Easy-PC

V26.0 Supplement

Copyright © 1998-2022 WestDev Ltd. All rights Reserved. E & O E

Number One Systems
20 Miller Court
Severn Drive
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: 08/08/22 Issue 1

Contents

CONTENTS **3**
CHAPTER 1. GETTING STARTED **4**
Installation 4
Running Easy-PC 26.0 5
CHAPTER 2. NEW FEATURES IN EASY-PC V26 ..... **6**
IPC-2581 Output 6
IDF Export – New Output Options 7
Remove illegal characters from Component Names 7
Include/Exclude Component Values 7
Exclude Off Board Items option for 3D View 8
Navigate to Folder feature in Document Properties dialog 8
Improved Toggle Layers dialog 9
Changes to Rulers 11
Bar Size 11
Ruler Colour 11
Interactive Ruler Stops 11
Cancel Move After Paste 12
Enhancements to the Goto Bar 12
Goto Text 12
Highlight Items in GoTo Bar 14
Plotting & Printing - Move plots up and down in the Plot List 14
Library Manager Report – Mark Missing Symbols 15
Library Manager Report – Only Show Errors 15
Library Editor Synchronise Library Names 15
Inherited Net Name and Bus Name from Bus 16
New DRC & DFM Checks 16
Min Solder Mask Width 16
Minimum Solder Mask To Track 17
Modified Pour Areas 18
Minimum Text Size 18
Ignore Same Component Errors 19
DRC only Check in Named Area 19
DRC in Schematics - Unconnected Gates 19
Invert Selection 21
Interactive Mitre/Fillet Any Angle in PCB 21
Apply Layout Pattern use Schematic Sheet 22
Track Editing - Finish Track on Start Pad not allowed 22
Grid Visibility Setting on Layers Bar 22
Updated Change Layer Span dialog 23
Updated Change Layer and Track Layer dialogs 24
Add Teardrop to Selected Pad only 24
Active Hyperlinks in Add Component and Add Component Bar 25
BOM Timestamp 26
BOM Cost Precision 27
BOM Multi-select Value Names 27

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

CDs are no longer supplied; a download link would have been supplied to you by email. Under special circumstances, product on CD media can be purchased if you contact our sales office.

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 26.0

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.

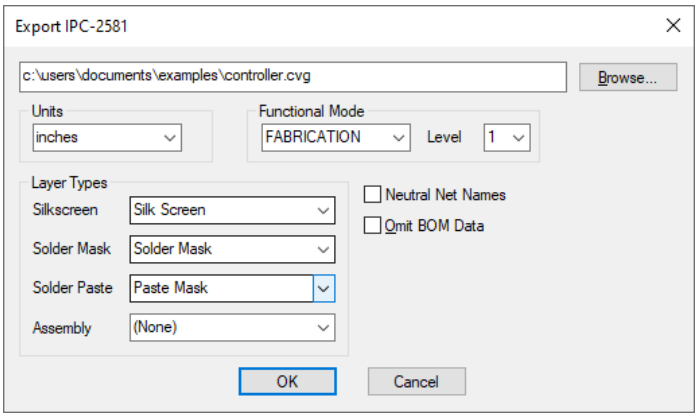
To start the program, double-click on the **Easy-PC** icon.

Chapter 2. New Features in Easy-PC V26

IPC-2581 Output

There is an **IPC-2581** export option on the **Output** menu. This option is available in the PCB Design Editor and in the PCB Panel Editor.

IPC-2581 is a modern CAD/CAM data exchange format which can be used as an alternative to Gerber, Excellon or ODB++ for manufacturing your PCBs. IPC-2581A format is exported.



Use **Browse** to change the destination of the output file name and location.

Choose the **Units** required for the IPC-2581 export. From the drop down list, choose **Inches** or **mm**.

Functional Mode

The IPC-2581 export is capable of outputting all or partial design files based on your requirements. From the drop down list, choose between **FULL**, **DESIGN**, **FABRICATION**, **ASSEMBLY** or **TEST**. Each of these modes will give you different portions of the design to suit the process it is intended for.

The **Level** drop down list is available for all outputs except FULL. This enables you to choose the detail level for each functional mode selected. Choose between levels 1, 2, or 3.

Layer Types

In the IPC-2581 format the **Silkscreen**, **Solder Mas**k, **Solder Paste** and **Assembly** layers need to be identified. Use the appropriate drop down list to select the **Layer Types** which will identify these layers. You can select **(None)** if you are not interested in the layer type.

If there are no power planes in the design these options are suppressed.

Neutral Net Names

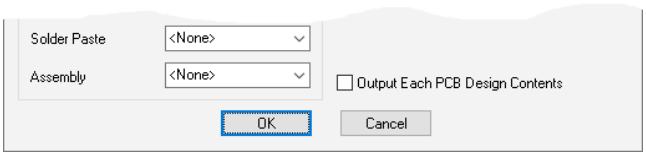
If this is checked then all user defined net names will output as numerical net names. This can be used to hide company specific net names, this will help protect your design IP.

Omit BOM Data

The **Omit BOM Data** switch is available with any **Fabrication Level** set. Use this to remove the BOM data from the IPC-2581 file to hide company specific Parts names, this will help protect your design IP.

Panel Editor Specific IPC-2581 Switches

When running the **Panel Editor**, there is a switch that is only visible when running the IPC-2581 option:



By selecting the **Output Each PCB Design Contents** check box, a copy of each of all the PCBs in the panel are exported. With it not selected, only the contents of one PCB is exported.

IPC-2581 Viewers

There are IPC-2581 viewers available on the web, these enable you to verify the output before sending to external sources, this is highly recommended.

IDF Export – New Output Options

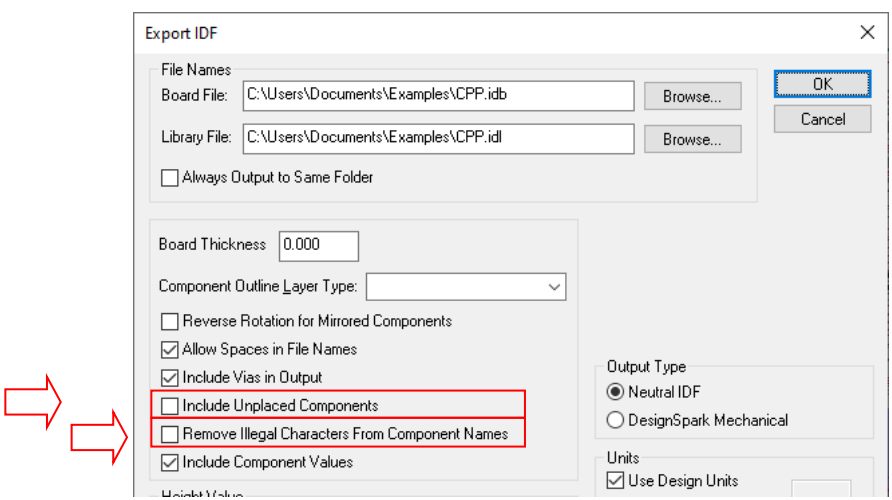
Remove illegal characters from Component Names

Include/Exclude Component Values

There are two new options in IDF export dialog:

Remove illegal characters from Component Names

Include (Exclude) Component Values



Remove Illegal Characters From Component Names - check this box to remove characters that are illegal in Component names for some programs, such as Inventor 3D. They will be replaced with an underscore. These characters include: \, /, :, *, ?, ", >, <, |

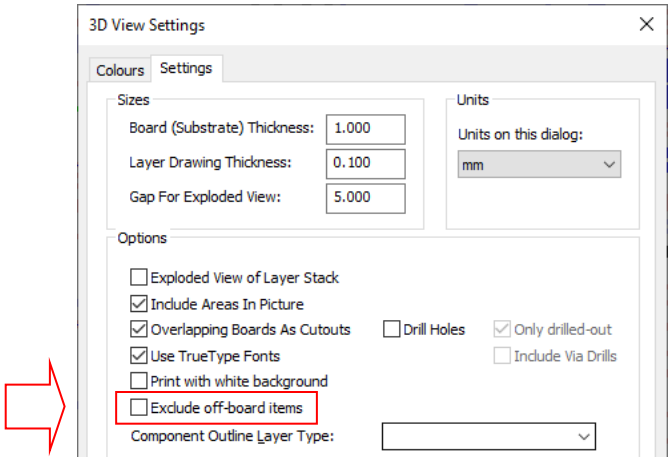
Note: The IDF export option already did this for Version 3.0 formats but this new option lets you switch it off if required.

