

Easy-PC

V24.0 Supplement

Copyright © 1998-2020 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: 03/08/20 Issue 1

Contents

| | |
|--|----------|
| CONTENTS | 3 |
| CHAPTER 1. GETTING STARTED..... | 5 |
| Installation..... | 5 |
| Running Easy-PC 24.0..... | 6 |
| CHAPTER 2. NEW FEATURES IN EASY-PC V24..... | 7 |
| Introduction..... | 7 |
| File Open..... | 7 |
| Recent Files Lists Projects | 9 |
| Add to Project – Other Files Relative Path Name | 9 |
| Save Project to Zip File..... | 10 |
| Intelligent Design Backup Interval..... | 10 |
| BOM Composer Update..... | 11 |
| Sort Values on columns | 11 |
| New Line Item Number Field | 12 |
| Cross-probe zoom After Probe..... | 13 |
| New Nets Bar..... | 13 |
| Net Properties – Nets shown on other sheets | 17 |
| Goto Bar shows DRC Errors for all Open SCM Sheets | 17 |
| New Disconnect From Signal Nets option | 18 |
| Changes to Component Values dialog | 19 |
| Editable Ref Name Column | 19 |
| Component Values dialog sorting | 19 |
| Ref Name Filter..... | 20 |
| Colouring for Read-only Controls..... | 20 |
| Drag Connected Items to Component Bin..... | 21 |
| Rules for Colouring Components..... | 21 |
| Move Components Names within a Tight Group..... | 22 |
| Changes to Change Component & Replace Component..... | 23 |
| Change & Replace Component dialogs are now resizable | 23 |
| Values are shown in the Preview list..... | 23 |
| Check box to “Keep Local Values” | 23 |
| Rename Components by List | 24 |
| Changes to the New Component Wizard | 25 |
| Library Item time stamps | 25 |
| Prompt for Net Name on Add Connection | 25 |
| Design Rules Check Changes | 26 |
| Component-to-Component SMT Check..... | 26 |
| Include Integrity Check..... | 27 |
| Check for Unpoured Areas..... | 28 |
| Silkscreen Overlap Check..... | 28 |
| Individual Item Selection in Silkscreen Checks | 28 |
| Min Via Hole Size..... | 29 |
| Integrity Check suppress PCB-only | 29 |
| Change to Preferences for ‘Cancel in Add Track’ | 30 |
| Paste Shapes in Footprints | 30 |
| Mirror Pad Style Exceptions | 32 |
| Allow Reposition of Component Pads | 33 |
| Options for Drawing Connections by Side in PCB | 33 |
| Pour Copper Avoid Same Net Tracks Setting | 34 |
| Thermal Spoke values by Pad Style..... | 35 |
| Plot Include/Omit Drilled-out Pads | 36 |

4 Easy-PC V24.0 Supplement

| | |
|---|----|
| Output Drill Data to Gerber | 36 |
| Extra header info in Excellon..... | 37 |
| Gerber Viewer embedded in NumberOne.com Web Site..... | 37 |
| ODB++ IP Protection..... | 38 |
| Layers Bar - Hide Power Plane | 38 |
| Layers Bar - Toggle True Width with Ctrl Key | 39 |
| Named Layer Toggles | 39 |
| Footprint check – Copper to Pad Association | 40 |
| Hexagonal Pad Shapes | 40 |
| Highlight Net during Add and Edit Track..... | 40 |
| Prefer Visible Layer in Add and Edit Track..... | 41 |
| Move Treat Vias as Fixed | 42 |

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 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

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.

With the installer running, once the **Welcome** screen is displayed, double-click on the **Install Easy-PC - Version 24.0** option, or click then press **Run**. Following the instructions on the screen, you should use the same **Destination Folder** for the Program Files as your existing Easy-PC program files.

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

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.

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.

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, “Users\Public\Documents\Easy-PC” on Windows 7, 8 or 10 (or the local language equivalents) if you are installing for All Users, or into your own Documents folder if installing for current user only.

Running Easy-PC 24.0

Once installed, an icon will appear in the **Number One Systems** folder and the **Start** pane if using Windows 10, you may also wish to create an **Easy-PC** Shortcut icon on your desktop.

To start the program, double-click on the **Easy-PC** icon from the **Number One Systems** folder or the **Start** pane if using Windows 10.

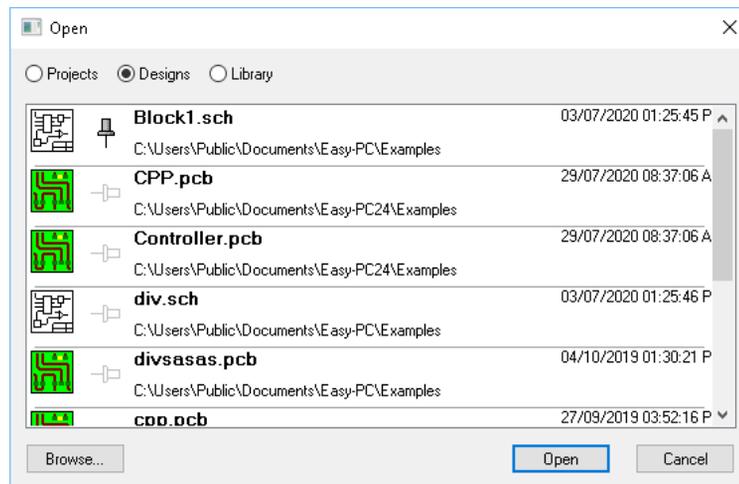
Chapter 2. New Features in Easy-PC V24

Introduction

All features are categorised as being applicable to Schematic (SCM), PCB, or more general use (unmarked).

File Open

A new **File Open** dialog that provides access to **recent Projects, Designs and Library** items has been added. This can also be optionally changed back to the standard **Open** dialog using **Preferences** if required (see below).



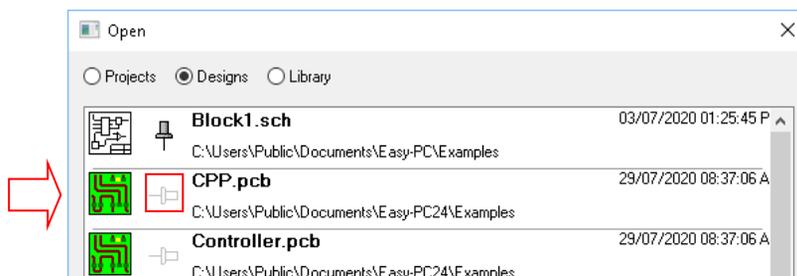
By using the **Ctrl** key, you can select or deselect more than one file at a time to open.

Once the file required is selected and press the **Open** button.

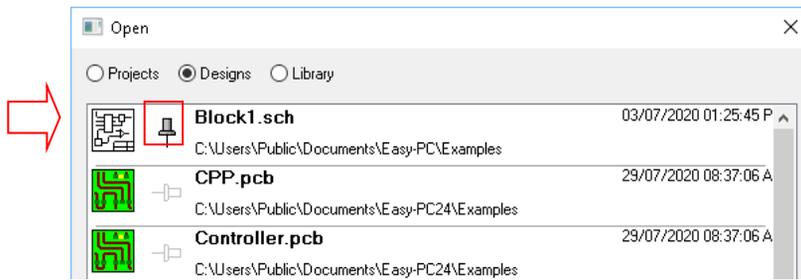
The **Browse** button will enable you to call up the standard **Open** dialog to search for your required design file.

'Pinning' items in the list

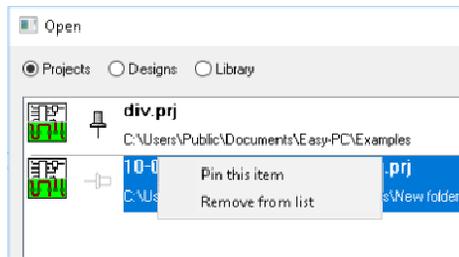
This new dialog includes the ability to 'pin' favourite items to the top of the list. This can be done by either clicking on the small pin next to the item name or by selecting the item once and right clicking to reveal the context menu.



On selection, the pin changes from unpinned to pinned:

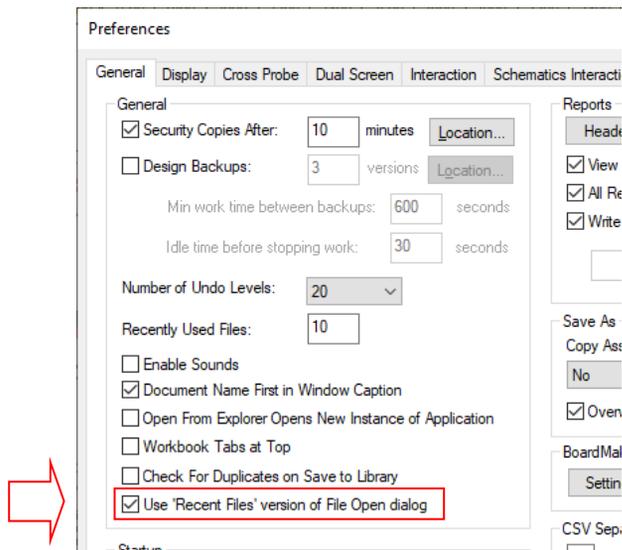


As well as **Unpinning this item** from the list, the context menu also enables you to **Remove** the item from the list (remember, the list is all recently used not a whole list of design items). There's no maximum number of entries for this feature, so it's up to you to remove any files that are no longer relevant. Use the **Remove from list** option on the context menu to do this.



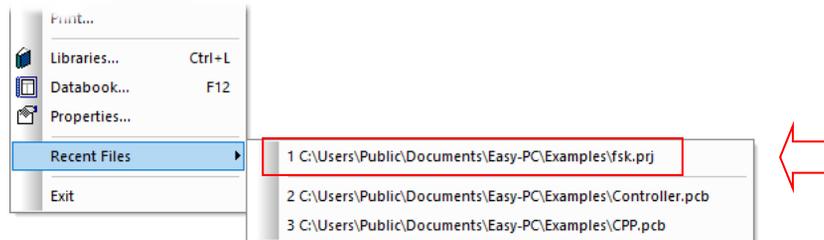
Disabling this option

If you wish to disable this option and just use the standard dialog, uncheck the option **Use Recent Files version of the Open dialog** check box in **Preferences** and **General**.



Recent Files Lists Projects

The **Recent Files** menu on the main **File** menu now lists **Projects** and **Design** files separately for ease of access.

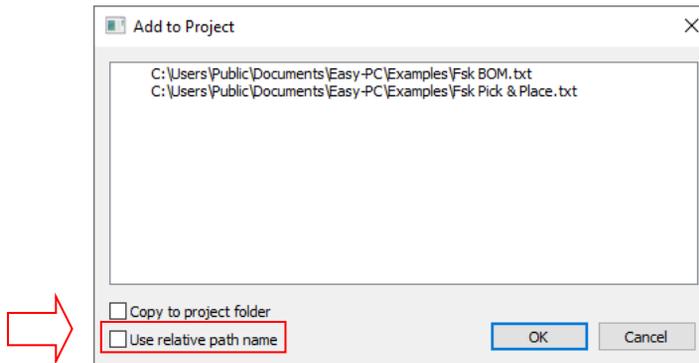


Add to Project – Other Files Relative Path Name

When using **Add to Project**, there is an option to reference ‘other’ files (not PCB or SCM) using their **relative path** if they are in the same folder as, or a folder below the Project.

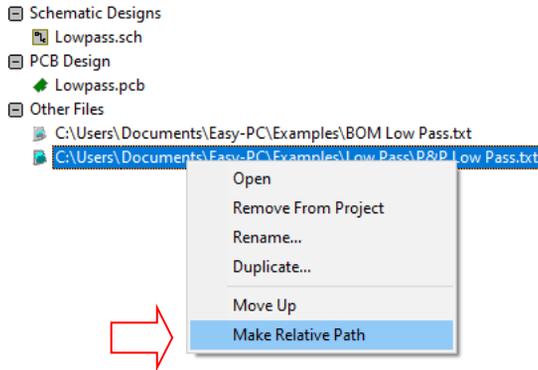
When adding other types of files to a Project, once a file is selected to add, a new dialog will show the list of files and allow you to add them in one go by selecting the ones you require using the **Ctrl** key and select.

You can also **Copy** the selected file(s) to the project folder or reference them using the **relative path** where relevant. This is instead of prompting you for each file as it is added to the project.



