Easy-PC

V27.0 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: 02/08/23 Issue 1

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.

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

The installation is the same for new and existing users alike. Existing users with versions prior to this latest version can install the new software over an existing installation without deleting the old one first.

Installation From CD

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

Installing over existing Easy-PC software

If you already have an earlier version of Easy-PC installed on your system and you wish to install the new version into the same folder as the earlier one, please note that you will then end up with both versions listed in the Windows Control Panel list of installed applications.

If you don’t want the earlier one to be listed in the **Control Panel**, you will need to un-install that version **before** you install the new one. If you install the new software into the same folder as the old version then try to un-install the old one, you will find that the new software will not run as the un-install will have removed many or all of the program files.

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

Uninstalling Existing Easy-PC Software

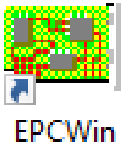
Uninstalling will still remove shared registry entries, so it is recommended that a configuration file be saved first using the **Configuration Files** option from the **Help** menu and **Support** option. This will provide a restore point for any settings which may be lost.

Data Files Location

There is a step in the **Setup** installation wizard that asks you where you want to place data files (for example, Libraries, Technology files, etc). The default is always to use the common documents folder, C:\Users\Public\Documents\Easy-PC on Windows 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 27.0

Once installed, an Easy-PC shortcut icon will appear on your desktop. This is also available on the **Start** panel in the **Number One Systems** folder.



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

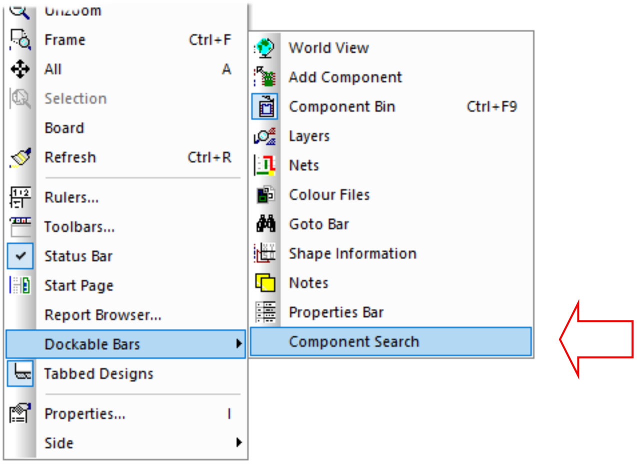
Chapter 2. New Features in Easy-PC V27

SnapEDA Component Content Interface

You can now access ready-made components from SnapEDA using a built-in interface. This enables you to search parts and download them directly into your Easy-PC designs and into your library.

Accessing the Interface

From the **View** menu, **Dockable Bars>** you can select the **Component Search** option. You can also select this option by right clicking on the Easy-PC framework and selecting **Component Search** from the menu.



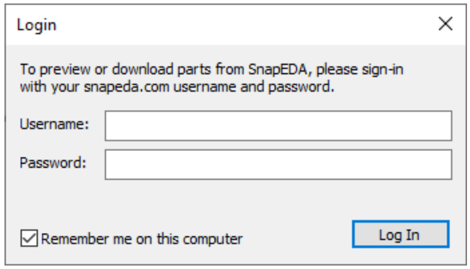
Prerequisites

You must first have a SnapEDA account in order to download searched contents using this interface. Go to the SnapEDA web site (www.snapeda.com) to create yourself an account. You only need to do this once. Without an account, you can search and view content but not download it, you will need a login

Running the Component Searcher

Once you have created the SnapEDA account you can run the browser interface in Easy-PC, it will be displayed, ready for searching.

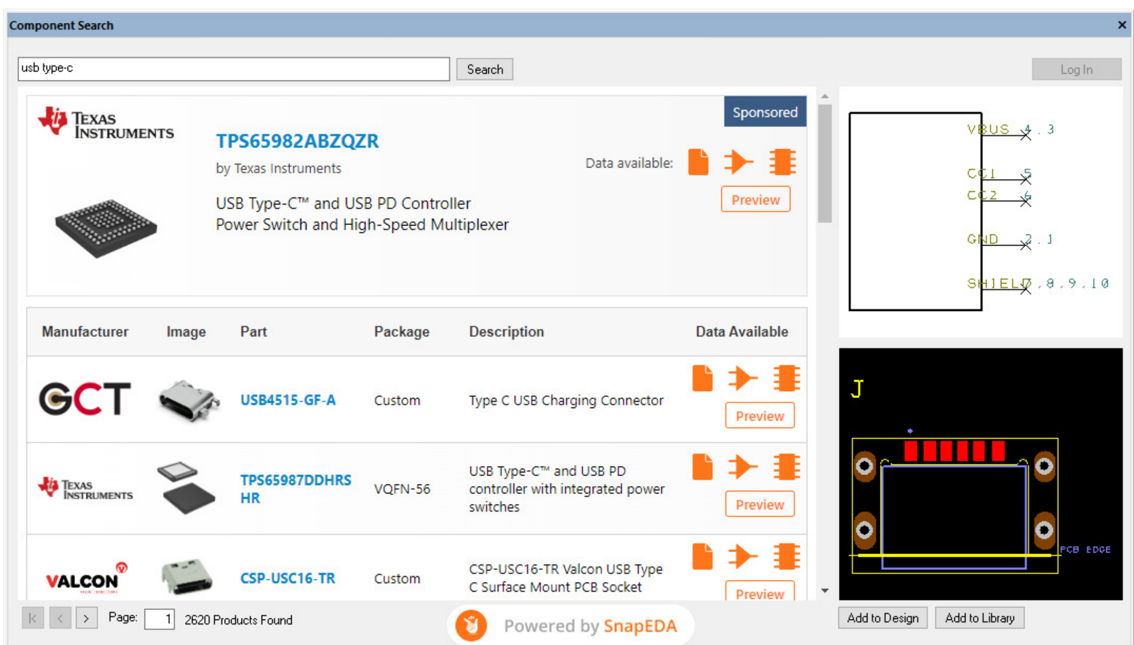
Use the **Log In** button on the SnapEDA Interface to access your login details page for the SnapEDA web site.



You must have a login to download library content from this web site (see above). You should enter your details for the first time of use, then these will be remembered for subsequent use. Without the account and login, you can search and view content but not download it without the login.

Running the Component Searcher

Once logged in, type your required Part name into the **Search** field and press <Enter>:



You will be presented with a list of found results, or a warning message if it doesn't find any.

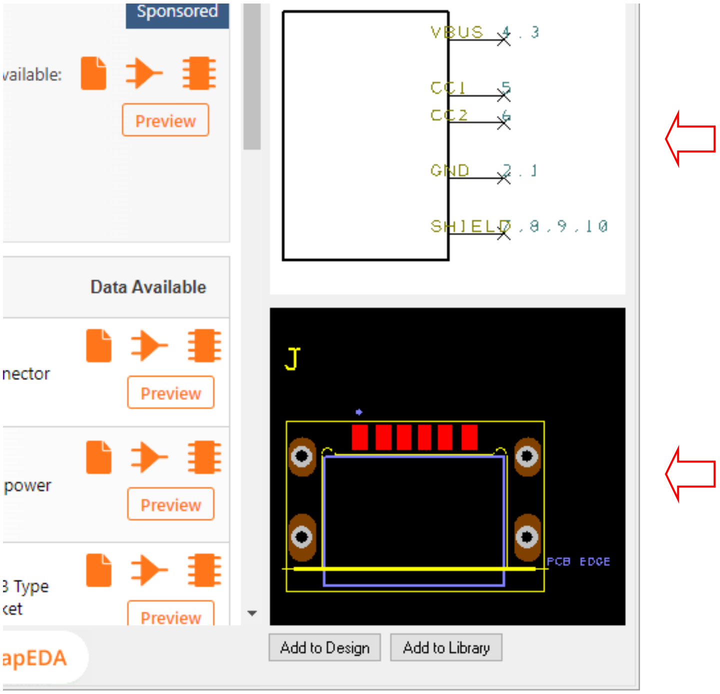
Once items are found based on your search criteria, the results window will display an image of the manufacturer for each item, an image of the item plus icons for the Part **Datasheet**, **Schematic Symbol** and **PCB Footprint** if these are available as downloads in Easy-PC format.

