Easy-PC

V25.0 Supplement

Copyright © 1998-2021 WestDev Ltd. All rights Reserved. E & O E

Number One Systems
20 Miller Court
Severn Drive
Tewkesbury Business Park
Tewkesbury
Glos GL20 8DN
United Kingdom

Phone: 01684 296 501
Email: sales@numberone.com

Technical: 01480 382 538
Email: support@numberone.com

Web site: www.numberone.com

The product is licensed by WestDev Ltd to the original purchaser of the product for use only on the terms set forth in the WestDev End User Licence Agreement. Copying, duplicating, selling or using the product contrary to these terms is strictly forbidden.

This Manual Copyright to WestDev Ltd. All Rights reserved. No part of it may be copied, photocopied, reproduced, translated, or reduced to any electronic medium or machine-readable form without WestDev Ltd. prior written permission.

Information in this manual is subject to change without notice and does not represent a commitment on the part of the vendor.

Number One Systems, Easy-PC, Easy-Spice, Easy-Router, ProRouter, Pulsar, Layan, Analyser, MultiRouter, Filtech, Z-Match, StockIt and all variants thereof are Trademarks of WestDev Ltd.

All other trademarks acknowledged to their rightful owners.

Number One Systems, a trading division of WestDev Ltd.

While every care has been taken in the preparation of this manual WestDev Ltd cannot be held responsible for any errors or omissions within it. If informed we will add comments and features which you may like to see written which may help others using this manual. Please send your comments through the technical support desk.

Manual date: 31/07/21 Issue 1

Contents

CONTENTS **3**
CHAPTER 1. GETTING STARTED **4**
Installation 4
Running Easy-PC 25.0 5
CHAPTER 2. NEW FEATURES IN EASY-PC V25 ..... **6**
Differential Pair Track Editing 6
Overview 6
What is a Differential Pair? 6
Setting up Diff Pairs 6
Creating Diff Pairs 8
Diff Pairs – Design Tooltips 11
Diff Pairs – Nets Bar 11
Highlight Colour of Diff Pairs 12
Design Rules Checking of Diff Pairs 12
Diff Pair Report 13
New Design Rule Checks 14
DRC check for Undrilled Pads 14
DRC check for Via to SMD Pad distance 14
DRC Track Spacings for Teardrops 15
Highlight Net across Project 17
Resize Shape 17
Delete in SCM, option to keep Net 18
Component Values – Rename option 19
Add Component Bar shows Values 21
Add Component – List of Recent items 21
Copper Coverage Report 23
Free Pads in Custom Report 24
Angle of PCB Item in a Panel Design 24
Shape To Pad 25
Component Suppression dialog 25
Place Component Names in PCB 26
Component Description on dialogs 28
Tented Vias 28
Export Pin Information from Component Editor dialog 31

Chapter 1. Getting Started

Installation

Backing up your files

If you already have Easy-PC installed, please remember to back up all your libraries, Technology files and any other data files before proceeding with the installation of the new version. The installer should not overwrite any of your own named files, but it can re-install new copies of our standard data files so if you have changed any of those files it is important to back them up first. If you are uncertain, check the time/date stamp on the file but in any case, make a back-up.

Of course, backing up your data is important not only for the upgrade but also at regular intervals during design.

Installation from a download link

A download link for the installation of Easy-PC would have been provided to you by email. Click on the link to download the executable named EasyPC.exe. This is the whole installation set and should be saved and backed up for future use. Any subsequent patches can be installed on top of this 'base' setup once installed.

Using Windows Explorer, find the executable in your *Downloads* folder and double-click it. You'll need to type (or copy/paste) the **password** provided to unpack this file. Once the unpack password has been successful, you will be allowed to continue with the installation. You will also need to have your **customer ID number** that will be in the download link and your **16-digit installation** code to fully install the product.

All other instructions should be followed until you click **Finish** to complete the installation.

The installation is the same for new and existing users alike. Existing users with versions prior to this latest version can install the new software over an existing installation without deleting the old one first.

Installation From CD

Note: CDs are no longer supplied by default, a download link would have been supplied to you to use unless you have purchased the CD media.

To use the CD-ROM - insert the CD-ROM and wait a short time. The CD-ROM will run up to speed and an Easy-PC Welcome screen will appear. If *autorun* has been disabled on your computer you must execute the 'setup.exe' program using the **Start** menu and **Run** command from the Windows task bar.

Installing over existing Easy-PC software

If you already have an earlier version of Easy-PC installed on your system and you wish to install the new version into the same folder as the earlier one, please note that you will then end up with both versions listed in the Windows Control Panel list of installed applications.

If you don't want the earlier one to be listed in the **Control Panel**, you will need to un-install that version **before** you install the new one. If you install the new software into the same folder as the old version then try to un-install the old one, you will find that the new software will not run as the un-install will have removed many or all of the program files.

If you wish to install and use the new version without removing the old one, you will need to install the new version into a different folder. The two versions will then operate independently and either can be un-installed without preventing the other from running.

Uninstalling Existing Easy-PC Software

Uninstalling will still remove shared registry entries, so it is recommended that a configuration file be saved first using the **Configuration Files** option from the **Help** menu and **Support** option. This will provide a restore point for any settings which may be lost.

Data Files Location

There is a step in the **Setup** installation wizard that asks you where you want to place data files (for example, Libraries, Technology files, etc). The default is always to use the common documents folder, C:\Users\Public\Documents\Easy-PC on Windows 8.1 or 10 if you are installing for All Users, or into your own Documents folder if installing for current user only.

Running Easy-PC 25.0

Once installed, an Easy-PC shortcut icon will appear on your desktop. This is also available on the **Start** panel in the **Number One Systems** folder.

To start the program, double-click on the **Easy-PC** icon.

Chapter 2. New Features in Easy-PC V25

Differential Pair Track Editing

Overview

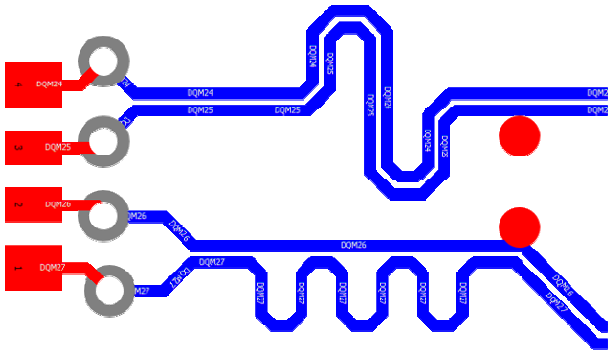
You can now use **Diff Pair** from the **Edit** menu to add **Differential Pairs** to your design.

Diff Pairs in Easy-PC are enabled and controlled from the **Design Technology** dialog and **Net Class** tab.

Controls are available for separate **Diff Pair Gap** and **Skew values** for pairs on a **Net Class**.

What is a Differential Pair?

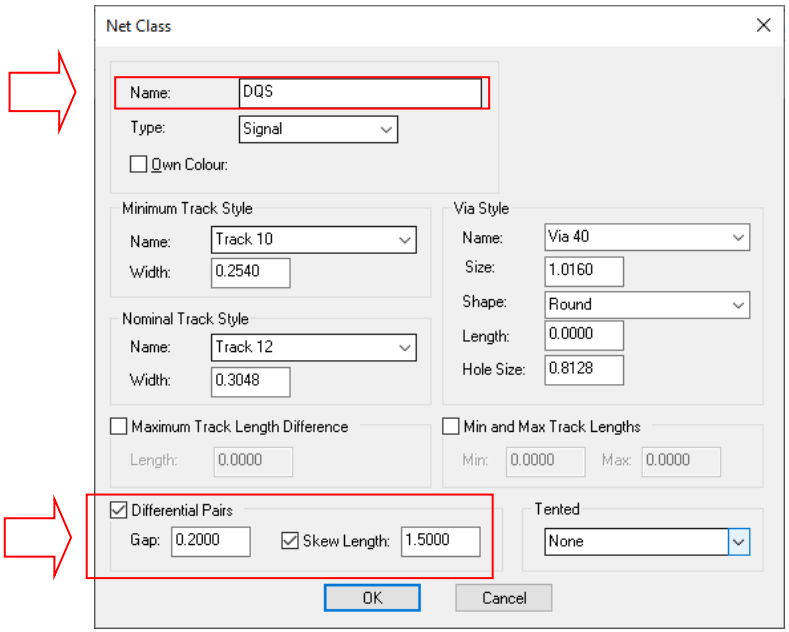
A Differential Pair is a term used in PCB design to define a pair of tracks that carry signals that are transmitted down them. These are usually tightly coupled tracks; one carries a positive signal and the other carries the returning signal, this tight coupling balances the pair.



Setting up Diff Pairs

Diff Pairs in Easy-PC are defined using **Net Class** settings within your **Design Technology** and they must be enabled for use by selecting the **Differential Pairs** check box.

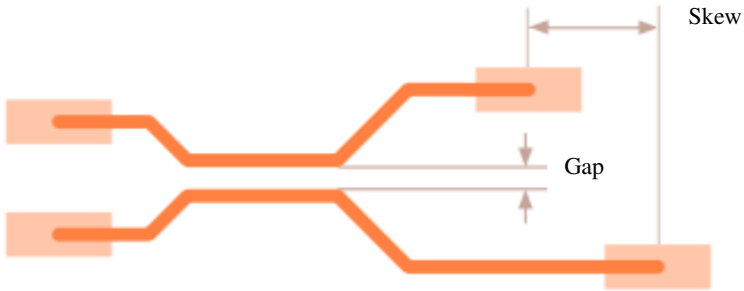
If **Differential Pairs** are enabled on a **Net Class**, Easy-PC will automatically detect pairs of nets using this Net Class and then based on their **Net Names**. It searches for pairs of nets with the suffix **_P** and **_N** and automatically designates those as a pair, for example, DQS_P and DQS_N



Once enabled, controls are available for separate **Diff Pair Gap** and **Skew Lengths** for pairs on the Net Class.

A **gap** different to the main Track to Track Spacing rule may be required for specific Diff Pairs.

