

Easy-PC

V20.0 Supplement

Copyright © 1998-2016 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: 01-Aug-2016 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 20.0.....	6
CHAPTER 2. NEW FEATURES IN EASY-PC V20	7
Introduction.....	7
Context menu on Document tabs.....	7
Add New Sheet For Project Command	7
Reset 'Editing Time' on Document Properties dialog.....	8
Tooltips Displaying Component Values.....	8
Display dialog – Show/Hide All.....	10
Crosshair Cursor – Large X.....	10
Replace Components - Global Find and Replace.....	11
Save To Library icon added to Toolbars	12
Centre After Zoom control	12
Preference to disable Zoom When Panning.....	13
Preferred Grid for Component Placement	14
Ability to output the BOM for a single Variant in BOM Composer.....	14
Next From Bin Command	15
Component Names character expansion.....	15
Switch Designs command Icon.....	15
Change Net dialog show/hide Auto-named Net Names.....	16
Popup Calculator for lengths/sizes.....	16
File Print – Print background.....	17
Add To Selection using Alt key.....	17
Find Library item by Description.....	18
Library Manager Enhancements	18
Library Manager – Importing to libraries	18
Symbol previews in Component Editor now use current Technology File	19
Add Components from Find (Library Item) dialog	19
Add Component from Library Manager	20
Pin Assignment Report in Component Editor	21
Highlight Creation Issues in Component Editor.....	21
Standard Display dialog used Symbol Preview windows in Component Editor.....	22
Tooltips displayed for Previews in Component Editor	23
Status bar shows gate or pin information in Component Editor	23
Symbol/Place Origin at Selection Centre [PCB].....	23
Auto-Mask Property for Component Copper [PCB]	24
Panel Wizard – Relative Path [PCB].....	25
Pads-only Plots allow over/under sizing in Plotting dialog [PCB]	25
Native Eagle Import.....	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 20.0** 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 20.0

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 that sits 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 or 10.

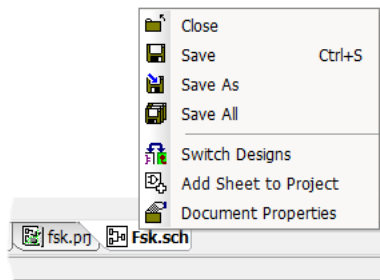
Chapter 2. New Features in Easy-PC V20

Introduction

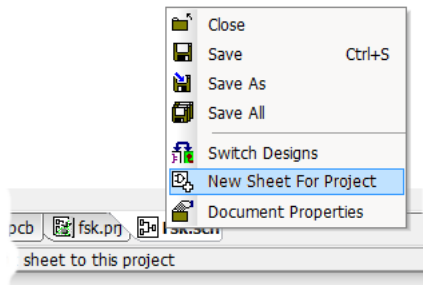
All features are categorised as being applicable for either products (unmarked), or [SCM] specific or [PCB] specific.

Context menu on Document tabs

When tabs are displayed at the bottom of the screen for each of the files currently open, you can now right-click on the tab to display a context menu of largely file-related commands relevant to the type of file.

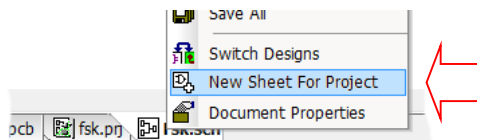


For an open **Project**, the context menu shows Project management functionality:



Add New Sheet For Project Command

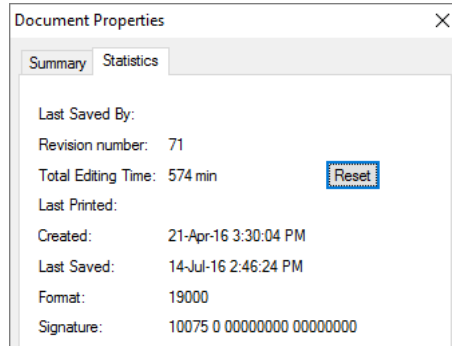
As part of the new context menu on the document tabs, with a Schematic sheet or Project open, there is a new command **New Sheet For Project**.



This can be used to add a new **Schematic** sheet to the currently open **Project**. Once selected, you will be directed through the normal **Add Sheet to Project** set of dialogs.

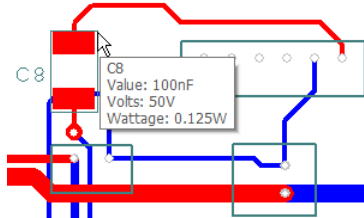
Reset 'Editing Time' on Document Properties dialog

A new **Reset** button has been added to the **Statistics** tab of the **Document Properties** dialog on the file menu. This will reset for the **Total Editing Time**. If current design is one sheet in a multi-sheet project, it will offer the choice of resetting the time in all sheets in the Project. This is available for both SCM and PCB designs but not Project files.

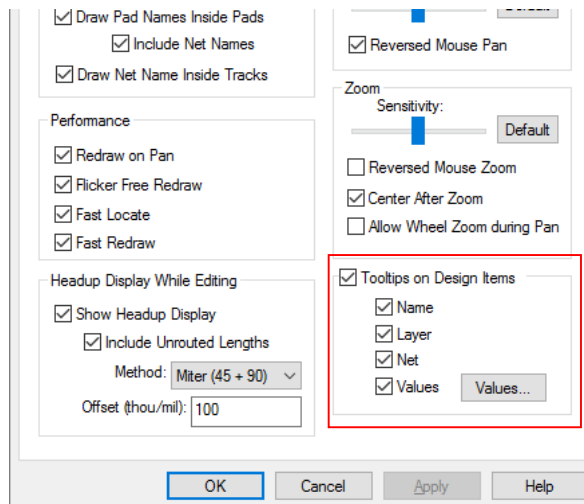


Tooltips Displaying Component Values

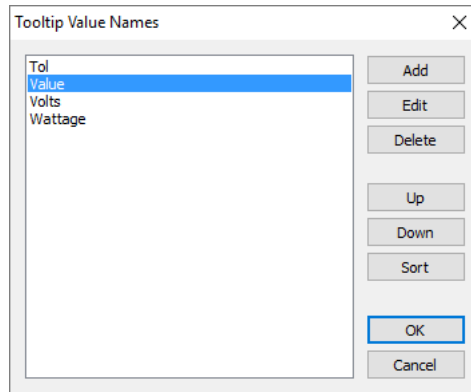
Design Tooltips can now display user-specified Values (Attributes) when hovering over components. The **Value Names** to display are configured on the **Display** page of the **Preferences** dialog.



