Easy-PC

V21.0 Supplement

Copyright © 1998-2017 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: info@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 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: 11-July-2017 Issue 1

Warranty and Software Licence Agreement

WestDev Ltd and Number One Systems

This document contains the WESTDEV LTD SOFTWARE LICENSE AGREEMENT which will govern the use of the Westdev and/or Number One Systems' products supplied with it.
You agree to the terms of this agreement by the act of opening the sealed package which contains the storage media on which the software is recorded. Do not open the sealed package without first reading, understanding and agreeing the terms and conditions of this agreement. If you do not agree to the license conditions you may return the SOFTWARE for a full refund before opening the sealed package.

1. LICENSE AGREEMENT

Ownership of the WESTDEV / NUMBER ONE SYSTEMS' Software Package and all its constituent parts supplied by Westdev Ltd ("LICENSED SOFTWARE") is not transferred to the Customer. Westdev Ltd hereby grants the Customer a non-exclusive license to use the LICENSED SOFTWARE on a single computer workstation. Each computer workstation, even on a network, must have its own separately LICENSED SOFTWARE.

2. OWNERSHIP OF SOFTWARE

As the LICENSEE, you own the magnetic, optical or other physical media on which the LICENSED SOFTWARE is originally or subsequently recorded or fixed, but an express condition of this License is that Westdev Ltd and/or the copyright owner retains title and ownership of the LICENSED SOFTWARE recorded on the original media and all subsequent copies of the SOFTWARE, regardless of the form or media in or on which the original and other copies may exist. This License is not a sale of the original SOFTWARE or any copy.

3. COPY RESTRICTIONS

This LICENSED SOFTWARE and all accompanying written materials and all other constituent parts are the subject of copyright. Unauthorised copying of the LICENSED SOFTWARE, including software which has been modified, merged, or included with other software, or of the written materials is expressly forbidden. You may be held legally responsible for any copyright infringement which is caused or encouraged by your failure to abide by the terms of the License.

4. TRANSFER OF LICENSED SOFTWARE

The Customer may transfer the LICENSED SOFTWARE provided that (i) this Software License Agreement is transferred with the LICENSED SOFTWARE, (ii) the transferee fully accepts the terms and conditions of this Agreement, and (iii) all complete or partial copies of the LICENSED SOFTWARE, including copies on data storage devices are also transferred (or destroyed).

5. MODIFICATION

You may not otherwise modify, alter, adapt, merge, de-compile or reverse-engineer the LICENSED SOFTWARE, and you may not remove or obscure any Westdev Ltd and/or Number One Systems' Copyright or Trademark notices. You must use all reasonable efforts to protect the LICENSED SOFTWARE, diskettes, and documentation from unauthorised use, reproduction, distribution or publication, or otherwise in violation of applicable law. Please contact our Customer Service department if you become aware of violations of Westdev Ltd's Copyright.

6. TERMINATION

The License is effective until terminated. You may terminate it at any time by destroying the LICENSED SOFTWARE and all complete or partial copies thereof. It will also terminate if you fail to comply with any term or condition of this Agreement. You agree upon such termination to destroy the LICENSED SOFTWARE and all complete or partial copies thereof.

7. WARRANTIES

No warranties are expressed or implied with respect to the LICENSED SOFTWARE described, its quality, performance, accuracy or suitability for any purpose. In no circumstances will the copyright holder be liable for direct, indirect, incidental or consequential damages resulting from the use of this SOFTWARE.

8. OTHER SOFTWARE

The supplied disks may contain software whose copyright is owned by third parties. This license agreement shall apply equally to this software except where the terms of this agreement are specifically modified by the copyright holder(s). Such modification(s) will be included in a machine readable file on the supplied disks.

9. GOVERNING LAW

The Governing Law of this Agreement shall be that of England.
Westdev Ltd will reward anyone giving information leading to successful prosecution regarding breach of copyright.

Contents

CONTENTS **4**
CHAPTER 1. GETTING STARTED **5**
Installation 5
Running Easy-PC 21.0 6
CHAPTER 2. NEW FEATURES IN EASY-PC V21 ..... **7**
Introduction 7
Dynamic Align – Interactive Guidelines & Snapping 7
New Dockable Properties Bar 8
Library Manager Additions 9
List of Values 9
Go To Symbol and Edit Symbol Context menu 9
Drag-and-drop Values To Properties 10
3D Library Report (PCB) 11
Pad Style Exceptions Report (PCB) 12
Zoom Level Increase 12
Fix and Unfix Dialog (PCB) 12
Design Tooltips Extended to include Net Class Name 13
Plotting - Excellon Output Leading/Trailing Zeroes (PCB) 13
IDF Output Enhancements (PCB) 14
DesignSpark Mechanical Output 14
User-defined Units 14
Reverse Rotation for Mirrored Components 15
Spaces in File Names 15
Include Vias in Output 15
Include Unplaced Components 15
File Naming and Locations 16
IPC-356 report – Neutral Net Names (PCB) 16
Plotting - Break Silk Screen (PCB) 17
Layers – Delete used (PCB) 17
BOM Composer Item Quantity 18
PDF Plot - Embed Fonts 18
Preferences – Workbook tabs Setting 19
Drawing – Display Hatched Drill Holes (PCB) 19
Change Shape Type (PCB) 20
Layers Bar Changes (PCB) 21
Defining Current Electrical and Documentation Layers 21
Showing and Hiding Powerplane Layers 22
Copper Pour Keepout Areas (PCB) 22
Shortcuts for Hollow Display of Items 23
Shortcuts for Text justification 24
Toggle units in Design Technology dialog 24
Unroute Nets - Ignore Fixed Items (PCB) 25
Display of Net Names (SCM) 25
Add Component – Values Preview 26

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.

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

Installation

Installation is via the *autorun* setup. Insert the CD-ROM into your CD-ROM drive 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.

If you are installing software downloaded from our web site rather than from CD, then you will simply need to run the installation program once it has finished downloading.

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.

With the installer running, once the **Welcome** screen is displayed, double-click on the **Install Easy-PC - Version 21.0** 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.

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 21.0

As with all **Easy-PC** programs, an icon will appear in the **Number One Systems** folder, or 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 7, 8, 8.1 or 10.

Chapter 2. New Features in Easy-PC V21

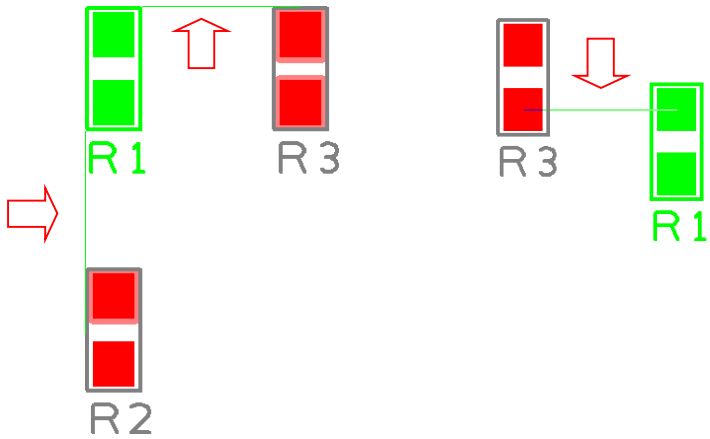
Introduction

All features are categorised as being applicable for either products (unmarked), or (SCM) specific or (PCB) specific.