The **Skew** value between each pair can be set, this effectively enables you to set a maximum delay distance. This can be as log or as short as required depending on your timing requirements.



A Net Class has several settings relating to Diff Pairs:

Differential Pairs: check this box to enable and disable Diff Pairs on this Net Class.

Diff Pair **Gap**: this is the gap between the edges of the tracks of a pair. This overrides the normal Track to Track Spacing rule.

Diff Pair **Skew Length**: the check box enables or disables this extra parameter and once enabled, provides you with a value for the calculation of skew between tracks of a routed pair.

The **Design Technology** dialog displays the **Diff Pair Gap,** whether **Diff Pair Skew** is enabled and the **Max Skew**.

Pad Styles | Text Styles | Line Styles | Hatch Styles | Track Styles | Layers | Layer Types | Layer Spans | Nets | Net Classes | Value Names

	LenDiff Enabled	Length Diff	MinMax Enabled	Min	Max	Own Colour	Colour	Tented	Diff Pairs	Diff Pair Gap	Diff Pair Skew	Max Skew
X	Yes	0.3000	No	0.0000	0.0000	No		None	Yes	0.2000	Yes	1.5000
X	No	0.0000	No	0.0000	0.0000	No		None	No	1.0000	No	0.0000
X	No	0.0000	No	0.0000	0.0000	No		None	No	0.2000	Yes	1.5000

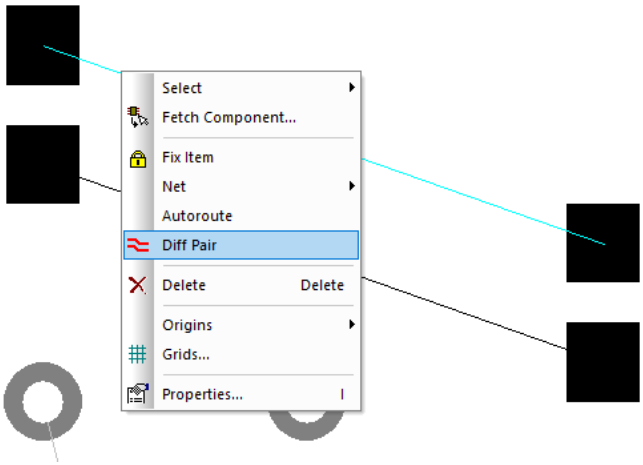
Other length rules for a Net Class can also be utilised within a Diff Pair, such as **Min/Max Track Lengths**.

Creating Diff Pairs

When a Diff Pair is added to the design, it will be the actual centre of the two lines that is on grid, not the Diff Pairs themselves. This means you should try and use grid multiples when routing so that odd segments are avoided.

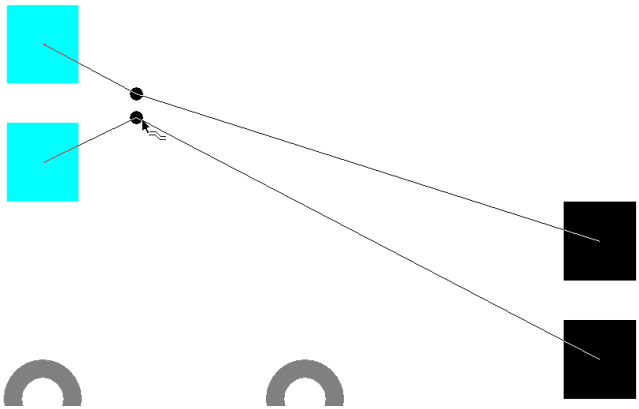
To start a Diff Pair, you can choose a number of options:

- Pick one of the Diff Pair connections and choose **Diff Pair** from the context menu:



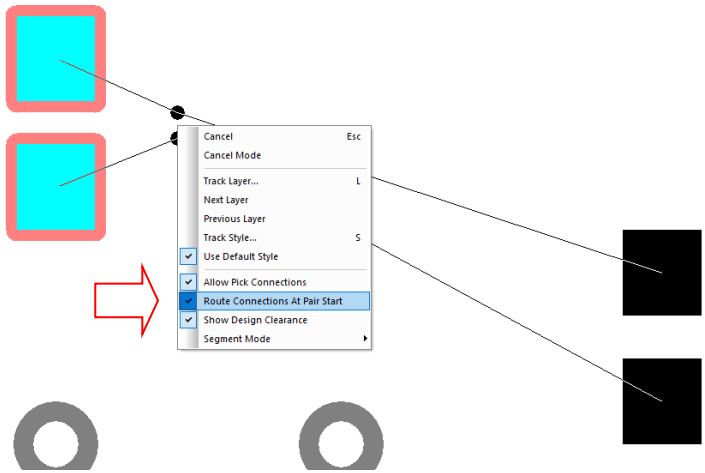
- Pick one of the pads of the Diff Pair connections and choose **Diff Pair** from the context menu.
- Pick either the connection or Pad of the Diff Pair connections and select **Diff Pairs** from the **Edit** menu.

Once the **Diff Pair** option is selected you will see two ‘dots’, these present the start of the Diff Pair, a modal cursor indicated you are in ‘pairing’ mode.



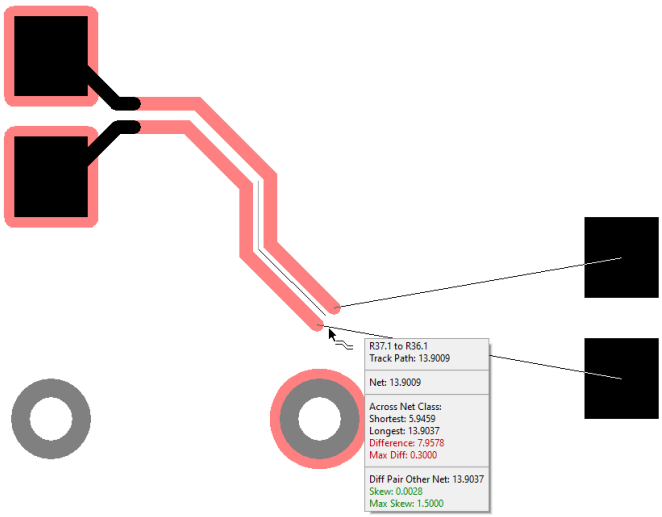
From here, you can ‘draw’ in your pair. This operates the same as when adding tracks with most tracking modes available; **Change Layer**, **Segment Modes, Display Clearances** etc. These are available on the context menu.

When you see the start points, you can also immediately right click to view the context menu. This contains the **Route Connection At Start Point** mode.



When this is selected, the start of the Diff Pair will be taken from the pads and will automatically route out (where it can) to the Diff Pair tracks.

Draw the tracking in as required:



During routing, this uses the **Diff Pair Gap** (on the Net Class, if defined) or if not defined, it uses the **Track to Track spacing** for the current Net Class and Layers (this way you can have different gaps on different layers).

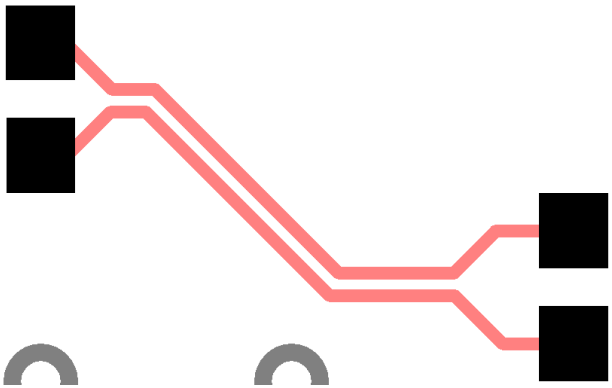
During editing, you can use **Change Layer** using the context menu option or <L> key. Two vias are inserted along with suitably spaced tracking. When using **Change layer,** if not **All Segments** then it inserts a 45-degree via pattern where vias are separated using the **Via to Via Spacing**.

Diff Pairs – Double Click mode

While editing a Diff Pair, it uses the **Edit Track** options in **Preferences** to decide what to do when **double-click** is used to finish, i.e. **Finish Here** or **Auto Complete**.

Use the **Show Legal Completion Path** option to identify the path that will be taken on completion.

Using **Auto Complete** will enable the pairing to finish on the target pads for you:

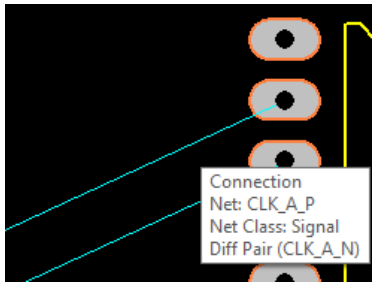


Diff Pairs – Partial routing

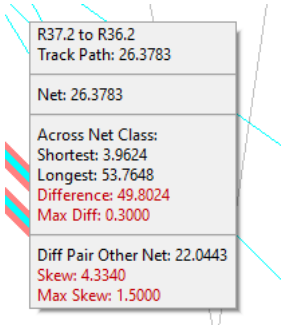
If a diff Pair has been partially completed, within Add Diff Pair mode, you can pick the two junctions at the end of the paired section to start from. Move the gather point back over the junctions to add paired segments to the existing ones and continue Diff Pair routing.

Diff Pairs – Design Tooltips

When selecting a connection that forms part of a Diff Pair, **Design Tooltips** will now show information about the Diff Pair if it detects a match on the Net Class name:

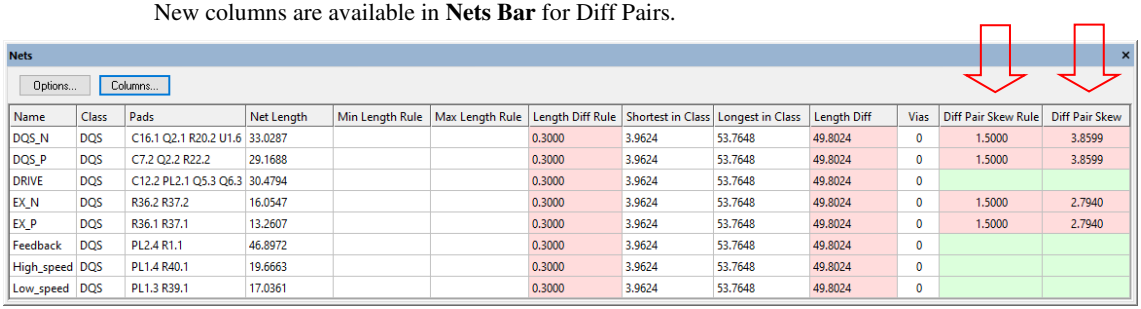


When editing any track that is in a Diff Pair the track limits show the length of the other net and, if defined, the skew and max skew values. The skew is measured as simply the difference in total net lengths.



