Easy-PC

V28.0 Supplement

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

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: 15/08/24

Contents

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. See below for more details.

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.

Installing over existing Easy-PC software

You cannot install Easy-PC into any folder that already has any contents. You must install into a new empty folder.

A new installation of Easy-PC 28 will present you with the default folder during installation. We recommend that you do not change this. If you do, please select an empty or non-existant folder to use.

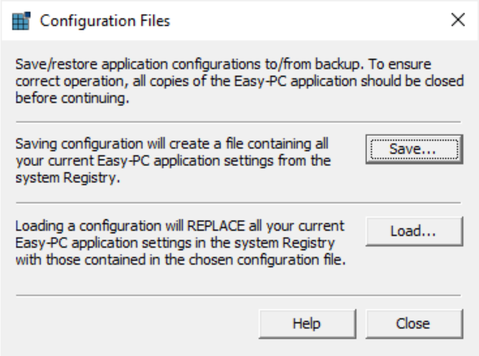
If you wish to install and use the new version without removing the old one, you will need to install the new version into a different folder. The two versions will then operate independently and either can be un-installed without preventing the other from running.

If you have customised any Easy-PC 27 menus, toolsbar or user defined settings in your old version, then create a Configuration file first that can then be read into V28 to retain these settings, see below. They will NOT be transported from V27 to V28 automatically.

Saving Previous Version Settings

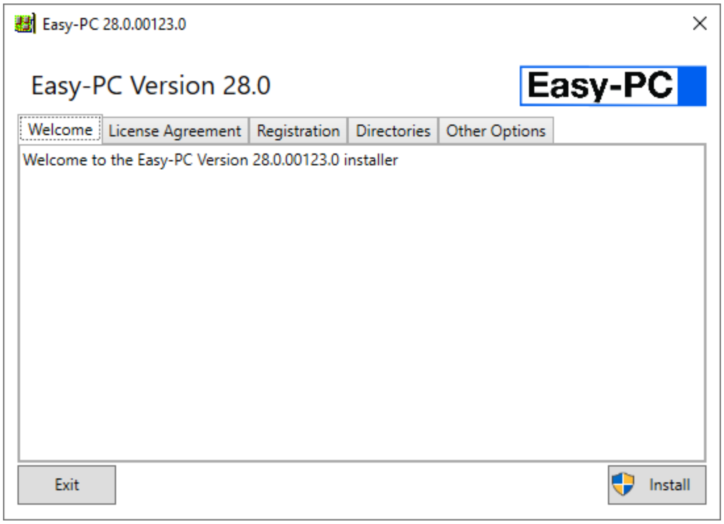
Version 28 now has its own registry settings. This means that if you have customised any **toolbars** or **Preferences** etc, these will not be passed forward into the new version. Going forward with the next version, they will be carried forward.

However, if you use **Configuration Files** and the **Save** option (from the **Start** menu, **Number One Systems** folder) you can save your setting to **Load** into the new version.



New Installer

Easy-PC 28 is provided with a new install program. It provides the same functionality but looks slightly different.



You will need enter each tab to provide the information required, such as agreeing to the License Agreement, your Registration details and product install code. The two other tabs inform you where the product will be installed and changing file extensions.

Uninstalling Existing Easy-PC Software

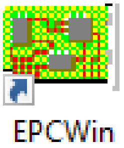
Uninstalling will still remove shared registry entries, so it is recommended that a configuration file be saved first using the **Configuration Files** option from the **Help** menu and **Support** option. This will provide a restore point for any settings which may be lost.

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 10 or 11 if you are installing for All Users, or into your own Documents folder if installing for current user only.

Running Easy-PC 28.0

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 on your desktop.

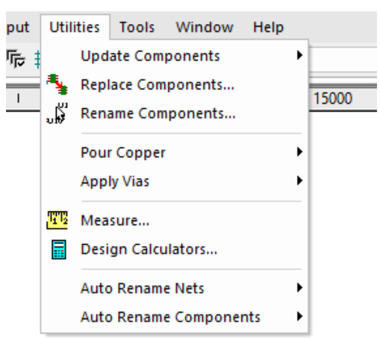
Chapter 2. New Features in Easy-PC V28

Menu and Toolbar Rationalisation

Changes have been made to the interface to tidy up and rationalise some of the menus and toolbars. The release of version 28 has also been an opportunity to add some essential functions to toolbars.

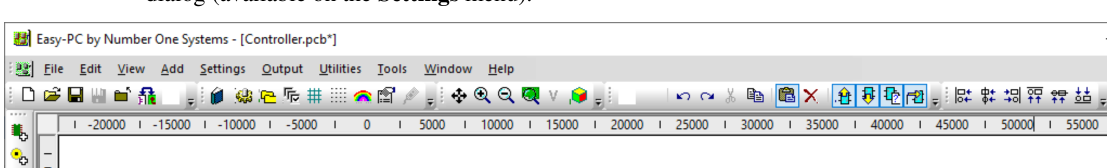
New Utilities Menu

A new **Utilities** menu has been added to PCB Design, PCB Symbol Design, PCB Technology, Schematic Design, Schematic Symbol Design and Panel Design contexts. This menu contains commands that have been moved from the **Tools** menu in order to rationalise functionality into more logical groups and to reduce the number of options on the **Tools** menu.



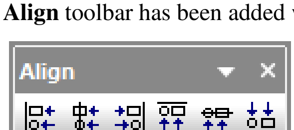
Rationalised Toolbars

The **File**, **View**, and **Settings** Toolbars have been modified to only include commonly used commands. This rationalisation will be more in-keeping with the design process. If you wish to add your own toolbar icons to these or to your own toolbars, use the **Toolbars** option from the **Customise** dialog (available on the **Settings** menu).



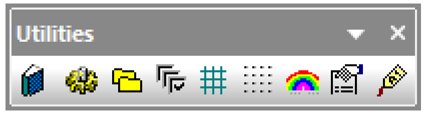
New Align Toolbar

A new **Align** toolbar has been added which contains all of the **Align** commands.



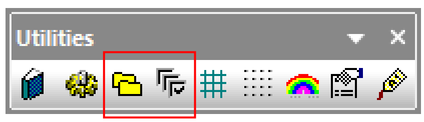
Tools Toolbar Renamed to Utilities

The **Tools** toolbar has been renamed to **Utilities** to better reflect its usage.



Folders and Preference Toolbar Shortcuts added to toolbar

Shortcut icons have been added to the **Utilties Toolbar** to include the **Preferences** and **Setup Folders** commands.



This will provide you quick access to these commonly used features.

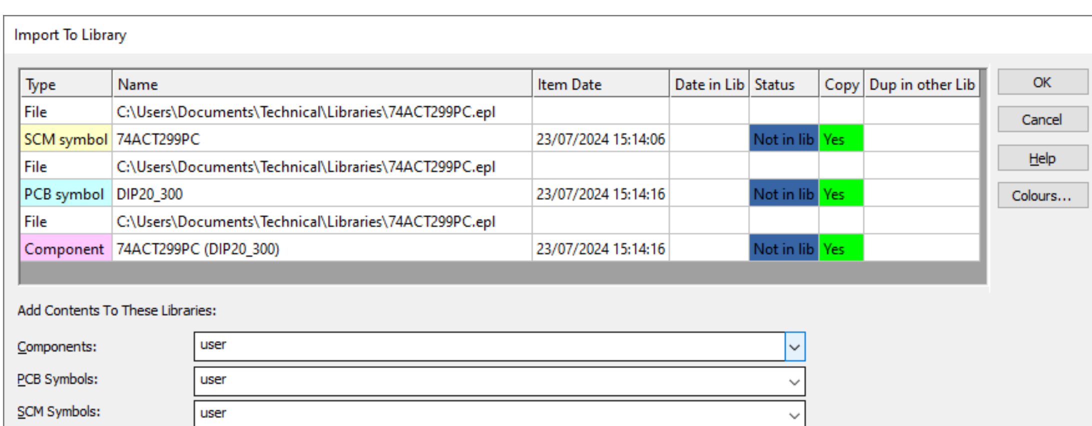
.EPL Files Imported to Create Full Library Content

When doing a library import (drag and drop) using an EPL file, you can now import Schematic Symbols, PCB Symbols and Components at the same time (provided the epl file contains them, some don't).

