	<input checked="" type="checkbox"/> Show Dynamic Connections Only	<input type="checkbox"/> Obstacle Avoid
Resolve Net Names	<input type="checkbox"/> Optimise After Edit	<input type="checkbox"/> Auto Finish
Track Length Limits		<input checked="" type="checkbox"/> Change Layer - Return To Previous Layer
Warnings		

## Zoom at Cursor feature

Within the **Options** dialog and **General** page, there is a new option to **Zoom at Cursor also re-centres window**.

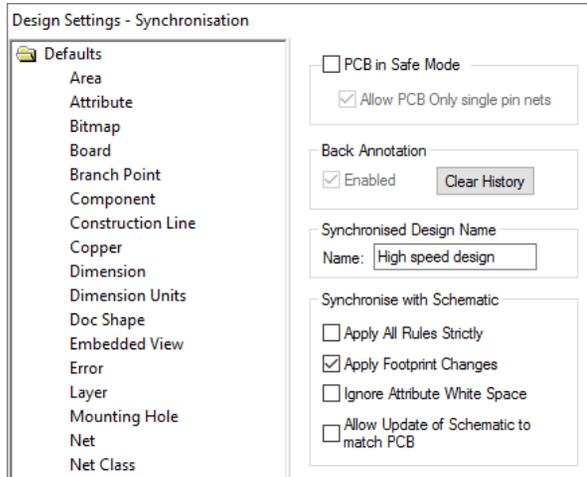
With this option enabled, Zoom In/Out at Cursor will re-centre the view window so the position of the cursor and the design item under it are shown in the centre of the window. By disabling this option,

the behaviour of Zoom In/Out at Cursor will be changed so that after zooming the position of the cursor and the design item under it remain unchanged in the view window.

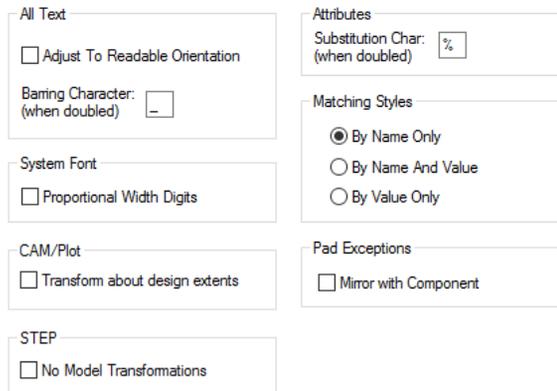


## Design Settings dialog Changes

Settings relating to design translation and synchronisation have been moved from the **General** tab to a new **Synchronisation** tab.

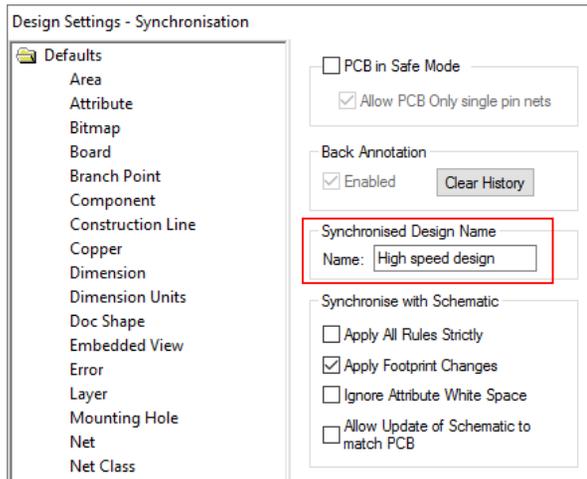


The **General** tab has been rationalised in the process.



### Design Settings – Set Synchronise Design Name

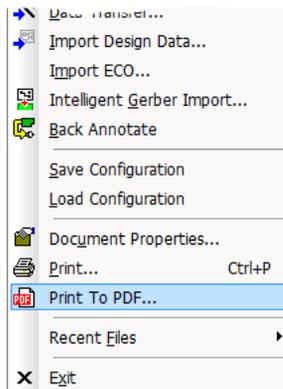
You can now specify a different design name to be used for **Synchronise Design**. For example, if you update the PCB to be revision B, you can still reference the original named Schematic by specifying its name in the **Design Settings** dialog and **Synchronisation** page.



### Changes to Print dialog

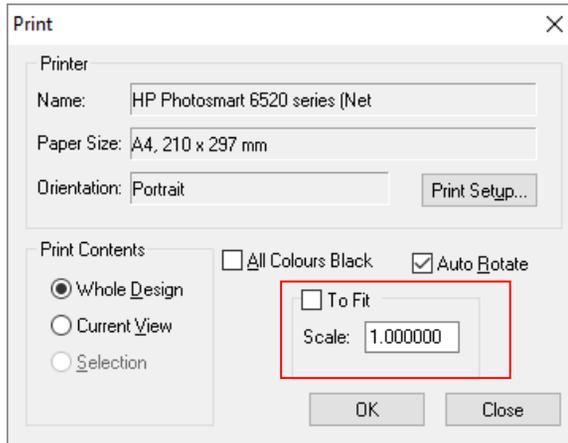
#### Print to PDF

From the **File** menu, there is now a **Print to PDF** option. This will provide a 'quick' way to generate a PDF file of your design. It prints the current page or currently displayed layers to a PDF file using the default PDF device options defined in the **CAM Plot** dialog.



## Print Scale

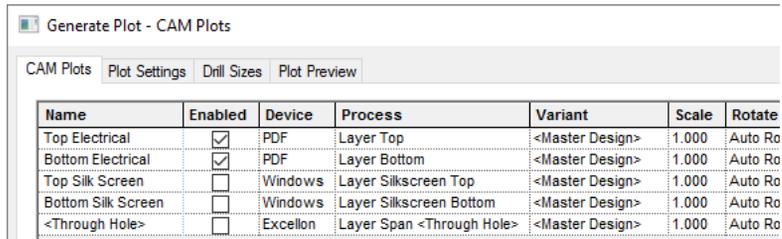
You can now specify a scale in the print option.



## CAM Plot Changes

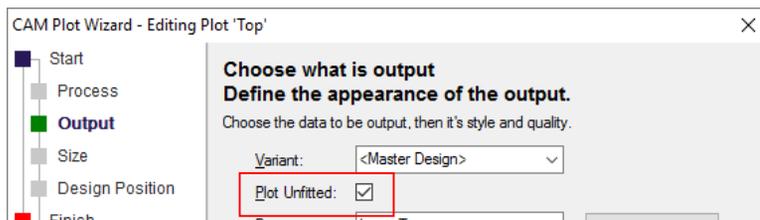
### Change to Tab Order

Within the **CAM Plots** dialog, the order of the tabs has been changed to be more logical.



