PULSONIX

Interactive High Speed Option

Users Guide

Copyright Notice

Copyright © WestDev Ltd. 2001-2021
Pulsonix is a Trademark of WestDev Ltd. All rights reserved. E&OE

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

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

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

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

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

Microsoft, Windows, 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: 26/01/21 Pulsonix iss 9

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

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

Contents

CONTENTS 3
CHAPTER 1. TECHNOLOGY SETUP 6
Technology File and Rules Management 6
Rules & Attribute Windows 6
Technology Layers - Reflect Layers 7
Import & Exporting Technology Rules 8
Import IPC-2581 Layer Stackup into Layers dialog 11
Import/Export CSV into Layers dialog 13
CHAPTER 2. RULES SPREADSHEET 15
Rules Spreadsheet 15
Editing Spreadsheet Columns 16
Rules Spreadsheet Options 16
Colours Used in the Spreadsheet 17
Highlighting Important Items 19
CHAPTER 3. NETS AND RULES FOR THE HIGH SPEED OPTION 21
Nets Items 21
Rules 22
Creating and applying Rules 23
Wildcard Wizard 25
Signal Paths 26
What is a Signal Path? 26
Creating Signal Paths 27
Interactively Creating Signal Paths 30
Sub Nets 30
Overview 30
Creating Sub Nets 31
Connect In Pin Order 33
Interactively Creating Sub Nets 34
Branch Points 34
Overview 34
CHAPTER 4. DIFFERENTIAL PAIR ROUTING 38
Overview of Differential Pairs 38
Defining Differential Pairs 39
Using the Differential Pairs dialog 45
Differential Pair Attributes 46
Differential Pair Gap 46
Differential Pair Skew 47
Pair to Pair Match Lengths 48
Defining Differential Pair Track Styles 48
Editing Differential Pairs into Areas 49
Using Edge Coupled or Broadside Differential Pairs 50
Use copy of existing Differential Pair Via Pattern 52
Legal Completion Path for Differential Pairs 54
Routing a Differential Pair 54
Changing Differential Pair Layers 57
Alternative method for starting Differential Pair Routing 59
Differential Pair Routing Functionality 59

Differential Pair Routing – Fillet Mode 60
Mirror Differential Pair routing from Angled Components 61
Acid trap removal using Minimum Allowed Gap Spacing Rule 62
Defining Differential Pair Colours 63
Differential Pair Routing Options 63
Differential Pair Chains 65
Why Diff Pair Chains? 65
CHAPTER 5. LENGTH BASED RULES 69
Length Based Rules 69
Interactive Net Length Indicators 69
Options to Display Track Length Indicators 69
Track Length Rule 71
Track Length Match Rule 75
Layer Change Length Rule 76
Extra Length Rule through Pin Package Attribute 77
CHAPTER 6. ADDITIONAL RULES 79
Track Parallel Segments Rule 79
Necked Length Rule 80
Track Impedance Rules 80
CHAPTER 7. SERPENTINE ROUTING 81
Fixed Rule Serpentine Routing 81
Serpentine Routing Shapes 81
Dynamic Serpentine Routing 85
Serpentine Skew Modes on Differential Pairs 86
CHAPTER 8. ANCILLARY FEATURE TO SUPPORT HIGH-SPEED DESIGN 88
DRC Checks for High Speed Rules 88
Reporting High Speed Rules and Results 88
Standard Reports 88
Custom Reports Using Report Maker 89
CHAPTER 9. BACK DRILLING 90
Back Drilling Overview 90
Technology Back Drilling Spans 90
Auto Generating Back Drilling Spans 91
Technology Back Drilling Rules 91
Identifying Back Drilled Vias in your Design 92
Back Drill Colours 93
Updating Back Drills 93
Back Drill Spacings 93
Back Drilling DRC 93
Via & Component Pad Properties 94
Back Drilling Pin Depth Attribute for Component Pads 94
Layer Span Layers automatically included in Layer Stack Preview 94
Back Drilling in the CAM Plot Wizard 96
Pad Exceptions 96
IPC-2581 and ODB++ Exports include Back Drills 97
Back Drilling Commands in the Report Maker 97
CHAPTER 10. RF DESIGN FEATURES 99
Square-ended Tracks 99
Chamfered Corners on Tracks 100
Spiral Tracks and Shapes 101
Importing Spirals 102

INDEX.....103

Chapter 1. Technology Setup

Technology File and Rules Management

All rules for definition and management of the Interactive High Speed option are available within the Technology file under their respective rule headings. Each page enables you to set constraints and rules for each feature.

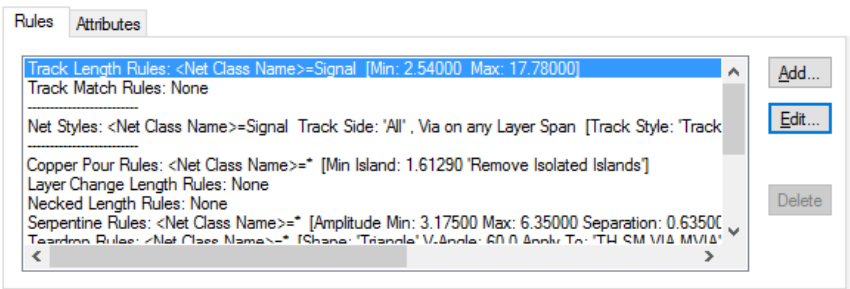
Rules & Attribute Windows

To introduce general functionality used throughout, the **Rules** and **Attributes** window is discussed below.

For each **Net** item page in the Technology that supports rules and attributes there is a 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.



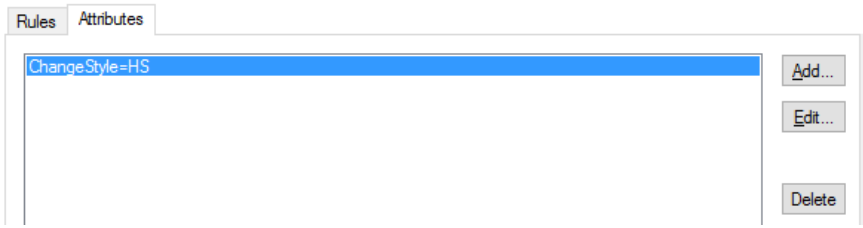
Add – this button is used to add a new rule for the selected rule item. By default, a new rule is created for <Net Name> and matching the net name selected, this is ready to add rule values to. The default net name rule is can be changed to your own criteria if required. If you press Add where the rule 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> CLK, then it can be deleted. If the rule applies to CL*, then it could belong to CLK and CLK1, in which case it cannot be deleted. The rule can be deleted of course from its own rule dialog.

Attributes

Net Attributes can be attached to a net. Rules can then be associated with that attribute. You might use an Attribute rule to attach to Nets that don't meet possible selection criteria. For example, a collection of nets that have different Net Names or different Net Classes but which still need **Track Length Matching** or a particular **Net Style** applied to them.



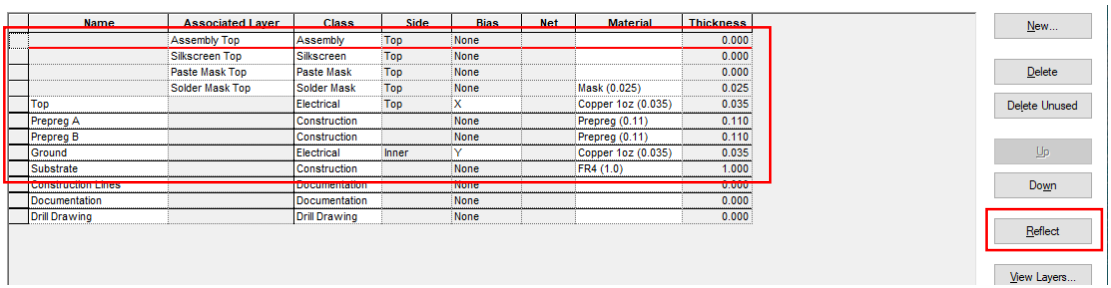
As with the **Rules** pane, there are buttons for **Adding**, **Editing** and **Deleting Net Attributes**. When adding an attribute here, if it is new to the design, it will be confirmed as being added to the **Attributes** dialog as well. This will be added as a Net Attribute for use on nets.

Once the attribute name has been added to the net, you must then create an appropriate rule for it to use. You may assign more than one Net to use this attribute. You may also assign more than one rule to an attribute so care should be taken if doing this.

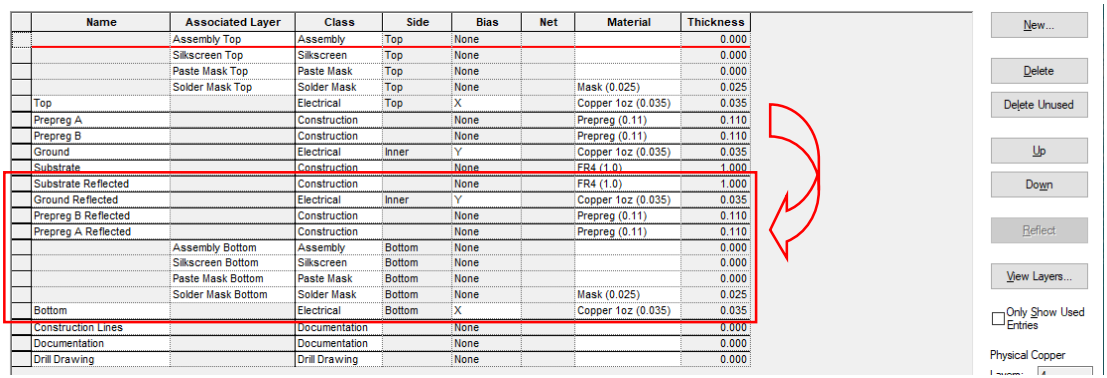
Technology Layers - Reflect Layers

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

This is a general PCB feature but is available to specifically help you create complex layer structures.



After using **Reflect**:



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

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

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

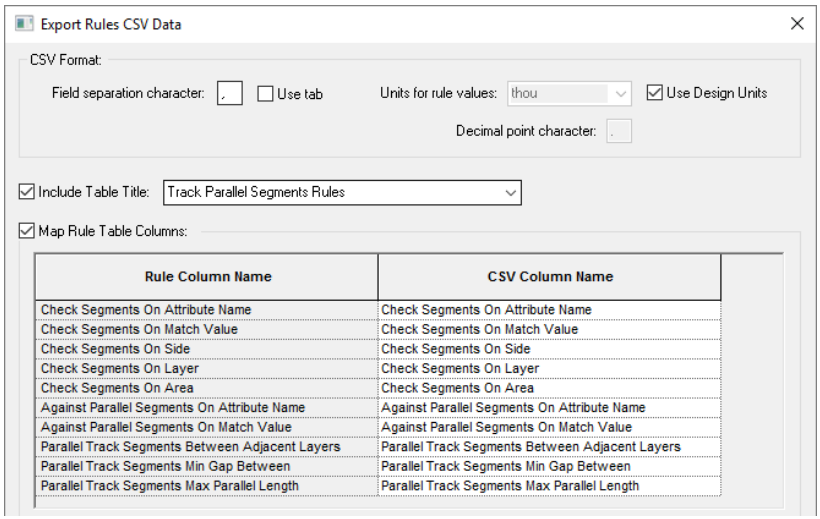
Import & Exporting Technology Rules

Within the **Technology** dialog, the **Export CSV** and **Import CSV** options are available on dialogs for **DFM/DFT rules** and **High Speed rules**. This means rules can be created externally using Excel for example, and imported into Pulsonix. Export of rules allows templates for each rule set to be exported before modification and import.

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 Technology Rules 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:



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

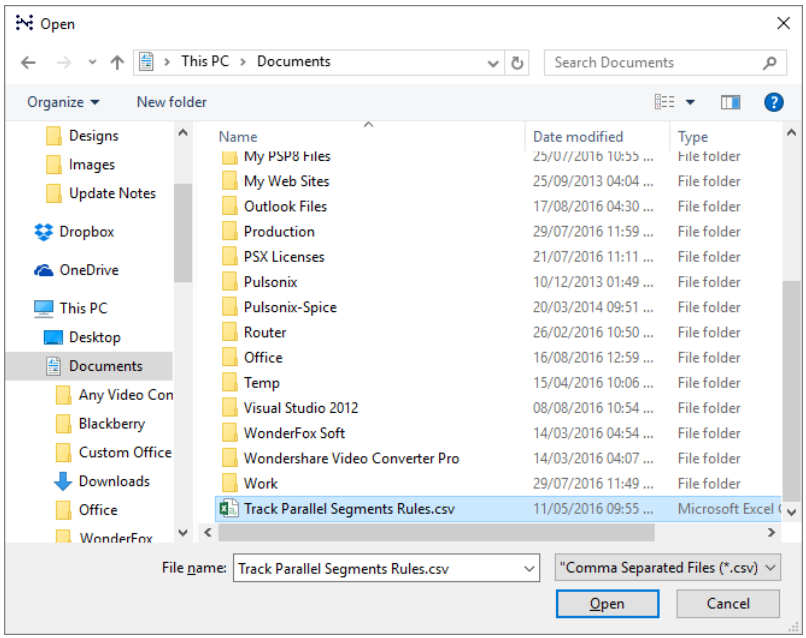
CSV Format – the contents under this header allow you to format the CSV file so that regional variations on the data format can be used.

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.

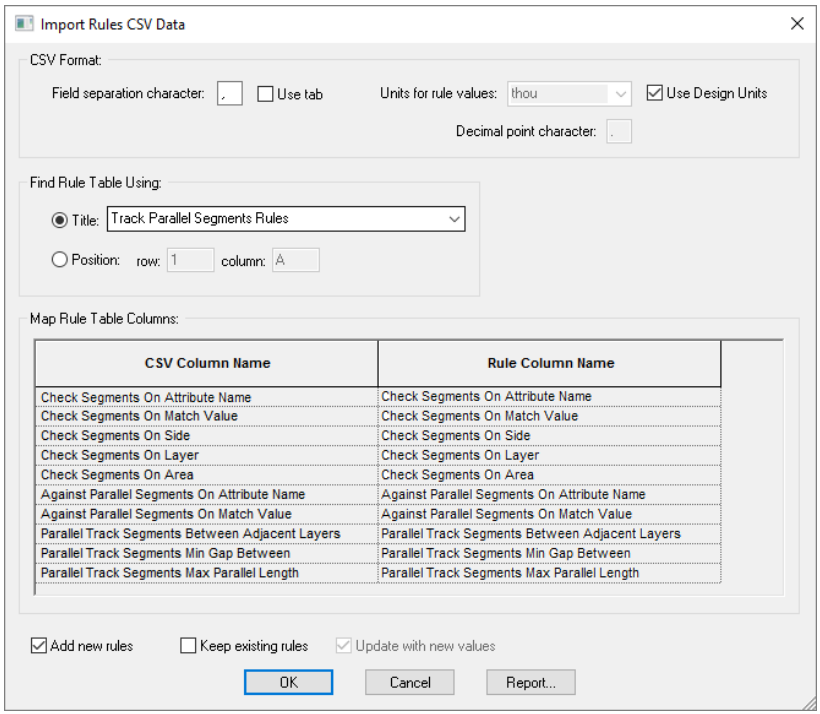
Map Rule Table Columns – this allows you to map the rule name being exported into the CSV file using either default column names provided or your own typed column names. These names appear in the CSV file, it may be that you wish to use shorter or abbreviated names or names in your local language.

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:



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.

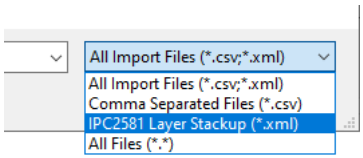
Import IPC-2581 Layer Stackup into Layers dialog

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

This feature can be accessed from the **Technology** page **Layers** and **Import…** and also from the **Import Design Data** option on the **File** menu.

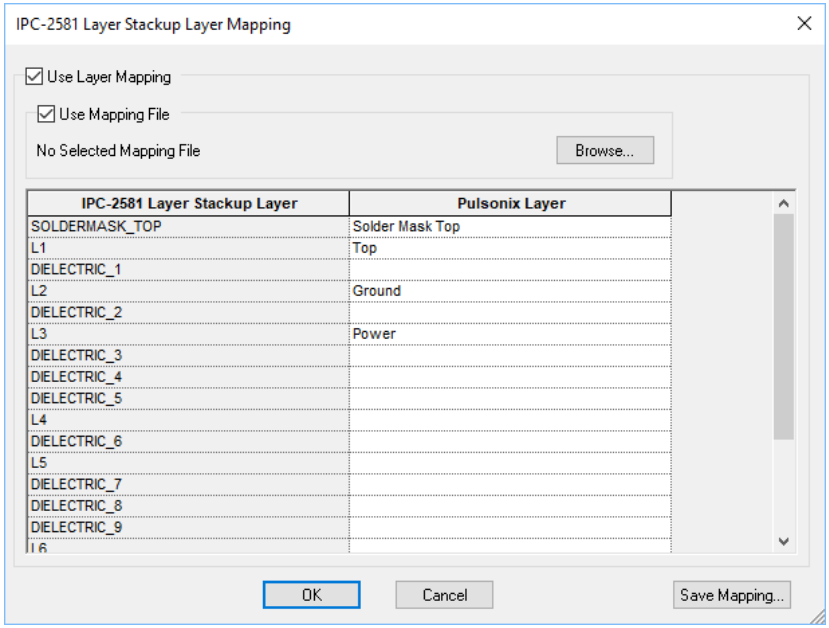
Layers Technology Page

When the **Import…** button is selected, you can import **CSV files** plus **Layer Stackup (*.xml)** files, (the .xml format is used by IPC-2581). The **Files of Type** filter allows you to choose the type required.



Import IPC-2581 Layer Stackup dialog

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


IPC-2581 Layer Stackup Layer Mapping
Use Layer Mapping
Use Mapping File
No Selected Mapping File
Browse...
IPC-2581 Layer Stackup Layer
Pulsonix Layer
SOLDERMASK_TOP
Solder Mask Top
L1
Top
DIELECTRIC_1
L2
Ground
DIELECTRIC_2
L3
Power
DIELECTRIC_3
DIELECTRIC_4
DIELECTRIC_5
L4
DIELECTRIC_6
L5
DIELECTRIC_7
DIELECTRIC_8
DIELECTRIC_9
L6
OK
Cancel
Save Mapping...

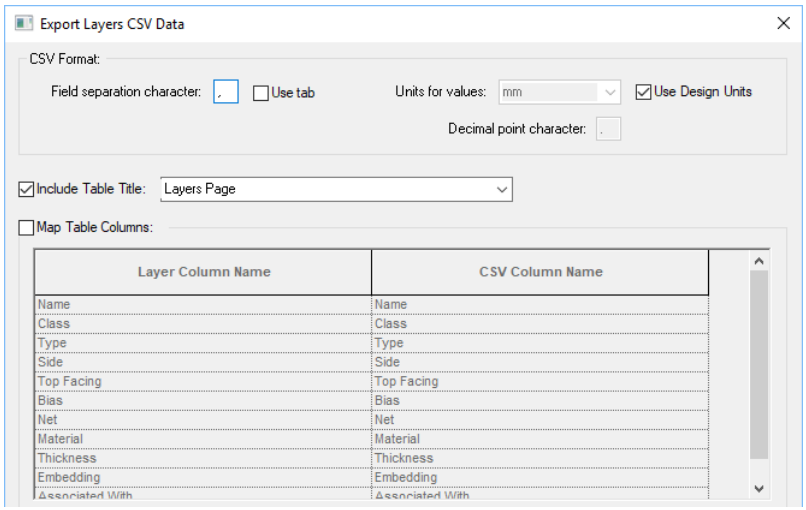
The imported file populates the **Layers** dialog:

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

Import/Export CSV into Layers dialog

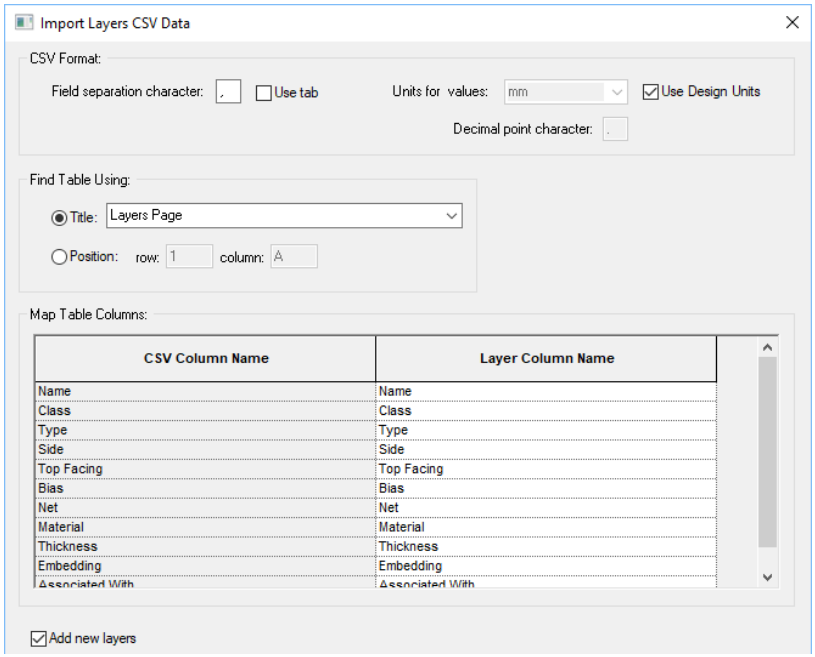
There are options on the **Layer Technology** dialog that enable you to **Export Layers** to a CSV file and **Import Layers** from a CSV file. This is intended to allow you to create layer stacks quickly or from an external resource.

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



Import dialog

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

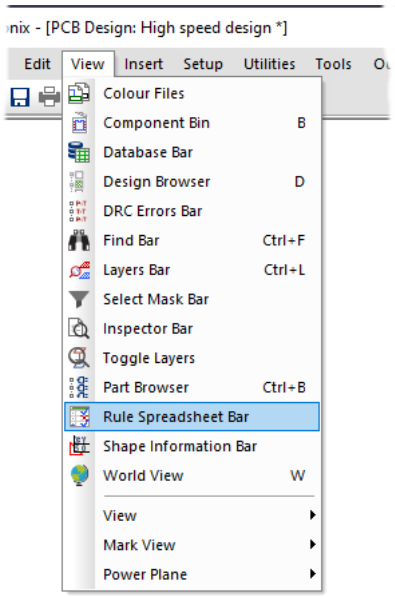


Chapter 2. Rules Spreadsheet

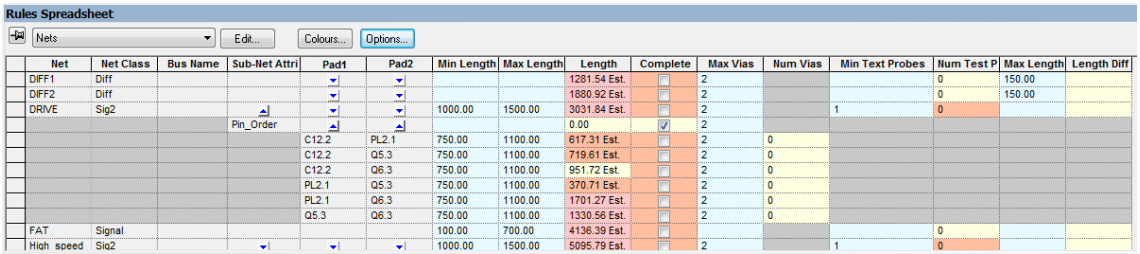
Rules Spreadsheet

As part of the **Interactive High Speed Rules** option, the **Rules Spreadsheet** dockable PCB bar displays information about Nets, their rules and where the rules are being violated. The display is updated dynamically as tracks are added and edited.

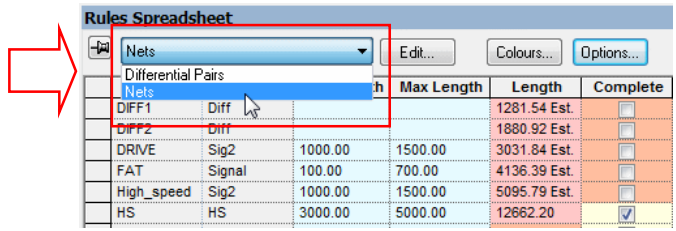
This option is available as the **Rules Spreadsheet Bar** on the **View** menu or from the context menu when right-clicking on the Pulsonix framework.



It is presented in the form of a spreadsheet in a dockable bar, you can switch between different information content.



Differential Pairs and **Nets** are available for selection on the drop down list.

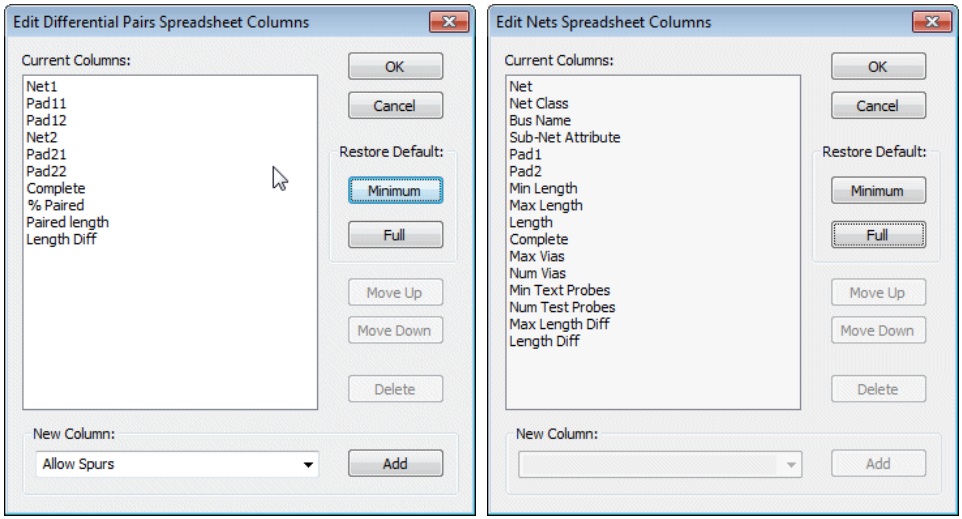


Choosing the spreadsheet type

Use the dropdown list in the dialog header to set the current spreadsheet type, or if the list is not visible select the required spreadsheet type from the shortcut menu. Choose between displaying a list of **Differential Pairs** or a list of **Nets**. Differential Pairs have one row per item, whereas Nets can have additional rows to display sub nets (defined by pin attribute) or to display pin pairs in the net. The existence of these rows depends on which columns have been included. The additional rows can be shown or hidden by pressing the small triangle icon in the owner row above them.