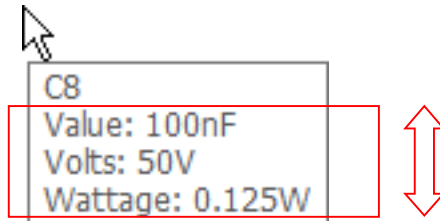
A new **Values** check box and button appear on this dialog:



From the **Values** button, you can select the **Value Names** to display on the tooltip.

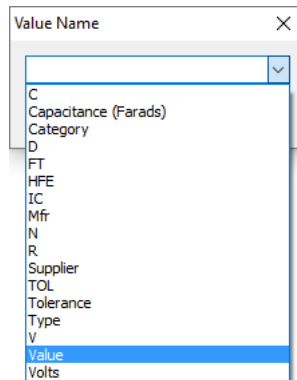


The **Up** and **Down** buttons will allow you to choose the order in which the values appear on the tooltip.



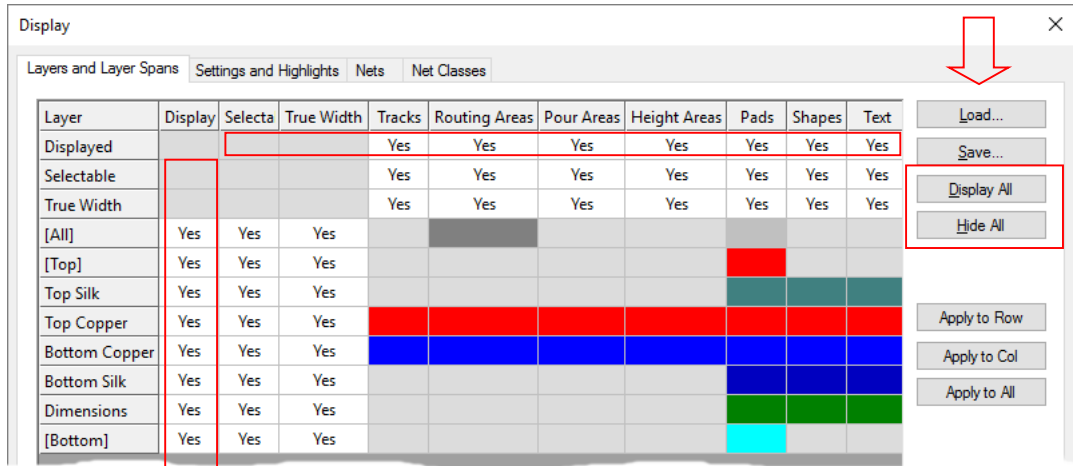
Pressing the **Sort** button will sort the list in alphabetical name order if required.

Using the **Add** button, choose the value required from the drop down list or type the value name you wish to use.



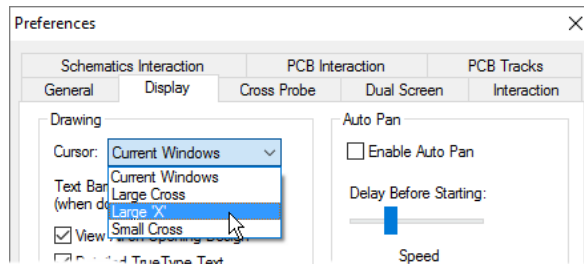
Display dialog – Show/Hide All

New buttons have been added to the **Display** dialog for **Display All** and **Hide All**. These will flip the displayed state of all items on the dialog on or off.

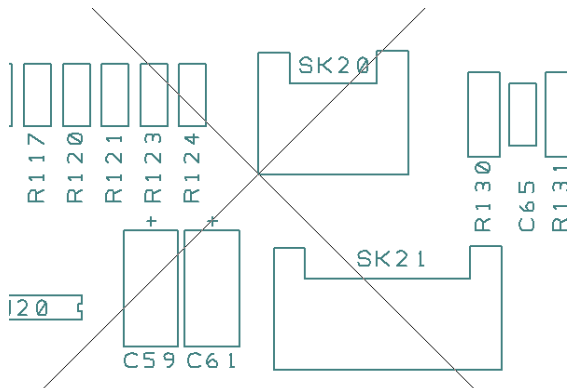


Crosshair Cursor – Large X

An additional cursor style has been added to Easy-PC 20. The **'Large X' 45-degree cross-hair** available from **Preferences, Display**, or by pressing shortcut key **C** to cycle through the available cursor types.



This cursor type is useful where a standard cross cursor is obscured by design objects.



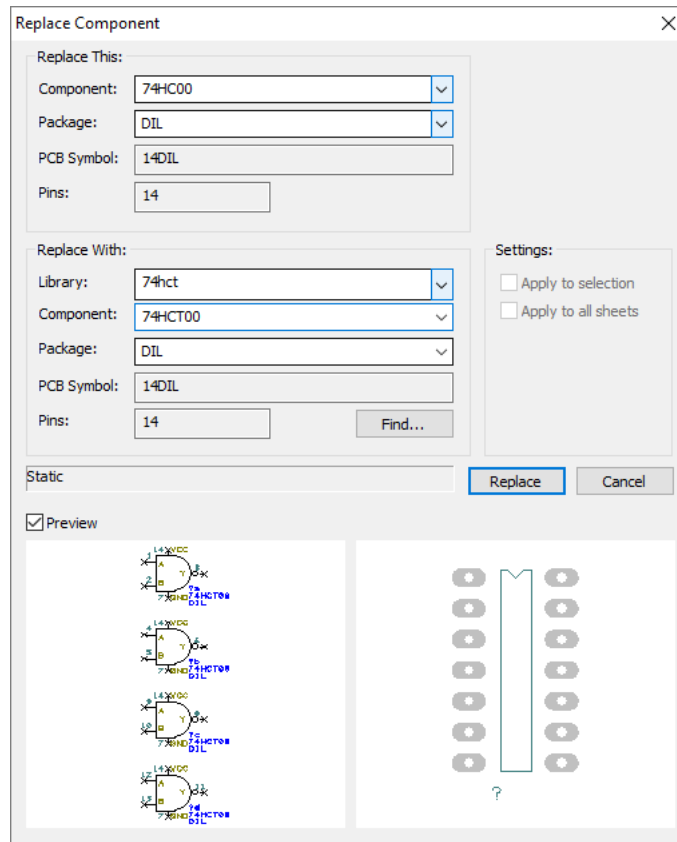
Replace Components - Global Find and Replace

There is a new command on **Tools** menu to **Replace Components**. This enables you to perform 'global' find and replace on components in a design or project. It allows you to choose a replacement component to apply across the design or project.