Context menu for Relative Path

Once in the Project file list, if the file lives somewhere relative to the Project file (in the same folder or below the Project), a new right-click context menu option will detect that the indicated file is below the Project and will allow you to convert their full absolute path into a **Relative Path**.

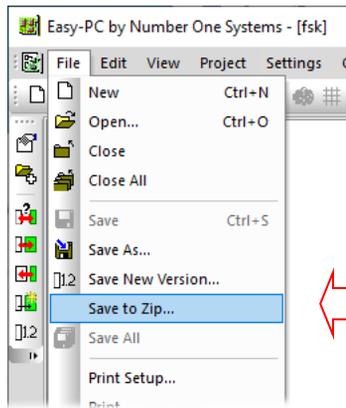


Once converted, if selected again using the context menu, you can return to the **Absolute Path** if required using the option that is now displayed.

Save Project to Zip File

There is a new command on the **File** menu of a **Project**, that saves all the files in a project into a single **Zip** file for ease of archiving or sending to colleagues, etc.

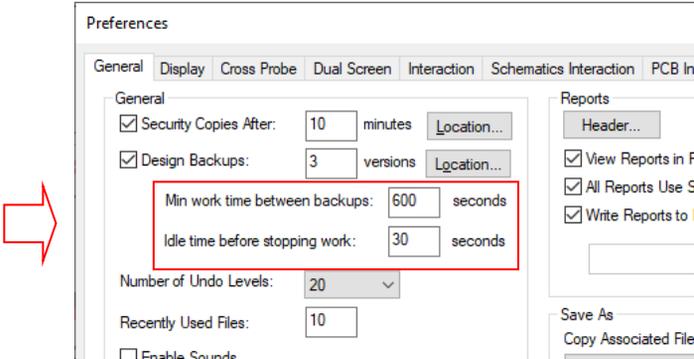
When selected, the **Save As** dialog is displayed where you can choose the folder to save the Zip file into. By default, it displays the current Project folder for saving.



Intelligent Design Backup Interval

Design Backups (if enabled) are written whenever you **save** a Schematic or PCB design. Using the new **Work Time** controls (enabled by default) the application will ensure that a meaningful amount of work is done between each backup regardless of how frequently you Save the design on which you are working. This means you can Save as often as you like without “filling up” your set of cyclic design backups with tiny incremental differences.

Two new settings on the **Preferences, General** dialog allow you to define the amount of work time has to elapse before making the next backup (**Min work time between backups**), and how long the design has to be unmodified before it is considered idle and thus the work timer stops ticking over (**Idle time before stopping work**). Work Time is triggered by any design editing that records or invokes an **Undo** action.

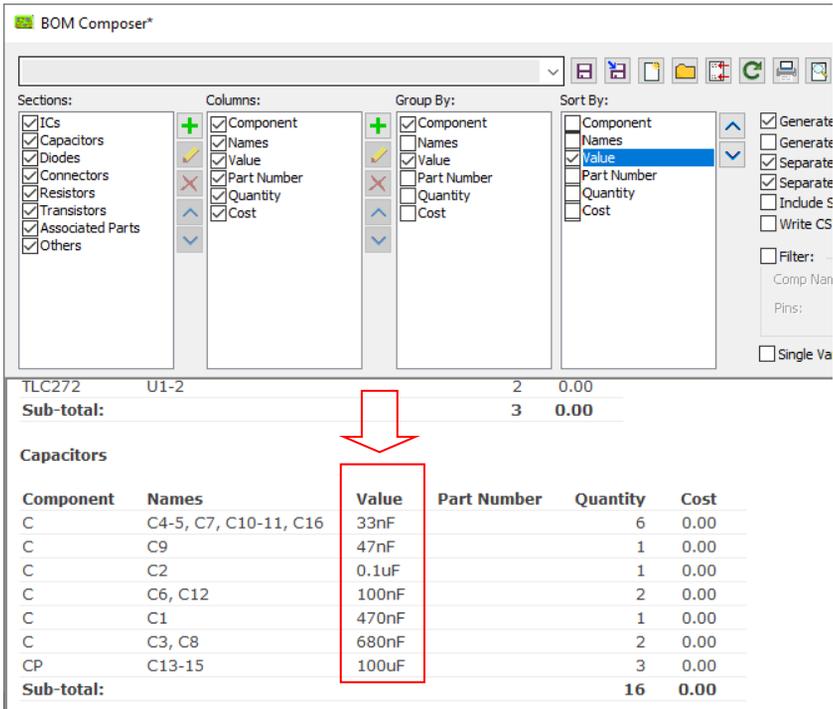


BOM Composer Update

Sort Values on columns

When sorting on the **Values** column it will now work out the actual electrical value and sort by that, smallest to largest. It does not understand exponents, so values such as 10e-9 or 2.2e3 will not be sorted.

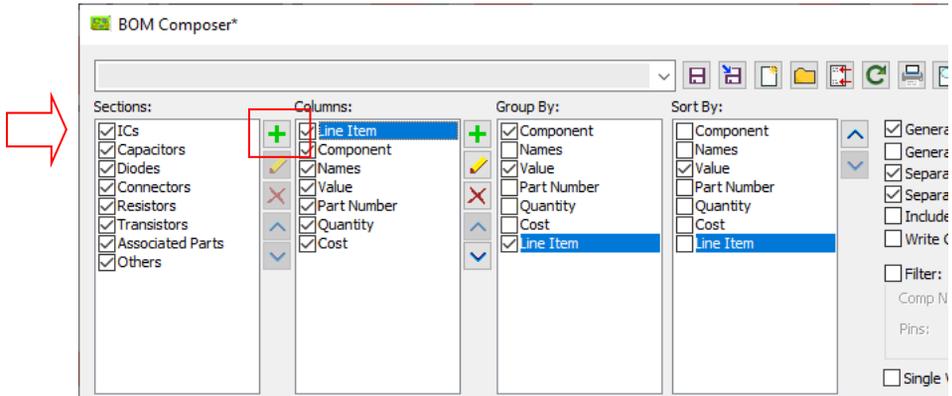
Thus 6.8pF will come before 4.7nF. It also understands absolute values too, for example 100pF = 0.1nF



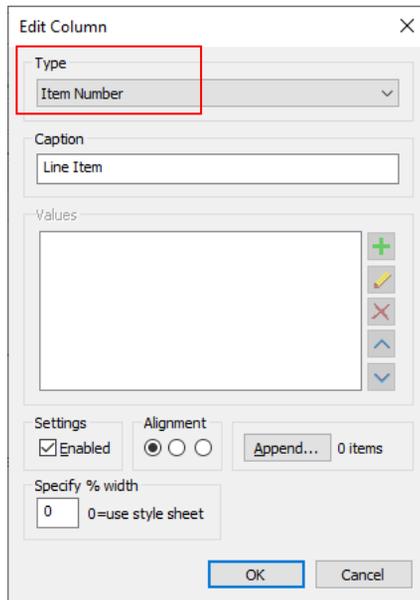
New Line Item Number Field

There is a new field available to add in BOM Composer that automatically writes a **line item number** for each row of data output to the report files.

To access the **Line Number** feature, click on the green + plus button on the central navigation bar.



From the **Edit Column** dialog displayed, select **Line Number** from the **Type** drop down list. You can also type your own **Caption** if you require one in the report.



The **Line Number** caption then appears in the **Columns**, **Group By** and **Sort By** columns, ready for positioning. As with all column items, move them in the report using the Up and Down arrows.

A sample report might look like this:

ICs

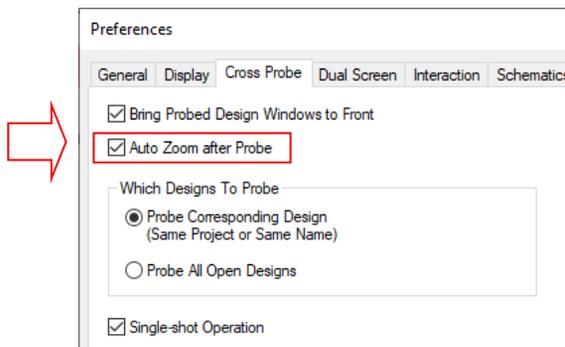
| Line Item | Component | Names | Value | Part Nu |
|-------------------|-----------|-------|-------|---------|
| 1 | 555 | U3 | | |
| 2 | TLC272 | U1-2 | | |
| Sub-total: | | | | |

Capacitors

| Line Item | Component | Names | Va |
|-----------|-----------|-----------------------|----|
| 3 | C | C4-5, C7, C10-11, C16 | 33 |
| 4 | C | C9 | 47 |
| 5 | C | C6, C12 | 10 |
| 6 | C | C1 | 47 |
| 7 | C | C8 | 68 |
| 8 | CP | C13-15 | 10 |
| 9 | C | C2 | 33 |

Cross-probe zoom After Probe

There is a new check box on **Cross Probe** page of **Preferences** dialog to **Auto Zoom** when probing.



This enables you to see what has actually been selected in the probe target. Previously, the default action was to pan the selected item into view if it is not already within the view. With this check box set (which is the default setting), the target design will zoom to show the selected item and a reasonable area around it.

New Nets Bar

A new dockable **Nets Bar** showing details of the nets in a **PCB design** has been added to Version 24, (note this is not available for Schematic Designs). The bar can display **Net Name**, **Net Class**, **Total Track Length**, **Min/Max Length Values** and **Rules**, Via Count and **Test Land Count**. These details are available already in the head-up display for a single track being edited, but the new bar allows you to view this information across multiple nets to assist with length matching etc.

Access the **Nets Bar** from the **View** menu and **Dockable Bars> Nets** menu option.

| Name | Class | Vias | Pads | Test Points | Min Length Rule | Max Length Rule | Net Length |
|------|-------|------|---------------------------|-------------|-----------------|-----------------|------------|
| A0 | Timed | 0 | U2.19 U3.10 U4.10 U31.10 | 0 | 2000.0000 | 6500.0000 | 2747.2795 |
| A1 | Timed | 0 | U2.18 U3.9 U4.9 U31.9 ... | 0 | 2000.0000 | 6500.0000 | 4282.8504 |
| A2 | Timed | 0 | U2.17 U3.8 U4.8 U31.8 ... | 0 | 2000.0000 | 6500.0000 | 3847.2795 |
| A3 | Timed | 0 | U2.16 U3.7 U4.7 U31.7 ... | 0 | 2000.0000 | 6500.0000 | 3375.9252 |
| A4 | Timed | 0 | U2.15 U3.6 U4.6 U31.6 | 0 | 2000.0000 | 6500.0000 | 1800.0000 |
| A5 | Timed | 0 | U3.5 U4.5 U31.5 | 0 | 2000.0000 | 6500.0000 | 1600.0000 |

There are various settings you can use to control what information is presented to you on the bar. The **Options** dialog is used to specify which nets to include in the table (for example by net class, net name or selection). The **Columns** dialog allows you to choose which of the available columns are displayed and in which order.

Once you have the bar set up to display the columns and nets that you are interested in monitoring, you should see the information updated automatically as you make changes in the design itself such as routing tracks, changing layers or adding connections.

For the **Net Properties** that are governed by rules on your **Net Classes**, the values in the bar may also be drawn with a coloured background to help you see which nets are within the rules (green) and which are outside (red). These colours can be changed using the **Options** button.

The **Options** button is used to control the Nets that are displayed on the Nets bar:

Choose from one of the available options to control which Nets are displayed. Empty (unused) nets are always excluded as they contain no useful information.

Include

- All Nets - every Net in the design.
- Selected Net - only the Net to which the currently selected item belongs.

- Selected Net Class - all Nets that belong to the same Net Class as the net of the currently selected item.
- Matched Nets - Nets whose name matches the filter or wildcard text given.
- This Net - one specific Net.
- This Net Class - Nets of one specific Net Class.

In addition to this you can opt to include or exclude Nets by other properties.

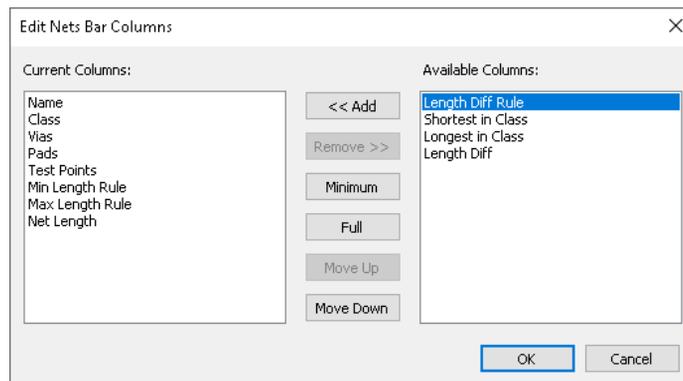
- Power/Ground Nets - those whose Net Class is of type Power or Ground.
- Nets without length rules - those whose Net Class does not specify any min/max or length difference rules.
- Default nets - those which have a default (not user-defined) name.