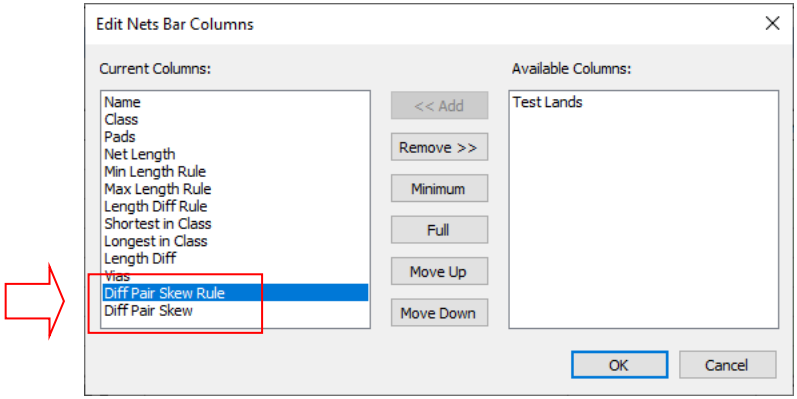
Diff Pairs – Nets Bar

New columns are available in **Nets Bar** for Diff Pairs.



Name	Class	Pads	Net Length	Min Length Rule	Max Length Rule	Length Diff Rule	Shortest in Class	Longest in Class	Length Diff	Vias	Diff Pair Skew Rule	Diff Pair Skew
DQS_N	DQS	C16.1 Q2.1 R20.2 U1.6	33.0287			0.3000	3.9624	53.7648	49.8024	0	1.5000	3.8599
DQS_P	DQS	C7.2 Q2.2 R22.2	29.1688			0.3000	3.9624	53.7648	49.8024	0	1.5000	3.8599
DRIVE	DQS	C12.2 PL2.1 Q5.3 Q6.3	30.4794			0.3000	3.9624	53.7648	49.8024	0		
EX_N	DQS	R36.2 R37.2	16.0547			0.3000	3.9624	53.7648	49.8024	0	1.5000	2.7940
EX_P	DQS	R36.1 R37.1	13.2607			0.3000	3.9624	53.7648	49.8024	0	1.5000	2.7940
Feedback	DQS	PL2.4 R1.1	46.8972			0.3000	3.9624	53.7648	49.8024	0		
High_speed	DQS	PL1.4 R40.1	19.6663			0.3000	3.9624	53.7648	49.8024	0		
Low_speed	DQS	PL1.3 R39.1	17.0361			0.3000	3.9624	53.7648	49.8024	0		

The **Edit Nets Bar Columns** dialog includes **Diff Pair Skew Rule** and **Diff Pair Skew.**

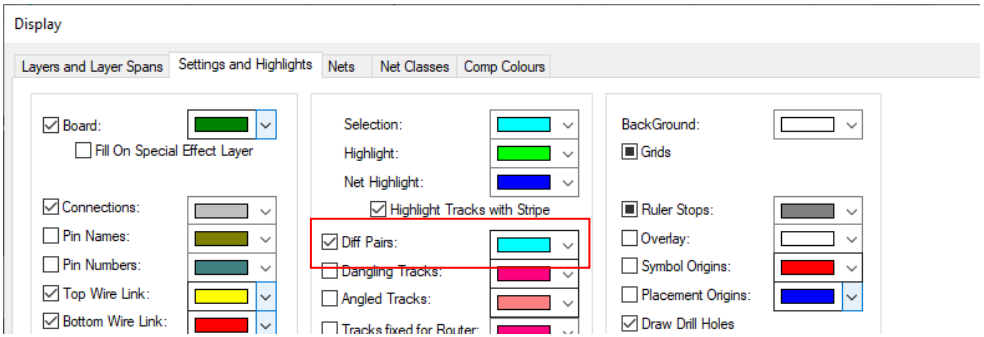


For a Diff Pair net, **Diff Pair Skew Rule** shows the rule on a net if its Net Class has a skew rule defined and **Diff Pair Skew** shows the actual difference between the lengths of the two nets forming the Diff Pair.

As with all Length columns in this dialog, the cells change colour to display whether the Skew is within the limits defined.

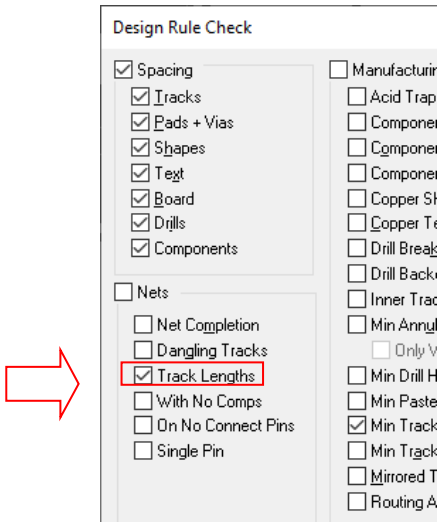
Highlight Colour of Diff Pairs

There is a highlight colour in the **Display** (Colours) dialog that allows you to see paired connections and tracks (using the centre stripe) for diff Pairs using a highlight colour to make them more visible.

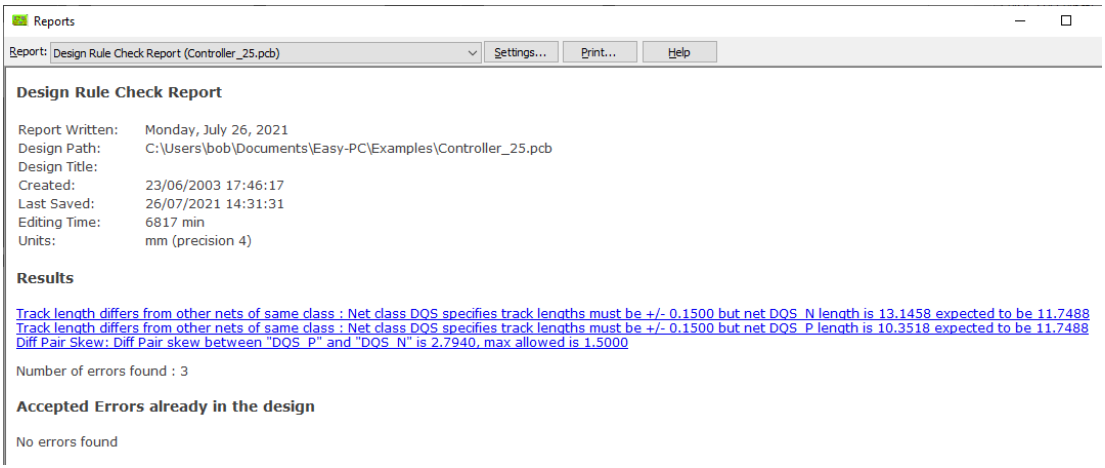


Design Rules Checking of Diff Pairs

The DRC **Track Lengths** check now checks for Diff Pair skew errors.

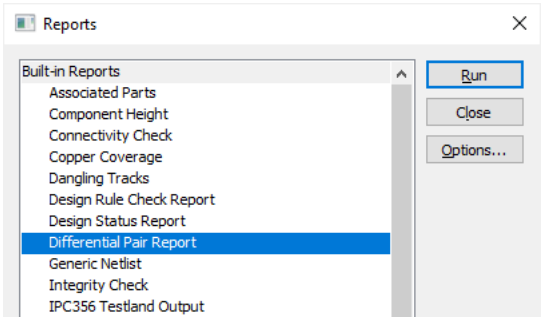


When run, the DRC report shows Skew differences:

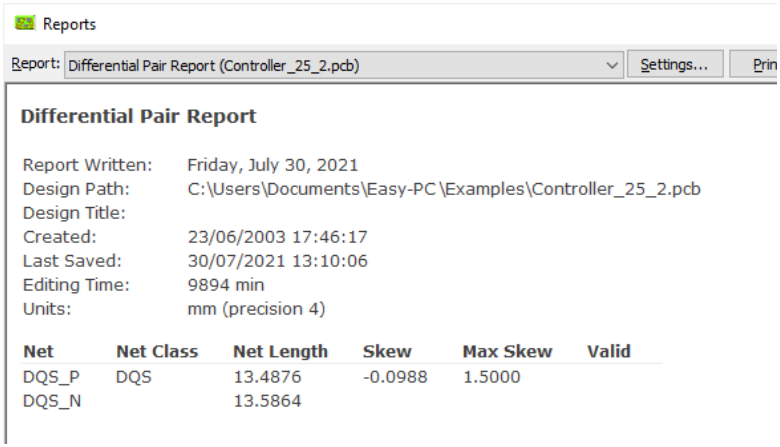


Diff Pair Report

A new report is available in the **Output** menu and **Reports** called **Differential Pairs Report**.

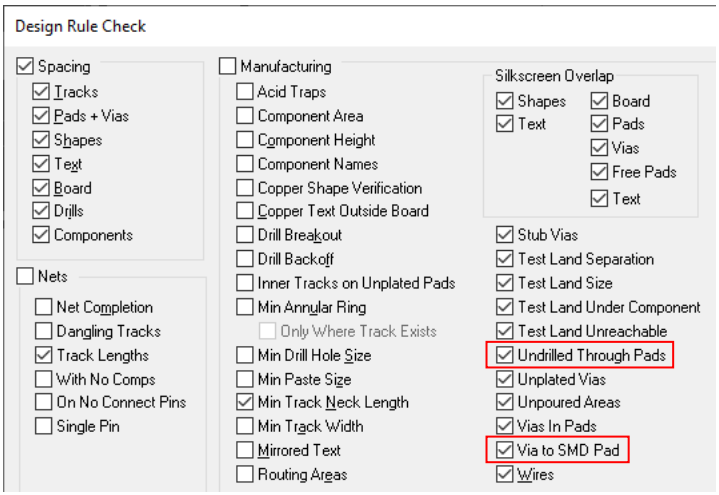


When run, the report will display values relating to the Diff Pair nets in your design:



New Design Rule Checks

New Design Rules Checks have been added for Undrilled Pads, Via to SMD Pad and a Track Spacing check for Teardrops.