Include Component Values - check this box to includes Component Values in the output file. This can only be for Version 3.0 formats.

Exclude Off Board Items option for 3D View

There is a new check box on the **Edit** menu, **3D Settings** dialog called **Exclude off-board items.** When selected, this will prevent components and other items that are not entirely within a board outline will be excluded from the view.

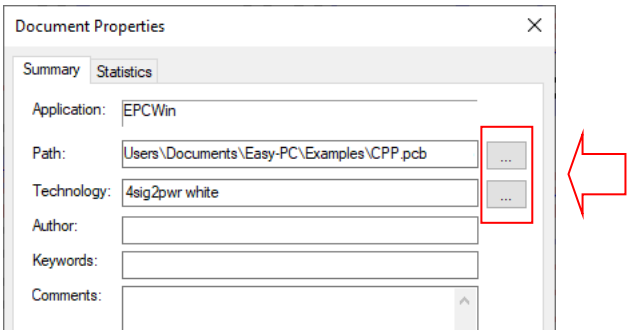
This can help for example if you have a standard drawing outline with title blocks, checking this box will exclude those portions that are not on a board.



Navigate to Folder feature in Document Properties dialog

Two small […] buttons have been added to the **Document Properties** dialog on the **File** menu, alongside the Path name of the design/project and the **Technology** file.

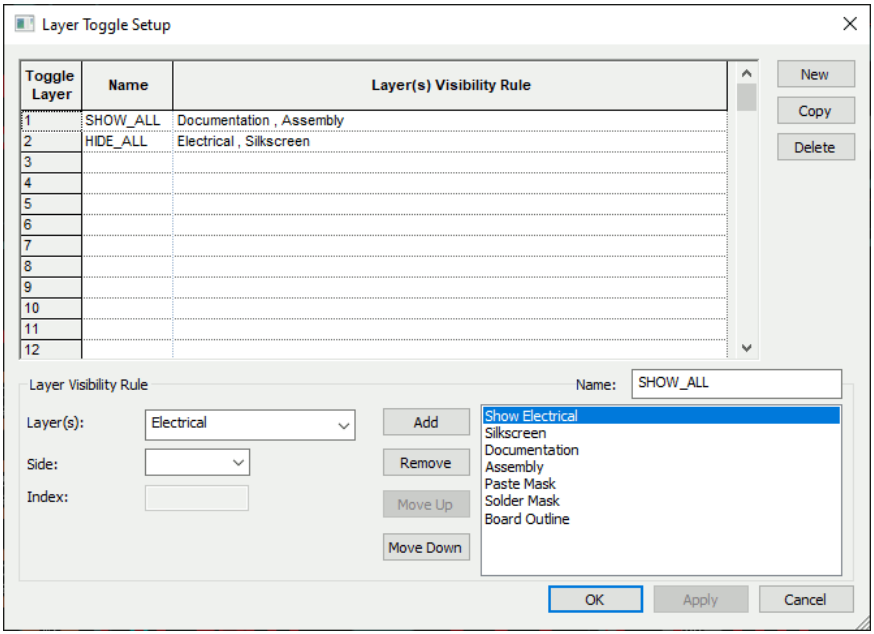
Selecting these will open the corresponding folder in Windows File Explorer.



Improved Toggle Layers dialog

The **Toggle Layers** dialog has been improved to make it more user-friendly.

The changes reflect the **Design Technology** dialog layout and use a grid to contain the different toggles, this includes the **Toggle Layer**, **Name** and **Layer/Side**.



Navigation

The **New** button is used to create a new row and will take the first available grid.

The **Copy** button will be enabled when the currently selected row (or cell within a row) contains commands. When clicked, the currently selected row will be copied to the next empty (free) row and will be selected.

The **Delete** button will be enabled when the currently selected row contains commands and is not the first row. When clicked, the currently selected row will be deleted and the last row to have commands will be selected.

The command list within the **Layer Visibility grid** section will be automatically filled with the commands in the newly selected row and the other controls will be populated with the details of the first layer rule in the list. If there are no commands in the newly selected row, then a new default layer rule will be added to the list and their details populated in the relevant controls. This is now ready to be edited.

This dialog shows all the available **Toggle Layer** commands and their, if any, **Name** down the left side of the grid alongside a slot for specifying a command detailing the layer or layers to which each should apply. The commands can be modified by typing directly into the cell or, alternatively, constructed by using the set of controls at the bottom of the dialog.

Layer Visibility Rule Controls

This section consists of a list of layer rules shown on the right side and controls to create or edit a layer rule on the left side. The list of layer rules make up a toggle layer command. The controls on the left edit the currently selected (blue highlighted) rule in the list on the right.

Controls on the left side of this section will be enabled and disabled depending on the selection in other controls. For example, you can only define a side when a Layer Type has been selected from the Layer combination box.

Like the controls on the left, the list buttons will be enabled and disabled based on the content in the list.

The **Add** button starts a new layer visibility rule and adds a default rule to the list on the right. This is now ready to edit.

The **Remove** button deletes the currently selected rule in the list. It will then select the next rule above (or below if nothing above) and automatically fill the controls on the left with their new values.

The **Move Up** button moves the currently selected layer rule up one position.

The **Move Down** button moves the currently selected layer rule down one position.

Layers, Layer Type and Side

The set of controls provided at the bottom of the dialog allow a new Layer Toggle command to be constructed or an existing Layer Toggle command to be edited.

The **Layer** combination box contains all the layer names and types in the current PCB design grouped together by whether it is a layer name or layer type. A **Layer** must be selected to create a valid toggle layer command and specifies what layer(s) the command is to act on.

As layer names can vary between designs it may be preferable to associate layers by characteristics rather than their explicit names.

Side provides the additional optional field which allow for more detailed control when a Layer Type toggle is being created.

Omitting Side and just specifying a Layer Type implies ALL layers of that type regardless of where they appear in the layer stack.

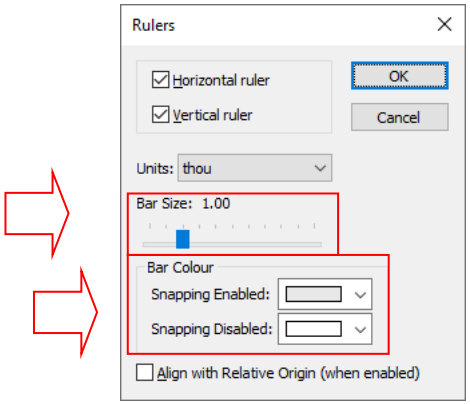
Index is available if the **Side** is selected as **Inner**. This will be a numeric value that refers to the Inner layer in the sequence it appears (in the Inner Layers list). It might be a more suitable alternative choice when specifying a specific inner layer (rather than all inner layers) to use a named layer command instead.

Omitting the Index when specifying **Inner** implies ALL inner layers, so "Electrical Inner" will toggle all the inner electrical layers.

The Layer Name, Layer Type and Side are defined in the **Design Technology** dialog.

Changes to Rulers

There are two new settings on the **Ruler Settings** context menu for **Bar Size** and **Ruler Colour**.



Bar Size

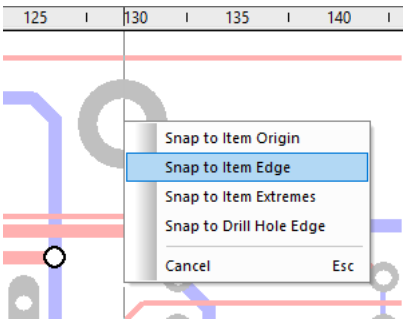
The **Bar Size** slider is used to increase or decrease the size of the ruler bar. This can be useful if you have a 4K monitor or if you wish to view it larger on a regular monitor.