Be aware that some Parts may not have downloadable content available. However, once you have an account, you can request content creation directly from the SnapEDA web site.

By selecting any of the orange icons to the right side of the results will display a quick preview of the item.

At this point, they have not been downloaded into the Easy-PC environment.

If you select the **Preview** button, this will download the Part data and display it in the **Preview** windows in the **Component Search Bar**.



Preview Windows

Once the Preview button is selected, the Symbol and Footprint will be displayed in the Preview windows to the right side of the dialog. This indicates how they will appear in Easy-PC in your design.

Once displayed, you can drag and drop the symbol into the design, or you can use the **Add To Design** button.

Add To Library

Use the **Add To Library** button to add the downloaded **Component**, **Schematic Symbol** and **PCB Footprint** into your own local Easy-PC library. Doing this means that you can also edit the Symbol or Footprint if you wish to refine it to suit your own needs. Remember also, these library items can be used on another design without having to download them again, whether you modify them or not.

The downloaded library contents will be placed in a newly created library named **DownloadedLibrary** under your **Library** folder.

Add to Design

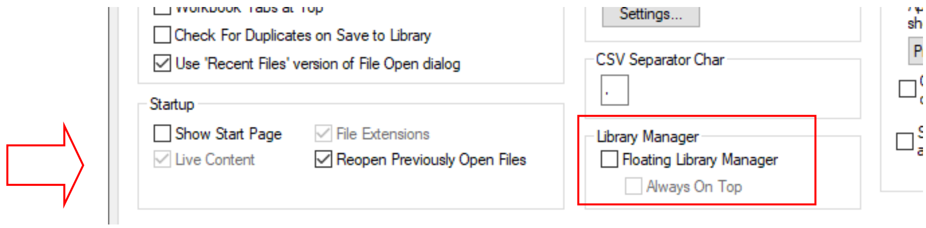
Use this button to add the downloaded Component into your design. This will place the Component at the end of your cursor in the design ready for placement plus the relevant symbol depending which editor you are currently using; Schematic or PCB. Another way to add the symbol is to select it in the preview window, then simply drag and drop it into your design.

Library Manager

Ability to Float Library Manager

The **Library Manager** dialog can now be made floating. This means, when editing a Component or Symbol, the dialog will still be visible.

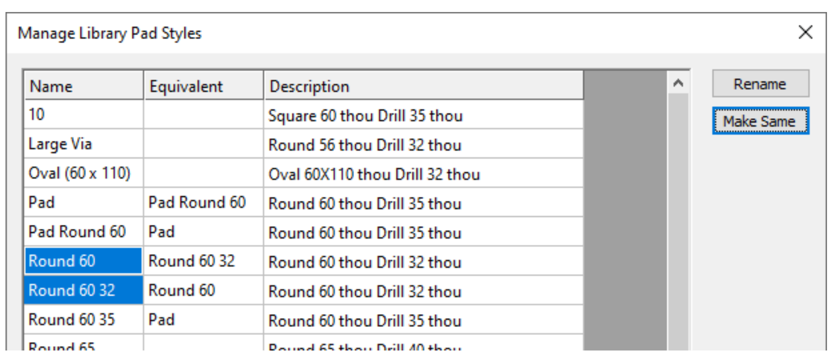
There are new check boxes to control this on the **Settings** menu, under **Preferences** and **General**.



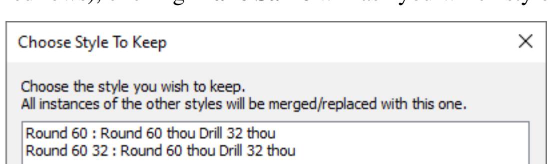
Two switches allow the **Library Manager** to be **Floating** and to **Always be On Top** (when floating).

Styles Tool Improvements

Improvements have been made to the way you select and apply style merging in the **Styles Tool**.



The new functionality operates like this: if you select more than one style (using Ctrl+Click to select the desired rows), clicking **Make Same** will ask you which style name you want to keep:



Selecting the style in the list will merge that style onto all the other ones you have selected once OK is pressed. This includes the ability to merge styles which are *not* equivalent, which you could not do previously.

Previously, if you selected one style that has equivalent(s), it would merge all equivalent styles with that one. There was no control over the name used.

Edit Component - Name Ranges and ‘Display As’ feature in Multiply Named Pins

There are changes to the Component editor and an extension to the definition of multiply named pins. This has been added to the **Pin Assignments** page of the **Component editor**.

Name Ranges

When creating multiply named pins in the Component editor, you can now use simple name ranges to specify lists of consecutive pin numbers. For example: 1,4,9-14,18,20-24 or 2+5-7+12. Previously, all pins had to be defined explicitly.

Note that there is one special case, where your entire set of pin numbers is in a single range. In this case the range is treated as if you have used commas to specify ‘same net’: 1-9

Gate Name	Sch Symbol Name	Sch Symbol Terminal Name	Sch Terminal Number	Pcb Symbol Pad Number	Component Pin Name/Number	Net (Cl Name
a	2NANDP	A	1	1	1	
		B	2	2	2	
		Y	3	3	3	
		VCC	4	14,8,11	14,8,11	
		GND	5	7	7	
b	2NANDP	A	1	4	4	

A comma will be used for externally connected pins, but if you want to specify internally connected pins instead, you will have to add the plus (+) sign as a suffix to the pin number string: 1-9+

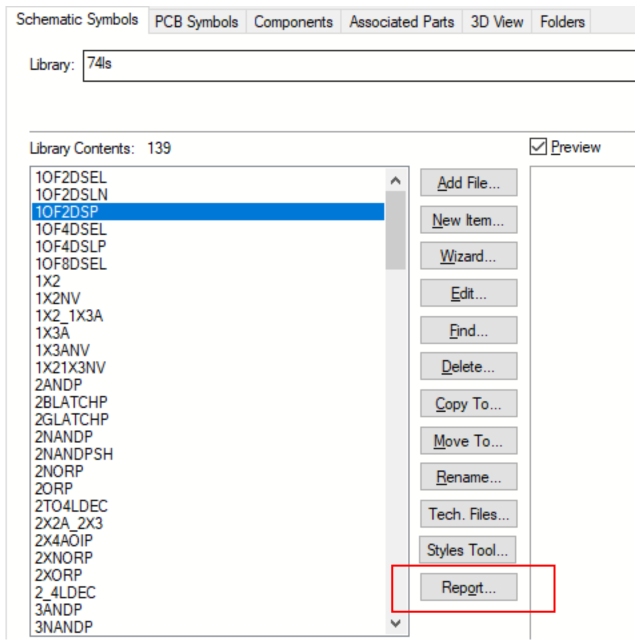
Name Prefix (Display AS)

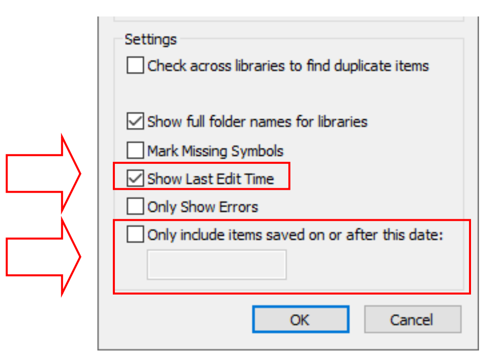
You can also prefix a multiply named pin with a string that you want to display instead of the list of actual pin numbers. To do this, put your display string first in the **Pad Number** column, followed by the equals sign, then the rest of the multiple pin string. For example: Vdd=2,9-13,7,12

Gate Name	Sch Symbol Name	Sch Symbol Terminal Name	Sch Terminal Number	Pcb Symbol Pad Number	Component Pin Name/Number	Net (Cl Name
a	2NANDP	A	1	1	1	
		B	2	2	2	
		Y	3	3	3	
		VCC	4	14,8,11	14,8,11	
		GND	5	7	7	
b	2NANDP	A	1	4	4	
		B	2	5	5	
		Y	3	6	6	
		VCC	4	14	14	
		GND	5	7	7	
c	2NANDP	A	1	Vdd=9-13	Vdd=9-13	
		B	2	10	10	
		Y	3	8	8	
		VCC	4	14	14	

Library Reports

From the **Libraries** option on the **File** menu, **Report** button, the **Library Reports** now have two additional options on the dialog.