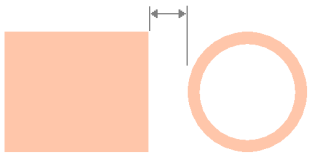


DRC check for Undrilled Pads

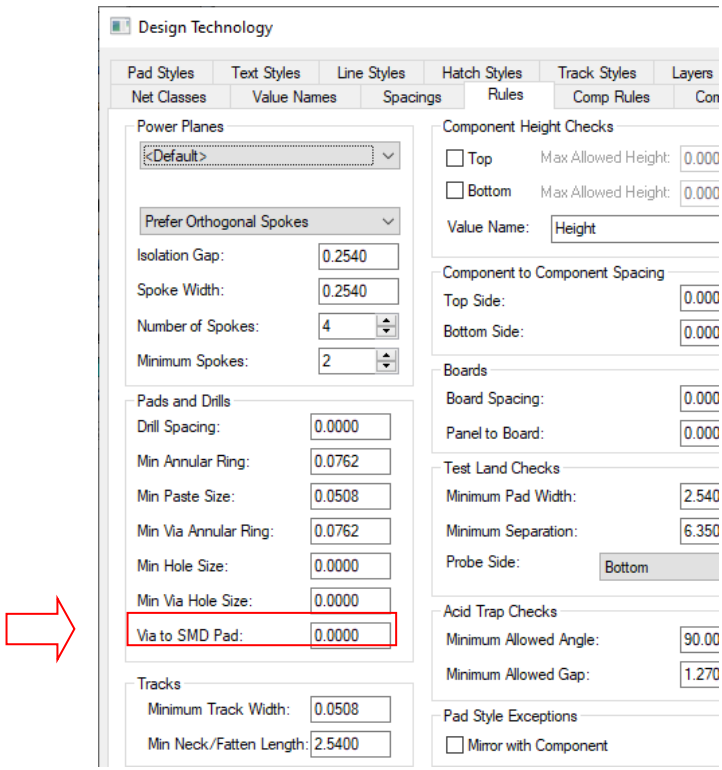
The new **Undrilled Pads** check in the **DRC** dialog looks for through-board pads (on layer [All]) that use a pad style with no drill hole. This avoids the mistakes where pads have not been specified for drilling and the final manufactured PCB has no drills or plating when required. This check is highly recommended on all boards that have any plated-through holes.

DRC check for Via to SMD Pad distance

There is a new check in **DRC** that looks for a minimum distance from a **Via to an SMD Pad** on the same net. This means you can set a value that is acceptable by the manufacturer to ensure that solder bridges don't exist.



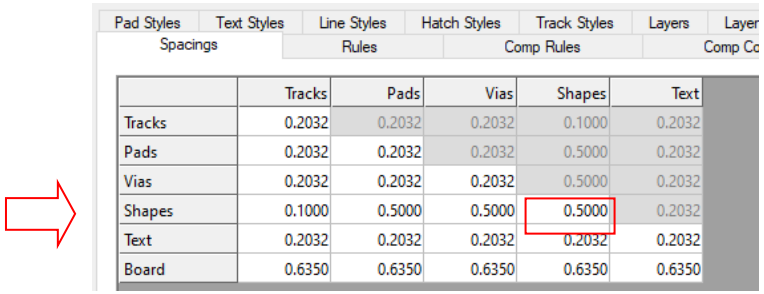
To use this option, it has to be enabled and a value defined on the **Rules** tab of the **Design Technology** dialog. This value specifies the distance from the SMD pad edge to the pad edge of the via.



DRC Track Spacings for Teardrops

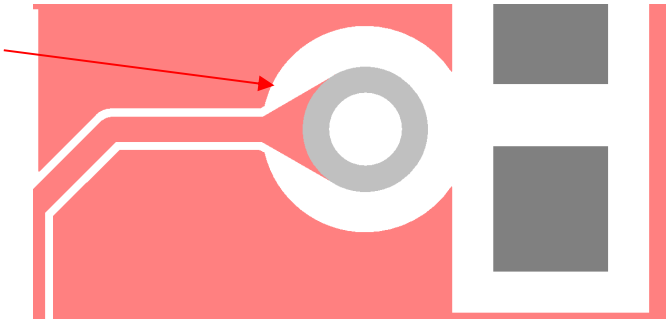
A new check box on **Spacings** page of **Design Technology** dialog instructs Easy-PC to use the spacings defined for tracks when dealing with teardrops. This is used instead of the default setting which is to use the Copper shape spacings.

The pouring will use the **Shape to Shape** spacing, like this if not checked:

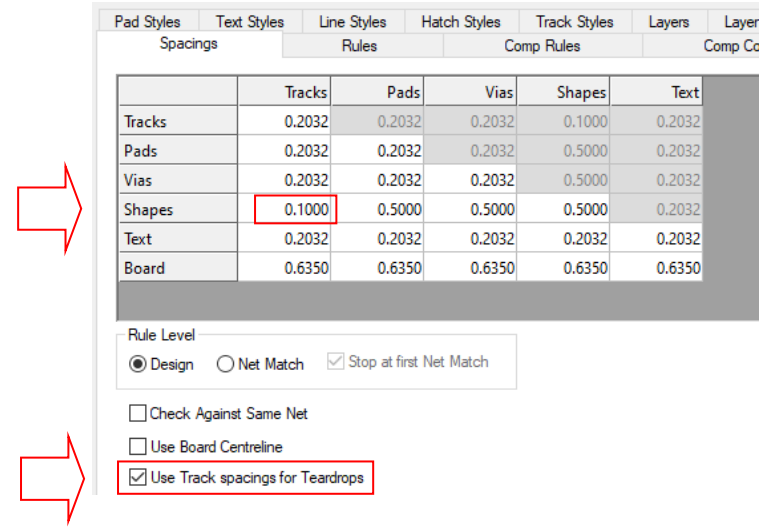


	Tracks	Pads	Vias	Shapes	Text
Tracks	0.2032	0.2032	0.2032	0.1000	0.2032
Pads	0.2032	0.2032	0.2032	0.5000	0.2032
Vias	0.2032	0.2032	0.2032	0.5000	0.2032
Shapes	0.1000	0.5000	0.5000	0.5000	0.2032
Text	0.2032	0.2032	0.2032	0.2032	0.2032
Board	0.6350	0.6350	0.6350	0.6350	0.6350

The result will look like this:

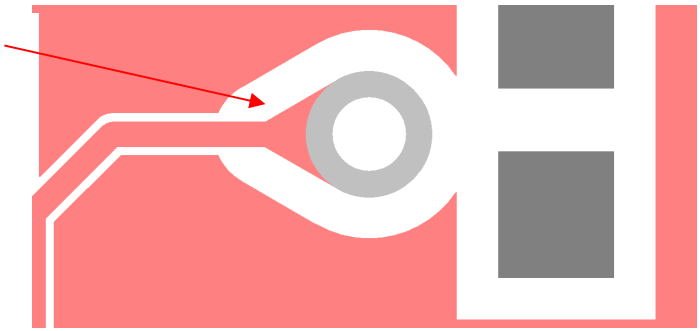


If the **Use Track spacing for Teardrops** check box is selected, it will use the **Shape to Track** spacing instead:



	Tracks	Pads	Vias	Shapes	Text
Tracks	0.2032	0.2032	0.2032	0.1000	0.2032
Pads	0.2032	0.2032	0.2032	0.5000	0.2032
Vias	0.2032	0.2032	0.2032	0.5000	0.2032
Shapes	0.1000	0.5000	0.5000	0.5000	0.2032
Text	0.2032	0.2032	0.2032	0.2032	0.2032
Board	0.6350	0.6350	0.6350	0.6350	0.6350

In our example, we've made the Shape to Track spacing larger but equally, it could well be smaller.



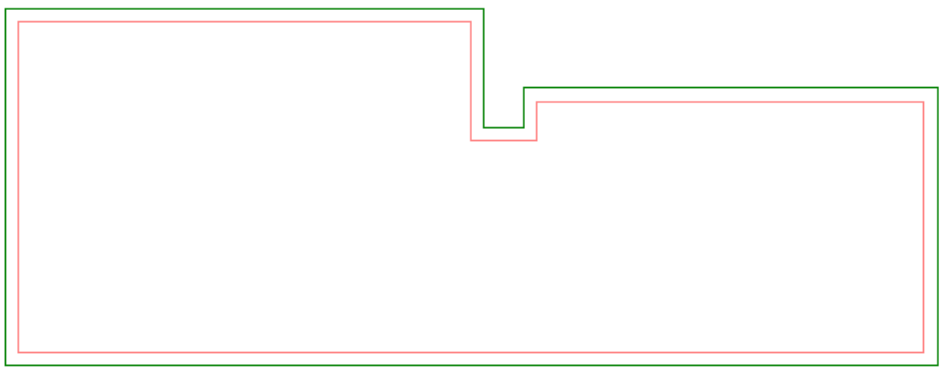
Highlight Net across Project

The **Net Highlight** command has been enhanced to now work across **all open designs** in a **Project** for user-defined net names.

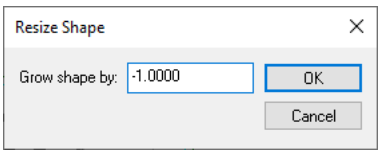
Resize Shape

The ability has been added to **grow** or **shrink Area**, **Board**, **Copper** or **Doc shapes** using the **Resize Shape** command.

This flexible new option enables you to select a Shape, the board outline for example, copy it, change its Type to a Copper Pour Area and then shrink it by a set value. An exact copy of the board outline will then be made but slightly smaller. This is shown in the example below:



To use this option, select the shape and right click, select **Resize Shape** from the context menu.



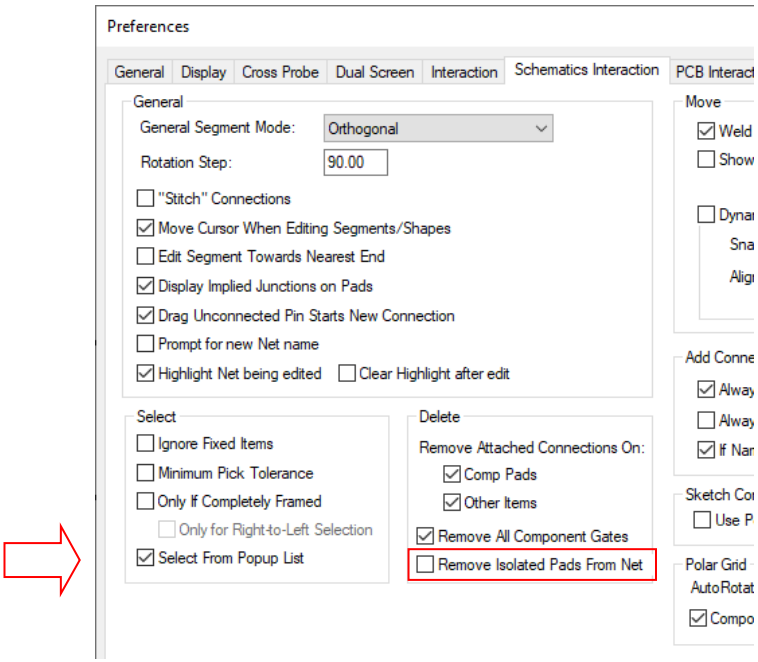
Type in the value required. A negative value using the **minus** – sign will **shrink** the shape. A positive value (with **no minus sign**) will **grow** the shape by the value typed. This will use the current design units as displayed in the bottom right-hand corner of the **Status** bar.

This option can be used in the Schematic design and symbol editors too.

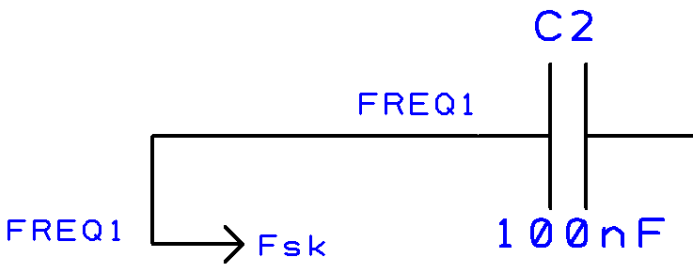
Delete in SCM, option to keep Net

There is a new check box **Remove isolated pads from net** in the **Delete** settings for **SCM Interaction Preferences**. Previously, this was only available in **PCB Interaction**. The default value is checked (to match current behaviour where net names are removed if isolated).

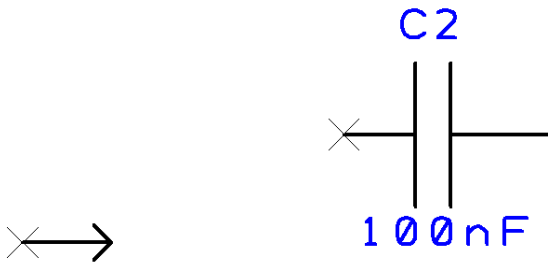
This option is used to (optionally) retain a net name on a pin when deleting the connection to it. The default action is for the net name to be removed.



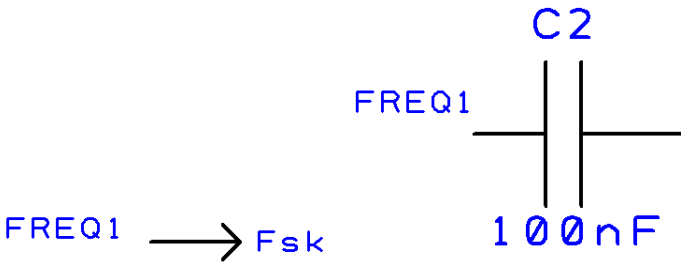
To demonstrate this, we start with the FREQ1 net between two pins:



With the option **checked**, the net and net names are fully removed.

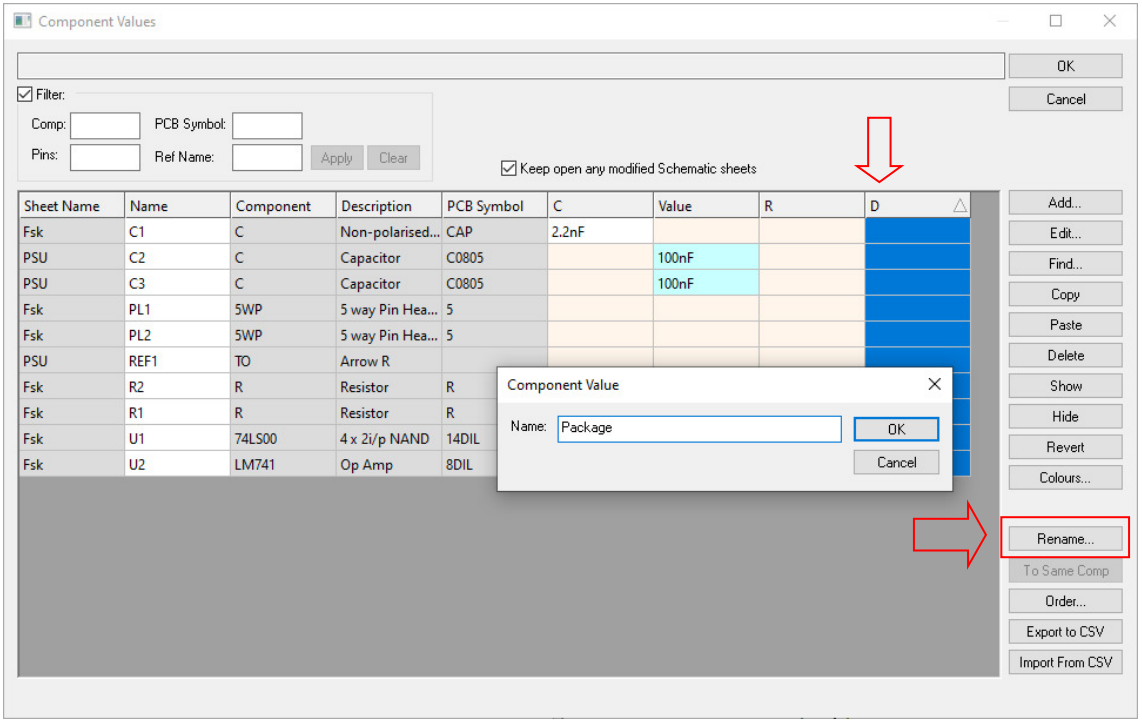


With the option left **unchecked**, the net names on the pins are retained when the connection is deleted.



Component Values – Rename option

In the **Component Values** dialog, there is a new **Rename** button that allows you to change the name of an existing attribute (value). This is available for both **Library Components** and Components in a **design**.



Sheet Name	Name	Component	Description	PCB Symbol	C	Value	R	D
Fsk	C1	C	Non-polarised...	CAP	2.2nF			
PSU	C2	C	Capacitor	C0805		100nF		
PSU	C3	C	Capacitor	C0805		100nF		
Fsk	PL1	5WP	5 way Pin Hea...	5				
Fsk	PL2	5WP	5 way Pin Hea...	5				
PSU	REF1	TO	Arrow R					
Fsk	R2	R	Resistor	R				
Fsk	R1	R	Resistor	R				
Fsk	U1	74LS00	4 x 2i/p NAND	14DIL				
Fsk	U2	LM741	Op Amp	8DIL				

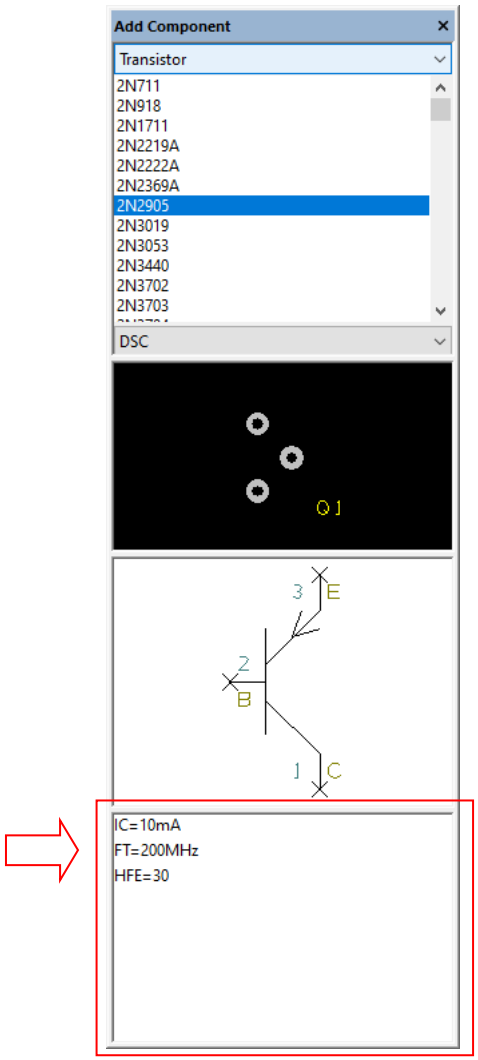
By selecting the **Value** name in the column header, the **Rename** option becomes available. Selecting this provides you with a **Name** dialog from which to type the new name.

The result is like this:

Sheet Name	Name	Component	Description	PCB Symbol	C	Value	R	Package
Fsk	C1	C	Non-polarised...	CAP	2.2nF			
PSU	C2	C	Capacitor	C0805		100nF		
PSU	C3	C	Capacitor	C0805		100nF		
Fsk	PL1	5WP	5 way Pin Hea...	5				
Fsk	PL2	5WP	5 way Pin Hea...	5				
PSU	REF1	TO	Arrow R					
Fsk	R2	R	Resistor	R			20K	
Fsk	R1	R	Resistor	R			10K	
Fsk	U1	74LS00	4 x 2i/p NAND	14DIL				SM
Fsk	U2	LM741	Op Amp	8DIL				DIL

Note, when used in the design and Values names are renamed, this is not reflected in the component Library.

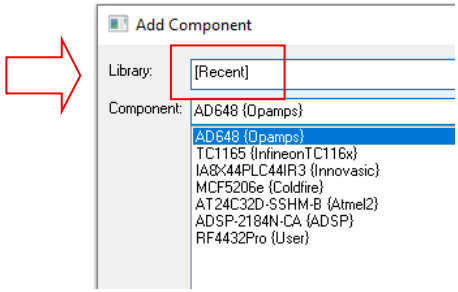
Add Component Bar shows Values



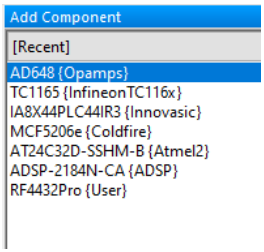
The **Add Component Bar** now displays values of the selected component in a list below the PCB and SCM preview windows. This is in common with the Add Component and other similar dialogs.

Add Component – List of Recent items

The **Add Component** dialog now has the ability to remember the most recent components you have added to your designs. It will present those as an easily accessible list when you select **[Recent]** from the library list in the Add Component dialog:



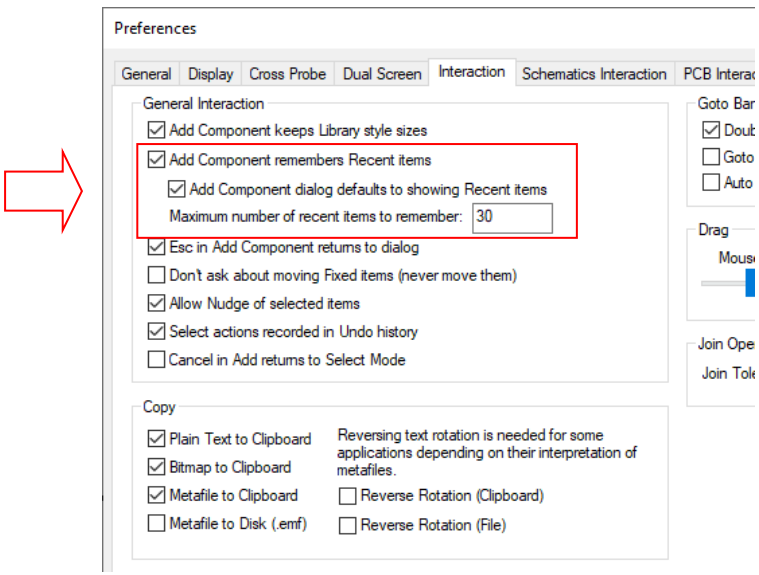
The same list is available in the **Add Component Bar** when you choose **[Recent]** from the top of the library list:



The **[Recent]** list is also available in the **Change Component** dialog accessed from **Component Properties**.

Each component you add or use from any of these locations will be automatically added to the top of this list, so you will always see your most recently added or used components right at the top.

The operation of this feature is controlled by settings on the **Interaction** tab of the **Preferences** dialog.



These allow you to enable/disable the whole ‘recent items’ feature, control how many items are to be remembered (the default is 30) and to specify whether to always display the [Recent] list when you open the Add Component dialog.

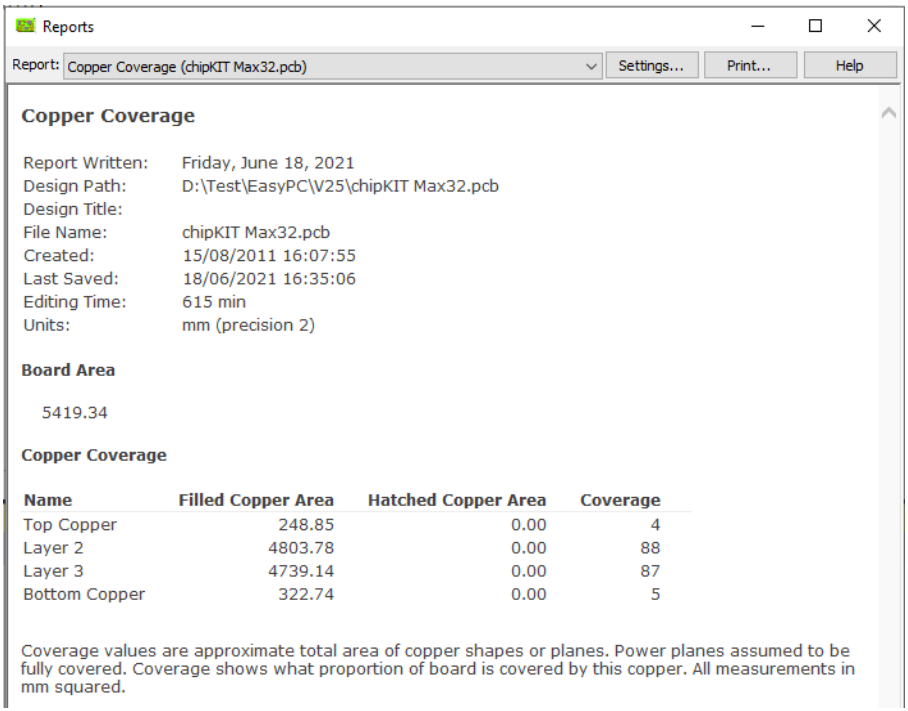
You may find it useful to use the Add Component dialog and bar together like this: in **Preferences**, **Interaction**, uncheck the box for showing recent items on entering the **Add Component** dialog. This allows you to use the dialog as before, browsing different libraries or using **Find** to locate items. On the **Add Component Bar**, set the library name to [Recent] to see your recently added items.

Now your recent items are available to drag into your design from the **Add Component Bar**, but if you want to add a component that you haven't recently accessed then open the Add Component dialog to find it. When you add that to your design it will automatically appear at the top of the Recent list in the dockable bar so it is readily accessible next time you want it.

Copper Coverage Report

A new report on the **Reports** dialog, **Copper Coverage**, lists the total area of filled/hatched copper shapes by layer, including the percentage of the board area covered by this copper.

When producing multi-layer PCBs, it is highly recommended to 'balance' the percentage of copper on each layer so that the board doesn't bend and warp during manufacture. When using copper areas, it is preferable to use hatched copper too. The report also shows hatched copper coverage.

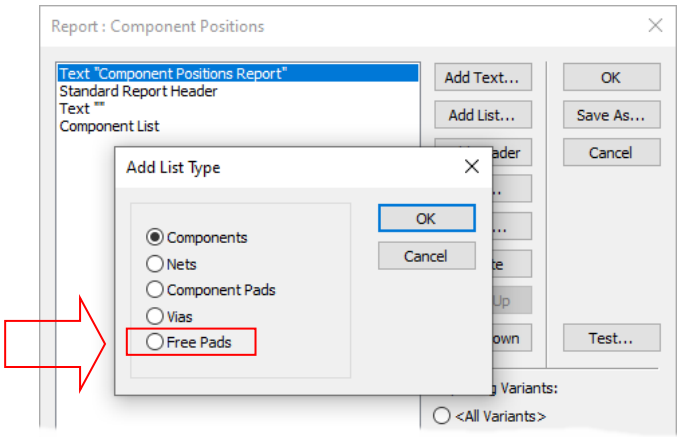


Note: it should be noted that as stated in the report, the coverage values are approximate as a best calculation. Power plane layers are assumed to be full coverage.

Further information can be found about calculating copper coverage and balancing at this web address: https://www.multi-circuit-boards.eu/en/pcb-design-aid/copper-balance.html
Acknowledgement and copyright to Multi Circuit Boards.

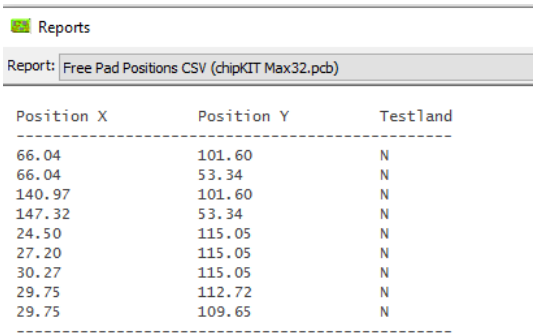
Free Pads in Custom Report

For **User Reports**, you can now produce a report showing the properties of **Free Pads**:



This is available when you use the **Add List** option.

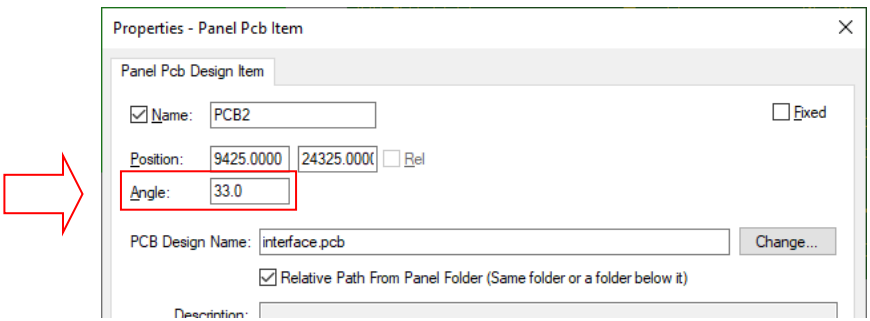
When run, the report can be used to produce a list of free pads that may have been marked as use for Testlands or fiducial markers and suchlike.



Position X	Position Y	Testland
66.04	101.60	N
66.04	53.34	N
140.97	101.60	N
147.32	53.34	N
24.50	115.05	N
27.20	115.05	N
30.27	115.05	N
29.75	112.72	N
29.75	109.65	N

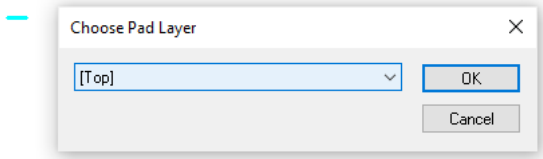
Angle of PCB Item in a Panel Design

In a **Panel** design, in **Properties** for a **Panel PCB Design** item, you can now set the **Rotation** to be any angle you want, you are no longer restricted to multiples of 90 degrees.