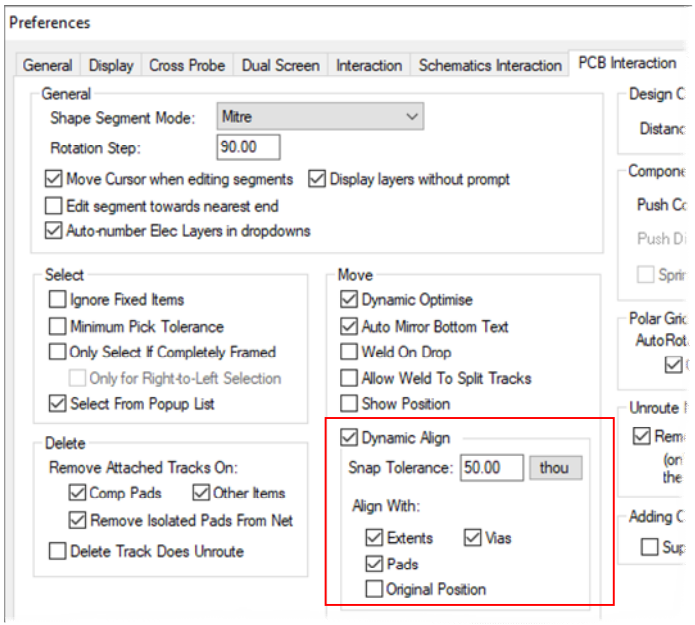
Dynamic Align – Interactive Guidelines & Snapping

Dynamic Align is a new concept of informing you when an item (Component, Pad, Via or Text) is in-line with another legal item that can snapped to, not all items can be snapped to each other.

During move, the item will 'snap' in-line and a graphical line(s) will be drawn in your selection colour to indicate when either a portion of the item extents, Pad, Via or Text is in-line with another similar item.



Options are available on the **Settings**, **Preferences**, **Schematic Interaction**/**PCB Interaction**, **Dynamic Align**. The settings are dependent on the design environment you are in, therefore allowing you to control the items that will be aligned to and control the **Snap Tolerance** in both the SCM and PCB. This option can also be enabled from the **Context** menu, when moving an item, right click, and select **Use Dynamic Align**.



When aligning an item, you can choose to enable the **Dynamic Align** option to align with an item's Extents, a Pad, Via, the item's original position or Text (only text items will align with another text item).

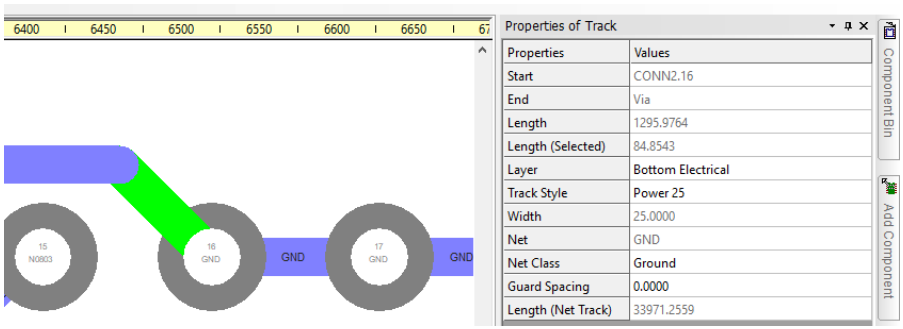
You are also provided with the ability to control the **Snap Tolerance** at which an item will snap to another item, this is the tolerance from the horizontal or vertical alignment of an item in your design.

Under the **Preferences** dialog, there are also options to set which items can be **Aligned With**, such as **Extents**, **Pads**, **Vias** and **Original Position**. You may wish to disable some of these if they are outside your area of interest.

New Dockable Properties Bar

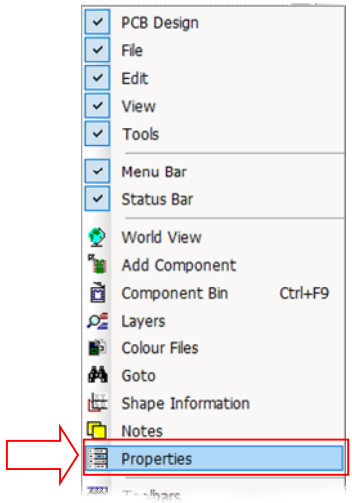
A dockable **Properties** bar has been introduced to allow you to see the properties of an item or multiple items in a modeless window. Many of the features available to you from within the existing **Properties** dialog are available in this dockable option, helping you to view and edit values for items without accessing a new dialog. Note that not all the properties are available.

The dockable **Properties** bar also provides the ability to show only the properties common to all the items in the selection.



To enable the **Properties Bar**, click on the **View** menu, **Dockable Bars**, and click **Properties Bar**. Alternatively, right click on the right-hand bar and select **Properties** as shown in the illustration to the right.

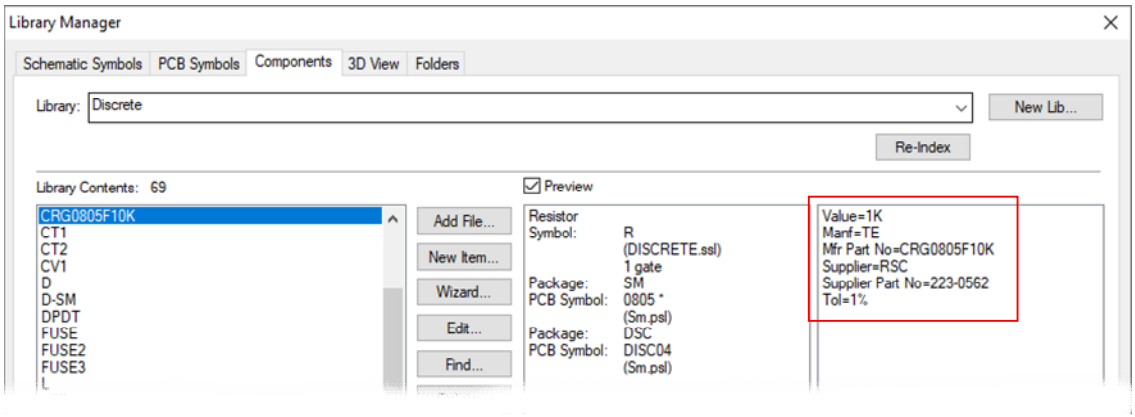
Because this is a floating bar, it can be undocked once selected for use.



Library Manager Additions

List of Values

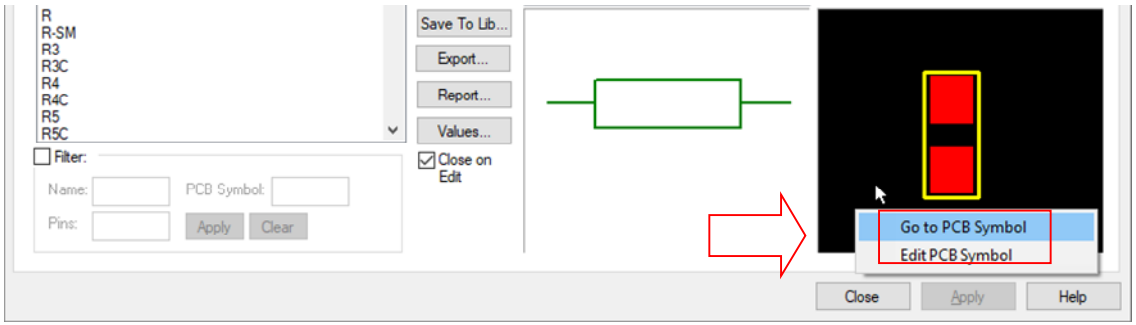
On the **Components** page of the **Library Manager** dialog, a new list of the **Component Values** is displayed in a list box to the right of the existing list of gate and footprint details.



Go To Symbol and Edit Symbol Context menu

On the **Components** page of the **Library Manager**, you can right-click in one of the symbol preview windows and choose **Go to Symbol** or **Edit Symbol** from the context menu. These new commands will jump to the corresponding tab on the library manager dialog and preselect the symbol that is shown in the preview, or it will open the corresponding symbol for edit without having to go and find it in the relevant symbol tab.

Note that when **Edit Symbol** is selected, the **Library Manager** will still be open but the symbol will be ready for editing in the window behind it.



Right clicking in the **Preview Symbol** windows will display a small context menu from which to choose the option required.

Drag-and-drop Values To Properties

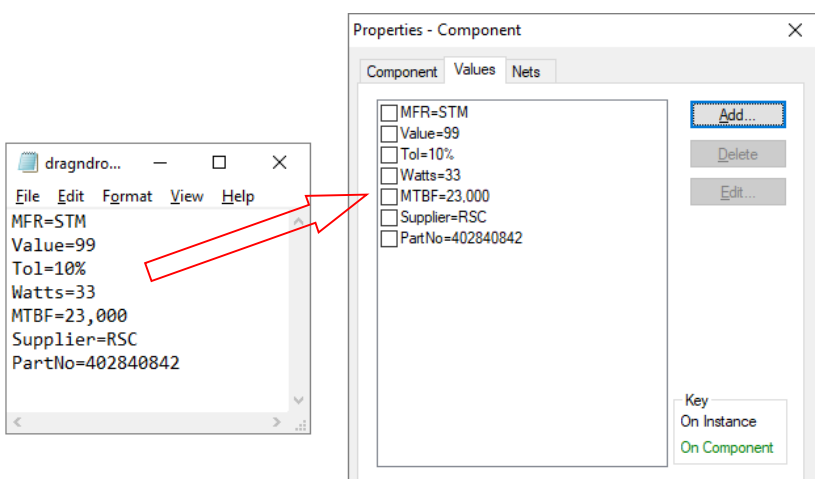
You can now drag-and-drop one or more text files, or a block of text from a text application such as Word or Notepad2, and drop it onto Easy-PC to add Values to Components. This feature can be used from the **Values** tab on the **Properties** dialog for **Components** and the **Values** page in the **Component Editor** (available from **Libraries**).

This facility is hard-wired to expect the attribute text in **name=value** form, using = as the separator. Any line of text that does not fit with this form will be ignored. Any name that already exists, will have the value replaced without prompt or warning. The syntax will accept the name=value format with or without spaces but nothing else.

Using Drag & Drop Values onto a component

There are two methods to drag and drop Values onto a Component:

1. Using the Windows explorer, you can select a text file containing legally formatted values text and drop it onto the **Values** tab of either the **Component edit** dialog or the **Component Properties** dialog, or;
2. Select portions of the required text from the file, again, formatted using the expected syntax and drag it out of the text file and onto the Values page for the desired item, such as this below:

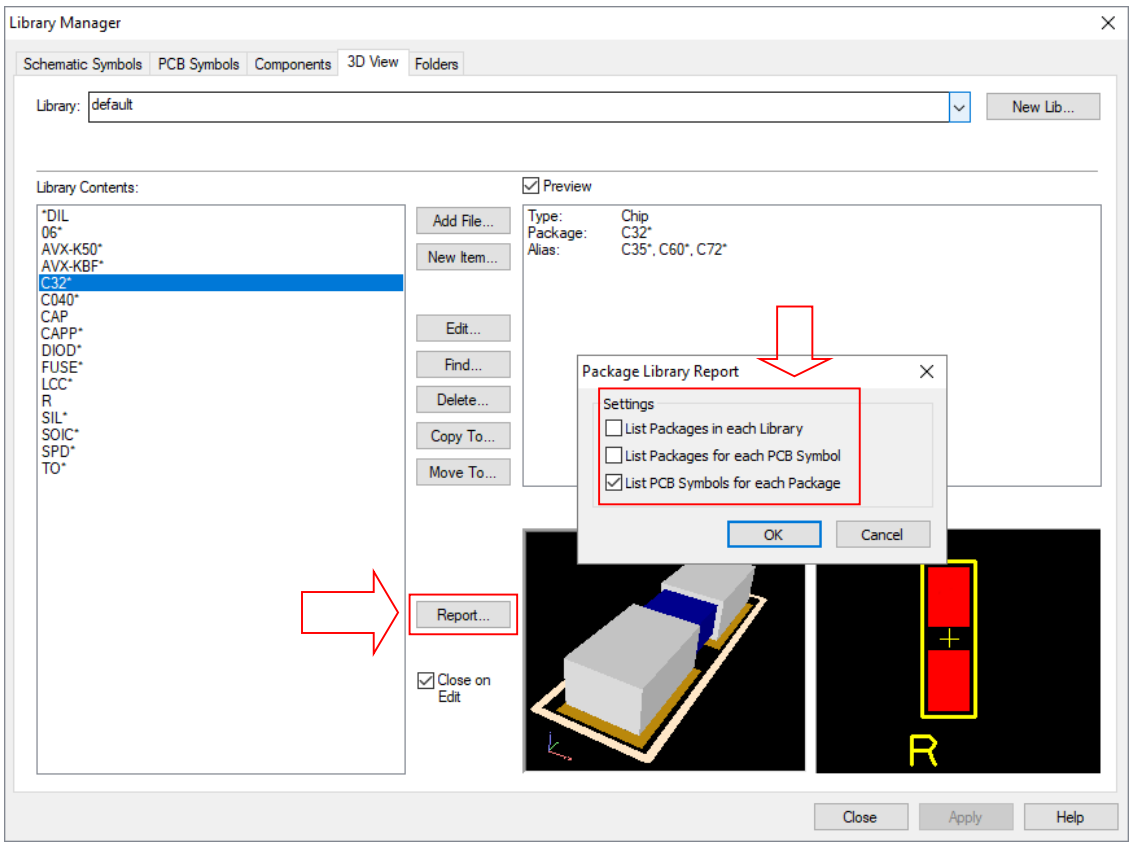


Note: Copy Paste does not work for this function, the text must first be selected, then **dragged** to the appropriate Easy-PC dialog.

Also note that the standard Windows Notepad program will **not** allow you to drag and drop individual text items due to the technical limitation of this program. Other text programs such as **Word** or **Notepad2** etc. do however allow this functionality.

3D Library Report (PCB)

There is now an extra section in the **3D libraries report** that lists the mappings from a 3D package to PCB symbol. The **Report** button on the **Library Manager** dialog now displays a **Settings** dialog that allows you to choose which sections to include in the report.



List Packages in each Library - this will report on the contents of each library, showing the package names and aliases present in each of your currently enabled 3D package libraries.

List Packages for each PCB Symbol - this will report how each PCB symbol is mapped (or not) to a 3D package.

List PCB Symbols for each Package - this will report the names of all PCB symbols that would use each of the 3D packages.

Pad Style Exceptions Report (PCB)

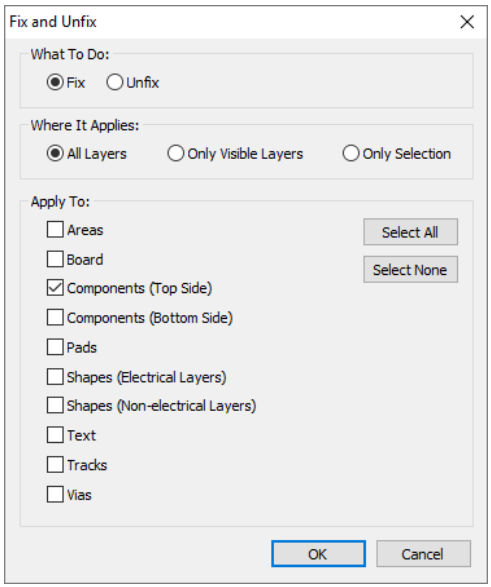
A new report, **Pad Style Exceptions Report**, has been added to the **Output** menu, **Reports** dialog within the Built-in Reports section. This will report all pad style exceptions in the current PCB design. Pad exceptions can be used when you wish to alter the style or size of a Pad by layer(s) within the Technology dialog.

Zoom Level Increase

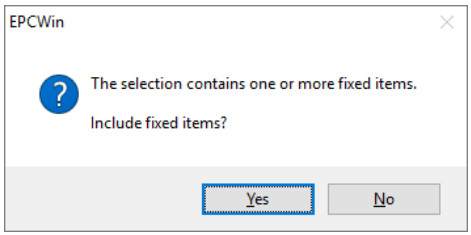
The maximum zoom level has been increased by a factor of 8. This will allow you to zoom in further to inspect critical areas/items in your designs.

Fix and Unfix Dialog (PCB)

There is a new dialog to **Fix** and **Unfix** items. Within a PCB design, this is available on the **Edit** menu. This allows you to fix or unfix multiple items by **layer**, **on only visible layers** or on **selection** of an item. The selection is then applied to an item category selected from the list. This can be items of different types, for example, **Areas, Components** and **Pads**.

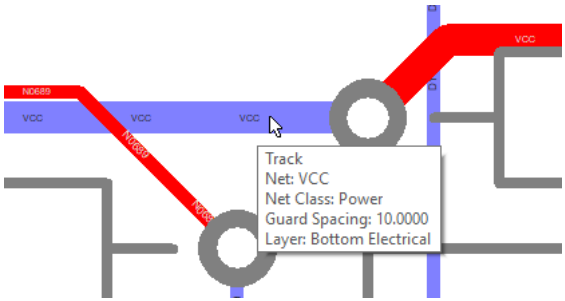


With an item is fixed, a warning dialog will be displayed if you try to move, edit or delete it.



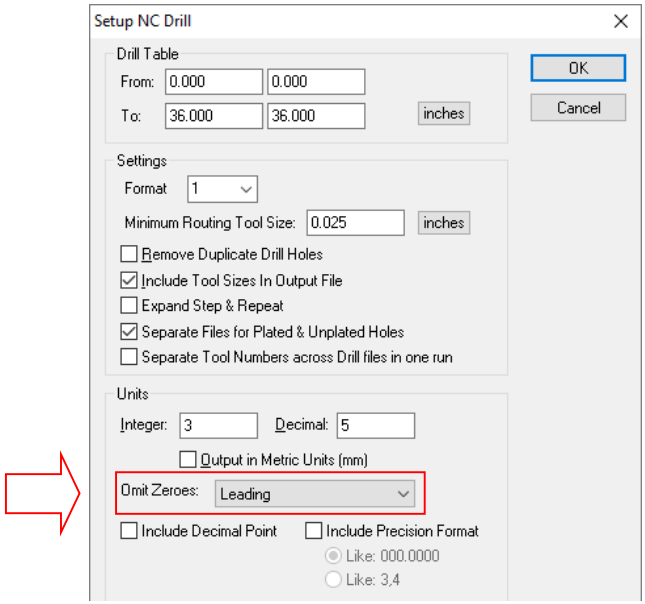
Design Tooltips Extended to include Net Class Name

The **Net** field on design tooltips now includes **Net Class Name** (if defined) and **Net Guard Spacing** (again, if defined). To enable the **Net** field, go to the **Settings** menu, **Preferences** dialog and **Display** then **Tooltips on Design Items** – check the **Net** box to enable it.



Plotting - Excellon Output Leading/Trailing Zeroes (PCB)

The new control within the **Plotting and Printing** option on the **Drill Data plot** and **Device Setup** under **NC Drill** dialog allows you to specify the setting for formatting coordinates. This can be set to **None** (the default), **Leading** or **Trailing**. Previously, coordinates were always output in full (Omit Zeroes = None) format.



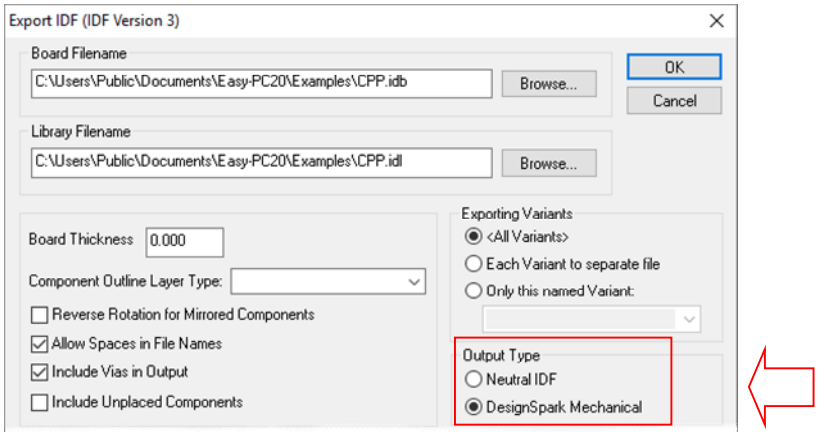
IDF Output Enhancements (PCB)

The IDF output dialog has been enhanced to now include the following features:

DesignSpark Mechanical Output

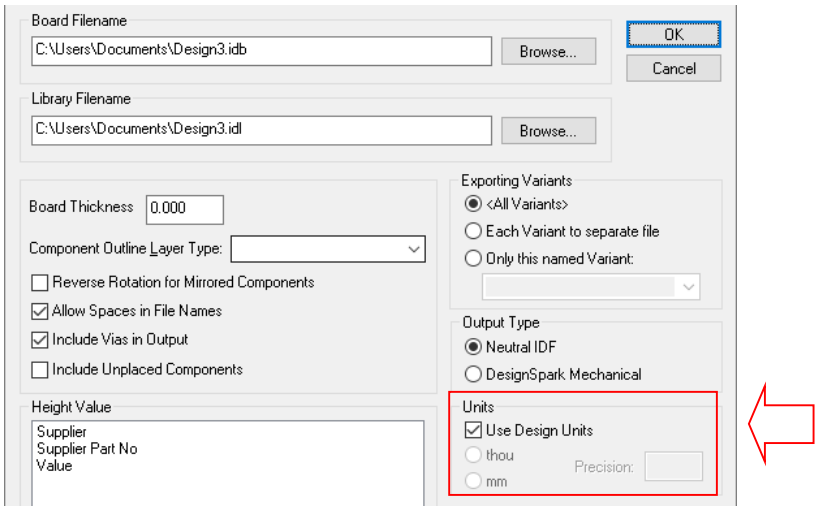
A new radio button has been introduced to select between **Neutral IDF** format and a specific format of IDF for the **DesignSpark Mechanical** product.

The DesignSpark Mechanical output is formatted to contain specific commands for this product to allow for smooth transfer between Easy-PC and DesignSpark Mechanical.



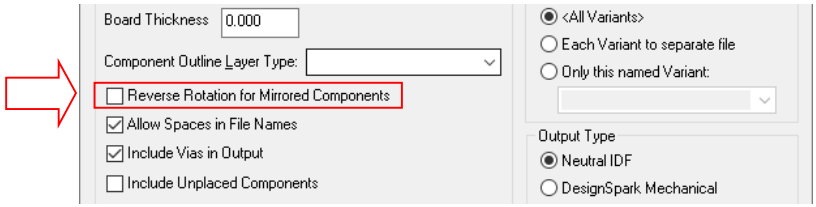
User-defined Units

A new **Units** section within the **IDF Export** dialog enables you to choose whether the units exported to the IDF file are taken as your **current design units** (defined in the **Settings** menu and **Units**), selecting the check box, or unchecking it and selecting one of the radio buttons to choose between **thou** or **mm**. You may also choose the **Precision** of the units with one of these radio buttons selected.



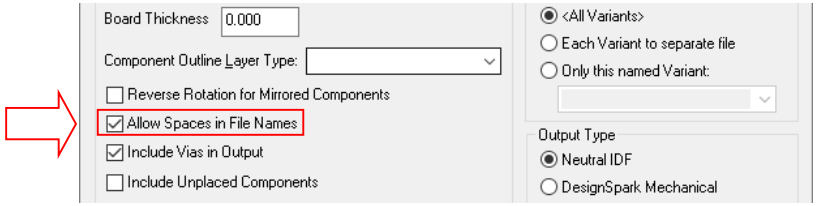
Reverse Rotation for Mirrored Components

Reverse Rotation of Mirrored Components, many software packages interpret the orientation of components on the back of the board as the angle you would see when looking at the board from that side. In this case, check this box (the default setting) to have component rotations reversed for mirrored components. If your software package does not interpret rotation of mirrored components in that way, uncheck the box to have component rotations output as defined in the design regardless of their mirrored state.



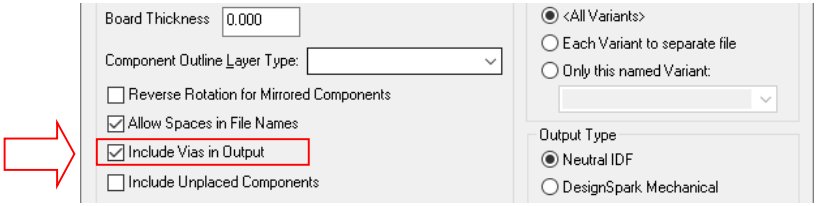
Spaces in File Names

The **Allow Spaces in File Names** check box lets you control the file names used for output, to cater for some applications which do/do not require spaces in the naming.



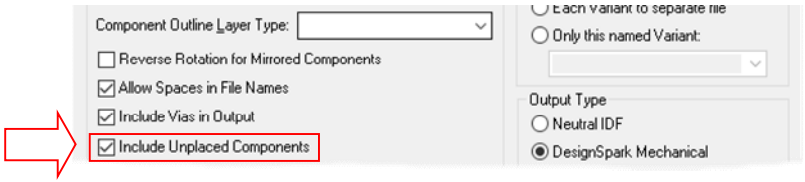
Include Vias in Output

By unchecking the **Include Vias in Output** check box, you can suppress the drilled holes for vias from the output file.



Include Unplaced Components

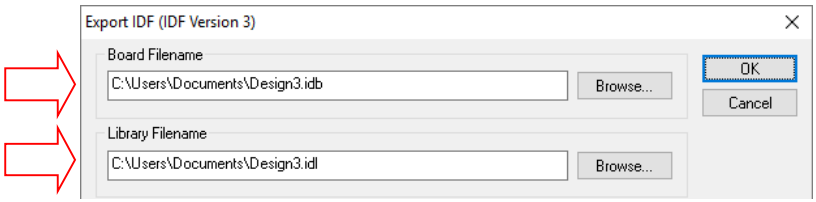
Select the **Include Unplaced Components** check box to include components from the component bin in the IDF output. Leave it unchecked to ignore them.



File Naming and Locations

Three small changes have also been made to the IDF Export dialog:

- Browsing to the **Board** or **Library** file locations will update the other control to match so that the locations are consistent and also to save you browsing to one when it has already been located in the other.

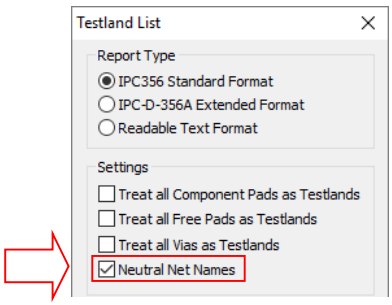


- **Path names** to output files are remembered for the next time you are using this dialog, provided it is the same design as last time.
- Changing the **file extensions** to something other than the standard ,idb or .idl will be remembered for future use of this option.

IPC-356 report – Neutral Net Names (PCB)

There is now an extra check box on the **IPC-356 report** dialog to allow for the output of **Neutral Net Names** in the IPC-356 and IPC-356A report formats.

Neutral Net Names cause all the net names in the output file to be named numerically (N1, N2, N3, etc.). This enables you to hide your design intent from recipients of the net list file by hiding your meaningful net names. It can still be used for tasks such as checking connectivity with a set of Gerber files though.

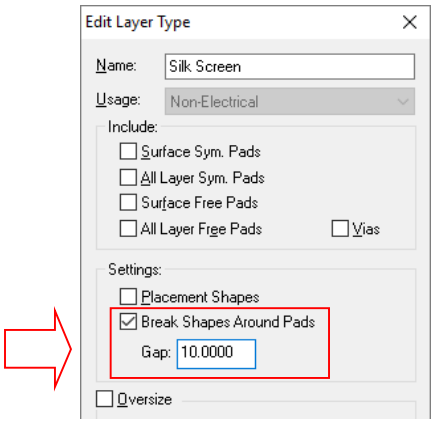


Plotting - Break Silk Screen (PCB)

New settings on a (non-electrical) **Layer Type** allow you to **Break Shapes Around Pads** and specify the **Gap** between the shape and the pad/via.

When plotting shapes on non-electrical layers such as Silkscreen will enable automatic breaks in the shape when the shape falls within the pad or via shape. Silkscreen plotted over pads or drill holes can cause poor coverage of solder on that pad and lead to electrical failure.

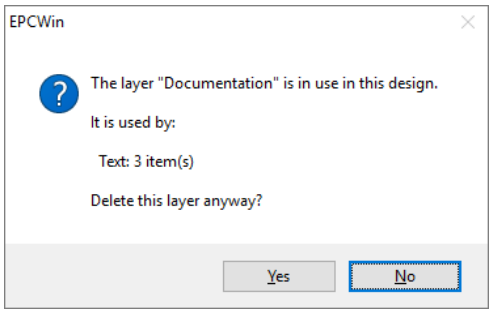
Changing the **Gap** value will enable you to increase or decrease the distance between the pad item and the shape.



Below shows an example of a silkscreen shape without and with a break:

Layers – Delete used (PCB)

The **Delete** button on the **Layers** page in **Design Technology** will allow you to delete a non-electrical layer even if items exist on the layer. It will prompt you with a message showing how many of each type of item would be removed, before actually removing them. This even allows removal of items such as shapes from a layer in a footprint in a design, and is therefore an easy way to clean up data imported from other systems which has superfluous information in it.

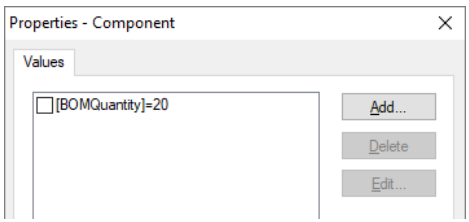


BOM Composer Item Quantity

From within **Component Properties**, a new 'built in' **value name** of **[BomQuantity]** is available to be added to Components.

This Value can be used to specify a number which is used for the quantity of this item to appear on the **BOM Composer** output. The default case with this value not set on a component is 1. This value can be used, for example, to specify a particular number of additional hardware elements (e.g. screws or washers) that have to be included for that component in the BOM without having to add multiple instances of that component to the design.

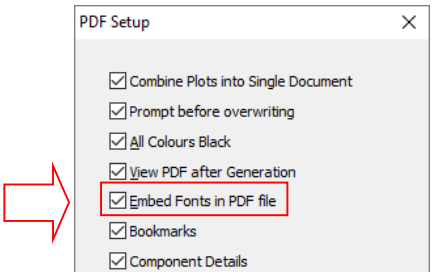
To add a [BomQuantity] Value to a Component, select the Component in a design, right click and select **Values.**



PDF Plot - Embed Fonts

You can now choose to embed fonts when outputting a PDF file.

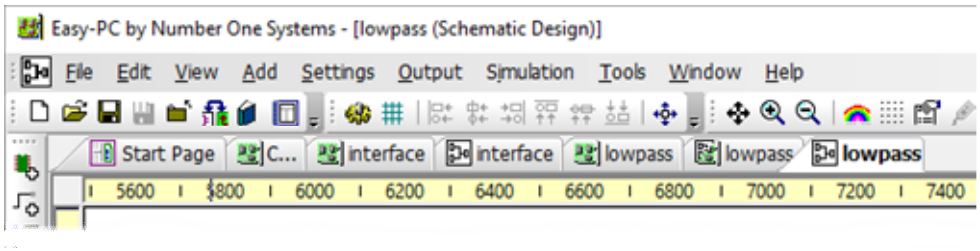
To access the **PDF Setup** dialog, go to the **Output** menu, **Plotting & Printing**, **Output** tab, **PDF**, click **Device Setup** and select the **Embed Fonts in PDF File** option.



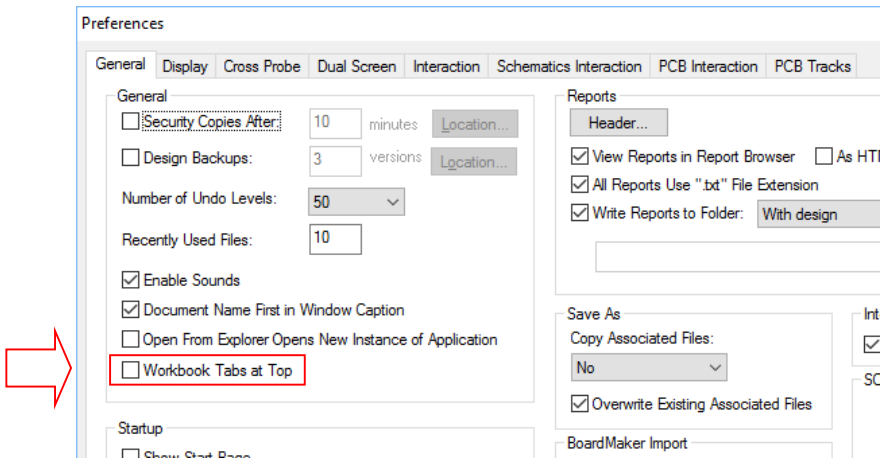
Select this option if have used true type fonts in the design and wish them to be saved into the PDF file. If this is not selected you rely on the fonts being available on other machines that the PDF file will be viewed on. The main reason that you may not want to select this is to reduce the size of the PDF file.

Preferences – Workbook tabs Setting

You can now control whether the tabs for open design files appear at the top or bottom of the drawing area. The default is for the tabs to be positioned at the bottom.

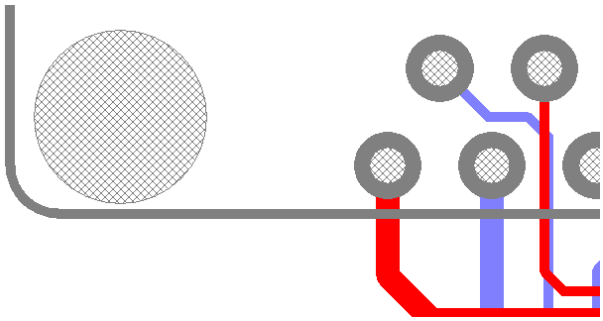


Within the **Settings** menu, **Preferences** dialog, **General** tab, by selecting the **Workbook Tabs at Top** option, you can now control whether the tabs for open design files appear at the top of the drawing area. The default is for the tabs to be positioned at the bottom.

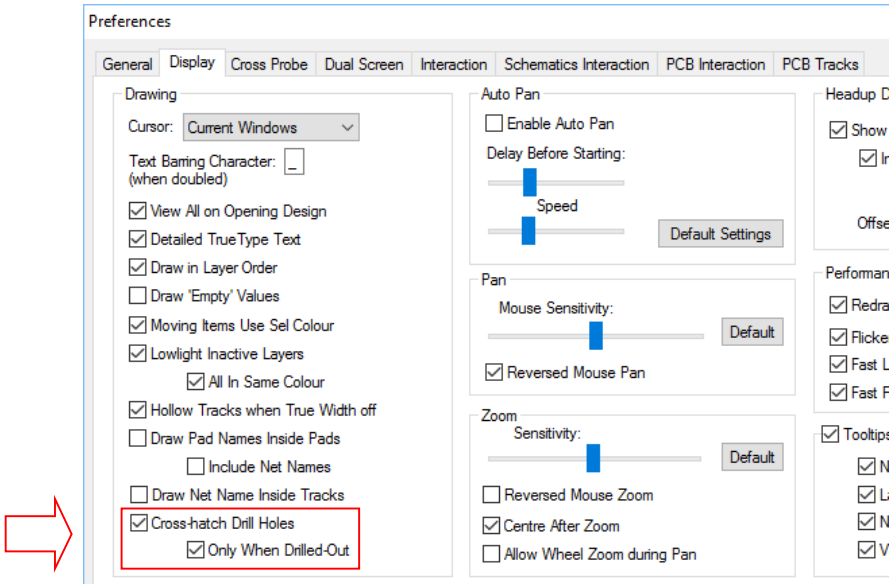


Drawing – Display Hatched Drill Holes (PCB)

A new drawing preference allows you to draw drill holes using a hatched pattern. A secondary option is available to specify whether this applies only to drilled-out holes.



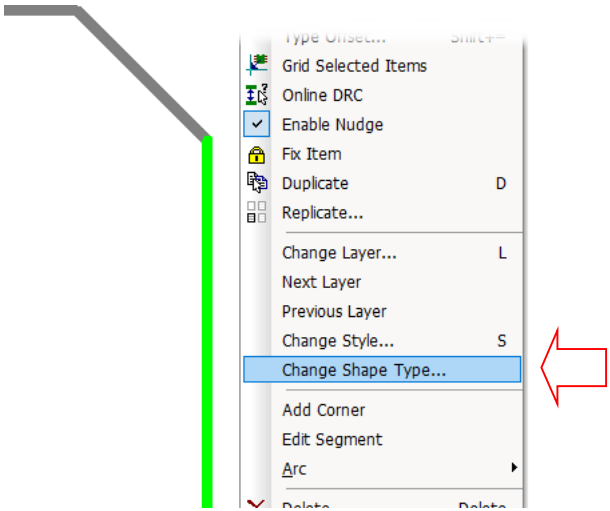
To access this option, go to the **Settings** menu, **Preferences** dialog, **Display** tab. Options are available at the bottom left hand side of the dialog for **Cross-Hath Drill Holes** with an option to **Only When Drilled Out**.



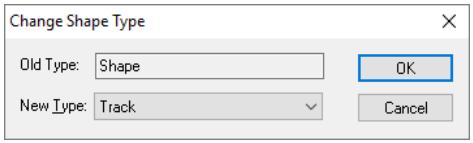
Change Shape Type (PCB)

Open shapes on an electrical layer can now be converted to be a **Track**. Also, if the current selection contains multiple items all of the same type, the **Change Shape Type** option will change them all in one go instead of having to select them one by one.

To do this, select the open shape in your design, right click and select **Change Shape Type** from the context menu.



Once the **Change Shape Dialog** appears, choose the new type required for the shape(s) in the selection.

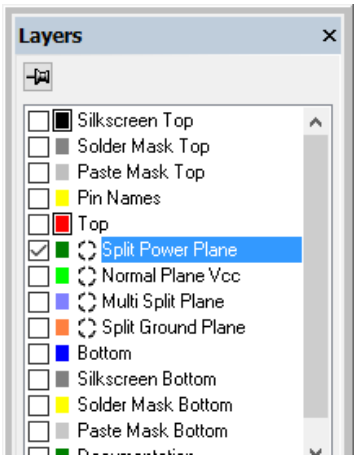


Layers Bar Changes (PCB)

New functionality has been introduced to the **Layers Bar**:

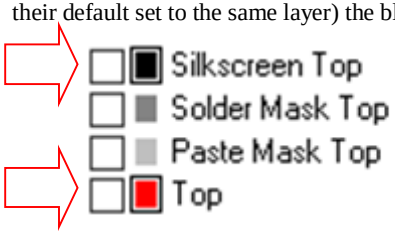
- Ability to define the current (default) electrical and documentation layers
- Ability to show and hide Powerplane layers
- Display notation of power plane layers

The amended layers bar now looks like this:



Defining Current Electrical and Documentation Layers

The colour squares are slightly smaller now allowing a black square outline around both the default electrical and documentation layers. Left click on the squares to set that layer as the default. The black ring will appear to indicate this. If there is no common default (i.e. all categories do not have their default set to the same layer) the black square is not drawn.



When starting a new track or adding a new doc shape, the default layer will be used.

Showing and Hiding Powerplane Layers

Powerplane layers are indicated by a small thermal pad icon to the right of the coloured square.

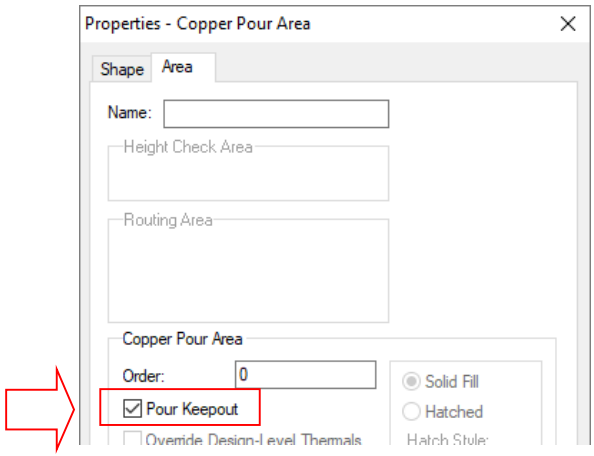
If the powerplane is being displayed in the design the icon is in negative. Left click on the icon to toggle the display to show/hide the powerplane.

Note, this option is a quick way to access the **View Powerplane** functionality. View Powerplane is still available from the **View** menu as before.

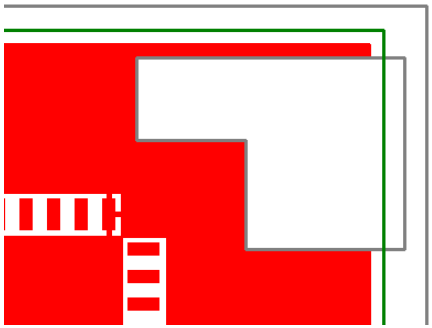
Copper Pour Keepout Areas (PCB)

Adding a **Copper Pour Area** to a power plane layer and designating it as a **Pour Keepout** will cause that region of the layer to be kept clear of copper on the resulting plane.

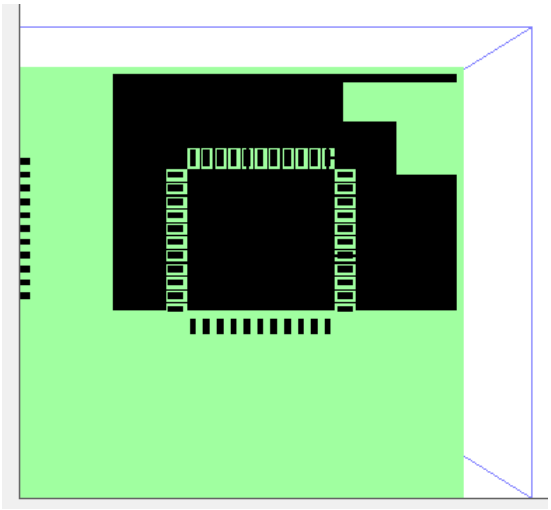
To designate the Copper Pour Area as a Keepout Area, select the area and then in **Properties**, select the **Pour Keepout** check box.



When the Pour Keepout area is added to the design, copper is avoided when poured.



The area will be shown on the **Preview** in the **Plotting & Printing** dialog to assist you in checking that the area is set up correctly.



Shortcuts for Hollow Display of Items

New commands have been added for the following actions, these can be assigned to a shortcut key when required:

- Display all layers
- Hide all layers
- Toggle True Size:
  - Pads
  - Shapes
  - Tracks
- Toggle Display:
  - Areas,
  - Connections
  - Copper
  - Pads
  - Routing
  - Text

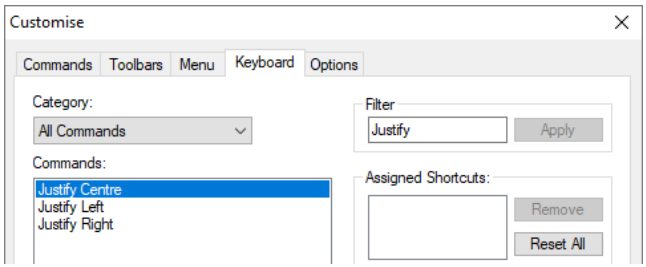
Where applicable, the Toggle Display commands can be used in both the Schematic and PCB design editors, providing you with greater flexibility over what items are displayed in your current view.

To configure shortcuts for your keyboard, from the **Settings** menu, select the **Customise** option and **Keyboard** tab.

Shortcuts for Text justification

There are new commands for justifying (changing **Alignment** setting) of text items in the design. This enables them to be assigned to shortcut keys rather than using the relevant location on the **Properties** dialog.

Three commands for **Justify Left**, **Justify Centre** and **Justify Right**, can be applied to Value Positions, Free Text, and all Value Positions attached to a Component.