Use the **Replace This:** selection of **Component** and/or **Package** to select the item to replace. The **Replace With:** option allows you to choose the **Library** and the **Component/Package** as the replacement. Once selected, the **PCB Symbol** name and number of **Pins** will be displayed as confirmation.

The **Preview** check box and windows allow you see previews of the new replacement **Schematic** and **PCB** symbols before replacing them in the design.

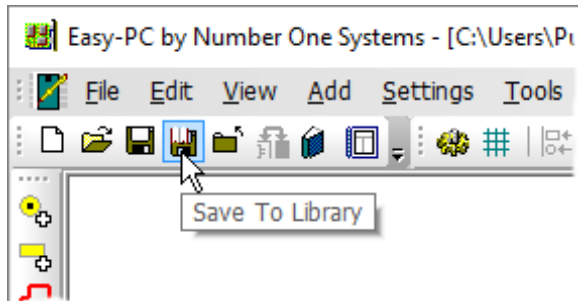
The **Find** button will display the standard **Find** dialog from which to make an advanced search and selection.



Settings switches are available to **Apply to selection** (rather than the whole design) and **Apply to all sheets** in a **Schematic** design.

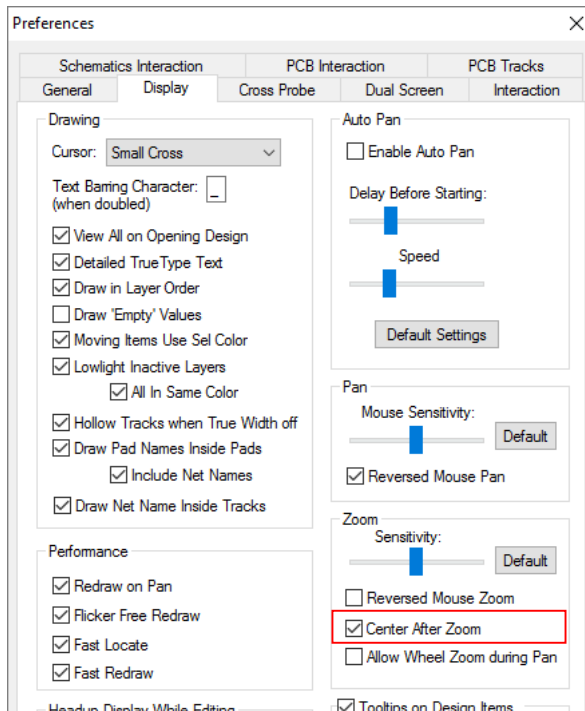
Save To Library icon added to Toolbars

From within the Symbol and Component editors, the default toolbars now include the **Save To Library** icon.



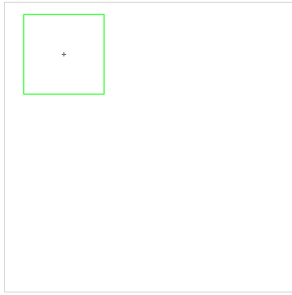
Centre After Zoom control

There is a new option on **Preferences, Display** dialog to choose whether or not to centre the cursor on the screen when Zooming in or out.

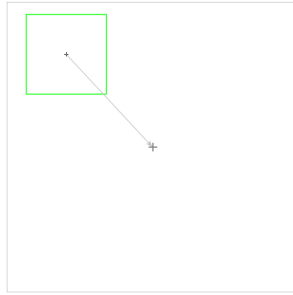


The default switch setting is to have **Centre After Zoom** enabled (checked) as it previously was. The new setting (unchecked) is more in keeping with the Windows style of zoom that other Office style products adopt. In Pulsonix the effect is to be able to zoom into an area and for the zoomed portion of the design (under the cursor) not to jump into the centre of the screen, effectively, it will stay where it is.

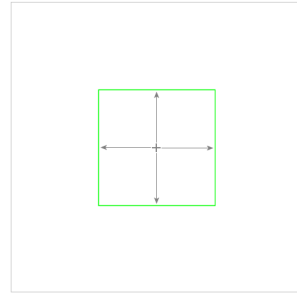
The illustrations below show how the old style zoom to centre works.



This picture shows the board items and the location of the cursor at the time of zoom.

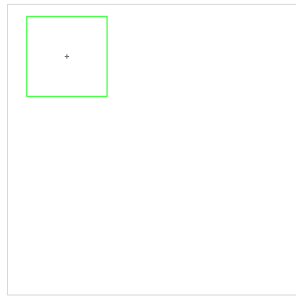


This picture shows the movement of the cursor to the screen centre when zoom is pressed.

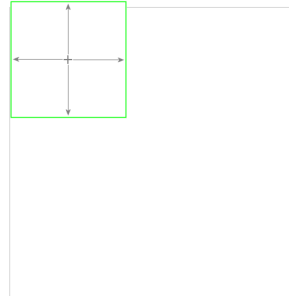


This picture shows the design area zoom expansion around the centre.

The illustrations below show how the new style zoom to centre works



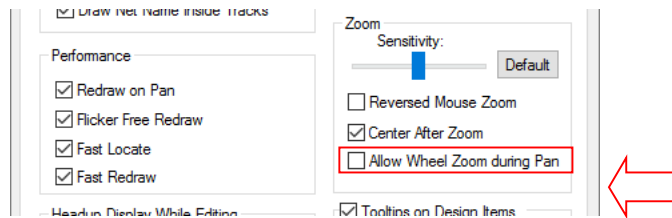
This picture shows the board items and the location of the cursor at the time of zoom.



This picture shows the movement of the cursor to the screen centre when zoom is pressed.

Preference to disable Zoom When Panning

There is a new option (**Allow Wheel Zoom during Pan**) on **Preferences, Display** dialog to disable the mouse-wheel zoom when already middle-button panning. Some mouse wheel/buttons are over-sensitive and a zoom can be performed accidentally, this preference prevents this.

