PULSONIX

High Speed Option
Users Guide

Vertrieb für Deutschland, Österreich und Schweiz:

PESCHGES VARIOMETER GmbH

D-52078 Aachen, Zieglerstr. 11

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

www. peschges-eda.de

E-mail: vertrieb@peschges-eda.de

Copyright Notice

Pulsonix, a division of WestDev Ltd.

Printed in the UK. Issue date: 15/04/11 Pulsonix iss 3

Pulsonix
Oak Lane
Bredon, Tewkesbury
Glos, GL20 7LR
United Kingdom

Contents

Chapter 1. Interactive High Speed Routing

Feature Summary

The **Interactive High Speed Routing** Option comprises a number of features and rules. This option is licensed separately as the High Speed Routing option and is not part of any other option, including the Advanced Technology package.

The following features are available:

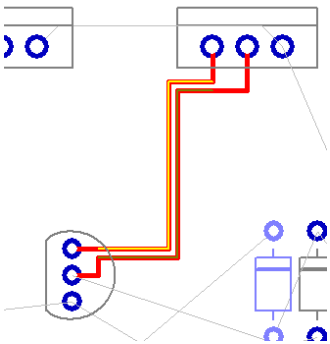
- Interactive Differential Pair Routing
- Rules-based Interactive Net Length Indicators
- Pin-to-Pin Rules Routing
- Daisy Chain (Pin Order) Routing
- Serpentine Routing
- Square-ended Tracks
- Chamfered Corners on Tracks
- Spiral Tracks and Shapes
- Specific High Speed Rules Design Rules Checking

Interactive Differential Pair Routing

Overview of Differential Pairs

A Differential Pair consists of two pairs of pins on two different nets which are be routed together as close as possible for the entirety of their track paths.

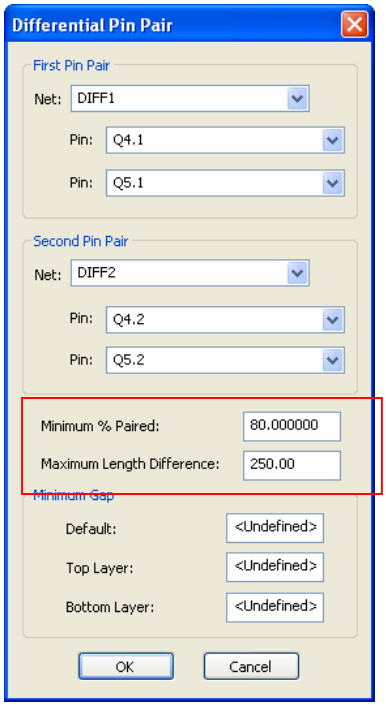
To aid the design of high speed circuits, you can define **Differential Pair Rules** in the design (at both Schematic and PCB stages). The interactive Differential Pair Routing feature is available in the PCB design editor.



It is possible to define Differential Pairs in Pulsonix using the **Technology** dialog and **Differential Pairs**. The Differential Pair have rules 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.

The basic rules are defined on the **Differential Pairs** dialog:



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

Differential Pair Rules

The spacing and length rules are defined in the **Technology** dialog. The spacing is either defined explicitly, or is taken from the **Spacing Rule** appropriate to the two nets. **Length rules**, if defined, can be checked as part of the **Design Rule Checking**.

A Differential Pair is defined in the **Technology** dialog under **Differential Pairs**.

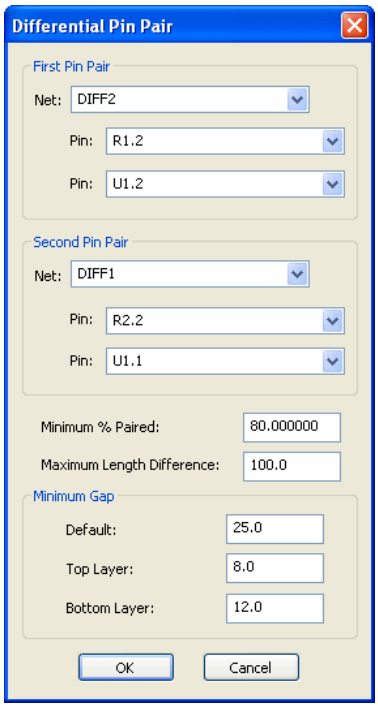
For each pair you provide:

- The net and pair of pins for the first master pin pair
- The net and pair of pins for the second pin pair
- The minimum percentage of how much the tracks are paired
- The maximum allowed track length difference between the pairs
- Optionally, the minimum default gap allowed between the tracks

Defining a Differential Pair

A Differential Pair requires two pairs of pins on two different nets, you can preselect these two pin pairs by firstly selecting each paired connection in the design (select the first then use <Ctrl-pick> to select the second one). You should ensure that the correct two pins are selected. If the two potential differential pair nets are selected, these net names will be entered directly as pre-selected in the **Differential Pairs** dialog when the **New** (Pairs) button is used.

If you wish, you can select the two connections from in the Differential Pairs dialog using the **Edit** button and the drop down list selection of the nets within the design. The first method of preselecting the nets is much easier though.



The **Minimum Gap** is the minimum distance the pair of tracks can be. This can be less than the normal spacing rules for these nets (which would otherwise be the value used).

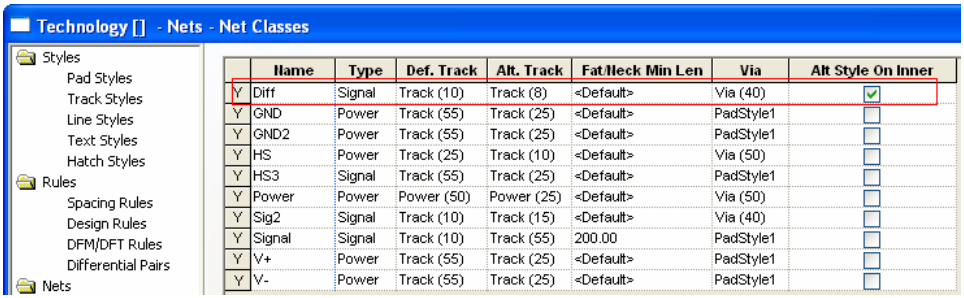
The **Minimum % Paired** is a rule which is checked by the design rules checker (under Nets, Differential Pairs). It ensures that of the total length of the two Paired tracks, the minimum % value defined for the proximity with each other is obeyed. If 80% is defined, then 80% of the track length must be within the minimum spacing rule.