Editing Spreadsheet Columns

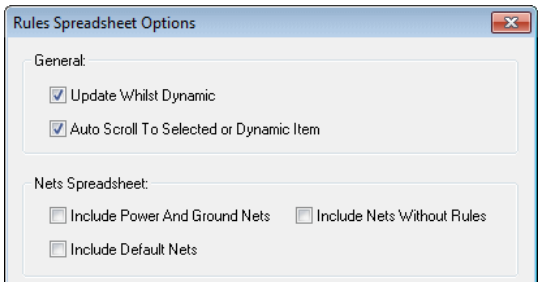
The **Edit** button is used to define what information (columns) are shown in the rules spreadsheet grid.



Using the **Minimum** button allows you to select a minimum number of fields to display. The **Full** button shows a more complete list. Both lists can be enhanced by selecting additional fields using the **New Columns** button and using the **Add** button to add them to the list. Once chosen, the order can be moved using the **Move Up** and **Move Down** buttons. Use the **Delete** button to remove the field from the display list.

Rules Spreadsheet Options

The **Options** button is used to set up your preferences for the rules bar behaviour and to specify which items are not to be included. If the button is not visible, Use **Spreadsheet Options** from the shortcut menu. The options are retained in the system registry so that your choice is preserved for future use.



Update Whilst Dynamic - Generally information in the spreadsheet is updated as soon as items in the design are changed. For large nets on some designs this can take a while, especially whilst placing items and adding track segments. Unchecking this box will stop the "continuous" updating that happens whilst moving items during interactive operations, and they will only be updated when the item is dropped or a corner is added. Leave the box checked if you want to see the track length changing "on the fly" as you move across the board, for example if you are using the spreadsheet instead of the head up track length display on the cursor.

Auto Scroll to Selected or Dynamic Item - Check this box to ensure the current item in the design is always visible in the spreadsheet by automatically performing a vertical scroll.

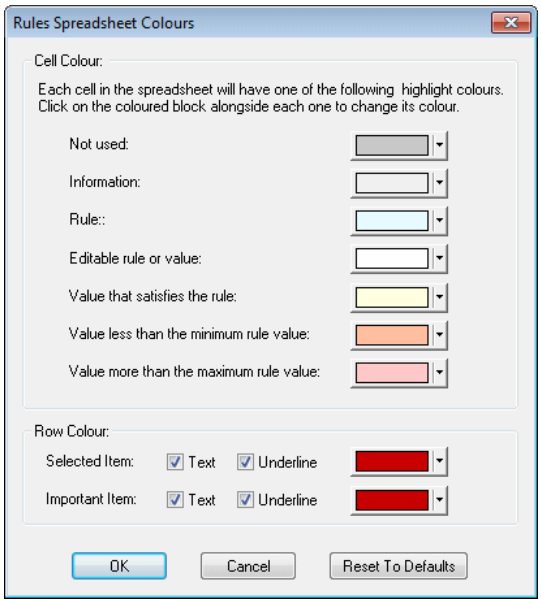
Include Power And Ground Nets - Check this box if you want to include nets that use a net class with type power or ground. These nets are often not required as net track length is not an issue with them. They usually have a lot of nodes and can take a long time when calculating values, like completion for example, as they need to check planes and poured copper.

Include Default Nets - Check this box if you want to include nets that don't have a user defined net name.

Include Nets Without Rules - Check this box if you want to include nets that don't use a net class containing track, via or probe rules. These nets are not normally required in the nets spreadsheet as they have no rules to check against.

Colours Used in the Spreadsheet

Use the **Colours** button to change the colours of the spreadsheet cells, and to define how to highlight the row that represents the item that is currently selected or being edited in the design. If the button is not visible, Use **Spreadsheet Colours** from the shortcut menu. All colours in this dialog are retained in the system registry so that your choice of colours is preserved for future use.



The top section shows a set of colour buttons showing the colours that will be used for each cell information type. Click on a colour block to change its colour.

Not Used - This colour is used for cells that are not needed, i.e. their column is not relevant for their row. For example, the "Net Name" cell on a pin pair row.

Information - Used for cells that represent non editable information about the row's data item, that is not directly associated with a rule. For example, the two net names for a differential pair.

Rule - This colour is used for non-editable values that represent a rule limit. For example, the maximum track length for a net. To change these values, you must visit the appropriate dialog in the technology.

Editable rule or value - Used for values that are editable. These usually represent a rule limit that is directly editable in the spreadsheet (like the differential pair rules) or an information cell that you can alter to change what data its row represents (pin name cells in a pin pair row on a large net for example).

Value that satisfies the rule - This colour is used for values that are within their corresponding rule limits, and therefore represent a rule pass. For example, a nets track length that is within the defined minimum and maximum rule values.

Value less than the minimum rule value - Used to highlight values that fall short of their corresponding minimum rule limit. For example, a pin to pin track length that is less than the minimum pin to pin length defined for a net, or a net complete cell on a net that has tracks missing.

Value more than the maximum rule value - Used to highlight values that exceed their corresponding maximum rule limit. For example, a pin to pin via count that is greater than the maximum number of vias allowed between pins on the net class.

The lower section shows how the selected or important items in the design have their rows highlighted in the spreadsheet. For each highlight type you choose a colour that will be shown as a block in the first column of the row, and choose whether the text in the row is displayed with this colour, and also if the row is underlined with the colour.

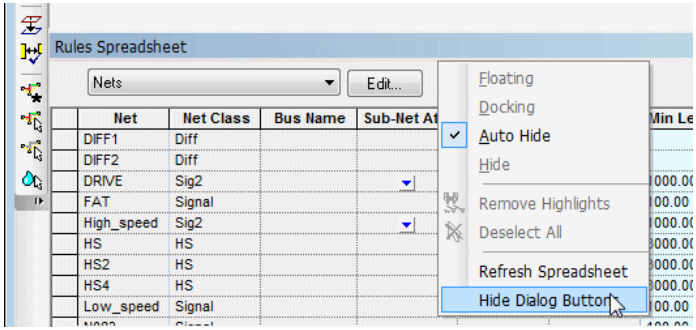
Selected Item - This row represents the single net or differential pair in the design that is selected, or a part of it that is being edited (a track being edited for example).

Important Item - These rows represents nets or Differential Pairs that have been marked as important in the spreadsheet. See above for information about highlighting rows for important items.

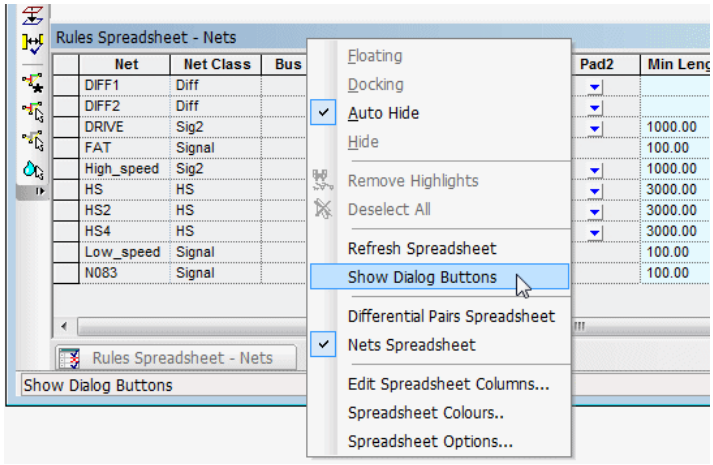
The **Reset To Defaults** button will return the colours and highlight check boxes to their original "factory" settings.

Hiding Dialog Buttons

You can maximise the space available for the spreadsheet grid by using **Hide Dialog Buttons** from the shortcut menu to remove the dialog controls above the spreadsheet and move the spreadsheet up.

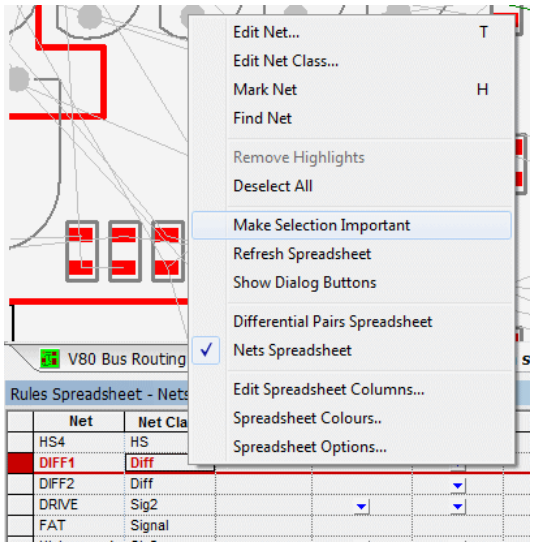


The **Show Dialog Buttons** option will reverse this. When hidden, you can still change the spreadsheet colours, options and columns by using commands on the shortcut menu. If the buttons are hidden, the title of the bar will include the spreadsheet type.



Highlighting Important Items

Using left click in any cell in the first column that represents a differential pair or net (i.e. not a sub row) will mark the item the row represents as "important" to you. The row will be highlighted to enable you to quickly see the information whilst editing the design. You can specify how the row is highlighted using the spreadsheet **Colours** dialog. Use **Remove Important Items** from the shortcut menu to clear the spreadsheet of any highlighting of important items. Use **Make Selection Important** from the shortcut menu to highlight all rows that have selected items in the design.



If a single differential pair or net is selected or is being edited, the row will be highlighted as the "selected" row. How the selected row is highlighted is defined in the spreadsheet **Colours** dialog. There is a spreadsheet option (see below) to ensure that the selected row is always scrolled to be visible.

Chapter 3. Nets and Rules for the High Speed Option

Nets Items

This is brief introduction to Nets, Diff Pairs, Diff Pair Chains, Signal Paths, Sub Nets and applying rules to them.

Within the **Technology**, there are 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 net items can have attributes assigned and rules added to them.

Why and when to use each category

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

Standard net items

Nets – All Nets (Connections) within the design have a **Net Name** with a **Type** *(see special note below about Type)* and (optionally) a **Guard Space**. They can 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 Names can be user defined or default names automatically assigned by Pulsonix.

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**. A Net Class is an alternative method of 'grouping' nets to which the same parameters can be associated. This is a general method for grouping categories of nets but can be achieved more efficiently using alternative methods. Nets do not require a Net Class, all rules can be attached to a single net or multiple nets.

Net Styles – All default [physical] 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**.

Net items for the High Speed Design Option

Signal Paths – 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. 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 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 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 – 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 Technology Rules, 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 so mention a few.

What can a rule be assigned to?

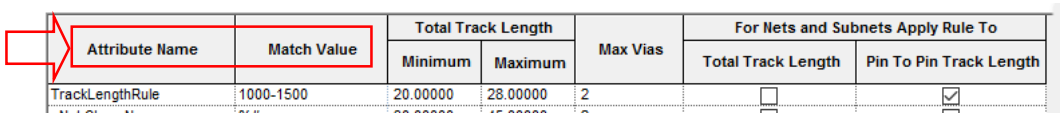
Rules can be assigned to any Net, Signal Path, Sub-Net, Differential Pair and Differential Pair Chain. This means one (or multiple) rules can be applied to multiple net categories. For example, a number of Thermal rules can be defined and applied to all signal or power style nets. Likewise, a Track Length 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 this style of rules structure is less likely to be used this way.

In addition to assigning rules to Net items, in the Thermal Rules and Teardrop Rules dialogs, you can also assign rules to <Pad Style Name>. This means specific rules can be added for pad or via styles for ranges or specific pads.

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	☐	☑

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.

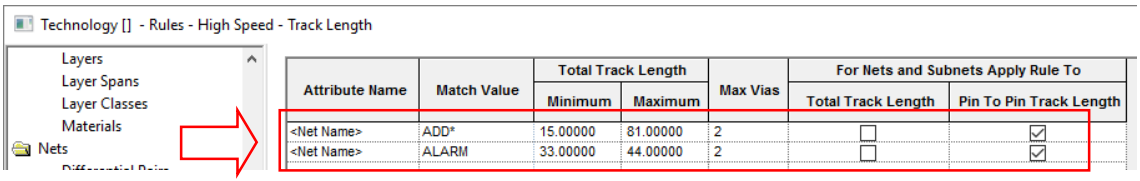
Creating and applying Rules

Rules can be created and applied using two methods:

Method 1

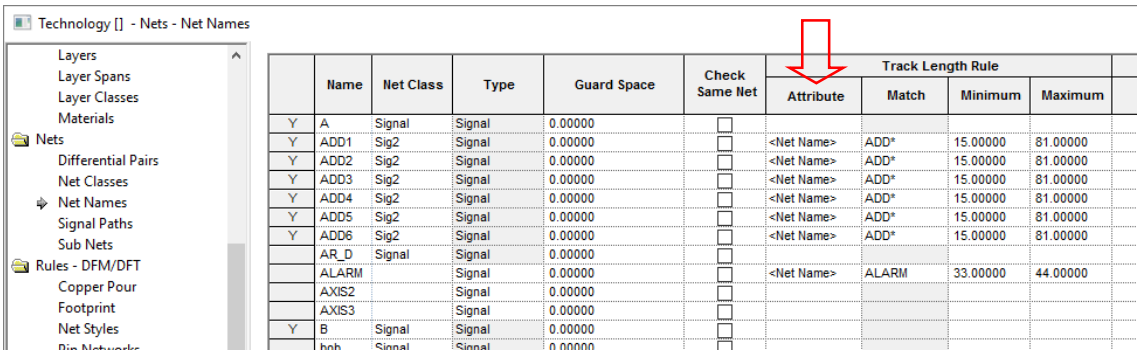
Create your rules first in the Rules sections (in the Technology dialog under DFM/DFT and High Speed) and apply them to the net(s) required.

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**:

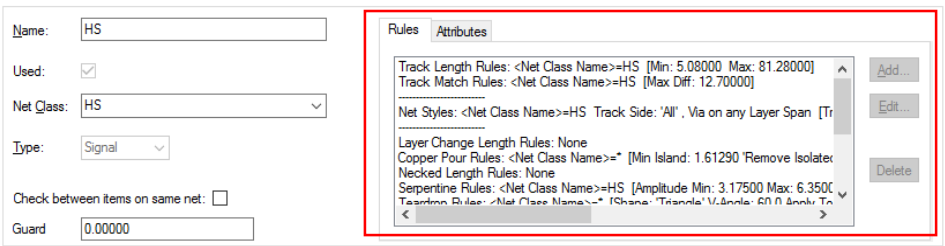


On the **Nets** page, if the new attribute matches the rule, the attribute name and value will be automatically adopted. If the rule doesn't match automatically, apply the rule by typing the rule name in the **Attribute** field (or selecting it from the drop down list which will be populated from the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

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



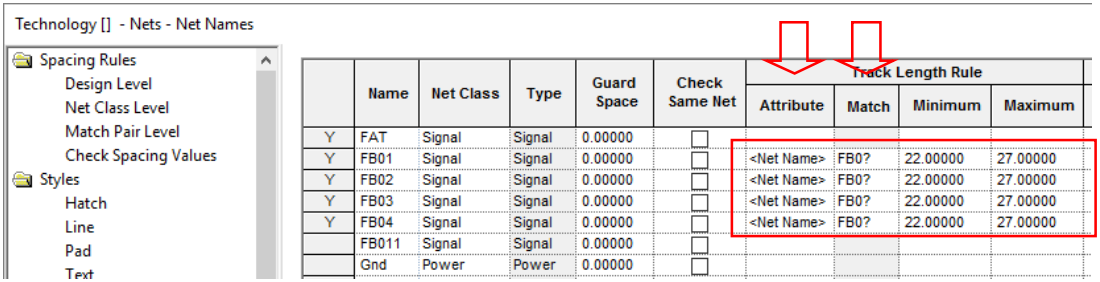
All rules added to a net-based item are displayed in the relevant dialog in the **Rules** and **Attributes** tabs:



Method 2 – Track Length Rule & Track Length Match Rule

When using the **Track Length Rule & Track Length Match Rule**s, 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).



	Name	Net Class	Type	Guard Space	Check Same Net	Track Length Rule			
						Attribute	Match	Minimum	Maximum
Y	FAT	Signal	Signal	0.00000	☐				
Y	FB01	Signal	Signal	0.00000	☐	<Net Name>	FB0?	22.00000	27.00000
Y	FB02	Signal	Signal	0.00000	☐	<Net Name>	FB0?	22.00000	27.00000
Y	FB03	Signal	Signal	0.00000	☐	<Net Name>	FB0?	22.00000	27.00000
Y	FB04	Signal	Signal	0.00000	☐	<Net Name>	FB0?	22.00000	27.00000
	FB011	Signal	Signal	0.00000	☐				
	Gnd	Power	Power	0.00000	☐				

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

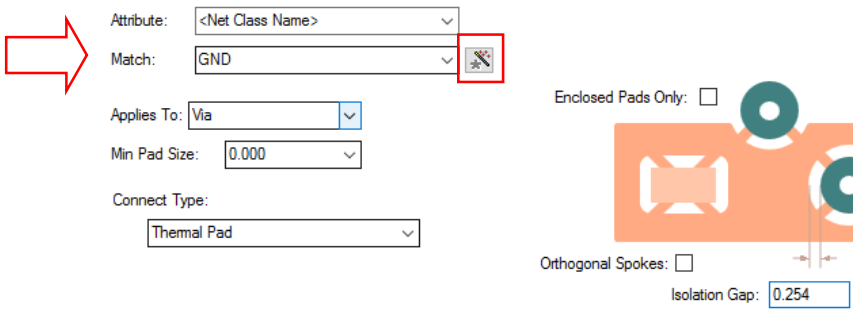
Wildcards are a powerful way to create ranges for selection, below shows a selection of wildcard ranges:

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.

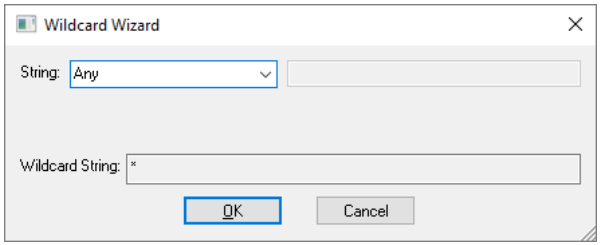
Wildcard Wizard

When a wildcard needs to be defined to create name ranges, you can use the **Wildcard Wizard**. This allows you to create wildcards, often, complex wildcard strings (or simple ones), that are presented in non-programming terms.

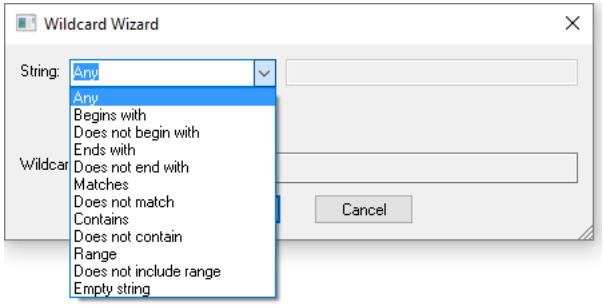
On dialogs where wildcards are accepted, there is a button to access the wizard.



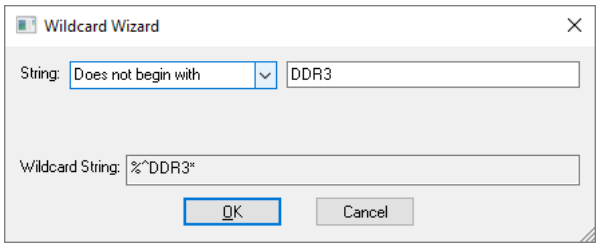
Pressing the button opens up the **Wildcard Wizard** dialog:



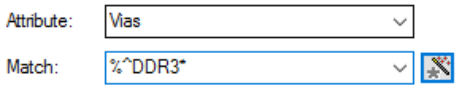
Wildcard String expressions are defined from the drop down list:



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?

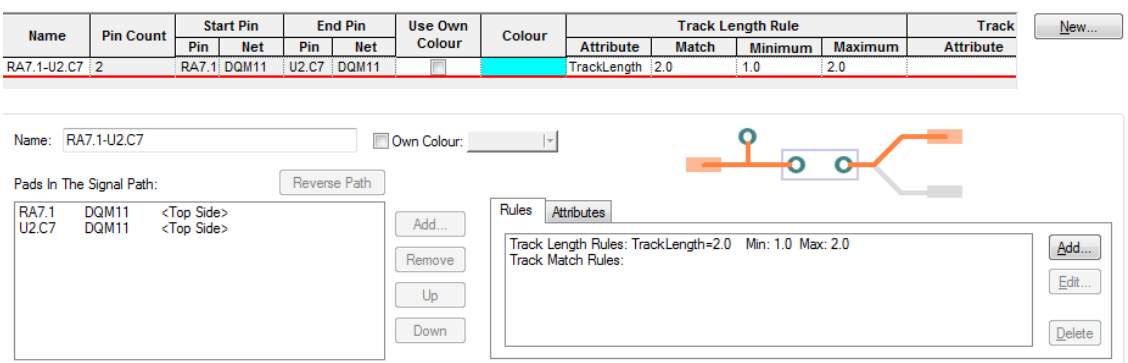
Signal Paths represent another level of net definition. The path is a named item containing an ordered list of pads that represent a signal path. For example, you can use this for High-Speed designs where constraints on specific signals are required or for defining portions of a net where specific rules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

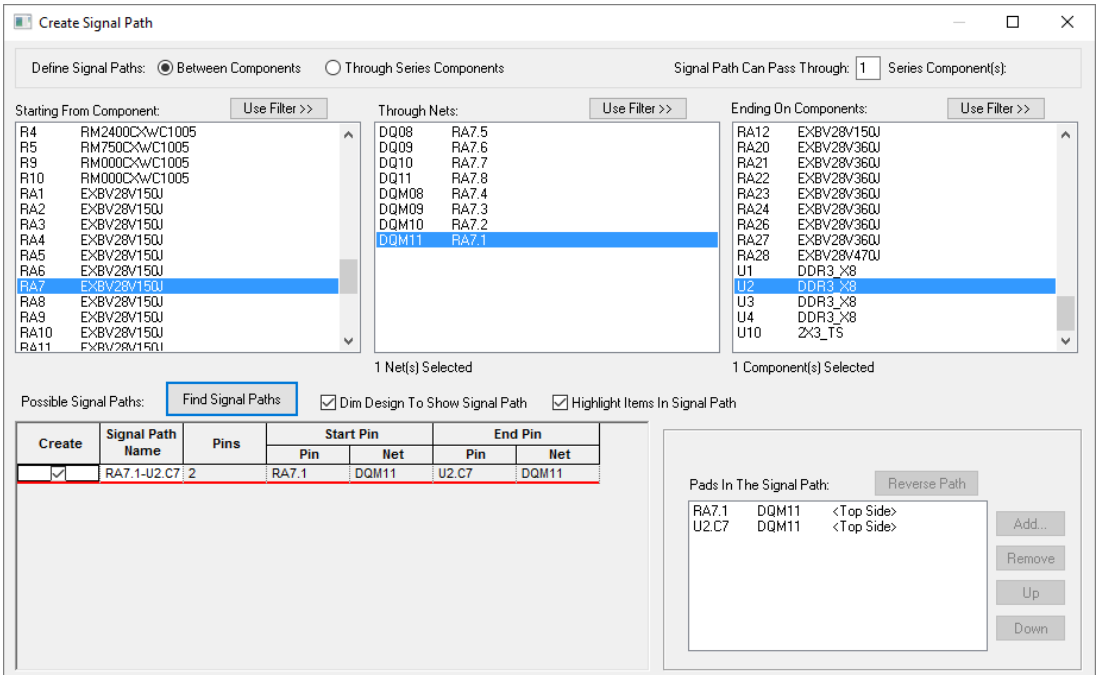
Signal paths are added in the Signal Paths page within the **Technology** dialog. From this page, **Track Length** and **Track Match Rules** can be applied if required or you can use the Signal Path name within other rules, such as **Net Styles**, **Serpentine Rules** or **Thermal Rules** etc.



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.

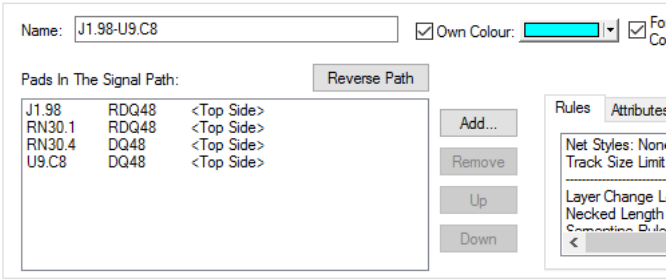
If satisfied with the signal path displayed, press the **Create** button. If this is your only selection at this point, press **OK** or **Apply** to add the selection to main Signal Paths dialog. If not satisfied with the selection, leave the Create button unchecked and make a new selection in the chooser.

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 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.

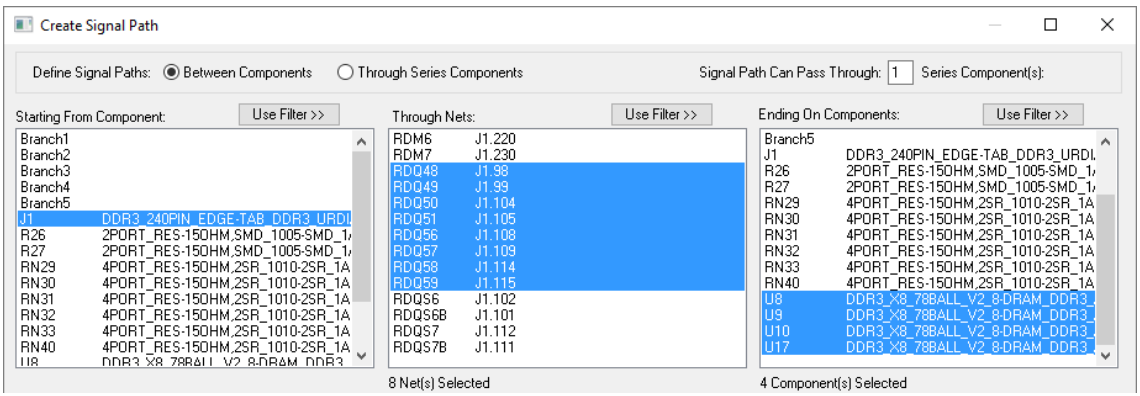


Use these to indicate you want to create the path, apply a name for the signal path and allocate a track length and length match rule to it. If the path is to pass through more than two pins within a particular net, the extra pins can be added into the correct place in a pad list.