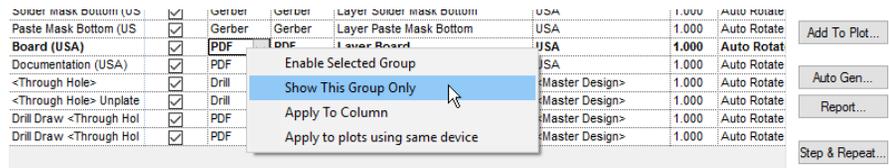
### Plot Unfitted Variants option in PCB

From within the **CAM Plot** option, using the **CAM Plot Wizard**, you can now force unfitted components to be plotted on a plot. On the **Output** page of the **Wizard**, select the **Plot Unfitted** check box if you wish them plotted.



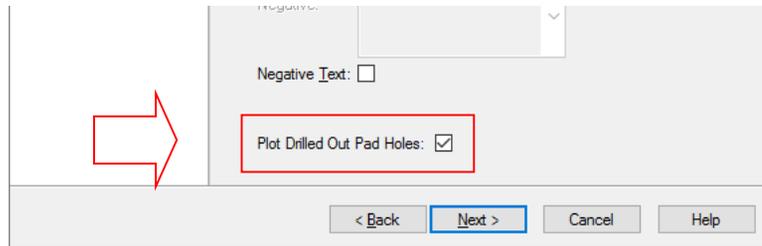
### Show Group option

From within the **CAM Plot** dialog, you can now just show the plots in a single group. By right clicking in a cell in the **Groups** column, you can select **Show This Group Only** using the context menu to show the selected group. Once enabled, right clicking again in the cell will enable **Show All Groups** to reveal all the other groups.



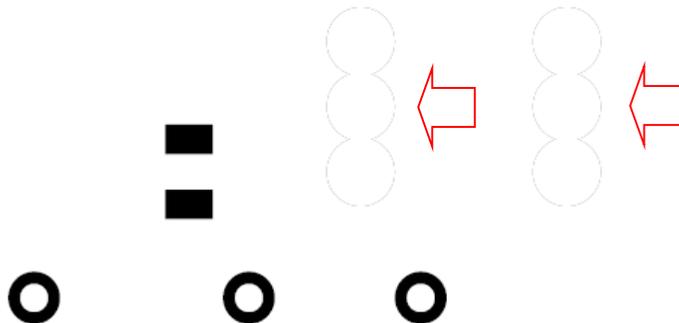
### Plotting Drilled Out Pad Holes

For drill holes which completely drill out the pad, there is now an option within the CAM Plot wizard on the Output page to opt whether to plot (or not) these holes.



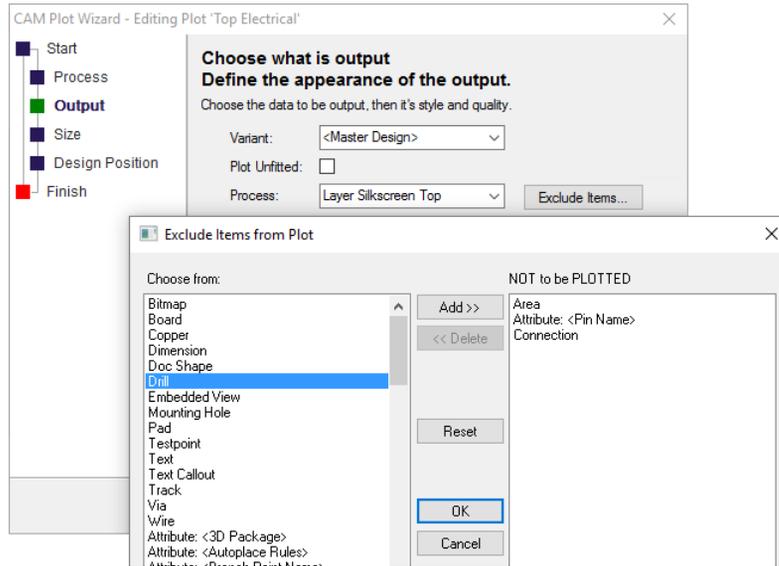
With this option selected, the drill is not plotted at all, nothing will be shown on the plot indicating a drilled out hole.

With this option not selected (the default selection), the plot will show the edge of the drill hole as shown below:



## Exclude Drill Holes from Plot

You can now exclude drill holes from a plot, this overrides any **Layer Class** setting. Using the **CAM Plot Wizard**, **Output** page, on the **Exclude Items** page, select **Drill** from the exclude list.

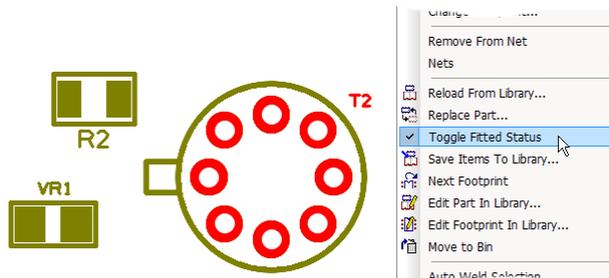


## Gerber x2 Output Update

To provide further support for the Gerber X2 format, the TF.GenerationSoftware X2 command has been added. This doesn't directly benefit you as the user but will benefit your manufacturer.

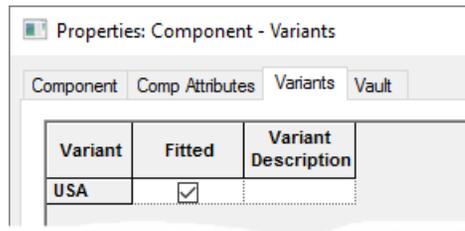
## Toggle Fitted Status of Component Variants

When in the current **Variant**, you can quickly toggle the **Fitted** status of selected components using a new command. Select the **Toggle Fitted Status** option from the **context** menu to achieve this.



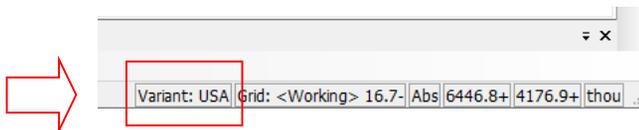
### Properties – Change Multiple Components Fitted Status

In the **Properties** dialog for multiple selected **components**, you can now set the **Fitted** status at the same time.



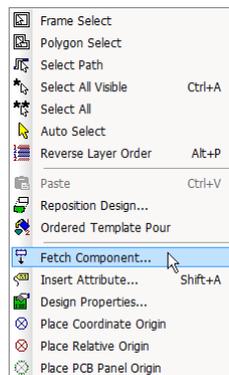
### Current Variant shown on Status Bar

The current variant is now displayed on the **Status bar**, double clicking it allows you to change the current variant using the **Variant Manager**.