The **Maximum Length Difference** is the maximum the two track paths are allowed to differ in length.

Once **OK** is pressed, the pin pairs and rules will be shown, you have more than one **Differential Pair**.

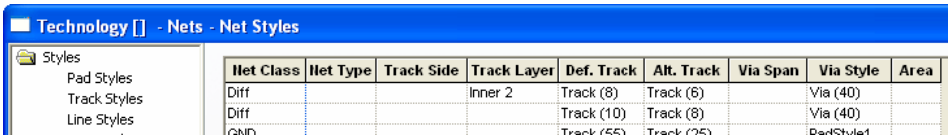
Track Widths uses for Differential Pairs

Differential Pairs use the **Net Class** or **Net Styles** (if defined) rules for tracks and vias. It is normal to create separate Net Class or Styles rules for Differential Pairs so that their specific characteristics and rules can be used.



	Name	Type	Def. Track	Alt. Track	Fat/Neck Min Len	Via	Alt Style On Inner
Y	Diff	Signal	Track (10)	Track (8)	<Default>	Via (40)	☑
Y	GND	Power	Track (55)	Track (25)	<Default>	PadStyle1	☐
Y	GND2	Power	Track (55)	Track (25)	<Default>	PadStyle1	☐
Y	HS	Power	Track (25)	Track (10)	<Default>	Via (50)	☐
Y	HS3	Signal	Track (55)	Track (25)	<Default>	PadStyle1	☐
Y	Power	Power	Power (50)	Power (25)	<Default>	Via (50)	☐
Y	Sig2	Signal	Track (10)	Track (15)	<Default>	Via (40)	☐
Y	Signal	Signal	Track (10)	Track (55)	200.00	PadStyle1	☐
Y	V+	Power	Track (55)	Track (25)	<Default>	PadStyle1	☐
Y	V-	Power	Track (55)	Track (25)	<Default>	PadStyle1	☐

If **Net Styles** has been used, these might look like this:

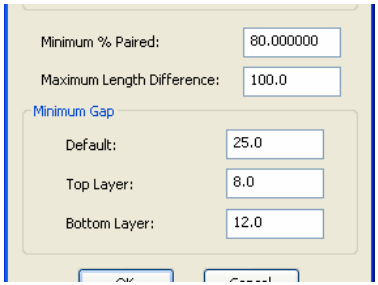


Net Class	Net Type	Track Side	Track Layer	Def. Track	Alt. Track	Via Span	Via Style	Area
Diff			Inner 2	Track (8)	Track (6)		Via (40)	
Diff				Track (10)	Track (8)		Via (40)	

Defining the Differential Pair Spacing Gap

Default Rules

Within the **Differential Pairs** option, you can define the **Minimum Default** pair **Gap** and for the **Top** and **Bottom** layers. The **Minimum Gap** is the minimum distance the pair of tracks can be. This can be less than the normal **Spacing Rules** defined for the nets. If the value is left as *Undefined*, then it takes the appropriate **Track to Track Spacing Rules** value.



Advanced Rules

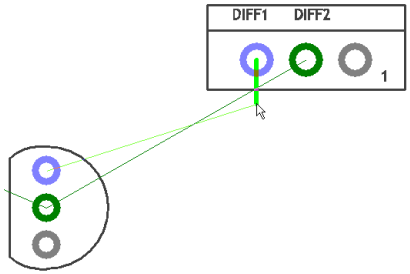
If you need more advanced rules, you can define these in the **Net Styles** dialog for the **Net Class** used by the **Differential Pairs**.

Routing a Differential Pair

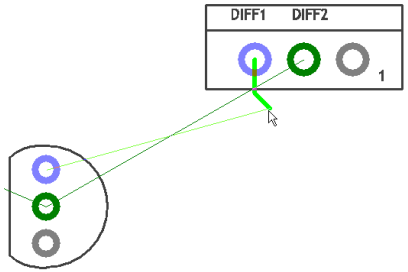
This is how you route Differential Pair tracks:

▶ To route a differential pair

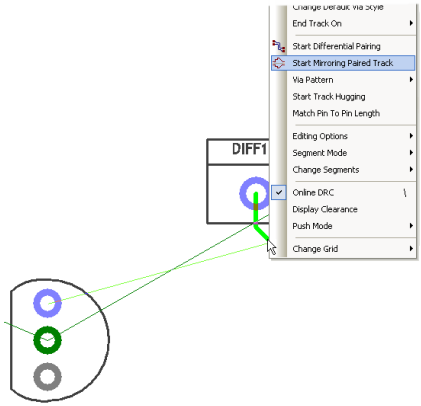
1. There are a number of ways to create differential pairs. We will use the more straight forward method to start with.
2. You would have set up your Differential Pairs already using the **Technology** dialog.
3. Begin to start routing from one of the pins in the set (double click on the net). Don't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



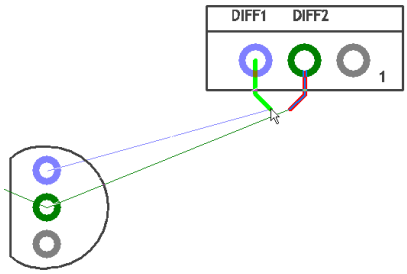
4. In this example, we will move the cursor inwards to create a 45 degree mitre as well. The routing mode is already enabled in **Angled (45)** mode (from the **Options** dialog and **Edit Tracks** page).



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

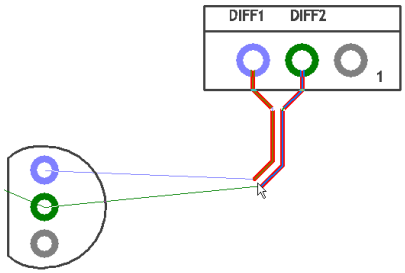


6. This will create the initial paired tracks using a mirror image of the first track you routed to start with.

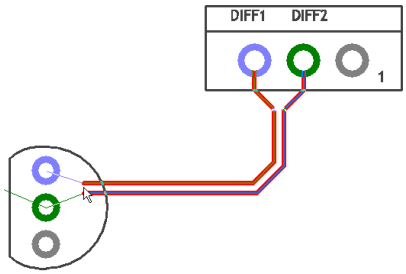


7. Moving the cursor closer to the paired track will now bring the tracks together using the **Track to Track** spacing rule defined in the **Technology** (or any additional **Net Class** or **Net Styles** rules or **exception rules** added to the net).