Note that if you have opted to include a specific named net then these additional check boxes are ignored.

Settings

- Auto Scroll - checking this option will ensure that the net on which you are working is always kept in view in the table.
- Valid Colour - this is the background colour for table cells that are within the specified highlight track lengths rules.
- Invalid Colour - this is the background colour for table cells that are breaking the specified highlight track lengths rules.

The **Columns** button enables the **Edit Nets Bar Columns** dialog. This is used to control which of the available Columns in the table you wish to see.



Choosing Columns

Use the buttons in the middle part of the dialog to manage the columns in your Nets bar table. Available Columns can be used by selecting them and pressing **<<Add** or simply double-clicking on the name. Likewise, to remove a column from the Current Columns list, double-click the name or use the **Remove>>** button.

The **Minimum** button will re-initialise the list to display a minimal set of information. Similarly, the **Full** button will re-initialise the list to display all available columns. Using the **Move Up** and **Move Down** arrows enables you to position the columns as required in the grid

Columns Available

These are the columns available for display:

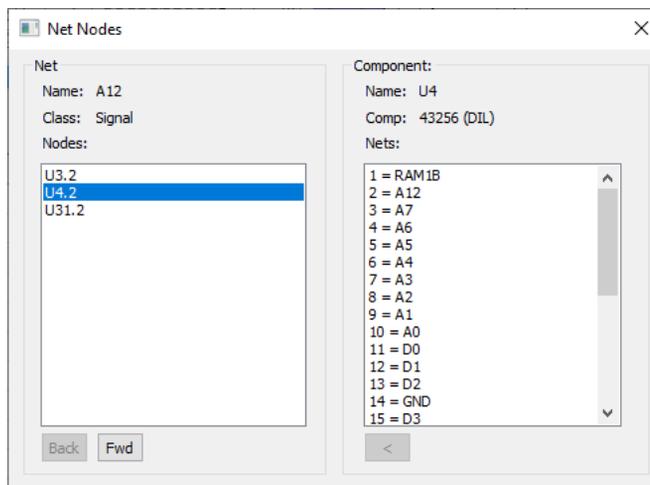
- Name - the name of the Net
- Class - the name of the Net Class
- Vias - the number of Vias in the Net
- Test Lands - the number of nodes (Pads or Vias) in the Net that are marked as Test Lands
- Pads - an abbreviated list of the component Pads
- Net Length - the total length of the Net (see the note below for Net Length calculation)
- Length Diff Rule - this is the Length Difference value if you have enabled this rule in Net Classes
- Shortest In Class - the shortest Net length across the whole net class.
- Longest In Class - the longest Net length across the whole net class.
- Length Diff - the difference between the shortest and longest Net length across the whole net class.
- Min Length Rule - the minimum length allowed for the Net as specified on the MinMax rule in Net Classes.
- Max Length Rule - the maximum length allowed for the Net as specified on the MinMax rule in Net Classes.

Net Length Calculation

The calculation of **Net Length** displayed in the Nets bar uses the same settings for unrouted connections as the **Headup Display**, which you can find on the **Display** tab of the **Preferences** dialog. This allows you to control how the remaining unrouted connections affect the overall net length value.

Net Nodes dialog

Double-clicking on a 'Pads' cell in the bar will open a new dialog allowing you to explore the pads on that net in more detail including being able to see the nets attached to all pads of a given component.



Information about the net is shown in the left-hand part of this dialog. This includes the name and class of the net, and a full list of all the 'named' nodes in that net (i.e. leaving out unnamed items such as junctions or vias).

Click on any node in the net to display information about the whole component to which that node belongs. This includes the component name and reference name, as well as a complete list of the nets connected to all the pads of that component.

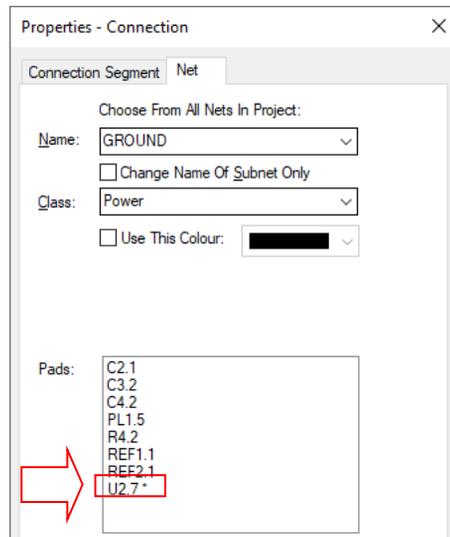
Double-click on any of the pads shown for the component (or select one and click the < button) and the attached net will become the 'current' net and the dialog will update to show information about that net.

You can use the **Back** and **Forward** buttons to move through the various nets you have inspected.

The dialog can be resized to allow you to view longer names.

Net Properties – Nets shown on other sheets

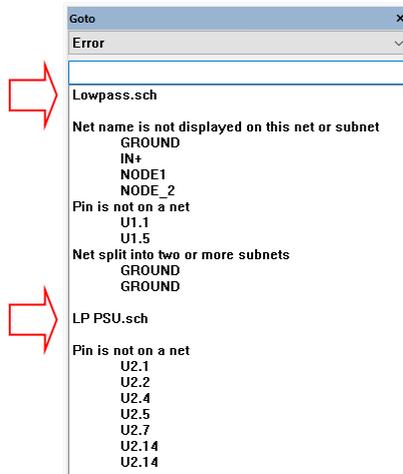
With a selected net, **Net Properties** has been extended in a multi-sheet project to include **Pads** on other sheets, these are shown with an asterisk suffix to differentiate them.



Goto Bar shows DRC Errors for all Open SCM Sheets

In a multi-sheet **Project**, the **Goto Bar** now shows **DRC** errors for **all open SCM sheets**.

Different sections for each page are shown:



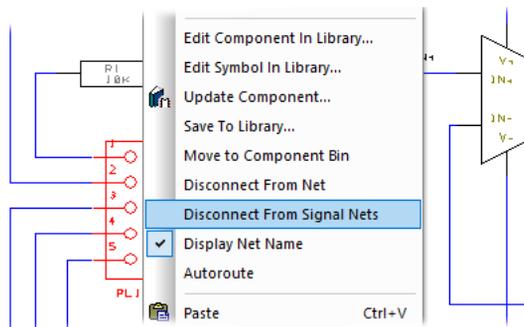
New Disconnect From Signal Nets option

Available in both Schematic and PCB designs, a new context menu option **Disconnect from Signal Nets** can be used in select mode. As the name suggests, this will do the same as Disconnect From Net but it will only do it to nets whose net class is of type Signal, thus leaving any power/ground nets still connected.

This option is useful if you need to swap gates for instance in a Schematic, it means that the power and ground nets are still connected without having to remember to connect them after the gate changes have been made.

Accessing it in Schematics

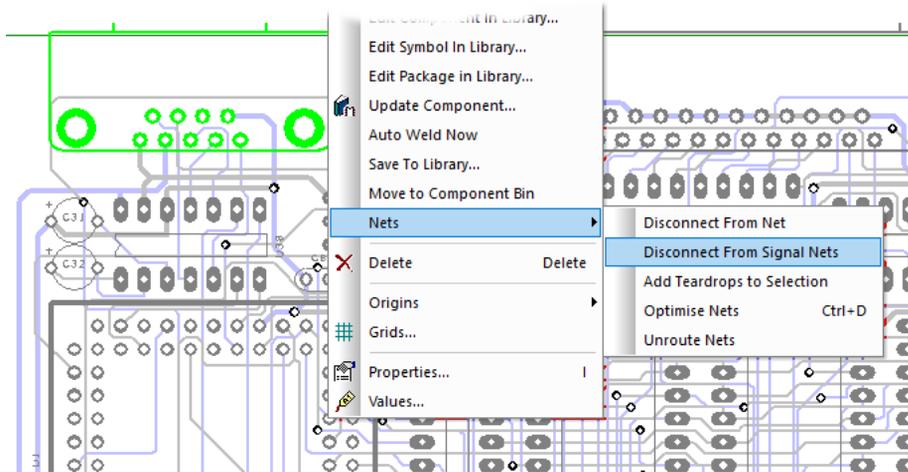
To access this new mode in the Schematic, select the Component and right click. From the context menu select **Disconnect from Signal Nets**.



Once the option is selected, the Component is disconnected and you can move or remove with the connections remaining in place ready for welding back in.

Accessing it in PCB

In PCB, this option is accessed through the **Nets** sub-menu for a selected Component.

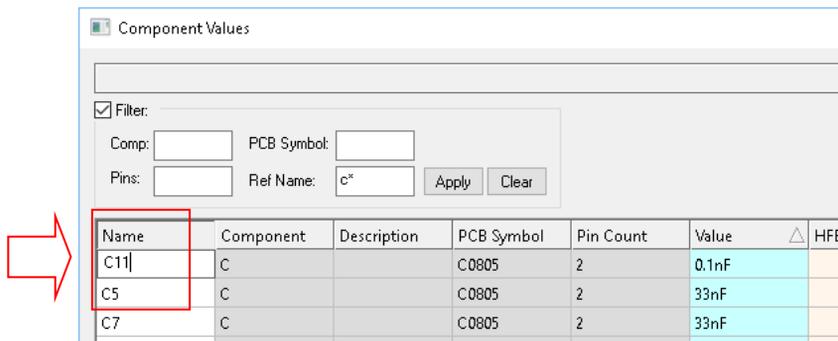


The option works for both routed and unrouted Components in PCB.

Changes to Component Values dialog

Editable Ref Name Column

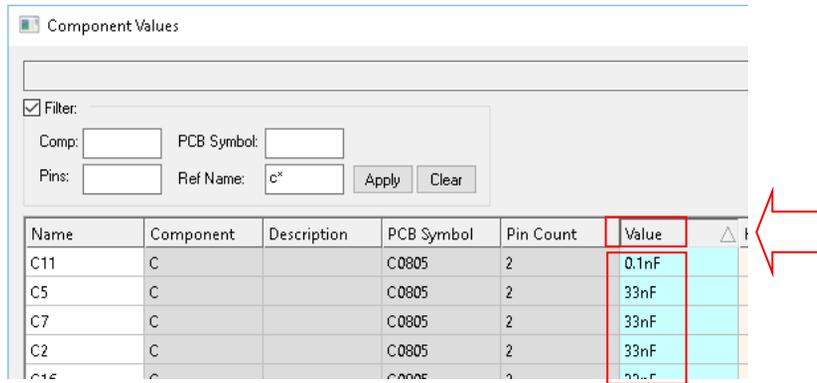
On the **Component Values** dialog for a design or Project, the **Ref Name** column is now editable. This allows you to alter the names of multiple components in one place. To edit a name, double click on the Ref Name cell and type the name required. On exit of this dialog, the name will be checked for validity.



Note, if you make changes in this dialog in a PCB design, these will also be recorded for back annotation to the schematic

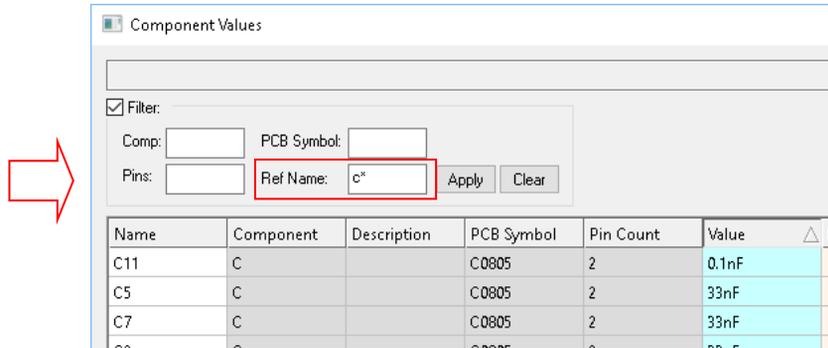
Component Values dialog sorting

Clicking on the column header for any column that is one of the recognised “value” names (Value, R, C, L etc.) will now sort on the ‘actual’ value taking into account series/numeric prefixes, thus putting 100pF before 100nF. It does not understand exponents, so values such as 10e-9 or 2.2e3 will not be sorted.