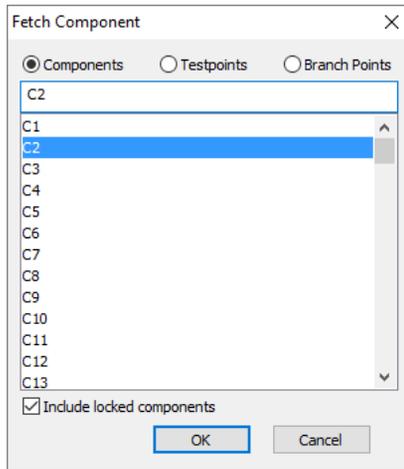


### Fetch Component

You can fetch a component to the cursor by selecting it from a list. This command is available in select mode when you have nothing selected in the design.



Once selected, you are presented with a list from which to make your choice:



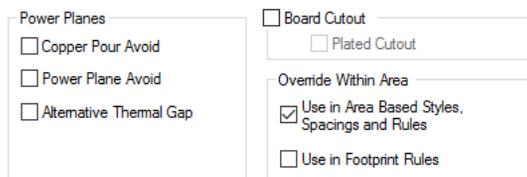
You can select an item from the list and press **OK** or you can double click the item to fetch it.

You can also use this option to fetch **Testpoints** and **Branch Points** if there are any in the design. If none are present in the design, the radio buttons are not displayed.

A check box on this dialog allows you to say whether the list includes **locked** items or not.

## Properties - Areas

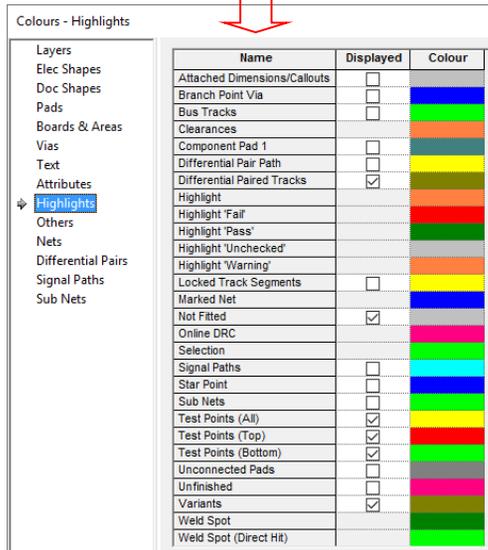
Within **Properties of Areas**, the check box within the **Override Within Area** section has been renamed to **Use in Area based Styles, Spacings and Rules**. This is also available in the **Area Keep In/Out** option dialog.



## Colour Dialog Changes

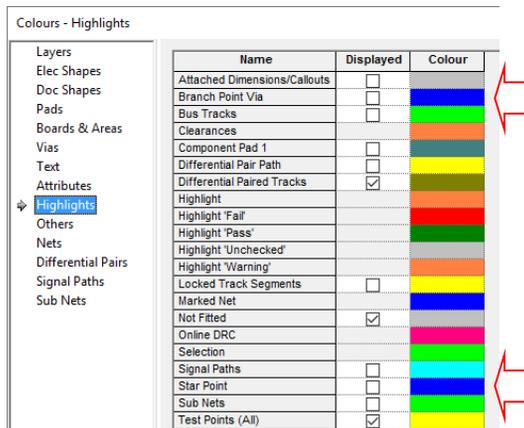
Highlights & Colours pages

Within the **Colours** dialog, the **Highlights** and **Others** pages are now sorted alphabetically.



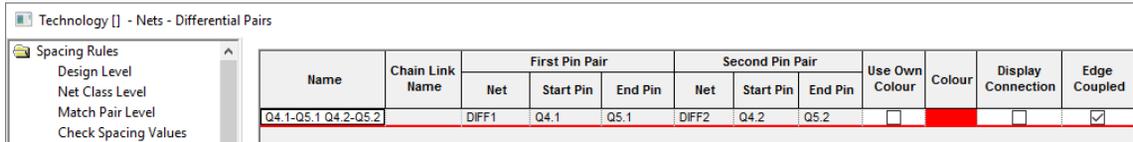
Colours added for Branch Point (Vias) and Star Points

Highlight colours have been added for **Branch Point (Vias)** and **Star Points**.



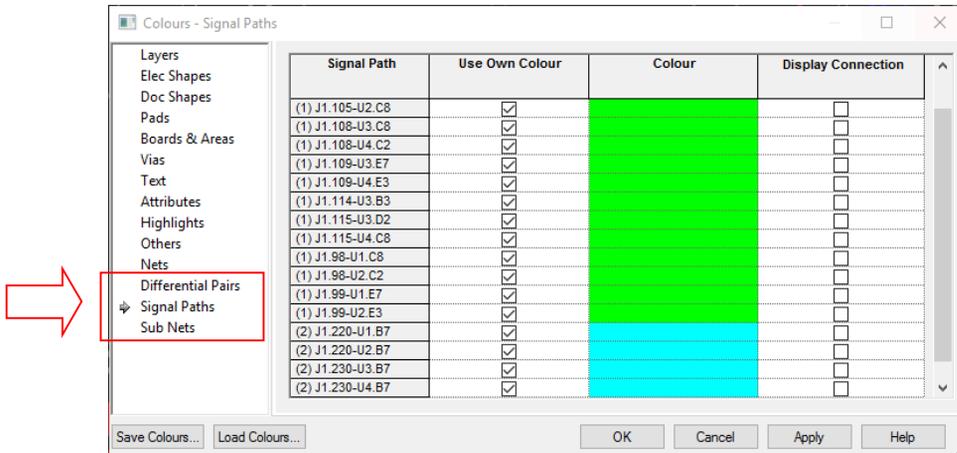
## Connection Colours in Technology dialog

Technology dialogs have been updated with the **Display Connection** column for use with net items (Net Names, Signal Paths, Sub Nets, Differential Pairs and Differential Pair Chains). Use this to selectively switch on or off the connection for that net item.



## New Tabs for Differential Pairs, Signal Paths and Sub Nets

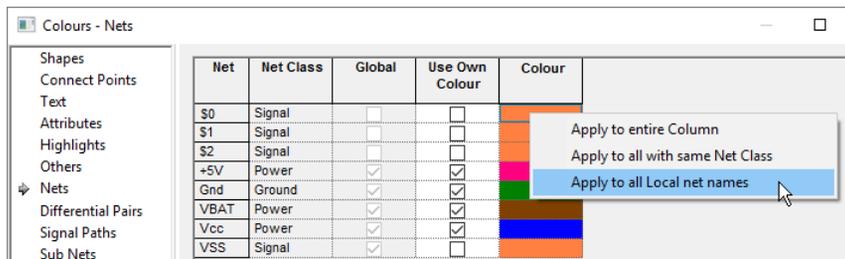
You can now set explicit colours for **Differential Pairs**, **Signal Paths** and **Sub Nets** in the **Colours** dialog. Colours for these items can also be set within the **Technology** dialog under their own items.