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.

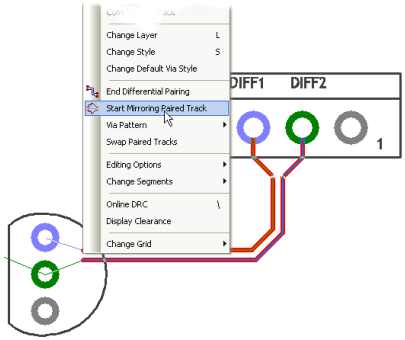


9. Move the cursor towards your end target, click once to add corners just like regular track editing. Changing layers will be demonstrated a bit further on.

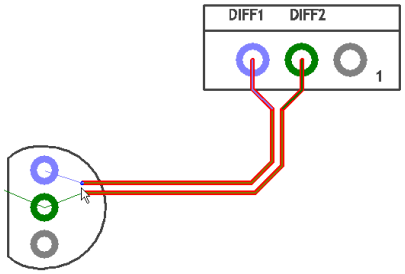


10. Position the cursor mid-way between the target pads.

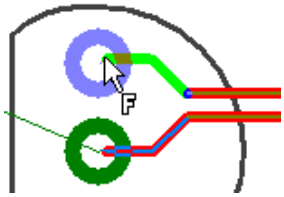
11. Right click the mouse and from the context menu select **Start Mirroring Paired Tracks** again.



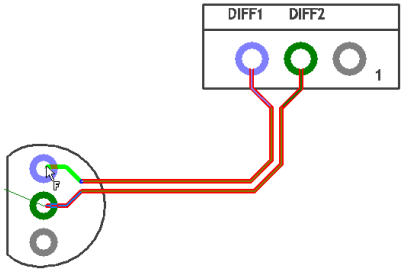
12. This will instruct this option that you wish to now finish the track pairing in a defined way.



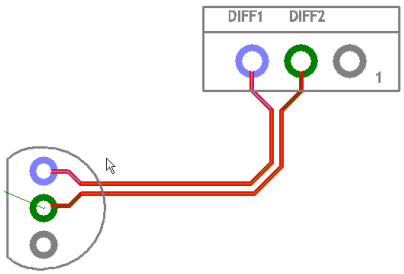
13. Assuming you are still using the **45 Degree Segment Mode**, moving the one track towards the target the other track will 'mirror' this image.



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



15. Click to finish.

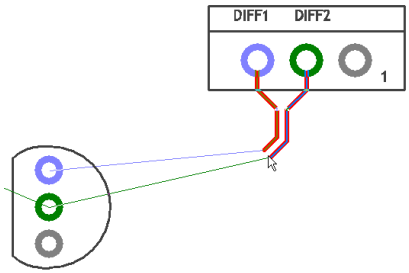


16. The completed result looks like the above example.
17. These two tracks will now behave as a differential pair and will use the rules provided. **Design Rules Check** etc. will also know about these tracked pairs.

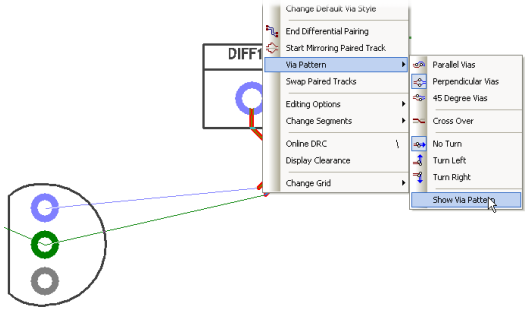
Changing Differential Pair Layers

▶ 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.

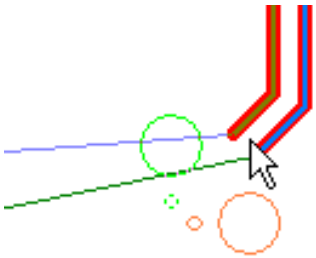


5. To show the vias, select the **Via Pattern>** option from the context menu.



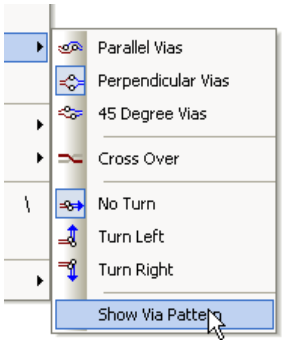
6. At the bottom of the sub-menu, select **Show Via Pattern**.
7. Now when you route, the via pattern and track positions will be shown. These will move dynamically with the routing to show their position if used.

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. Also from the context menu, you can select the type of via pattern to be generated during a layer change.

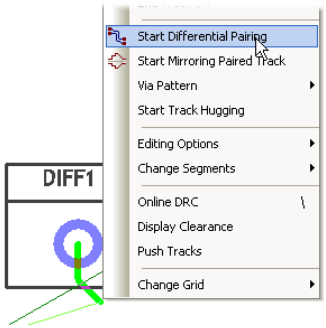


11. The patterns will be applied to subsequent routing after selecting from this menu so it is possible to use multiple-pattern types if required.

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.

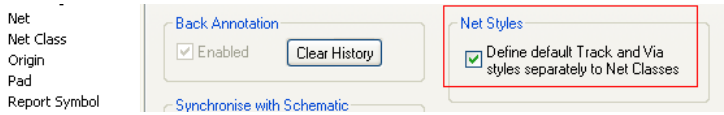
Using Alternative Gaps and Track Widths on Inner Layers

Generally, when creating Differential Pairs, it is expected that you would normally define and use a unique **Net Class** per Differential Pairs.

There are two methods for defining the widths used on Differential Pairs.

Method one uses the normal **default track** style, and a width change to the **Alternate** style as defined in the **Net Class** when this is required. This relies on you changing the width as and when needed. The second method uses the settings defined in the **Net Styles** dialog. As well as the **Track width** used on each **Layer** as well as the **Top** and **Bottom** layers you can also define the **Spacing Gap** used on those layers.

If using the **Net Styles** method, you must first enable the use of **Net Styles** by selecting the **Net Styles** check box in the **Design Settings** dialog under the **General** tab.