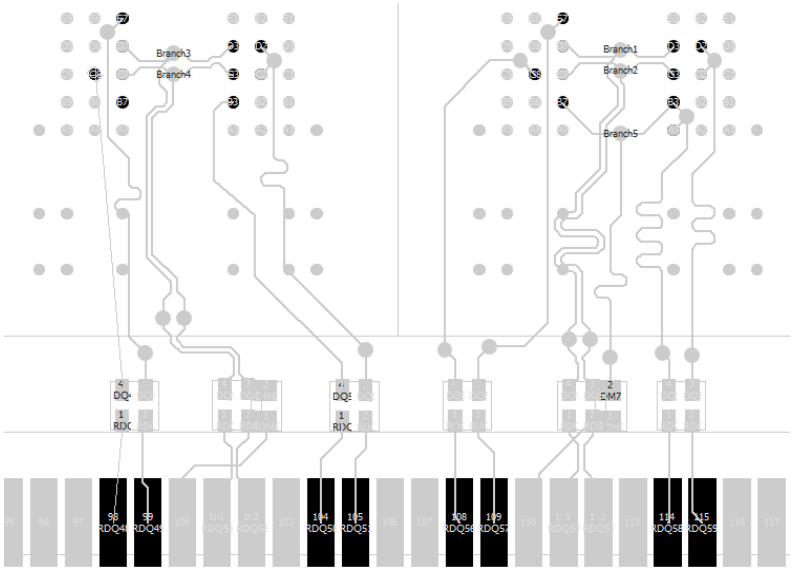
Adding Multiple Signal Paths in one hit

Multiple signal paths can be created at the same time.

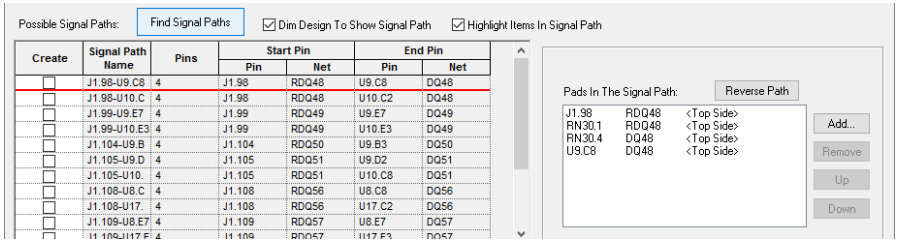
In the example below, Connector J1 has been chosen as the **Start Component**. As it happens, this has all the nets required exiting from it. You must then choose the nets required for the paths in **Through Nets**. Multiple nets can be chosen at one time by dragging down the list or using standard Shift/Ctrl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



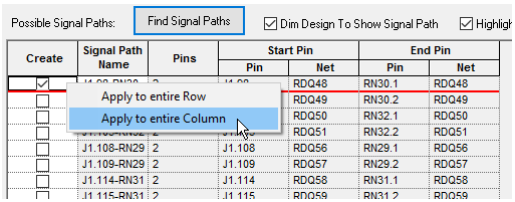
With this selection made, with the design dimmed, the proposed signal paths are displayed. You can use zoom and pan to navigate the design at this point.



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



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.



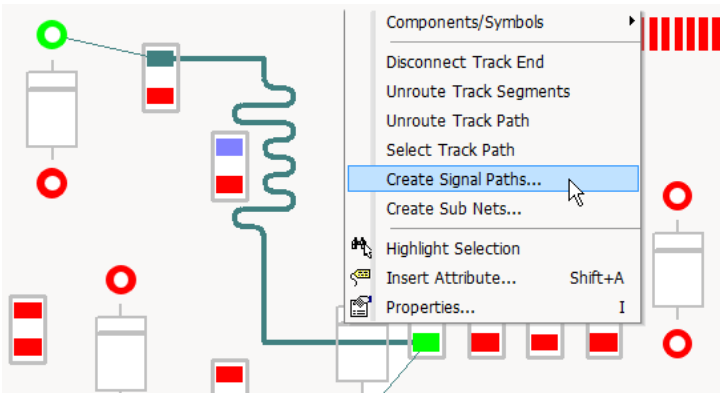
Once created, the main dialog displays the Signal Paths:

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	RDQS6	RN33.	RDQS6	☐		☐	<Signal Path N	J1*	12.000	13
J1.104-RN32.1	2	J1.10	RDQ50	RN32.	RDQ50	☐		☐	<Signal Path N	J1*	12.000	13
J1.105-RN32.2	2	J1.10	RDQ51	RN32.	RDQ51	☐		☐	<Signal Path N	J1*	12.000	13
J1.108-RN29.1	2	J1.10	RDQ56	RN29.	RDQ56	☐		☐	<Signal Path N	J1*	12.000	13
J1.109-RN29.2	2	J1.10	RDQ57	RN29.	RDQ57	☐		☐	<Signal Path N	J1*	12.000	13
J1.114-RN31.1	2	J1.11	RDQ58	RN31.	RDQ58	☐		☐	<Signal Path N	J1*	12.000	13
J1.115-RN31.2	2	J1.11	RDQ59	RN31.	RDQ59	☐		☐	<Signal Path N	J1*	12.000	13
J1.98-RN30.1	2	J1.98	RDQ48	RN30.	RDQ48	☐		☐	<Signal Path N	J1*	12.000	13
J1.99-RN30.2	2	J1.99	RDQ49	RN30.	RDQ49	☐		☐	<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

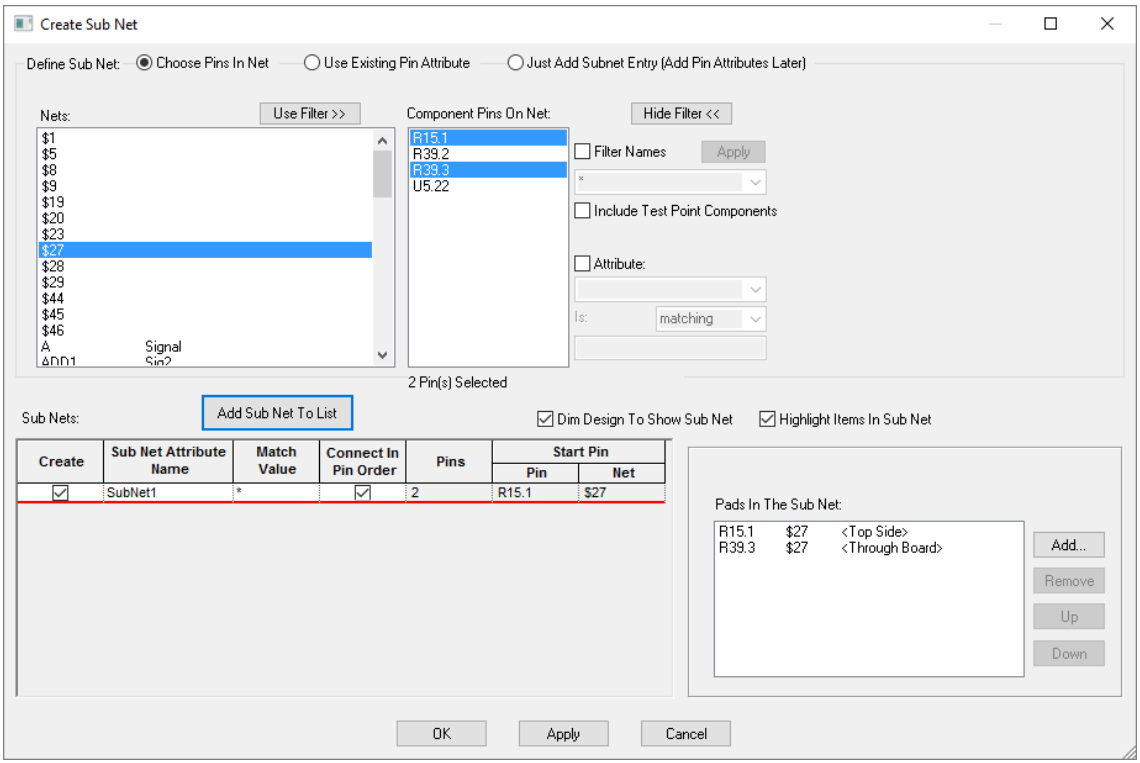
Overview

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

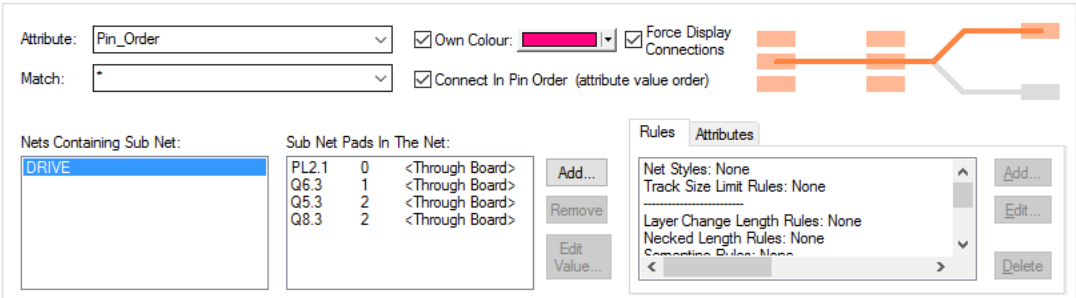
A new tool provides an interactive mode and modeless dialog to create sub nets. It can be invoked from the **New** button in the **Sub Nets** page within the **Technology** dialog. It can also be accessed from the context menu when a component pin(s) or net is selected.



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	*	☐	☐		☐	<Sub Net Name>	HS1	6.35000	12.70000
Pin_Order	*	☑	☑		☑	<Sub Net Name>	Pin_Order	19.05000	27.94000



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.

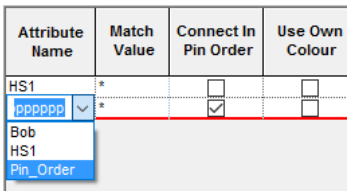
If creating sub nets in a Technology file there are no pins, so pressing the **New** button directly adds a blank sub net to the Technology page.

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.

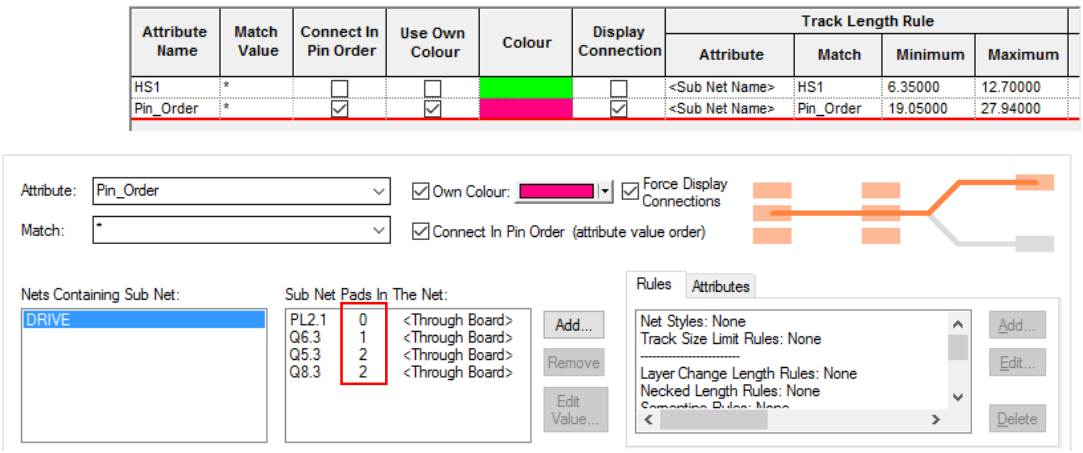
If you wish to change the attribute name, this must be done in the Attributes page of the Technology. Once the name has been changed, you will then need to select it using the Sub Nets dialog and Attribute Name.

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.

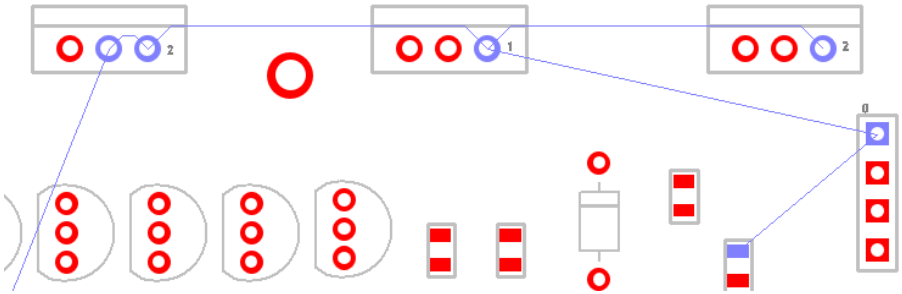


Connect In Pin Order

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.



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



Existing Pulsonix V8.5 Designs

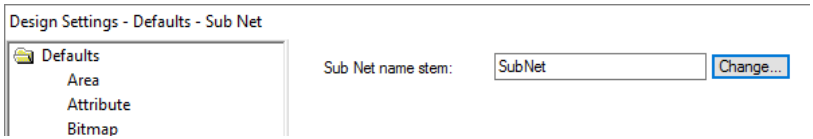
If you have already used **pin attributes** to define sub nets in a previous version of Pulsonix, i.e. V8.5 or older, then these would now be converted into sub nets.

New Designs

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

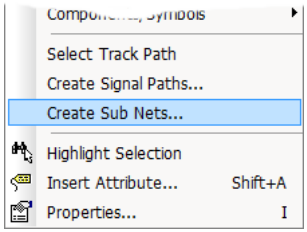
Designs Settings and Default Sub Net names

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.



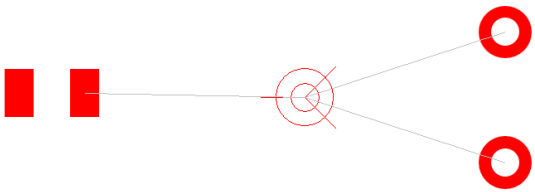
Once selected, the **Create Sub Net** dialog is available with the sub net preselected from where you can add this sub net entry to the **Sub Net** rules within the **Technology**.

Branch Points

Overview

Branch points 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.

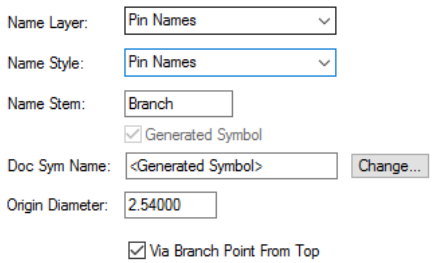


When added to a Schematic design they are automatically transferred to the PCB. Once added they can be used on **Signal Paths, Sub Nets** and **Differential Pairs**.

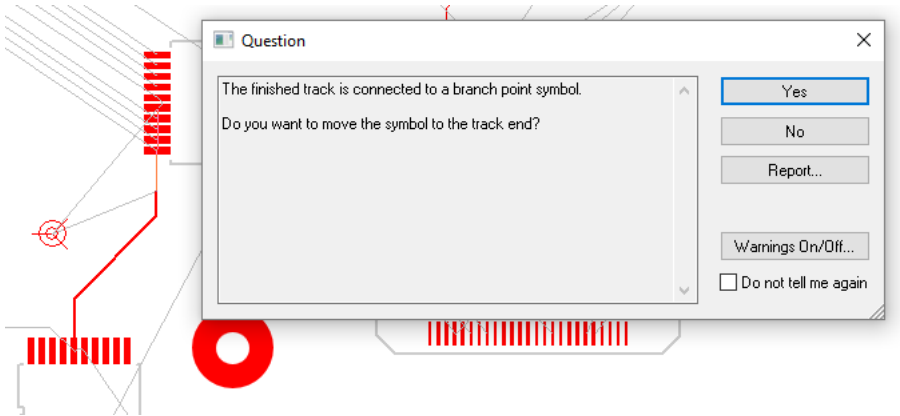
Branch points are added to the design using the **Add Branch Point** option or by using **Change Branch Point** on a selected **via**. When added, you are required to attach the branch point to a pad or connection.

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.

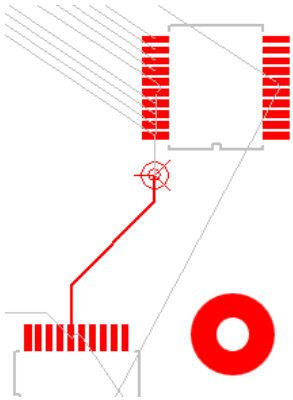
The default symbol or auto-Generated symbol is specified in the **Design Settings** dialog and **Branch Point** page. You might use your own **Branch Point Doc Symbol** to add extra detail for plotting or some other highlighting. When branch points are used in the design, they are zero size items and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



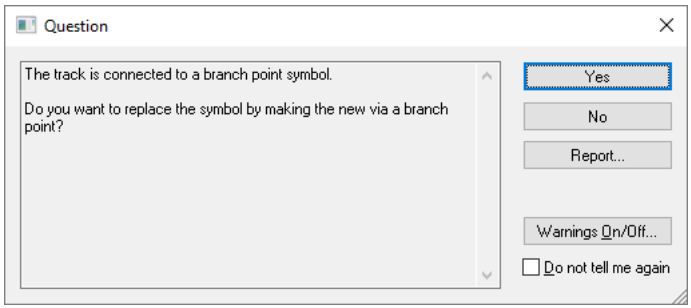
When adding a track (or a differential paired track) with a connection attached to the end that ends on a branch point, after finishing the track at a set position you will be asked if you want to move the branch point to the end of the track:



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



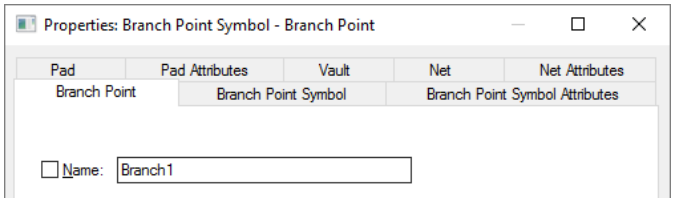
If the track ends on a via then you will be asked if you want to change the via to a branch point via, replacing the doc symbol branch point.



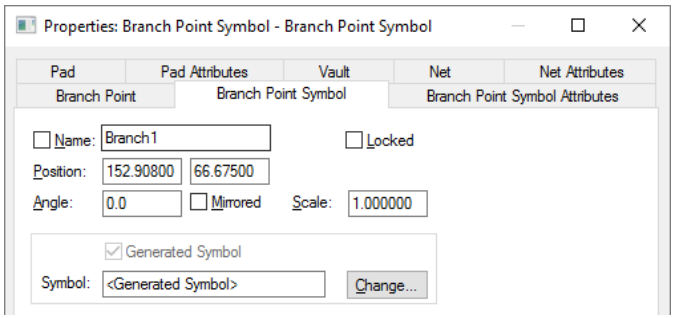
Branch Point Properties

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.



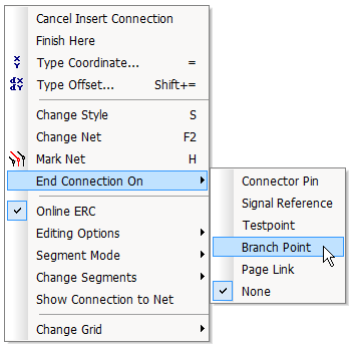
The **Branch Point Symbol** tab displays information about the position of the branch point and the symbol used.



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 Fetaures

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.



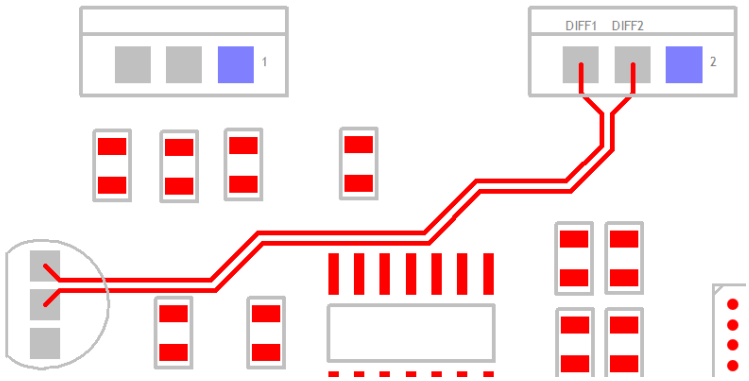
You can **cross probe** branch points between Schematic and PCB, and vice-versa.

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

Chapter 4. Differential Pair Routing

Overview of Differential Pairs

Differential Pairs and **Differential Pair Chains** can be specifically defined at both Schematic and PCB stages. Functionality within Pulsonix allows parameters to be created so that **Differential Pairs** behave and are routed as required. The interactive Differential Pair Routing feature is available in the PCB design editor.



Differential Pairs are defined in Pulsonix using the **Technology** dialog and **Differential Pairs**. Rules can be defined for them, such as how close the tracks should be and how much they are allowed to differ in length as two rule examples. Other rules may be applied to Differential Pairs, such as the default Minimum Gap and Minimum Layered Gap. Rules from other high speed options can also be used, such as overall Track Length of the net and Serpentine Routing.

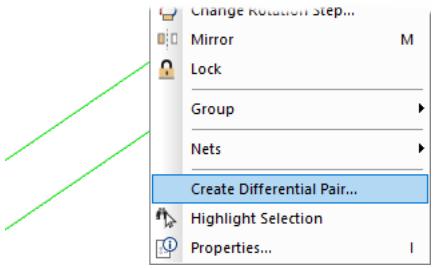
Once the rules are defined, a special manual routing mode for differential Pairs can be used to route the two track pairs at the same time. Paired track sections are locked together using the rules gap defined in the Differential Pairs dialog. Whilst paired, subsequent editing of the tracks will keep them locked together using the functionality provided.

Defining Differential Pairs

A Differential Pair requires two pairs of pins on two different nets. There are various methods for selecting the nets depending on where you start from. You can create these in your Schematic design or in your PCB.

Manual Process

You can select two nets in a PCB or Schematic design and from the context menu use **Create Differential Pair** to create the 'pair'. Once these nets are selected, it will open up the **Differential Pair Rules** dialog in your **Technology** with these nets pre-selected and ready to define. If four pads are selected, they will be used for the pair, otherwise the pads will be automatically allocated. You will not see this command on the context menu if the nets already form a Differential Pair, or don't have at least two pins on each net.



Similar to above but you can preselect the two connections to be paired in the design, (as before, select the first then use <Ctrl-select> to select the second one). Then go into the **Technology** dialog and **Differential Pairs** page, pressing the **New** button will automatically add these two nets as the new 'pair'. Obviously, the above method is easier.

Both of the above processes can be done manually by using the **Differential Pairs** dialog to select both nets using **First Pin Pair** and **Second Pin Pair** drop down boxes. The drop-down list selection allows picking nets and pins on that net within the design. The first method of preselecting the nets is much easier though.

Create Differential Pairs From Rules

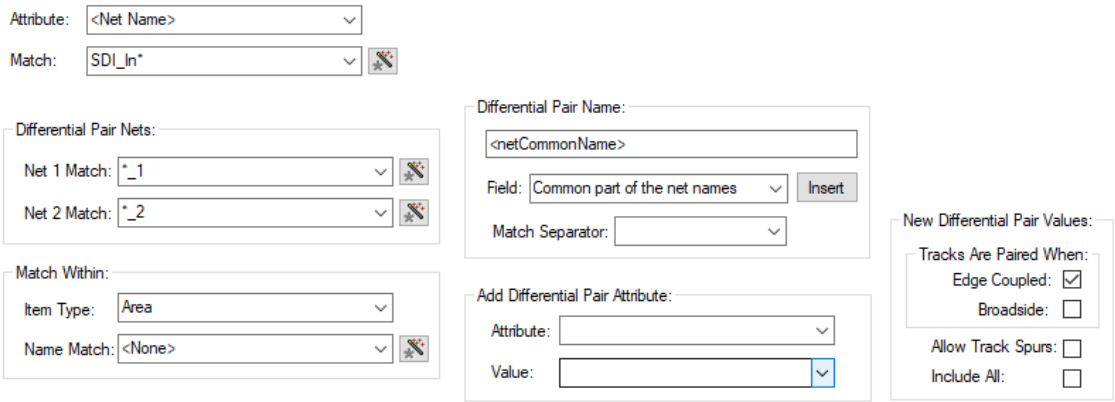
You can create Differential Pairs in a Schematic or PCB design by setting up and then applying new **Differential Pair Nets** rules, defined in the **Rules - High Speed** section of the **Technology** dialog. This enables you to create a rule or rules that capture the net names in your design to apply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

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

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

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

Enable	Attribute Name	Match Value	Differential Pair Nets		Match Within		Template String	Edge Coupled	Broadside	Allow Spurs	Include All	Add Differential Pair Attribute	
			Net 1 Match	Net 2 Match	Item Type	Name Match						Attribute Name	Attribute Value
☑	<Net Name>	DQS_*	*P	*N	Area	<None>	<netCommonName>	☑	☐	☐	☐		
☑	<Net Name>	LCO_D*	*1	*2	Area	<None>	<netCommonName>	☑	☐	☐	☐		
☑	<Net Name>	SDI_In*	*_1	*_2	Area	<None>	<netCommonName>	☑	☐	☐	☐		
☑	<Net Name>	SDQ0*	*_P	*_N	Area	<None>	<netCommonName>	☑	☐	☐	☐		



Attribute

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

Match

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

Differential Pair Nets

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

The Net 1 Match

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

The Net 2 Match

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

Match Within (for a PCB design)

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

Differential Pair Name

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

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

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

Common part of the net names - use this to name the Differential Pair with the common part of the two net names. If the Attribute Match string contains a stem before the wildcard * you can choose a separator to be used after that stem in the resultant name. If this is required, type the **Match Separator** or select it from the drop-down list. For example, if the Attribute Match string is SDQ* and *_P and *_N are used to match nets SDQ0_P and SDQ0_N, then the common part of the names will be SDQ0, and if the Match Separator is "_" it inserted after the stem to give a Differential Pair name of SD_Q0.

First net name.

Second Net Name.