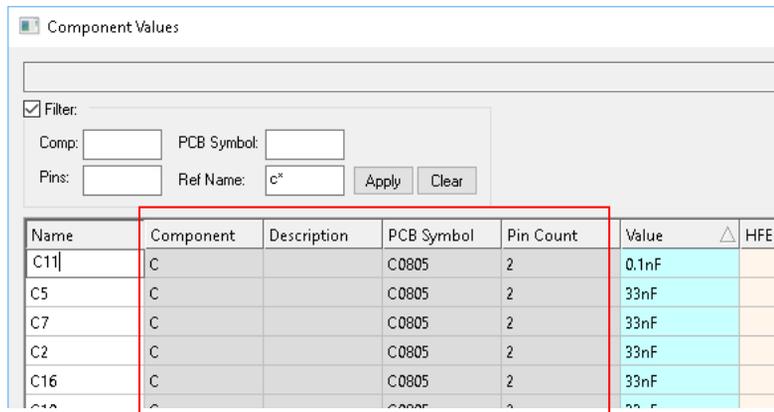
Ref Name Filter

A field for **Ref Name** has also been added to the **Filter** controls, allowing you to filter the component table based on a wildcard of component ref names.



Colouring for Read-only Controls

Read-only edit boxes now have greyed background to make it clear that they are not currently editable.



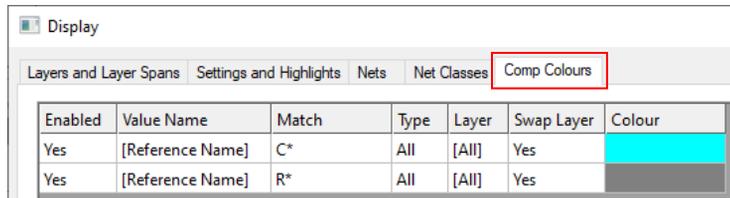
Drag Connected Items to Component Bin

Dragging items to the **Component Bin** has now been extended to allow you to drag components to the bin even if other design items, like Connections or Tracks, are also in the selection. Previously, moving connected items to the bin was only possible using Move To Bin. The bin must be pinned for this option to work.

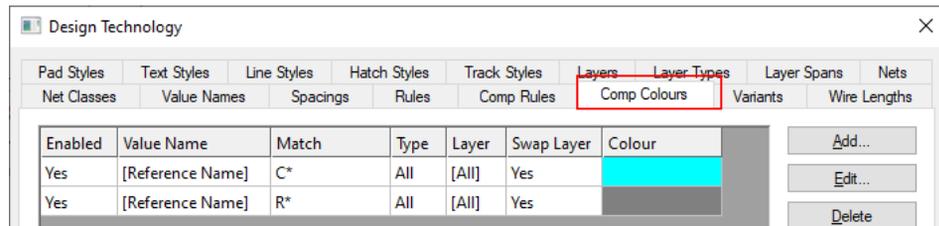
Rules for Colouring Components

A new **Component Colours** tab is available in both the **Design Technology** dialog and **Display** (Colours) dialog to allow settings for over-riding the colours used to draw Component shapes and/or text.

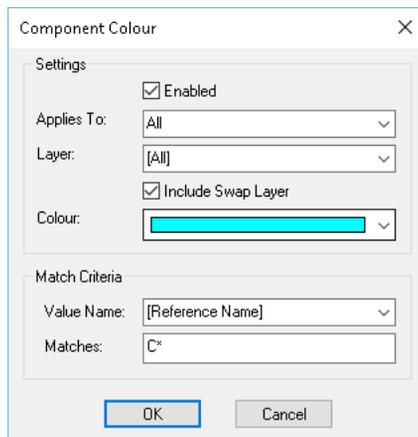
Match components by wildcard on chosen property/attribute and specify colours to be used for matching components by side/layer. This can be used for things like producing a colour-coded assembly drawing, highlighting components with an 'obsolete' value, etc.



The **Design Technology** dialog reflects the same colours as **Display**:



Using the **Add** button displays the **Component Colour** dialog:



Applies To: is used to select the item the colour applies to, choose **All**, **Shape** or **Text**.

Layer: choose the layer that the colour is applied to. This can be any layer in the design.

Include Swap Layer will automatically do the same colouring on the equivalent layer on the bottom side as you've set on the top. When set to No, the colour you've defined will only apply to the layer you have chosen

Colour: choose the alternative colour required for the item.

Match Criteria: match based on a Value Name, for example [Reference Name]. The **Matches:** can be specific names or wildcard names.

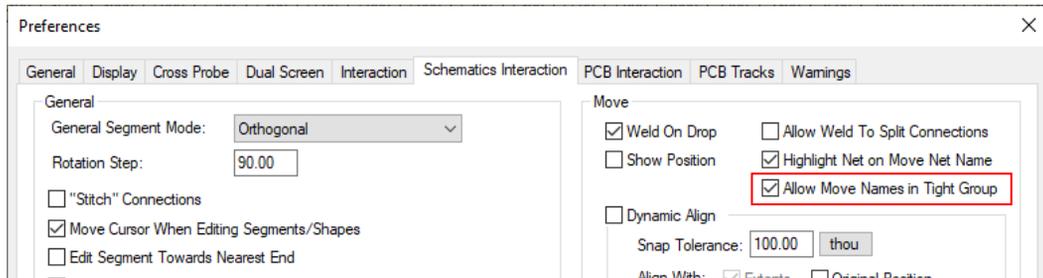
Move Components Names within a Tight Group

If you select one or more **Component Names** (and nothing else) where those Components are part of a **tight group**, this allows you to move those names without disturbing the position of the Components.

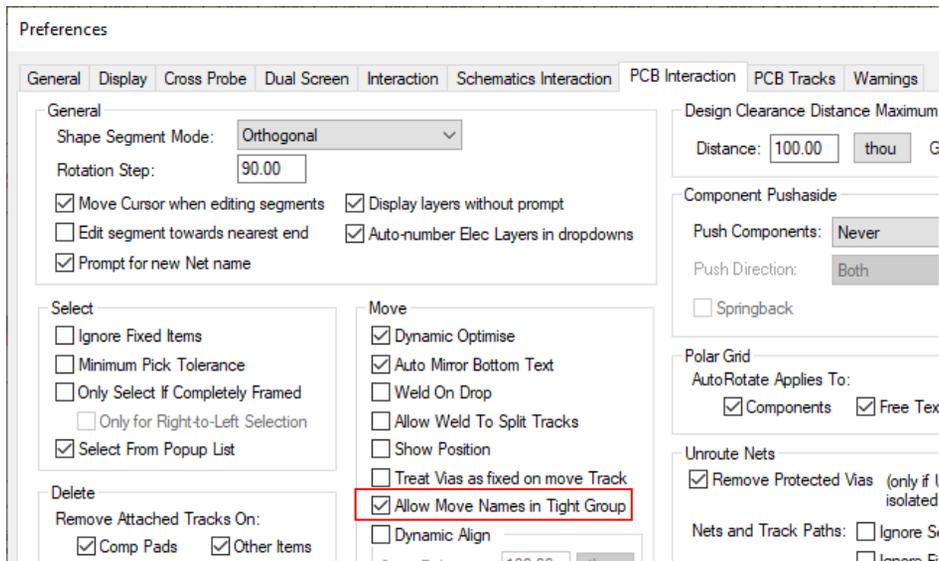
New Preference – ‘Allow Move Names in Tight Group’

Under **Preferences** and **Move** in both **SCM Interaction** and **PCB Interaction**, there a new check box, **Allow Move Names in Tight Group** to use when dragging Component Names (as described above).

SCM Interaction dialog



PCB Interaction dialog



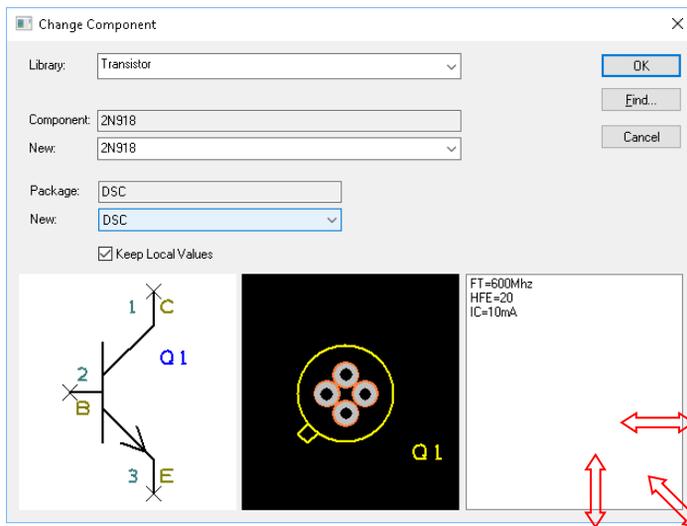
Changes to Change Component & Replace Component

Changes have been made to the **Change Component** dialog (from **Component Properties**) and **Replace Component** (from **Tools** menu):

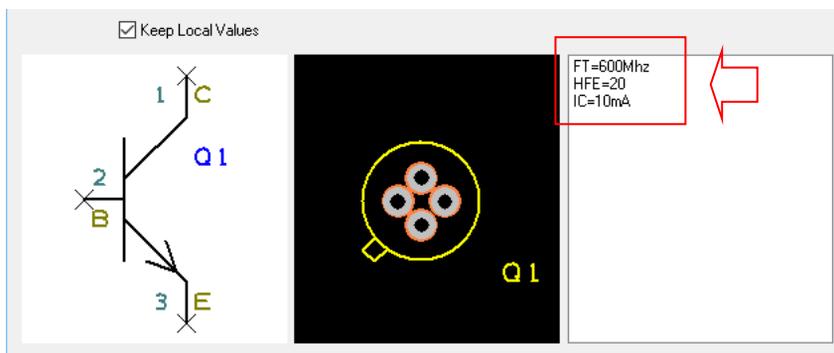
Change & Replace Component dialogs are now resizable

You can resize the dialog by grabbing the edges or corner of the dialog and dragging it. This is useful if you have large components that cannot be viewed using the default size.

Values are shown in the Preview list

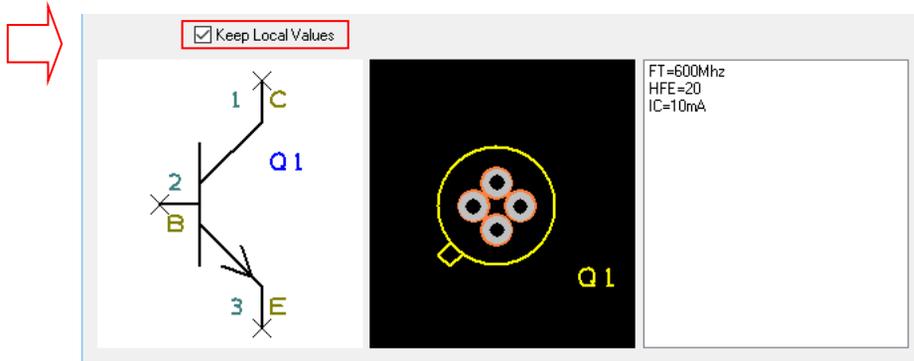
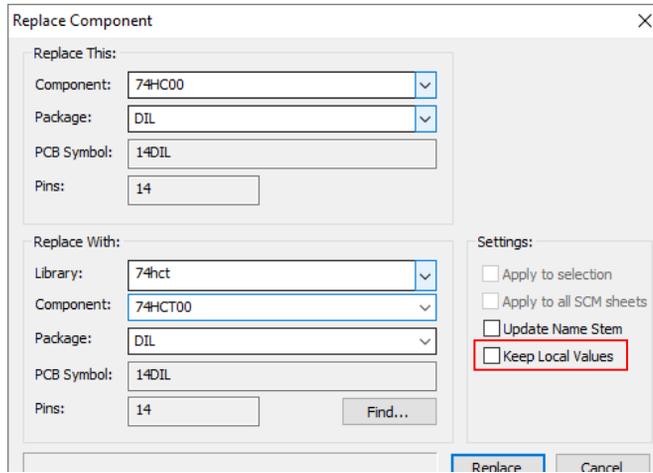


The preview for the selected Component shows you **Values**. This is the same as it does already in the **Add Component** dialog.