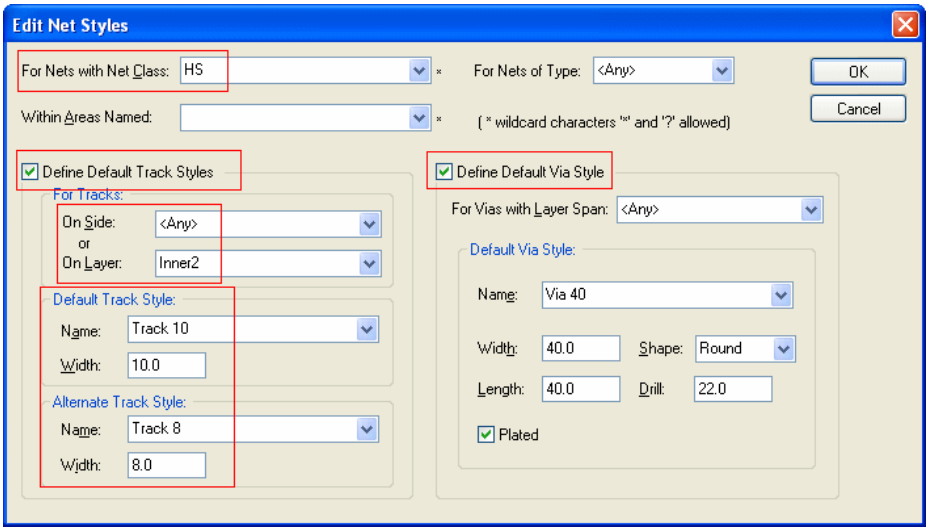


Once this is enabled, the **Net Styles** page is available in the **Technology**.

Net Class	Net Type	Track Side	Track Layer	Def. Track	Alt. Track	Via Span	Via Style
Ground				Power (50)	Power (20)		Via 55
HS				Track 8	Track 8		Via 40
HS2				Track 8	Track 20		Via 40
Power				Power (50)	Power (20)		Via 55

From this dialog, select the **Net Class Name** and press the **New** button on the dialog.

The **Edit Net Styles** dialog is displayed:



From here you can add varying levels of complexity to your Differential Pairs for the selected **Net Class**. The effects any Net Class that uses this Net Class Name, not just the Differential Pair.

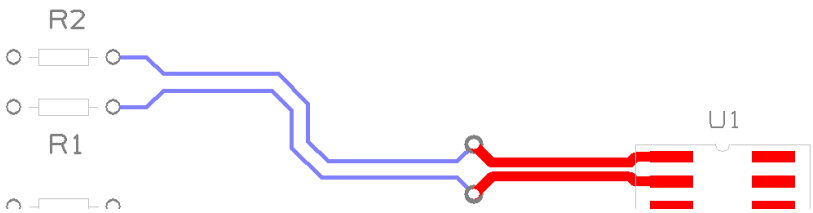
The **For Nets with Net Class:** displays the Net Class Name that will use this set of rules. You can choose to Define Track Styles and Define Default Via Styles.

Using the **For Tracks:** choose the **Side:** you require, or choose the **Layer** for which this rule applies. The **Default Track Style** and **Alternate Track Style** displayed will be used for this rule. Likewise, the **Default Via Style** rule will also be applied if a rule has been defined.

Once edited, the rule might look something like this in the **Net Styles** dialog. Additional rules have been defined for the **HS Net Class** for the **Top** and **Inner 2** layers.

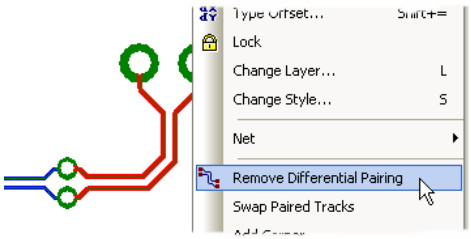
Net Class	Net Type	Track Side	Track Layer	Def. Track	Alt. Track	Via Span	Via Style
Ground				Power (50)	Power (20)		Via 55
HS				Track 8	Track 8		Via 40
HS			Inner2	Track 10	Track 8		Via 40
HS		Top		Track 15	Track 8		Via 40
HS2				Track 8	Track 20		Via 40
Power				Power (50)	Power (20)		Via 55

When the Differential Pair using these rules is edited in the design, it would look something like this:



Removing Differential Pair Routing

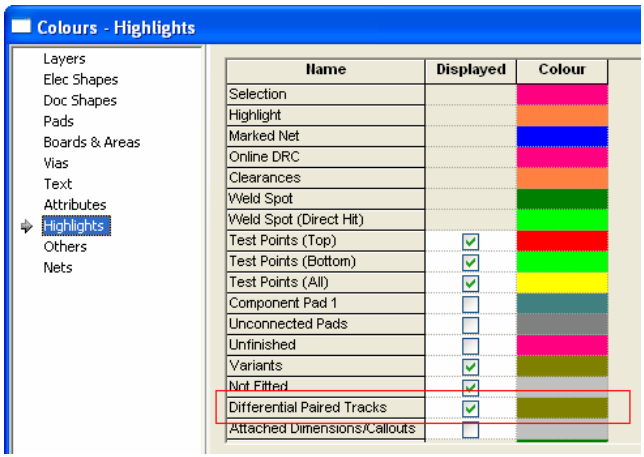
A paired section of track (or a selected part of it) can be unpaired using the context menu command **Remove Differential Pairing** whilst in select or edit modes on the selected track.



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

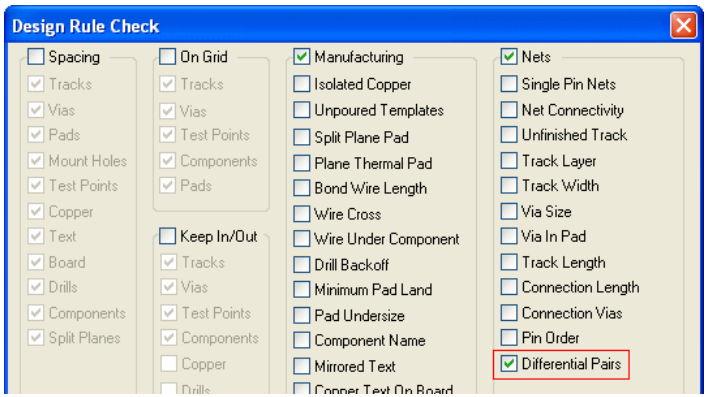
Differential Pair Routing Colours

You can draw **Differential Paired Tracks** in s 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.