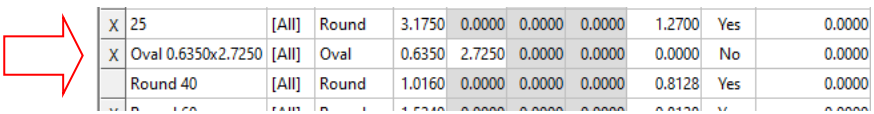
Shape To Pad

The new context menu command **Shape to Pad** in **Select** mode is available when there is at least one **single line shape** or **unattached track** selected.



This command will ask you to choose the layer for the pads (based on the layer of the first valid selected line), then it will take each line and replace it with an oval pad of the same dimensions on the chosen layer.

For each pad created, a new **Pad Style** will be added to the **Design Technology** using the dimension of the shape selected.

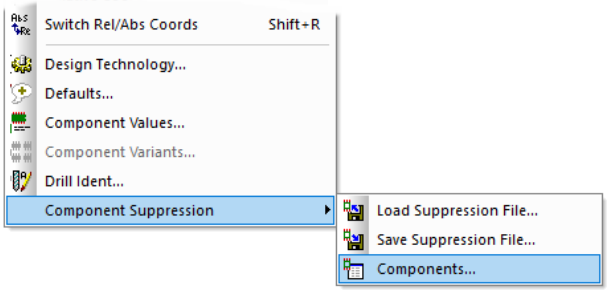


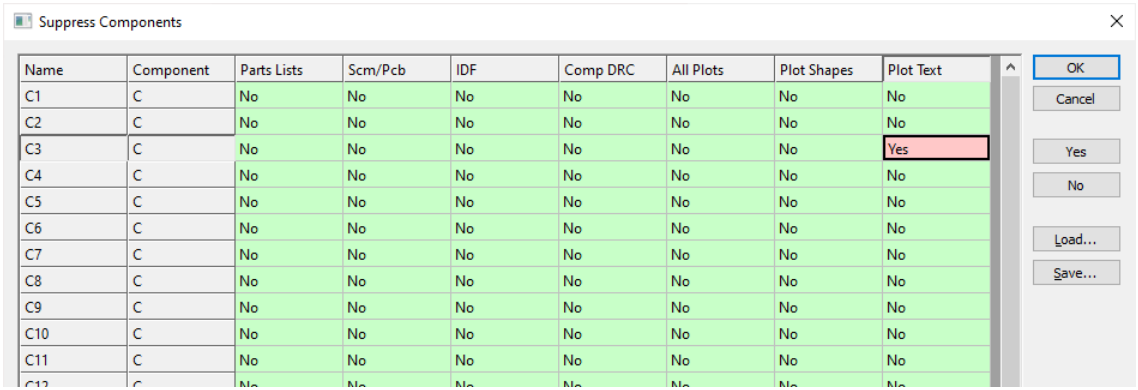
X	25	[All]	Round	3.1750	0.0000	0.0000	0.0000	1.2700	Yes	0.0000
X	Oval 0.6350x2.7250	[All]	Oval	0.6350	2.7250	0.0000	0.0000	0.0000	No	0.0000
	Round 40	[All]	Round	1.0160	0.0000	0.0000	0.0000	0.8128	Yes	0.0000

This function is intended for use after importing Gerber files and the need to convert short track segments into pads. These would be isolated segments that would have originally been pads in the host system before being written to Gerber. During import, this cannot be done automatically and is ideally suited to a manual ‘by eye’ observation, this new option will aid this.

Component Suppression dialog

A new command is available on the **Settings** menu and **Component Suppression** option called **components**. On selection, this displays a dialog with a table of components and their current suppression settings to make it easy to manage them all in one location. In previous releases, you are restricted to doing them through the **Component Properties** dialog.



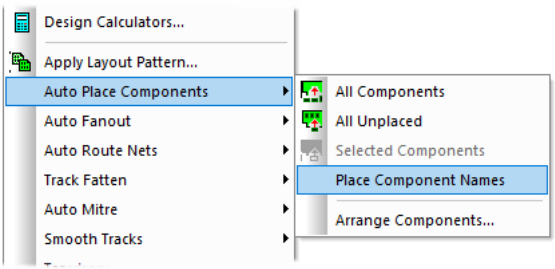


The **Yes** and **No** buttons toggle the status of a selected cell. The cell can be toggled by double clicking it also.

The **Load** and **Save** buttons on this dialog do the same as the existing Load/Save Suppression File commands that were available in previous version. This allows you to save settings to a text file, manipulate them, and read them back in.

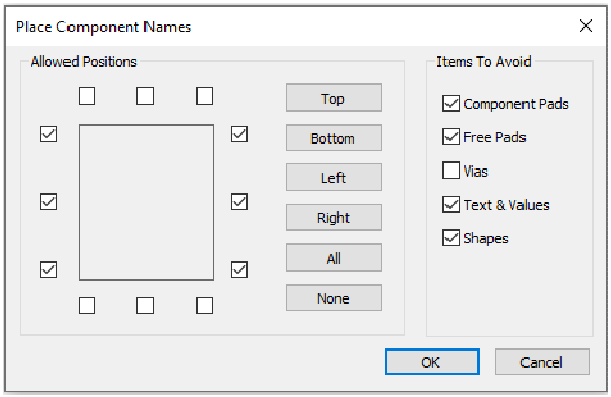
Place Component Names in PCB

There is a new command on the **Tools** menu, **Auto Place Components** option, called **Place Component Names**. This will run spin through all the components in the design (or only the selected ones) and will rationalise the orientation and position of all the value positions.

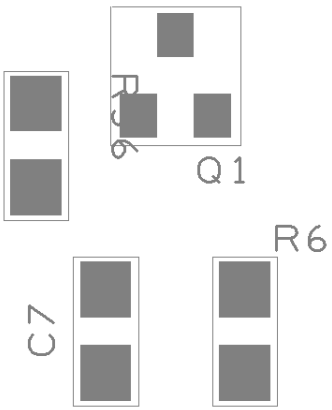


The means you can place all of your component names in a PCB for example, based on the dialog settings. If you have rotated components, the component name will also be rotated. This dialog enables them all to be repositioned and rotated automatically.

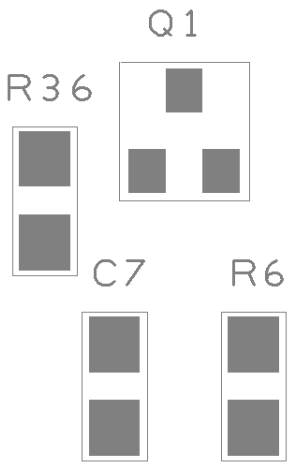
A dialog is presented allowing you to select any of 12 possible positions for the names around the component, and which types of items to avoid when attempting to place them. The component position is the one used on the component in the design and not the symbol origin.



In the example below, the component position setting has been set to the top right corner (the Allowed Position), below is the starting design with a good mix of component names with various rotations and pad violations:



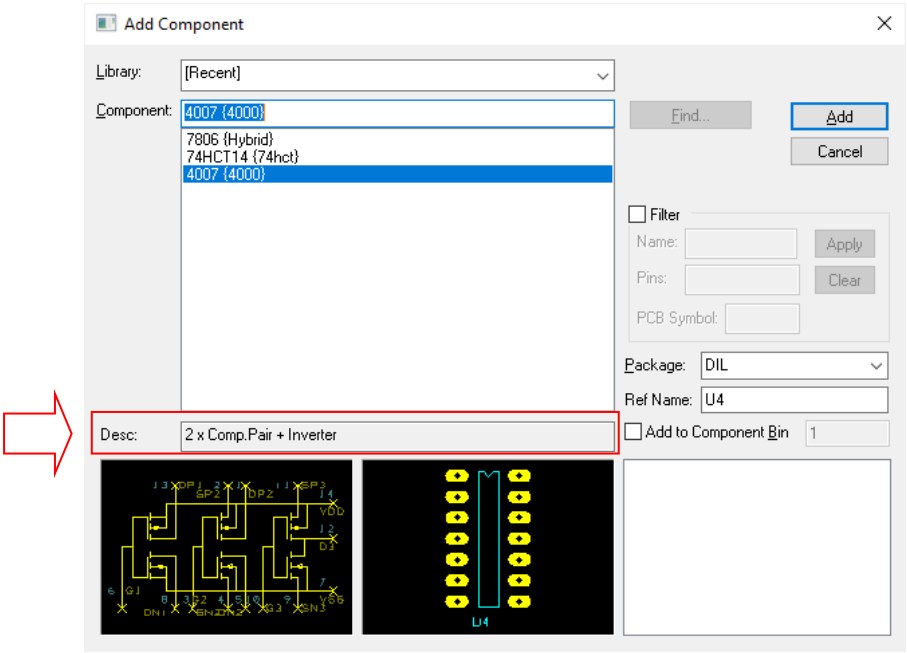
The image below shows the affect after running this tool:



Component Description on dialogs

On dialogs where components are added or replaced, the **Description** from the component is now displayed along with the existing information (package etc.).

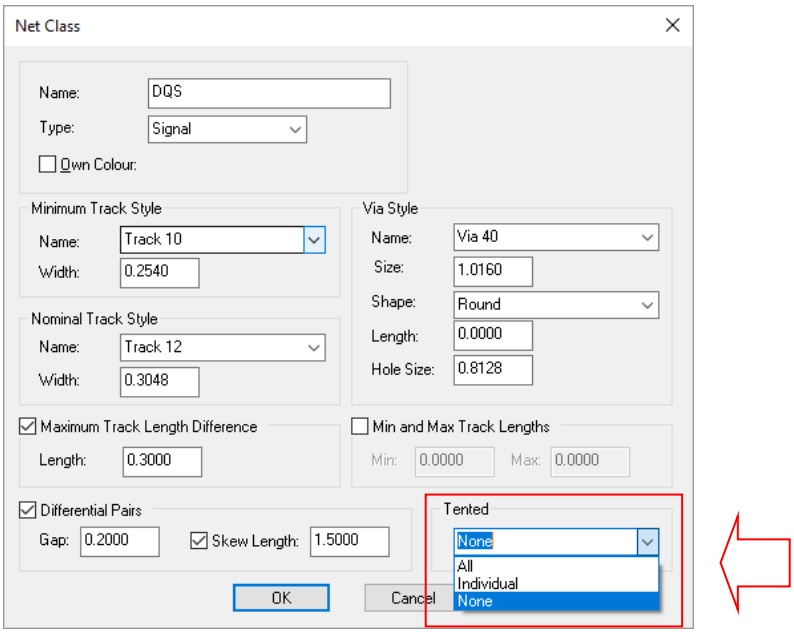
This change applies to **Add Component**, **Change Component** (from **Properties**) and **Replace Component**.