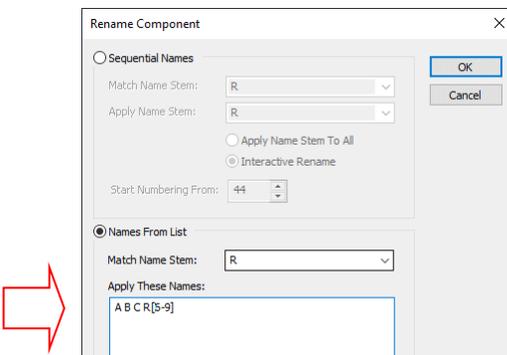
Check box to “Keep Local Values”

Once run with the chosen new Component, the choices dialog now enables you to **Keep Local Values**. This will ensure that any values you have defined or altered locally on the Component instance in the design will be retained when the new Component definition is applied.



Rename Components by List

The **Rename Components** option (on the **Tools** menu) has been extended to allow you to provide a list of new Component Names (including name ranges) that are used in the point-and-click stage instead of simply renaming sequentially. As well as typing the list, you can also import a list using Copy/Paste.



When all the names have been “used up”, you will be asked if you want to name some more, pressing Yes will return to the dialog for you to enter more names.

Changes to the New Component Wizard

Description box

The **Details** tab of the **New Component Wizard** now has **Description** box from which to enter the Component description.

Component Wizard - Details

Start
Type
Details
Schematic Symbol
PCB Symbol
Assign Pins
Values
Finish

Enter the details of your new component

Component Name: PIC32MX795F512L-80I/PT

Description: PIC32 Series

Package: QFP

Default Reference: U

Values Tab

The **New Component Wizard** now has a new tab for **Values** to allow you to add values to the Component you are building.

Component Wizard - Values

Start
Type
Details
Schematic Symbol
PCB Symbol
Assign Pins
Values
Finish

Create Values to add properties to your Component

MFR=Mttrata

Supplier PartNo=AS-334-87251

Supplier=RS

Tolerance=1%

Value=10K

Wattage=0.125W

Library Item time stamps

When copying items between libraries, or from designs to libraries, the existing time stamp ('version') on those library items will now be preserved rather than set to the current time. This helps with getting the correct comparison of library items when running options such as **Update Component** regardless of any library item re-arrangement that has been done.

Prompt for Net Name on Add Connection

When starting a new **Connection**, the new **Add Net** option will display a net name dialog allowing you to supply a name straight away, instead of having to finish the connection and/or do **Change Net**. This is a switchable option and is on by default. It can be switched off in **Preferences**, see below.

Add Net

Enter name for new net, or Cancel to leave unnamed

ADD2

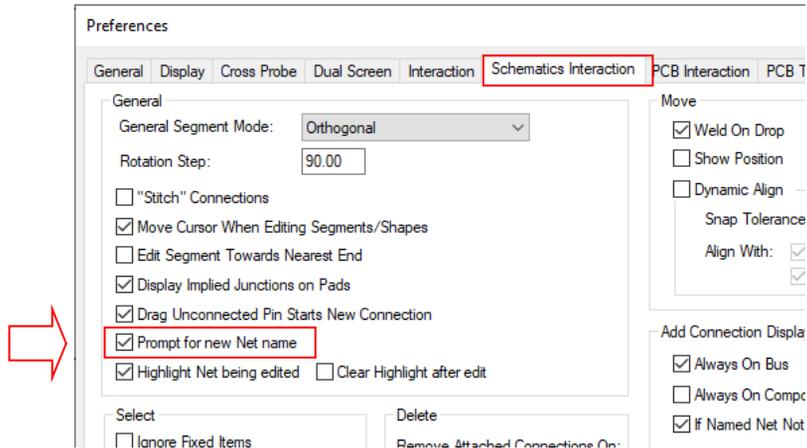
OK Cancel

The dialog will auto-increment if possible, to assist with adding a series of net with sequential names. If you used ADD1 as your net name for example, using Add Net a second time will increment the net name to ADD2.

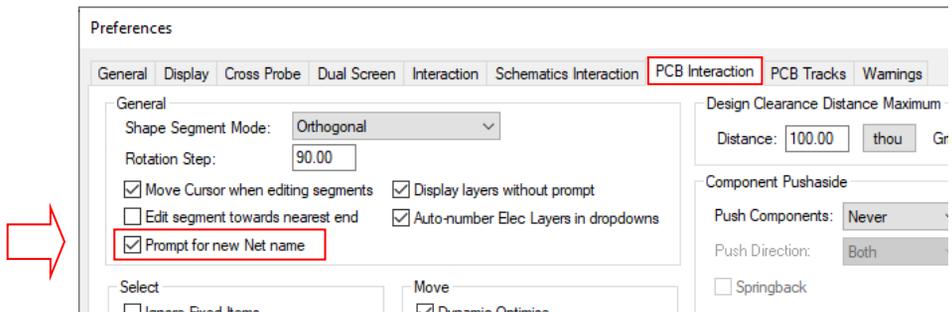
Preference to switch off prompt

If you do not want to be prompted for a Net Name, you can switch it off using **Preferences** and **Schematics Interaction** and **PCB Interaction** and unchecking the **Prompt for new Net Name** box.

Schematic Interaction tab



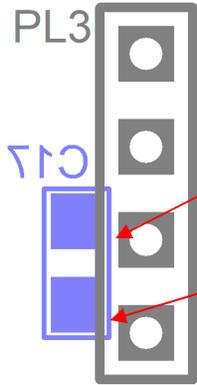
PCB Interaction tab



Design Rules Check Changes

Component-to-Component SMT Check

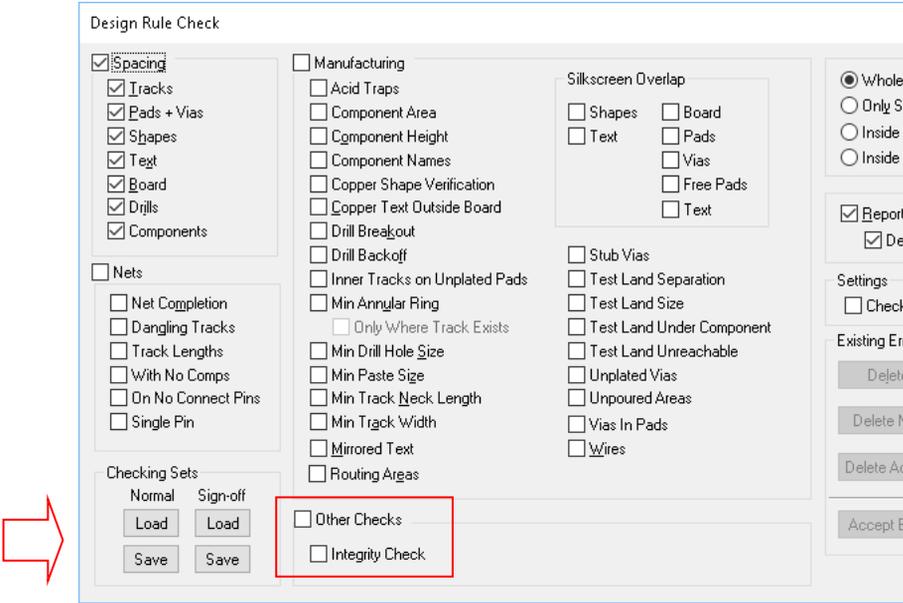
When doing a **Component-to-Component** check for SMT components on the opposite side of the board against a through-hole component, if **no placement shapes** are defined for the **through-hole component's bottom side** then **Spacing** checks are now done only against the space occupied by the **pins** protruding through that side of the board, rather than using the bounds of the entire component on that side.



In the diagram left, the C17 SM component is on the bottom side, the connector PL3 is through-hole. C17 has no placement area and as such can be placed closer to the pins of PL3 on the bottom side. If PL3 has a bottom side placement area, then this would be used for the spacing rule.

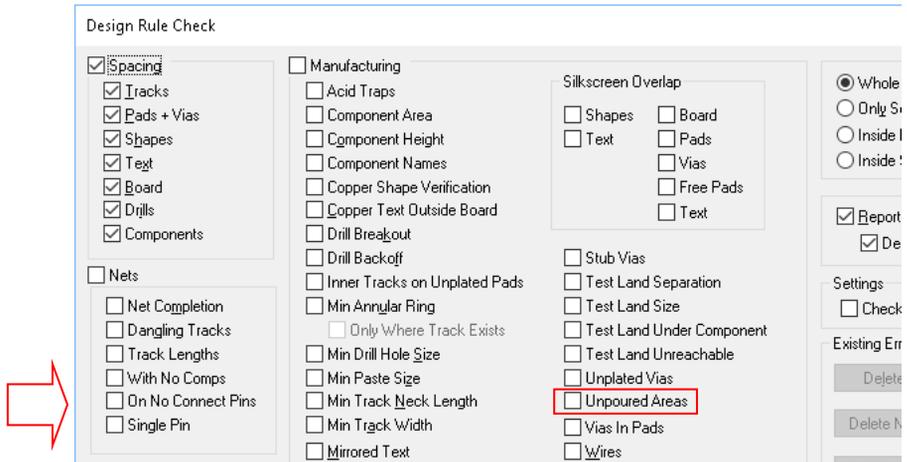
Include Integrity Check

Integrity Check is now available within the **Design Rules Check** dialog. Its presence here allows it to be included in your saved **Sign-off** checks.



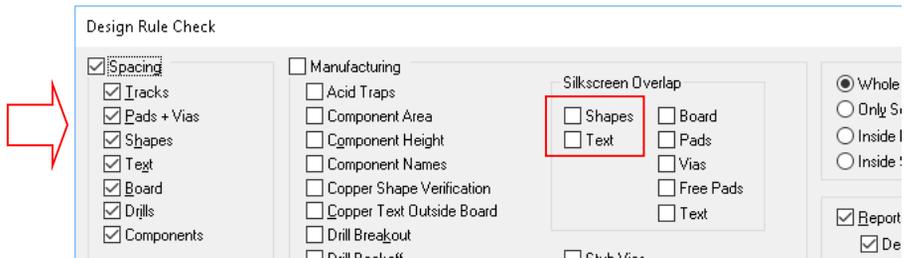
Check for Unpoured Areas

There is a new check under the **Manufacturing** header that checks for **Copper Pour Areas** that have no copper attached to them (**Unpoured Areas**). This is an important check to ensure full design and electrical integrity within your design before sending away for manufacture.



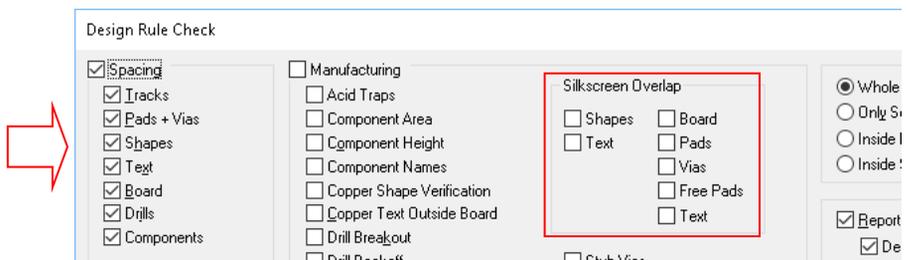
Silkscreen Overlap Check

The existing **Silkscreen Overlap** check has been split into separate checks for **Shapes** and **Text**.



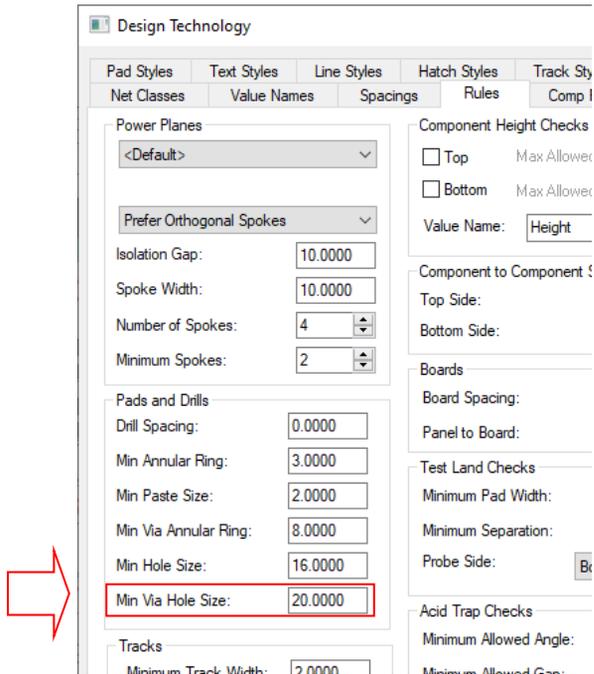
Individual Item Selection in Silkscreen Checks

Further extensions to **DRC Silkscreen Overlap** checks means you can now split them by checking **Shapes** and/or text against any of **Board, Pads, Free Pads, Vias** and **Text**.