Design Rules Checking Differential Pairs

The **Design Rules Checking** dialog can be used to check **Differential Pairs** using rules defined in the **Technology,** this is available as a **Nets** check.



The checker will report an error if:

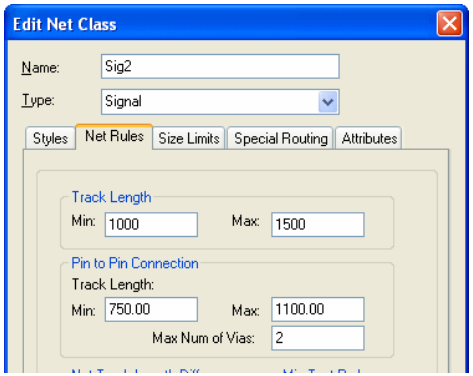
- The percentage of paired track length to the total track length is less than the minimum defined in the technology.
- The maximum total track length of the pairs is less than the max defined in the **Technology**.
- The tracks keep the specified distance apart.

Rules-based Interactive Net Length Indicators

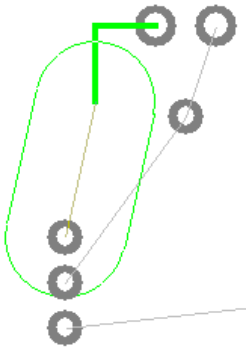
The interactive net length indicators allow you to define length rules in both the Schematic and PCB Designs, if defined in the Schematic, these rules are translated into the PCB design. In the PCB design interactive track editing displays the rules as restriction boundaries.

Track Length and Pin to Pin Connection Length Rules

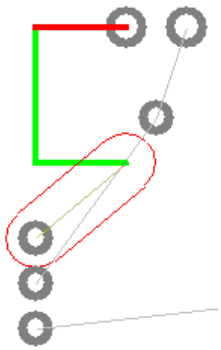
Rules to control **Min** and **Max Track lengths**, and **Min** and **Max Pin to Pin Connection Lengths** are defined in the **Net Class** dialog on the **Net Rules** page. If the **Track Length** rule has been defined on the **Net Class** for the track, or if the track is part of a pin network, during 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.



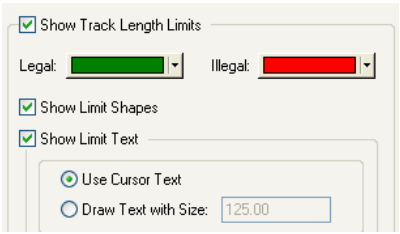
The large oval area shows the length of the track is within the minimum track length set.



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

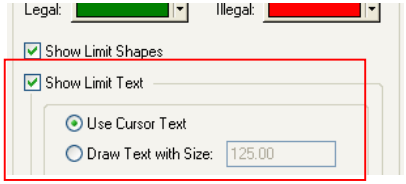
Options for displaying Net Length Rules

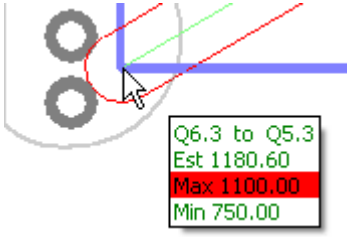
The colours for the **Legal** and **Illegal** track lengths can be defined in the **Options** dialog under **Edit Track**. Using **Show Limit Shapes**, the display of the length indicators can also be switched on and off.



Track Length Limits

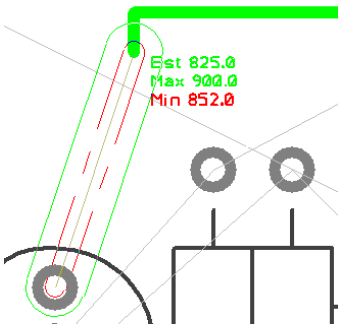
As well as the visual indicators for legal and illegal values, the actual rule values used can also be displayed using the **Show Limit Text** check box.



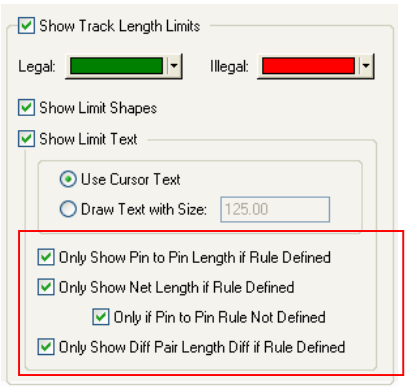


The **Use Cursor Text** option displays the limit text similar to a tooltip always on top of your design keeping it more legible especially in dense areas of the design. You can alter the distance the text box is from the cursor using the **Interaction Options**. You can also use the **Reposition Cursor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

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



'Only Show' Rules

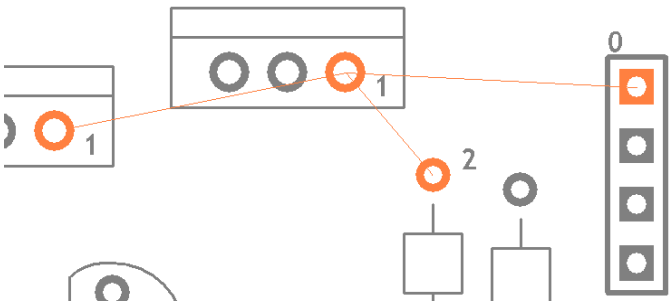


Use the three **Only Show...** text boxes to decide if you always want to show the appropriate length, or only show the text if there is a maximum or minimum length limit rule defined. If you are only showing the net length if there is a rule defined, you also choose if you want to display this as well 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.

Daisy Chain (Pin Order) Routing

Often with high speed design you will need to add nets in a specific pin order so the electrical path is exact and not left to the 'closest' point optimisation. Using advanced rules in the **Net Class** dialog, daisy chain rules can be created with **Net Optimisation** obeying these rules.



Adding the Pin Order Rule

The Pin Order is controlled using a nominated attribute to a subset of the pins in a net the additional restrictions will apply to these nominated pins and the connections between them. The value of the attribute is only relevant to the Pin Order check.

During **Net Optimisation** a minimum set of paths between the nominated pins will be calculated and the rules applied to these paths. These paths may pass through other un-nominated pins, which will be considered as vias in the path. Paths to other parts of the net, or which make a longer alternative path between pins in the subnet, are not included.