Ruler Colour

Use the **Bar Colour** to change the background colour of the Ruler. Care should be taken when changing this so that you don't loose the text colour on a dark background.

Interactive Ruler Stops

When you drag a ruler stop you can now see it move dynamically. You start the drag from the bar with menu items for additional functionality available on the context menu if you right click the mouse:

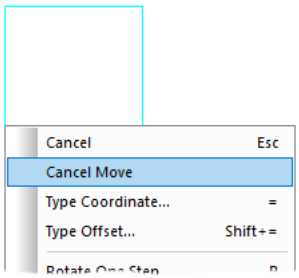


There are options on the context menu to snap the ruler stop to the item you finish on. The snap to origin, extremes and drill edge work the same as the existing options from the ruler bar. The snap to item edge option will snap to the closest edge on the item.

There are options on the context menu to snap the ruler stop to the item you finish on. Use the **Snap To Origin**, **Snap To Item Edge**, **Snap To Item Extremes** and **Snap To Item Drill Edge Hole** to snap to exact item positions.

Cancel Move After Paste

There is a new option on the context menu called Cancel Move.



This can be used when an item is copied and after paste. This will cancel the move of the item and release it back in its original position.

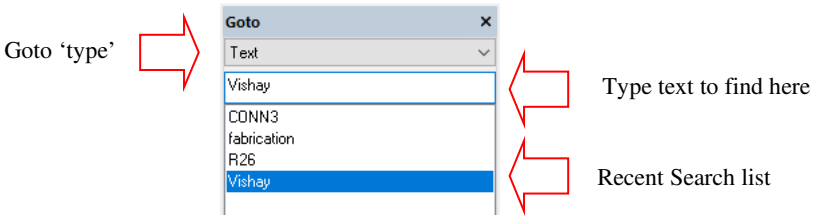
If a section of design is cut from one design and pasted to another, the offset is not preserved. That means that if the original section has been critically positioned, the pasted section must be manually repositioned to the required location without the ability to use properties and type in exact co-ordinates. The **Cancel Move** option allows the original position to be preserved.

If the item is copied using **Duplicate** instead of **Copy**, the menu entry is still present but only while the item is on the cursor and dynamic, it disappears once the item is placed.

Enhancements to the Goto Bar

Goto Text

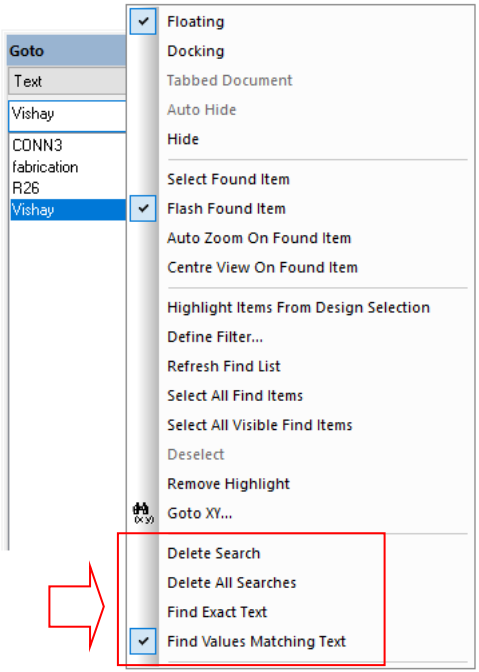
A **Text** search has been added to the **Goto** Bar, this can find all 'free' Text and Values text in your current design.



Choose **Text** in the Goto Bar as the '**type'** and enter the text to find in the box. As you search for different text items in your design, the searched list will be populated. This can be refined using the **Delete Search** option from the context menu on the selected text string.

Note, to find Component Names, like, R1, C1 etc. choose Values Text.

New options on the context menu are available for **Goto Text**.



This option can also be used to search for Values by selecting **Find Values Matching Text** from the context.

Context menu commands

Delete Search - Deletes the selected search from the list.

Delete All Searches - Deletes all previous searches from the list.

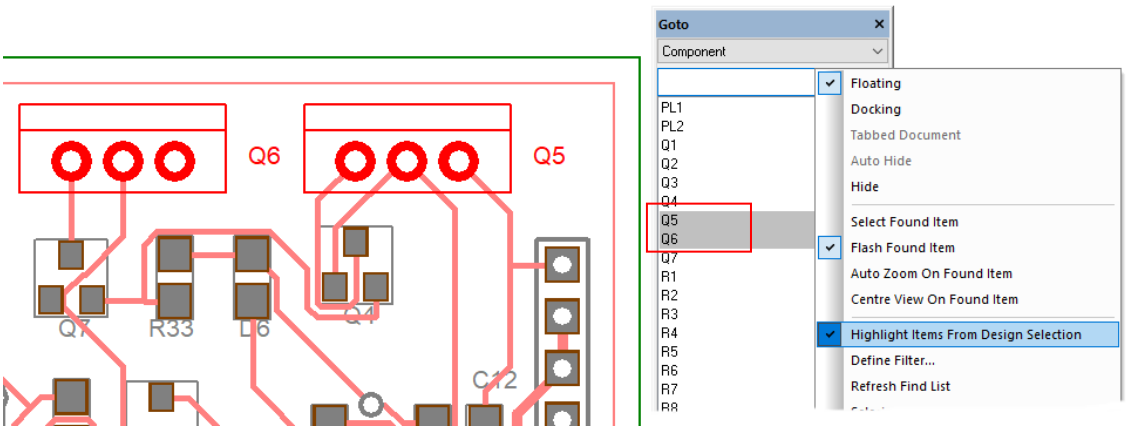
Find Exact Text - Only text matching the exact search string will be found.

Find Values Matching Text - Use this to find items with Values containing text matching the search string.

Highlight Items in GoTo Bar

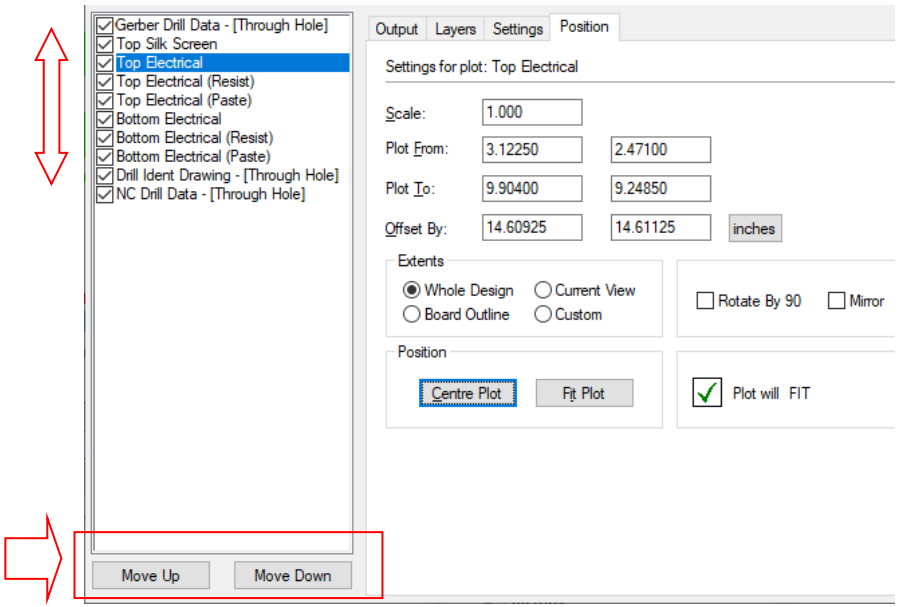
There is a new option on **Goto Bar** context menu called **Highlight Items From Design Selection**.

If you select items in the design, they will be highlighted in the Goto Bar (if it is pinned). It will highlight the relevant item e.g. if you select a Component and have Nets select in the Goto Bar, it will highlight the component net(s).