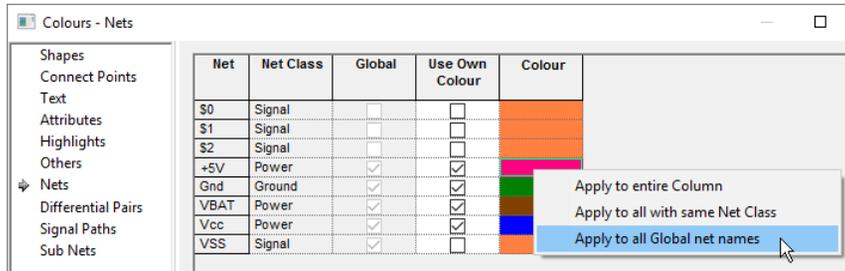


## Selectively set the colour of a net

When changing Colour selections in a Schematic design, using the context menu selected on a colour, you can apply the colour of a net to all other local/global nets.

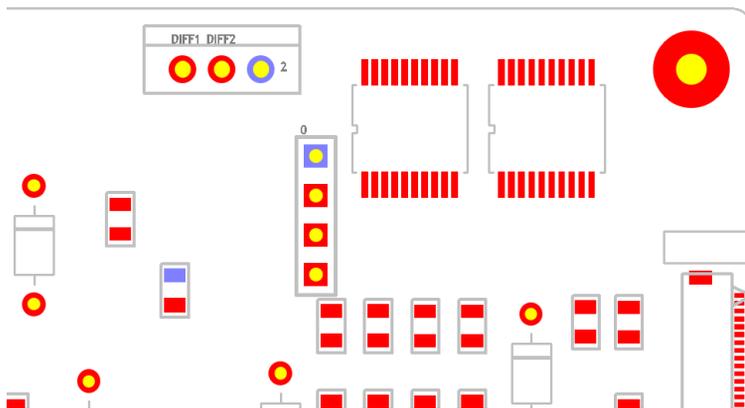
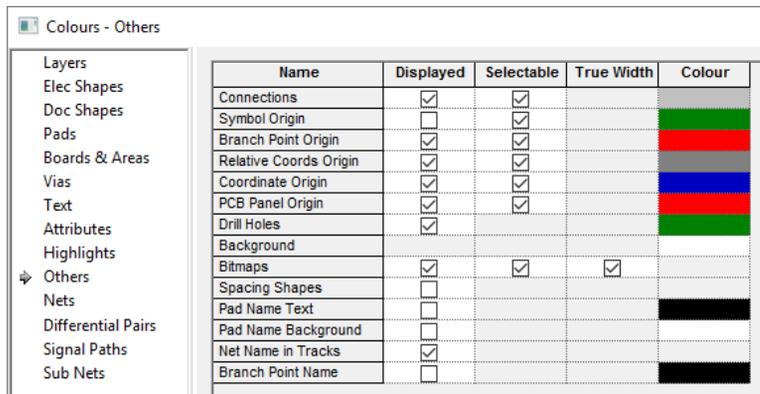
**Apply to all local net names** and **Apply to all Global net names** will enable you to make the colour selection.





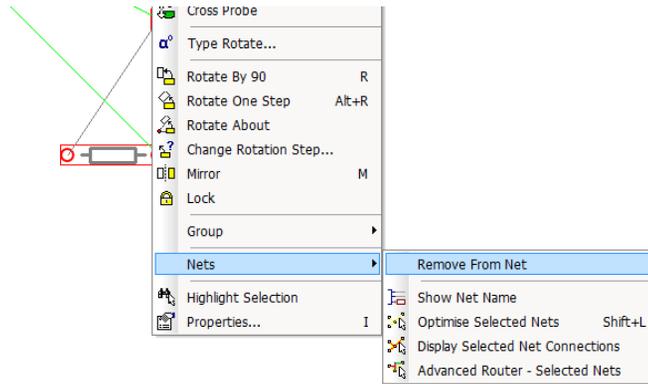
### Set the colour of Drill Holes in PCB

You can now set the explicit colour of drill holes in the PCB design (in addition to just using the background colour). Select the **Drill** option and **colour** from the **Colours** dialog and **Others** page.



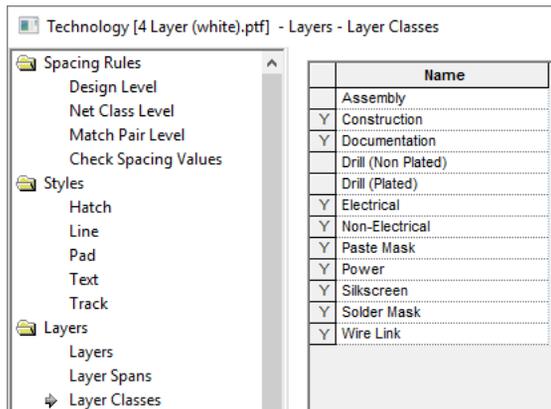
## Remove From Net for Multiple-selected Tracks/Connections or Components

The **Remove From Net** option is now available if multiple tracks or connections are selected, or if multiple other items selected, such as Components. With multiple items selected, the context menu displays the **Remove From Net** option.



## Layer Classes now in alphanumeric name order

**Layer Classes** in the **Technology** dialog are now listed in alphanumeric order to make them more logical and easier to find.

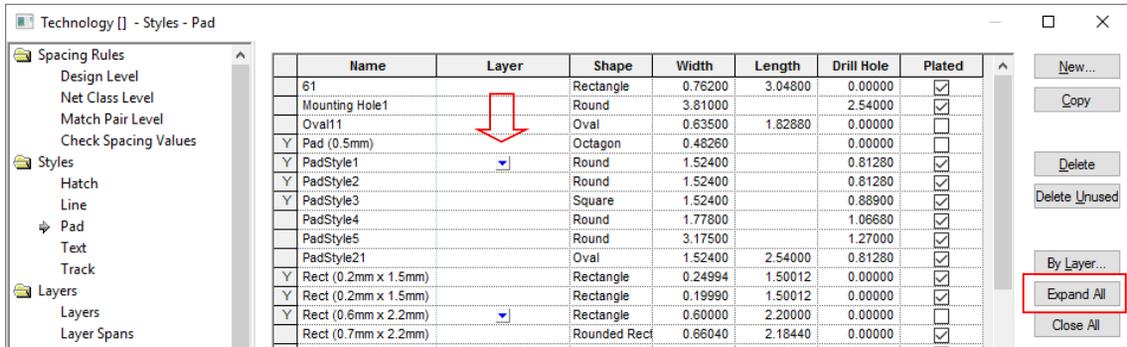


## Pad Styles - 'Expand All' Exceptions

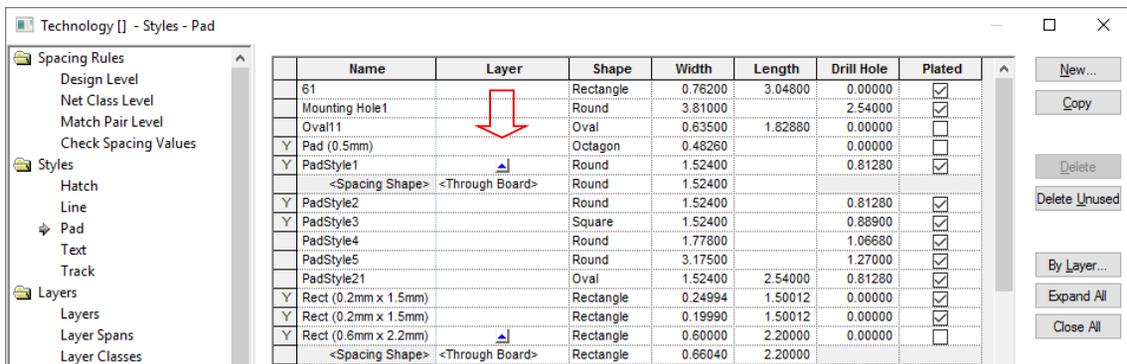
A new **Expand All** button on the **Pad Styles** page will unfold all **By Layer** exceptions in one click. This is an alternative to having to go through and click each blue triangle to expand its view. Once expanded, each blue triangle can be clicked to close it or you can use the **Close All** button to close all **Pad Style By Layer** exceptions.

The grid will look like this folded up until the **Expand All** button is pressed:

## 84 Pulsonix Version 9.0 Update Notes



Expanded, the **Pad Styles** grid now looks like this showing each of the **By Layer** exceptions:



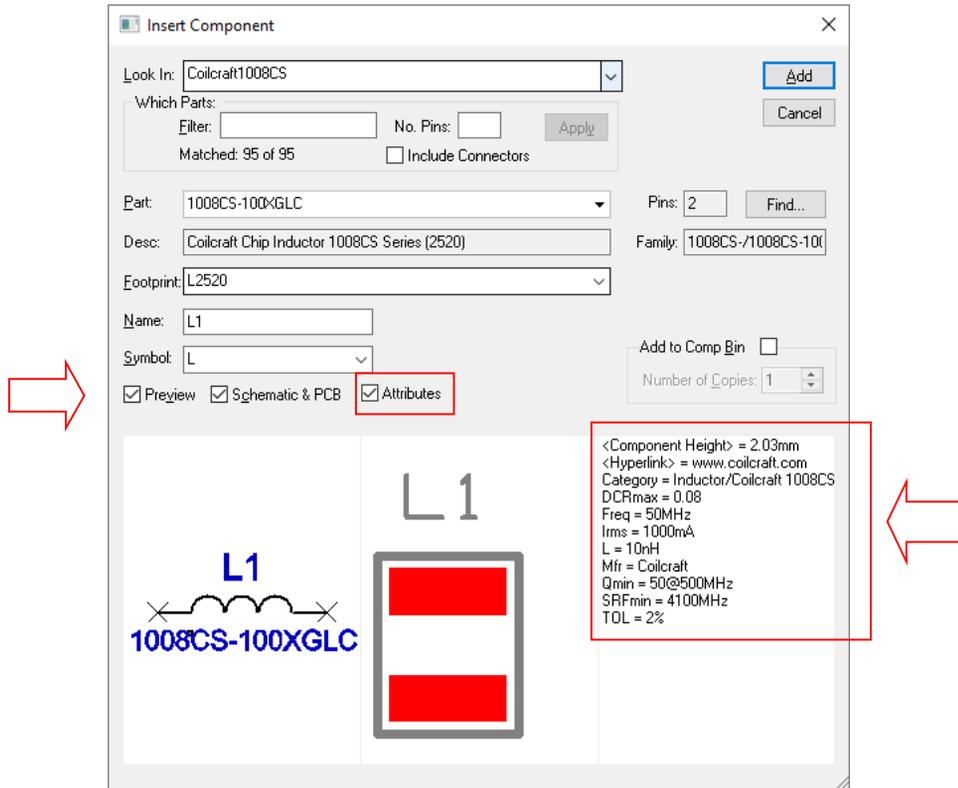
## Changes to the Design Browser

Some minor features have been improved within the **Design Browser** to ensure better behaviour consistency. Any action that results in a change of the active schematic page view in the main application window will result in the appropriate branch being expanded to show that page in the browser. For example, when swapping pages using the design tabs, the browser will result in the appropriate branch being expanded to show that page in the browser.

Also, when the Design Browser is in **Auto-Hide** mode, whenever it is displayed it will only show the branches expanded as they were when it was last hidden, except for any changes that have happened while it was in a hidden state. If the active Schematic page view in the main application window has changed, the appropriate branch will be expanded to show that page.

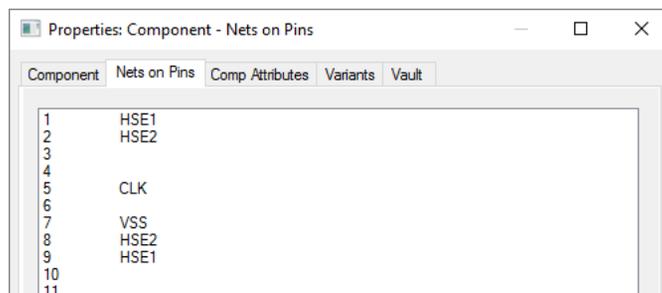
## Displayed Attributes on Insert Component dialog

The **Insert Component** dialog now has a check box alongside the **Symbol/Footprint Previews** that lists the **Attributes** of the selected item. This will help you confirm that the chosen part is the right one before it is added to the design.



## Nets on Pins in Component Properties

The **Properties** dialog for **Components** now has an extra tab (**Nets on Pins**) that lists the net on each pin on the selected component. This is useful when verifying pin number connectivity against net name usage.