How to add daisy chain rules

There are a number of short additions required to invoke daisy chain rules:

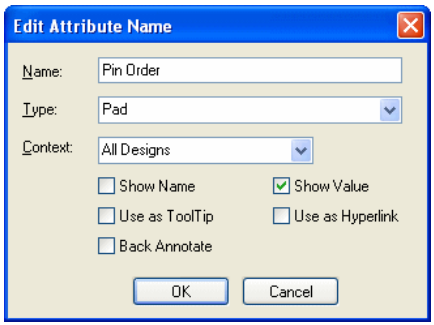
- Add a suitable **Attribute Name** to the **Technology** file, for example, **Pin_Order**. Make this a **Pad** attribute type.
- Use an existing **Net Class** or create a new one to contain the daisy chain rule.
- Edit the **Net Class** and from the **Net Rules** page, add an **Additional Restriction**.
- Add a **New Attribute Name**, use the **Pad** attribute **Pin_Order** for example, exit the dialog.
- In the design, add a **Pin Attribute** of **Pin_Order** with suitable values to the pins which require it. The attribute values determine the pin order that the net will adhere to. Using values like 0, 1, 2, 3, 4, etc. will make the pin order easier to define. These will be alpha-numerical order.

Once these are added, **Net Optimise** can be run to make use of these specific paths. **Design Rules Check** can be performed to check the pin order values.

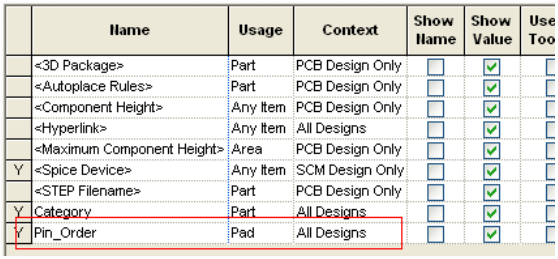
▶ Adding an appropriate attribute to the Technology

1. Select **Technology** <**T**> from the **Setup** menu.
2. Select **Attribute Names** from the browser list.
3. Select the **New** button.
4. In the **Name:** box type a suitable name, e.g. **Pin_Order**
5. In the **Type:** box, select **Pad** as the item type.

6. In the **Context:** drop down list, select **All Designs**.



7. Click **OK** to exit.

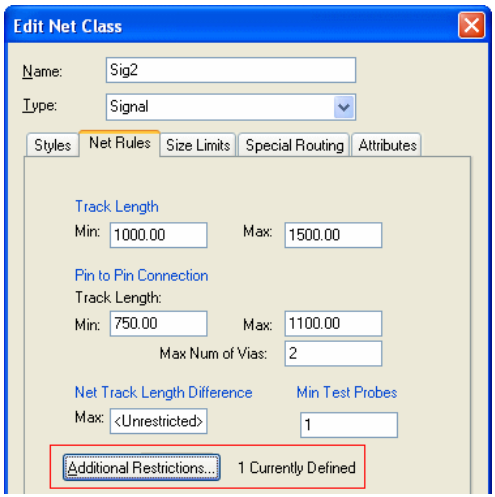


	Name	Usage	Context	Show Name	Show Value	Use Too
	<3D Package>	Part	PCB Design Only	☐	☑	☐
	<Autoplace Rules>	Part	PCB Design Only	☐	☑	☐
	<Component Height>	Any Item	PCB Design Only	☐	☑	☐
	<Hyperlink>	Any Item	All Designs	☐	☑	☐
	<Maximum Component Height>	Area	PCB Design Only	☐	☑	☐
Y	<Spice Device>	Any Item	SCM Design Only	☐	☑	☐
	<STEP Filename>	Part	PCB Design Only	☐	☑	☐
Y	Category	Part	All Designs	☐	☑	☐
Y	Pin_Order	Pad	All Designs	☐	☑	☐

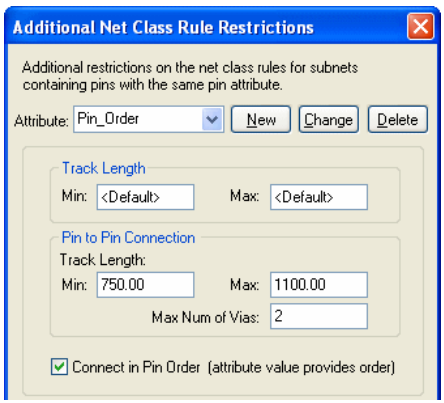
8. Click **Apply** to save the **Attribute Name** in the Technology.

▶ Setting up extra rules for the Net Class

1. In the **Technology** dialog, select **Net Class**.
2. Select a **Net Class** and select the **Edit** button.
3. Select the **Net Rules** tab. The daisy chain rules are added in a dialog accessed from **Additional Restrictions**.



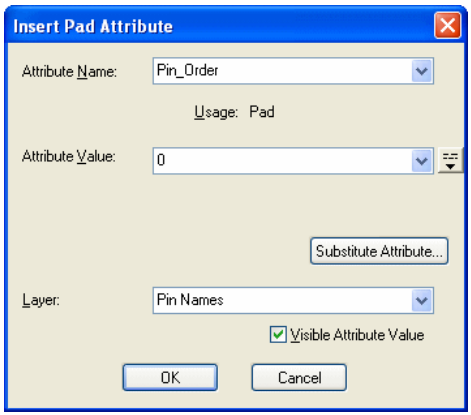
4. Select the **New** button next to **Attribute:**



5. You need to select the Attribute created previously (**Pin_Order**) from the drop down list.
6. The rules on this dialog are for use when pins have the **Pin Attribute** defined.
7. To enable daisy chain rules you must select the **Connect in Pin Order** check box at the bottom of the dialog.
8. Click **OK** to exit.
9. You must use **OK** to save these settings to the **Technology** ready for use in the design.
10. The **Pin_Order** attribute is now available for use in the in the design.