EPL files are created and downloaded from third party component content creators, such as Samacsys, SnapMagic, Ultra Librarian or PCB Libraries. This content is ready to use once stored in the Easy-PC library system. EPL files provide an alternative if you are not using one of the integrated systems using the Library Loader from Samacsys or the Component Search Engine from SnapMagic.

When the file is dropped, you will be prompted to select a technology file for your Schematic and PCB Symbols (it prompts once for each type). Once selected, the dialog will then populate with the library items that are included in the file, ready to be imported.

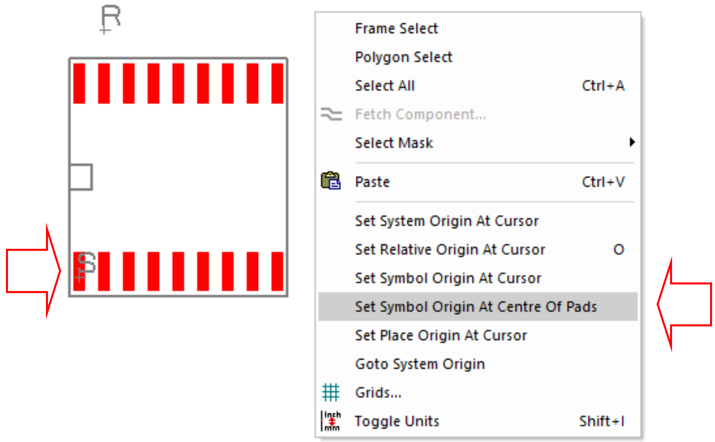
You can select which library should be used for each library type on this dialog.



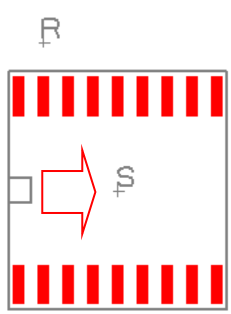
Set Symbol Origin at Centre of Pads

A new context menu command, **Set Symbol Origin at Centre of Pads**, has been added. This is availabel when editing PCB Symbols and Schematic Symbols.

When editing a symbol, with nothing selected, right click and select **Set Symbol Origin at Centre of Pads**. On selection, a centre point between all the pads in the symbol will be calculated and the Symbol's origin will be reset to this calculated point.



The result resets the symbol origin to the centre of the pad area:



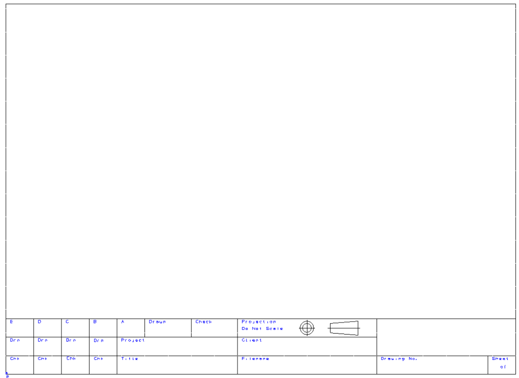
Save Copy As for Components

The **Save Copy As** command has been added to the **Component Editor**.

When editing a Component, this command will prompt you with the standard file **Save** dialog, from where you specify the path where you want the copy of the current file to be saved. The file will be saved as a self-contained .cmp file that can be imported into another library.

Import option added to Schematic Symbol Editor

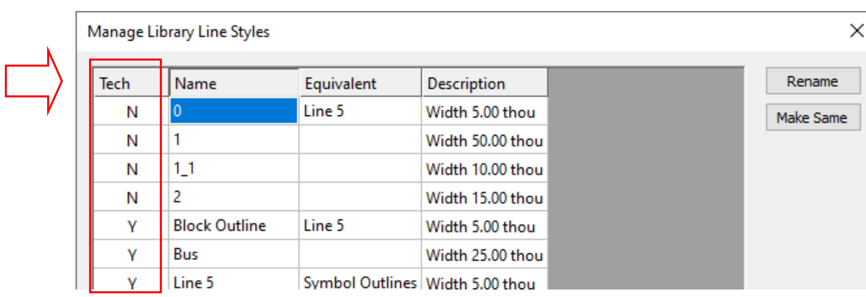
The **Import** option has been added to the **File** menu when in the **Schematic Symbol Editor**. This enables you to import DXF files to create items such as drawing borders.



Styles Tool Changes in the Library Manager

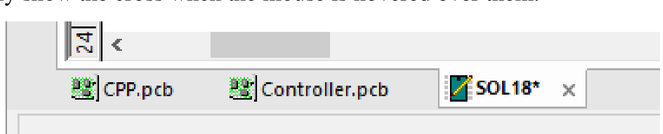
In the **Library Manager** and **Styles Tool**, the **Manage Styles** dialogs now have an extra column in the grid (Tech) indicating if the selected style exists in the **Technology** file or not. This helps you choose the merge; you may have a preference for name styles used in the Technology over local names, enabling you to keep the style consistent with the design.

The **Tech** column shows **N** for not from the Technology file or **Y** or in the **Technology** file.



Close using Cross on Window Tabs

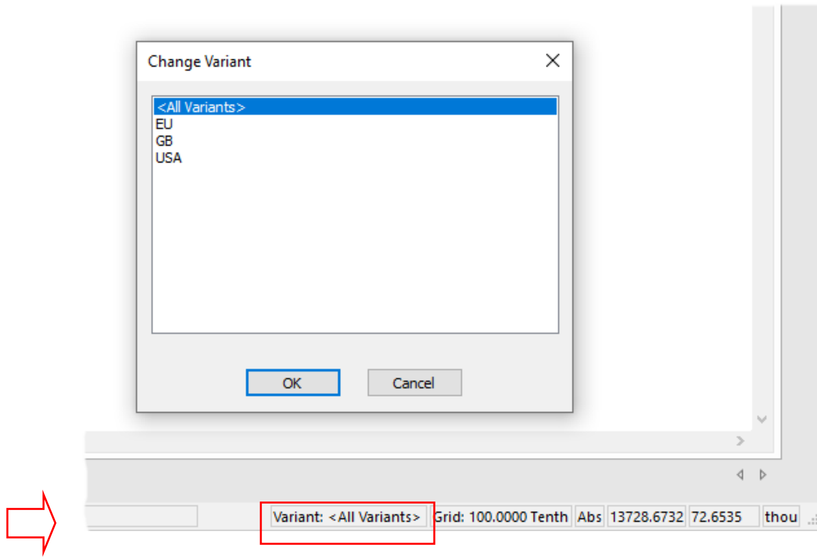
Window tabs now have a **Close** cross. The active window will show the cross, whilst other windows will only show the cross when the mouse is hovered over them.