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

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

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

Add Differential Pair Attribute

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

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

New Differential Pair Values

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

Allow Track Spurs

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

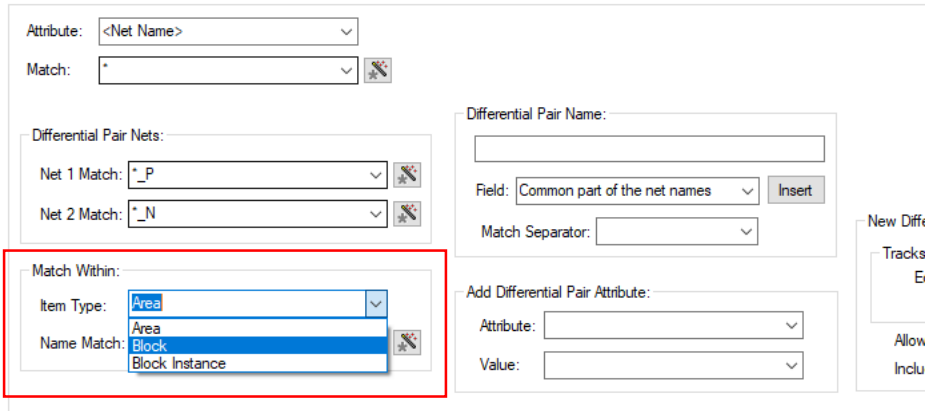
Include All

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

Match Within Schematic

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

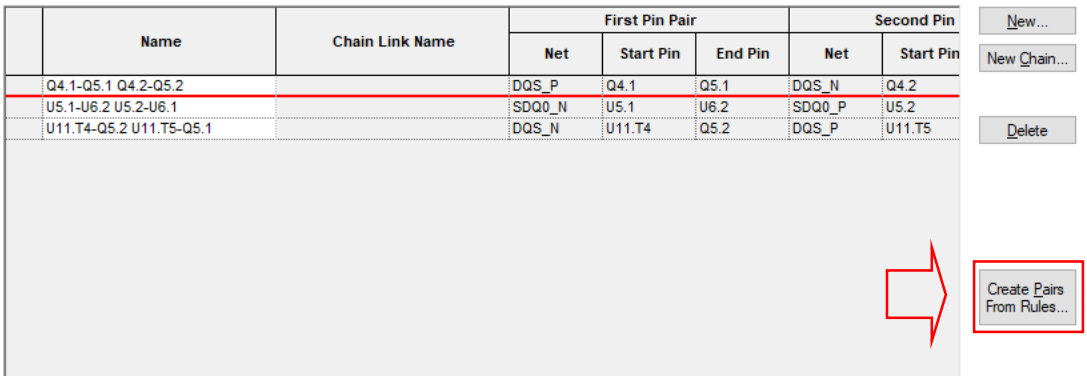
In this case, Block Instance Name is the name of an instance of a block in the design. Block Name is the name of a block used by the instance; several block instances may use the same named block. The nets may be across the design but the reason for the area is only pins defined in that area, block,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



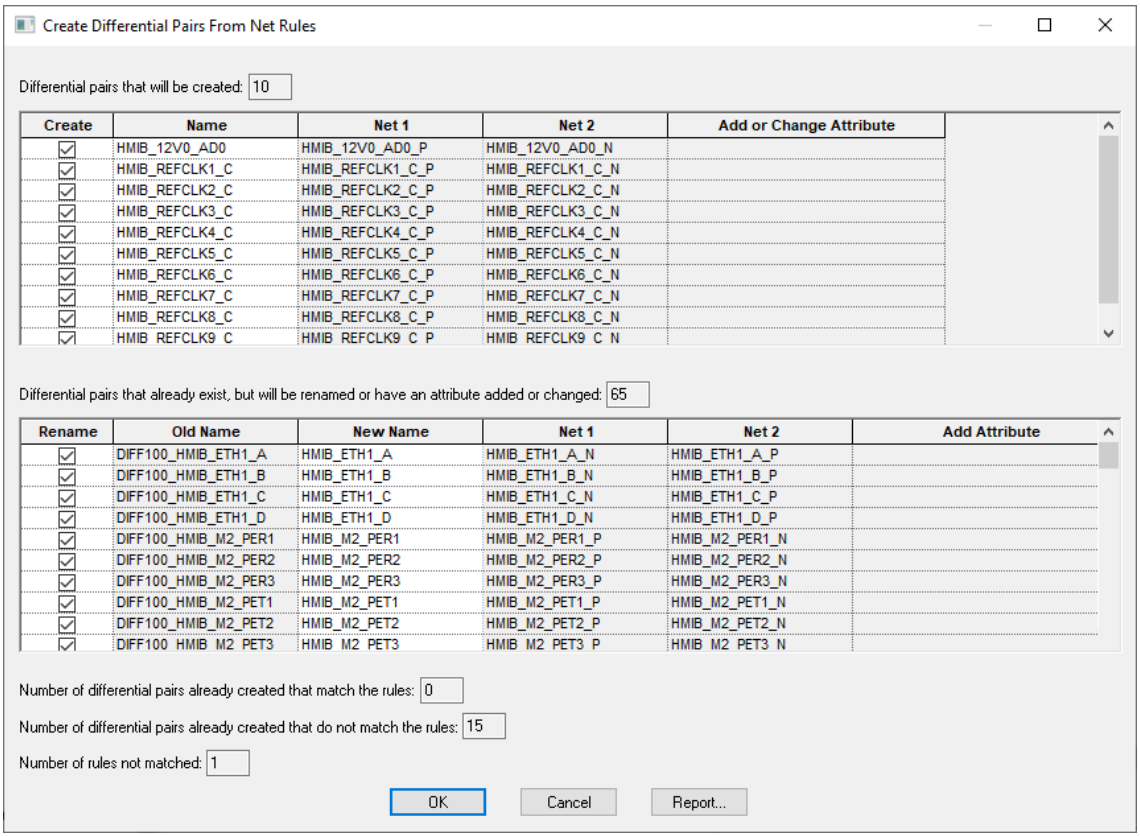
Create Pairs From Rules Dialog

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

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



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



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

Differential Pairs that will be created

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

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

Differential Pair that already exist

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

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

Summary boxes

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

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

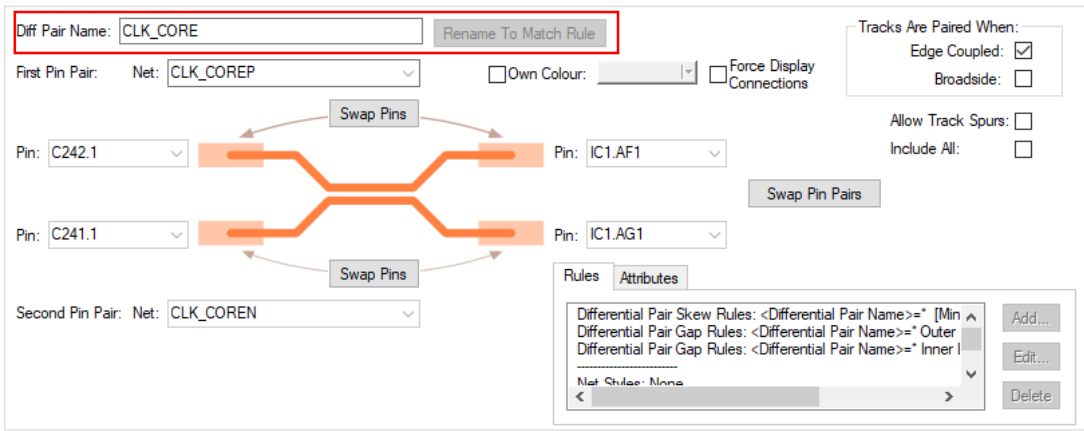
Number of Differential Pairs already created that do not match the rules, shows how many Differential Pairs that are currently in your design that do not match any rules. This is a very useful check to ensure that if you do want to cover all Differential Pairs using rules, then this should read zero.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

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

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

Reapplying Rules

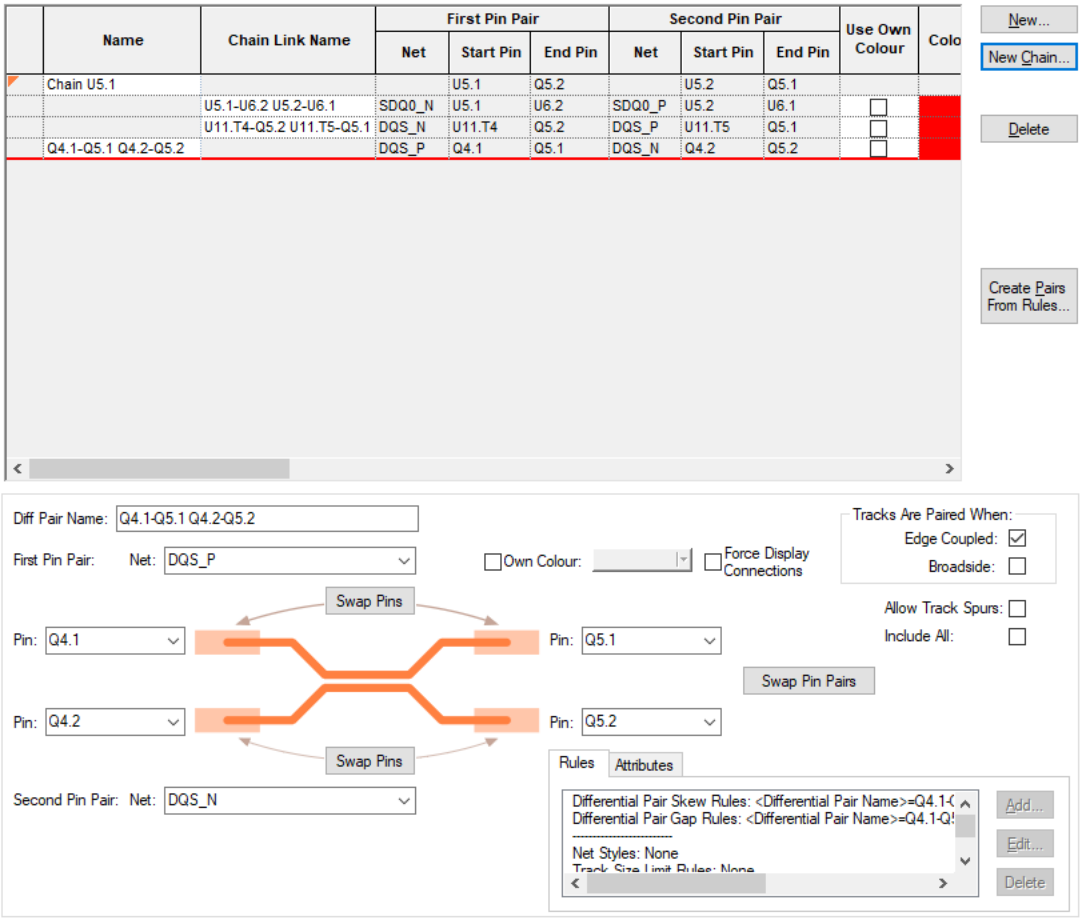
A button on the **Differential Pairs Nets** dialog next to the **Diff Pair Name** enables you to **Rename to Apply Rule** to match any new rules defined in the **Differential Pair Net Rules** dialog.



Using the Differential Pairs dialog

Differential Pairs can have a unique name to help identify them. The default name will be constructed from the four pin names, but can be changed to a name of your own choice.

As well as Differential Pairs, you can also define Differential Pair Chains, these are discussed in more detail later on in the section.



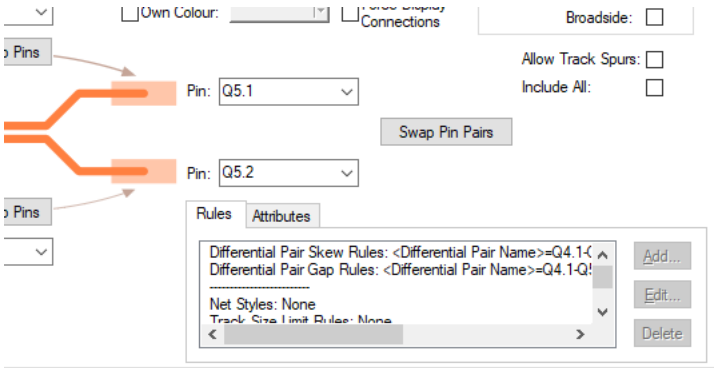
Choose the type of Differential Pair that the pairs are allowed to use; **Edge Coupled** (side-by-side) or **Broadside** (one on top of the other on different layers).

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

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

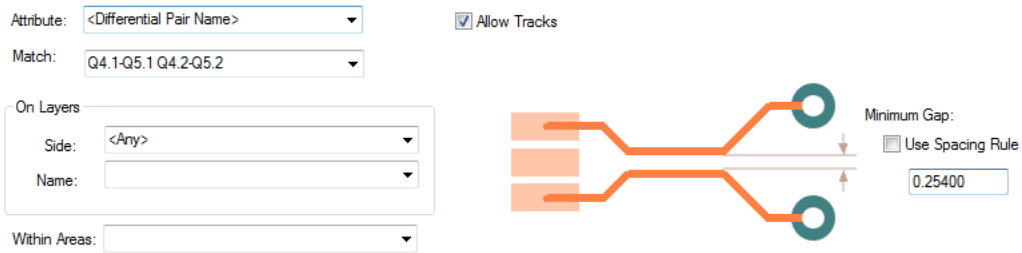
Differential Pair Attributes

Differential Pairs can 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.



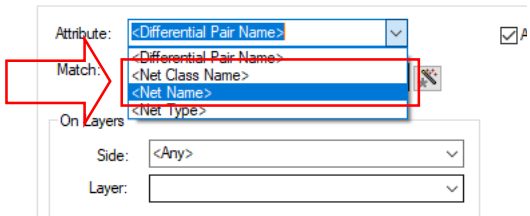
Differential Pair Gap

Once the Differential Pair has been created, you may create the gaps (separation) that the pair will use. These are defined in the **Technology** under the **Differential Pair Gaps** page.



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

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



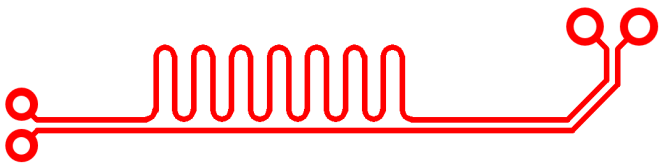
The gaps can be defined for layers or areas if required. Wildcards are allowed, for example Layer = Inner*

Allow Tracks - Use the **Allow Tracks** in conjunction with **Layers** and/or **Area** to create a rule that allows or disallows tracks on that layer or area. You could, for example, disallow differential pairs on Inner layers. This dialog allows you to create a rule for it.

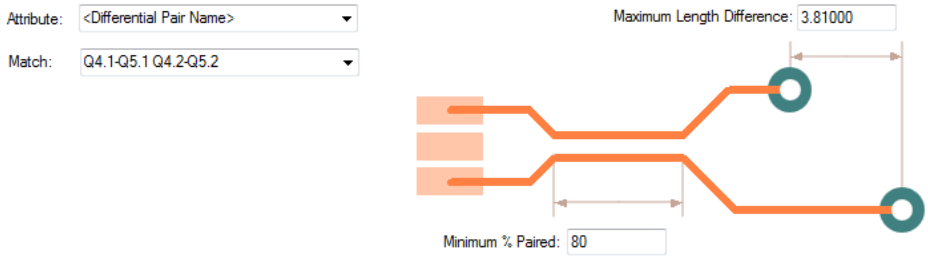
Minimum Gap - The Minimum Gap is the distance used when a differential pair is considered paired (i.e. it runs parallel). You can choose to use the **Use the Spacing Rules** to derive the gap, or specify a minimum gap. Note that the gap specified here can be smaller than that derived from the spacing rules.

Differential Pair Skew

The **Differential Pair Skew** page is used to define the **Minimum Percentage** pairing of tracks. An additional rule defines the **Maximum Length Difference** between the Differential Pair tracks. This defines the 'skew' between the two tracks in the pair.



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



The appropriate skew rule is found for each differential pair by working down the list of rules until the first match is found. Hence, the order of the rules is important. A match is when the specified **Attribute Name** and **Match Value** match an attribute of a differential pair, or an attribute on both nets of a differential pair. So the differential pair or nets must have the given attribute, and it's value must wildcard match the value. In particular, you can match the inbuilt attribute <Differential Pair Name>.

Min % Paired specifies the minimum percentage of the total track length which must be paired. Paired means that it should be parallel and at the gap specified by the **Differential Pair Gap** rule, or otherwise, the track to track **spacing rule**. The percentage is calculated using unadjusted lengths (see below for adjustments applied for length difference).

Max Length Diff specifies the maximum difference (or skew) of the total track length of each side of the differential pair. This length includes any length adjustments caused by **Track Length Factor**, **Layer Change Length** or **<Pin Package Length>** Attribute on a pad.

Skew can also be achieved when matching Differential Pairs to other Pairs and the pattern can also be adjusted for a single track skew. These features are both covered later on in this manual under *Pair to Pair Match Lengths* (directly below) and under *Chapter 7, Serpentine Routing and Serpentine Skew Modes on Differential Pairs* respectively.

Pair to Pair Match Lengths

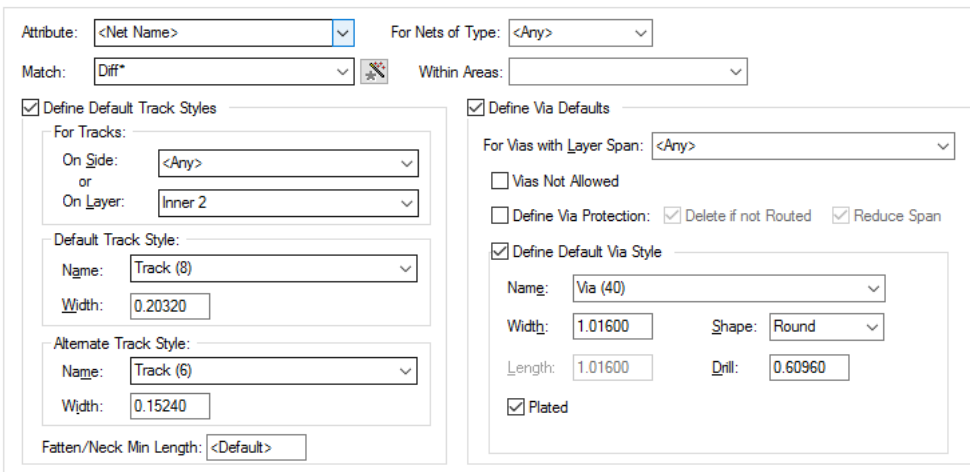
If you need to match a pair of Differential Paired tracks against another set or multiple sets, then you can use the **Track Length Match Rule**. This allows you to define multiple Differential Pairs against each other.

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

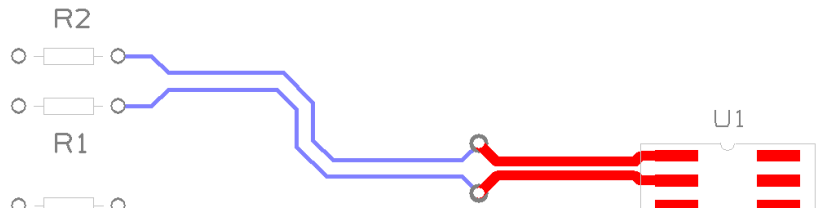
Defining Differential Pair Track Styles

As with other nets, Differential Pair track thicknesses and Via sizes are defined in the **Net Styles** dialog.

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	V Prot
<Net Name>	Diff*				Inner 2	Track (8)	Track (6)	<Default>		Via (40)	
<Net Name>	Diff*			Inner		Track (8)	Track (10)	<Default>			
<Net Class Name>	Diff					Track (10)	Track (8)	<Default>		Via (40)	
<Net Class Name>	GND					Track (55)	Track (25)	<Default>		PadStyle1	
<Net Class Name>	GND2					Track (55)	Track (25)	<Default>		PadStyle1	
<Net Class Name>	HS					Track (10)	Track (15)	2.54000		Via (40)	
<Net Class Name>	HS3					Track (55)	Track (25)	<Default>		PadStyle1	
<Net Class Name>	Power					Power (50)	Power (25)	<Default>		Via (50)	
<Net Class Name>	Sig2					Signal (8)	Signal (6)	<Default>		Via (40)	

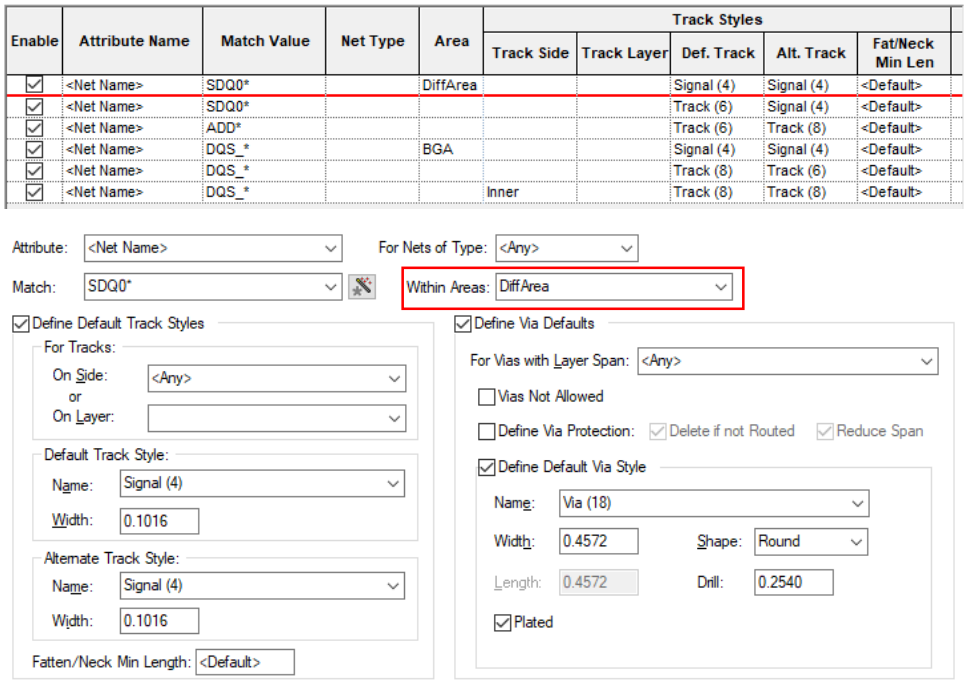


As well as defining the Track and Via styles to be used, you can also define how the Differential Pair track style will behave on other layers or within areas. This is normal track and via style definition but also equally applies to Differential Pairs where this is critical.

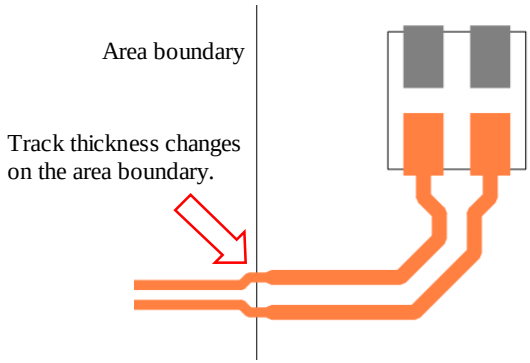


Editing Differential Pairs into Areas

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

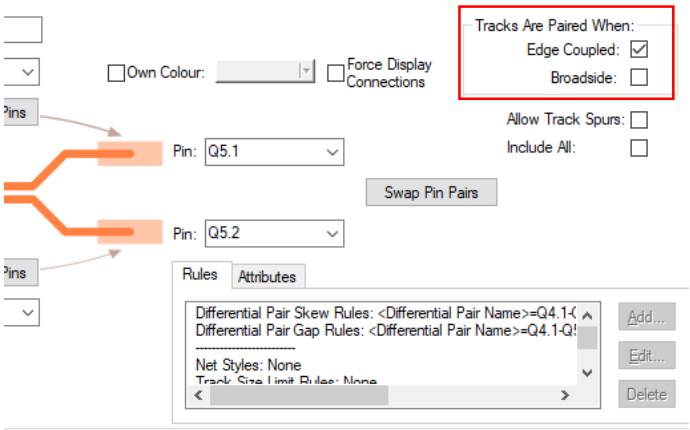


As the Differential Pair enters the named **Area**, it changes to the styles defined in the **Net Styles** dialog.



Using Edge Coupled or Broadside Differential Pairs

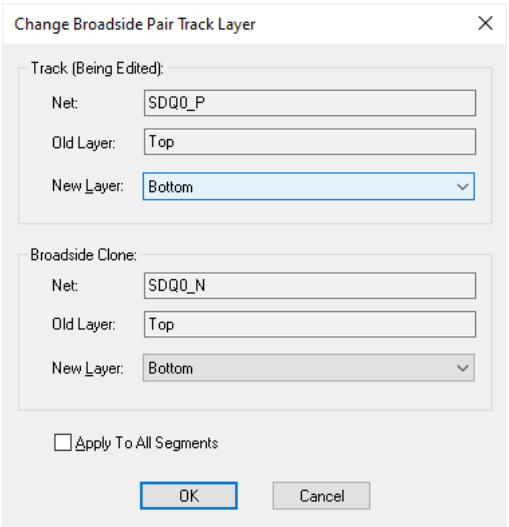
You can define how the Differential Pair tracks are 'paired'. **Edge Coupled** are the usual Differential Pairs, the edges of the two tracks separated by the specified gap and appear on the same layer side-by-side. **Broadside** Differential Pairs are paired vertically, the tracks laid on top of each other on different layers. It is possible to allow both types of pairing on a differential pair. The choice of which type to use will depend on your design and the technology being used. Usually, you would use one or the other method, but Pulsonix also allows mixing the track pairing methods.



For **Broadside** only coupling, the gap can still be provided and will be used for the minimum separation of the tracks when they are on the same layer, for example when routing out of surface mounted pads but this will not contribute to the % paired.

Adding Broadside Differential Pair Tracks

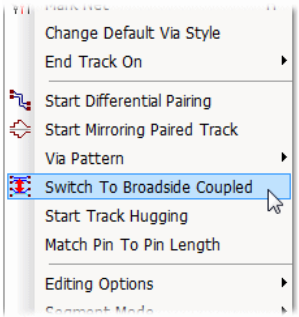
When adding a Broadside Differential Pair track, using **Start Differential Pairing** will prompt for the **Broadside Clone Layer**. This is the layer that the 'copy' of the track being added will go on.



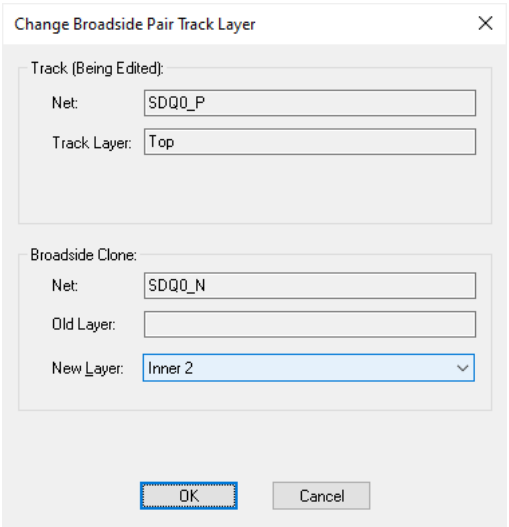
When using **Start Mirroring Paired Track** if both tracks are attached to surface mounted pads, mirrored tracks will be added, but you will not be able to start pairing yet and the defined gap will be maintained. If you start mirroring from through-hole pads or vias, you will be asked for the layer for the clone and the mirrored track will change to that layer. The mirrored tracks will be allowed to come together and when this happens a click will start broadside pairing, one track on top of the other.