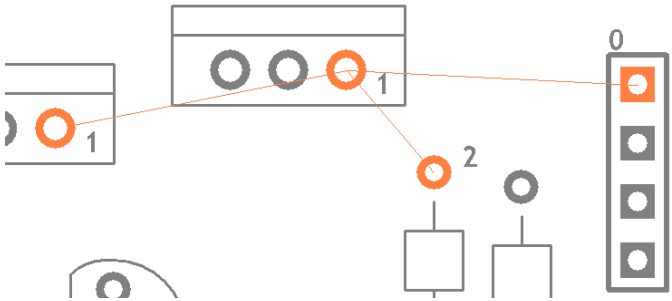
▶ Adding the pin attribute to the design

1. Select a pin in the design, one which requires this attribute.
2. From the context menu, select **Insert Attribute**.



3. From the **Attribute Name:** drop down list, select the **Pin_Order** attribute name.
4. In the **Attribute Value:** box, type the pin order value required. It is recommended that you use sequential numbers or letters to define the order, such as 1, 2, 3, or a, b, c etc.
5. For **Layer:** (for the attribute value if you wish to view it), select a non-printed layer such as **Pin Names**. One that can be viewed but isn't significant in the design (unless you wish to plot the value names).

6. Click **OK** to save the attribute value to the pin.
7. Position the **Attribute Value**, near the pin is recommended.
8. Press the <**Esc**> key to exit this mode.
9. Do this for all pins on the same net which are pin order specific.
10. When you use the **Net Optimise** option on the design, your new connection pattern will adhere to the **Pin Order** rules.

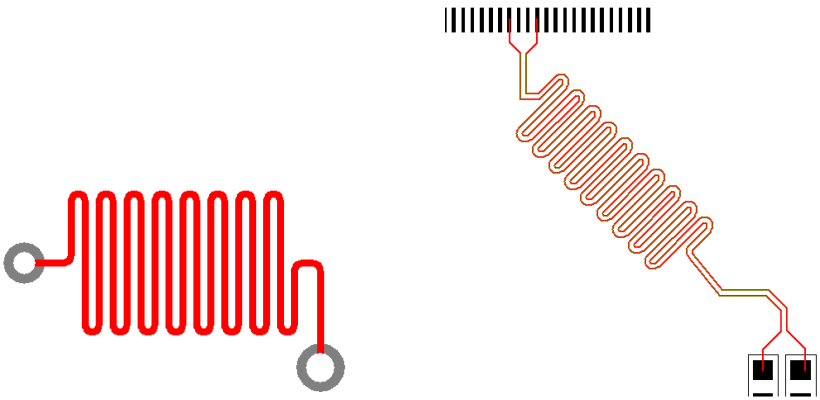


Design Rules Checking Pin Order rules

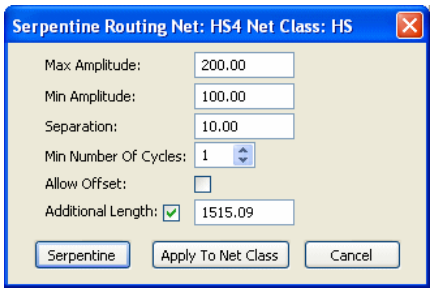
During **Design Rules Checking**, if you select the **Pin Order** option from the **Nets** selection, DRC will check for correct pin ordering as defined by your pin order rules.

Serpentine Routing

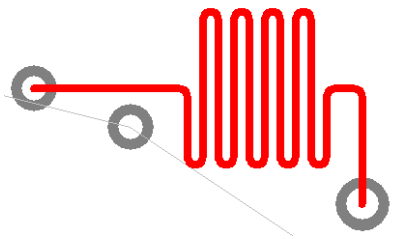
For balancing the length of high speed nets is the insertion of track 'length' without introducing spacing errors. This is commonly known as **Serpentine Routing**. Serpentine Routing can be applied across 90 degree or 45 degree track segments (as shown below). It can also be applied to differential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

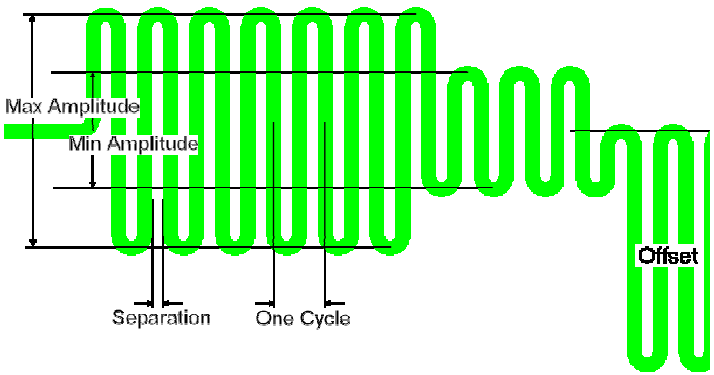


You can define two amplitudes that it can use to automatically reduce the amplitude to avoid obstacles, shown below.



Net Class Special Routing

The serpentine parameters can be defined in the **Net Class** dialog under **Special Routing**, these values are used to prime the dialog. Net Class net and connection rules will be applied to prime the dialog with the required amount of additional track.

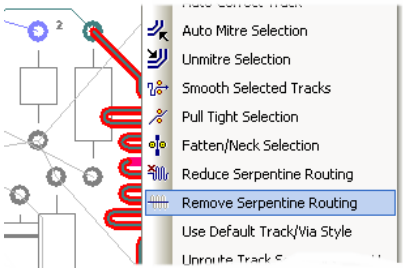


Additional Length

If **Additional Length** is required this can also be defined. If enabled, the serpentine routing option will stop once this length has been exceeded, otherwise the whole of the selected segments will be considered for serpentine routing. If a track minimum length rule has been defined on the Net Class, then the dialog will be primed with the Additional Length required to satisfy this rule.

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.

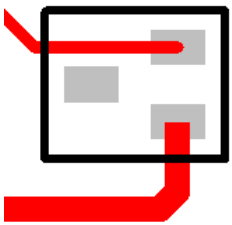


Reducing Serpentine Routing

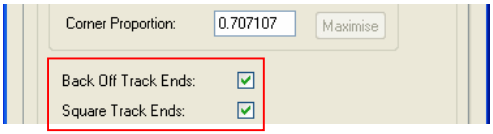
Also on the context menu is the **Reduce Serpentine Routing** feature to reduce a selected section of serpentine routing by one 'loop'. This enables it to be easily trimmed to the correct length without interactively editing the track.

Square-ended Tracks

To support the creation of RF designs, tracks can have square ends instead of rounded ends. This feature is available in the Advanced Technology option.



These is defined on a **Net Class** basis in the **Technology** dialog. For each net class that requires this track style, edit the **Net Class** and on the **Special Routing** page, select the **Square Ended Track** option.