### Net Name in Pad option

The **Pad Name in Pad** feature has been extended to also show the net name for pads that are connected. It will be enabled by default but may be disabled via a new option on the **In-Place Names** tab of the **Options** dialog and deselecting the **Additionally Show Net Name in Pad** option.

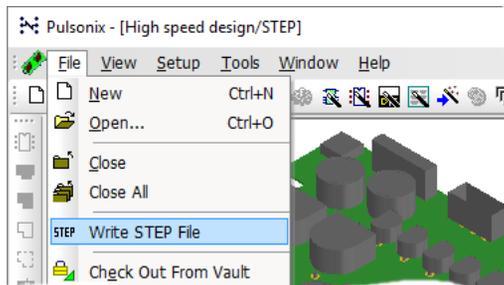


This option also now has an option to confine the names to just the extents of the pad on screen. Check the **Truncate Names to Pad Size** to enable this option.



### Write STEP file from Preview window

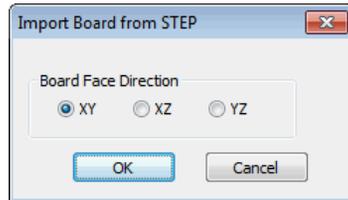
Within the **STEP Preview** window, on the **File** menu a **Write STEP File** command has been added. This will write a STEP file for the view which means that the data does not have to be regenerated before writing making it much faster. Once the preview has been verified, you can simply commit to the STEP file by using this option.



## Board Face Direction in STEP File Import

The ability to choose which plane to examine when looking for board outline in imported STEP file has been added.

From the **File** menu, choosing **Import Design Data** and a valid STEP file will display this dialog:

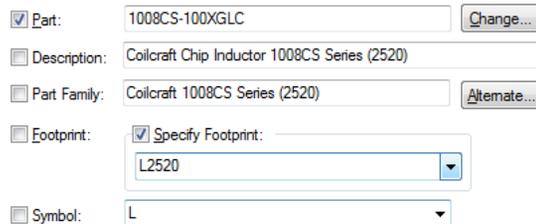


By default the system will try to find board shapes on XY plane. If the shapes have been defined in another orientation you can choose the wanted plane from the dialog.

## Specifying Footprints in the Schematic

In schematics, you can now choose if you want to specify the footprint for use in the PCB design. If the footprint is subsequently changed in the PCB, **Synchronise Designs** will force a change back to that used (and specified) in the Schematic.

Use the **Specify Footprint** check box in the **Component Properties** dialog to make a specific choice.

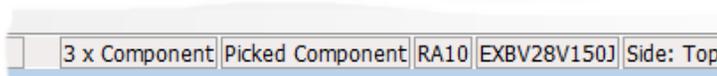


In Previous versions, the dialog stated **Default** Footprint, this really meant *unspecified*, but now you can choose not to specify (which is the default).

## Status Bar Improvements

There is now additional information and fields that have been added to **Status Bar**:

- When selecting more than one of same type, the **Status Bar** now shows N x type (e.g. **3 x Component**) instead of simply saying **Multiple items** as it did previously.



- When selecting an item that has a position (e.g. Component, Pad, Via) the XY position is shown.

Side: Top Pos: 51.9-,4.9 Grid: <W

- When selecting an item that has an overall size (e.g. shape), the XY extents are shown.

Area Style: LineStyle1 Size: 66.6 x 29.0 Layer: <Top

### Find Bar Improvements

**Differential Pairs, Branch Points Signal Paths** and **Sub Nets** can now be found using the **Find** bar.



### Find Bar Changes

You can now find **Templates, Copper, Component Pads** and **Mounting Holes** that are not on a net. To use this, select <No Net> in the net names list and use the **First/Next** buttons to find the items that are not on a net.



## Report Maker Changes

New commands within the **Report Maker** option have been added for the following:

### Track Length Rules

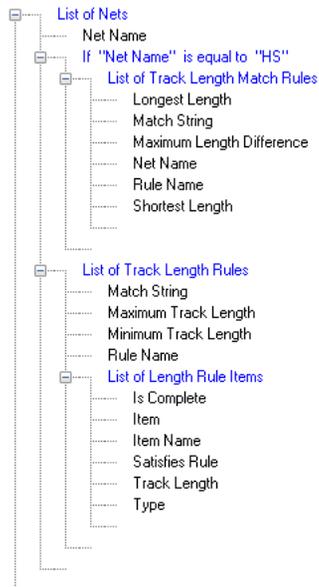
For reporting **Track Length Match Rules** and **Track Length Rules**. These can report the rule details and all items that use the rule and their lengths and length differences. This will replace the **Critical Nets Report** under **Standard Reports**.

Now accept the following commands:

```
List of Track Length Match Rules
  Longest Length
  Match String
  Etc.

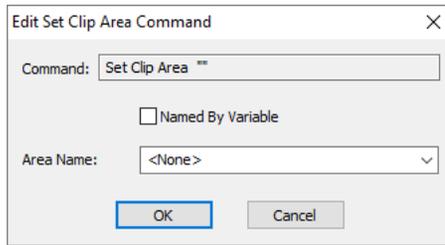
List of Track Length Match Rules
  Match String
  Maximum Track Length
  Etc.
```

**Track Length Rules** and **Track Length Match Rules** examples:



### Setting Clip Area

There is a new command for **setting a clip area**. All physical design items will only be reported if part of them is within the area. This allows you to report all components within a named area. The clip area will be used by any nested reports.



When this command is selected, the dialog will ask you to choose the **Area Name** from which to set the clip area to. If the **Named By Variable** check box is selected, the Area Name drop down list will be empty and the selection will be made by a **Variable** name.

This command would be used at the beginning of a report sequence to define the area to be clipped. Multiple Set Clip Area commands can be used within a report with each one resetting the area for the next sequence of commands. For example, you may have three PCBs in one PCB design for which to export Pick & Place information for. Adding and naming an area around each PCB outline will allow each PCBs information to be exported.

A **Set Clip Area** example:

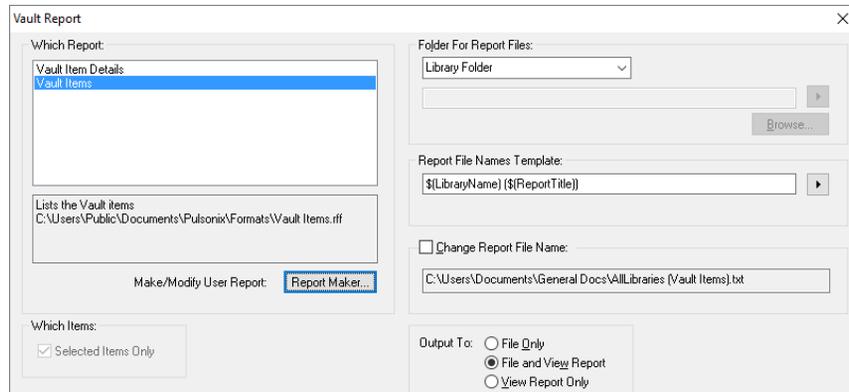


### Clipped Areas in CAM Plot

When using a format file in the **CAM Plot** option that has **clip to named area** defined, this will set up the clip area when generating the report. If in the **CAM Plot Wizard**, you have the **Crop Output To** enabled and an area defined, this will overwrite the format file **Clip Area** in the format file.