Switching between Differential Pair types

When adding differential pair tracks that allow **both** coupling types, you can switch between them using **Switch To Broadside Coupling** or **Switch to Edge Coupling** from the context menu.



This can only be done when not actually adding paired tracks, so you have to un-pair, change coupling type and re-pair again. When switching to Broadside coupling you will again be asked for the layer for the broadside clone track.



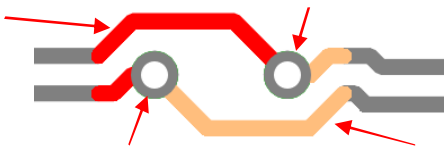
For Broadside coupling the paired length will be calculated where one track is completely over the other, but if both edge and broadside coupling is allowed, both types of paired sections will be added together to give the total figure.

Use copy of existing Differential Pair Via Pattern

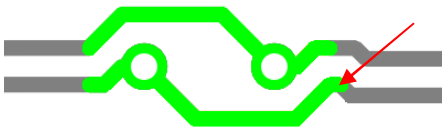
You can choose to use a copy of an existing Differential Pair Via pattern when changing layer whilst adding paired tracks. You may have already created a pattern that you want to reuse, this can be copied and reused.

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

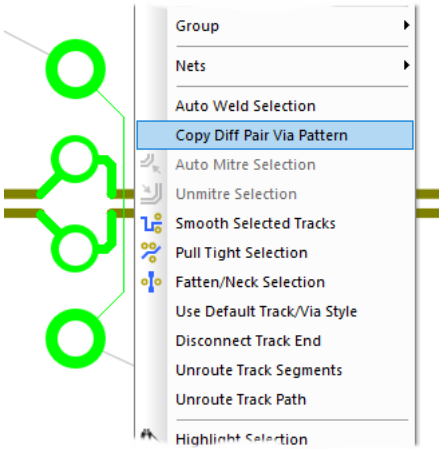
In the example below, selecting the red and yellow tracks plus the vias would be suitable:



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

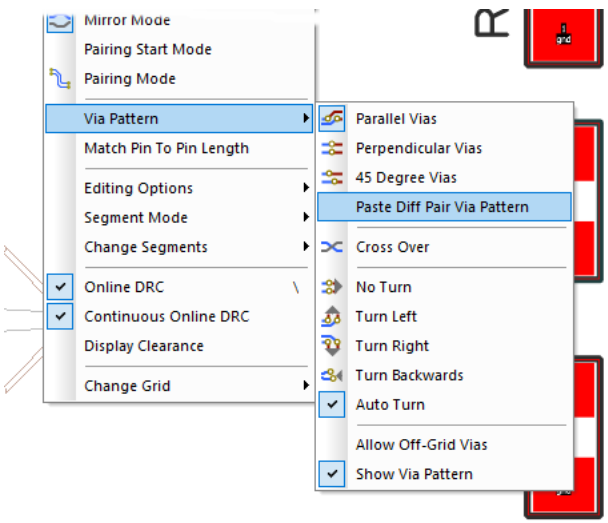


If the pattern is as required, you can then use **Copy Diff completion Pair Pattern** from the context menu to save it to a special location on the clipboard.



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

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



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

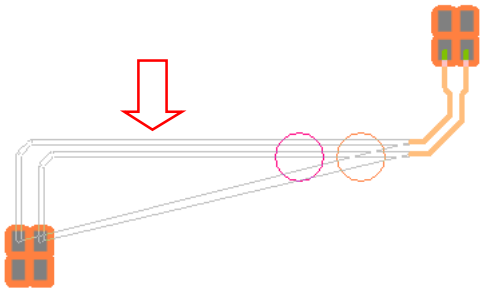
The via pattern option **Paste Diff Pair Via Pattern** will be remembered and will be used in future whilst the pattern is available on the clipboard. Changing layer back to the start layer will use the same pattern rotated about 180 degrees.

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

Legal Completion Path for Differential Pairs

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

This mode also now works when in **Mirror** mode.

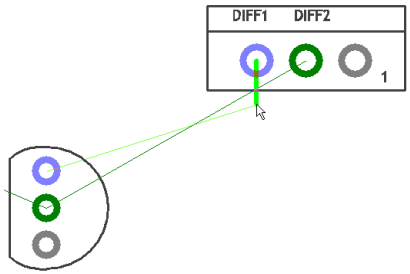


Routing a Differential Pair

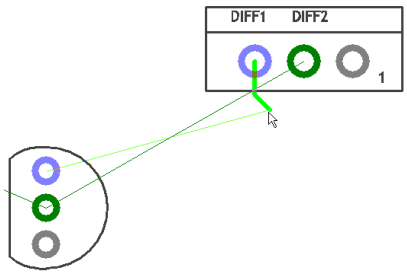
This is how you route Differential Pair tracks:

▶ To route a differential pair

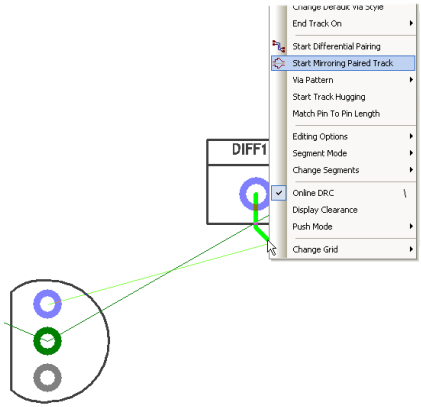
1. There are a number of ways to create Differential Pairs. We will use the more straight forward method to start with.
2. You would have set up your **Differential Pairs** already using the **Technology** dialog.
3. Begin to start routing from one of the pins in the set (double click on the net). Don't add any corners, the Start Mirror Paired Tracks option only works from the initial track segment off the source pad.



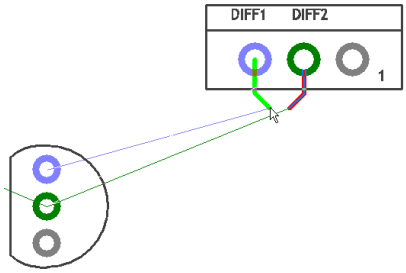
4. In this example, we will move the cursor inwards to create a 45 degree mitre as well. The routing mode is already enabled in **Angled (45)** mode (from the **Options** dialog and **Edit Tracks** page).



5. Right click the mouse and from the context menu, select **Start Mirroring Paired Tracks**.

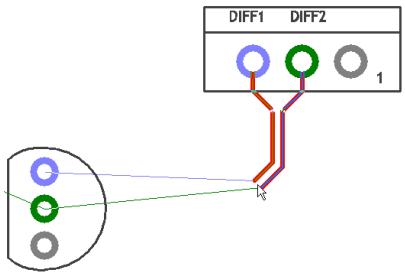


6. This will create the initial paired tracks using a mirror image of the first track you routed to start with.

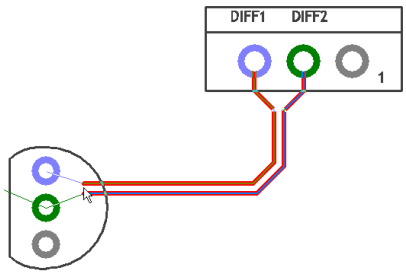


7. Moving the cursor closer to the paired track will now bring the tracks together using the **Track to Track** spacing rule defined in the **Technology** (or any additional **Net Class** or **Net Styles** rules or **exception rules** added to the net).

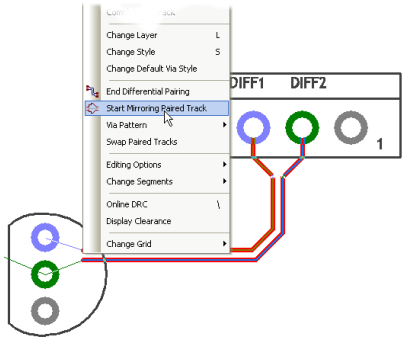
8. Move the mouse closer to the second track and click to add the first corner, it will not move closer than the **Spacing Gap** defined.



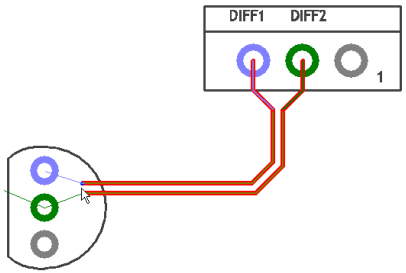
9. Move the cursor towards your end target, click once to add corners just like regular track editing. Changing layers will be demonstrated a bit further on.



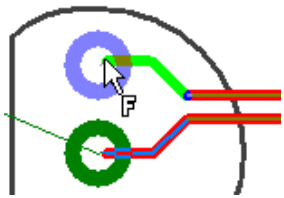
10. Position the cursor mid-way between the target pads.
11. Right click the mouse and from the context menu select **Start Mirroring Paired Tracks** again.



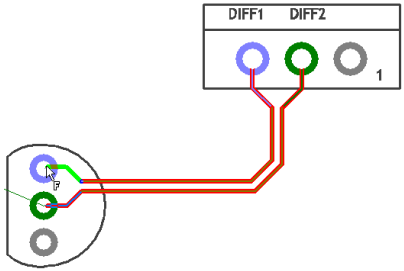
12. This will instruct this option that you wish to now finish the track pairing in a defined way.



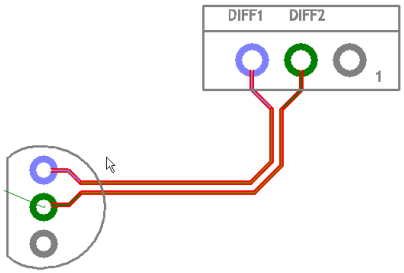
13. Assuming you are still using the **45 Degree Segment Mode**, moving the one track towards the target the other track will ‘mirror’ this image.



14. With the cursor now over the target, the **Finish** marker will be shown.



15. Click to finish.

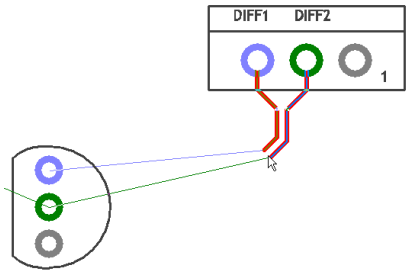


16. The completed result looks like the above example.
17. These two tracks will now behave as a differential pair and will use the rules provided. **Design Rules Check** etc. will also know about these tracked pairs.

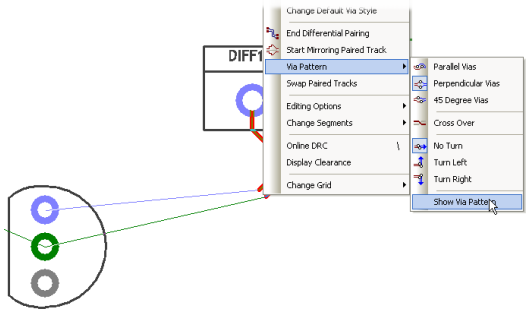
Changing Differential Pair Layers

▶ To change layer during differential pairing

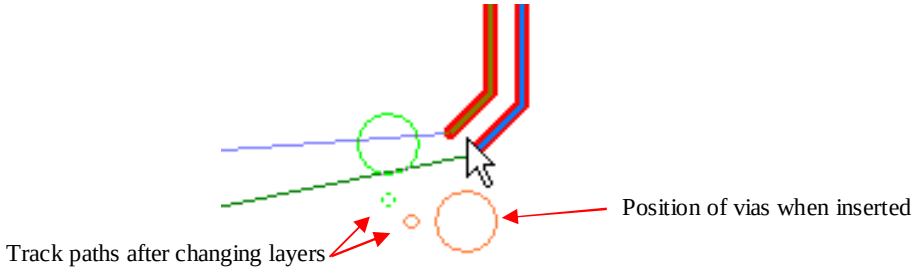
1. During differential pairing you may wish to change layers.
2. To do this, right click and select **Change Layer** from the menu or the shortcut <**L**>. as normal
3. Because you are already in Differential Pairing mode, you can opt to show the position of the vias and the next track segments.
4. Once your routing has been started (using **Start** or **Mirror Differential Pairs**), right click.



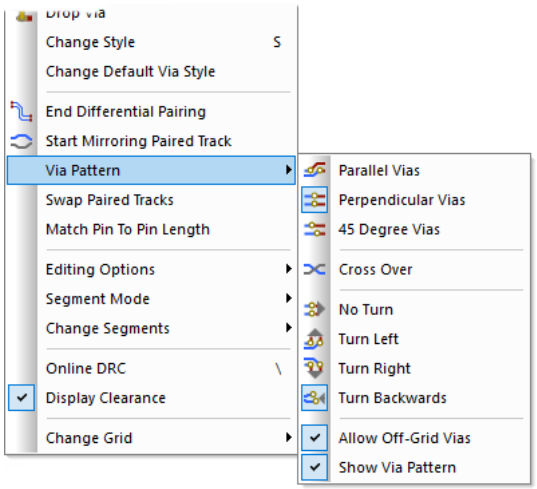
5. To show the vias, select the **Via Pattern>** option from the context menu.



6. At the bottom of the sub-menu, select **Show Via Pattern**.
7. Now when you route, the via pattern and track positions will be shown. These will move dynamically with the routing to show their position if used.
8. These are shown as outline shapes for clarification. The larger circles will indicate the Via position, the smaller circles indicate the next start position of the tracks on the other layer.



9. The via and track positions will depend on the via and track grids and the **Spacing Rules** defined in the **Technology**.
10. From the context menu, you can also select the type of via pattern to be generated during a layer change.

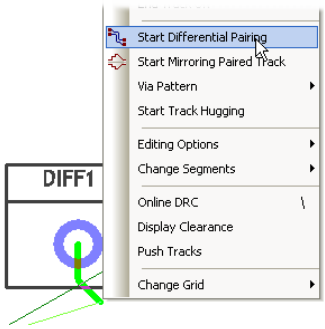


11. The patterns will be applied to subsequent routing after selecting from this menu so it is possible to use multiple-pattern types if required.

Alternative method for starting Differential Pair Routing

▶ Alternative method of starting differential pair routing

1. An alternative method to create Differential Pairs is to start the routing, then from the context menu, select **Start Differential Pairing**.



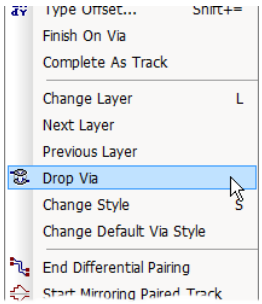
2. Any routing in this current session you add now will be paired.
3. Use the **End Differential Pairing** mode form the context menu to exit this mode.

Differential Pair Routing Functionality

Drop Via

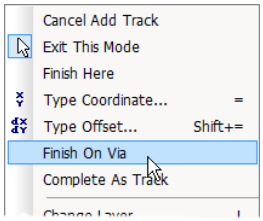
When adding a PCB track (or differential paired track), there is a new option on the context menu to **Drop Via**. This operates the same as **Change Layer;** adding a via at the end of the track but for this option, not changing the layer of the new track. A via is added along the track as you edit,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



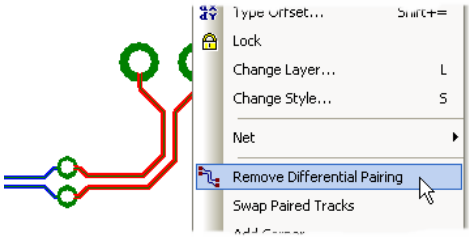
Finish On Via

The **Finish On Via** option is 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 also available when adding differential paired tracks.



Removing Differential Pair Routing

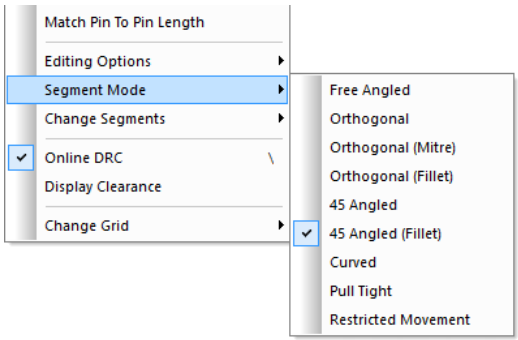
A paired section of track (or a selected part of it) can be unpaired using the context menu command **Remove Differential Pairing** whilst in select or edit modes on the selected track.



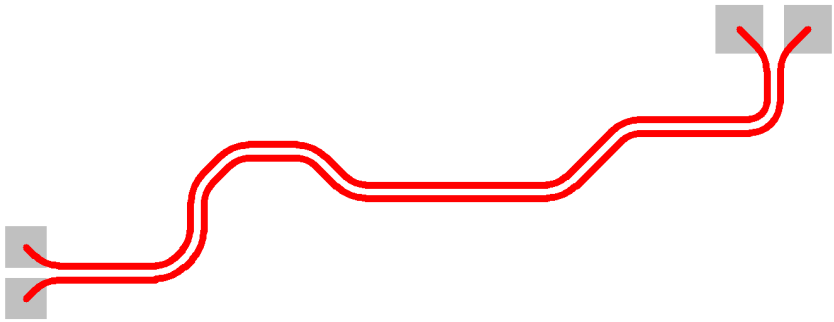
You can create any number of paired sections of track along the path but you must complete the gaps manually.

Differential Pair Routing – Fillet Mode

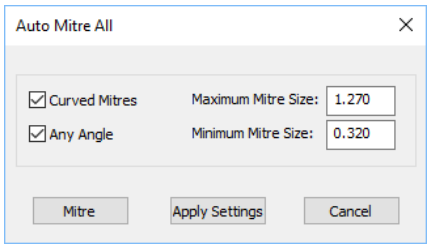
While routing Differential Pairs, you can switch **Segment Mode** to use one of the two routing modes that allow you to add Fillets: **Orthogonal (Fillet)** and **45 Angled (Fillet)**.



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

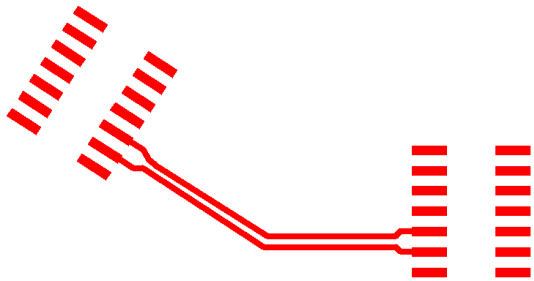


Fillets can also be applied to Differential Pairs on existing tracks that are already routed by selecting the corner and using **Edit Mitre** from the context menu. The Mitre used (45 degree or Fillet) will depend on the setting using from the **Tools** menu, **Auto Mitre**, **Auto Mitre All**. Both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

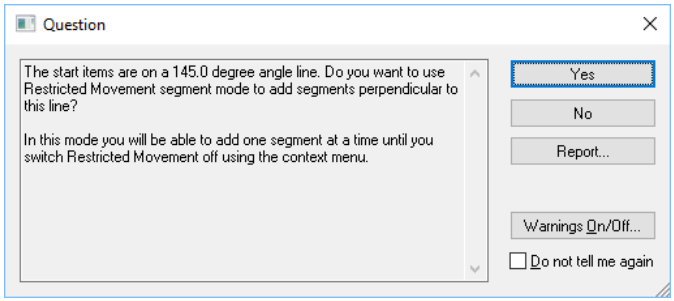


Mirror Differential Pair routing from Angled Components

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



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



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

Using this mode

After cornering, the mode stays and you can turn in 45 degree steps relative to the start angle. When close together and you click, it will enter adding paired tracks still in the restrictive mode. Add a segment or two and then use the context menu to switch **Restrictive Movement** off and now you can add the segments in your normal segment mode.

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

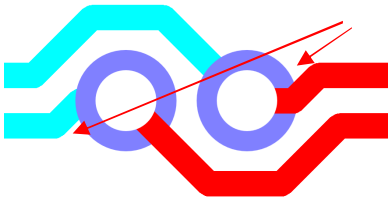
Note: **Restrictive Movement** mode always allows you to turn a multiple of 45 degrees now. In 9.1 this only worked if you set the angle to zero.

Acid trap removal using Minimum Allowed Gap Spacing Rule

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

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

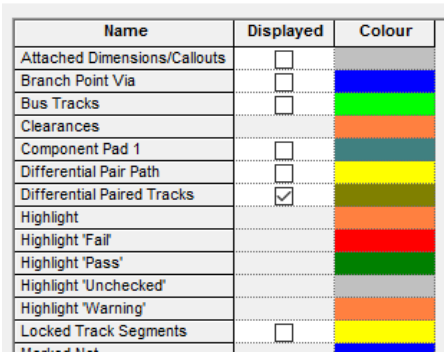
Potential acid traps



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

Defining Differential Pair Colours

You can draw **Differential Paired Tracks** in a different colour using the **Colours** dialog and **Highlights** page. These are in addition to the normal **Track** and **Net** colours which can also apply to Differential Pairs.



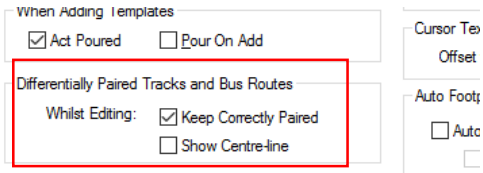
Name	Displayed	Colour
Attached Dimensions/Callouts	☐	
Branch Point Via	☐	
Bus Tracks	☐	
Clearances		
Component Pad 1	☐	
Differential Pair Path	☐	
Differential Paired Tracks	☑	
Highlight		
Highlight 'Fail'		
Highlight 'Pass'		
Highlight 'Unchecked'		
Highlight 'Warning'		
Locked Track Segments	☐	

Differential Paired Tracks defines the colour of paired tracks in a Differential Pair.

Differential Pair Path defines the colour of the pad to pad path of tracks and connections in a Differential Pair. An individual Differential Pair colour can be defined from within the **Technology** dialog and **Differential Pairs** page.

Differential Pair Routing Options

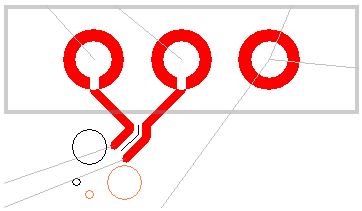
Check boxes are available on the **Options** dialog and **Interaction** page are available for **Differentially Paired Tracks and Bus Routes**:



Correctly Paired

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

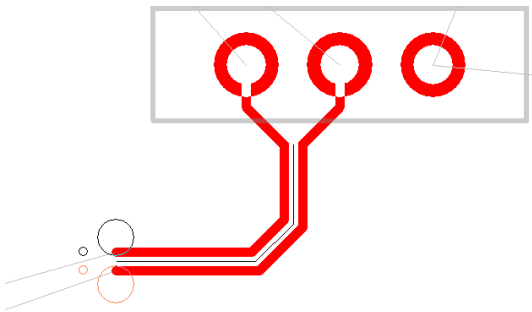
Without **Keep correctly Paired** checked. Tracks are allowed to be edited in illegally.



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

Show Centre-line

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

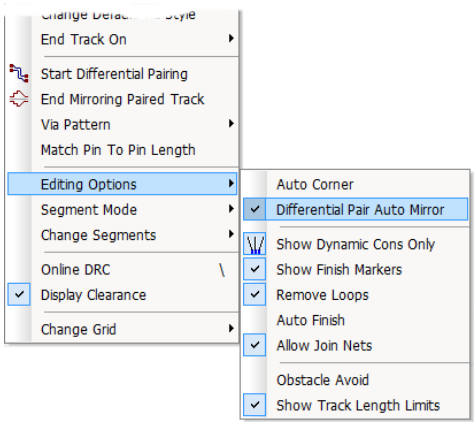


The line is removed as soon as the edit, move or move corner is complete on the Differential Pairs.

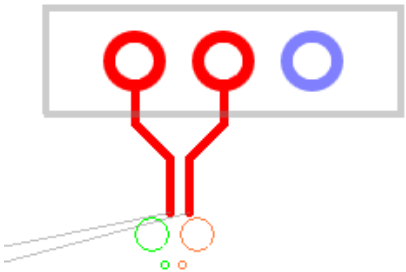
Automatically starting Diff Pairs as Mirrored

You can define if a Differential Pair is mirrored by default when the pair routing is started.

The option is on the context menu when in Insert Track for a Differential Pair. It is also available in the options sub menu on the context menu when editing the track.



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

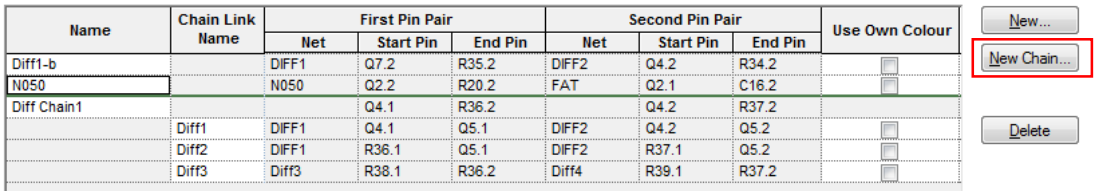


Differential Pair Chains

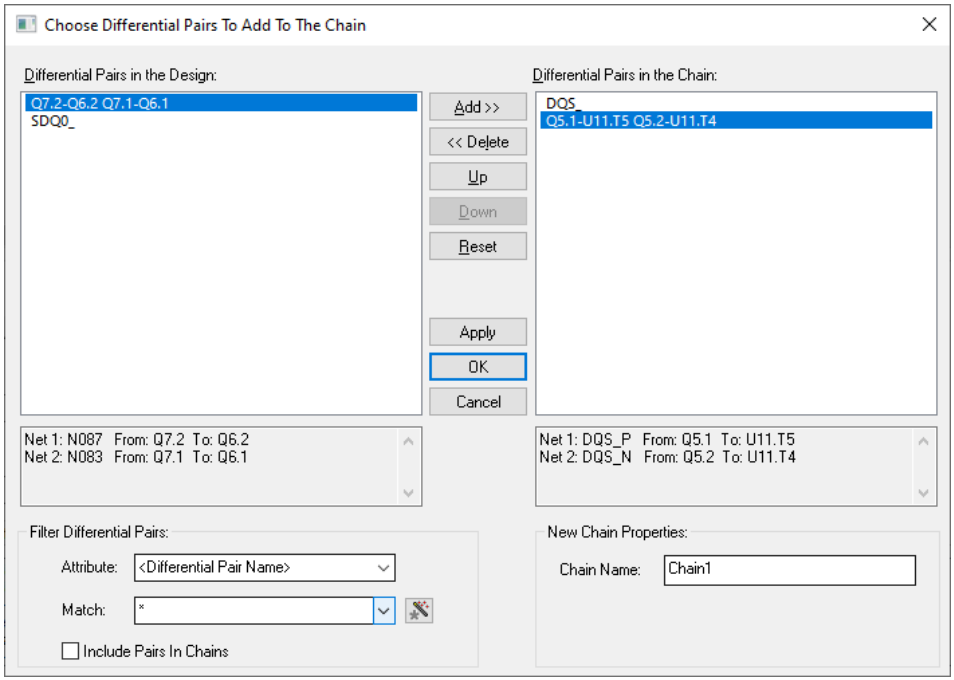
Why Diff Pair Chains?

Differential Pairs can 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.



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:



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

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

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

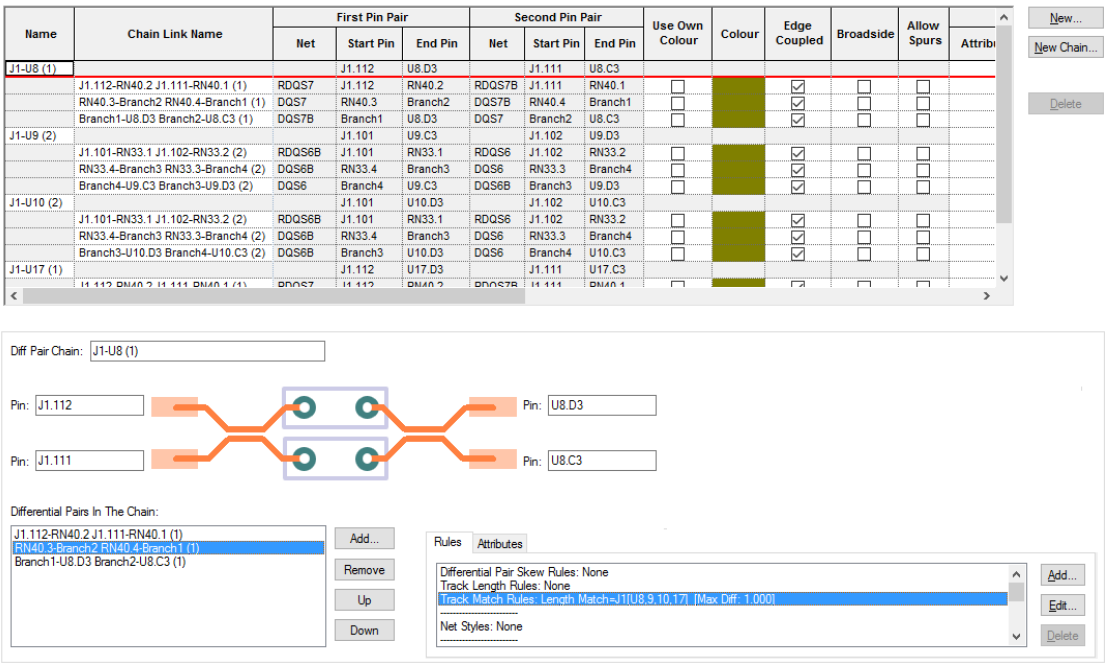
The **Chain Name** text box allows you to set a name for the newly created Differential Pair chain when you exit this dialog. If this box is left blank, then a default unique chain name will be generated for you.

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

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

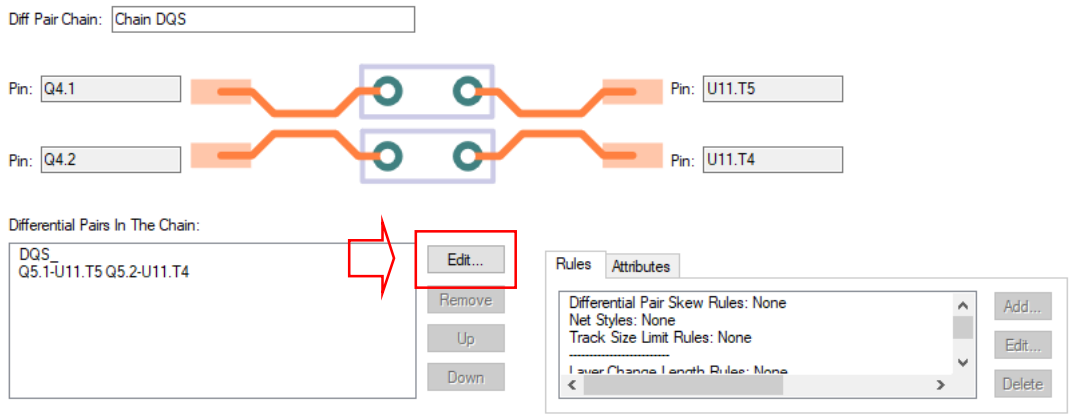
	Name	Chain Link Name	First Pin Pair			Second Pin Pair		
			Net	Start Pin	End Pin	Net	Start Pin	End Pin
	Chain DQS			Q4.1	U11.T5		Q4.2	U11.T4
		DQS_	DQS_P	Q4.1	Q5.1	DQS_N	Q4.2	Q5.2
		Q5.1-U11.T5 Q5.2-U11.T4	DQS_P	Q5.1	U11.T5	DQS_N	Q5.2	U11.T4
	Q7.2-Q6.2 Q7.1-Q6.1		N087	Q7.2	Q6.2	N083	Q7.1	Q6.1
	SDQ0_		SDQ0_N	U5.1	U6.2	SDQ0_P	U5.2	U6.1

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



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**.

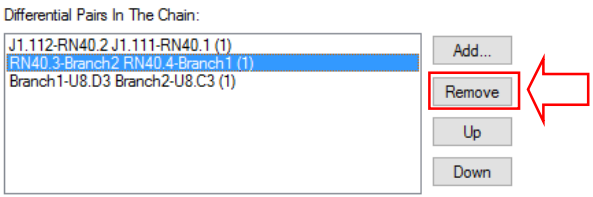


Once you have created the chain, selecting it in the grid still displays the chain controls, but the **Add…** button has been replaced with an **Edit…** button. Selecting it displays the New Chain dialog where you can edit the chain.

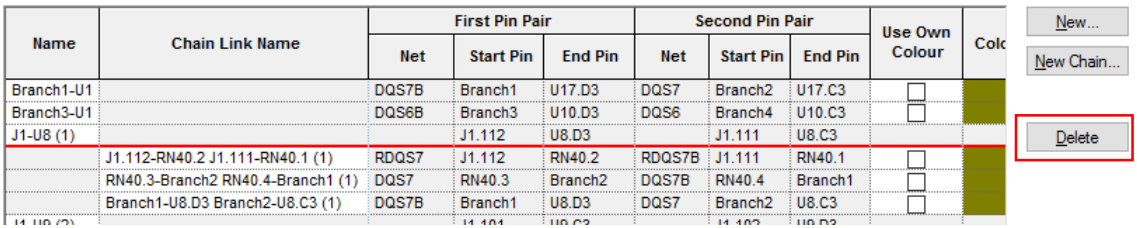
Deleting Differential Pair Chains

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	☐	
Branch3-U1		DQS6B	Branch3	U10.D3	DQS6	Branch4	U10.C3	☐	
J1-U8 (1)			J1.112	U8.D3		J1.111	U8.C3		
	J1.112-RN40.2 J1.111-RN40.1 (1)	RDQS7	J1.112	RN40.2	RDQS7B	J1.111	RN40.1	☐	
	RN40.3-Branch2 RN40.4-Branch1 (1)	DQS7	RN40.3	Branch2	DQS7B	RN40.4	Branch1	☐	
	Branch1-U8.D3 Branch2-U8.C3 (1)	DQS7B	Branch1	U8.D3	DQS7	Branch2	U8.C3	☐	

<Component Pin Name> Attribute

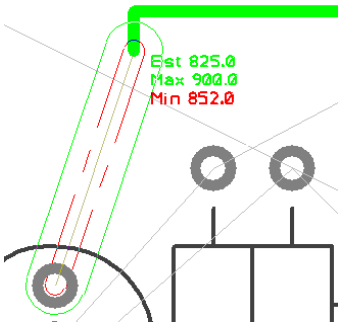
To make this chain easier to locate, a pin attribute is available **<Component Pin Name>,** for example **U12.6**. Use this if you need to find the pair that connects to that pin. This attribute is available throughout Pulsonix.

Chapter 5. Length Based Rules

Length Based Rules

Interactive Net Length Indicators

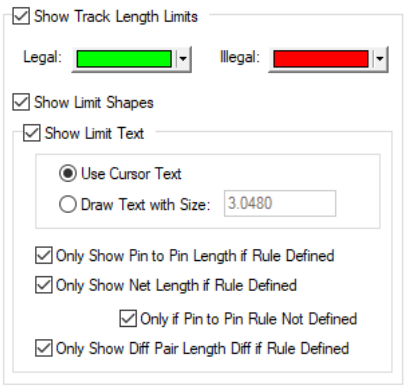
The interactive net length indicators allow you to display length rules in the PCB Design. The rules can be defined in the Schematic and passed through to the PCB during the translate stage. In the PCB design interactive track editing displays the rules as restriction boundaries and a head-up display.



Options to Display Track Length Indicators

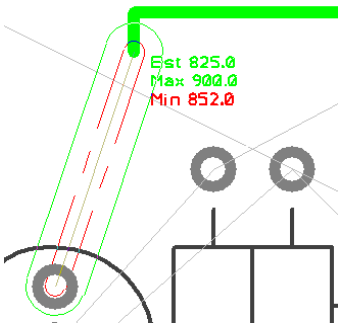
Options for displaying Net Length Rules

The colours for the **Legal** and **Illegal** track lengths can be defined in the **Options** dialog under **Track Length Limits**. Using **Show Limit Shapes**, the display of the length indicators can also be switched on and off.



Track Length Limit Shapes

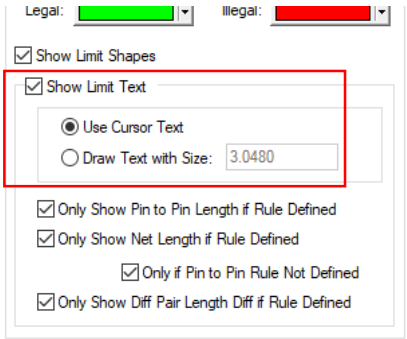
The Min and Max values can be displayed as numerical figures but also shapes. Coloured shapes indicate the extents of the limits.

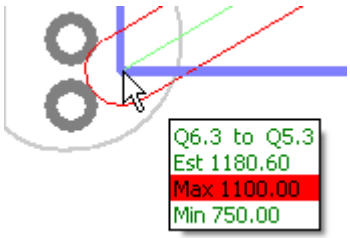


In the illustration above, the red box 'inside' the green box indicates that the minimum length has not yet been met, the green box indicates the extents to which the track must be routed to meet this. Once the min limit has been met, the red box then changes to be outside of the green shape to indicate the outer max limit of the track. These shapes change dynamically as the track us routed.

Track Length Limit Text (head-up display)

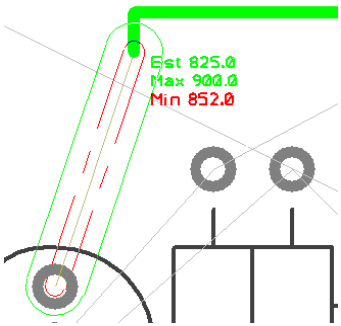
As well as the visual indicators for legal and illegal values, the actual rule values used can also be displayed using the **Show Limit Text** check box.



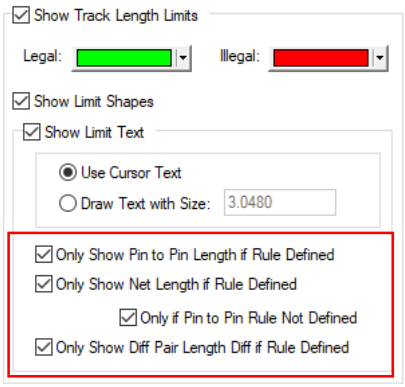


The **Use Cursor Text** option displays the limit text similar to a tooltip always on top of your design keeping it more legible especially in dense areas of the design. You can alter the distance the text box is from the cursor using the **Options** dialog and **Track Length Limits** page. You can also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

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



'Only Show' Rules



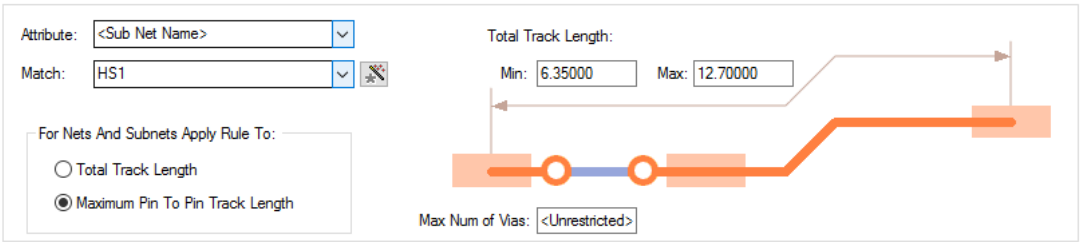
Use the three **Only Show...** text boxes to decide if you always want to show the appropriate length, or only show the text if there is a maximum or minimum length limit rule defined. If you are only showing the net length if there is a rule defined, you also choose if you want to display this as well as the pin to pin length or only if the pin to pin length is not displayed.

The Pin to Pin length is the sum of all tracks and unrouted connections in the path between the pins. All unrouted connections have their length estimated assuming they will be routed using the angled segment mode, except the connection at the end of the track being added that might instead use the orthogonal length or direct connection length depending on the current segment mode.

Track Length Rule

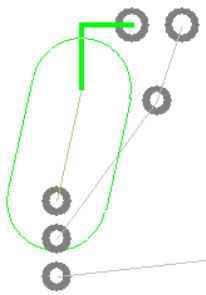
The **Track Length Rules** page enables you to control the length limits of the overall track length and the pin to pin lengths required. You can define **Min** and **Max Track Lengths which** can be applied to any Net-based items and as Attributes to the items. Within this rule there is a sub-category 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.

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	☐	☑
Track_Length	HS	5.08000	81.28000	2	☑	☐
<Net Class Name>	HS1	63.50000	76.20000	<Unrestricted>	☑	☐
<Net Class Name>	HSE	8.00000	14.00000	1	☐	☑
<Net Class Name>	PAIR	38.10000	45.72000	<Unrestricted>	☑	☐
<Net Class Name>	Sig2	19.05000	27.94000	2	☐	☑
<Net Class Name>	Signal	2.54000	17.78000	<Unrestricted>	☑	☐
<Sub Net Name>	HS1	6.35000	12.70000	<Unrestricted>	☐	☑
<Sub Net Name>	Pin_Order	19.05000	27.94000	2	☐	☑

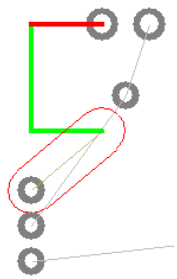


When the **Track Length rule** has been defined for a **Net item** or **net attribute,** during editing, a coloured graphical shape is displayed indicating the track length limits based on the rules defined.

Separate shapes are drawn to indicate minimum and maximum track length rules.

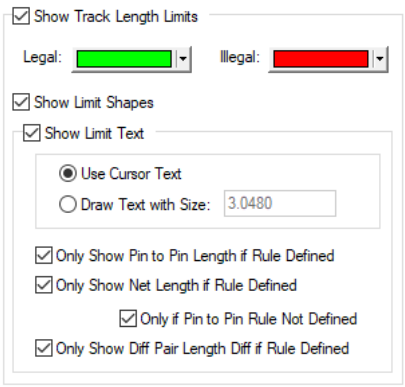


The large oval area shows the length of the track is within the minimum track length set.



The small oval area shows that the maximum length rule has been exceeded.

Parameters for defining the display of the length rules can be set in the **Options** dialog and **Track Length Limits** tab.



Show Track Length Limits

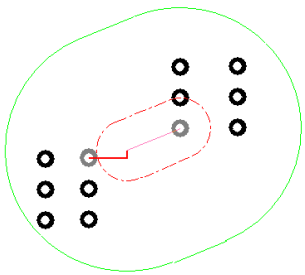
Check this option to show length limit shapes or text in general. Whilst using **Edit Track**, the length limits can also be turned on or off using the shortcut menu **Editing Options - Show Track Length Limits**.

The text and shapes are colour coded to show **Legal** and **Illegal** values. Green and Red are the defaults, but you can use any colours.

When you are within the maximum track length limits, the limit shape or text is drawn in the legal colour. When the maximum limit is exceeded, the shape or text is drawn in the illegal colour and the shape will grow as the excess increases. Minimum limits are always drawn in the illegal colour in a dashed style, until the minimum is exceeded, when the shape is not drawn.

Show Limit Shapes

Limit Shapes give you a visual indication of how far the track can extend and still be legal. You should be able to legally reach any point between the inner and outer shapes



Show Limit Text

The limit text gives information about the current estimated length and length limits that apply.

Use Cursor Text, uses the system defined tooltip font and text size. The text has a background box drawn in the current background colour.

Draw Text with Size, inbuilt font drawn at the given text size. The size is given in the current design units, but you can specify other units on input (e.g. 3mm). The text has no background so improves visibility of surrounding items at the expense of readability.

You can alter the distance the text box is from the cursor using the **Interaction Option - Cursor Text Offset From Cursor**. You can also use the **Reposition Cursor Text** command (available from the context menu - Editing Options), which shifts the cursor text into the next quadrant.

Pin to Pin Length

The Pin to Pin Length is the track length between the two *nearest* **Component Pins** or **Branch Point** by tracking a path from each end of the connection being edited. This length will follow through vias. The names of the two pins will be show along with the length.

You can choose to **Only Show Pin to Pin Length if a rule is Defined** which limits the length of the Pin to Pin connection.

Net Length

The Net Length is the total track length within the current net.

You can choose to **Only Show Net Length if a rule is Defined** which limits the length of the Net. More specifically, you can choose to Only Show Net Length **If a Pin to Pin Rule** is defined which limits the length of the connection.

Differential Pair Length

The length of a Differential Pair is the track length of the path between the start and end pins of the selected *side* of the pair.

You can choose to **Only Show Differential Pair Length if a rule is Defined** which limits the length of the Differential Pair.

Length Rules

You can define **Track Length Rules** (explicit length limits) and **Track Length Match Rules** (length Limits defined by comparing with other lengths) in the technology.

You can also define track length limits on a **Pin Network Part**.

You can also show pin to pin length limits without rules being defined in the technology by using the **Match Pin To Pin Length** option when editing tracks.

How Limits are Calculated

The connection you are editing may be part of a number of length limited items. The lengths and applied rules are evaluated for all of these, and the most limiting rules are found. The rule which most limits the minimum length may not be the same rule which most limits the maximum length. The two most limiting rules are the ones which are displayed. It could be possible that the minimum and maximum are such that it is impossible to satisfy both without making wider changes to the design.

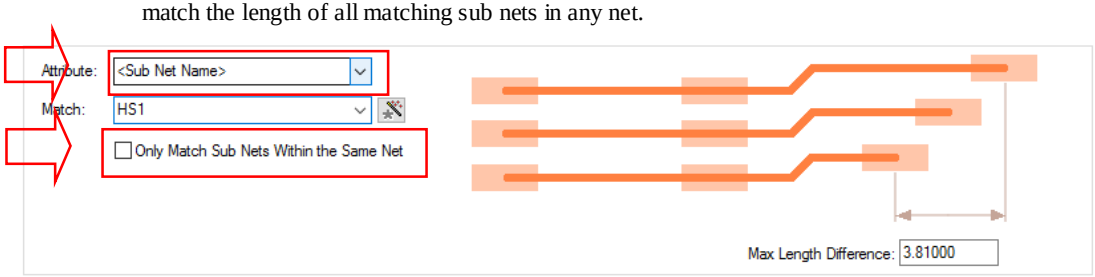
How the Lengths are Calculated

The lengths are calculated by adding the lengths of all the tracks which make up the Total or Pin to Pin Length of the length limited item (as required for the rule being evaluated). Unrouted sections, or trailing segments are estimated using a 45 degree plus orthogonal projection. Length adjustments are also applied using **Track Length Factor Rules**, **Layer Change Length Rules** and **<Pin Package Length> Attribute** on a pad.

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 displays 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.



Track Length Match Rule

The **Track Length Match** rule is used to define length differences between different nets. You could 'cluster' nets and associate them using an attribute name which can then be matched or you can use standard design items such as <Net Name>, <Diff Pair Name>, <Net Class>, <Signal Path Name> and <Sub Net Name>. You can also add a net attribute to provide a tag to add to any required nets.

Attribute Name	Match Value	Extra Delay Length	Max Length Difference
<Net Name>	Diff	0.00000	3.81000
<Net Name>	HS*	0.00000	12.70000
<Signal Path Name>	HSE	0.00000	2.00000
<Differential Pair Name>	DIFF*	0.00000	6.35000

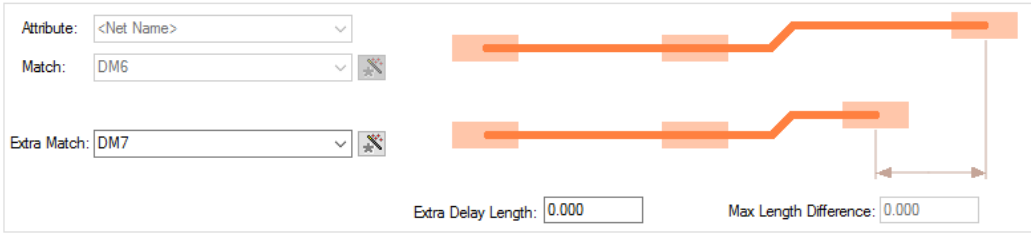
The **Match Length Difference** value is the maximum length difference (or skew) between any item in the match group.

Use the **Extra Match** button to add an extra match group to the currently selected group, this adds a new line to the grid. The **Attribute Name** cell is blank because it is the same as the first line, but you must add a different **Match Value**.

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	

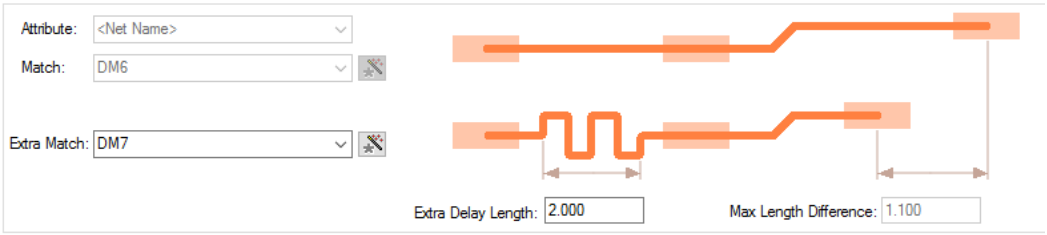


When the **Extra Match** rule is set to zero, this means simply ensure that the explicit nets are matched using the Max Length Difference. If, however, an explicit override rule is required, then the **Extra Delay Length** can be assigned (see below).

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 that the base item.

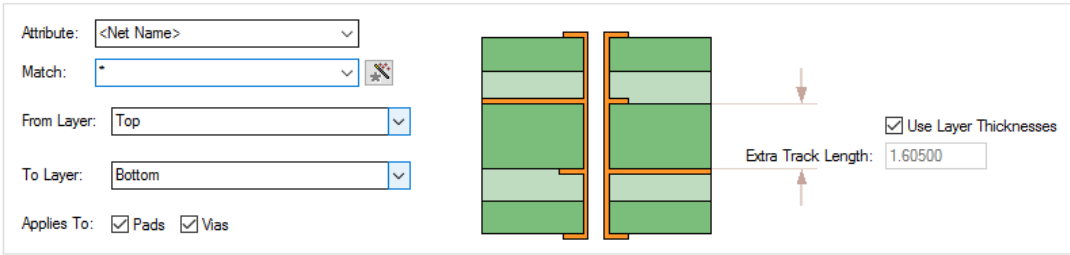
Attribute Name	Match Value	Extra Delay Length	Max Length Difference
Length Match	RDQ48-59	0.000	1.000
<Signal Path Name>	*Branch*	0.000	0.500
Length Match	J1[U8,9,10,17]	0.000	1.000
<Net Name>	DM6	0.000	1.100
	DM7	2.000	



Layer Change Length Rule

The **Layer Change Length Rule** allows you to add an extra length to vias or pads using a layer span rule. By default, track lengths are calculated without taking any account of this additional length. These rules allow you to specify this. This rule can be used in conjunction with the **Track Length Rules** or **Track Length Match Rules**.

Attribute Name	Match Value	From Layer	To Layer	Applies To		Extra Track Length	
				Pads	Vias	Use Layer Thicknesses	Length
<Net Name>	*	Top	Bottom	☑	☑	☑	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.

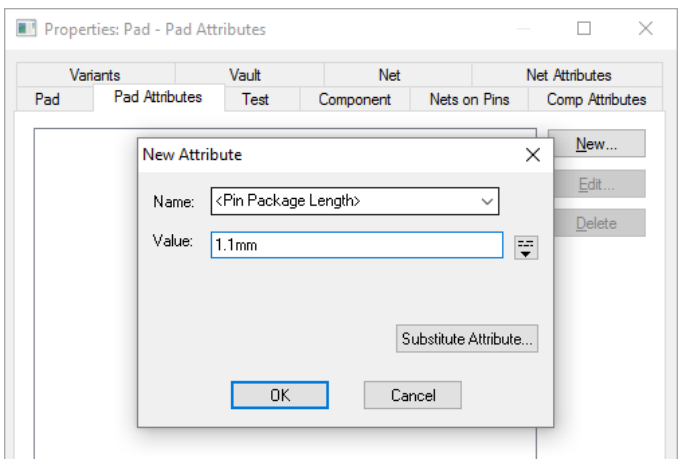
Extra Length Rule through Pin Package Attribute

As well as the general **Layer Change Length Rule** that can be added to layer spans, Pulsonix also provides you with a built-in **Pad Attribute** that can be used on a pad. The attribute <Pin Package Length> can be used to define a pin length or the internal length between two pins say, for a Signal Path 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 rules 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.

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.