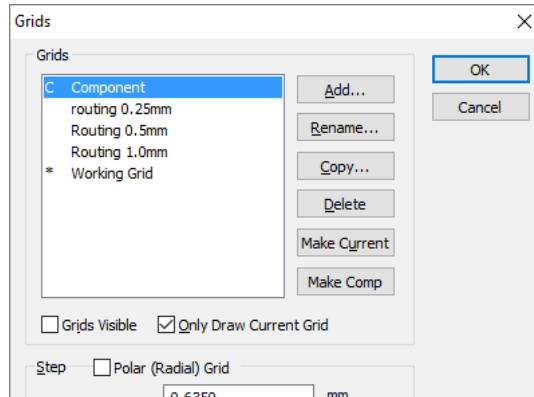


Preferred Grid for Component Placement

From within the **Grids** dialog, there is an extra property of a grid that defines it as the preferred grid for placing Components.

This property is set using the **Make Comp** button. When moving any selection that includes Components, the preferred component grid (if defined) will be used in preference to the current working grid.

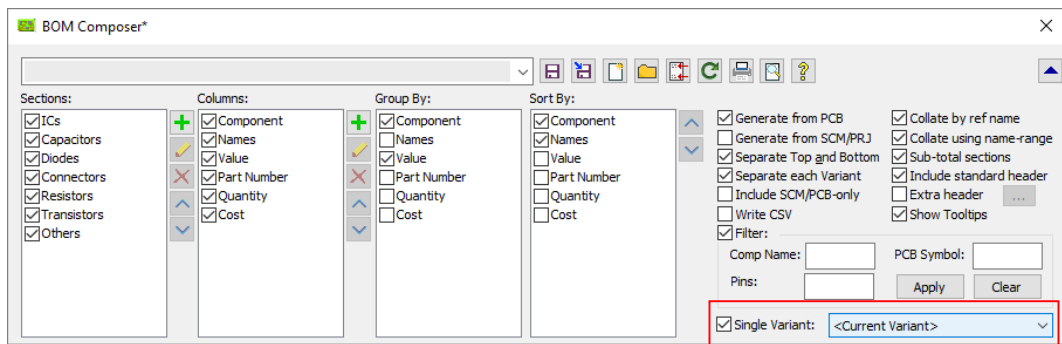
Once **Make Comp** is selected, the grid name selected is highlighted with the **C** character next to it.



Ability to output the BOM for a single Variant in BOM Composer

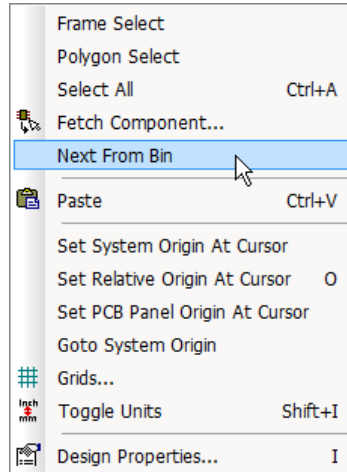
Within the **BOM Composer**, the ability to output the BOM for a **Single Variant** has been added.

A check box to select **Single Variant** plus the actual variant name using a drop down list box have been added to use for the BOM.



Next From Bin Command

With nothing selected in the design, right clicking in free space will now provide you with an option (**Next From Bin**) on the context menu to fetch the items from the **Component Bin** one-by-one onto the cursor for placing. This option is only available provided items exist in the Component Bin.



Component Names character expansion

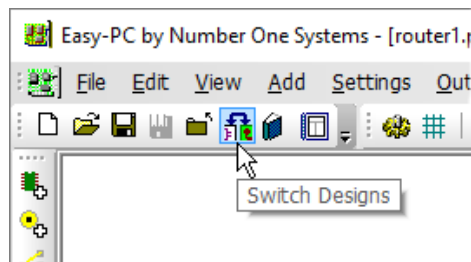
Components in a design can now be named using more than just alphanumeric characters. Up to now you had to name them with alphabetic first and alphanumeric for the rest. Now it will allow you to use alphanumeric anywhere, plus any of the following, in any order:

- _ \$ £ & + = ! @ # ~ \ /

NB only use these special characters if you are fully aware of the implications of doing so.

Switch Designs command Icon

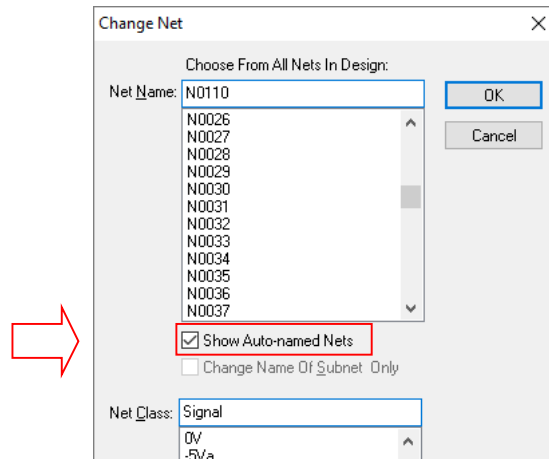
The command **Switch Designs** was introduced a while back but has now been given its own icon and added to the **File Toolbars** to give you instant access to the 'opposite' design. From PCB, the Schematic design and from Schematic, the PCB design.



This command is also available on the **File** menu.

Change Net dialog show/hide Auto-named Net Names

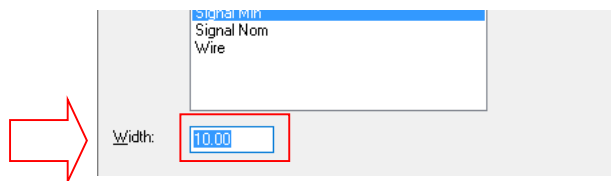
There is an additional check box on the **Change Net** dialog to show/hide auto-named (default) net names from the list.



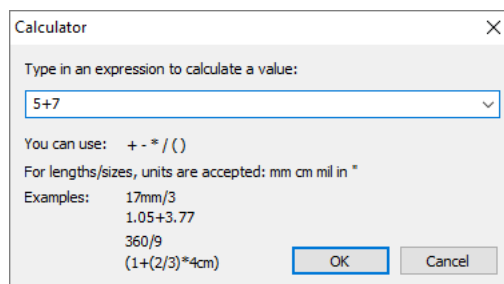
Popup Calculator for lengths/sizes

To assist in the calculation of length/size/angle in edit boxes on various dialogs or in grids such as those on the **Shape Information** bar and the **Design Technology** dialog, a new calculator feature has been introduced.

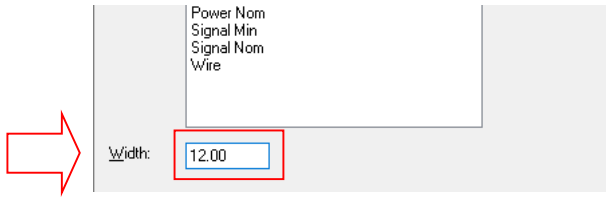
Using the example below, with the Width column selected ready to enter a new value, type =



This displays the **Calculator** dialog into which you can type using the allowed characters.

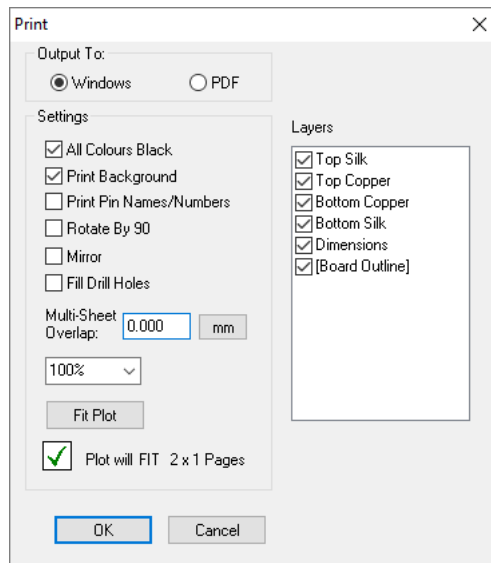


Once the dialog is confirmed using the OK button, the new calculated value is displayed:



File Print – Print background

There is a new check box on all **Print** dialogs allowing you to choose whether or not to print the background as well.



Previously this was hard-wired to print the background for a PDF print and no background for a Windows print.

Add To Selection using Alt key

When using **Frame Select** to select items, in addition to using **Shift** (to select whole items) and **Ctrl** (to toggle selection state) you can now also use the **Alt** key which will **Add To Selection**. This will add the framed items to the current selection without disturbing the selection state of items that are already selected.

Find Library item by Description

In the **Find** dialog, you can now search for components by their **Description**. From the Value names section, use the special value [Description] from the drop down. Find will search based on the text you supply.

The screenshot shows the 'Find' dialog box with the following settings:

- Name: Contains, HCT
- Number of terminals/pads is: 1
- Pcb Sym: Is Exactly
- Value: Is Exactly
- Value: [Description], Is Exactly
- Value: Is Exactly
- Value: Is Exactly
- Only search in Enabled libraries

Note: as this requires access to the Description field in each Component, this information has to be present in the library index files for it to show up in **Find**. The easiest way to ensure this is to go to **Libraries, Components** and select [All Libraries]. Click the **Re-Index** button. You should only have to do this once, the extra index information for Description should be automatically included for any components that you add or edit in the future.

Library Manager Enhancements

Library Manager – Importing to libraries

When using the **Add File** option on any of the **Library Manager** dialogs to add designs or libraries to the selected library, you are now presented with the **Import To Library** dialog.

The screenshot shows the 'Import To Library' dialog box with the following data:

Type	Name	Item Date	Date in Lib	Status	Copy
File	C:\Users\Public\Documents\Easy-PC\Examples\Ex.pcb				
Component	74HC00 (DIL)	15-07-2016 12:00:27	15-07-2016 12:00:27	Same version	No
Component	74HC04 (DIL)	21-05-2007 13:32:31	21-05-2007 13:32:31	Same version	No
PCB symbol	14DIL	01-03-2007 15:04:02		Not in lib	Yes

Buttons: OK, Cancel, Help, Colors...

Add Contents To These Libraries:

- Components: 2, 74hc
- PCB Symbols: Qbsm
- SMC Symbols:

This allows you to see the net effect of importing the selected data to your libraries, as well as importing to all relevant library types in one go.

Instead of simply displaying a list of the library item names that are to be imported, then offering a sequence of “overwrite, yes/no/all/cancel” dialogs, the status of all incoming items are shown in a colour-coded grid.

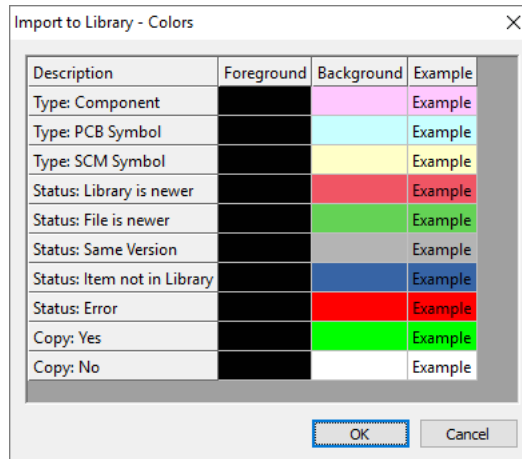
This lets you see at a glance which items are already in the library, which are older or newer in the incoming data, or are only in the incoming data and not in the library.

By using the **Yes** or **No** toggle box in the **Copy** column, you can decide at that point which ones are to be written to the libraries, instead of being prompted one at a time when you press the OK button. Double-click on the Yes/No button to toggle its status.

Where the **Date in Lib** box is empty, this shows that the library item being imported is not in that library file. Care should be taken not to assume that because it doesn't show here, that it isn't in any of the other libraries available on your system.

Colour Coding

Each of the status within this dialog are colour coded. These can be set up to your own preference using the Colours button.



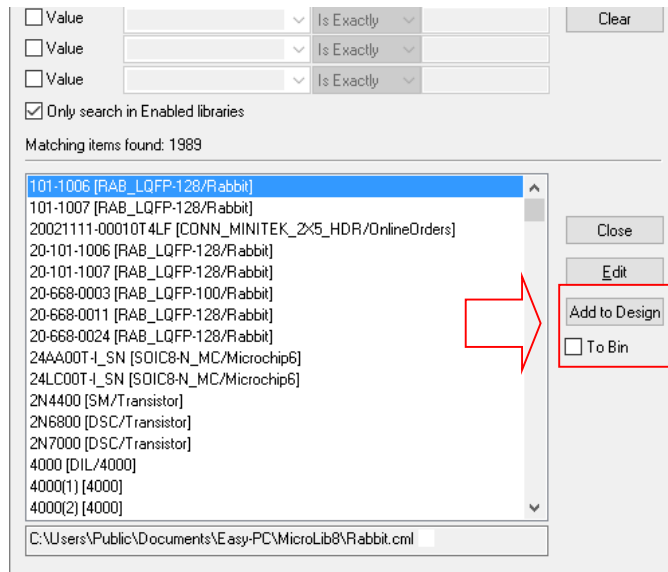
Symbol previews in Component Editor now use current Technology File

Symbol Previews within the **Component Editor** now use current Technology file colours used for the Symbol Editors, like the Library Manager and other dialogs.

Add Components from Find (Library Item) dialog

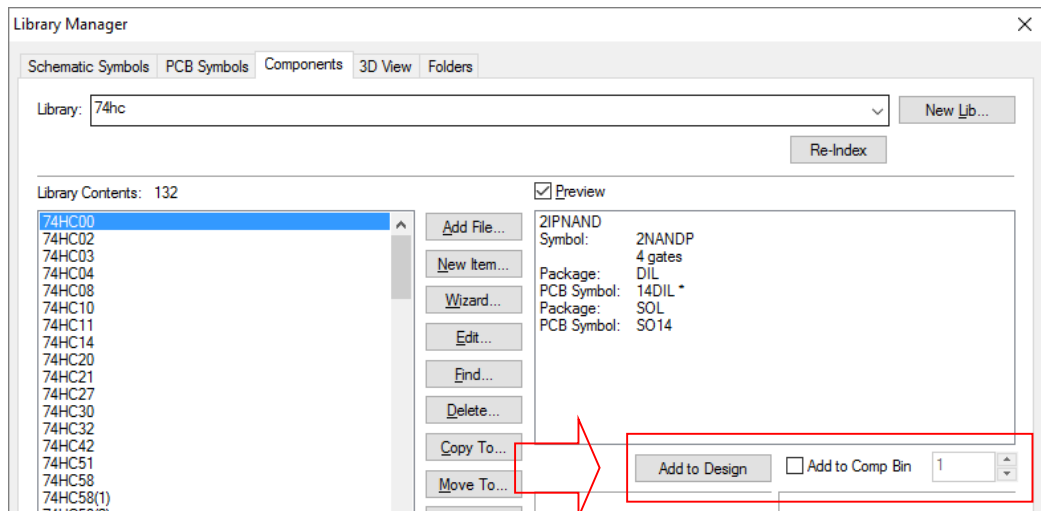
When using the **Find** dialog to search libraries for particular items in the **Library Manager**, new buttons allow you to **add a single component to the design** or to the **Component Bin** directly from this dialog. Only once instance of a found item can be added at a time.

Use the **Add to Design** button from the main dialog to add more.



Add Component from Library Manager

From the **Components** tab of the **Library Manager** it is now possible to add components to the current design directly from the dialog. They can either be added as a quantity to the **Component Bin**, in which case the dialog will stay open, or the dialog will close and the chosen component will be interactively added as if it had been selected from **Add Component**.



Once the component has been added to the design and cancel is pressed, you are exited into the design.

Pin Assignment Report in Component Editor

From within the **Component Editor**, there is a new **Pins Report** on the **View** menu. This lists key information about the pin assignment in the current Component. This includes details of common pins, multiple mapped pins, and any pins that are not assigned.

An example report looks like this:

```

Component Pins Report
-----

Report File:           C:\MicroLib\Rabbit (CML - Component Pins Report).txt
Report Written:       Friday, July 15, 2016
Design Path:         C:\MicroLib\Rabbit.cml
Design Title:
Created:             15-Jul-16 3:28:25 PM
Last Saved:
Editing Time:        0 min
Units:              mm (precision 3)

PCB SYMBOL PIN NAMES APPEARING MORE THAN ONCE
=====
None
COMPONENT PIN NAMES APPEARING MORE THAN ONCE
=====
None
GATE A
=====
Symbol: 101-1006_A
Pins: 64
GATE B
=====
Symbol: 101-1006_B
Pins: 64
End Of Report.
    
```

Highlight Creation Issues in Component Editor

Within the **Component Editor**, grid cells now include small coloured bars at their left edge to highlight issues. These will highlight items such as unassigned, multiple mapped, and common pins. The bar is shown in the left hand side of the cell to which it represents.

Below is an example of the unassigned pins using a red 'bar':

Gate	Sch Symbol	Sch Symbol	Sch Terminal	Pcb Symbol	Component Pin	Net (Clas
Name	Name	Terminal Name	Number	Pad Number	Name/Number	Name
a	R	A	1			
		B	2			

Below is an example of the multiple mapped pins using a light blue 'bar':

Gate	Sch Symbol	Sch Symbol	Sch Terminal	Pcb Symbol	Component Pin	Net (Clas
Name	Name	Terminal Name	Number	Pad Number	Name/Number	Name
a	R	A	1	1	1	
		B	2	1	1	