Switch Variants on the Status Bar

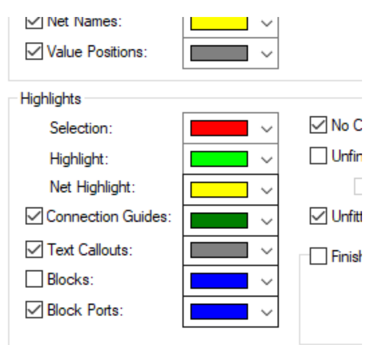
With a design containing Variants, the **Status Bar** will now show you which variant is currently active.

If you double click on the **Variant Name** in the status bar, it will open up the **Change Variants** dialog which allows you to select a new current variant.



Connection Guides Colour Option Added in Schematics

The **Schematic Colours** page now has a **visibility** and **colour** option for **Connection Guides**. Connection Guides are available when you have used the **Reverse Engineer** option to create a Schematic design from the PCB when the Schematic doesn't exist.

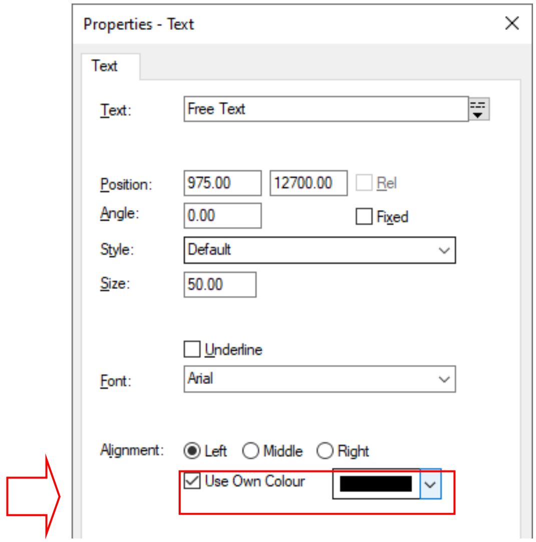


Text – Own Colour Selection

You can now set a custom colour on **free text**.

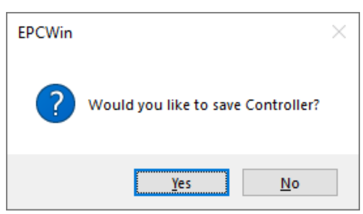
Free Text is coloured using the **Colours** dialog, but this change enables you to override that colour using the **Text Properties** dialog.

By selecting the **Use Own Colour** check box, you can then choose a colour for that selected text item or items.With this selected, you can use the drop down colours box to specify the text colour.



Forward Design Changes – Option to Save Schematic

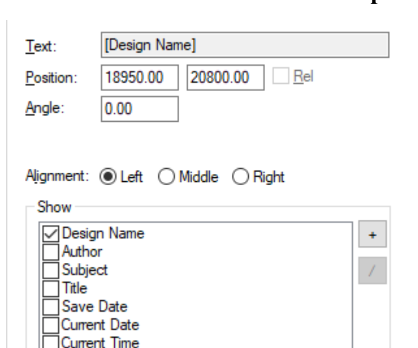
When running the **Forward Design Changes** option from Schematic to PCB, you will be prompted with the option to save any current changes to the Schematic design before continuing with the process.



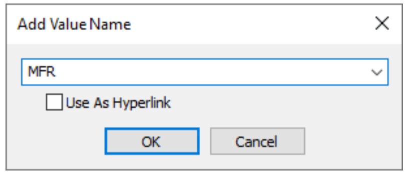
Can now add new Value Names in Value Properties Dialog

From within a **Symbol** (using the **Schematic** or **PCB Symbol Editor**) and the **Values Properties** dialog of a selected **value**, you can now directly add additional **Values**. Previously, this was only possible using the **Design Technology** dialog and **Values** page; this option is still available.

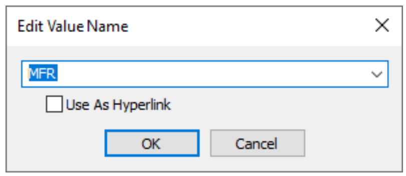
There are two new buttons on the **Values Properties** page, **+** and **/**



Clicking **+** will open the **Add Value Name** dialog where you can create a new Value name.



Clicking **/** will open the **Edit Value Name** dialog where you can edit the selected Value.



Flip Vertical and Flip Horizontal

There are some changes to the **Flip** option:

Flip is now **Flip Horizontal** in **Select** mode and **Move**, and is available on the context menus. This does exactly what the previous Flip option did.

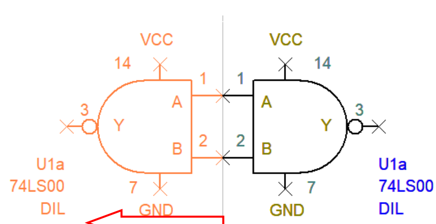
Flip has been renamed to **Flip Segments** on the **Edit Track**, **Edit Connection** and **Edit Shape** context menus.

A new option, **Flip Vertical,** has been added to **Select** mode and **Move** on the context menus. This does Flip Horizontal followed by a rotation of 180 degrees.

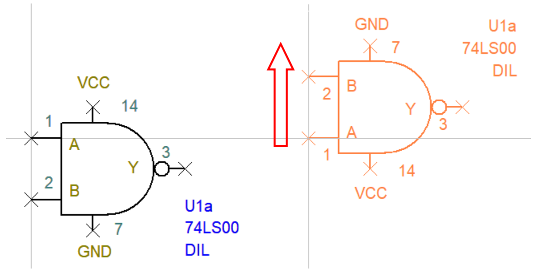
A new option, **Mirror and Swap Layer** has been added to PCB, see below.

To summarise the changes to Flip:

Flip Horizontal – this mirrors the item to the other side of the board (for a PCB). In a Schematic design, it will mirror the symbol in the X direction about the symbol origin, pin 1 in the example below, flipping the symbol from right to left.



Flip Vertical – this mirrors the item to the other side of the board and applies a rotation (for a PCB). In a Schematic design, it will mirror the symbol in the Y direction about the symbol origin, pin 1 in the example below, flipping the symbol bottom to top.



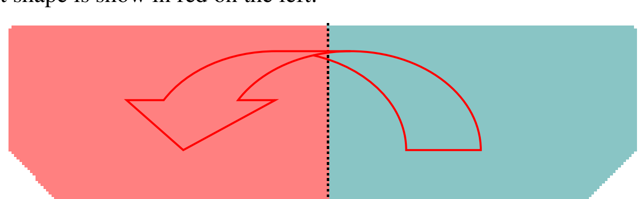
Mirror and Swap Layer Command Added in PCB

A new command, **Mirror and Swap Layer**, appears in context menu when in PCB with a layer item selected, such as Copper. This allows you to Flip the item position and Swap the layer of a selection of items assuming the design layer check is passed, meaning it can be swapped. This command is not available for a Schematic design.

The starting item is selected:

The command is executed and the resultant item is now Flipped in the X direction and mirrored to the the opposite layer:

To show the shape before and after, superimposed together, the right shape is selected, and the resulant shape is show in red on the left:

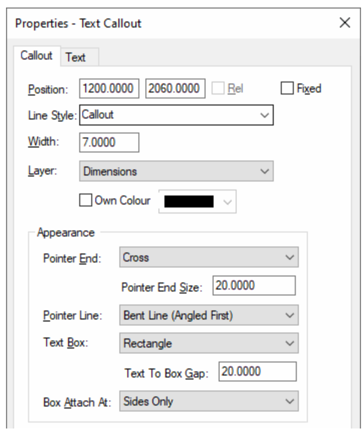


New Callout Pointer Shapes

Two new shapes have been added to Callout pointers - **Cross** and **Plus**. These are added in addition to the existing shapes available.