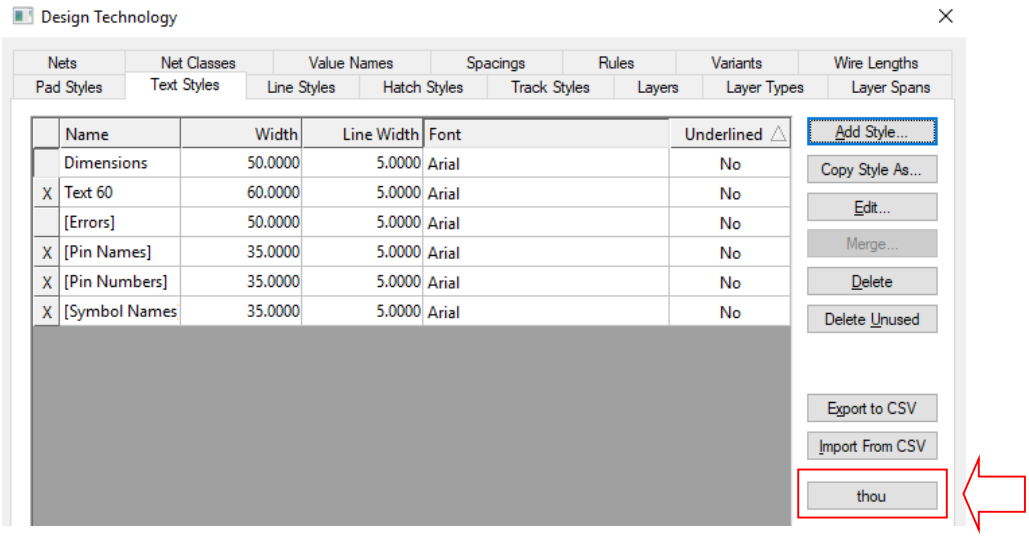


These commands are available from the **Settings** menu, **Customise** option and **Keyboard** page.

Toggle units in Design Technology dialog

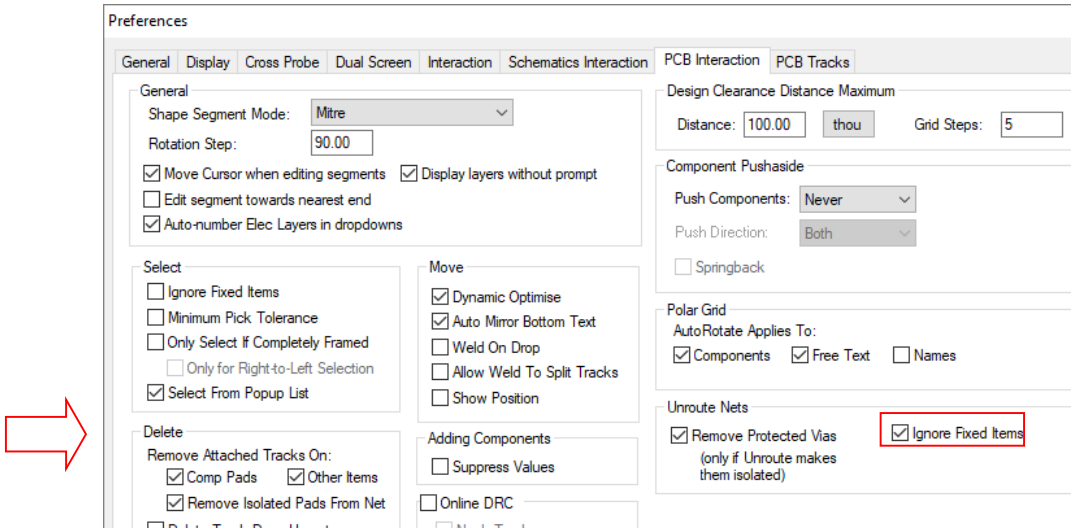
For all tabs on the **Design Technology** dialog where lengths/sizes are displayed (Styles, Rules, Spacings, etc.) there is now a button allowing you to toggle the units between **thou** and **mm**.

You can of course, still add values in the 'other' units from those defined by typing the unit when you define the value, i.e. 1mm when using units of thou. This will be resolved to 39.37 thou when you exit the dialog and enter it again.



Unroute Nets - Ignore Fixed Items (PCB)

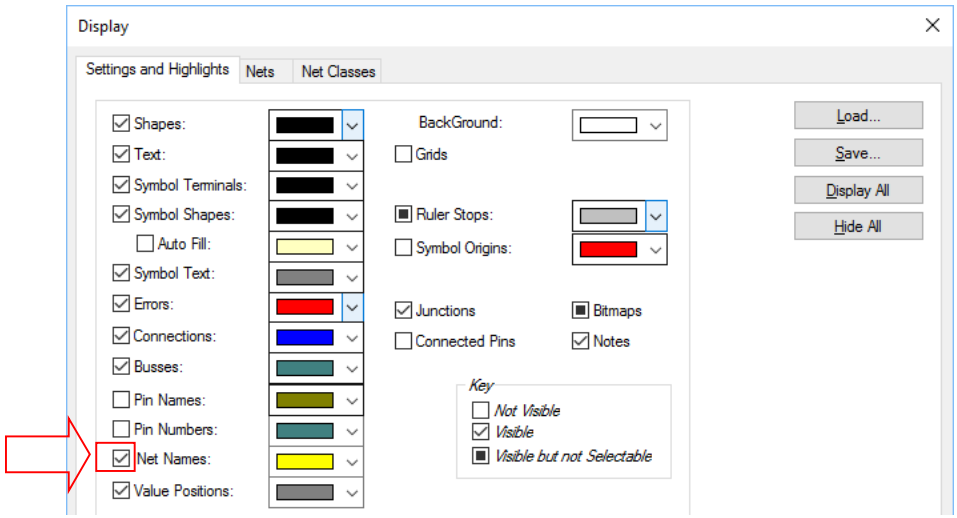
There is a new check box on the **Settings** menu, **Preferences** option, **PCB Interaction** tab, under **Unroute Nets** to **Ignore Fixed Items**. When selected, this means when **Unroute Nets** is run, it will ignore fixed tracks/segments.

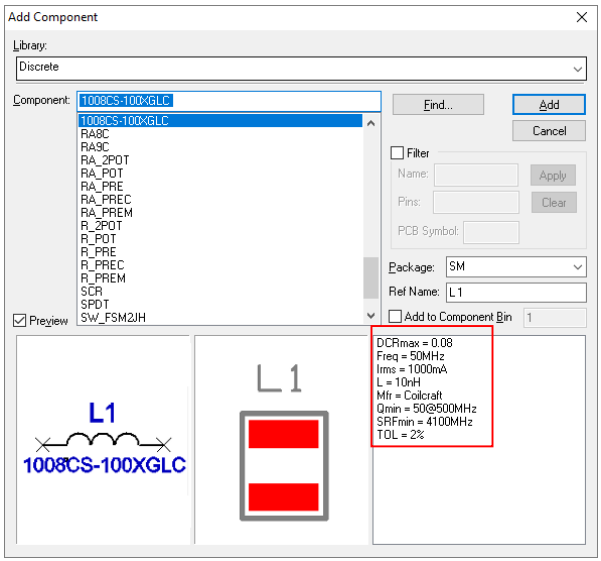


Note: Some exceptions to this still apply: the Auto Routers have their own settings already for keeping fixed items, and Swap Pins will always ignore the fixed status as it unroutes the first segment of tracks joining to the swapped pins.

Display of Net Names (SCM)

From within the **Display** dialog, **Settings & Highlights** page, you can now switch off **Net Names** using the new check box as well as being able to just select the colour for Net Names.



Add Component – Values Preview

On the **Add Component** dialog, a new list of the **Component Values** are displayed in a box to the right of the existing Schematic and PCB symbol Preview windows. This will further help you confirm the chosen component is the correct one before you add it to your design.