Tented Vias

This is a simple method of specifying whether or not vias are to be 'tented'. A tented via is one that is covered with solder resist during the manufacturing stage. In order for this to happen, the via that is to be tented must be identified in the design and a specific output generated.

The **'tented'** property is defined in the **Net Class**. When vias in the design use this Net Class, the output file generated in the **Plotting and Printing** dialog can use this flag as a selection option.



Each Net Class has a new property of **Tented** which can be set to one of three values:

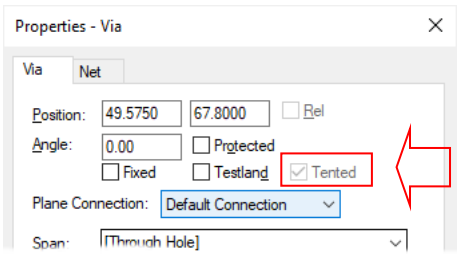
None: vias on this Net Class are automatically not tented so they will appear on plots as normal.

All: all vias on this Net Class are automatically classified as tented.

Individual: the Tented flag is set by you as required. This is set on **Via Properties** for each via

Via Properties

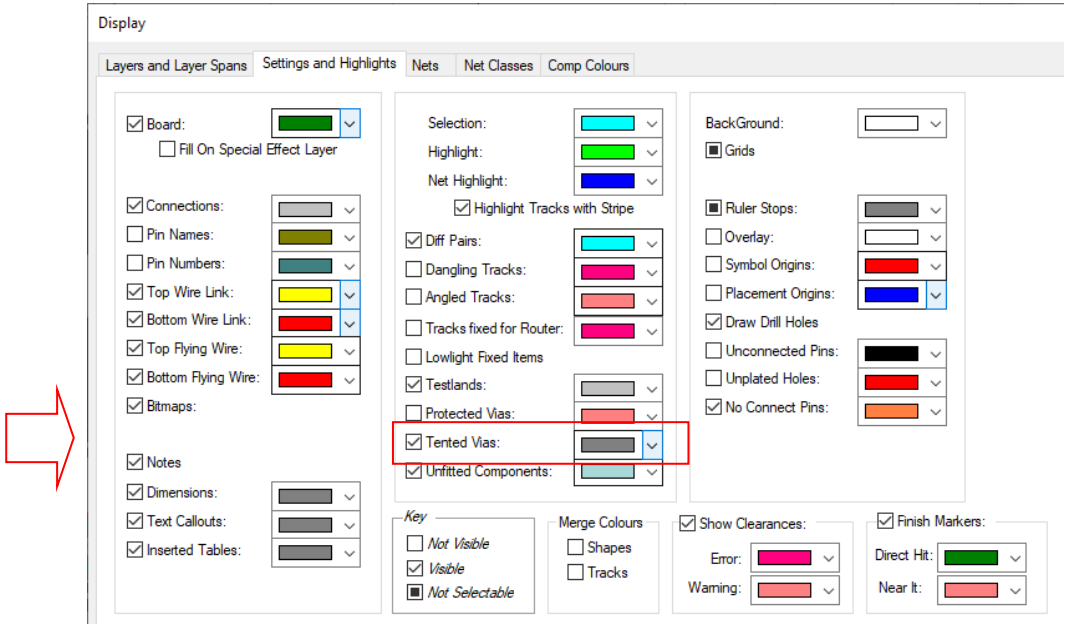
When **Tented** is set to **None** or **All** on the Net Class, the tented state of a via is displayed on the **Via Properties** dialog but the check box is greyed out because it is inheriting its setting from the Net Class.



When the **Tented** property is set to **Individual**, this check box then becomes available for setting.

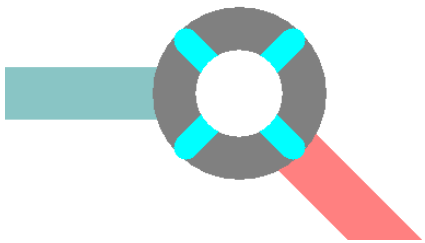
Tented Via Colours

On the **Display** (Colours) dialog there is a new check box and colour setting for **Tented Vias**:



Tented Vias in the Design

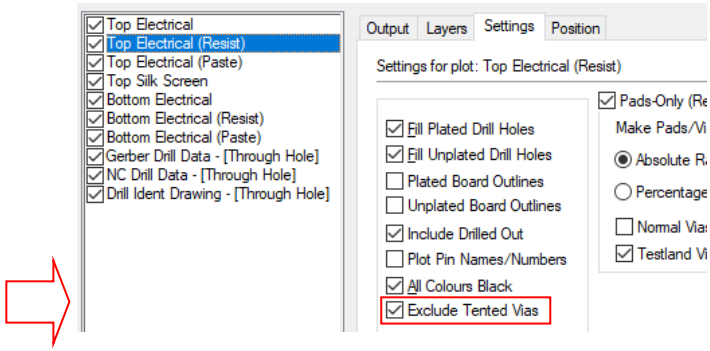
If this highlight is enabled (checked) in Display, any via that is tented (either individually on **Properties** or from its **Net Class**) will be marked with an X (cross) in the same way as we do to highlight unconnected pads:



Note, this is the only visual indication of the ‘presence’ of the via shape on a resist layer, the rest of the processing is done at the plotting stage. In other words, if you display only your top resist layer, vias that are marked as tented will still be drawn.

Plotting Tented Vias

In **Plotting and Printing**, there is a new check box on the **Settings** tab for each plot that allows you to decide whether or not to exclude tented vias from that plot.

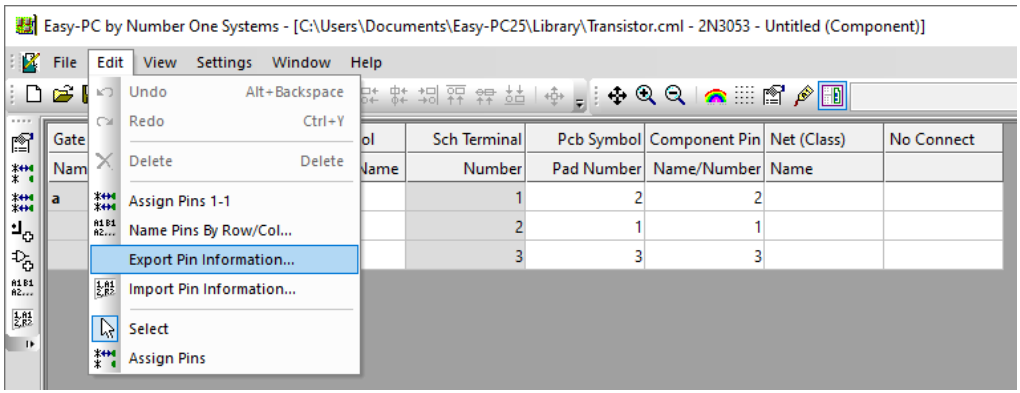


The default setting for this is unchecked (plots include tented vias) but when auto-generating plots it will try to determine which plots are Resist and will thus tick this box for you. This setting is also saved in **Plot Jobs** so it is easy to prepare a preferred set of plots and settings and save it for future use.

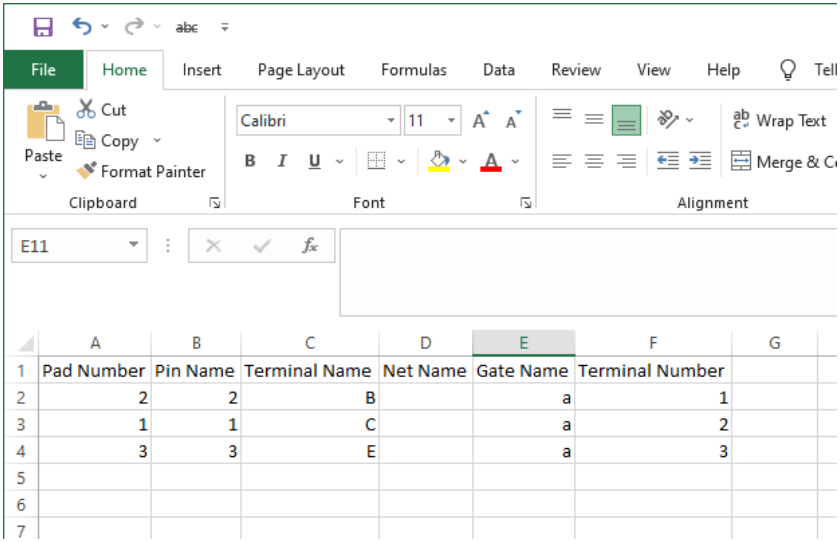
Export Pin Information from Component Editor dialog

In the **Component Editor**, it is already possible to import a CSV file of pin information to help assign names and numbers to pins in the component. Easy-PC V25 now allows you to **Export Pin** information in a CSV.

The new **Export Pin Information** command is available in the **Component Editor** on the **Edit** menu.



This helps you prepare this data by outputting a CSV file containing the existing pin details from the component, which you can then edit in a program like Excel, and read back into the component using the existing **Import Pin Information** command.



	A	B	C	D	E	F	G
1	Pad Number	Pin Name	Terminal Name	Net Name	Gate Name	Terminal Number	
2	2	2	B		a	1	
3	1	1	C		a	2	
4	3	3	E		a	3	
5							
6							
7							