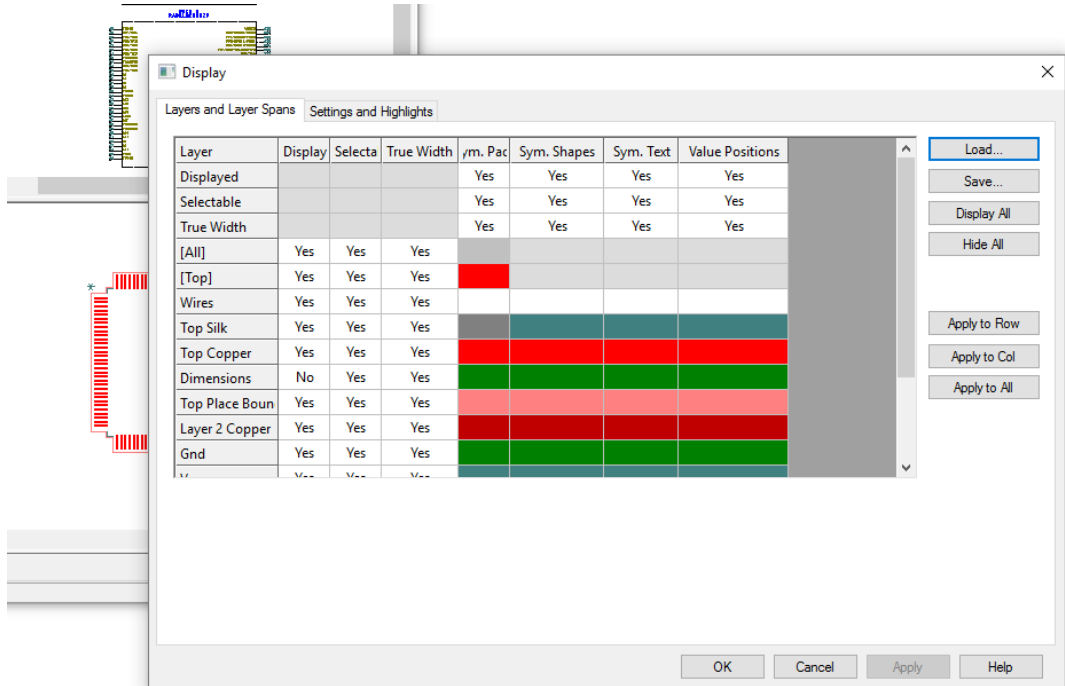
Below is an example of the multiple mapped pins using a double, purple/blue 'bar':

Gate	Sch Symbol	Sch Symbol	Sch Terminal	Pcb Symbol	Component Pin	Net (Clas
Name	Name	Terminal Name	Number	Pad Number	Name/Number	Name
a	2NANDP	A	1	1	1	
		B	2	2	2	
		Y	3	3	3	
		VCC	4	14	14	
		GND	5	8,9	8,9	
b	2NANDP	A	1	4	4	

Once the issue has been corrected, the bars are removed dynamically. There is no manual user update to make.

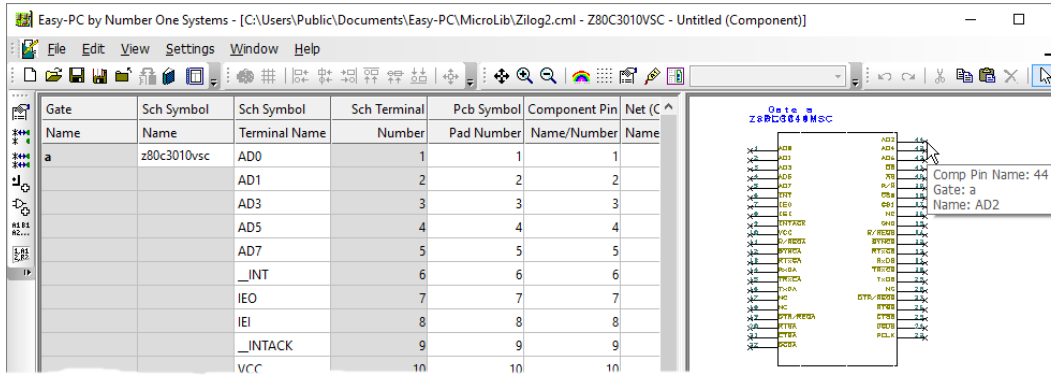
Standard Display dialog used Symbol Preview windows in Component Editor

Within the **Component Editor**, the **Display** dialog is now the same standard colour dialog as used in the SCM or PCB editors. This gives you more flexibility on how items are displayed.



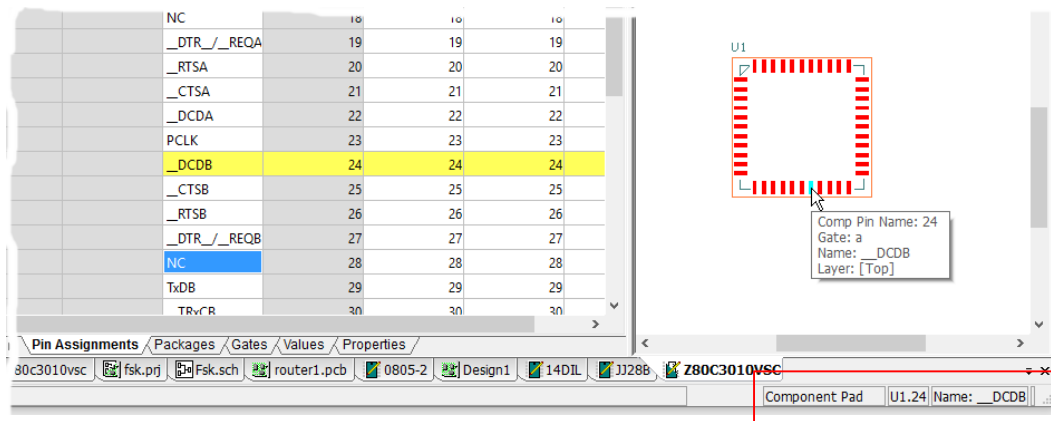
Tooltips displayed for Previews in Component Editor

Tooltips are now displayed when hovering the cursor over **Pins** in both the Schematic and PCB symbol **Previews** within the **Component Editor**.



Status bar shows gate or pin information in Component Editor

Information about a **selected Pin** or **Gate** in the **Preview** window is now displayed on the **Status Bar** (as well as the tooltip).

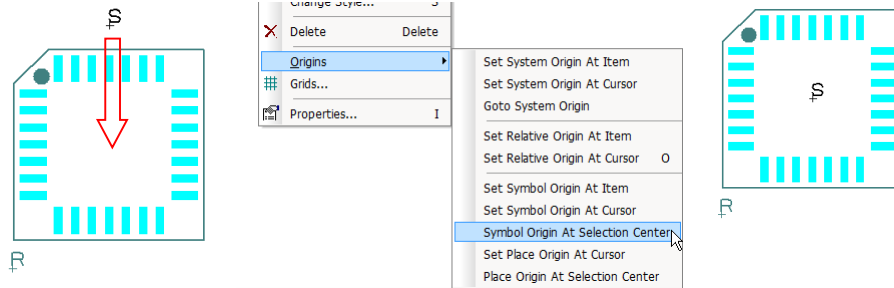


Symbol/Place Origin at Selection Centre [PCB]

From within the **Symbol Editors**, there are new context menu commands for placing origins. These now include **Symbol Origin At Selection Centre** and similar **Place Origin At Selection Centre** (for PCB symbols).