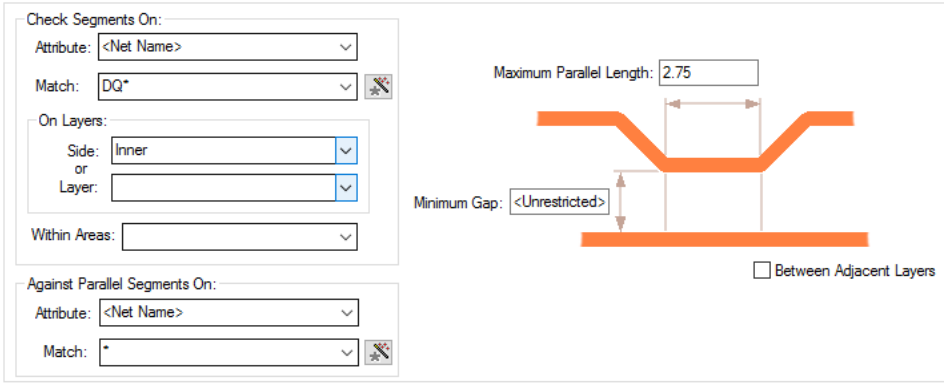


Chapter 6. Additional Rules

Track Parallel Segments Rule

A rule is available in **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>	*	☐	<Unrestricted>	3.00000
<Net Name>	DQ*	Inner			<Net Name>	*	☐	<Unrestricted>	2.75000



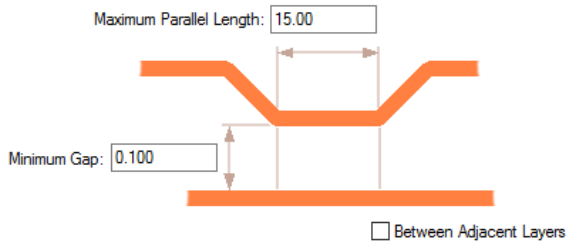
If the **Minimum Gap** is left undefined <Unrestricted>, it will use the default Track to Track spacing rule defined in the **Spacing Rules** dialog.

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>	*	☐	0.100	15.000
<Net Name>	DQ48				<Net Name>	*	☐	0.150	17.000

How the rule works

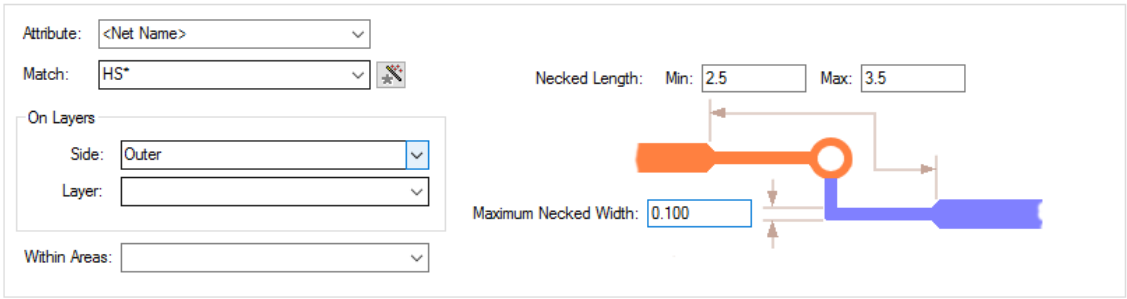
For the example below, the rule is: if the two tracks are within the minimum gap of 0.100mm, you cannot have a run of longer than 15.00mm



Necked Length Rule

Where designs require restrictions applied for the min and max lengths of necked lengths, these can be defined in the **Necked Length Rules** dialog.

Attribute Name	Match Value	Side	Layer	Area	Max Necked Width	Necked Length	
						Minimum	Maximum
<Net Name>	HS*	Outer			0.10000	2.50000	3.50000



Minimum & Maximum Necked Length defines the minimum and maximum lengths that a track can be necked in a single run before ending or returning to a width thicker than the Maximum Necked Width.

The **Maximum Necked Width** defines the maximum track width that is considered necked. This is an actual width regardless of the default track styles of the actual track style used.

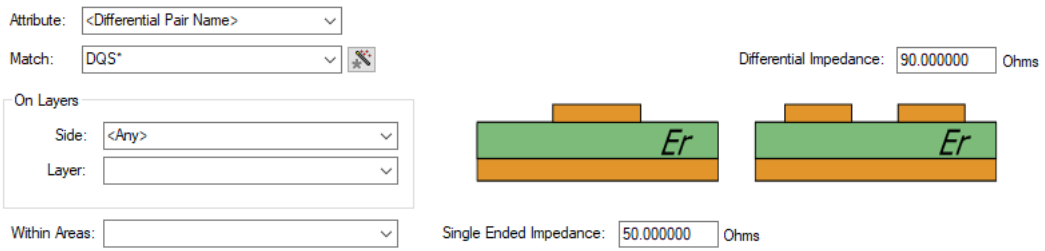
Track Impedance Rules

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

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

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

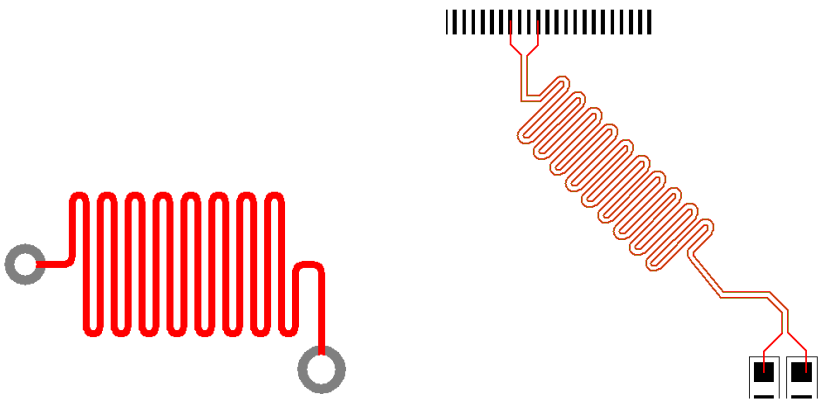
Enable	Attribute Name	Match Value	Side	Layer	Area	Single Ended Impedance	Differential Impedance
☑	<Differential Pair Name>	DQS*				50.000000	90.000000



Chapter 7. Serpentine Routing

Fixed Rule Serpentine Routing

For balancing the length of high speed nets is the insertion of track ‘length’ without introducing spacing errors. This is commonly known as **Serpentine Routing**. Serpentine Routing can be applied across 90 degree or 45-degree track segments. It can also be applied to differential pair routing (shown below, right).



You can select a track segment (or segments) and run the **Serpentine Routing** command from the context menu which prompts for the serpentine parameters. Parameters are defined for the amplitude and separation of each loop. You can also define the minimum number of loop cycles to insert, and also the amount of additional length required (otherwise it will do as much as possible).

Serpentine shapes and parameters are defined in the **Technology** dialog and **Serpentine**. If you require length rules to be applied to the tracks, then also other rules available such as **Track Length Rules**, **Track Length Factor Rules** and **Track Length Match Rules**.

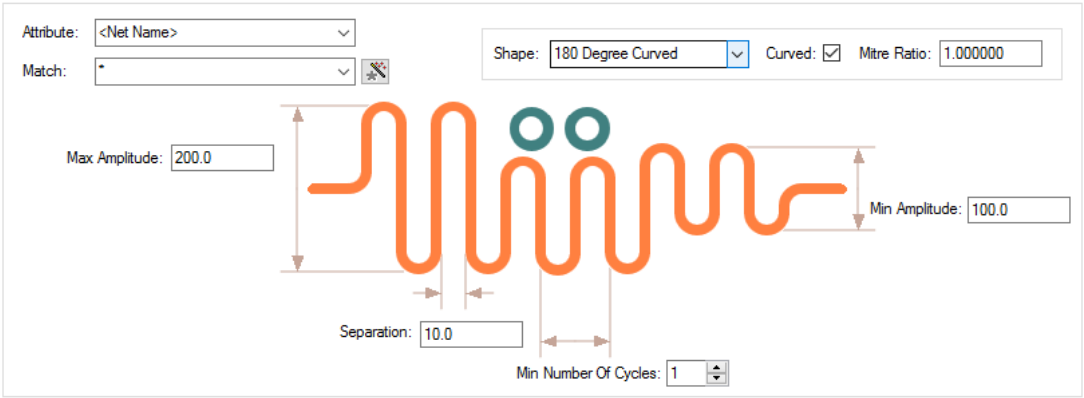
Serpentine Routing Shapes

Using the **Shape:** drop down list, you can choose from different styles of serpentine shapes available or you can customize the ones available to create a variation of these.

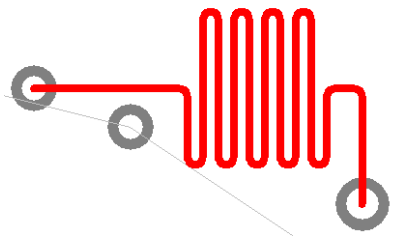
180 Degree Curved

180 Degree Curved will produce a standard serpentine with curved corners. This presets the **Mitre Ratio** to 1.0 and produces 180 degree curved corners. If the **Curved** button is unchecked, it will produce an Octagonal serpentine around the top of the loop although, because it has been ‘customised’, it will show as User Defined.

Attribute Name	Match Value	Min Amplitude	Max Amplitude	Separation	Min Cycles	Shape	Curved	Mitre Ratio
<Net Name>	*	100.0	200.0	10.0	1	180 Degree C	☑	1.000000

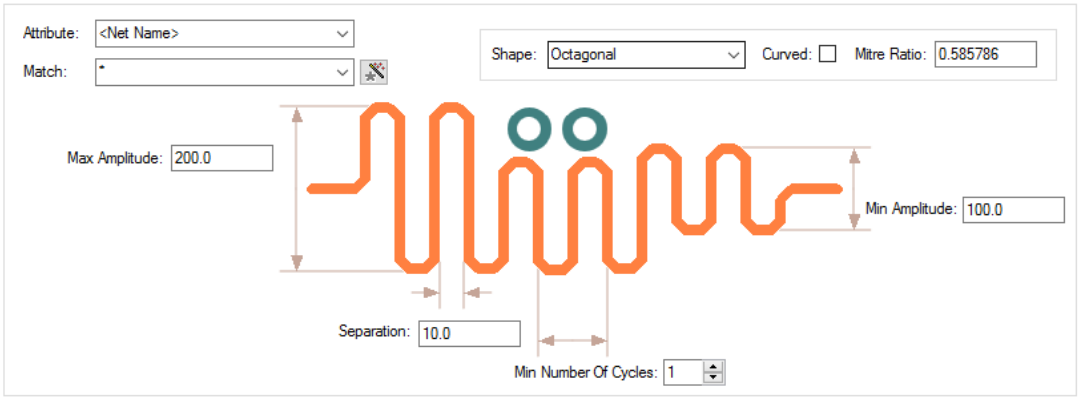


You can define two amplitudes that it can use to automatically reduce the amplitude to avoid obstacles, shown below.



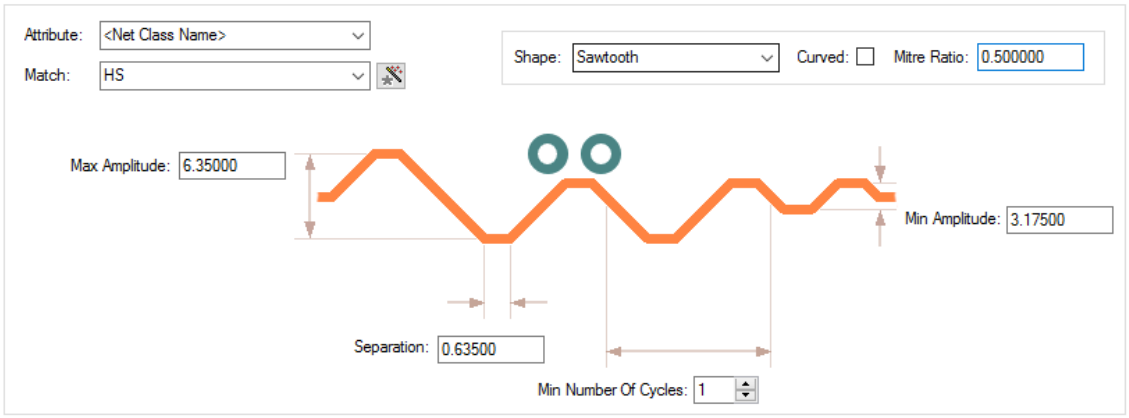
Octagonal

Octagonal produces a serpentine with the 45-degree mitre and flat 'top' lengths the same value.



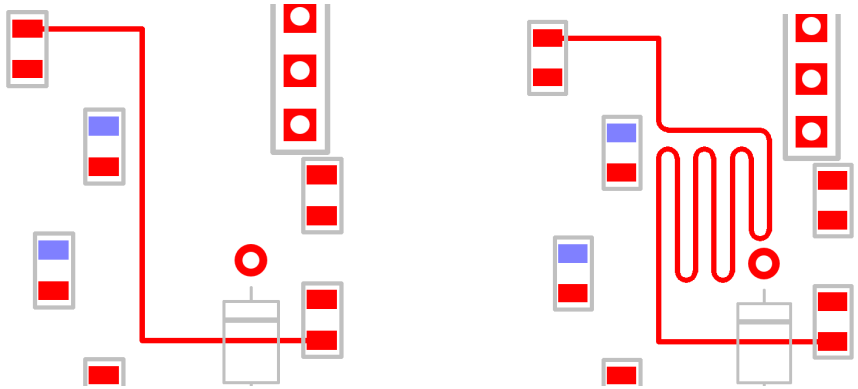
Sawtooth

The Sawtooth shape allows you to define **Min** and **Max amplitude** as well as **separation**. Changing the **Mitre Ratio** will adjust the sawtooth shape allowing more 'flat' spots on the waveform.

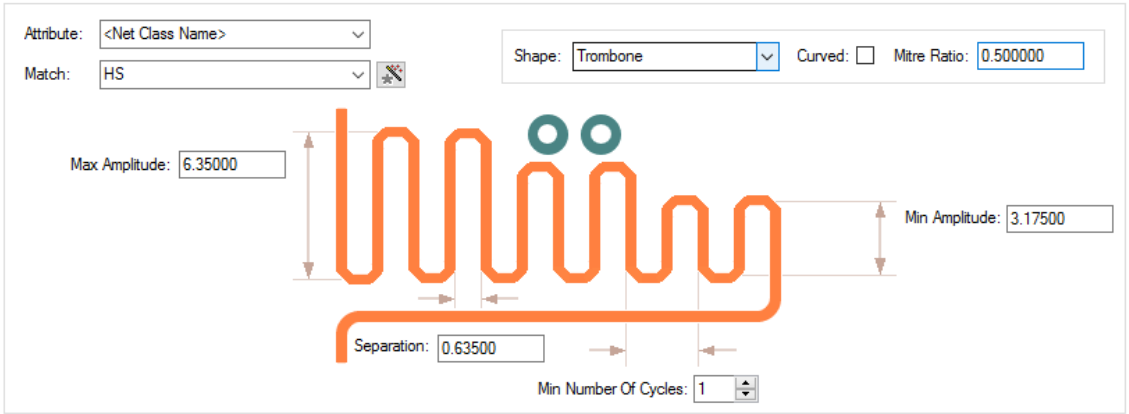


Trombone

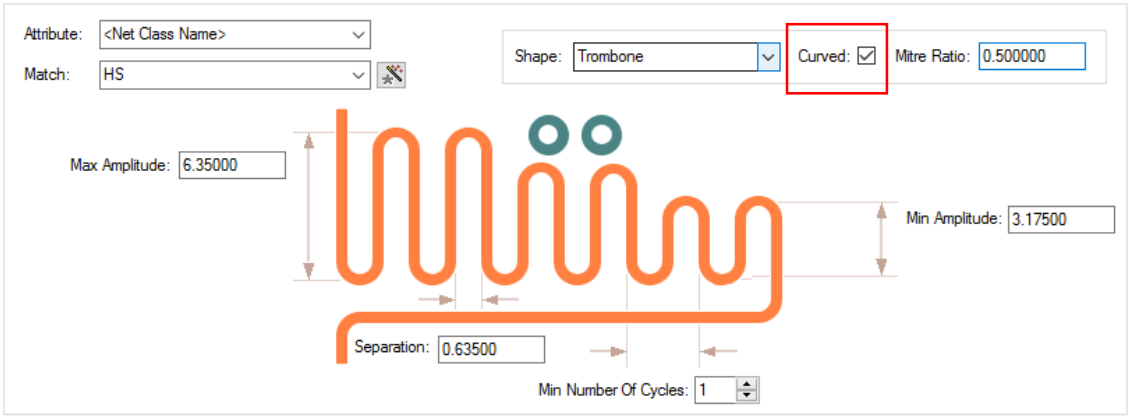
The **Trombone** style breaks into the track and edits it into the trombone shape. The pictures below show the track before serpentine routing and after:



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



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

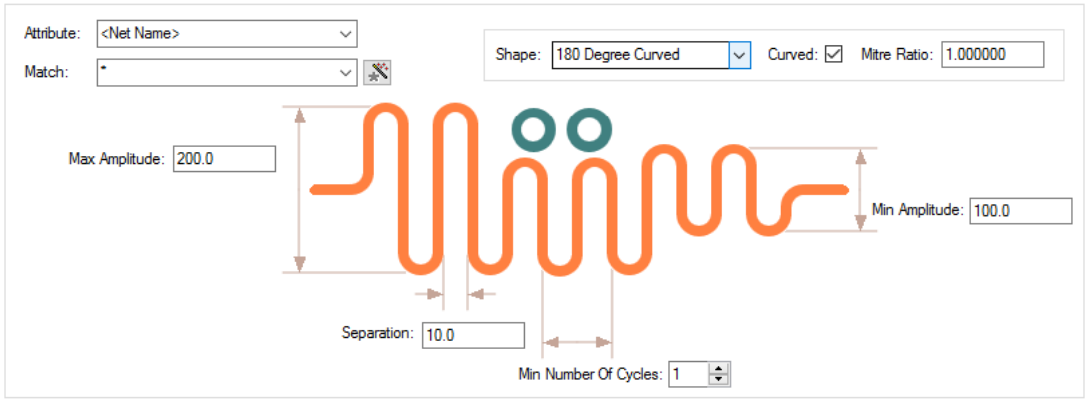


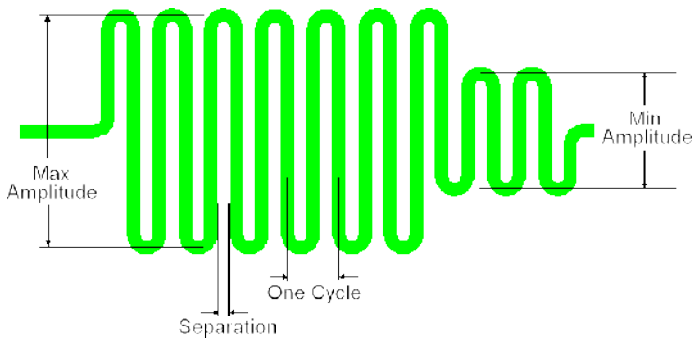
User Defined

If the value is set to a value not matching the above cases, the shape drop down is displayed as **User Defined**.

Shape Parameters

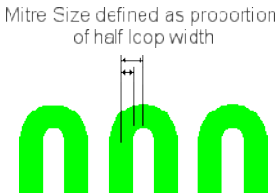
You can define the serpentine with parameters that control the Amplitude and shape of the mitred corners. Mitred or curved corners can range from 180 degree curves to 90 degree corners.





Shape

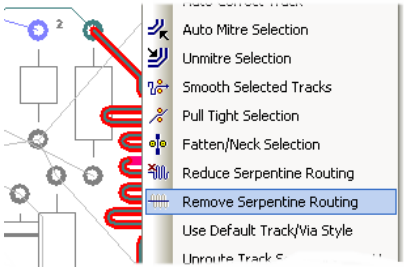
You can control the shape of the top of each loop. The mitre around each turn can be curved or straight, and the size of the mitre is defined by the **Mitre Ratio**, which is the proportion of the 90 degree corner taken up by the mitre. A value of 1.0 gives a complete 180 degree curve (or *sawtooth* if straight) around the top of the loop; a value of 0.0 gives a squared off top to the loop; values in between give a loop of two curves or 45 degree angled lines with a flat top between. The default is **Curved** with a **Mitre Ratio** of 1.0, which results in 180 degree curve around the loop.



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

Remove Serpentine Routing

From the context menu, for a selected track the **Remove Serpentine Routing** feature can be used to remove a selected section of serpentine routing. It can also be used to remove all serpentine routing from the design.

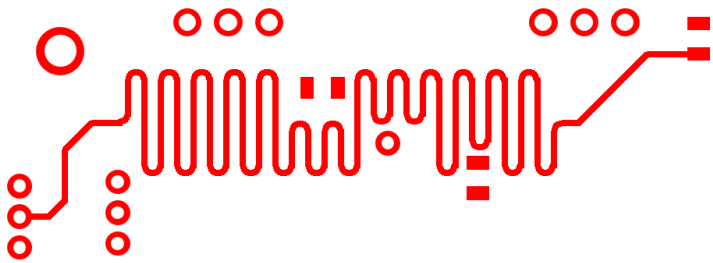


Reducing Serpentine Routing

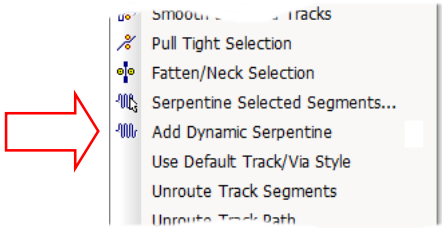
On the context menu is the **Reduce Serpentine Routing** feature to reduce a selected section of serpentine routing by one 'loop'. This enables it to be easily trimmed to the correct length without interactively editing the track.

Dynamic Serpentine Routing

Serpentines tracks can be created and edited using the 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, the **Add Dynamic Serpentine** mode 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 mode can also be invoked from the **Utilities** menu, **Serpentine** >, **Serpentine Mode** option.

You can 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**).

Serpentine Skew Modes on Differential Pairs

Selection of single track on a differential Pair

When a single track segment of a Differential Pair is selected, when using the **Dynamic Serpentine** mode**,** regardless of the position of the cursor relative to the 'other' Differential Pair track, the Serpentine will be applied to the selected segment.

Serpentine Skew Modes on Differential Pairs

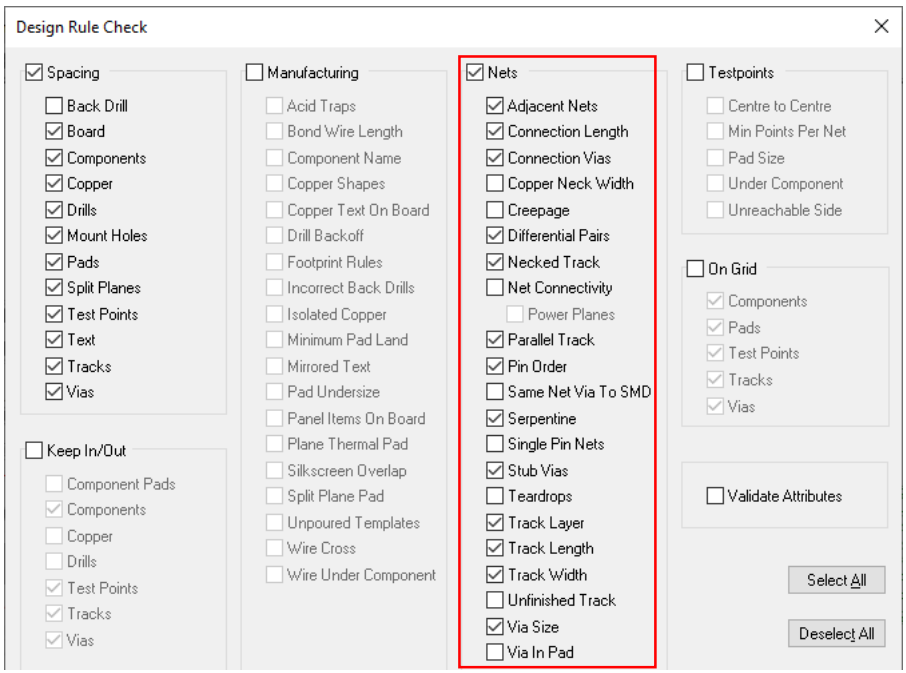
When using the Serpentine mode for Differential Paired tracks you can adjust the pair biasing. Selectable from the context menu, this enables you to bias the track upwards so that the bottom track is not affected, to centre the bias around both tracks of the pair and to bias the serpentine track to the bottom and to have the bottom paired track move to allow space for routing.

Biased Top Biased Both Biased Bottom

Chapter 8. Ancillary Feature to Support High-Speed Design

DRC Checks for High Speed Rules

Various checks can be carried out by the **Design Rules Check** dialog for High-Speed rules. Use the check boxes under the **Nets** section to define the checks required.



Design Rules for checking available for the High-Speed option are: **Adjacent Nets, Connection Length**, **Connection Vias**, **Differential Pair**, **Necked Track, Parallel Track**, **Pin Order**, **Serpentine, Stub Vias, Track Layer**, **Track Length**, **Track Width**, and **Via Size**,

Reporting High Speed Rules and Results

Standard Reports

Standard reports are supplied for reporting rules with high speed parameters assigned to them:

Critical Nets Report

Differential Pairs Report

The reports are additional to the reporting on each of the **Rules** pages within the **Technology**. Each one of these will report its own rules specialty, such as **Where Used** for **Serpentine Rules**.

Custom Reports Using Report Maker

Features within the **Report Maker** enable all aspects of high speed design nets to be reported using commands available.