Plotting & Printing - Move plots up and down in the Plot List

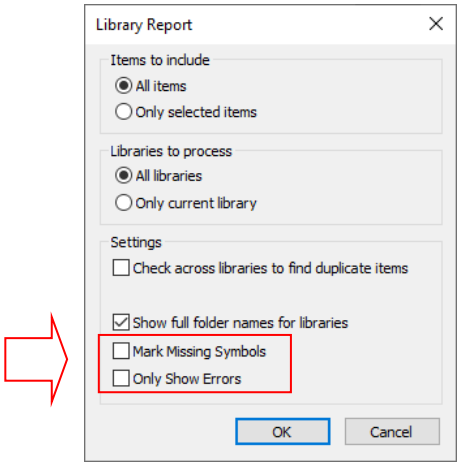
There are two new buttons on the **Plotting & Printing** dialog available from the **Output** menu - **Move Up** and **Move Down**. These buttons allow you to reorder plots in the list, to sort them by type (Gerber etc.) for example.



Library Manager Report – Mark Missing Symbols

Library Manager Report – Only Show Errors

There are two new settings in Library report dialog - **Mark Missing Symbols** and **Only Show Errors**. This report is available from within the **Library Manager** on the **Symbol** and **Component** tabs.

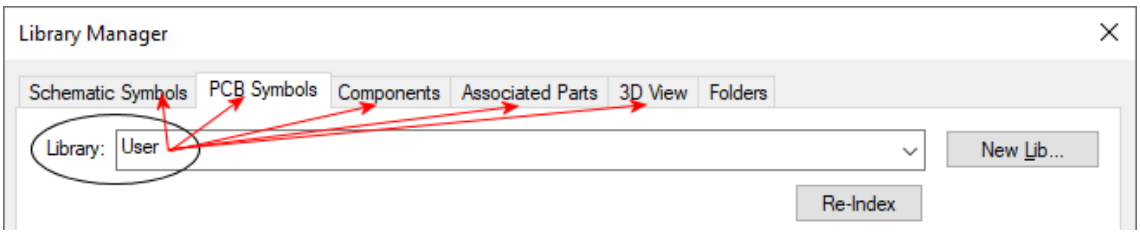


Mark Missing Symbols, will add the word, 'Missing', to the report if the symbol can't be found.

Only Show Errors, only reports missing symbols and reports where a symbol is defined, but can't be found in an accessible library.

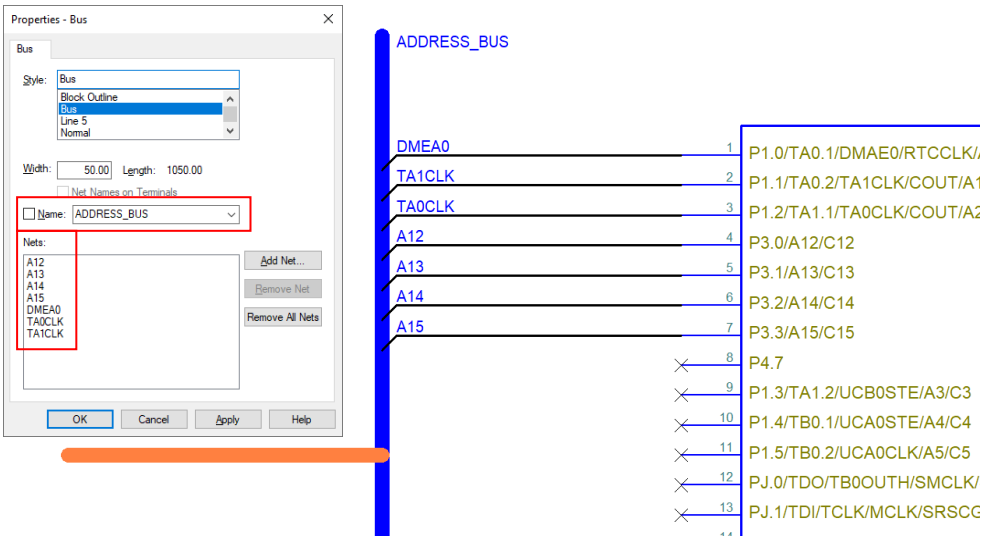
Library Editor Synchronise Library Names

When using the **Library Manager**, if a specific library is selected, **User** on the **Component** tab for example, when you now swap to another tab, **Schematic Symbols** for example, the program will try and choose the corresponding User symbol library if the same name exists. The Library Manager synchronises the library name across all tabs including 3D Packages and Associated Parts.



Inherited Net Name and Bus Name from Bus

When starting or ending a **Bus** on the segment of an existing **Closed Bus**, it will now inherit the **Net Name** and **Bus Name**.

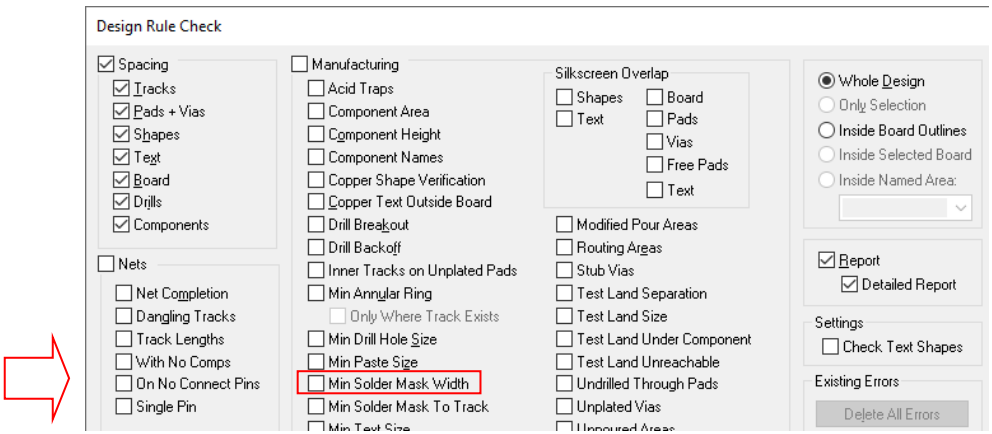


New DRC & DFM Checks

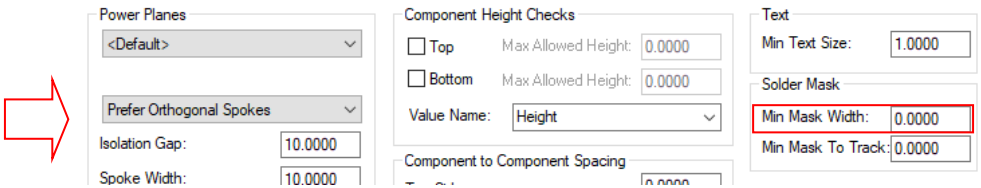
New checks have been added to enhance the Design Rules Checking for various design rules and Design For Manufacturing (DFM). When run, these checks will validate your design so that you have an additional level of confidence when sending it off for manufacture.

Min Solder Mask Width

A new check for **Min Solder Mask Width** has been added to the **Manufacturing** section in **DRC**.



The minimum mask to track value can be set in the **Design Technology** under **Rules** and **Solder Mask**.

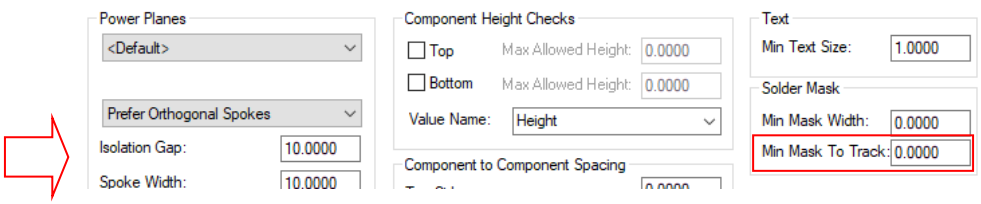


Long thin slivers of solder mask can potentially peel off during manufacturing, especially when under heat and should therefore be avoided.

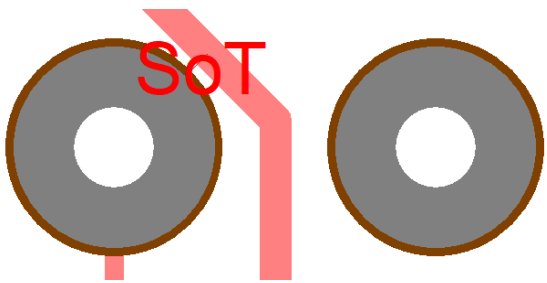
Minimum Solder Mask To Track

A new check for **Min Solder Mask To Track** has been added to the **Manufacturing** section in **DRC**. The solder mask to track check will check any layer with a layer type that has the Min Mask To Track value defined (this value can only be defined on Non-Electrical layer classes).

The minimum mask to track value can be set in the **Design Technology** under **Rules** and **Solder Mask**.

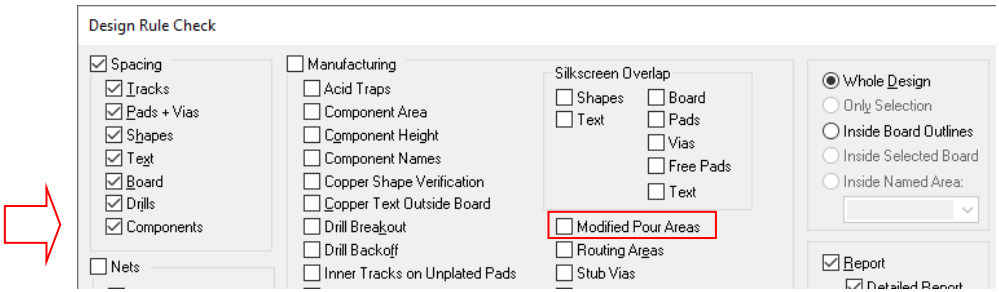


This check ensures there is a minimum distance between a track and a solder mask opening (not including the opening for the tracks start and end nodes if they exist).



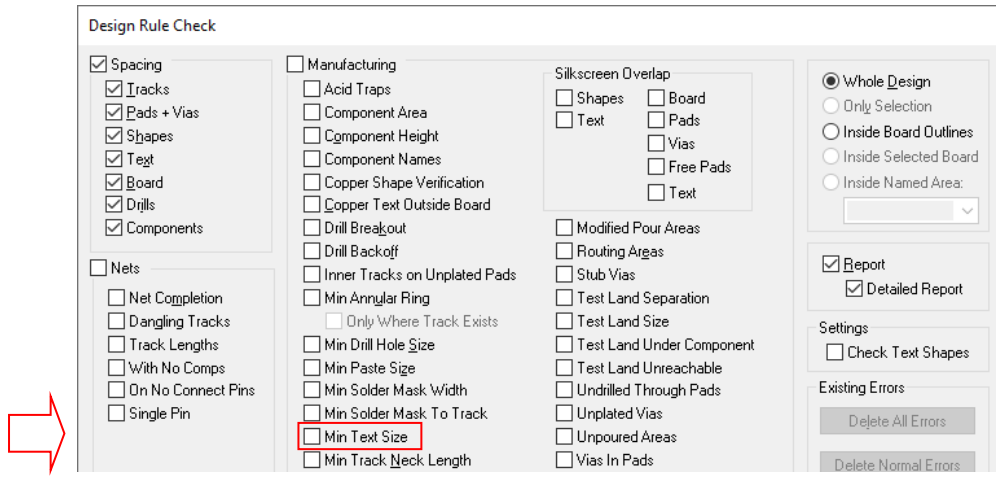
Modified Pour Areas

A new check for **Modified Pour Areas** has been added to the **Manufacturing** section in **DRC**. When run, this check will give an error if a area has been modified (i.e. the area shape has changed, items inside area have changed or moved etc.) but the area has not been repoured. It requires the area to be poured first though. In addition, also run the additional existing DRC check - Unpoured Areas.

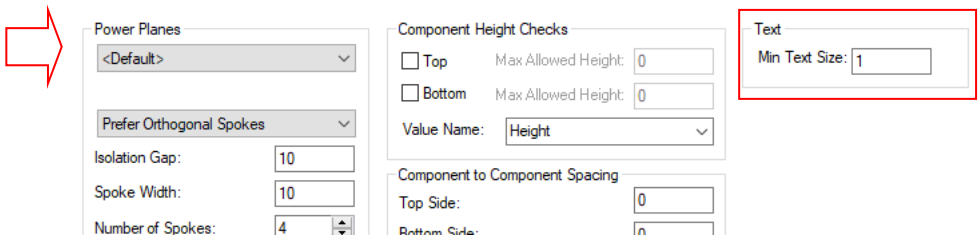


Minimum Text Size

A new check for **Min Text Size** has been added to the **Manufacturing** section in **DRC**.



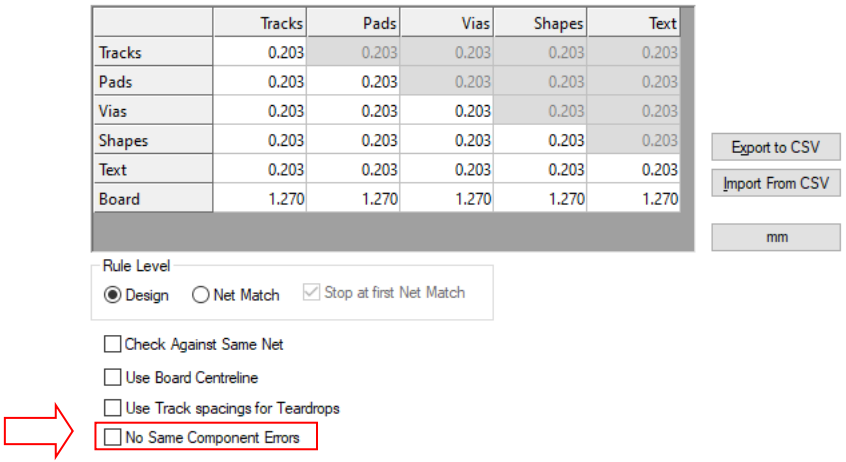
The min text size is defined in the **Design Technology**, **Rules** dialog on the **Settings** menu. Defining and running this check will ensure that all text added to your design is manufacturable, for example, copper or silkscreen text.



Ignore Same Component Errors

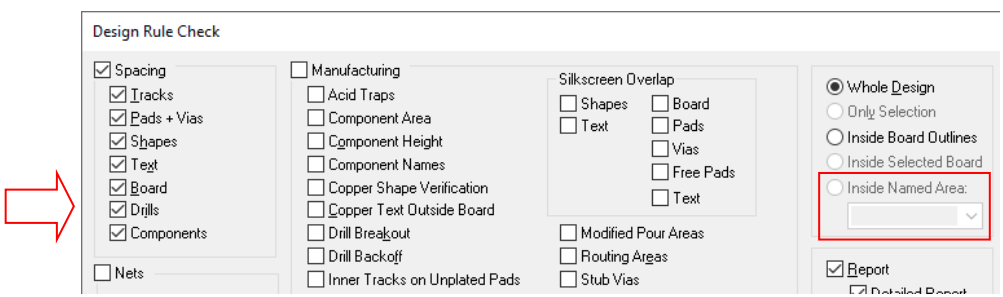
There is a new setting in **Design Technology** and **Spacings** for **No Same Component Errors**.

When set, DRC will ignore errors if they are in the same component, for example pad to pad spacing, or Silkscreen overlap checks.



DRC only Check in Named Area

From within the **DRC** option, there is now a check box to only check for errors inside a specific **Named Area**.