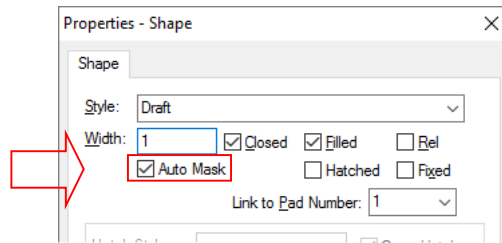
These calculate the centre of the bounding box (excluding any pin names or numbers but including any origins themselves) as the location for the particular type of origin. This will assist with tasks such as positioning the origin in the centre of the body or only some of the pads on a footprint if the pad placement is not symmetrical.

In the example below, the Symbol Origin will be placed in the centre of the footprint by selecting the pads only first, then choosing the option from the context menu.



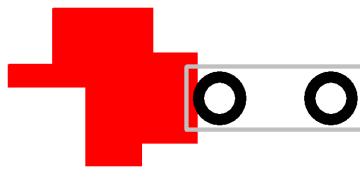
Auto-Mask Property for Component Copper [PCB]

From within the **Footprint Editor**, for a selected **shape** on **copper layers**, the **Shape Properties** has an extra check box – **Auto Mask**.

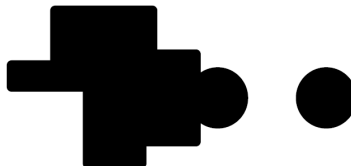


This is available when the copper shape has a pad number assigned to make it into a 'custom' extension of the pad shape. With the new check box selected, whenever the pad is plotted on a **solder** or **paste mask** layer, a shape corresponding to that copper shape will automatically be generated on the mask as well.

Shape in design looks like this:

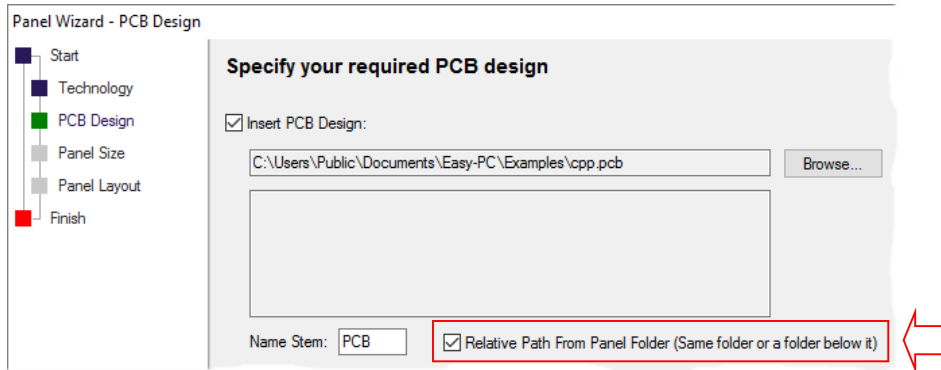


Shape plotted oversize to a solder mask plot looks like this:



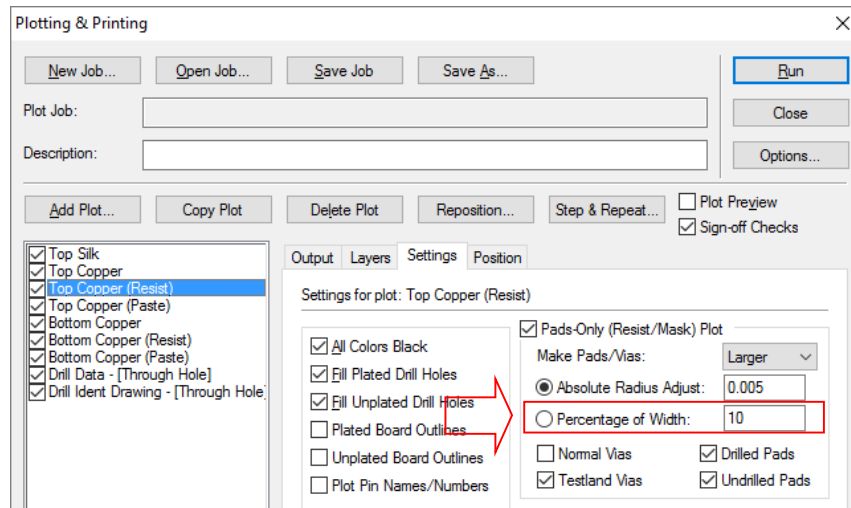
Panel Wizard – Relative Path [PCB]

On the **PCB Design** page of the **Panel Wizard**, it now defaults to **Relative paths** for inserted PCB files.



Pads-only Plots allow over/under sizing in Plotting dialog [PCB]

Within the **Plotting and Printing** dialog, on the **Settings** page, there are extra controls for a pads-only plot to allow resist/paste over/under sizing to be specified by **percentage** of the pad size.

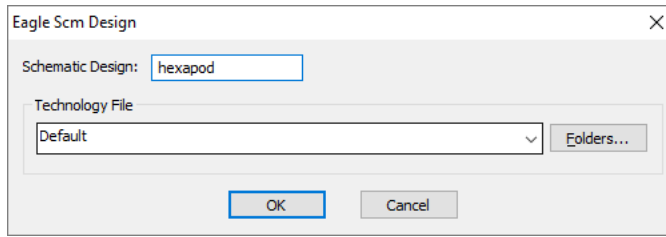


This brings the ‘old style’ auto-generated pads-only plots in line with what can already be done for user-defined layers in the **Layer Type** settings, but it also provides consistent functionality within the **Plotting and Printing** dialog.

Native Eagle Import

The standard Cadsoft Eagle design and library import mechanism has been improved with the introduction of 'native' format import. This means an Eagle file does not need to be converted now to import it into Easy-PC making the process much easier.

To convert Eagle files, simply drag the .sch or .brd file from Windows Explorer to the open Easy-PC program. The standard import dialog will be displayed to confirm the file format. You can choose to import the file using a **Technology** file at this point also.



Note: files older than Eagle V6 must still be imported using the previous Eagle intermediate ULP format files which are still supplied.