Show Last Edit Time will cause the last edit time of library item (if defined) to be included as a column in the report.

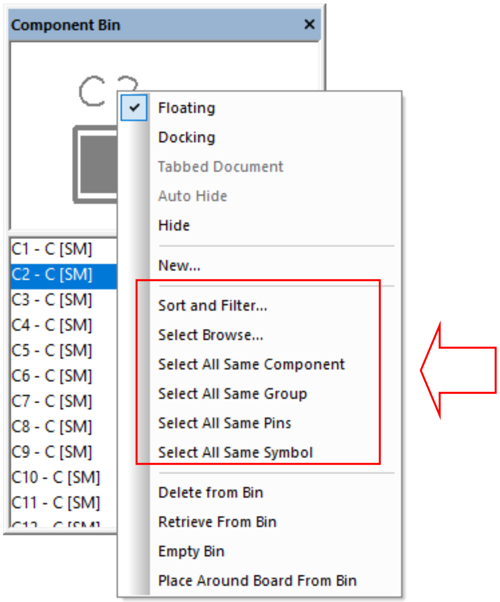
Only include items saved on or after this date which means the report will only include items saved on or after the date specified in the date box below it.

Component Bin

New Selection Options

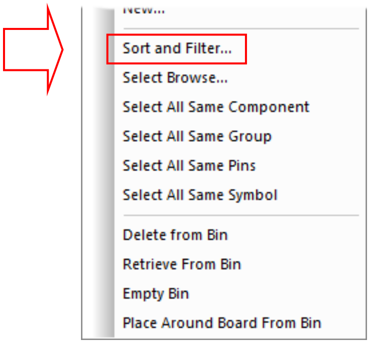
From the context menu in the **Component Bin**, there are new commands to assist you with selection, sorting and filtering, dragging the desired set of Components out of the bin and for placement around the board outline:

- Sort and Filter
- Select 'Same' Commands
- Place Around Board From Bin

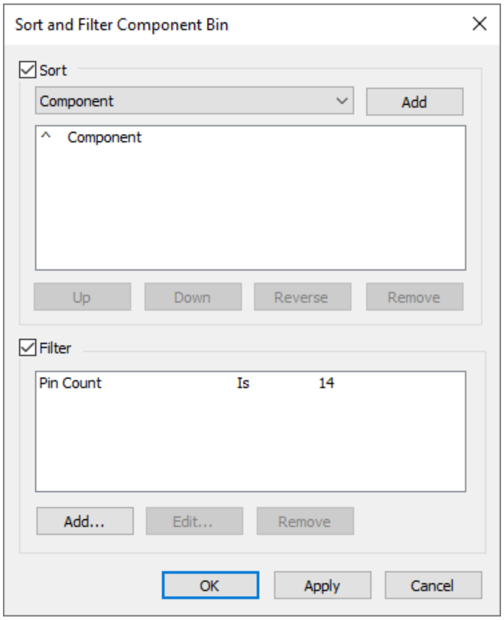


Sort and Filter

When you right-click on the **Component Bin** list, the command, **Sort and Filter** is available.



This invokes a new dialog from where you can specify parameters for sorting and filtering the Components shown in the bin.

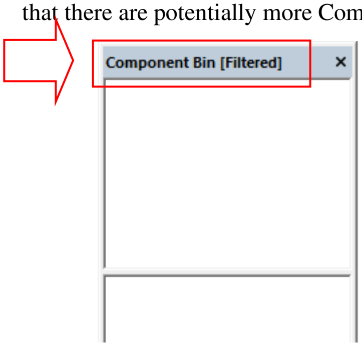


Sorting and **Filtering** options allow you to sort in your preferred order based on **Component** (name), **Group**, **Name**, **Pin Count** and **Sheet** (for a Schematic), in whatever order or combination is required.

Both categories are slightly different; **Sorting** enables list of priority to be created in ascending or descending order using the **Reverse** button. The sort direction is indicated with the small arrow before the sort name.

Filter enables you to remove or include items that you want to see in the list.

When filtering options are enabled, the caption of the component bin shows **[Filtered]** to remind you that there are potentially more Components in the bin than are currently shown.

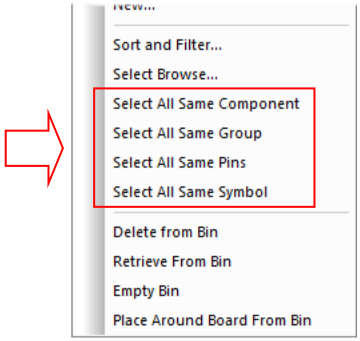


Select 'Same' Commands

When you right-click on the **Component Bin** list, there are new commands available to **Select All Same xxx** to assist you when dragging the desired set of Components out onto the design:

- Select All Same Component

- Select All Same Group
- Select All Same Pins
- Select All Same Symbol

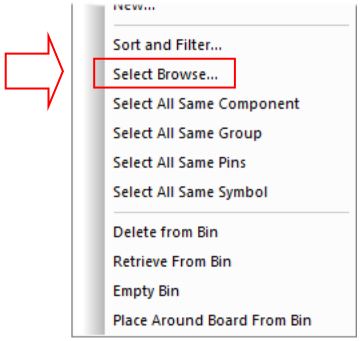


These options look at the item(s) already selected in the Bin, and go through the other items in the Bin to select the ones that match on the chosen criteria.

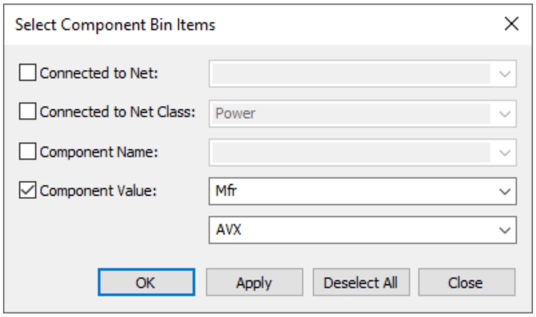
For example, select one simple resistor, right click and **Select All Same Pins**, and all two-pin components will become selected.

Select Browse

When working with the **Component Bin**, you can select items in the bin based on criteria chosen using the new option, **Select Browse**.



This option is available from the context menu when right clicking in the list.



This displays a dialog from where you can make a selection based on:

Connected to Net - One or more pins on a Component connected to specified Net.

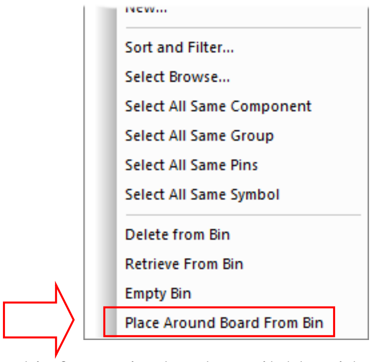
Connected to Net Class - One or more pins on a Component connected to specified Net Class.

Component Name - Match by Component Name.

Component Value - Match the Value, Select the **Value Name** from the drop down list or type a name in. Type in the **Value** or select it from the drop down list.

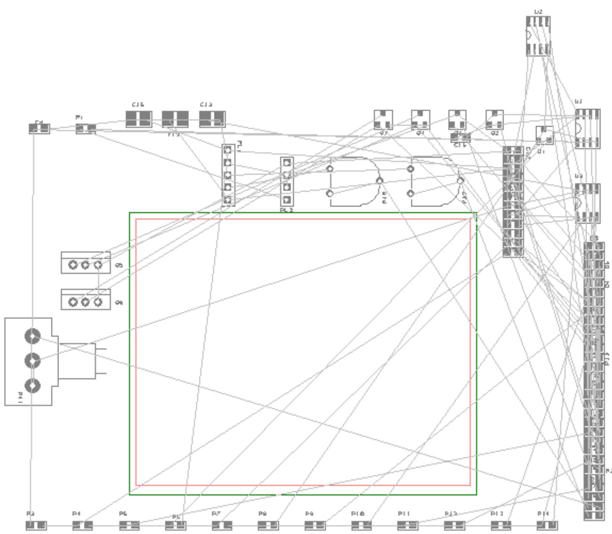
Place Around Board From Bin

From the **Component Bin**, there is a new command on the context menu called **Place Around Board From Bin.**



This feature is already available with the existing **Arrange Components** dialog on the **Tools** menu but this allows you to make the placement directly from the Bin without further prompting or selection of options for placement. It will however use the settings on the main Arrange Component, including **Component Stacking Above or Around the Board Area**.