### Report Maker Vault Reports

You can now report items in the vault from the **Vault Manager** using the **Report Maker**. Check the Vault context check box in a formatted report to make a report appear in the list in the vault manager reports dialog. See me for details of commands.



## Technology File Changes

### Changes to existing Technology files

All existing Technology files (SCM and PCB) have been enhanced to add new Styles to accommodate new functionality in version 9.0.

## Library Changes

As part of version 9.0, all library contents supplied have been updated. If you have used or modified any of the previous MasterLibraries selection, you will need to ensure these are backed up if overwriting your library location with the new version libraries.

---

*Note: it is always recommended that any contents used from the MasterLibraries are copied into your own library in another location. We also recommend that any used libraries are backed up periodically, i.e. every night.*

---

## Library Consolidation

Pulsonix Parts libraries have been consolidated, the following changes have been made:

Atmel8051 + AtmelAm + AtmelATSam7 + AtmelAVR into Atmel

Coilcraft1008CS + CoilcraftLPS4018 into a new CoilCraft library

Fairchildfdc2512 + Fairchild1 + FairchildACEX into Fairchild

Infineontc116x renamed to Infineon

LT2 into LT

Maxim8051 + Maximmp into Maxim6

Microchip2 into Microchip

Miscconn2 into Miscconn

TI TMS320 dsp renamed to TI2

MSP430 + CSD8 into TI2

NXP-LPC32x0 renamed to NXP

Philips80C51 + PhilipsARM + PhilipsP89 + PhilipsXA into NXP

RenesasR8C renamed to Renesas

SonyCX renamed to Sony

ST-arm7 renamed to ST2

TMS470 into TI2

ToshibaTMP86 renamed to Toshiba

New library 'Misc'

Intel8051 renamed to Intel

MC68HC into NXP

### LT Spice Libraries Added

LTSpice.ssl and LTSpice.pal have been added to support the LT Spice netlist export from Pulsonix Schematics.

# Pulsonix Vault

This is a very brief introduction to the new Pulsonix Vault. The Vault has its own full product Users Guide which can be found from within Pulsonix on the **Help** menu and **Online Manuals> Pulsonix Vault Users Guide.pdf** or within the online Help system at any time.

## What is the Vault?

Pulsonix Vault is a data management system that can help you organise and manage your Pulsonix data in a safe and secure way.

In its simplest form it could be viewed as a 'managed library', holding Parts, Footprints and other data in the same way as your existing Pulsonix libraries. However, it is much more than that, as it can hold design files as well as library data, it controls how they are updated and who can update them, understands the relationships between items, and maintains a revision history of each item. Pulsonix Vault can help you organise, manage and track the creation and modification of your important Pulsonix data, at the same time making that data available where it is needed across your organisation.

Pulsonix Vault consists of two main elements, a server and a client (or indeed many clients). Additional elements are used to assist with tasks such as installation, configuration and backup.

## Do I need to use the Vault?

You do not have to use the Vault. If for example you wish to use native Pulsonix libraries for your Part and other library data, and the normal Windows file store for your design files, then that is fine. There is even a single checkbox on the Options dialog that will hide away or turn off the Vault features and commands in the user interface.

You may also decide not to use the Vault because your company already has a data or document management system in place. There is probably no point confusing the issue by trying to use two different systems to manage your Pulsonix data. If you are unsure about this, we can advise you.

However, even for a single user who does not share their Pulsonix data with anyone else, the Vault offers data management facilities that should help you maintain your library and design data over time.

## System Requirements

The Vault uses an industry-standard Postgres database to store its information. This database will run on any Windows system that is supported for use with the main Pulsonix application, although you should consider the information under 'Deployment Planning' below before deciding where to install the server.

Connection to the database from the Pulsonix 'client' application is via an ODBC connection, using a driver that will be installed by the Pulsonix setup program if it is not already there on your system.

To store the actual data files in the Vault, access to a shared network folder is required. This needs to be a folder that is accessible with both read and write permission directly over your Windows network.

### Deployment planning

Before installing the Vault database server, there are several things you should consider to ensure you achieve an optimal experience when using the Vault.

The Postgres database used for the Vault server has a long history of reliable operation across a wide range of platforms, so the choice is more about the ability of the selected Windows platform to service the demands of the database server and the clients who wish to use it.

If you expect to have a small number of connected clients, making occasional demands on the Vault to provide data, then it is feasible to install the server on a ‘workstation’ edition of Windows, provided you can satisfy the requirements of folder sharing for the Data Files (see below).

If instead you are contemplating large numbers of connected clients making heavy use of the Vault data, or checking many files in and out, then it is wise to consider installing on a Server edition of Windows such as Server 2012.

Similarly, the underlying hardware choice largely depends on how heavy you expect your demand to be. A server that is configured without sufficient memory to cope with the load can cause the response of the Vault operation to be degraded.

Network speed should not be a significant factor, as it will not be any different from using a network server for file storage. Storing files – even large design files – via the Vault is no more time-consuming than saving them to a network folder via the normal File Save commands. However, if you have been used to storing all your files locally and are now moving to a server-based Vault configuration, it is quite possible that you may see some increase in file read and save times due to network latency or bandwidth issues.

### Key Concepts of the Vault

#### What can be stored in the Vault?

The Vault can hold the following types of ‘library’ data: Parts, Footprints, Schematic Symbols, and Schematic and PCB Doc Symbols.

It can also hold the following types of file-based data: Schematic and PCB Designs, Schematic and PCB Technology Files, Schematic and PCB Profiles, Schematic Block Files and PCB Panels.

The Vault is designed to provide you with a secure environment for managing your ‘published’ data. You may still use locally-stored design files and native libraries for developing data, and release your completed items to the Vault when you are ready to do so.

#### What is an ‘item’?

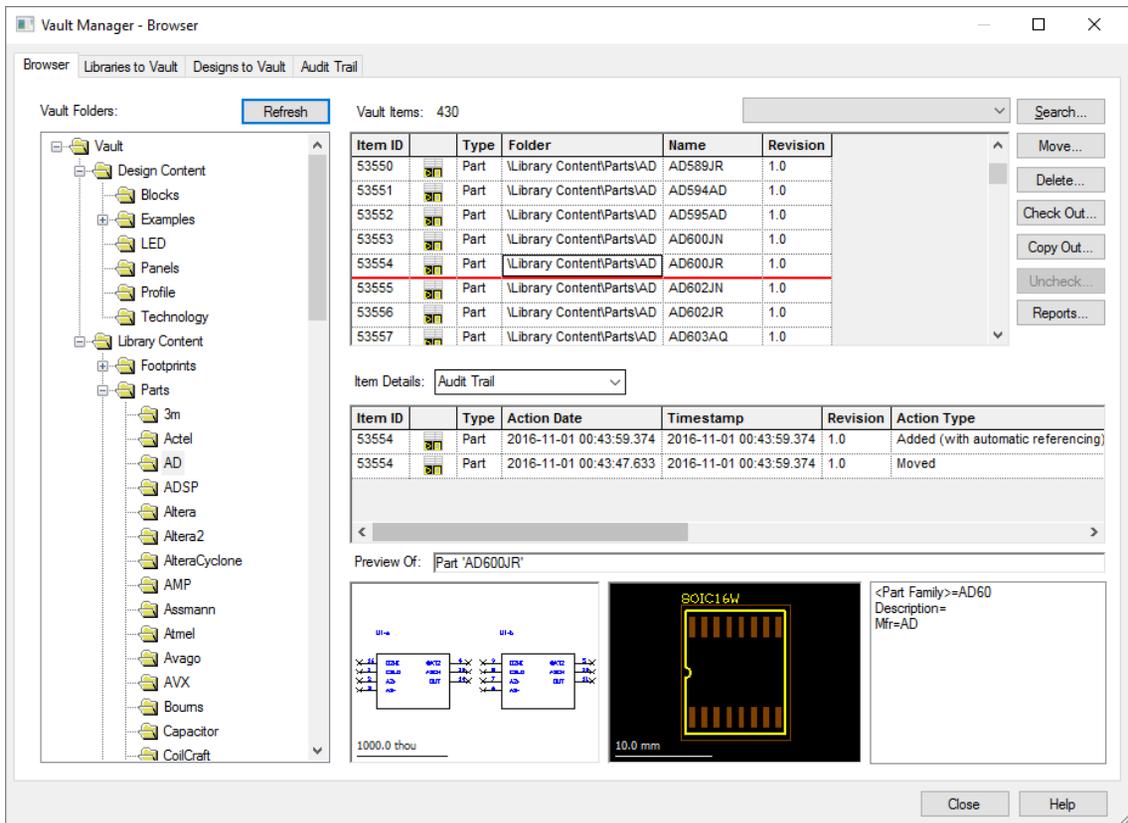
Each item that is placed into the Vault is given a unique item identifier, or “IID”. Each time that item is saved to the Vault, a new version of the item is created, with this latest version having a new version timestamp or “Version”. This is distinct from the “Revision”, which we will hear more about in a moment. In this way, the Vault contains a full history of all revisions of an item.

The IID combined with the Version uniquely identifies a specific version of an item. In this way a Part for example can know which exact version of a Footprint it is to use. Not just any version of the Footprint, only the specific version that is indicated by the IID and Version listed in the information held along with the Footprint name inside the Part.

## Vault Tools

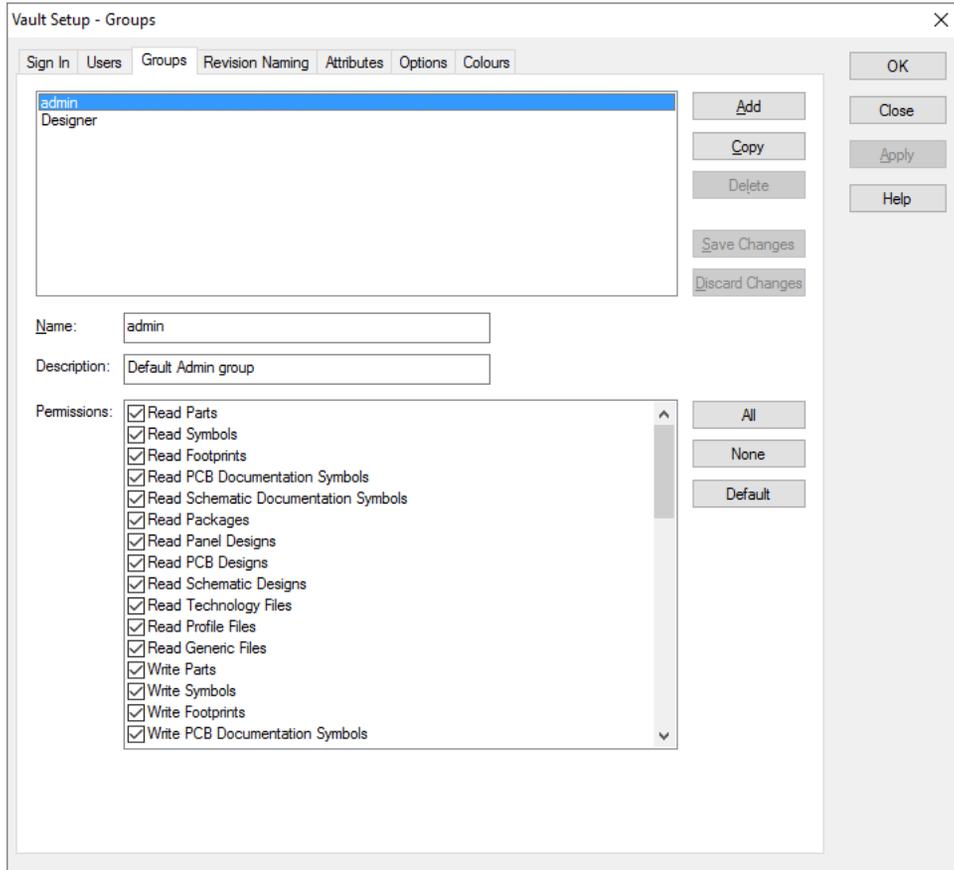
### The Vault Manager dialog

The **Vault Manager** dialog is accessed from the **File** menu or the **Vault** toolbar. It is the main interface between Pulsonix and the connected Vault. From here you can browse the Vault contents, examine the revision history of items, search for items, and carry out other management tasks.



### The Vault Admin Tool

From the **Setup** menu, you can access the **Vault** admin dialog. This allows you to administer and manage the Vault system.



The Vault has its own full product Users Guide which can be found from within Pulsonix on the **Help** menu and **Online Manuals > Pulsonix Vault Users Guide.pdf** or within the online Help system at any time.