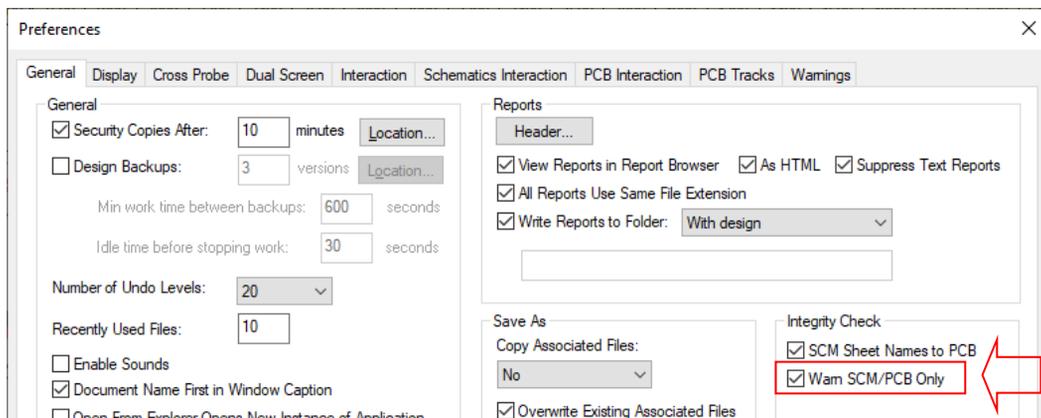
Min Via Hole Size

There is now a separate value on the **Design Technology, Rules** for **Min Via Hole Size** so you can have two different values for the minimum hole size used for **Vias** as well as Pads (using Min Hole Size option).



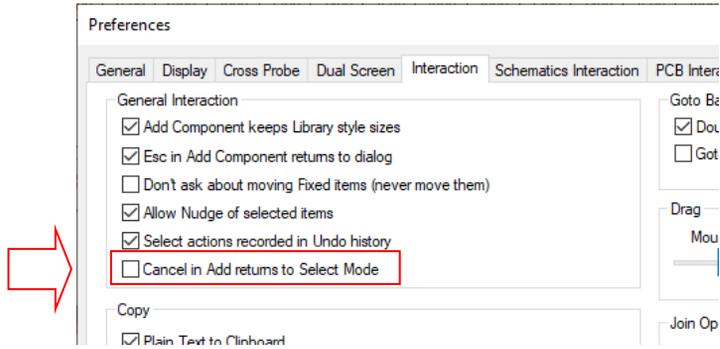
Integrity Check suppress PCB-only

There is a new setting in the **Integrity Check** section of **General Preferences**. With the **Warn SCM/PCB only** check box unchecked, any warning messages about ignoring **SCM-only** or **PCB-only Components** during Integrity Check will be suppressed. This means where there are many warnings that you know to be correct and irrelevant to you will not be written to the report, thus decluttering it.



Change to Preferences for ‘Cancel in Add Track’

The existing preference for **Cancel in Add Track returns to Select Mode** has now moved to the **Interaction** tab of **Preferences** dialog (it was previously on the PCB Tracks tab). This makes it more meaningful as it applies to other options (see below).



Applies to other items too

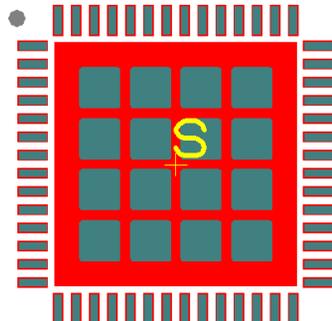
The **Cancel in Add Track returns to Select Mode** option has also been generalised to apply to Add Track, Add Connection, Add Shape, Add Line and Add Text.

Paste Shapes in Footprints

Within the **PCB Symbol (Footprint)** editor, in the **Tools** menu there is a new command, **Apply Paste**. This is used to assist in creating a rectangular pattern of paste shapes for items such as thermal pads. This is an automated way to create something you can already do by hand but this tool makes it much easier.

For example, it can be used to add a 4x4 Paste layer cutout matrix to the central thermal pad. A practical example Component using thermal cutouts shapes is the NXP device MC34GD3000EP using the QFN50P800X800X100-57N-D footprint. It helps you create the shape sizes, position them, add quantities, create the gaps etc. The dialog lets you create the shapes based on manufacturer's datasheets and also lets you do this iteratively by trial and error. Each time a value is altered, it can be applied to the shapes to enable you to see what has been created.

Paste defined on the paste layer will reflow during manufacture to cover the whole thermal pad. You must also have a paste layer in your design for this to become useful.



Select a single pad in your PCB symbol to enable this command on the **Tools** menu. The pad selected will be the one that the shapes are applied to. A dialog will be displayed allowing you to control the

pattern of shapes to be created. Existing shapes can also be removed, allowing you to easily try several patterns until you are happy with the result.

The top portion of the dialog shows you the **Pad Style** you have selected; it shows you the Pad Style Name and the actual style used.

Shape Details

Choose the **Layer** on which the shapes are to be created, and the **Line Style** to be used for those shapes. A narrower line style will produce smaller rounded corners on each shape. You'll probably find that you'll need a very fine line style because of the size of the shapes. You'll need to add one if it doesn't already exist in your **Design Technology Line Styles**.

Enter the **Width** and **Height** for an individual paste shape. You can also choose to define the **Width** and **Height** based on the **Centre Line** of the shape, or by **Outside Edges** when the application will work out the position of the shape segments for you. The options you choose may depend on what information is provided on the datasheet for the component you are building.

Arrangement

Use the controls in this portion of the dialog to define how the paste shapes are going to be arranged over your pad.

Gap allows you to define the gap between adjacent edges of shapes, whereas **Step** allows you to specify the distance between the same point (e.g. lower left corner) of each shape.

Fill Pad will allow the application to work out how many shapes are required to match the Width, Height and Gap/Step you have defined, whereas **Count** allows you to specify an actual number of shapes in each direction.

The tool will discard any shapes that are not entirely within the confines of the pad, even if you specify too many shapes or too large a size.

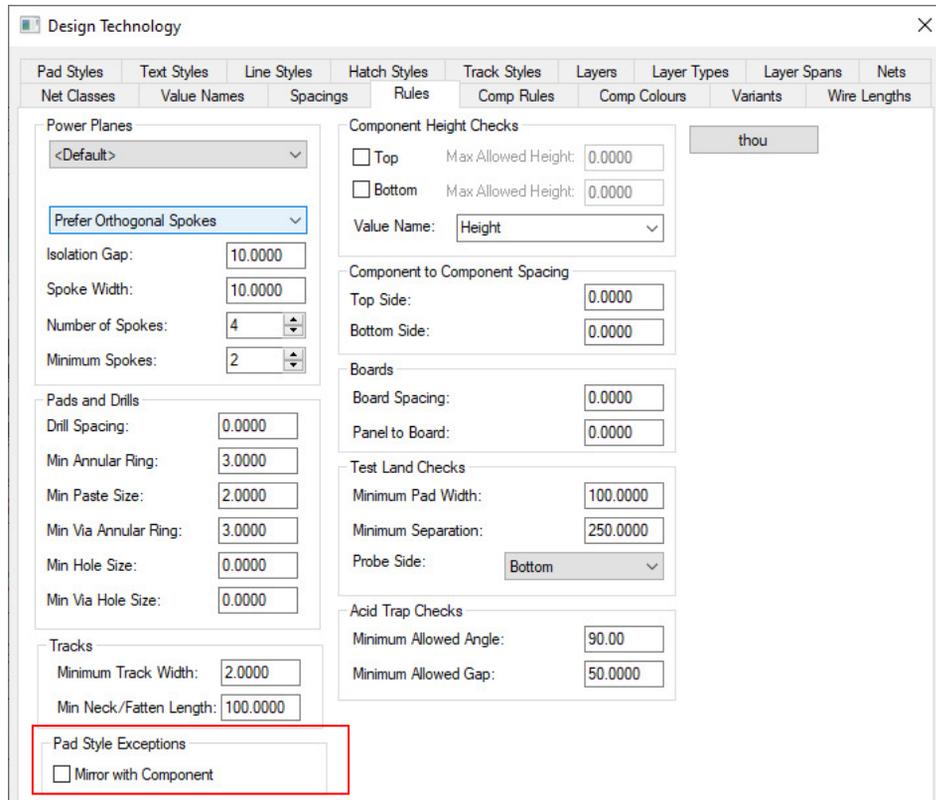
Settings

Remove Existing Shapes will delete any shapes that already exist within the pad on the chosen layer. This is useful when trying out various patterns, as it will discard the previous pattern before creating the new shapes.

Apply Pad Style Exception will create a "by layer" pad style exception for the pad over which you are placing paste shapes, to suppress the application-generated solder paste that would normally happen during plotting and printing. You should not enable this setting if there are other pads in your PCB Symbol that use the same pad style, otherwise no paste may be generated for those pads. This is not generally an issue, as you would almost certainly be generating paste shapes for a large central pad that is the only one using that pad style.

Mirror Pad Style Exceptions

A new check box **Mirror with Component** on the **Design Technology Rules** page has been added. With the **Mirror with Component** check box selected, when a Component is flipped (mirrored), it flips over the pad style stack so that any pad style exceptions are applied to the swap layer automatically instead of having to define two sets of exceptions for the Top and Bottom layers.



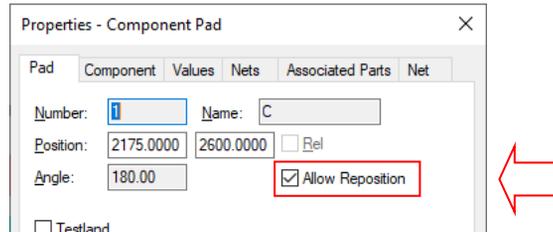
The image shows a screenshot of the 'Design Technology' dialog box. The 'Pad Style Exceptions' section is highlighted with a red box, and a red arrow points to the 'Mirror with Component' checkbox. The dialog box contains various settings for design technology rules, including Power Planes, Component Height Checks, Component to Component Spacing, Boards, Test Land Checks, Acid Trap Checks, Pads and Drills, Tracks, and Net Classes.

| Section | Property | Value |
|--------------------------------|----------------------------|--------------------------|
| Power Planes | Power Planes | <Default> |
| | Isolation Gap: | 10.0000 |
| | Spoke Width: | 10.0000 |
| | Number of Spokes: | 4 |
| Pads and Drills | Drill Spacing: | 0.0000 |
| | Min Annular Ring: | 3.0000 |
| | Min Paste Size: | 2.0000 |
| | Min Via Annular Ring: | 3.0000 |
| | Min Hole Size: | 0.0000 |
| | Min Via Hole Size: | 0.0000 |
| Tracks | Minimum Track Width: | 2.0000 |
| | Min Neck/Fatten Length: | 100.0000 |
| Component Height Checks | Top Max Allowed Height: | 0.0000 |
| | Bottom Max Allowed Height: | 0.0000 |
| Component to Component Spacing | Top Side: | 0.0000 |
| | Bottom Side: | 0.0000 |
| Boards | Board Spacing: | 0.0000 |
| | Panel to Board: | 0.0000 |
| Test Land Checks | Minimum Pad Width: | 100.0000 |
| | Minimum Separation: | 250.0000 |
| | Probe Side: | Bottom |
| Acid Trap Checks | Minimum Allowed Angle: | 90.00 |
| | Minimum Allowed Gap: | 50.0000 |
| Pad Style Exceptions | Mirror with Component | <input type="checkbox"/> |

This setting applies to the whole design, so care should be taken when checking the box in an existing design to make sure that any existing pad style exceptions are still working as expected.