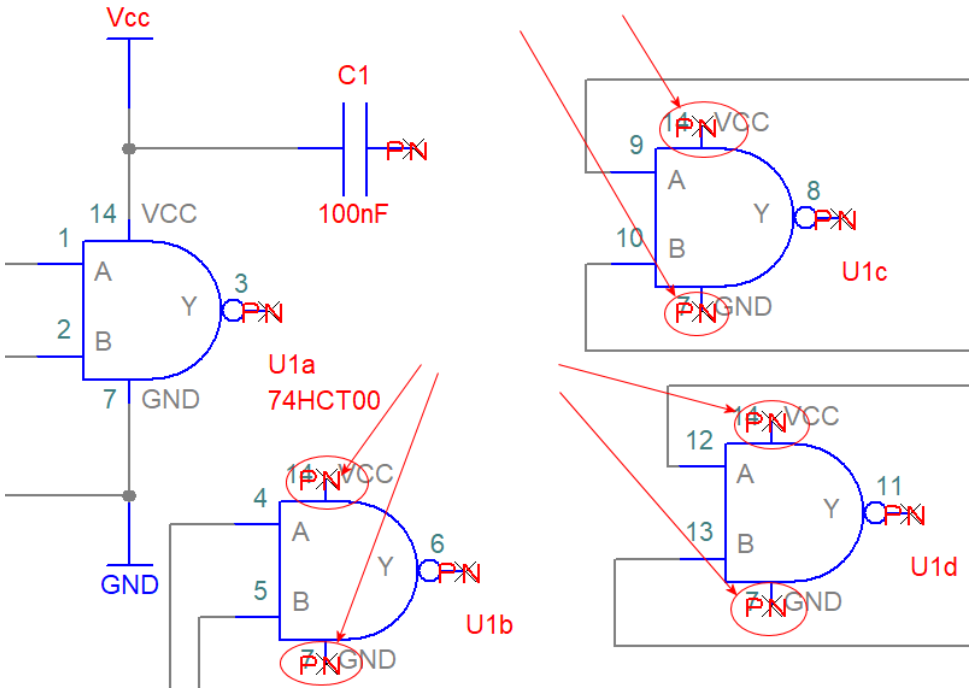
Named areas are added using **Add Component Area** and then named using **Properties** and setting the name there.

DRC in Schematics - Unconnected Gates

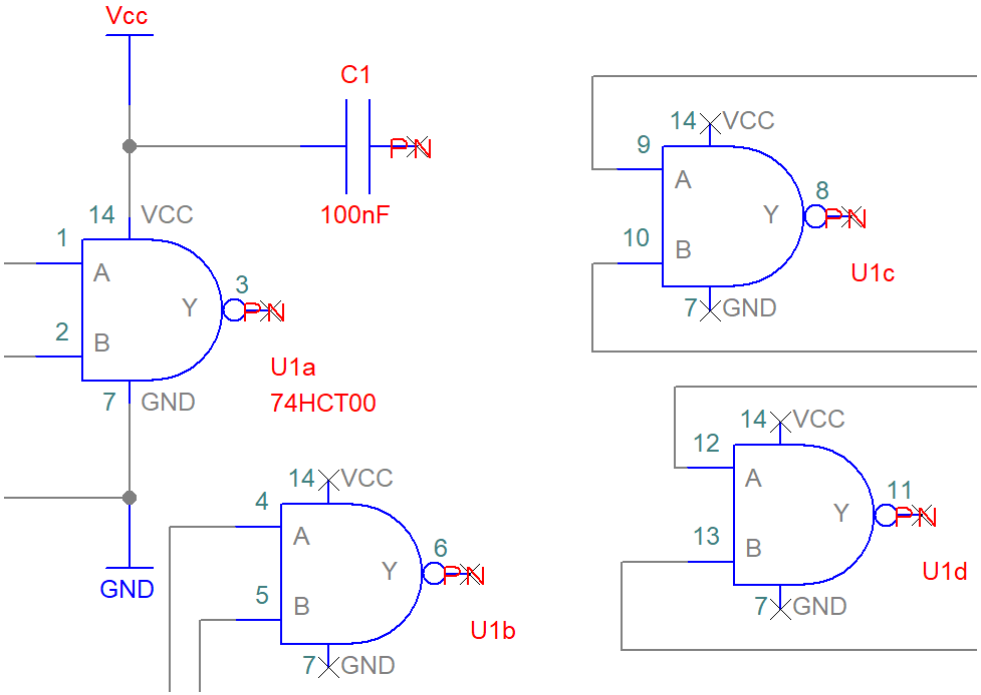
Within the **DRC** option in a **Schematic** design, reporting common power pins as **unconnected** on additional gates now no longer reports this as an issue.

Where gates are defined separately for heterogeneous devices, they may contain a power pin, especially if the same symbol has been used multiple times. Where the gate contains power pins for example, for the device to function, only one set of power pins must be connected on one of the gates. In this instance, this flagged an error in DRC. In Version 25 and previous version, common pins 7 and 14 (for example) are connected on one gate but are flagged as "unconnected" on the other gates even though the device overall has the correct power pins connected.

In Version 25, it would be reported as an error like this on the additional three gates:



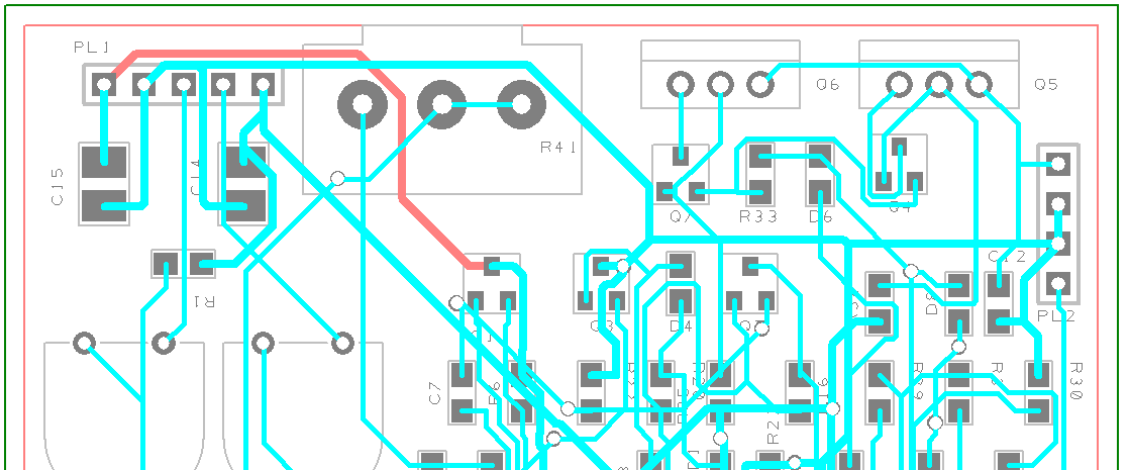
In Version 26, these errors are now longer reported when DRC is run:



Invert Selection

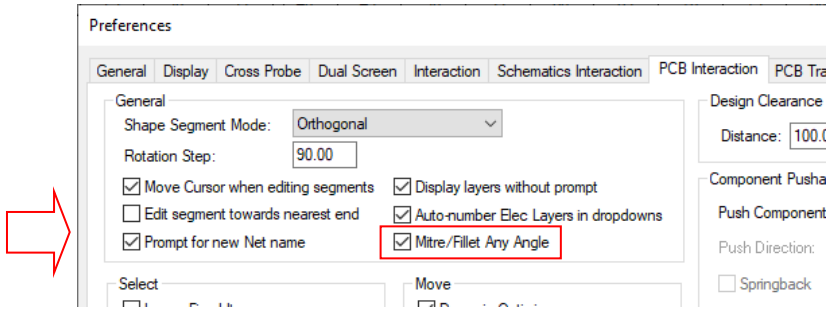
There is a new option on the context menu under **Select**, named **Invert Selection**. If you have items all of the same type selected, the selection will invert all items of that type. If you have different types selected, all unselected items will be selected.

A practical example might include using **Shift-select** to pick one track path, **Invert Selection** to select all other tracks and then run the **Mitre Selected Tracks** or **Smooth Selected Tracks** options.



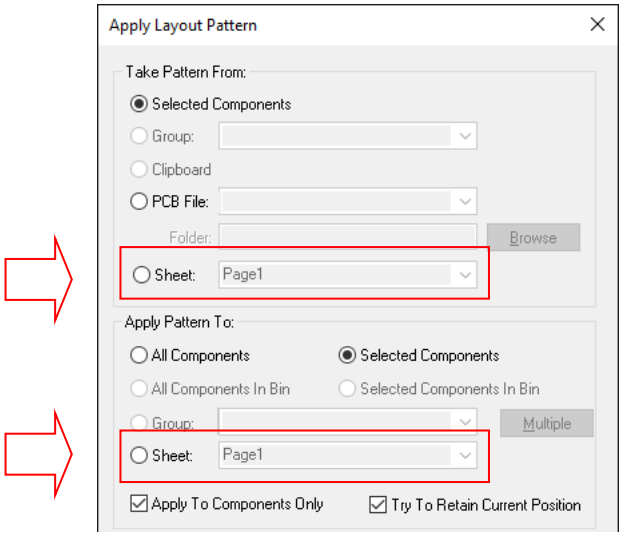
Interactive Mitre/Fillet Any Angle in PCB

In PCB, you can now interactively add/edit Mitres and Fillets for track that are at any angle instead of just right angles. To enable this option, use the **Mitre/Fillet Any Angle** switch on the **Preferences PCB Interaction** dialog. With this option disabled, you can only mitre orthogonal corners.