An example of the **Cross** shape is shown below:

Use the **Callout Properties** dialog to change the **Pointer End** type:

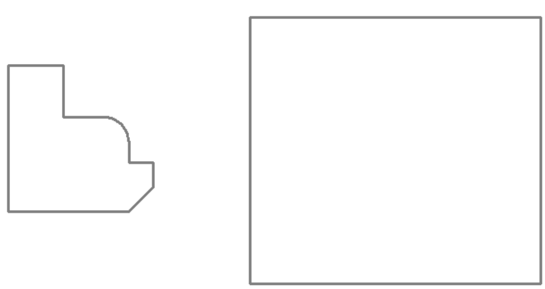


Paste copied Shape as a Cutout

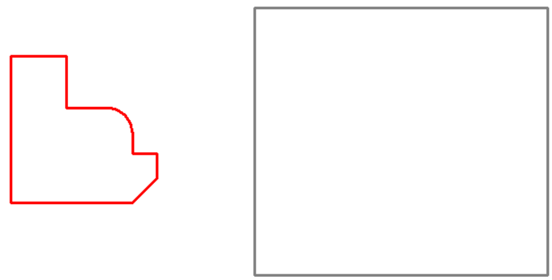
You can now **paste** a copied shape as a **Cutout**.

How to do this…

As an example below, starting with the shape you wish to copy on the left side and the shape to which you wish to add the cutout on the right side of the illustration:

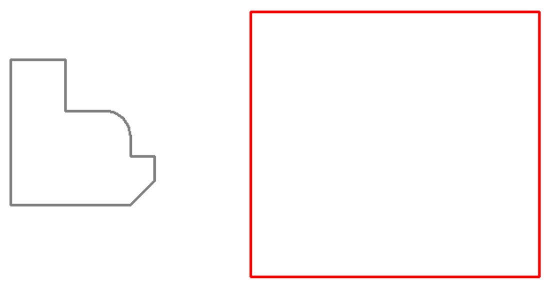


Select and **copy** the shape that you require for the cutout, select the whole shape and press **Ctrl-C** to copy it:

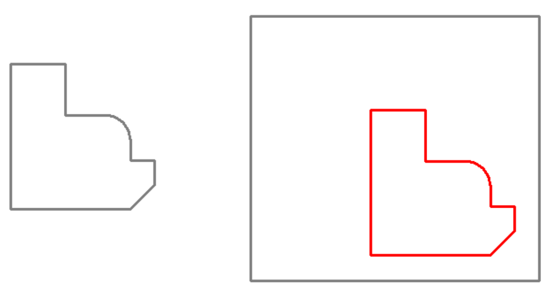


From the **Add** menu, select **Add Cutout** and select any sub option (**Shape**, for example).

Select the shape you wish to add the cutout to:



Now paste (using **Ctrl-V**) and position the cutout shape.

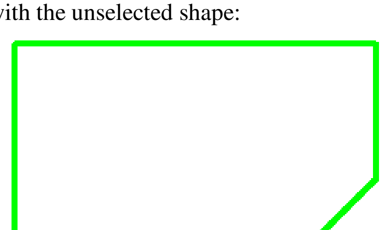


Cut Shape

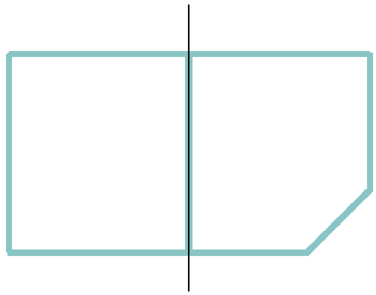
The **Cut Shape** option has been added to the **Edit** menu. This allows you to cut a shape into two or more individual closed shapes.

When run, the ’cut’ line will create two (or more) overlapping shapes, where the centre line of each shape overlap.

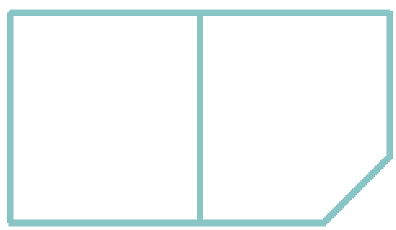
Start with the unselected shape:



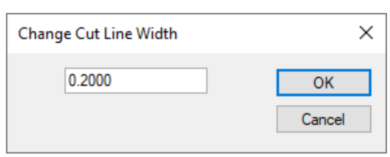
Select **Cut Shape** from the **Edit** menu and select the shape(s) to change it to. Draw the 'cut' shape:



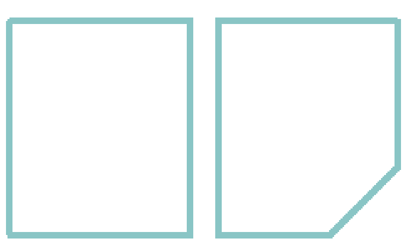
The shape will split to create two or more overlapping shapes. The centre lines of the overlapping shapes are coincident.



An alternative to having overlapping shapes is to right click with this option enabled, and to select **Change Cut Width** using option on context menu.

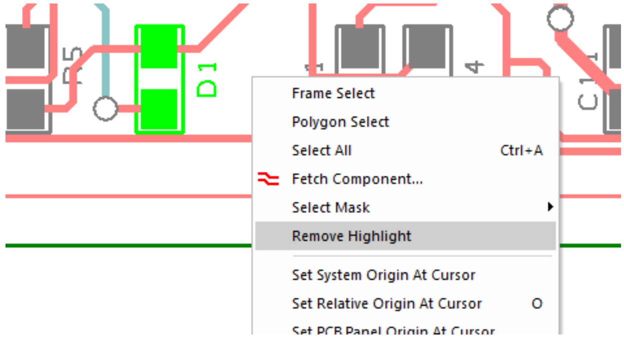


Type the cut line width, this will effectively be the 'gap' between the edges of the two shapes once applied.



Remove Highlight added to the context menu

There is a new context menu command, **Remove Highlight**, that will unhighlight when any items are highlighted in the design. This is available when an item is highlighted and nothing else is selected. This option is also available for **Net Highlights**. This is particularly useful in conjunction with the **Goto Bar**.



Component Positions Reports name changes

The **Component Positions** BOM and standard reports have been renamed to **Pick and Place** to reflect a more accurate description of their usage.

Under **BOM Composer Reports** - Component Positions renamed to Pick and Place

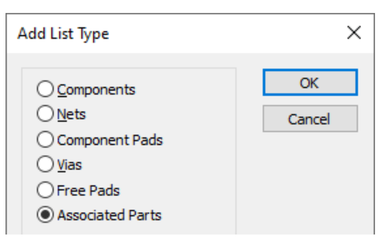
Under **User Reports** - Component Positions renamed to Pick and Place

Under **User Reports** - Component Positions CSV renamed to Pick and Place CSV

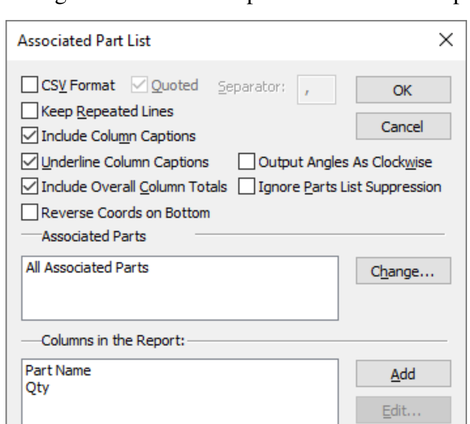
User Reports - Associated Parts List

You can now add **Associated Parts List** to your user report.

From within the **Report Editor** dialog, select **Add List…** and select **Associated Parts**.

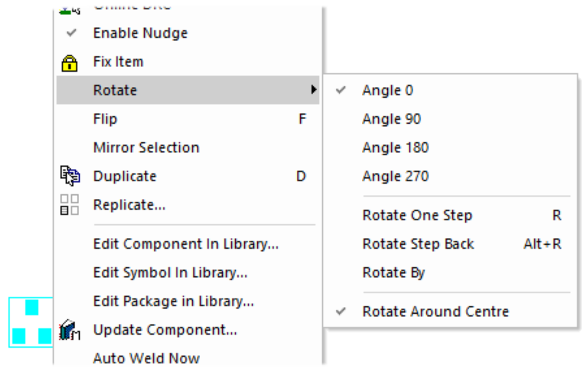


The following columns can be reported for associated parts: **Part Name**, **Quantity** and **Description**.



Rotate Around Centre

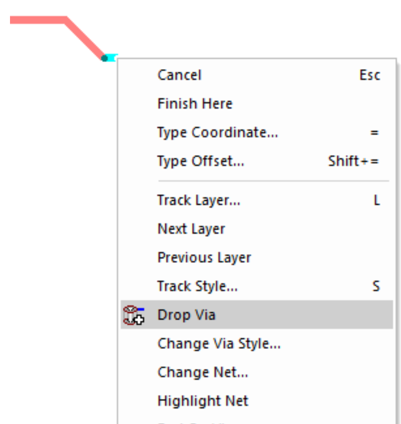
A new context menu command, **Rotate Around Centre**, has been added to the **Rotate** section of the context menu when placing or when using select mode with an item selected.



The command acts as a tick box, when enabled, rotation of the selected item will rotate around the centre point of the selection (rather than the origin).

Drop Via Option Added to Add/Edit Track

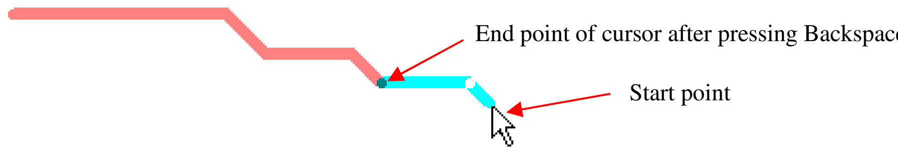
When **adding** or **editing** a **track**, you can now right click and select **Drop Via** from the context menu.



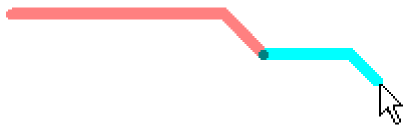
Using this feature will add a via at the location of your cursor, and continue with the track on the same layer, no layer change is made. This is a useful option if you wish to stitch in vias for shielding purposes or for enhancing power distribution paths.

Edit Track – Backspace moves the cursor to the end segment

If **backspace** pressed it will still go back to the previous segment corner (as before), but now the cursor will move to the **start of the previous segment**.



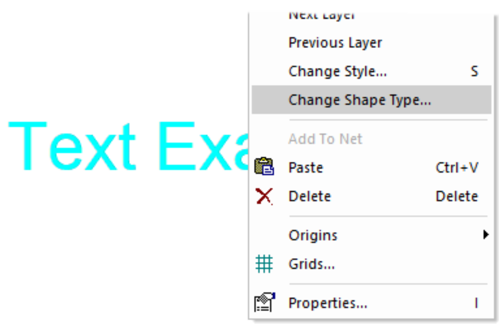
While editing, pressing the backspace key removes the track segments and moves the cursor back to the last segment corner:



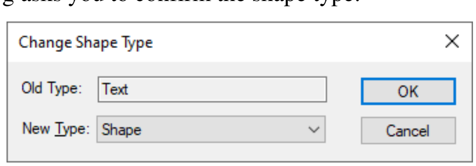
Change Text to Shapes

You can now convert True Type **text** into **shapes**, (Note, this doesn't work with the system 'stick' font). You might do this if you wish to mill the text in copper if using a PCB prototyping machine for example.

To do this, select a piece of text that uses a true type font. Right Click, and from the context menu, select **Change Shape Type**. A dialog will appear to request which shape type to use. When **OK** is pressed, the text will be converted into that type.



A dialog asks you to confirm the shape type:



The text is now individual shapes that can be edited or deleted when selected:

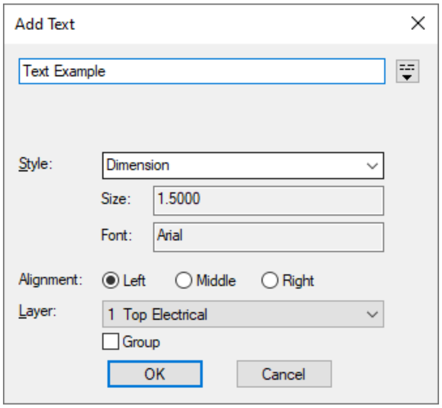
Add Text Cutout

There is a new **Cutout** option on the **Add** menu for **Cutout Text**. This allows you to create a cutout from any true type font text.

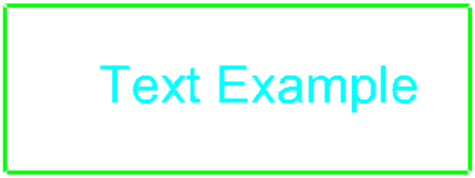
To use this option, select a **shape** and go to the **Add** menu and **Cutout Text**. You could also start with an unselected shape and use **Add Cutout Text**, and then select the **shape** to apply it to.

Select a target shape:

This will prompt you with the **Add Text** dialog. You can specify the text you want to add and whether you want the final shape to be **grouped** together. Grouping it means you can move the text along with the shape if you wish to reposition it.

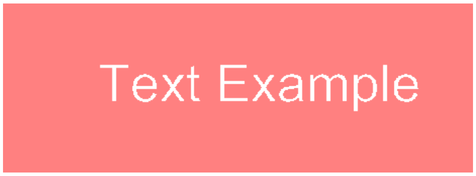


Once confirmed using the **OK** button, select the position that you want the cutout text to be placed



Once released, the shape now includes the text:

After it is deselected, the shape and text look like this:

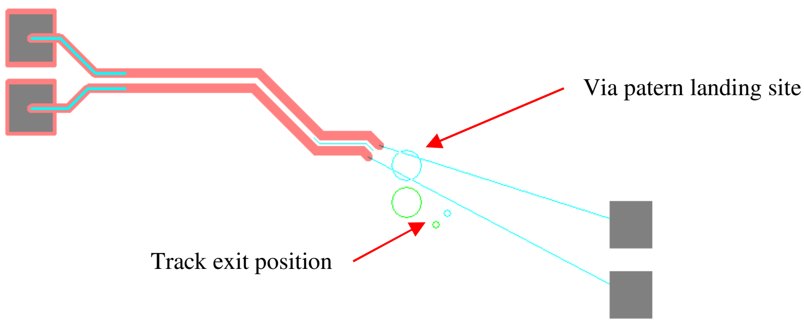


Copy / Paste multiple document shapes between design types

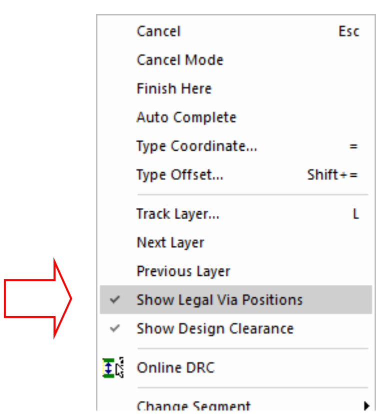
You can now copy and paste multiple selected shapes between design types. For example, a Schematic shape defining your company drawing border can be copied and pasted into your PCB design (to create a PCB drawing border). The shape will be added using the **Styles** and **Layers** from **Shape** defaults under the **Settings** menu in the target design if copying to a PCB design.