Chapter 9. Back Drilling

Back Drilling Overview

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

The process of Back Drilling

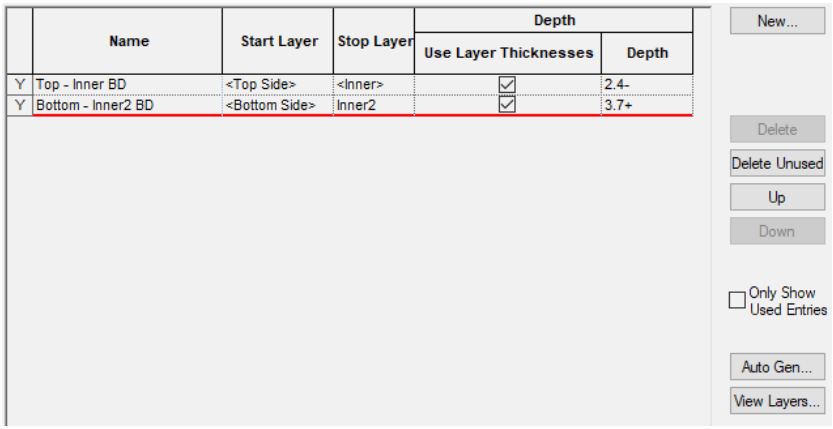
In order to utilise Back Drilling, the summary below informs you of the general process:

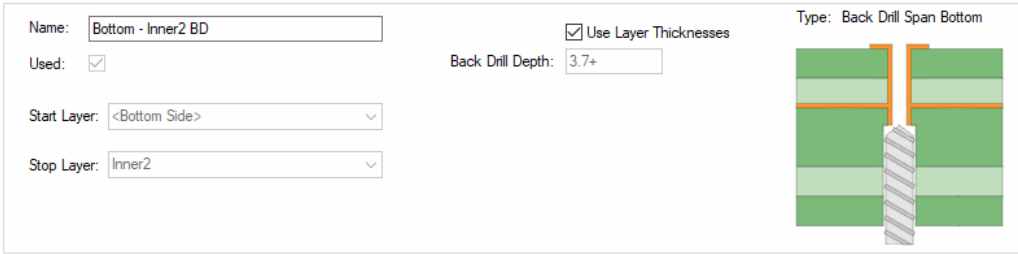
- Identify a requirement for it, then:
- Add Back Drill Spans to accommodate Back Drilling in your Technology
- Create Back Drilling Rules in your Technology
- Route your design, vias and spans that match the Back Drilling rules will be created
- Export the Back Drilled Vias to the Excellon NC Drilling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

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

Technology Back Drilling Spans

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





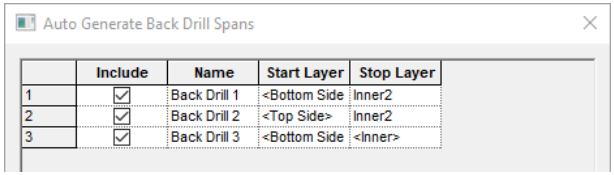
The **Back Drill Depth** is the depth of the oversized drill hole after Back Drilling. This value takes into account the **Back Drill Stub Length** (taken from the **Back Drill Rules** page), and is measured from the top or bottom of the board to the inner electrical 'Stop' layer of the Back Drill span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

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

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

Auto Generating Back Drilling Spans

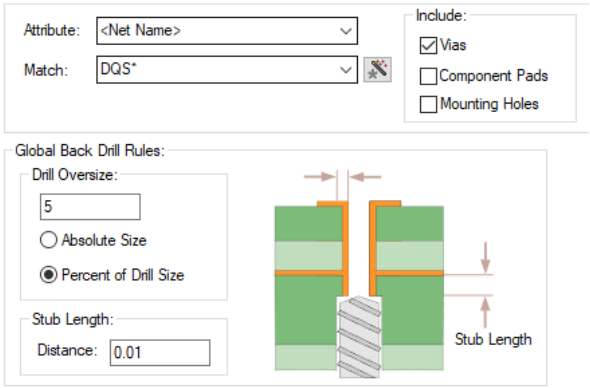
The **Auto Gen** button on this page enables you to automatically generate a list of Back Drill spans that are usable by your current design. These will be generated from the rules defined in the **Back Drill Rules**. The grid is filled with Back Drill spans that can be used in the design.



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

Technology Back Drilling Rules

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



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

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

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

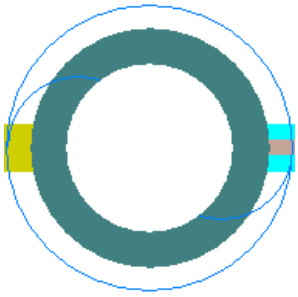
You also define the remaining **Stub Length** required after Back Drilling has been performed. This is your safety margin to absolutely ensure the via stop pad required isn't drilled out.

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

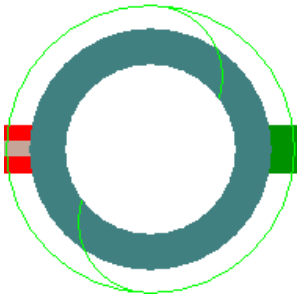
Identifying Back Drilled Vias in your Design

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

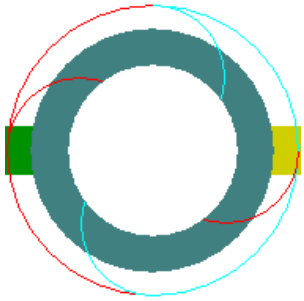
The colour of the circle is dependent on whether it is a Top Back Drill (same colour as Top layer) or a Bottom Back Drill (same colour as Bottom layer), with the curved line being the same colour as the Stop layer of the Back Drill. The curved line will be horizontal for a Top Back Drill and vertical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



Top Back Drill



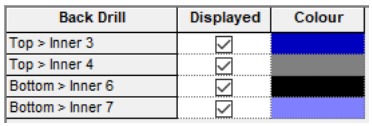
Bottom Back Drill



Both Top and Bottom Back Drill

Back Drill Colours

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



Updating Back Drills

The **Update Back Drills** feature is available in the **Utilities** menu. Use this option to check and assign the most appropriate Back Drill spans to the vias that are on Back Drilled nets and that currently don't have this status. This option is also automatically run when the **CAM Plot** option is initially run.

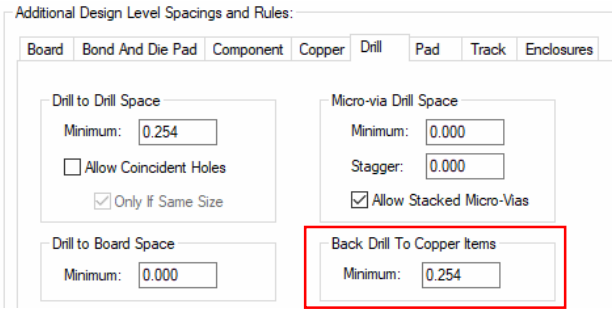
Back Drilled vias can be checked for using new checks in the **DRC** option, see below.

Back Drill Spacings

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

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

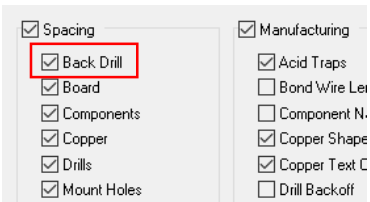
The 'general' **Drill to Drill** spacing handles Back Drill to Back Drill spacing, and Drill to Board handles Back Drills to Board spacing.



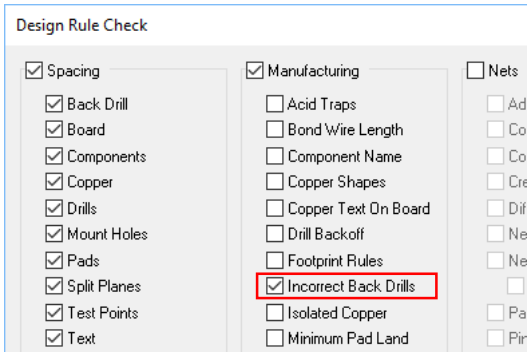
Back Drilling DRC

The **Design Rules Check** dialog has a **Back Drill** check box under the **Spacing** section.

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



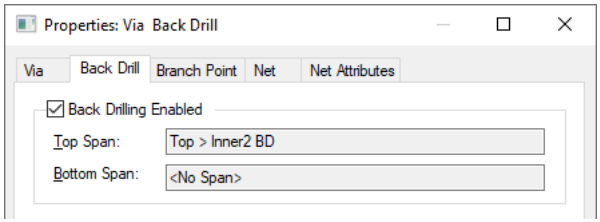
A **DRC** check is also available under the **Manufacturing** section and is named **Incorrect Back Drills**. This check will report an error if a Via Back Drill span is currently incorrect (checked against the **Back Drill Rules**) and requires updating using the **Update Back Drills** option.



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

Via & Component Pad Properties

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



Back Drilling Pin Depth Attribute for Component Pads

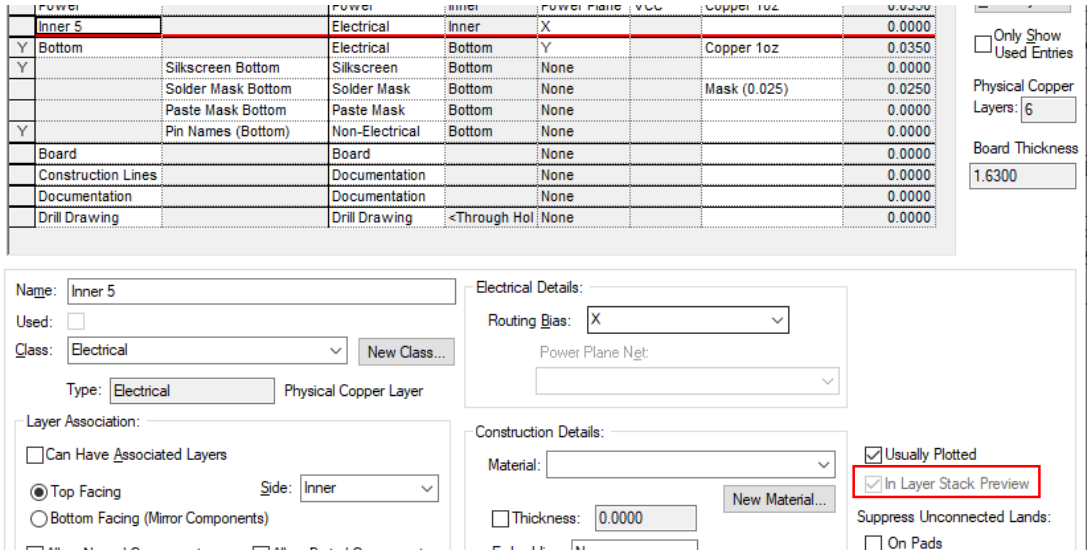
An attribute on a Component pin called **Pin Depth** is available and allows you to define the depth of an **individual** pin so that you know the maximum depth allowed for Back Drilling.

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

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

Layer Span Layers automatically included in Layer Stack Preview

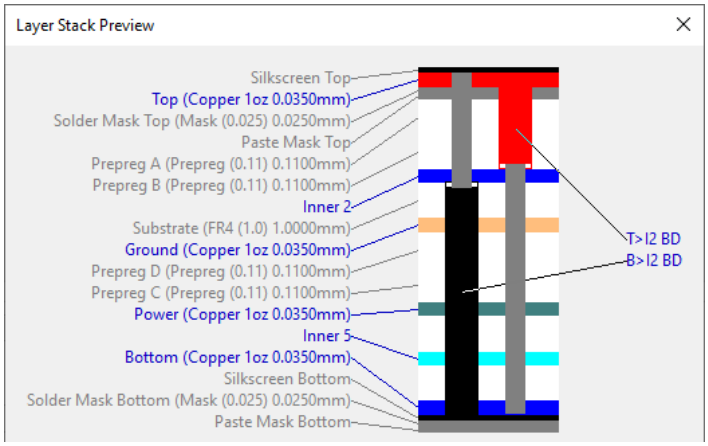
The **From Layer** or **To Layer** of a Layer Span, or the **Stop Layer** of a Back Drill span is included in the **layer stack preview**. For the applicable layer where these are used, the **In Layer Stack Preview** check box on the **Technology Layers** page will be checked and disabled (greyed out).



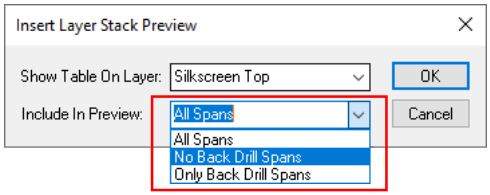
Inserted Layer Stack Previews

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

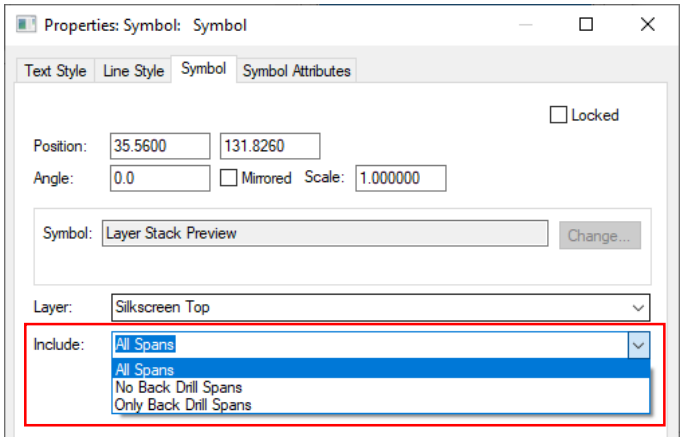
In the Technology, the **View Layers** button works the same on the Layers and Layer Spans pages (shows layers and regular layer spans) but on the Back Drill span page, it will show the layers along with the Back Drill spans, no regular layer spans are shown.



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



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



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

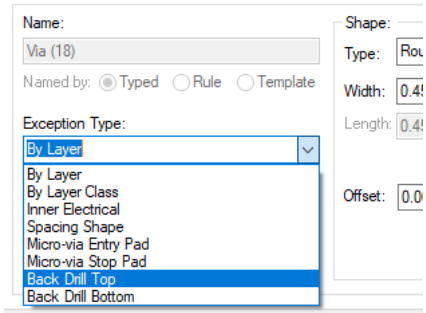
Back Drilling in the CAM Plot Wizard

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

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

Pad Exceptions

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



Excellon Report Chanegs

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

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

```
Plot Name:        Excellon
Variant:          <Master Design>
Contains:         Bottom >Inner 2 BD          Unplated
Side:             Bottom
Drill Depth:      0.0500

Drill Type:       Unplated
Scale:            1.000
Mirrored:         No
Rotated:          No
Offset:           -41.9040     -65.1905mm
Step & Repeat:    Not used
```

IPC-2581 and ODB++ Exports include Back Drills

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

Back Drilling Commands in the Report Maker

Commands are available to enable Back Drilled Vias to be reported and/or added as a user report to the design to display manufacturing information about them.

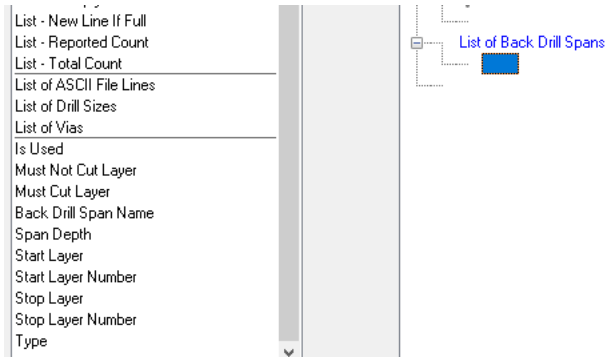
List of Vias and **List of Pins** support the **Back Drilling** feature:

Is Back Drilled

Top Back Drill Span Name

Bottom Back Drill Span Name

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



Additional commands are available:

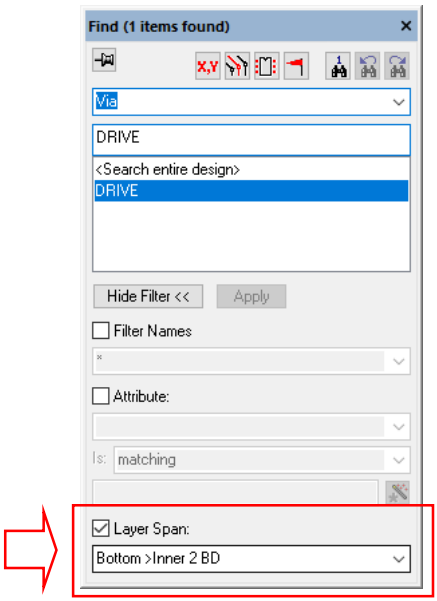
On a via - Is Back Drilled, Top Back Drill Span Name, Bottom Back Drill Span Name.

On a layer - Layer Depth.

On a Layer Span - Span Depth.

Find Bar

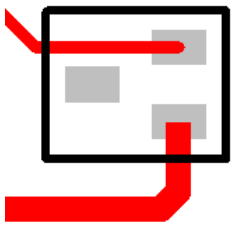
You can use the **Find Bar** to locate any vias that currently have a Back Drill span. When searching for vias, check the **Layer Span** box at the bottom of the bar, and select a Back Drill span from the drop down list.



Chapter 10. RF Design Features

Square-ended Tracks

To support the creation of RF designs, tracks can have square ends instead of rounded ends.



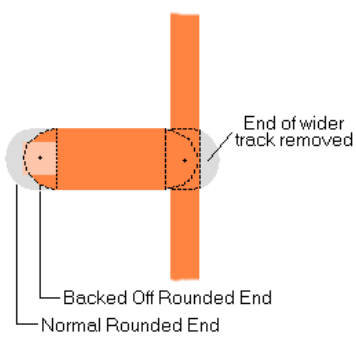
These is defined in the **Technology** dialog under the **Track RF** rules.

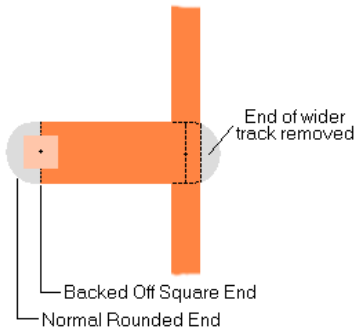
You must have the **Back Off Track Ends** enabled for the **Square Track Ends** to also be enabled.

Back Off Track Ends

Enabling **Back Off Track Ends** will cause track ends to be moved back so there is no overshoot. This is only applied to tracks which would otherwise cause an overshoot, and which have sufficient length for the track to be backed off. **Design Rule Checking** will take the back off into account, allowing tracks to end more closely to other obstacles than would normally be the case. Tracks will be backed off at width changes and T-Junctions, as well as where they terminate at a pad.

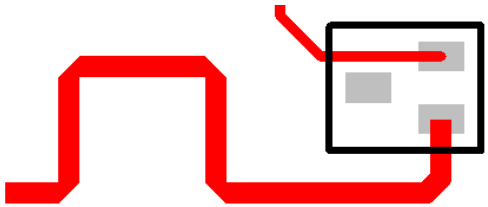
You have a choice of how these backed off track ends are finished. They are either **Round Track Ends** or **Square Track Ends**, see the diagrams below to see the difference.



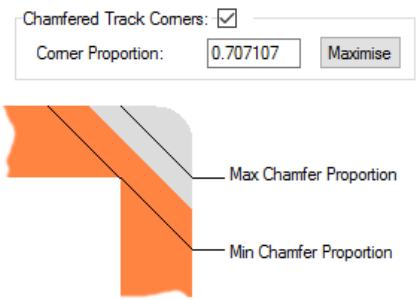


Chamfered Corners on Tracks

For the support of RF designs you can create RF mitres in Pulsonix, these are called **Chamfered Corners**. Parameters allow Track corners to be chamfered (45 degree outside corner and 90 degree inside corner). These are in addition to Curved tracks and Filleted tracks.



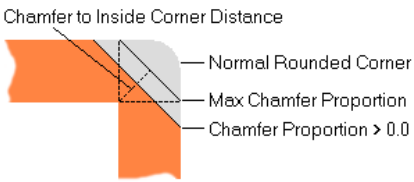
The Chamfered Corner feature is available as a rule that can be set on any net item using the **Track RF** rules page within the **Technology** dialog. For each net that requires this style of track corner, edit the **Attribute Name** and for selected nets, select the **Chamfered Track Corners** check box.



Enabling **Chamfered Track Corners** causes orthogonal corners to be drawn with a 45 degree chamfer instead of the normal rounded corner. Note that the corner is still considered rounded for spatial checking purposes. This gives an over estimate of the space occupied by the chamfered corner. Corners are only chamfered if the track is orthogonal and there is sufficient distance to complete the chamfer before the next corner.

The size of the chamfer is controlled using the **Chamfer Proportion**. The value gives the distance between the inside corner and the outside chamfer as a proportion of the track width (or the minimum width if the two track segments are different widths). A value of 0.5 and a track width of 20 thou would cause the distance between the inside corner and the chamfer to be 0.5 x 20 = 10 thou. The value can be between 0.0 and the reciprocal of the square root of two (0.707107).

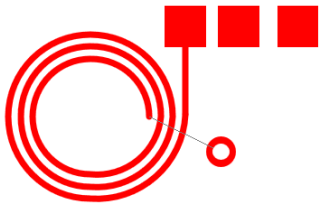
Use the **Maximise** button to set the maximum value. The maximum value gives a chamfer across the width of the track. A value nearer to 0.0 will lengthen the chamfer and decrease the distance between the chamfer and the inside corner.



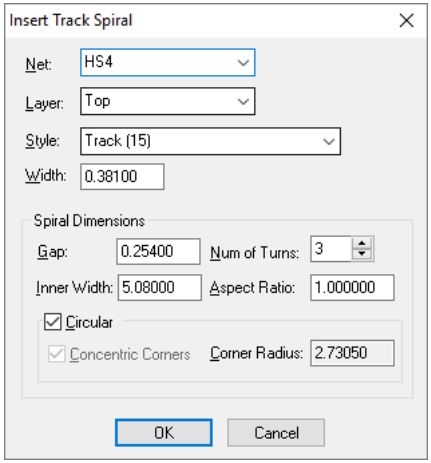
Spiral Tracks and Shapes

As part of the increasing support for RF designers, you can use Pulsonix to insert a **Copper** or **Track Spiral** (and **Breakouts** in footprints).

Insert Spiral is available on the **Insert** menu for **Shapes** and **Tracks**. This feature can be used for designing spiral inductors and planar transformers for example.



When adding the spiral, various parameters allow you to control its shape.



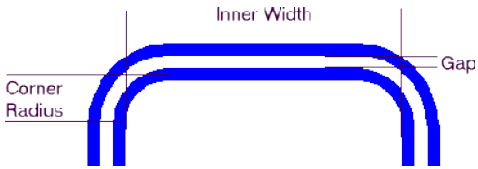
Select the **Net** to connect the spiral to. For tracks you must select an existing net, but Copper does not have to be on a net.

The **Layer** box is used to select the layer to place the spiral on.

Style is used to select the track/copper Style of the spiral. You can also type a **Width**.

Spiral Dimensions

The spiral always begins at the inner right side and ends at the outer right side (but you can rotate or mirror it later).



The **Gap** is the distance between each *turn* of the spiral excluding the segments **Widths**.

The **Num of Turns** is the number of complete *loops* of the spiral.

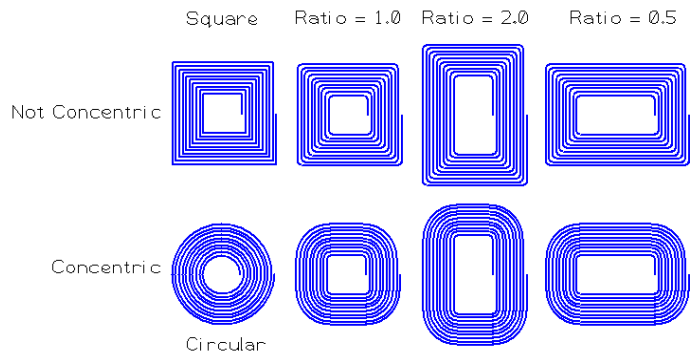
The **Inner Width** is the distance across the inner void of the spiral.

The **Aspect Ratio** allows you to create *rectangular* spirals and is the ratio of **Height / Width**. An **Aspect Ratio** > 1.0 gives a **tall** spiral, and < 1.0 gives a *Wide* spiral.

The **Corner Radius** is the initial inner radius of the corners of the spiral.

Concentric Corners gives you *tight* corners which are properly nested, increasing in radius as the number of turns increases. Otherwise, the corners are fixed at the **Corner Radius**.

The **Circular** option fixes **Concentric Corners** on, and the **Corner Radius** to be half the **Width** plus the **Inner Width**. This has the effect of giving near circular spirals. A zero **Corner Radius** and not **Concentric Corners** will give you square corners.



Note: Spirals are added normal track and copper, to modify, you should use delete and then add another one.

Importing Spirals

You can also import spirals from other systems, such as AutoCAD or SolidWorks using the **Import Design Data** option from the **File** menu and choosing their respective formats, DXF or STEP.

These shapes can be imported directly into the design or into a Doc Symbol or into the Footprint editor depending on how you wish to use them or how many times they will be required.

Index

B

Back drilling, 90
- auto gen, 91
- CAM plot, 96
- colours, 93
- depth, 91
- DRC, 93
- drill oversize, 92
- excellon export, 96
- find bar, 98
- gloabel back drilling rules, 92
- IPC-2851 export, 97
- layer stack preview, 95
- OBD++ export, 97
- pad exceptions, 96
- part pin depth attribute, 94
- pin depth attribute, 94
- properties, 94
- report maker commands, 97
- spacing, 93
- span, 90
- span rules, 91
- start layer, 90
- stop layer, 90
- stub length, 91, 92
- update, 93

Back off track ends, 99
Broadside Coupled, 50

C

Chamfer proportion, 100
Chamfered track corners, 100
Concentric corners, 102
Constraint manager, 6
Critical nets report, 88

D

Design rules checking
- high speed rules, 88

Differential pair chains, 22
Differential pairs, 21
Differential pairs
- edge coupled, 50

Differential pairs
- broadside coupled, 50

Differential pairs
- removing, 63

Differential pairs
- interactive net length indicators, 71

Differential pairs
- track length rules, 72

Differential pairs
- pin to pin length rules, 72

Differential pirs report, 88
Dynamic serpentine routing, 85

E

Edge Coupled, 50
Export layers, 13
Extra delay length rule, 76
Extra length rule, 77
Extra match, 76

I

Import CSV, 11
Import impedance values, 11
Import IPC-2581, 11
Import layers, 14
Interactive net length indicators, 71

L

Layer change length rule, 76

M

Make selection important, 19
Mitre ratio, 84

N

Necked length rules, 80
Net classes, 21
Net styles, 21
Net type, 22
Nets, 21

P

Pin package length, 77
Pin to pin length rules, 72
Planar
- transformers, 101

R

Reduce serpentine routing, 85
Reflect layers, 7
Remove important items, 19
Remove serpentine routing, 85
Removing differential pairs, 63
Report
- critical nets, 88
- Differential pirs, 88
- where used, 88

Report maker, 89

Routing
  serpentine, 81
  serpentine mitre shape, 84
Rules, 22
  layer change length, 76
  necked length, 80
  serpentine, 81
  track length match, 75
  track parallel segments, 79
Rules management, 6
Rules spreadsheet, 15

S

Serpentine routing, 81
  dynamic, 85
  reduce routing, 85
  remove routing, 85
Shapes
  spirals, 101
Show limit shapes, 69
Show limit text, 70
Signal paths, 21
Spirals
  dimensions, 101
  importing, 102
  inductors, 101
  tracks/shapes, 101
Square ended tracks, 99
Start differential pairing, 59
Start mirroring paired tracks, 56
Sub Nets, 21

T

Technology
  rules, 22
Technology files, 6
Track length match rule, 75
Track length rule, 72
Track parallel segments rule, 79
Tracks
  chamfered corners, 100
  spirals, 101
Type, 22

U

User defined mitre shape, 84

W

Where used report, 88
Wildcard
  examples, 24
Wildcard wizard, 25