Apply Layout Pattern use Schematic Sheet

On the **Apply Layout Pattern** dialog, there is now an option to select components from a Schematic **Sheet**. This is available for the Source pattern and the Target pattern.



Select an existing Schematic design sheet from the **Project**. If the PCB was created without a project, only one sheet name will be available.

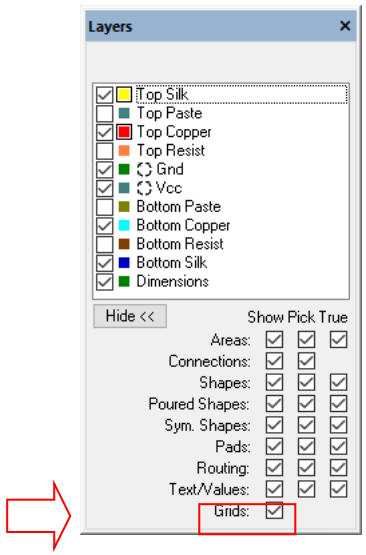
Track Editing - Finish Track on Start Pad not allowed

You can no longer finish a track when the track still inside the pad that it starts on. There are no practical situations where a track ever needs to start and finish within the same pad area, so the editing function has been tightened up to avoid this. Generally speaking, these are called dangling tracks and should be avoided.

If you have created this situation on existing designs, they can be detected using the **Design Rules Check** option and subsequently resolved by manually editing the offending track. Use 'Next' from the context menu to select buried track (key <N>) or hold <Alt> when left clicking.

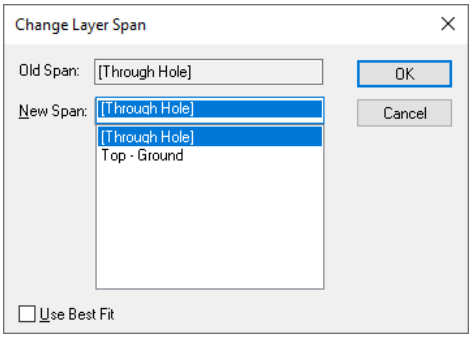
Grid Visibility Setting on Layers Bar

The **Layers Bar** has been updated to include the **Grids** check box on the bottom pane. Checking or unchecking this will change the display status of the Grids to either on or off.



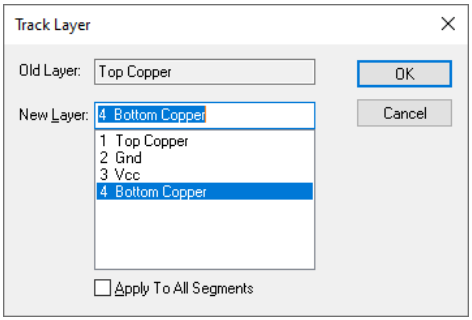
Updated Change Layer Span dialog

The **Change Layer Span** combo box is now a list and matches **Change Style,** it is a simple selection rather than a drop down, this will help save on mouse clicks.



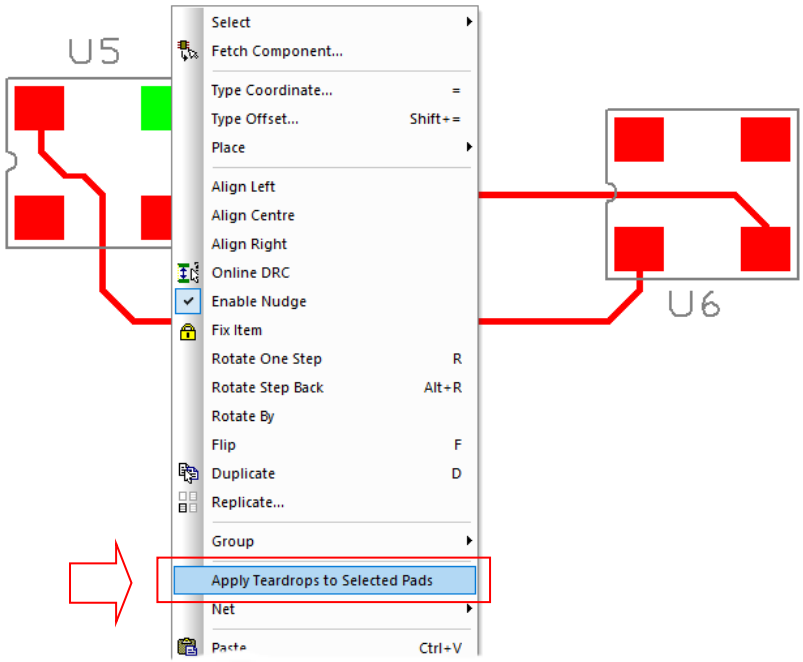
Updated Change Layer and Track Layer dialogs

The **Change Layer** combo box is now a list and matches **Change Style,** it is a simple selection rather than a drop down. This new style is used on **Change Layer** and **Change Track Layer**.

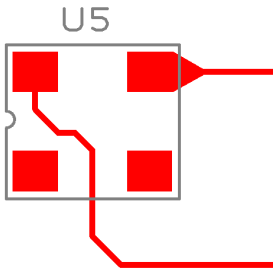


Add Teardrop to Selected Pad only

If you have a pad selected, you can use the new option to **Apply Teardrop to Selected Pad**.



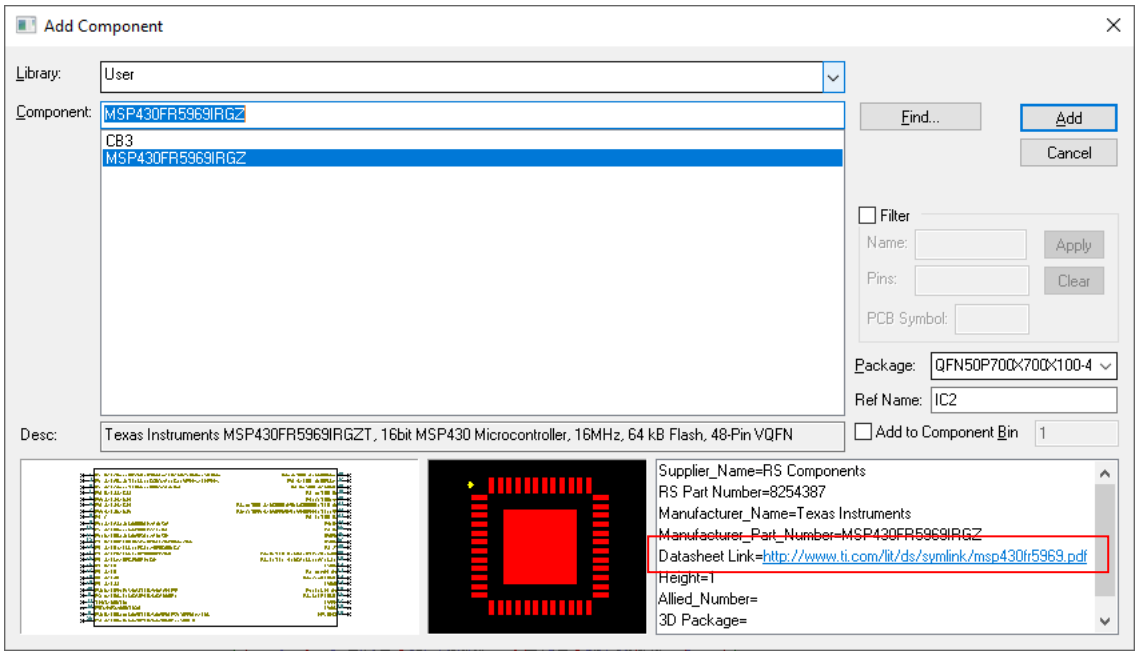
The result will look like this:



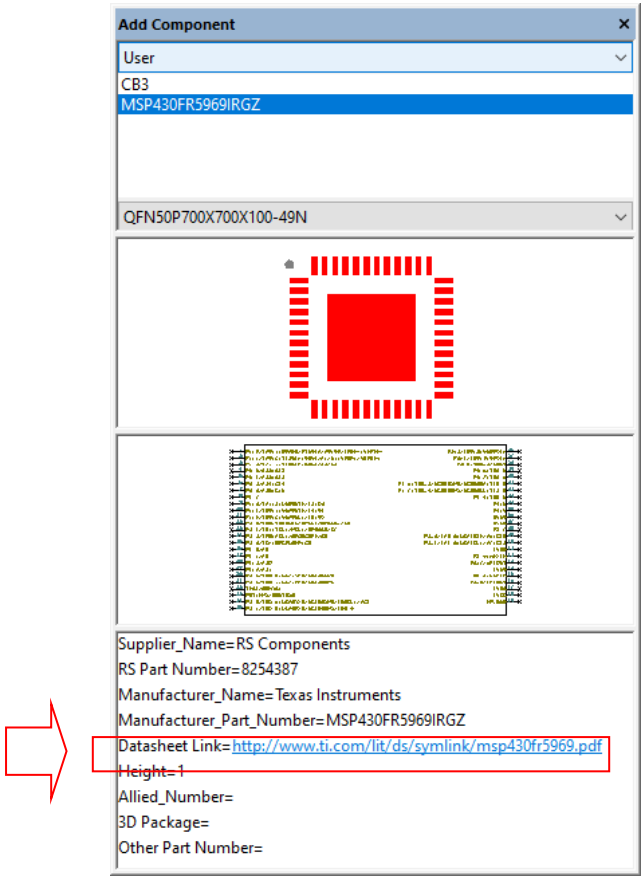
Likewise, once the teardrop is added, you can select the pad and remove the teardrop using the **Remove Teardrop from Selected Pad** option.

Active Hyperlinks in Add Component and Add Component Bar

In the **Add Component** dialog and **Add Component Bar**, Values that are hyperlinks will now be shown highlighted in blue. If you double click on the link, it will open it.

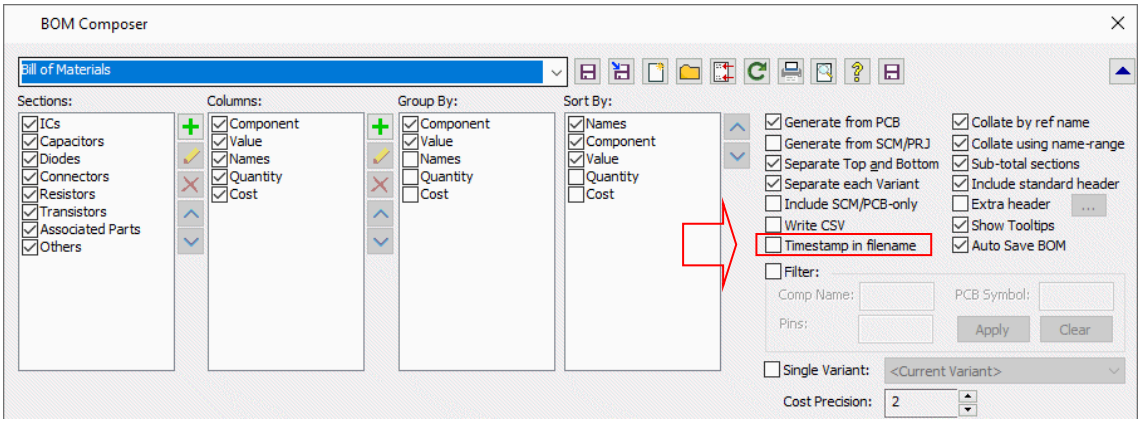


The **Add Component Bar** shows this in the Values pane:



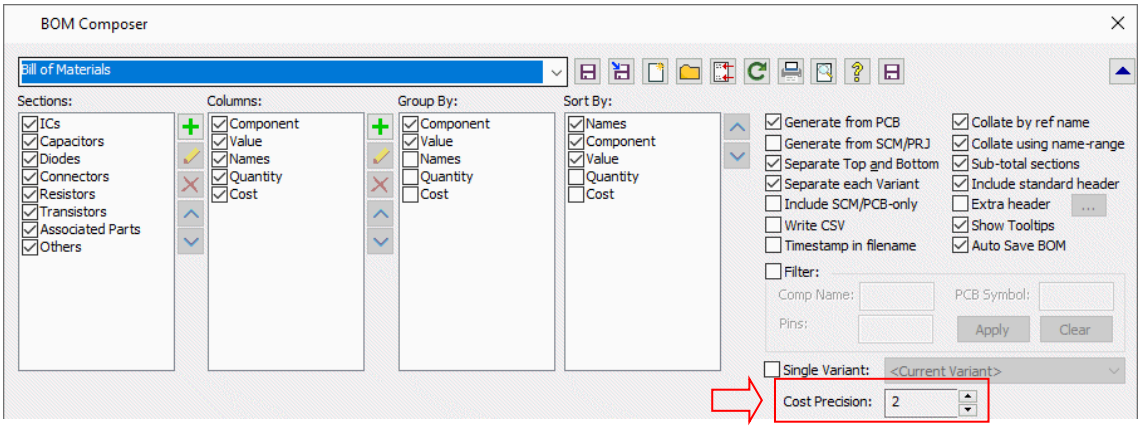
BOM Timestamp

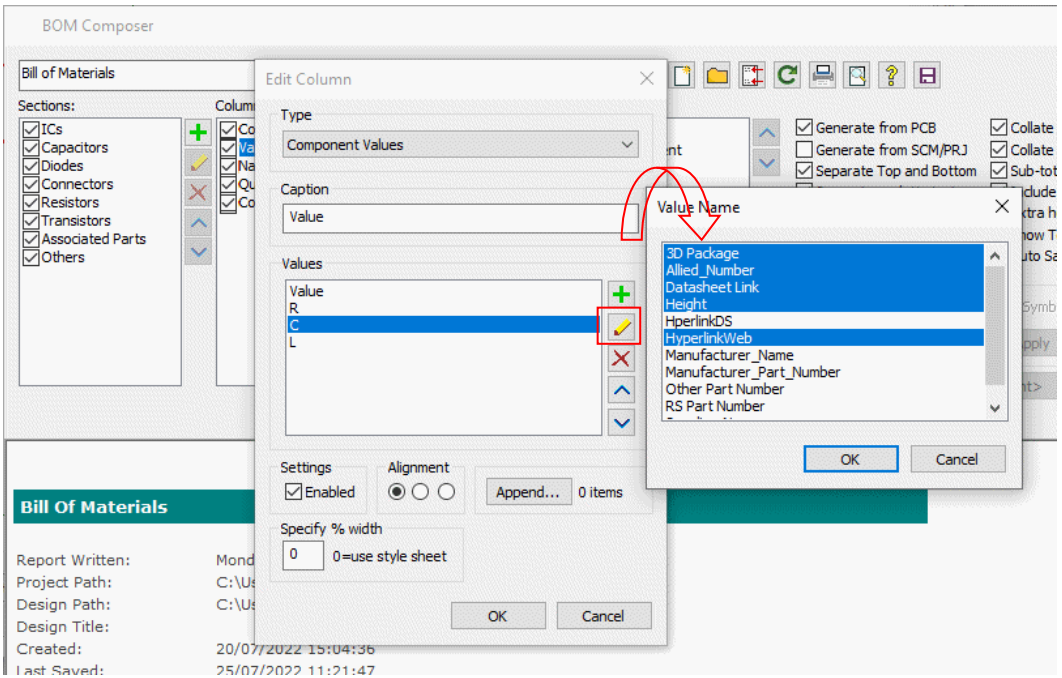
You can now add a **timestamp** to the BOM filename to make it unique.



BOM Cost Precision

You can now change the precision of the component cost.



BOM Multi-select Value Names

When setting up the **Values** column, you can now select and add multiple Value names using the **Shift** and **Ctrl** keys.