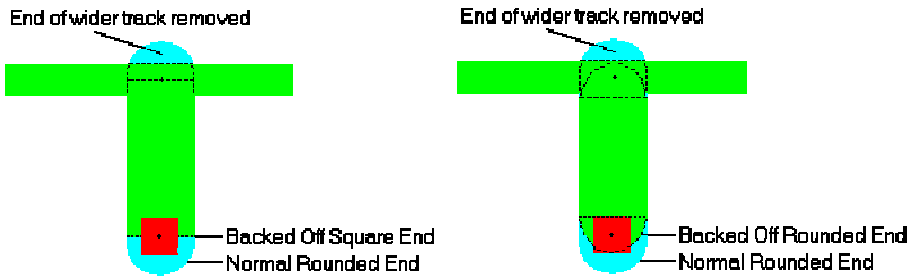


You must have the **Back Off Track Ends** enabled for the Square Track Ends to also be enabled.

Back Off Track Ends

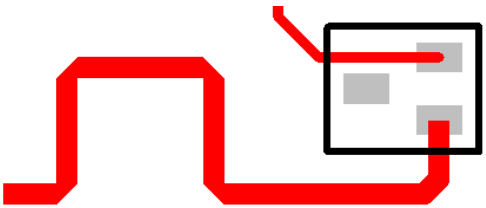
Enabling **Back Off Track Ends** will cause track ends to be moved back so there is no overshoot. This is only applied to tracks which would otherwise cause an overshoot, and which have sufficient length for the track to be backed off. **Design Rule Checking** will take the back off into account, allowing 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.

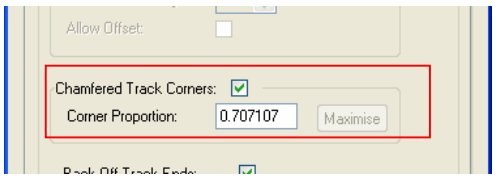


Chamfered Corners on Tracks

To additionally support RF designs you can create RF Mitres in Pulsonix, these are called **Chamfered Corners,** this is particularly useful for high speed signals. Track corners can be chamfered (45 degree outside corner and 90 degree inside corner).



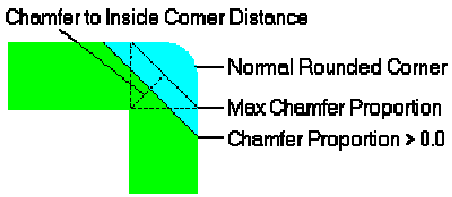
This feature is defined on a **Net Class** basis in the **Technology** dialog. For each net class that requires this track style, edit the **Net Class** and on the **Special Routing** page, select the **Chamfered Track Corners** option.



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.

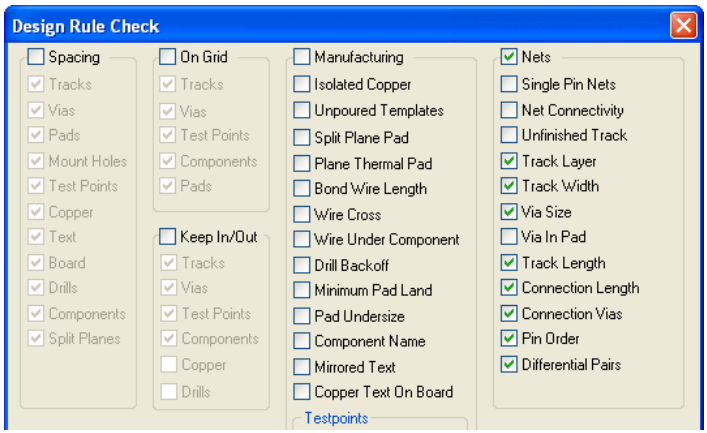
The size of the chamfer is controlled using the **Chamfer Proportion**. The value gives the distance between the inside corner and the outside chamfer as a proportion of the track width (or the minimum width if the two track segments are different widths). So a value of 0.5 and a track width of 20 thou 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.



Interactive High Speed Rules DRC Checks

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

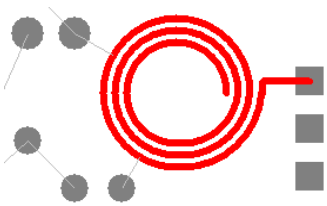


Design Rules for checking available for the High Speed option are: **Track Layer**, **Track Width**, **Via size**, **Track Length**, **Connection Length**, **Connection Vias**, **Pin Order** and **Differential Pair**.

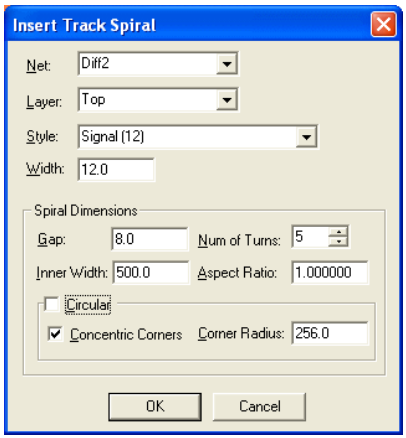
Spiral Tracks and Shapes

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.



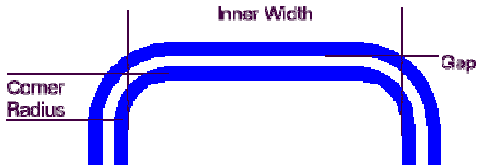
Select the **Net** to connect the spiral to. For tracks you must select an existing net, but Copper does not have to be on a net.

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.

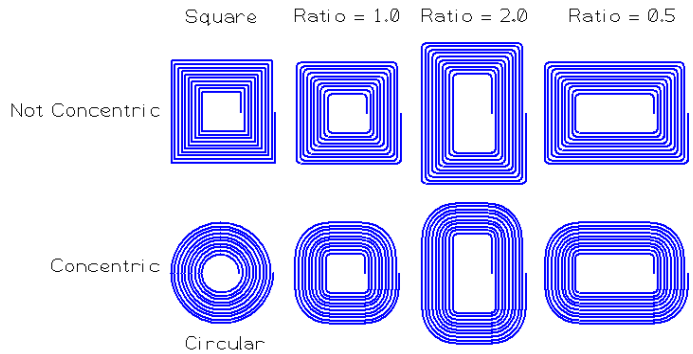
The **Inner Width** is the distance across the inner void of the spiral.

The **Aspect Ratio** allows you to create *rectangular* spirals and is the ratio of **Height / Width**. So 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.

Index