Add Diff Pair – Show Legal Via Positions

Whilst adding the paired track segments you can use **Show Legal Via Positions** to display circles where the vias will land if you decide to change layer. It also shows smaller circles where the tracks will start on the next layer after adding the vias and the track transition pattern.



This option is available to toggle on or off on the context menu when in Diff Pair mode:



If this option is on and **Online DRC** is on, and the vias would not be legal due to obstacles on the other side, the circles will not be shown.

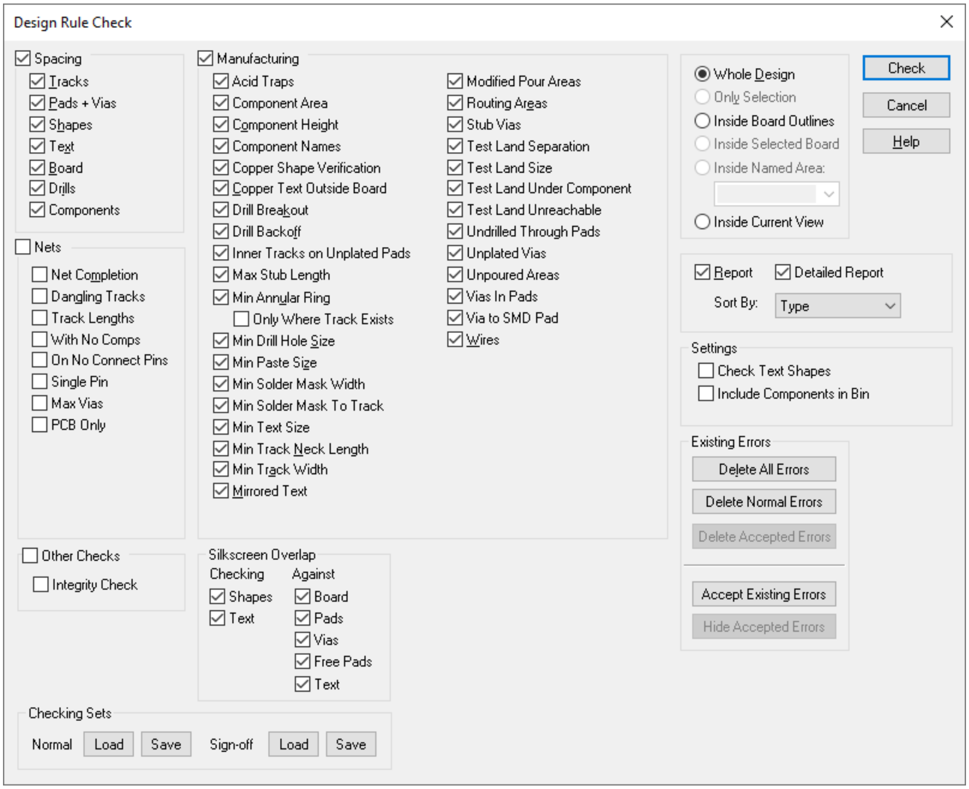
Low Level Geometry Overhaul

As information for you, and with no visible dialog or functional changes, the underlying geometry used within the Copper Pouring algorithm has been updated and improved. One of the main benefits to you is the stability when running this routine. You will also see a greater accuracy of pouring into areas where you might have expected it to pour but where it hadn't previously. This is the main area where the new code impacts design, but it is also used in other lesser visible areas of the product, such as DRC.

Design Rules Checking (DRC)

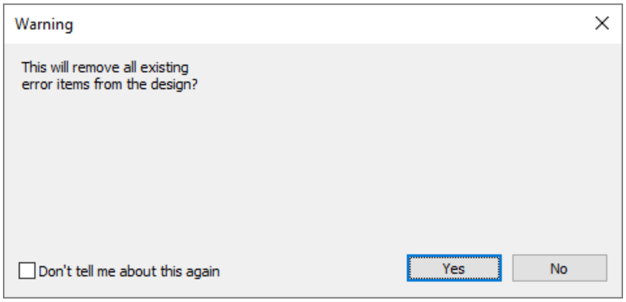
DRC Dialog Reorganised

The DRC dialog has been reworked to simplify and organise the existing controls.



Remove Errors Warning

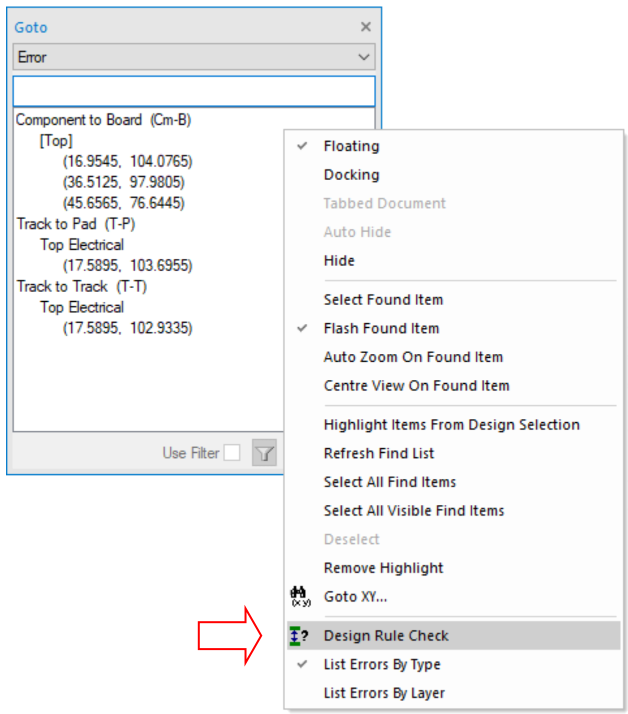
In the **DRC** dialog, when removing existing errors (any of the three **Delete Errors** buttons), a warning will prompt you instead of the normal message box. This allows you to turn off this message. The message can be restored using the **Preferences** dialog and **Warnings** page.



Goto Bar

Shortcut to DRC Option from Goto Bar

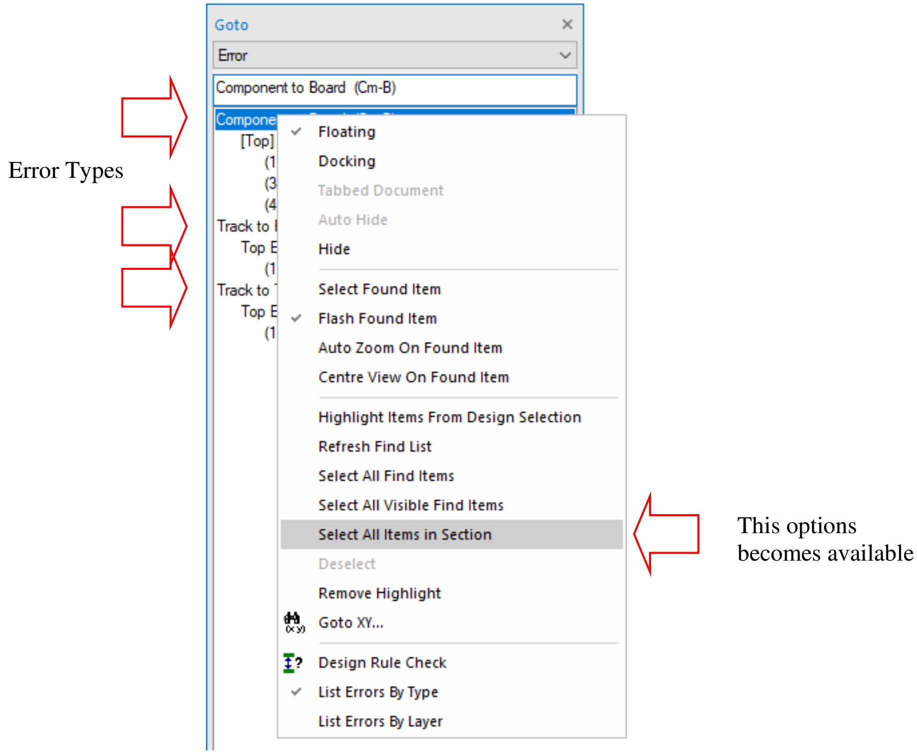
In the **Goto** dockbar under **Errors**, there is a new context menu command, **Design Rule Check**. This provides you with quick access to the Design Rule Check dialog.



Select All Items in Section Context Option for Errors

In the **Goto** – **Errors** dockbar, there is a new context menu command **Select All Items in Section**.

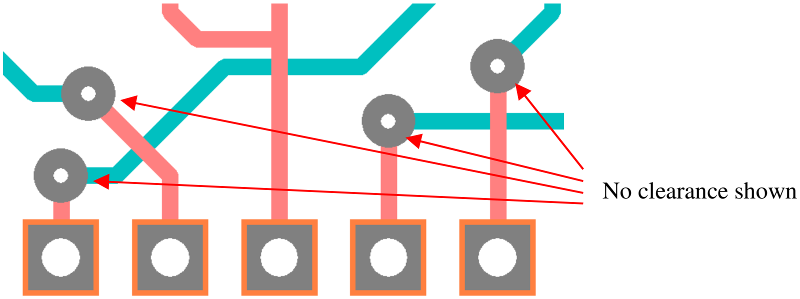
This command is available when a section (such as an error type or layer, not an individual error) is selected, for example the Error Type being checked. This command will select all the errors within this category (Error Types).



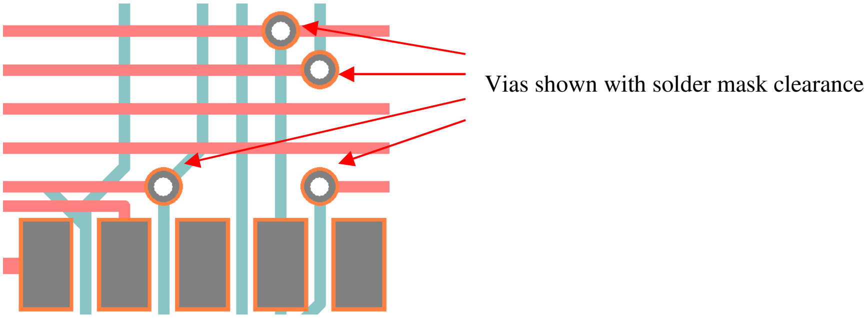
Via Oversize Displayed

Via oversize (when defined) is now shown and coloured in the **Solder Mask** colour using the same colour as pads. This prevents issues such as the vias appearing bigger than their actual size when plotted.

In previous versions, vias were shown like this, with **no** solder mask clearance displayed:



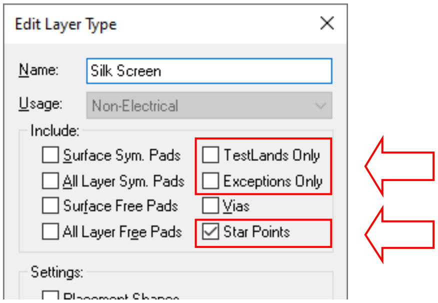
Vias will now be displayed showing the solder mask clearance:



Layer Type Filters added for Testlands, Layer Exceptions and Star Points

Options have been added to the **Layer Type** dialog within the **Design Technology** to enable you to filter the pads that appear on layers in a particular class meeting certain conditions.

New conditional switches have been added for **Testlands**, Layer **Exceptions** and **Star Points**.



These options work in conjunction with the pad filters (Surface Sym. Pads etc.).

Testlands Only and **Exceptions Only** will only affect pads that would appear anyway due to the selections made with the pad 'type' switches.

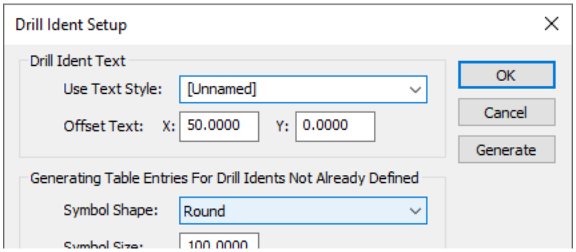
Vias is existing functionality**.**

Star Points is the only switch which will force pads to display (or not) regardless of other switches.

Drill Ident Setup Dialog – Generate from design and improvements

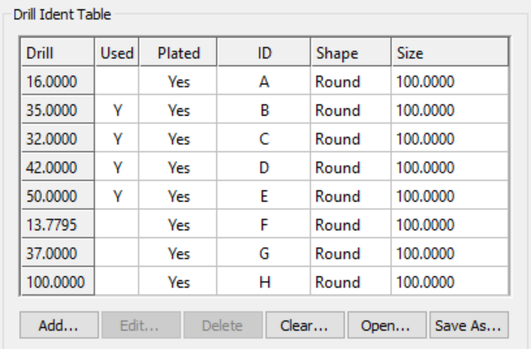
The drill ident setup dialog has been improved in order to facilitate and manage your drill idents more efficiently. This is available under **Output**, **Plotting and Printing, Options button**, **Setup Sizes and Symbols button** under **Drill Ident Drawing.**

A new **Generate** button has been added. When pressed, it will replace the current items in the grid and add all the drill sizes in the current design using the default drill ident values. If you subsequently add another pad or component that contains a drill size not already in the list, it will automatically be added as normal when the drill table is updated or when the plotting dialog is entered.



Drill Ident Setup Dialog – Used column

The **drill ident setup** dialog now shows you which drill idents are being using by the current design. In the **Drill Itent Table** on the dialog, there is a new column, **Used**. For each drill size defined in the grid, a **Y** will appear in the used column indicating that this drill size is used in the design.


Drill Ident Table

Drill	Used	Plated	ID	Shape	Size
16.0000		Yes	A	Round	100.0000
35.0000	Y	Yes	B	Round	100.0000
32.0000	Y	Yes	C	Round	100.0000
42.0000	Y	Yes	D	Round	100.0000
50.0000	Y	Yes	E	Round	100.0000
13.7795		Yes	F	Round	100.0000
37.0000		Yes	G	Round	100.0000
100.0000		Yes	H	Round	100.0000

Add... Edit... Delete Clear... Open... Save As...

Drill Ident Drawing – Allow 0 size and/or no ident plots

Plotting **drill ident drawings** can now handle 0 (zero) size and empty drill idents.

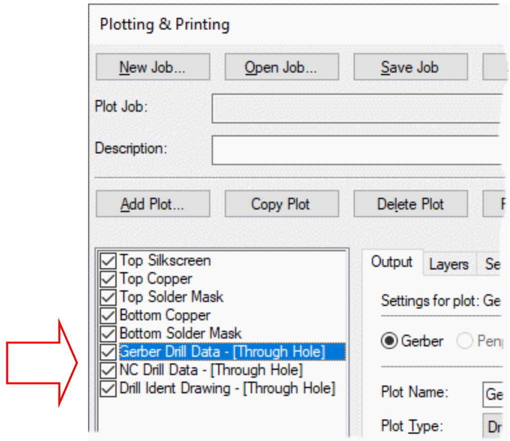
A 0-size drill ident (meaning an ident symbol of size 0) is used to indicate that the symbol should not be plotted.

An empty drill ident string is used to indicate that the drill ident text should not be plotted.

These new settings, together or separately, allow you more control on what drill ident information will be plotted without the need for deleting idents (grid rows) on the **Drill Ident Setup** dialog.

Auto Generated Gerber Drill Plot Removed

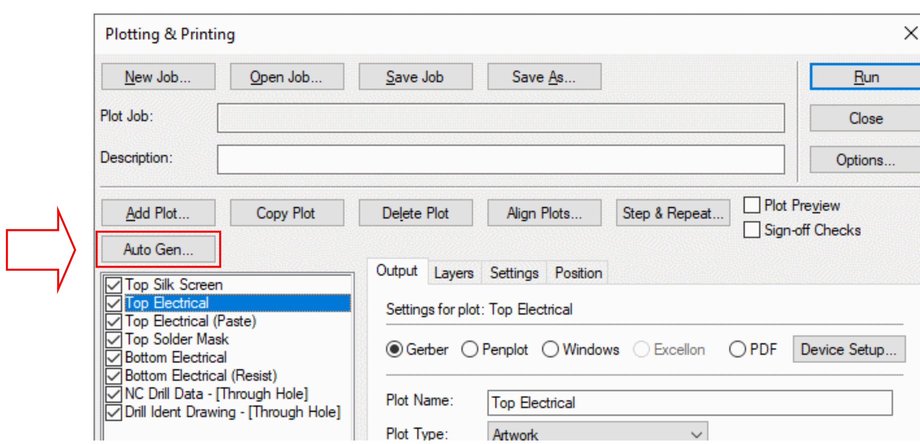
In previous versions, auto generated plots in the plotting dialog used to include a **Gerber Drill** plot. This has led to confusion and is **not** generally required for manufacturing. This no longer occurs for auto generated plots in Version 28. However, if you already have this plot type, it can be removed from your plotting set by simply selecting it and using the **Delete Plot** option.



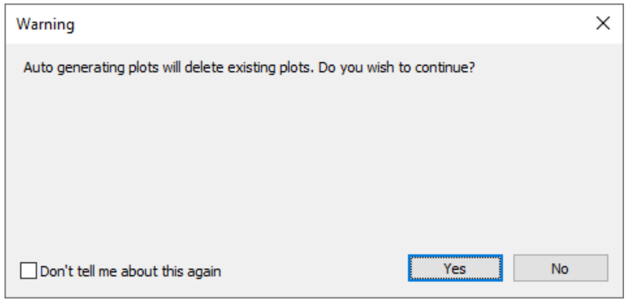
If you still require this plot, it can be generated manually using the **Add Plot** option.

Plotting – Automatic Generation of All Plots option

There is a new button on the **Plotting and Printing** dialog - **Auto Gen…**. This will generate a set of plots for every Electrical and Non-electrical layer in your design, plus drilling and Drill Ident layers. Electrical and Non-electrical plots are generated in Gerber format, with NC Drill files generated in Excellon format. The Drill Ident plot is generated in Gerber format also but can be reconfigured if required.

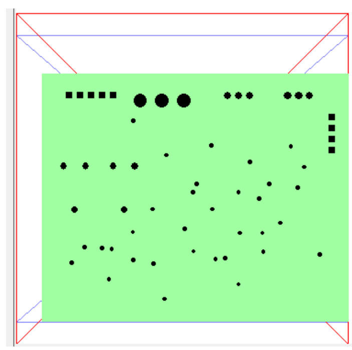


When selected, a warning will prompt you to confirm the removal of all existing plots, if OK is pressed, new plots will be automatically generated.



Plot Preview – Shows Pads-Only Normal Vias and Testland Vias

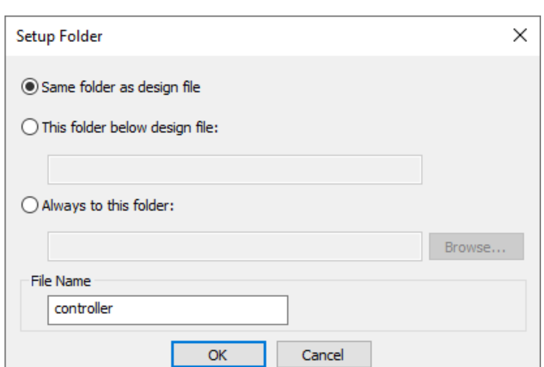
When previewing a Pads-Only plot in the plotting dialog, such as a Solder Mask plot, the **Plot Preview** now shows and/or hides normal vias and Testland vias depending on the settings you've selected for the **Layer Type**.



ODB++ - Dialog Folder Customisation

A new button has been added to the ODB++, **Setup Folder**. Use this to define the folder path when exporting non-compressed ODB++ files.

Clicking this button will open the **Setup Folder** dialog from where you can choose various location settings for the output folder.



Same folder as design file - This option saves the file into the same folder as the design file.

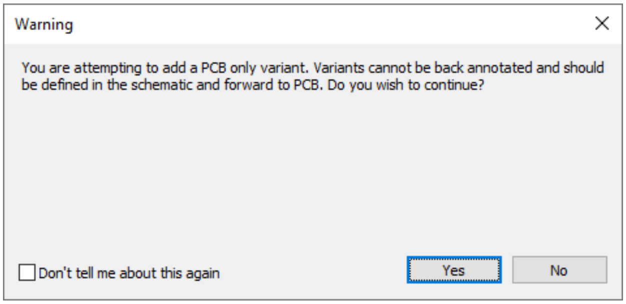
This folder below the design file - This option saves the file into a named folder below the design file. If the folder doesn't exist, the program will create it for you when the export is generated.

Always to this folder - This option saves the file into a named folder, regardless of where the design file is. The **Browse** button allows you to locate the folder.

File Name - Additionally, if you have the **Compressed** option ticked before entering the **Setup Folder** dialog, you will be given another option to specify the **File Name** of the compressed files.

Variants – Warning when creating new PCB only Variants

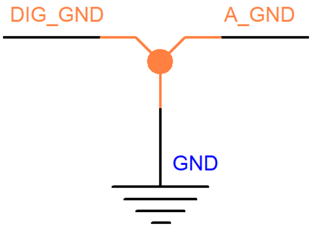
When attempting to create a new variant in a **PCB design** using the **Design Technology**, **Variants** option, you will be prompted with a warning describing the possible problems in doing this. This helps to prevent issues with back annotation failing due to the design having **PCB only variants**.



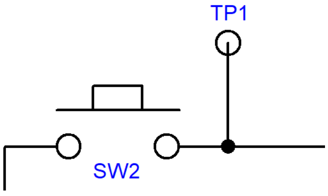
New & Modified Library Content

Changes have been made to the standard library installation shipped with Easy-PC Version 28:

Two Star Point Component examples have been added, along with associated Schematic symbol and footprint. Examples are provided for a three-point star and 8-point star.



A surface mount Testland example has been added to the library. The Component is available alone with the associated Schematic symbol and footprint.



The PCB symbol will exist as part of the track (unless you specifically require it to be visible). We have provided a PCB symbol that is 10 Thou (0.250mm) but it can be edited to suit your requirements.