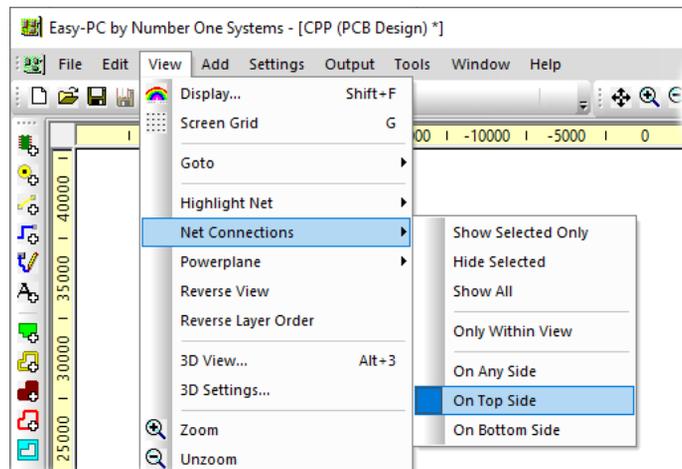
Allow Reposition of Component Pads

An additional check box on **Pad Properties** called **Allow Reposition** has been added. When this is checked, the pad on this instance of the Component can be independently moved or rotated, either on the **Properties** dialog or interactively.



Options for Drawing Connections by Side in PCB

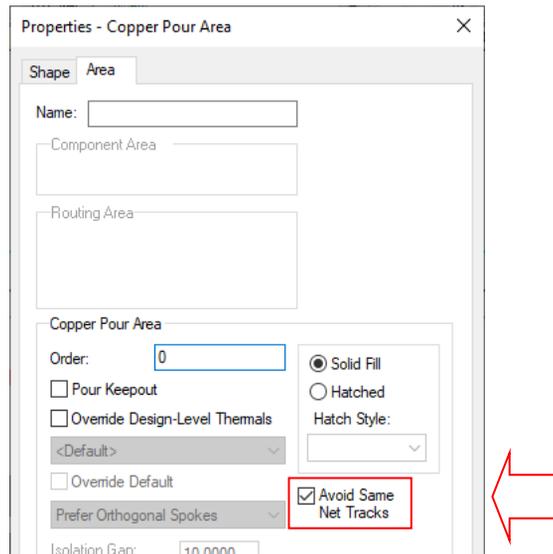
In a PCB design on the **View** menu and **Net Connections**, new commands are available to set connections to be drawn only on one side (e.g. only those between Components on the Top Side) and only when both ends are within the current view.



This only applies to the pads to which the connection is applied. The connection itself remains independent of any specific layer as normal.

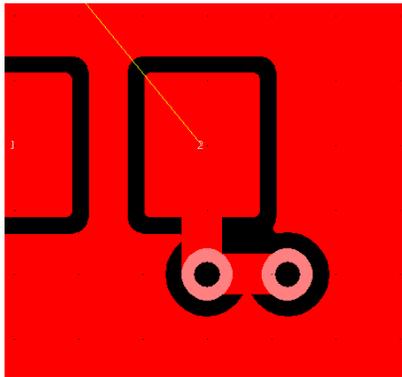
Pour Copper Avoid Same Net Tracks Setting

A new check box has been added to **Copper Pour Area Properties** to **Avoid Same Net Tracks**. When checked, this will instruct Pour to treat all tracks, even those on the same net as the Area, as obstacles to avoid.

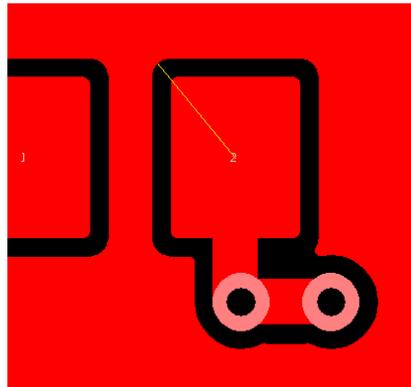


Examples of pouring below:

Without this setting



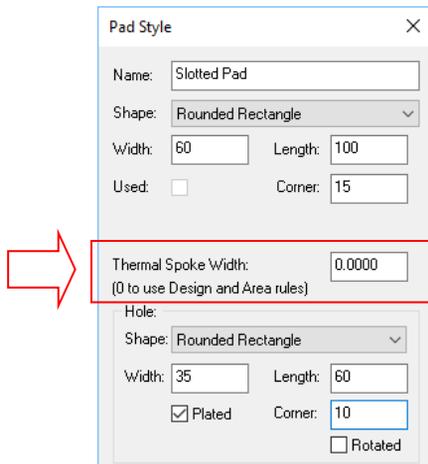
With this setting



Thermal Spoke values by Pad Style

In the PCB **Design Technology**, **Pad Styles** now have an extra field for **Thermal Spoke Width**. If this value is set to something greater than zero (the default) then this size will be used in preference to any design, or area-level rules defining thermal spoke width (in **Design Settings, Rules**).

This allows you to override the normal size of thermals for larger pads to ensure a better dispersal of heat from the pad during soldering.



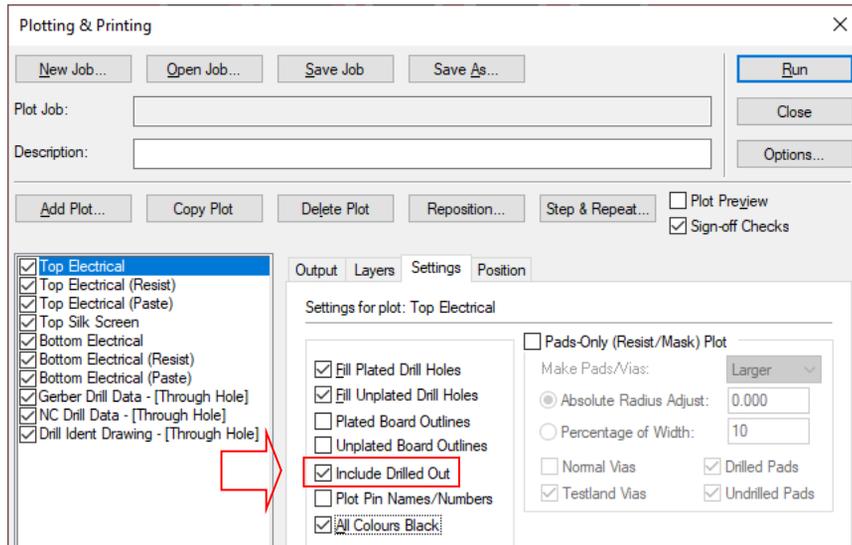
The **Thermal Spoke** value is now also displayed in the **Design Technology Pad Styles** dialog for ease of checking.

| Design Technology | | | | | | | | | | | | | |
|-------------------|-----------------|-------------|--------|-------------|----------|--------------|--------|--------------|--------|---------------|--|-------------|---|
| Net Classes | | Value Names | | Spacings | | Rules | | Comp Rules | | Comp Colours | | Varian | |
| Pad Styles | | Text Styles | | Line Styles | | Hatch Styles | | Track Styles | | Layers | | Layer Types | L |
| | Name | Layer | Shape | Width | Length | Corner | Width2 | Hole Size | Plated | Thermal Spoke | | | |
| X | Oval (60 x 110) | [All] | Oval | 60.0000 | 110.0000 | 0.0000 | 0.0000 | 32.0000 | Yes | 0.0000 | | | |
| X | Round 60 | [All] | Round | 60.0000 | 0.0000 | 0.0000 | 0.0000 | 32.0000 | Yes | 0.0000 | | | |
| X | Round 60 32 | [All] | Round | 60.0000 | 0.0000 | 0.0000 | 0.0000 | 32.0000 | Yes | 0.0000 | | | |
| X | Round 60 35 | [All] | Round | 60.0000 | 0.0000 | 0.0000 | 0.0000 | 35.0000 | Yes | 0.0000 | | | |
| X | Round 65 | [All] | Round | 65.0000 | 0.0000 | 0.0000 | 0.0000 | 40.0000 | Yes | 0.0000 | | | |
| X | Round 70 | [All] | Round | 70.0000 | 0.0000 | 0.0000 | 0.0000 | 37.0000 | Yes | 0.0000 | | | |
| X | Round 175 | [All] | Round | 175.0000 | 0.0000 | 0.0000 | 0.0000 | 100.0000 | Yes | 0.0000 | | | |
| X | Square 65 | [All] | Square | 65.0000 | 0.0000 | 0.0000 | 0.0000 | 40.0000 | Yes | 0.0000 | | | |
| X | Square 125 | [All] | Square | 125.0000 | 0.0000 | 0.0000 | 0.0000 | 45.0000 | Yes | 0.0000 | | | |
| X | Via 40 | [All] | Round | 40.0000 | 0.0000 | 0.0000 | 0.0000 | 25.0000 | Yes | 0.0000 | | | |

Plot Include/Omit Drilled-out Pads

An additional check box on the **Settings** tab of the **Plotting & Printing** dialog has been added.

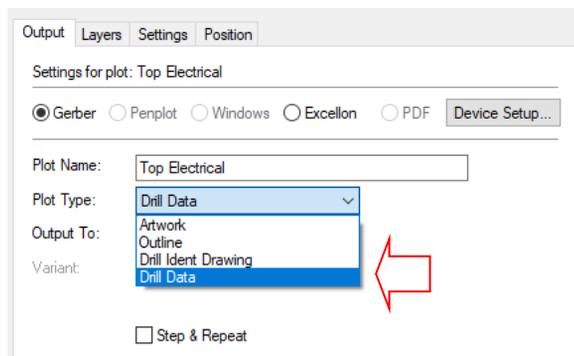
If you have pad styles where the drill size is larger than the pad, this produces an effect called 'drilled out pads' where any copper land for the pad would be completely removed by the drilling process. If you want to suppress the pad land from these drilled out pads, then uncheck this check box.



Output Drill Data to Gerber

Drill data plots, previously only written to Excellon format, can now write in **Gerber** format as well. This means drill data can be exported using Gerber X2 format as required by some manufacturers.

From the **Output** tab of the **Plotting & Printing** dialog, with Gerber selected using the **Plot Type** drop down list, select the new option, **Drill Data**. This applies to new plots as well as existing plots.



Extra header info in Excellon

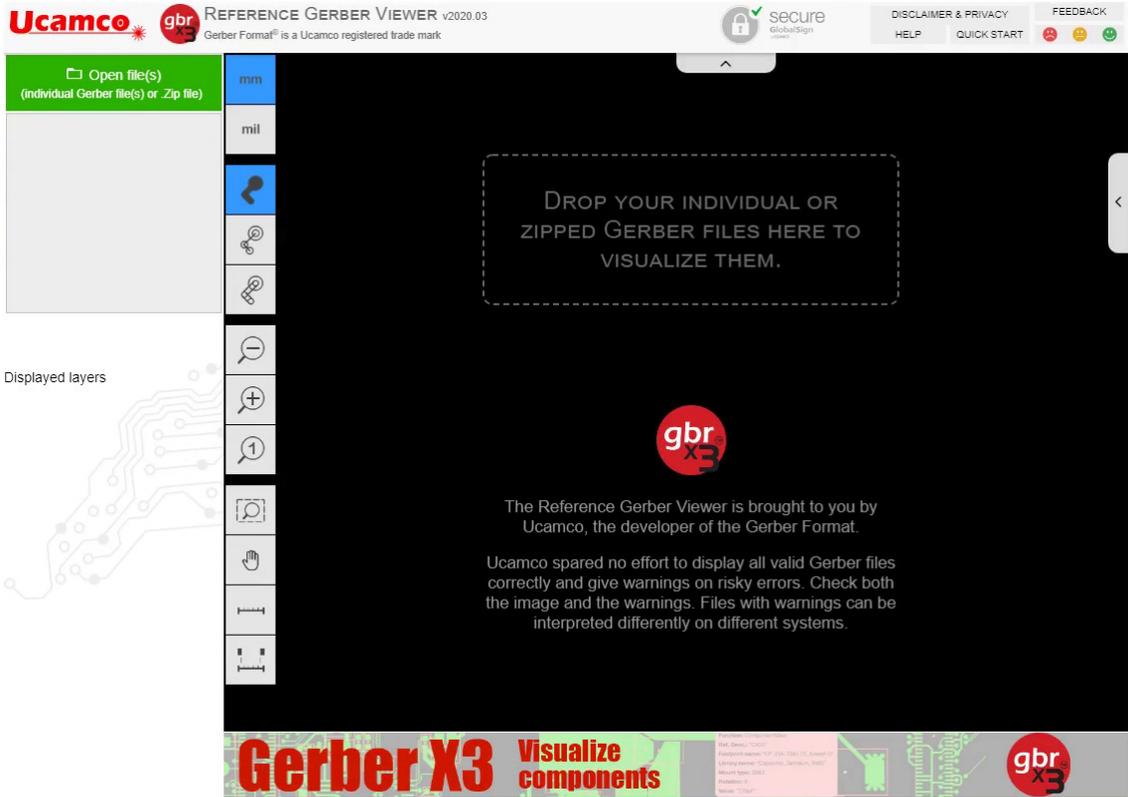
Excellon files can now contain extra header information to specify coordinate formatting and file contents, including file function and other details compatible with the **Gerber X2** standard as advised by Ucamco.