The result will look like this:



Save Copy As

There is a new command, **Save Copy As** on the **File** menu for **Designs**, **Symbols**, **Technology Files**, but not **Projects**.

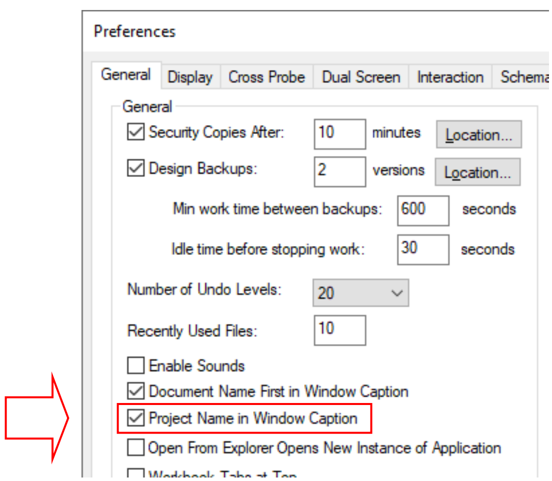
This command will prompt you with a standard file **Save** dialog, from where you specify the path where you want the copy of the current file to be saved.

Note, if the file is currently part of a Project then it will be disconnected from that project, to prevent issues with the new and original design files being linked back to the same project.

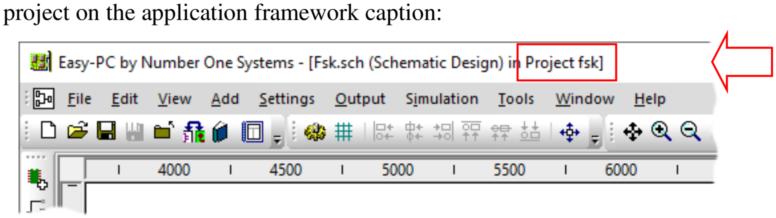
Project Improvements

Project Name shows on Window Caption

There is a new check box on the **Settings** menu under **Preferences**, **General**, named **Project Name in Window Caption**.



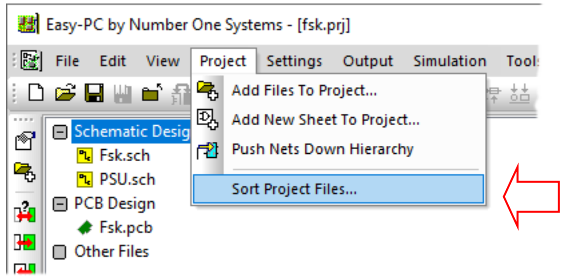
If checked (the default setting), it will include the name of the project for any design files that belong to the project on the application framework caption:



Sort Project Files

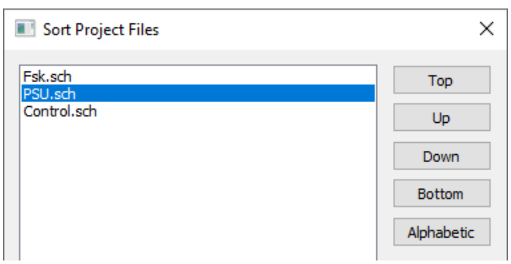
Uses Project Menu

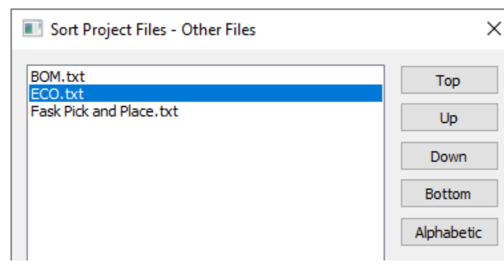
The new option, **Sort Project Files**, is available on the **Project** menu of a Project.



Sort Other Files

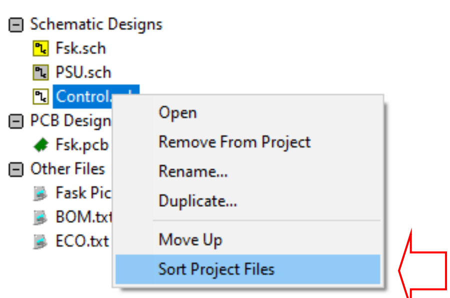
To sort **Schematic** sheets and **Other Files**, select a file in the **Project** tree view prior to using the option from the menu or context menu.





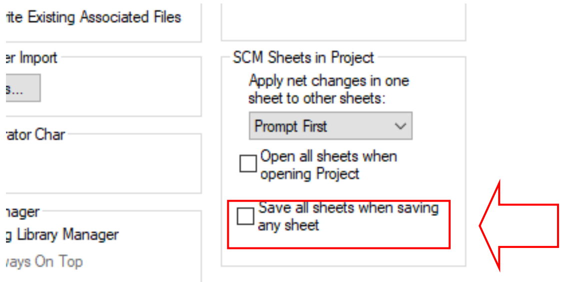
The functionality on this dialog enable file list sorting to be performed manually using the **Up** and **Down** buttons and to promote a file to the **Top** or **Bottom** of the list. The list can also be sorted **Alphabetically** using the button available.

This feature is also available by right-clicking on a selected Schematic in the tree view of a Project.



Save Project Sheet Saves All Sheets

From the **Settings** menu and **Preferences** option, **General** page, there is a new check box, **Save all sheets when saving any sheet**. If checked (the default setting is unchecked), when you save any **Schematic** that belongs to a **Project**, all Schematics in the Project will be saved.



Save Project Libraries

There is a new command on the **File** menu named **Save Project Libraries**. This is very useful if you want to create a Project or 'design' based library set.

When selected, this will create a new **Component**, **PCB Symbol** and **Schematic Symbol library** file, and copy all the library items to those libraries from every design in the project.

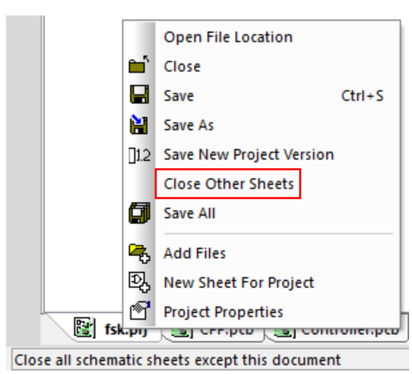
If run from a Schematic or PCB design (if the design does not belong to a Project and its schematic/PCB 'pair' cannot be found), it will only create the Component library plus the appropriate symbol library (Schematic or PCB Footprint). For example, this option will not create Schematic .ssl library if running from a PCB design

When selected, this option will prompt you to provide the name of the library set in which to save each library type and the location of the new library files. This will default to the same location as the Project itself.

It will also ask if the path is to be added to libraries folders if it is not already in there.

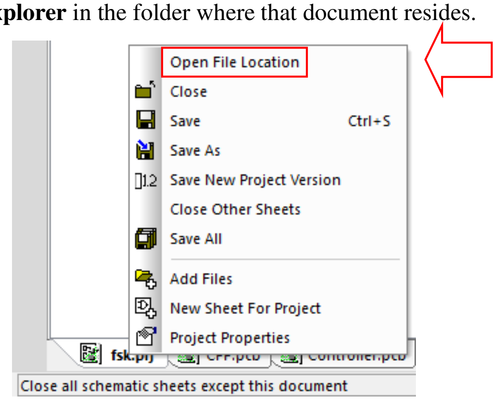
Close Other Sheets

There is a new context menu item available on workbook tabs for a Project and Schematic sheets. This command allows you to close all the open sheets for a Project (when current document is the Project), or all sheets other than the current sheet (where the current document is a Schematic sheet).



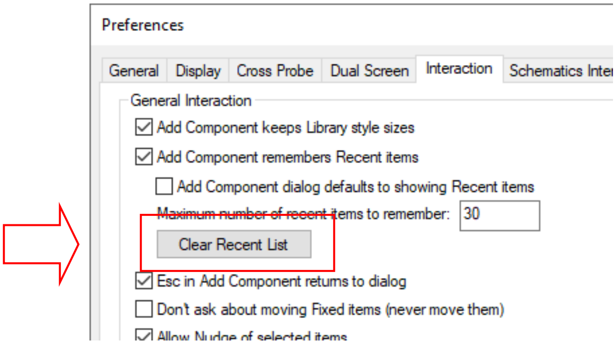
Open File Location

A new context menu item is available on a workbook tab for all documents. This opens the **Windows file Explorer** in the folder where that document resides.

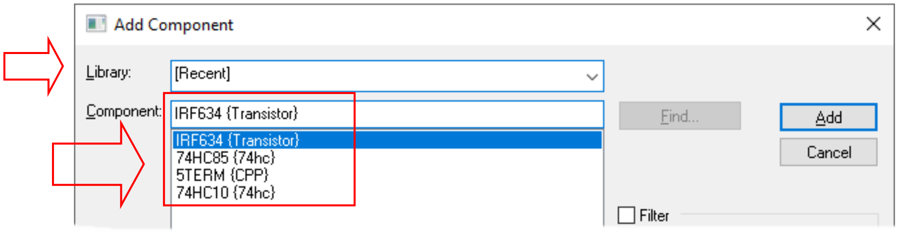


Clear Recent List from Add Component

There is a new button on the **Settings** menu under **Preferences** and **Interaction**, use to **Clear Recent List** from the **Add Component** dialog. This will empty the list of remembered recent items (on the Add Component dialog) and start the list again.

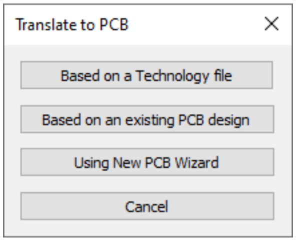


The **Add Component, Recent** list will be cleared:



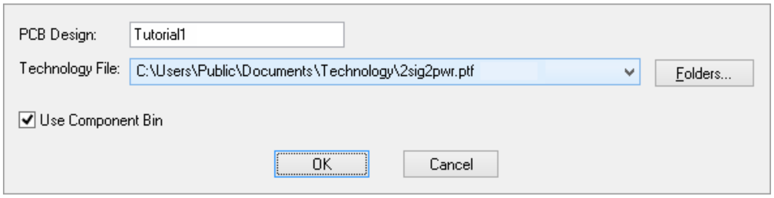
Translate to PCB with additional options

When choosing **Translate To PCB** from a Schematic or Project, you now have three options for creating your new PCB design:



Based on a Technology File

The first opens the existing dialog that allows you to choose a **Technology File** and **Use** the **Component Bin**.



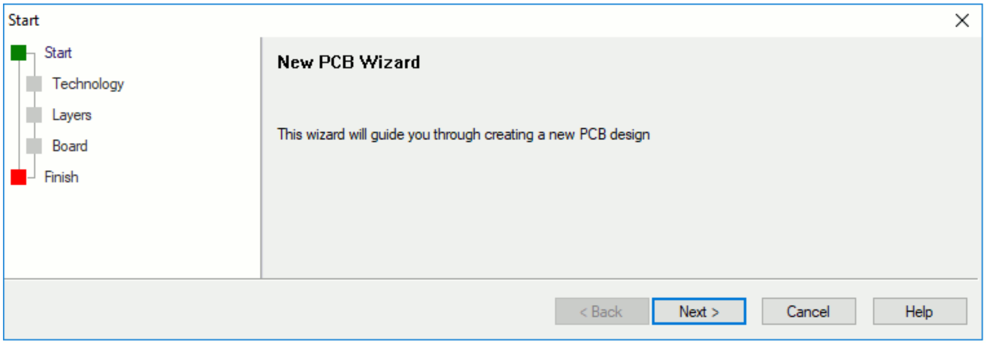
Based on an existing PCB Design

The second allows you to find an **existing PCB Design**, from which it will copy the technology and related information as well as any board outlines, free copper, areas, text, etc. On selection of this option, you will select the PCB design from the **Open** dialog.

This will have the same result as if the other design had been saved with **File**, **Save Technology File**, then the new design was created using this but without the need to create a technology file.

Using New PCB Wizard

The third option enables you to invoke the **New PCB Wizard** to create the new PCB. Here you can access a Technology file, choose various options for your layers, and define a basic board outline.

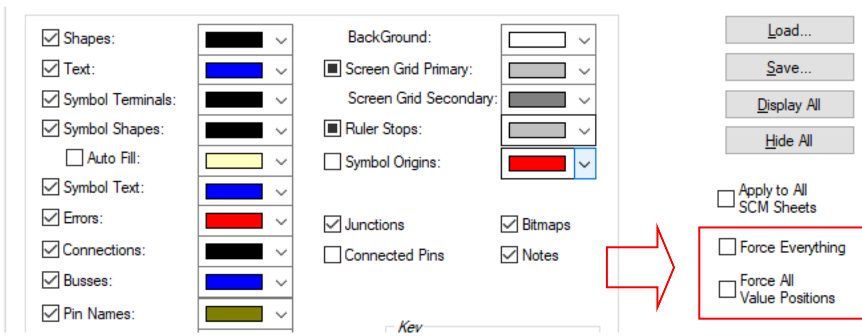


Colours – Force Everything On & Force All Value Positions

Two new switches are available in the **Colours** dialog, under **Settings & Highlights**:

- **Force Everything** on
- **Force All Value Positions** on

These switches work at a high level so they override all other switches in the Colours dialog. However, once used and set unchecked again, the previous colour settings are totally restored.



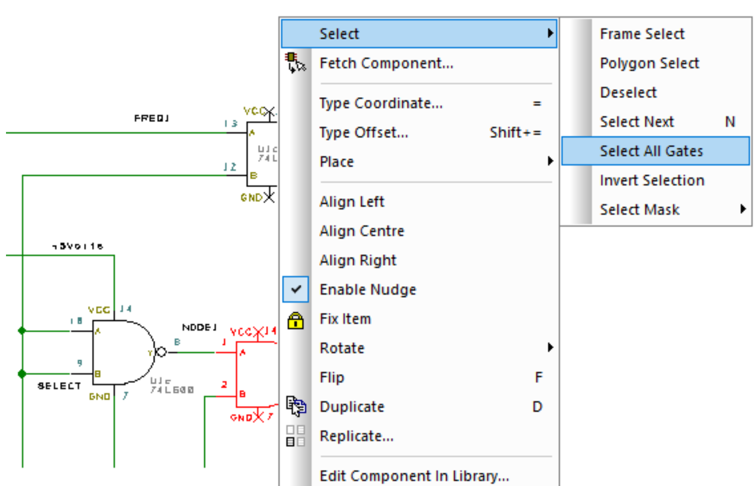
Use the **Force Everything** check box to force everything on so that it is visible, true size and selectable. This will be everything except Values which has it own check box.

Use the **Force All Value Positions** check box to force all Values in the design to be displayed.

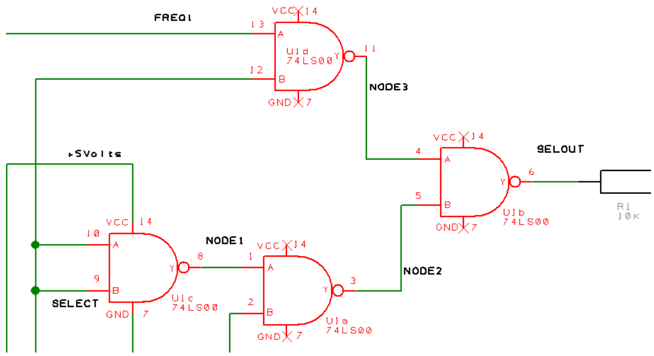
*Note, Because the installer respects menu customisation, upgrade installations from some older versions may still show **View**, **Display** rather than **View**, **Colours**.*

Select All Gates

If you are in a Schematic design, and have a single item selected that is all or part of a multi-gate Component, by right clicking the mouse, from the **Select >** sub-menu, there is a new command named **Select All Gates.** When selected, it will select all of the Gates on this Component.



Selecting it, will select all the gates on the same Component:



Copy Shapes Between Different Design Types

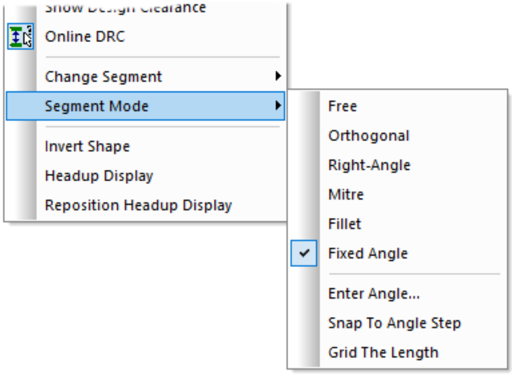
You can now copy a shape between two designs of different types (for example, between Schematic and PCB designs). The shape will be added using the styles and layers from shape defaults in the target design.

For example, you might define a drawing blank as a Schematic Symbol, this can now be selected and copied into the Footprint editor where it can be saved for use (with a Component) in a PCB design.

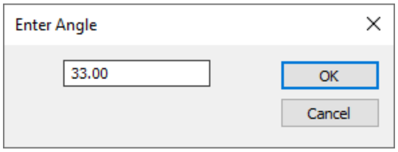
New Segment Mode for Shape/Track Editing

New Segment Modes have been added for use when editing Shapes or Tracks.

When editing a shape/track, from the context menu option **Segment Mode>,** you can now select some new modes:



Fixed Angle – this segment mode can be used for adding single segments at a supplied angle. The default will be to add single segments at 45 degrees. Selecting this mode again from the menu returns it to the previous segment mode you were using. Use **Enter Angle** from the context menu to change to the required angle, you will then be able to add segments at this angle and at 45 degree rotation increments from that angle.



Select **Snap to Angle Step** from the context menu to snap to steps equal to the angle originally supplied. For example if the angle supplied was 15 degrees then you will be able to add segments at multiples of 15 degrees.

Select **Grid the Length** from the context menu to grid the length of the angled segments being added using the current grid value. For example, if your working grid is set to 100, then the length of the segment in this mode will be 100. If this feature is not selected the line ends will snap orthogonally to the nearest grid point.

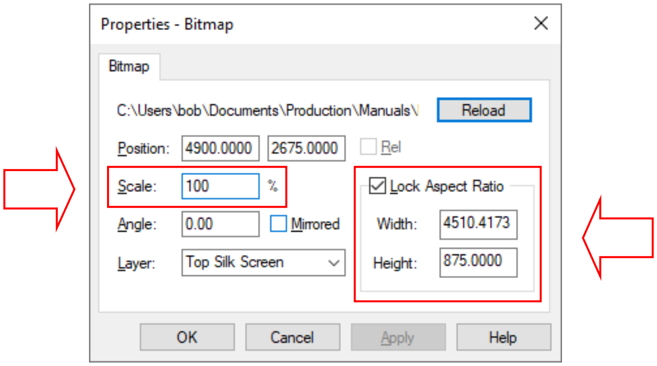
Bitmap Properties Scale and Aspect Ratio

The **Bitmap Properties** dialog for bitmaps has been updated.

A **scale box** has been added which can be used to adjust the scale of the bitmap based on its original size.

A check box to lock the **Aspect Ratio** has been added.

Both **Scale** and also interactively **dragging** a corner to resize (see below) will maintain the **Aspect Ratio** if this button is checked on.



Drag to Resize Bitmap

Bitmaps can now be easily resized in the design by either dragging and edge or a corner.

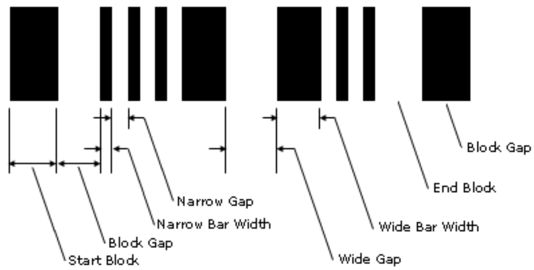
To move a corner, pick and drag it with nothing else selected in the design.

The aspect ratio of a bitmap can be locked in **Bitmap Properties** so that when a corner is dragged the aspect ratio of the image is preserved.

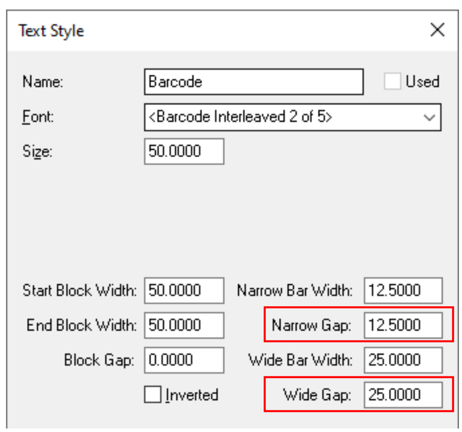
Previously bitmap sizes had to be changed in the Properties dialog.

Barcode Line Gap Additions

The **Narrow Gap** and **Wide Gap** parameters can now be specified. These are used to add extra definition to the barcode format.

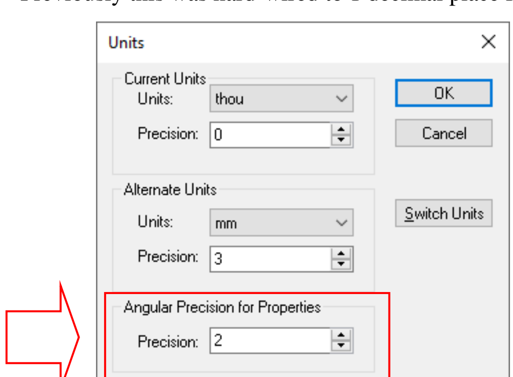


The dialog has also been rearranged to be more logical in its layout and the **Underlined** check box removed for barcodes.

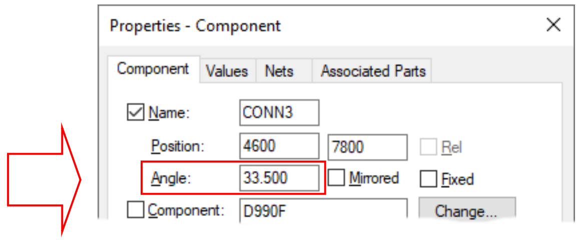


Set Angular Precision in Units dialog

On the **Units** dialog for a design, in addition to being able to set up two sets of units, you can now also specify the **Angular Precision** for display of angles in options such as **Properties** and **Measure**. Previously this was hard-wired to 1 decimal place for Measure and 2 decimal places for Properties.



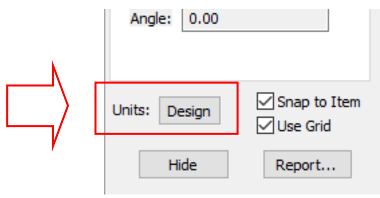
On **Properties**, this shows as:



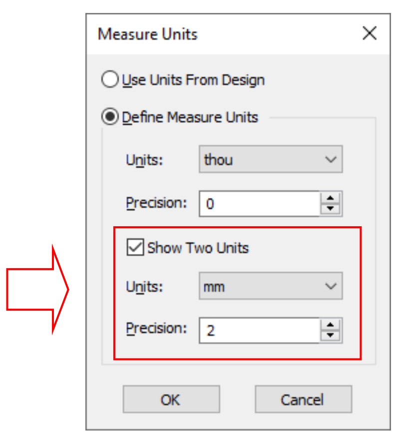
Measure – Dual Units

The **Measure** dialog/panel can now display lengths/areas in two different units.

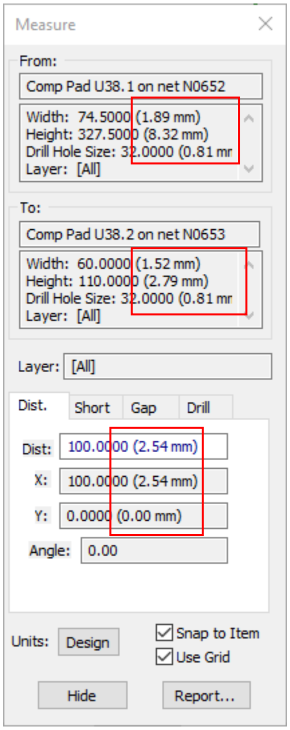
From the main **Measure** dialog, click on the **Units:** button (the one that shows the current units) to access the **Measure Units** dialog.



This dialog now has an extra check box **Show Two Units** and a second pair of controls to let you set up the secondary units.



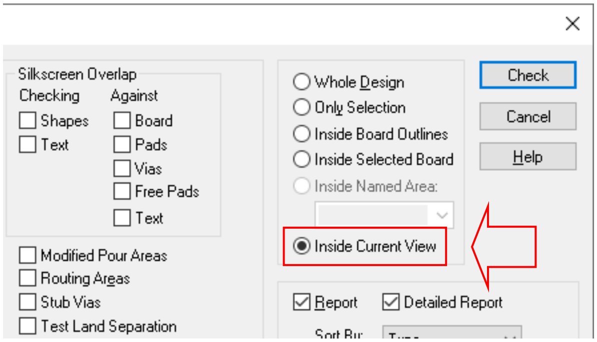
Once this is set up, any measurements you do will be displayed in both sets of units. The second unit set being shown in brackets ( ) on the Measure dialog:



Design Rule Check

DRC Within Current View

There is an additional button on DRC dialog to allow checking within current view.

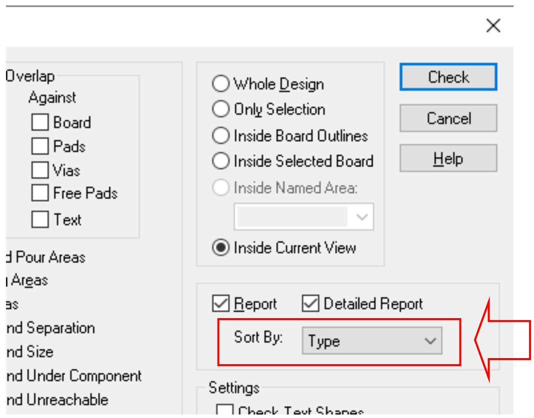


Note that in common with some of the other settings (e.g. Inside named area) some of the checks still apply to the whole board, so 'check where' largely applies to the Spacing checks (as it did in previous versions).

Sort Report by Layer or Type

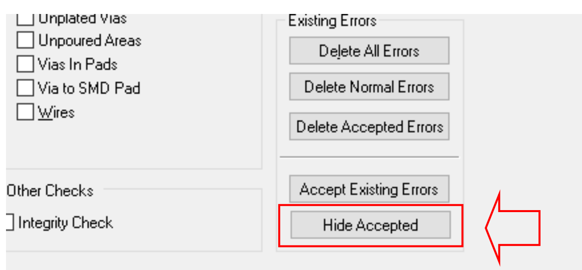
The **DRC report** (either generated from the **DRC** dialog or later when run from the **Output** menu and **Reports**) will now be sorted into order.

You can choose on the DRC dialog to sort by **Layer** or by **Type**.



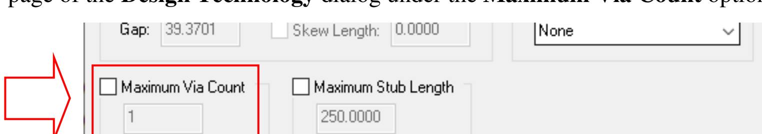
Show/Hide Accepted Errors

A new button on the DRC dialog enables you to toggle between **Showing** or **Hiding Accepted** errors in the design.

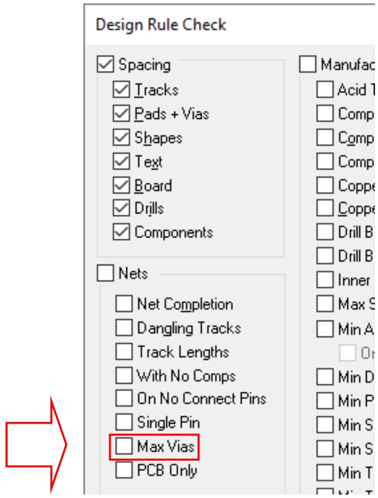


Nets – Max Vias

This is a new check that can be used to ensure that certain nets do not exceed a required limit on the number of vias. The parameters are defined on a Net Class and can be edited through the **Net Classes** page of the **Design Technology** dialog under the **Maximum Via Count** option.



The check box **Max Vias** under heading **Nets** on the **DRC** dialog is used to enable this check.



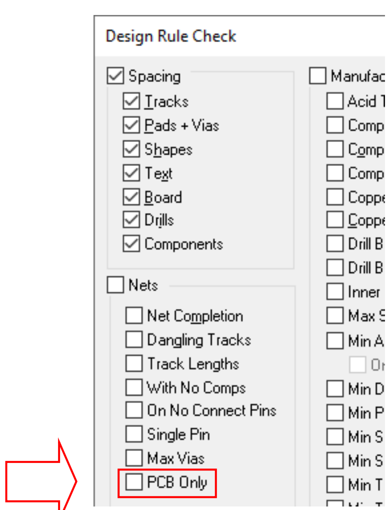
When checked, any net that has more vias than are allowed by its Net Class will have DRC error markers **MV** added at the position of the offending vias. Note that because there is no way to put vias on a net in a particular order, the error markers will be added to the excess vias in the order they happen to be defined in the design data.

Design Rule Check – PCB Only nets

There is a new DRC check for nets that are deemed **pcb only**. These are nets that are **not** connected to **any** component pin. Note that pins on PCB-only components are ignored when looking for valid component pins in the net. This check does not include free pads or vias.

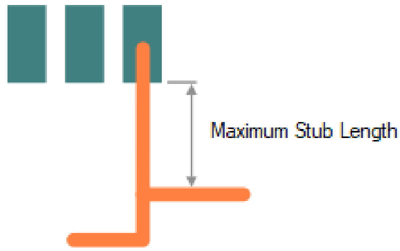
This check might be useful if you have free pads (as mounting holes) that are connected together but not to a component for example.

The check box **PCB Only** under heading **Nets** on the **DRC** dialog is used to enable this check.

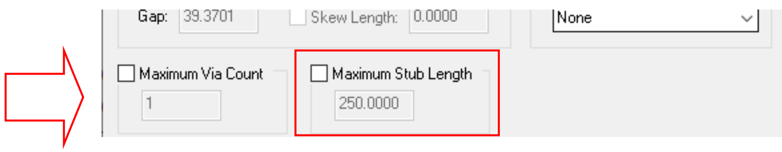


Max Stub Length

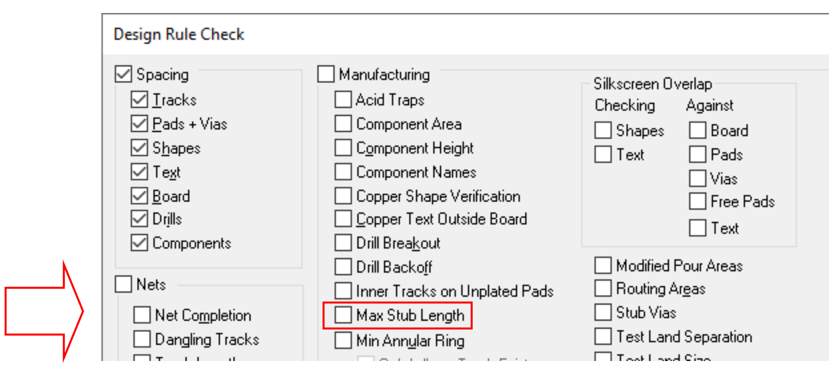
There is a new DRC check that looks at the distance from the edge of a component pad to the next 'node' (pad, via or junction), this is classed as a 'stub' route.



The parameters are defined on a Net Class and can be edited through the **Net Classes** page of the **Design Technology** dialog under the **Maximum Stub Length** option.



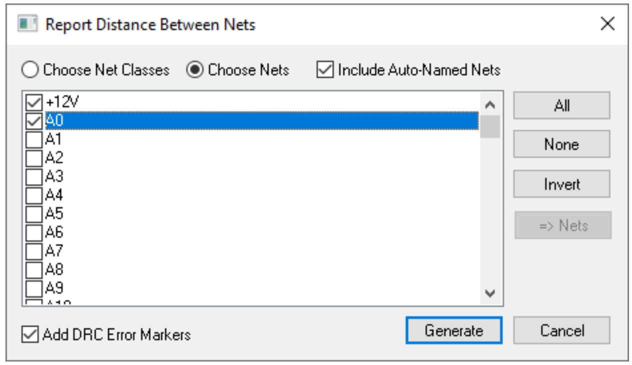
The check box **Max Stub Length** under heading **Manufacturing** on the **DRC** dialog is used to enable this check.



New Report - 'Net Distances'

Available from the **Output** menu, **Report** option, the new **Net Distances** report enables you to choose two or more nets from the PCB design and the program will calculate the minimum distance between each pair of nets. It will also indicate where that smallest distance occurs by use of optional error markers.

Once run, the **Report Distance Between Nets** dialog is displayed:



Options allow you to select the **Net Class** or **Net Name** and to select the nets to be checked.

The **Add DRC Error Markers** button can be ticked to cause the application to add a DRC error of type "ND" (Net Distance) for the location where each pair of nets comes closest together. Although these are not strictly errors, it does allow you to use the other facilities of the application (such as the Goto bar) to locate the places where the report indicates each pair of nets are closest together.

Note: each time you run this report, all existing "ND" errors will be deleted from the design, regardless of which nets or net classes are chosen in each run.

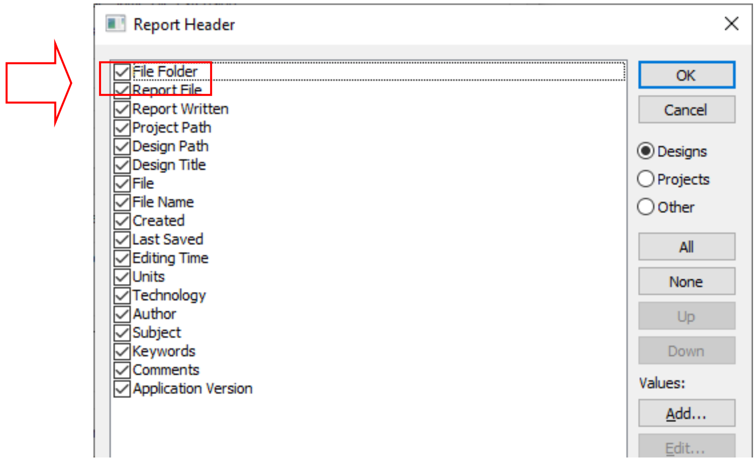
The report generated will look like this:

Net 1	Net 2	Distance	Layer	Item 1	Item 2	Location
+12V						
	-12V	12.5000	Bottom Electrical	Track	Track	1687.5000,5887.5000 to 1687.5000,5887.5000
	GND	12.5000	Bottom Electrical	Track	Track	1687.5000,6725.0000 to 1687.5000,6725.0000
	VCC	11.5000	Bottom Electrical	Track	CONN3.1	4325.0000,7800.0000 to 4325.0000,7800.0000
-12V						
	+12V	12.5000	Bottom Electrical	Track	Track	1650.0000,5887.5000 to 1650.0000,5887.5000
	GND	38.0000	[All]	Track	CONN3.9	4689.0000,7900.0000 to 4689.0000,7900.0000
	VCC	107.5000	Bottom Electrical	Track	C47.2	1650.0000,6100.0000 to 1650.0000,6100.0000
GND						
	+12V	12.5000	Bottom Electrical	Track	Track	1725.0000,6725.0000 to 1725.0000,6725.0000
	-12V	38.0000	[All]	Track	CONN3.8	4727.0000,7900.0000 to 4727.0000,7900.0000
	VCC	12.0000	Top Electrical	Track	Track	1770.0000,6500.0000 to 1770.0000,6500.0000

New Report Headers

File Folder Command

An extra **File Folder** command to output just the last folder name of the file path has been added to the **Settings** menu, **Preferences** dialog under **General** and **Report Headers**.



Report Header – Date Commands now also show Time

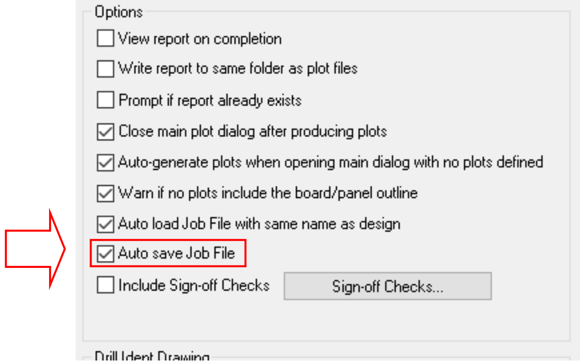
In **Report Headers**, where a **date** command is included (such as **Creation Date**, **Last Saved** or **Date Written**), this will now also show the **time**.

```
Created:        23/06/2003 17:46:17
Last Saved:     27/07/2023 14:32:12
```

Plotting

Auto Save Job File

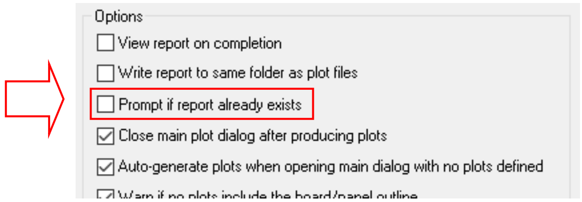
There is a new check box on the **Plotting and Printing** dialog, under **Options**, **Auto Save Job File**, which gives you the ability to disable/enable automatic saving of the job file on exit from the dialog.



Plot Report Append/Overwrite

From the **Plotting and Printing** dialog, when plots are generated, if the plot report already exists then you have the option of being prompted about overwriting the existing report or appending to it (or even cancelling opening the report). This is in addition to options already available to compose the plot report folder/file with things like a timestamp to get a separate report each time.

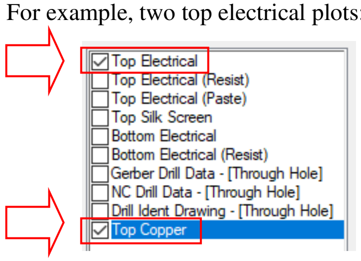
To enable this prompt, go to **Plotting and Printing** dialog, **Options** and select the **Prompt if report already exists** option.



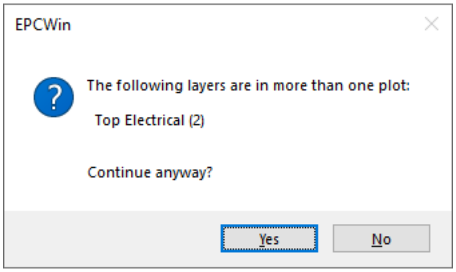
Pre Plot Checks

When running plots from the **Plotting and Printing** dialog, the layers selected for each plot will be checked, and any layers that appear in more than one plot will be highlighted for you to confirm before carrying on with processing. This is to help avoid accidentally selecting the wrong layers when setting up the plots.

For example, two top electrical plots:



This will be warned to you like this:



Note that this check will ignore any layers that are not single layers (e.g. [Through Hole]). It will also ignore 'pads only' plots as these will by their very nature involve outputting the same layer (e.g. Top) as it already used by a normal electrical plot.

New & Modified Library Content

Changes have been made to the standard library installation shipped with Easy-PC Version 27:

Existing PCB outline templates have been removed from the discrete footprint library.

PCB only drawing borders have been added and are included in a new library, **Frames.cml**. Drawing borders are already available for Schematics in the Schema library.

A change has been made to the PCB footprint library sm.psl. All footprints have been edited so that they are all in the same, horizontal, orientation. Previously, some were vertically orientated.