Gerber Viewer embedded in NumberOne.com Web Site

In partnership with Ucamco, the owners of the Gerber format, you now have access to the **Gerber viewer** tool embedded within the NumberOne.com web site. This viewer can be reached at <https://www.numberone.com/gerbviewer.asp>

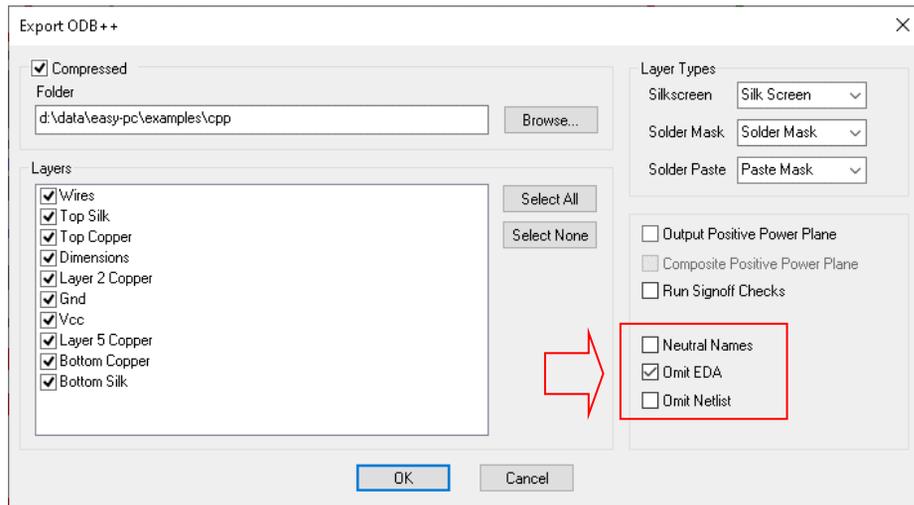
This is an important verification tool as it allows you to preview your Gerber files before you send them out for manufacturer.

To use this tool, simply drop your Gerber files or collection of Gerber files in a ZIP file onto the web site where they will be opened to view.



ODB++ IP Protection

Three new options are available on the **Export ODB++** dialog that help with protecting your intellectual property (IP) whilst still allowing manufacturing data to be produced and sent externally to your company.



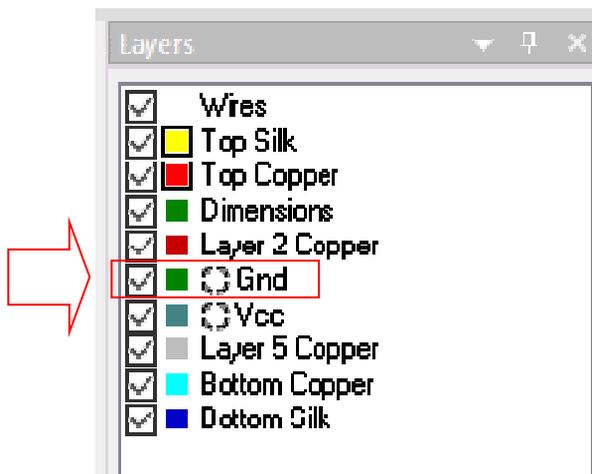
Neutral Names: net and Component (part) names are replaced with plain sequential names.

Omit EDA: leaves out the 'EDA' and Parts list (BOM) sections of the data.

Omit Netlist: leaves out the 'cad netlist' section so no net names are available.

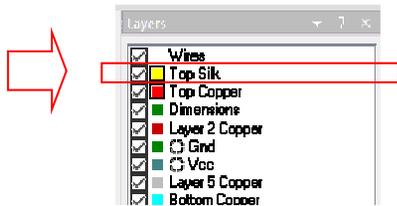
Layers Bar - Hide Power Plane

In the **Layers Bar**, when double-clicking on a **Power Plane** layer to make only that layer visible, any currently displayed power plane will be turned off. This equates to automatically using the **Hide Powerplane** option when displaying a single layer.

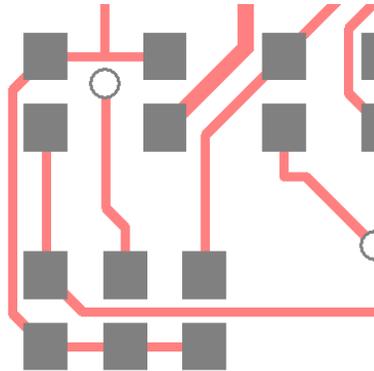


Layers Bar - Toggle True Width with Ctrl Key

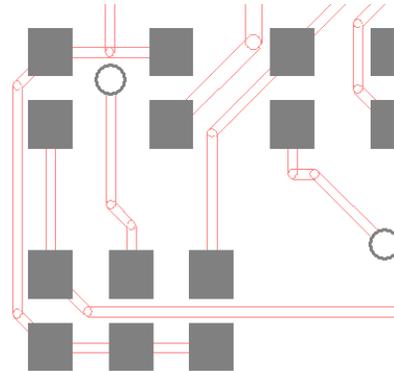
In **Layers bar**, if you single **Ctrl-Click** on the layer name, it will toggle the true size state on that layer (e.g. Tracks and Copper shapes).



With true size on



Without true size on

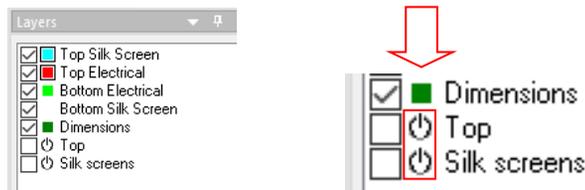


Named Layer Toggles

The **Layer Toggle** entries now have an optional **Name**. Access this by right-clicking on the **Layers Bar**, **Set Toggle Layers**.



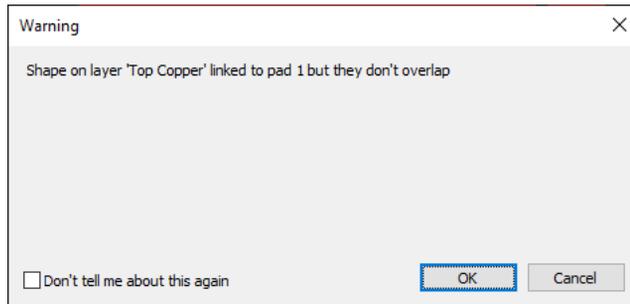
Those that are named will be listed on the **Layers Bar** below the actual layers. This allows the toggles to be enacted from there without having to be assigned to shortcut keys. Toggle entries are indicated by the “on-off” icon alongside their name:



To action a layer toggle, either click the check box (which will momentarily show as *checked* then uncheck itself again) or double-click on the name.

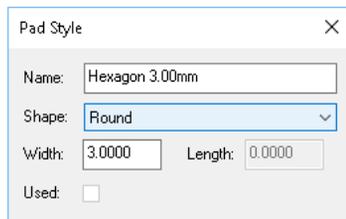
Footprint check – Copper to Pad Association

When saving a **PCB Symbol** any **copper shapes assigned to pads** will be checked to make sure that their associated pad is actually inside the shape. If they aren't, a warning dialog is displayed:



Hexagonal Pad Shapes

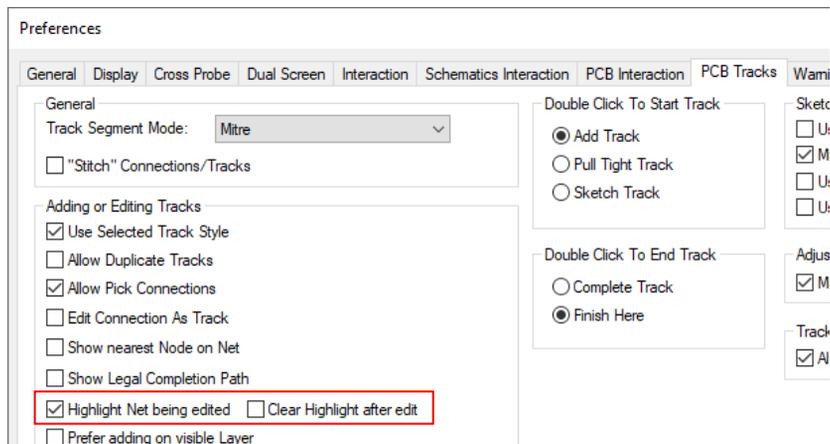
A regular **hexagon** shape has been added to the list of available **Pad Shapes** available. This can be used in both the PCB and Schematic editors.



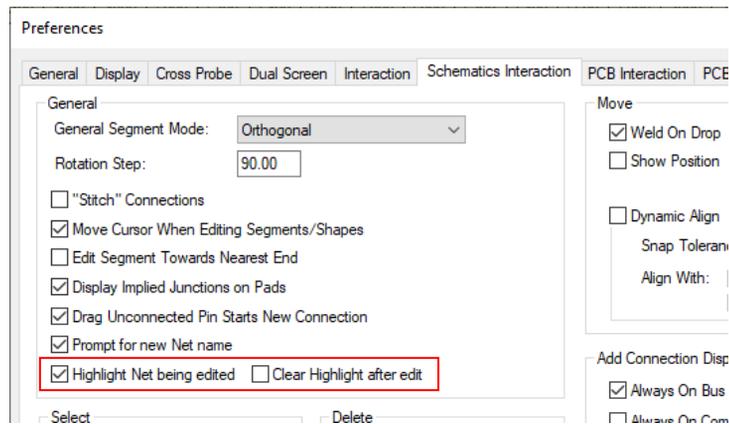
Highlight Net during Add and Edit Track

When adding or editing a **PCB Track**, **PCB Connection** or **SCM Connection**, new **Preferences** will automatically highlight the net being edited to make it easy to see the rest of the items on the net.

Preferences, PCB Tracks page:

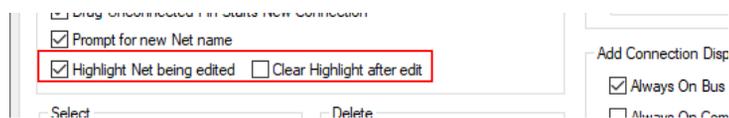


Preferences, Schematics Interaction page:



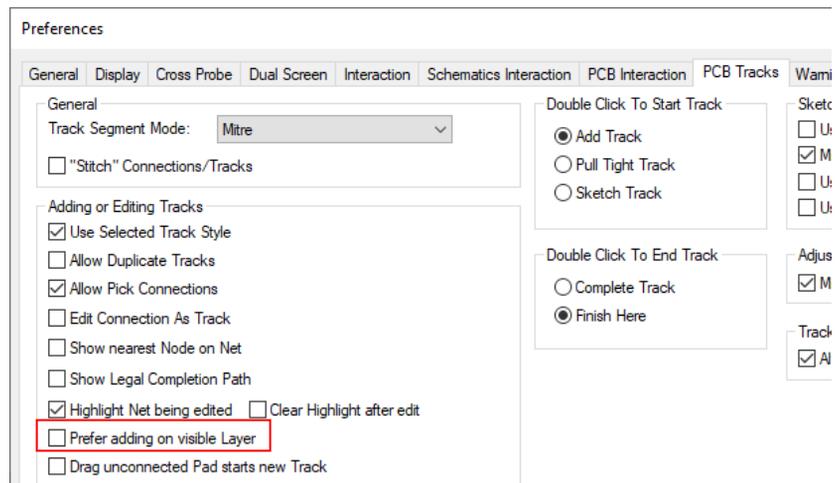
Clear Highlight After Edit option

As well as the ability to highlight during edit, there is also a new complementary option called **Clear Highlight After Edit** that when checked, will remove the highlight once editing is finished.



Prefer Visible Layer in Add and Edit Track

When adding or editing a PCB track, if the expected/default layer is not visible, checking the **Prefer adding on visible Layer** option (set in **Preferences, PCB Tracks**) will use the first visible electrical layer instead of attempting to make the hidden default layer visible.



Move Treat Vias as Fixed

When moving PCB track segments, the **Treat Vias as fixed on move Track** option causes unselected vias at track ends to be treated as fixed, instead of picking them up with the moving tracks. This avoids disturbing track patterns on other layers.

This option can be accessed from **Preferences** on the **PCB Interaction** page:

