Easy-PC

V22.0 Supplement

Number One Systems
20 Miller Court
Severn Drive
Tewkesbury Business Park
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.

Manual date: 25-July-2018 Issue 2

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

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

If you have been provided with download link, use this to download the installation executable, named EasyPC.exe. Find it in your *Downloads* folder using Windows Explorer 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.

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

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.

With the installer running, once the **Welcome** screen is displayed, double-click on the **Install Easy-PC - Version 22.0** 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 22.0

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 10.

Chapter 2. New Features in Easy-PC V22

Introduction

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

Library Integration with the Component Search Engine

Easy-PC Version 22 will be delivered with free access to the Component Search Engine.

The **Component Search Engine** web site enables you to search Components (along with Schematic and PCB Symbols) and download them to your Easy-PC application ready to use. Not only do they appear on the end of your cursor but are also automatically added to your library to use on another design.

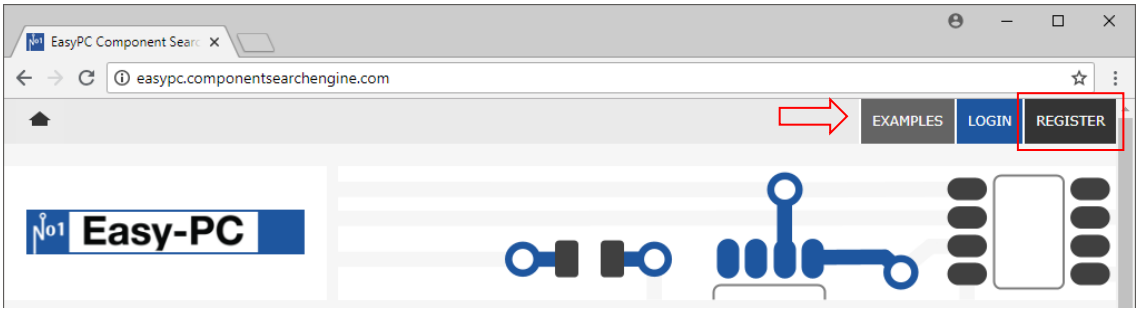
Product Version/Build Requirements

You will need to be running Easy-PC 22.0 to benefit from the **Component Search Engine** integration. This feature will only ever run on the latest major version of Easy-PC.

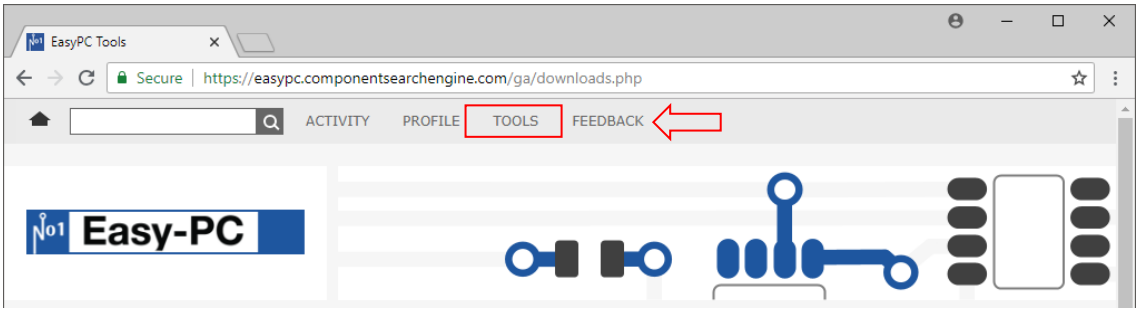
Registering with Component Search Engine

To use the Component Search Engine, you will first need to register an account.

Go here - https://easypc.componentsearchengine.com/ and click **Register.**

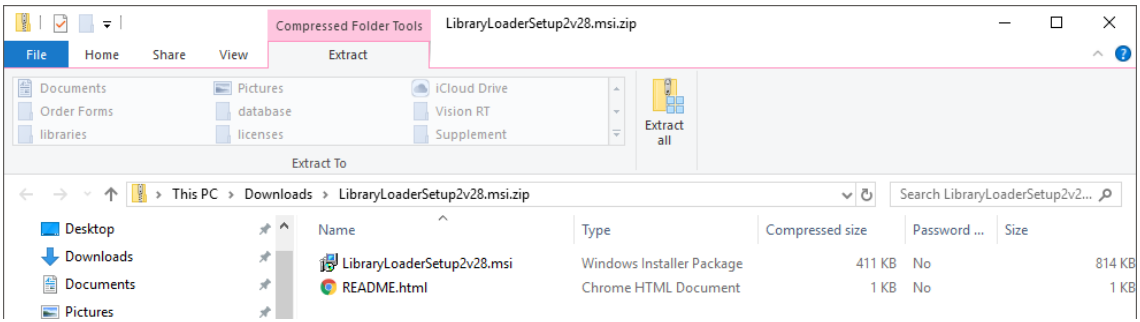


On the **Tools** page, scroll down to the **Library Loader** section.



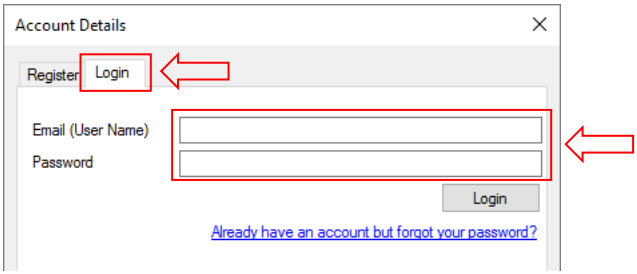
Click the **Download Library Loader** button to start the download.

Once downloaded, run the **LibraryLoaderSetup.msi** program from your download folder.



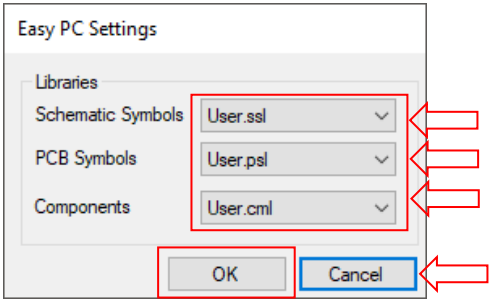
Once the **Library Loader** has been successfully installed, open the **Library Loader** from your Windows **Start** menu.

You will be asked to either **Register** or **Login**. As you have already registered you can select the **Login** Tab and **Login** using your **User Name** and **Password**.



Once logged in, select **Your ECAD Tool** as **Easy-PC** using the drop-down list box. Now select the **Settings** button next to it.

From within the **Settings** dialog you will now be able to set the destination for where new Schematic Symbols, PCB Symbols and Components will be saved to.



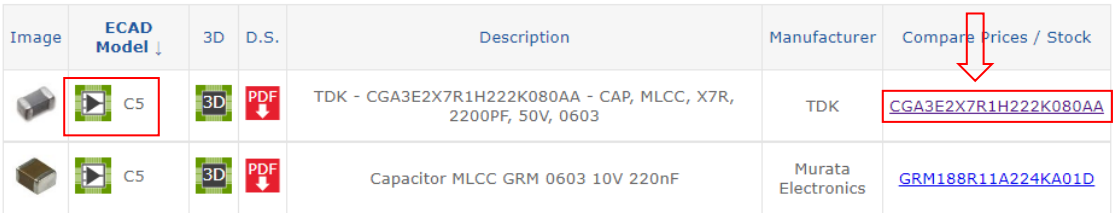
Now you've set up the Library Loader you can now download Components from the Easy-PC Component Search Engine web site and they will load directly into your libraries and onto the end of your cursor ready for placement in a design.

Using the Easy-PC Component Search Engine

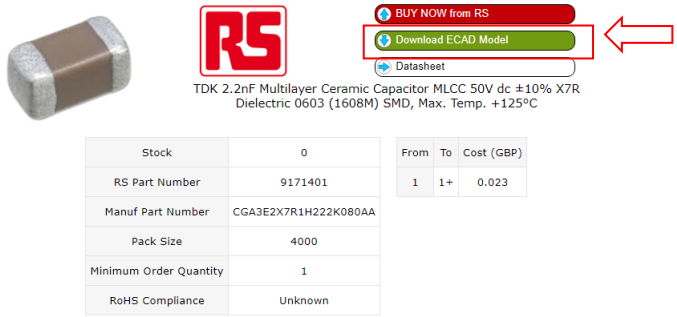
From the **Easy-PC Component Search Engine** website (https://easypc.componentsearchengine.com/), enter the name of your Component or a description of

it in the **Search** section. Once the search is complete you will see six columns presented including an ECAD Model, Datasheet and Pricing/Stock information for the Component from various suppliers.

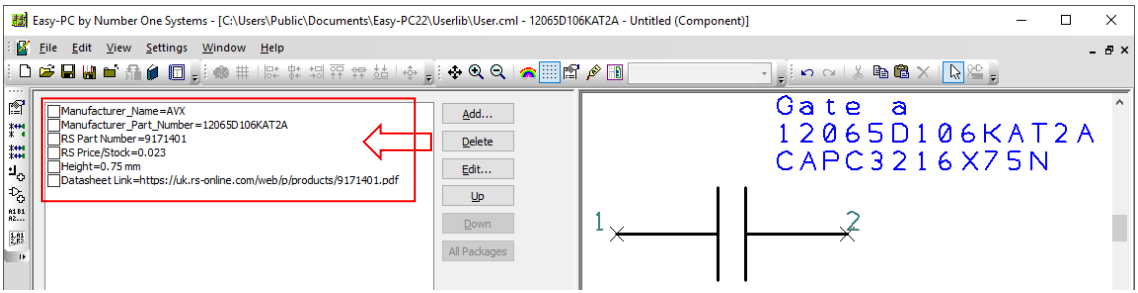
Clicking the **Price/Stock** link will allow you to select and download the Part with pre-populated attributes from a specific supplier.



Clicking the **Download ECAD Model** button will download the model from the website into your libraries and onto the end of your cursor ready for placement in the design. Attributes are automatically added to the Part depending on the section you download it from, therefore if we click **Download ECAD model** from RS Components, we would then also see the RS Part Number on the Part in Easy-PC.



Now the Component has been downloaded, you can investigate its values from the **Edit Component** dialog in Easy-PC under the **Library Manager** to see it has inherited the values from the online model, including the Supplier name and Supplier Part number.

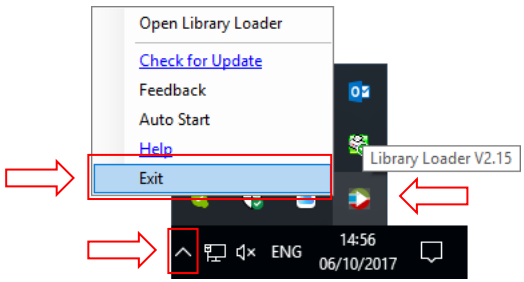


Updating Imported Parts to Company Standards

It is always worth remembering that imported parts may need to be updated to your company standards. For example, this may mean adding attributes to Components or changing text, pin and line styles on Symbols and Footprints to be in-line with your other library items.

Closing the Library Loader Application

As the Library Loader will run in the background, you will need to close the background application if you wish to close it completely. To do this, select the **up arrow** on your **taskbar**, select the **Library Loader** icon and from the menu displayed, click **Exit**.



Supported Web Browsers

The following browsers are support by the Component Search engine integration:

- Google Chrome
- Mozilla Firefox
- Microsoft Edge
- Apple Safari

*Note: **Internet Explorer** is not supported for the full integrated searching and download capability.*

Associated Parts

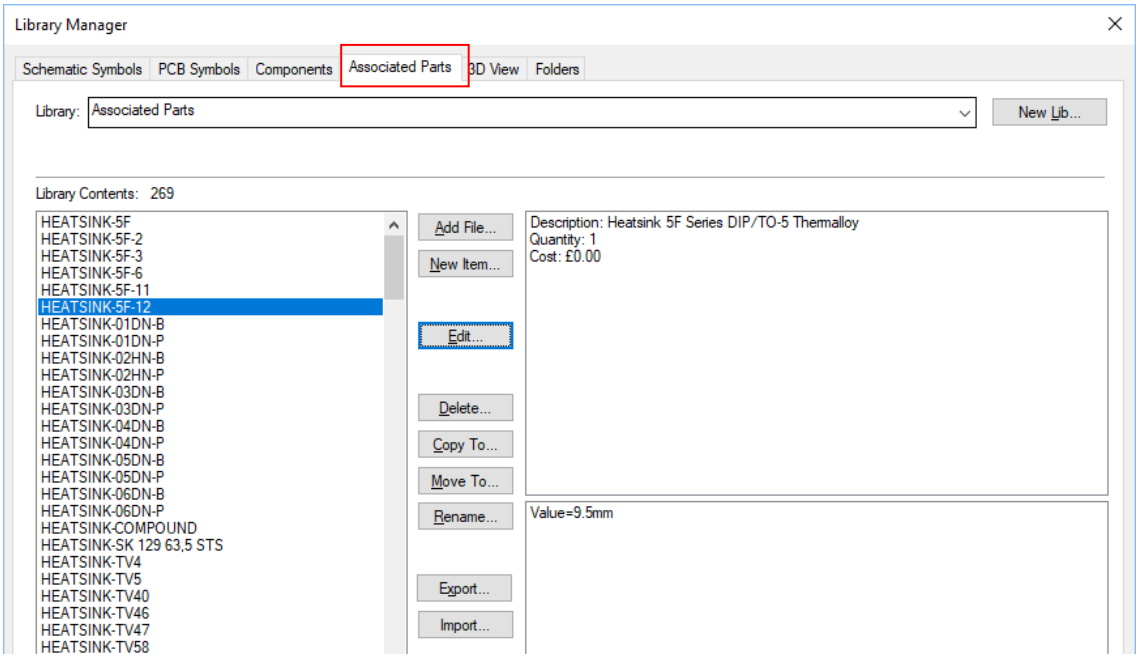
There is a new type of Library item introduced for version 22 called an **Associated Part**. This is used when a component requires additional items to be provided on the Bill Of Materials (BOM) but where you do not wish them to physically appear in the design. The additional items attached to a TO-220 transistor for example, would need to include the heatsink, nut, bolt, nylon washer and thermal paste.

There are a number of mechanisms for associating a Part with a Component; create a new list of **Associated Parts** in their own library and add these to the Component that requires it, or add an Associated Part by typing in its details on the fly when editing a Component. The advantage of using

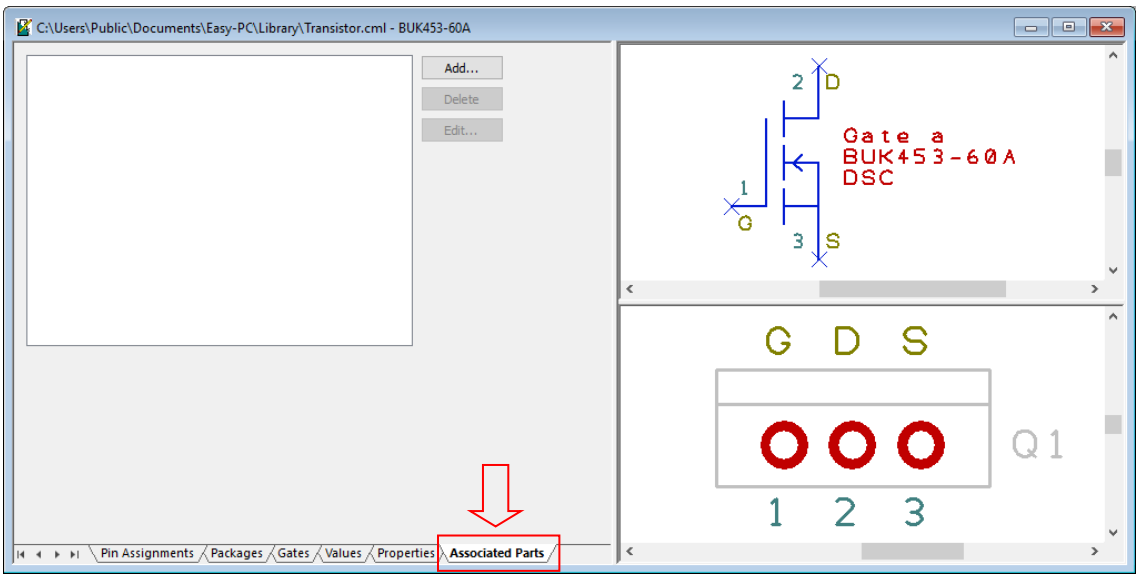
an Associated Part from a library is that it will be more consistent and can be used again for multiple Components. You can also add Associated Part information to a Component in the design using its **Properties** or you can add design-level Associated Parts, these might be a 'higher' level than a Component, for the board construction for example, like specifying FR4 or a board part number.

Associated Parts and Libraries

For library items, a new tab for **Associated Parts** has been added to the **Library Manager** dialog:

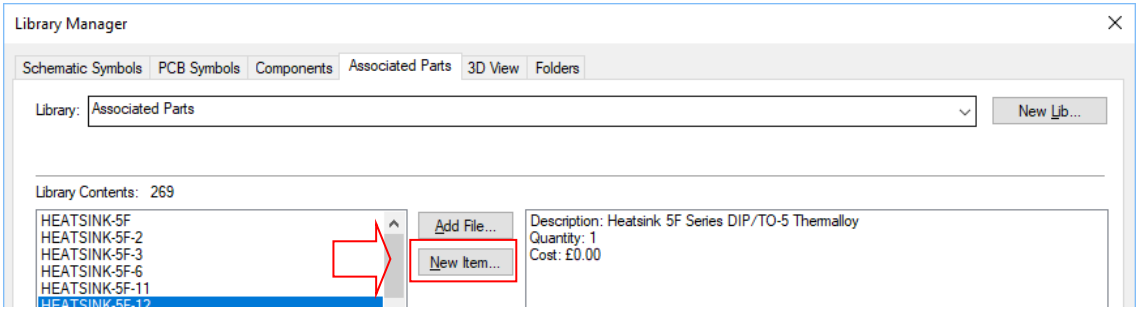


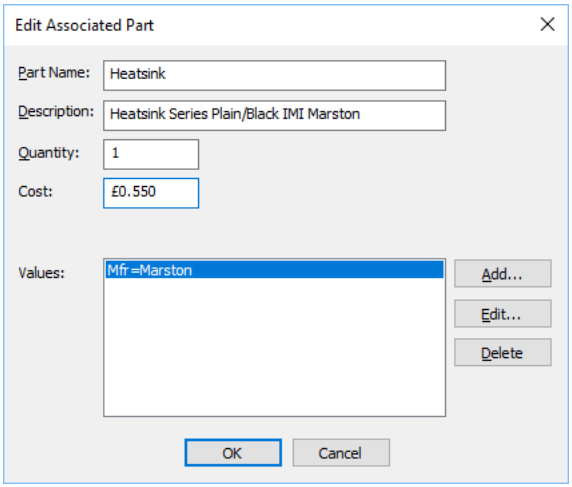
Once an **Associated Parts library** and **Associated Parts library item** have been created, this item can then be added to a **Component** using the **Associated Part** tab on the **Component Editor** dialog:



Creating New Associated Parts Libraries and Items

From the **Associated Parts** tab in the **Library Manager** select the **New Item** button:

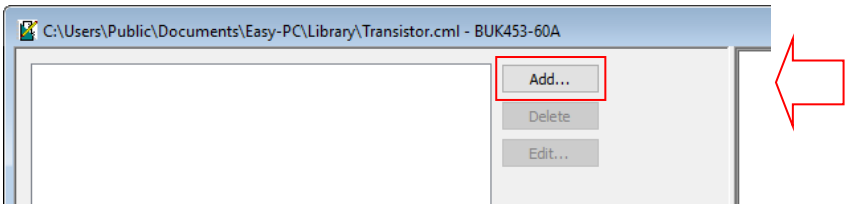




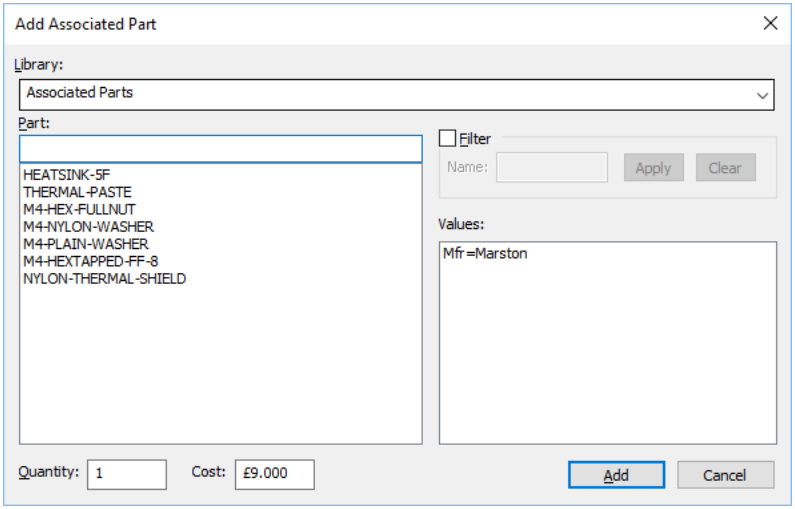
All information on this dialog is saved with the Associated Part and 'attached to the Component once associated.

Adding Associated Parts items to Components in the Library

From the **Associated Parts** tab of the **Component Editor**, use the **Add** button.



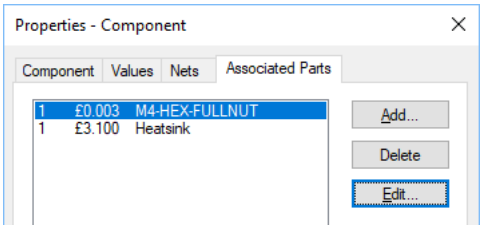
This opens the **Add Associated Part** dialog:



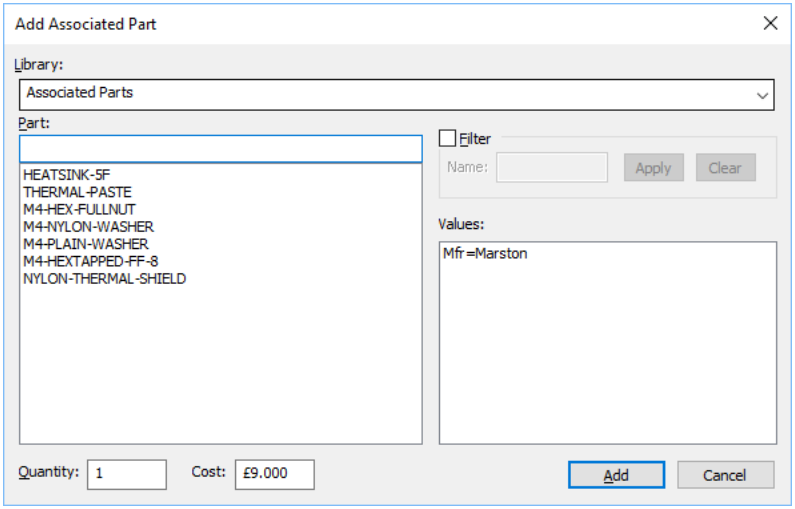
From here, you can add an Associated Part from an Associated Parts library, or you can type in the details directly. Either way is fine, however, if you wish to use the same Associated Part again, saving it into a library might save you time later on. It might also be more efficient for consistency too.

Adding Associated Parts items to Components in the Design

For a design that does not have Associated Part, these can be added using the **Component Properties** dialog and **Associated Parts** tab.



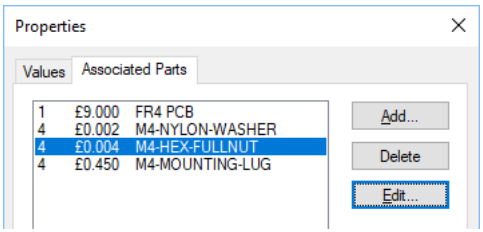
Use the **Add** button to add Associated Parts to the list.



Adding Design-Level Associated Parts

Associated Parts can be added to the design, for items such as the PCB itself, board mounts etc.

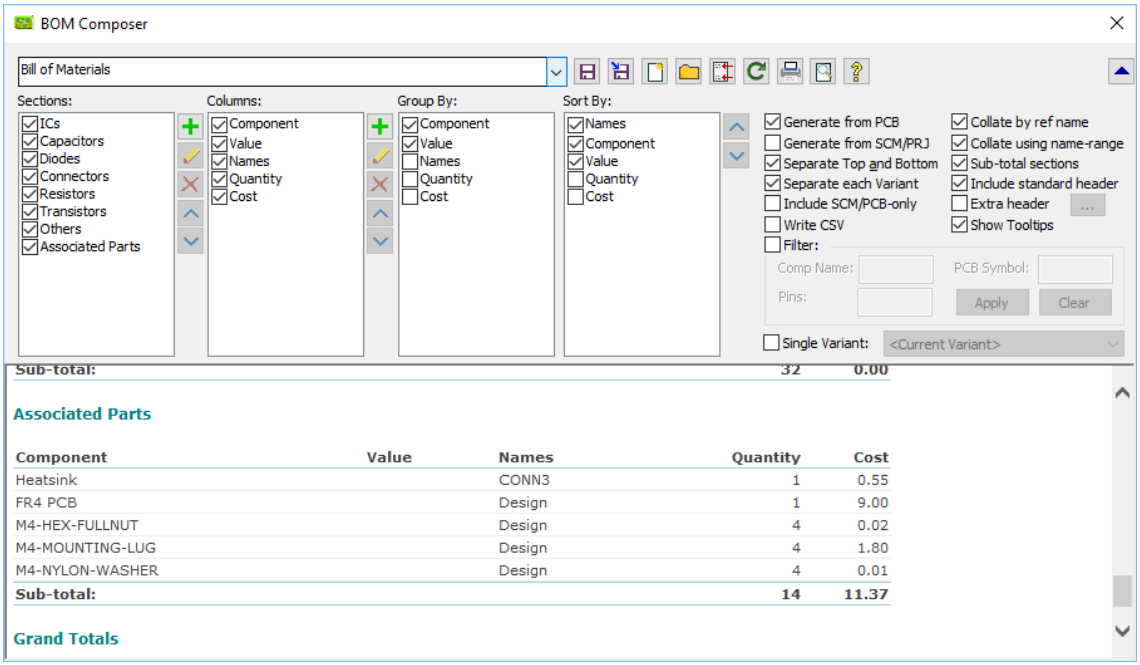
To add these, use design **Properties** from the **View** menu.



Easy-PC knows and reports the difference between Component Associated Parts and design-level Associated Parts.

Associated Parts

Once Associated Parts have been included in a design, they will appear as a separate section for Associated Parts in the BOM Composer report.

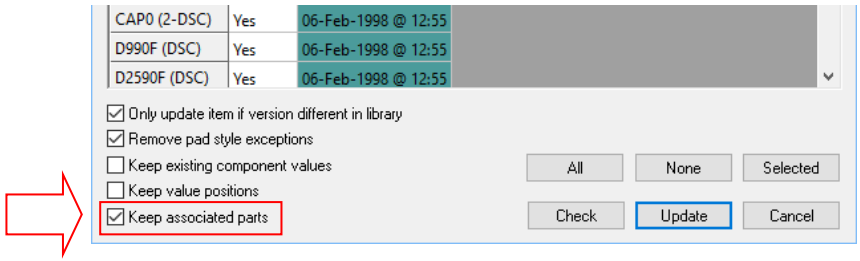


Using Associated Parts in the design

Other features within Easy-PC also understand how to handle the new Associated Parts:

Translate to PCB – Components added to the PCB will have instance-level Associated Parts copied from the Schematic.

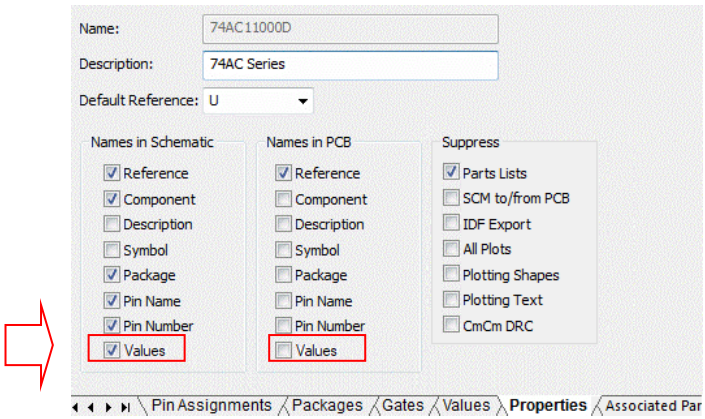
Update Component – when updating the design from the library, a new check box exists to **Keep Associated Parts.**



Duplicate/Copy/Paste – any Associated Parts defined on Component instances will also be copied across to the new Component.

Value Properties in Component Editor

For the **Properties** page in the **Component Editor**, you can now set the display of **Values** at the Component library level.

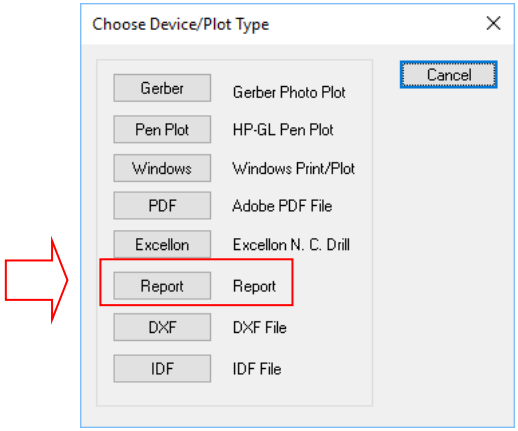


Changes to Plotting and Printing Dialog

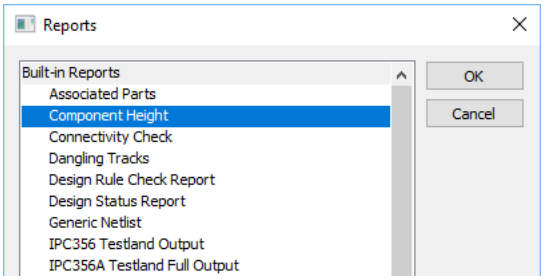
Reports in Plotting

The ability to specify **Reports** when generating a ‘set’ of plots has been added to the **Plotting and Printing** dialog.

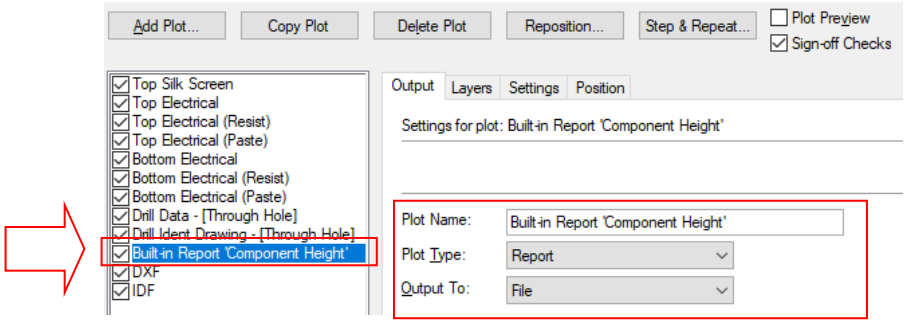
From within the **Add Plot** dialog, using the new **Report** button, you can now choose the report you wish to run when generating plots.



Selecting the **Report** button takes you to a version of the **Report Manager** dialog from where you select which report you want to run. The list includes built-in reports, user-defined reports and BOM composer outputs.

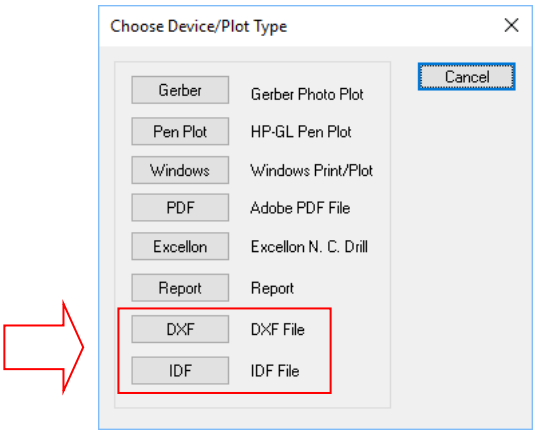


Once selected. The report appears in the main **Plotting and Printing** dialog. This setting is also saved into the **Plot Job** using **Save Job**.

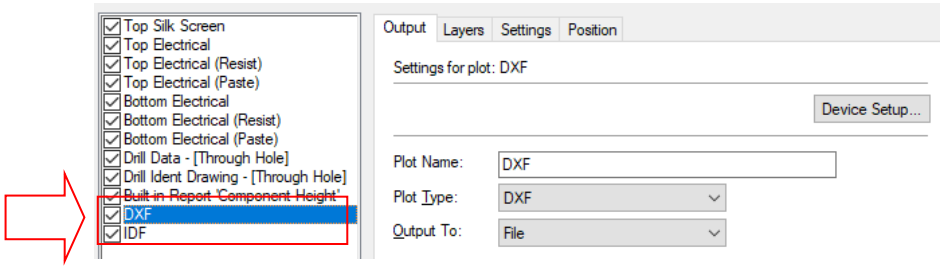


IDF and DXF in Plotting

You can now create plots that generate **DXF** and **IDF** outputs as part of the **Plotting and Printing** mechanism. This includes saving all the relevant settings from the IDF and DXF dialogs into a **Plot Job** for easy production of different outputs.



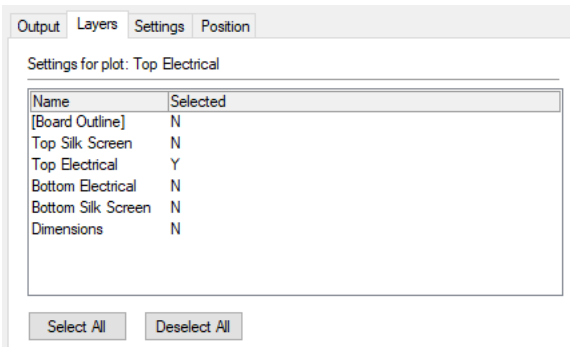
When either of these options is selected, it will simply add a plot into your plot list. Any settings defined in the **DXF** or **IDF** options from the **Output** menu will be used.



*Note: **IDF** Export is a cost option feature and will only appear on this dialog if you have purchased it and if it is enabled from **Settings, Optional Features**.*

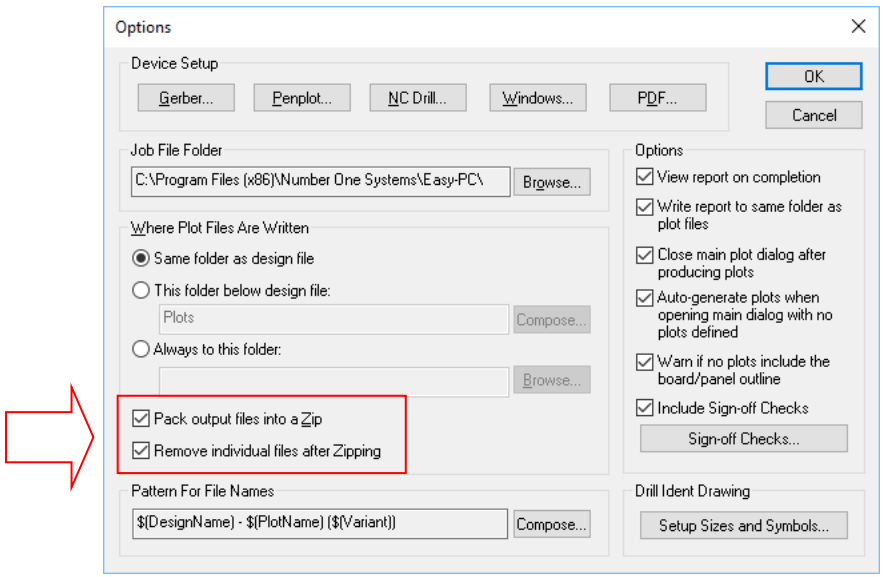
Plotting Layers – Select/Deselect All

There are new buttons for **Select All** and **Deselect All** on the **Layers** tab of the settings for a plot. Use these to quickly make selections.



Output Plots to Zip file

There are two new check boxes on the **Options** dialog of the Plotting dialog. This allows you to automatically save the generated output files (Gerber, NC Drill, Reports, etc.) directly into a **Zip** file, and optionally **remove** the individual files afterwards.



Pack output files into a Zip – output files are exported as normal, then saved into a Zip file. Without the Remove check (see below) selected, all the files will remain saved on your hard drive along with the Zip file also. The Zip file will be named *DesignName -.zip* following your file name pattern for plots.

Remove individual files after Zipping – if the check box to Zip the output files has been selected, selecting this option will then remove them from your hard drive. All plots will be removed and just the Zip file (containing the plots and reports) plus the Plot Report will remain.

Customisation of Plotting File and Folder Names

Version 22 introduces user-defined formatting of **Folder** and F**ile** names in the **Options** dialog of **Plotting and Printing**. This uses a ‘tag’ scheme with user-defined key words to format the output filename. This applies to the folder name when the folder is set to ‘this folder below design’, and to file names for each plot.

Tags can include Design Name, Plot, Plot Job and Project Name, Device Type, etc. To help you construct the tag pattern, syntax and creation, there is a new **Compose** dialog.

Where patterns are used

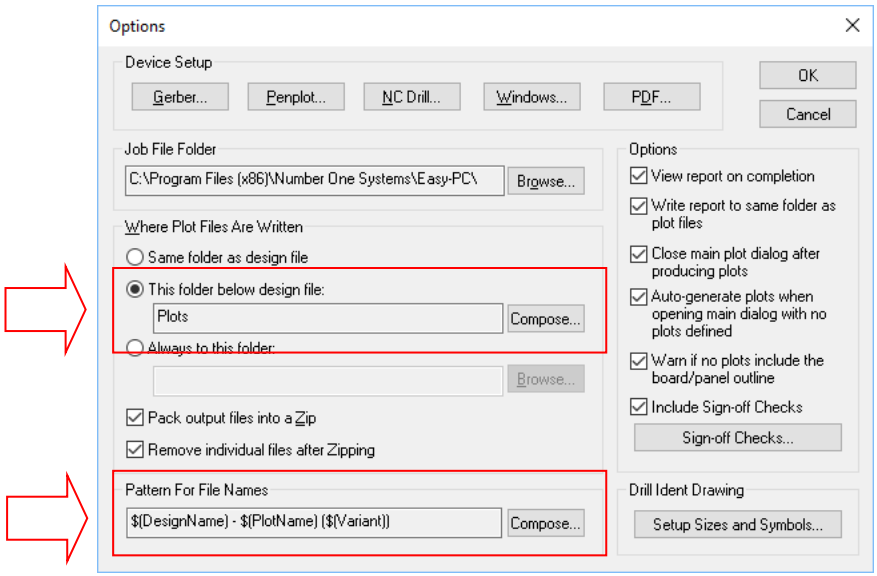
These patterns are used to construct the folder name for storing plots and other files generated during **Plotting and Printing**, and also to construct the actual names for these files. This allows you to apply some quite sophisticated rules to name and organise your output files.

When composing the pattern for the output folder, the default value is simply the text "Plots" which will place the output files in the Plots folder below the design file.

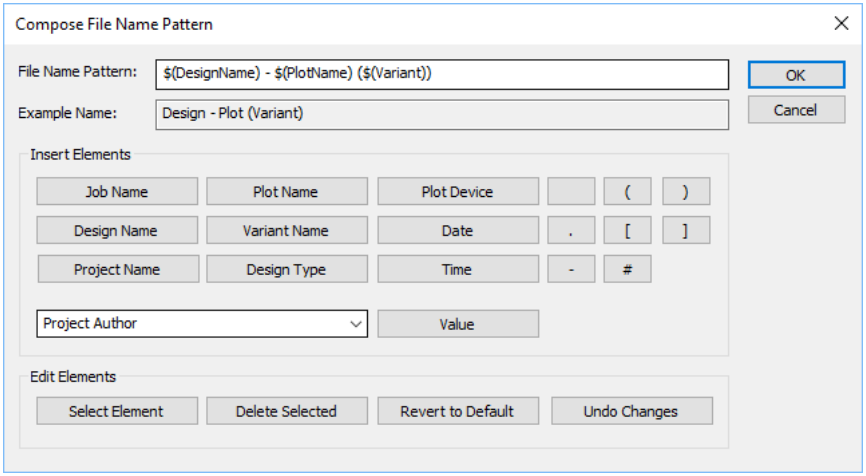
When composing the pattern for actual output files, the default value combines the design name, the plot name, and (if variants are used) the variant name.

You can use the controls on this dialog to alter the pattern to suit your preferences.

From the **Options** dialog, a defined pattern is displayed:



By selecting the **Compose** button, this reveals the **Compose File Name Pattern** dialog. From here, you can make choices that will customise your output filenames.



The **File Name Pattern** box on the dialog shows you the current pattern. These patterns are made up of 'tags' or 'fields' that are replaced at runtime with the relevant values, as well as any text required to complete the name.

Example Name: will display a preview that will show you what the final composed name might look like when the current pattern is applied. This will help you to quickly see the results of manipulating the pattern by altering the text or adding or removing elements.

Click on any of the buttons in the **Insert Elements** area to add the relevant item to the pattern at the current location of the text cursor.

When you click any function button that is not a text character, you will see the relevant tag added to the pattern. The 'character' buttons are there for convenience to simply save you typing the character yourself. They do not form part of the element but are used for filename formatting.

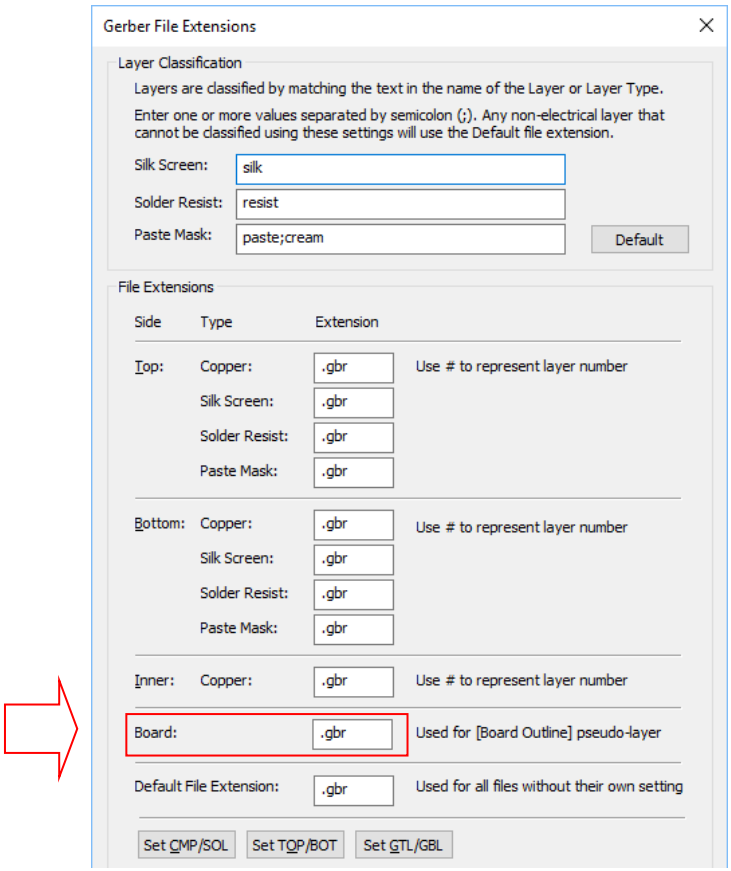
As well as design elements, you can also choose to add a **Value** to the pattern. The drop down list will provide you with all the value names defined in your design. Not only does this give you system values but also component values should you so wish to use them.

To remove elements from the pattern, click within the tag on the pattern box, then click the **Select Element** button. The text for that tag will become selected, and you can then either click one of the **Insert** buttons to replace that tag with the one you really want, or click **Delete Selected** to remove it from the pattern.

You can also edit the pattern by hand, to insert, alter or remove plain text items.

Gerber Output File Extensions for Board Outline

When setting up custom file extensions for Gerber files (through Plotting & Printing, Device Setup, Gerber Setup, Setup, Gerber File Extensions), an extra setting on the dialog allows you to specify the file extension for a plot containing the pseudo-layer [Board Outline].



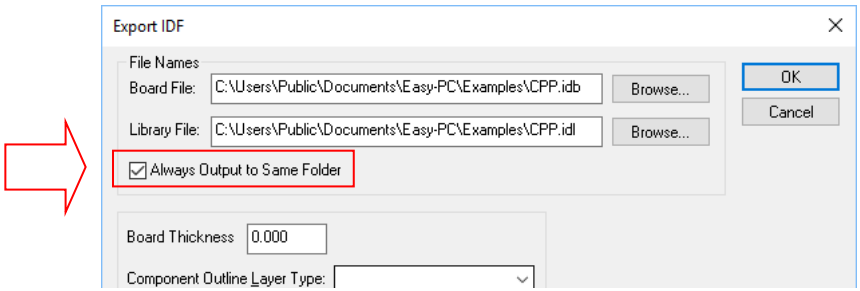
Plotting - Auto Mask

The ability to automatically create mask (solder resist/paste) for Copper Shapes attached and associated to pads in PCB Symbols has been extended to work on user-created mask layers in the PCB (layers which use a layer type which has a defined over or under-size). Any shape defined on a copper layer as part of a symbol will now create an appropriate under or oversized shape on an auto-mask layer. Previously, the Auto Mask facility was only available for pads-only plots generated at the Plotting stage.

IDF Export Dialog Changes

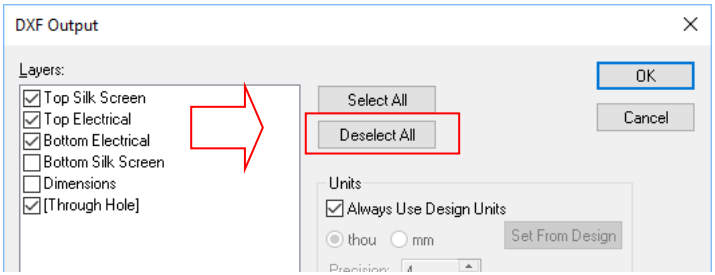
When using the **IDF Export** cost option, there is an extra check box to **Always output to same folder**. File names will default to the name of the design, but with this check box selected then once you have chosen a folder for your output files that folder will override the design folder next time you open the dialog.

This will be useful for users using SolidWorks which likes to import only from one specific folder.



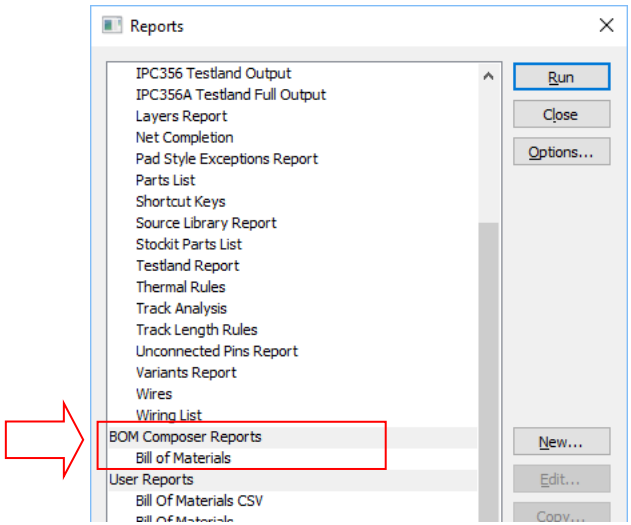
DXF Export Dialog Changes

There is an extra button to **Deselect** (uncheck) all the layers in the list. Initial default layer selection will check those layers that are currently used (have data present) in the design.



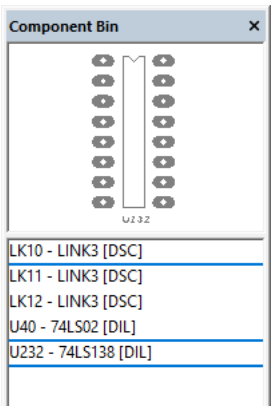
Create BOM Composer Reports from Reports Dialog

The **Report Manager** dialog when selected from Outputs menu now lists saved **BOM Composer Reports**. This means you can generate BOM Composer Reports from the same dialog as all the other reports.

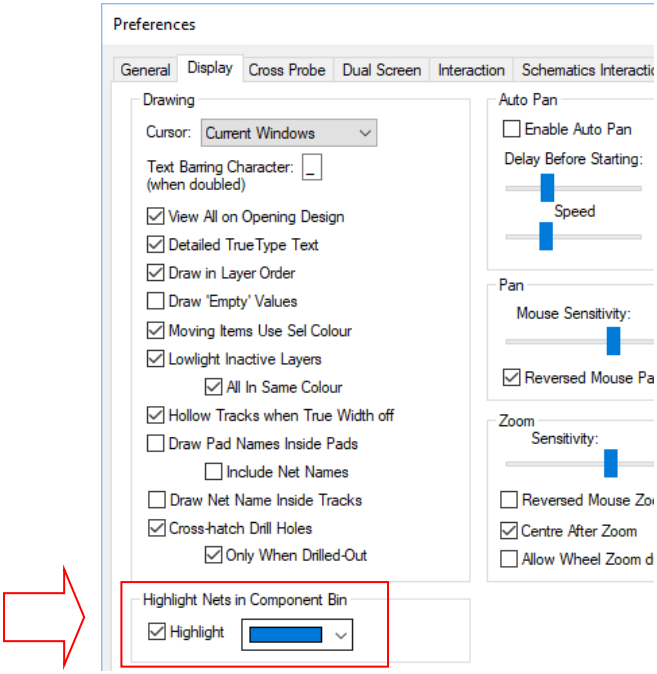


Component Bin – Highlight Nets

Within the **PCB Design** editor, Components which are attached to the currently highlighted net are highlighted in the **Component Bin** by underlining them with a coloured bar. If the net is large, such as for a GND signal, then it may be possible that multiple components are highlighted with the underline.

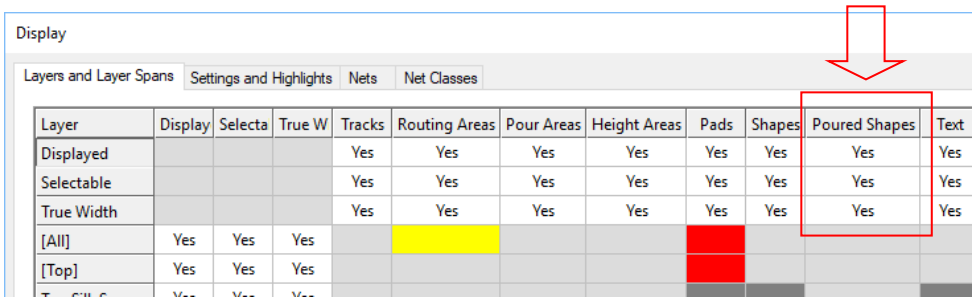


This option can be enabled in the **Preferences** dialog under **Display**. The underline colour for **Highlight Nets in Component Bin** can also be chosen here.



Colours (PCB) – Display of Poured Shapes

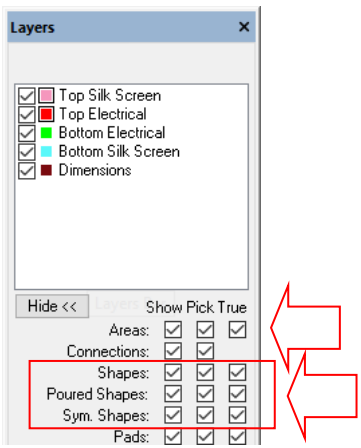
There is now a separate column in the **Display** (colours) dialog for **Poured Shapes** allowing them to have different colour and visibility settings from other shapes.



Layers Bar – Additional Check Boxes

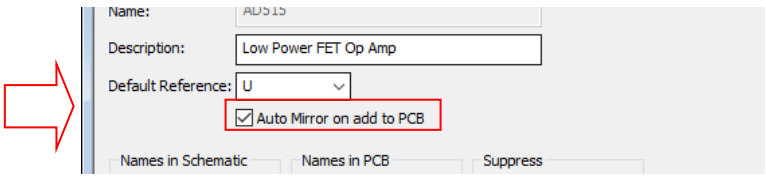
Additional check boxes to separate **Shapes**, **Poured Shapes** and **Symbol Shapes** have been added to the **Layers Bar**.

A new set of check boxes alongside the existing ones have been added to toggle the state of **True** width settings.



Auto Mirror Component Setting in Component Editor

From within the **Component Editor** dialog in the **Library Manager**, on the **Properties** tab, a new setting, **Auto Mirror on add to PCB** has been added. When checked, when this component is added to a PCB design (through the normal mechanisms - Add Component, Translate to PCB etc.), it will be flipped (mirrored) to the other side of the board automatically.



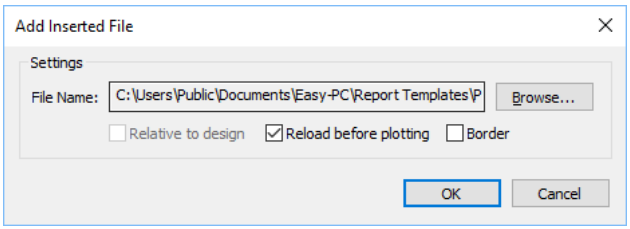
Inserted Files and Reports into a Design

Two new features, **Insert File** and **Insert Report**, are available on the **Add** menu for both Schematic and PCB designs.

Add – Insert File

From the **Add** menu, you can use **Insert File** to insert the contents of an existing text file into the design as a visible item. This is similar to what you can already do by pasting in the text of the file into a free text item, except that this new item retains the link to the underlying file and can reload it into the design.

Once the required file has been selected using the standard Open dialog, the **Add Inserted File** dialog is displayed:



This allows you add additional detail to the file, such as a **Border**, and to **Reload** it before plotting to ensure it is up to date.

Relative to design : check this box to specify that the application should look for the file relative to the location of the design file if the file is reloaded in the future. With this box unchecked, the application will always use the full absolute path of the file you selected.

Reload before plotting : check this box to tell the application that the file should be re-loaded and the inserted table re-generated whenever plots are run in the Plotting and Printing dialog.

Border : check this box to add a border (box) around the outside of the text.

Change settings if you wish, and press OK. The contents of the selected file will be read in, and a visible item built from free text with an enclosing border (if selected) will appear on the cursor for you to place. This will now act as one complete ‘unit’.

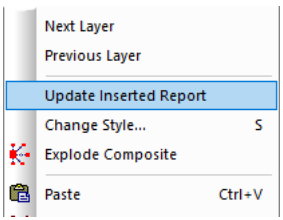
Add – Insert Report

A similar command is also available that allows you to **Insert Reports** as visible items in the design. You can insert most built-in reports and user-defined (both custom and BOM Composer) reports.

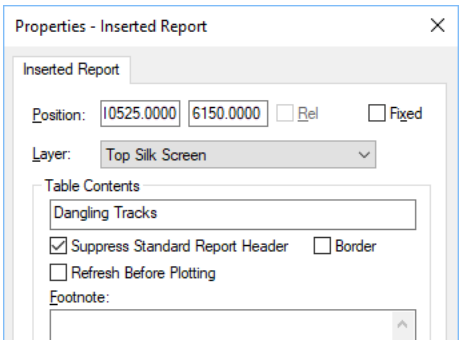
When this option is selected, the standard Reports dialog is displayed from which to make the report selection.

Properties of Inserted Files/Reports

Corresponding commands exist to update the inserted files using the context menu for a selected report:



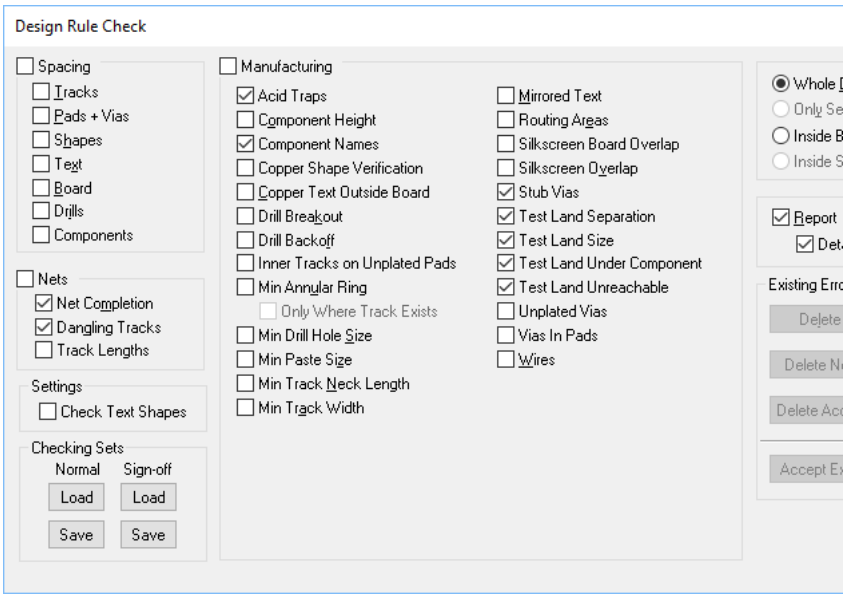
A **Properties** tabs for selected inserted reports provides access to the settings.



DRC Dialog Changes

Dialog Layout

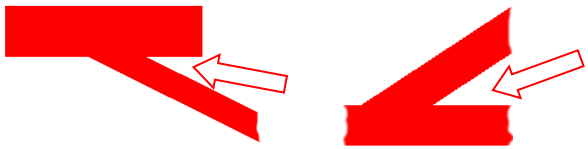
New checks have been added to the **Design Rules Checking** dialog and the Manufacturing section of the dialog has been split as well as sorting it alphabetically.



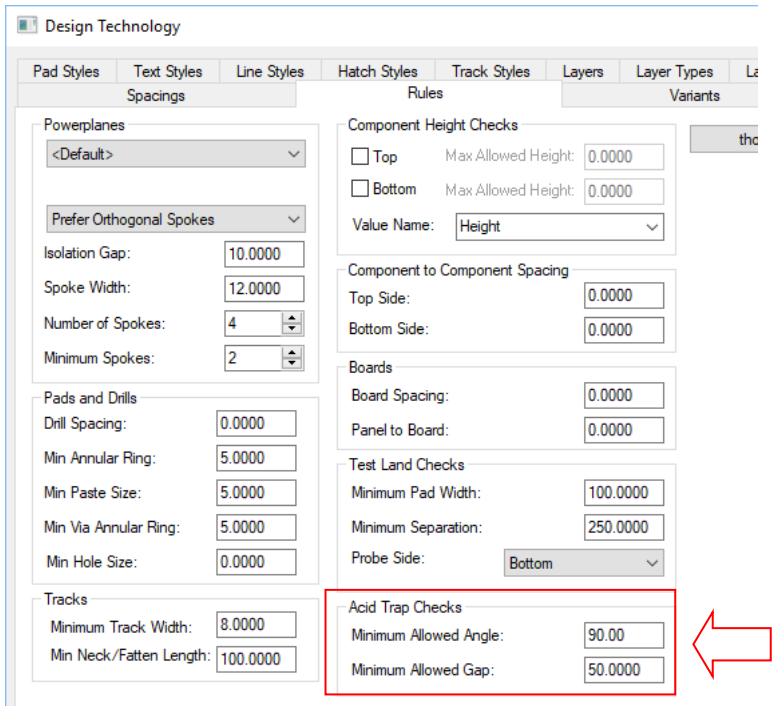
Acid Trap Check

Acid Traps: This checks for acid traps between Track segments and between Tracks and Pads.

An acid trap is considered to be an acute angle created between design items, usually tracks and pads (as shown below).

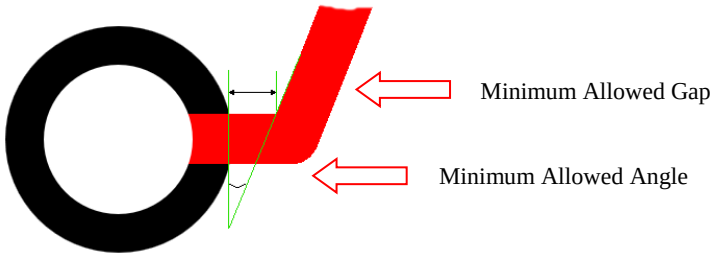


There are two new settings on **Rules** tab of the **Design Technology** page to set up the angle and gap.



Minimum Allowed Angle is defined between track segments or between the edge of a track segment and the pad shape it exits from. An angle of 0 disables this check. An angle greater than 90 degrees will only apply to track segments and not the exit angle from a pad.

As well as the detection of acute angles (less than the **Minimum Allowed Angle** defined), the angle also represents calculated angles based on tracks closer than the gap between the pad and the offending segment. This is defined as the **Minimum Allowed Gap**.



Minimum Allowed Gap is the gap between a pad and a track segment after the previous segment has exited the pad. If this second segment had turned to below the Maximum Allowed Angle relative to the edge of the pad, then an acid trap has been formed. Setting a gap of 0 disables this check.

Component Names Check

Component Names: checks for component names that are closer to a component other than the one to which they really belong. This is to avoid misunderstanding when the silk screen looks like the name belongs to the wrong component.

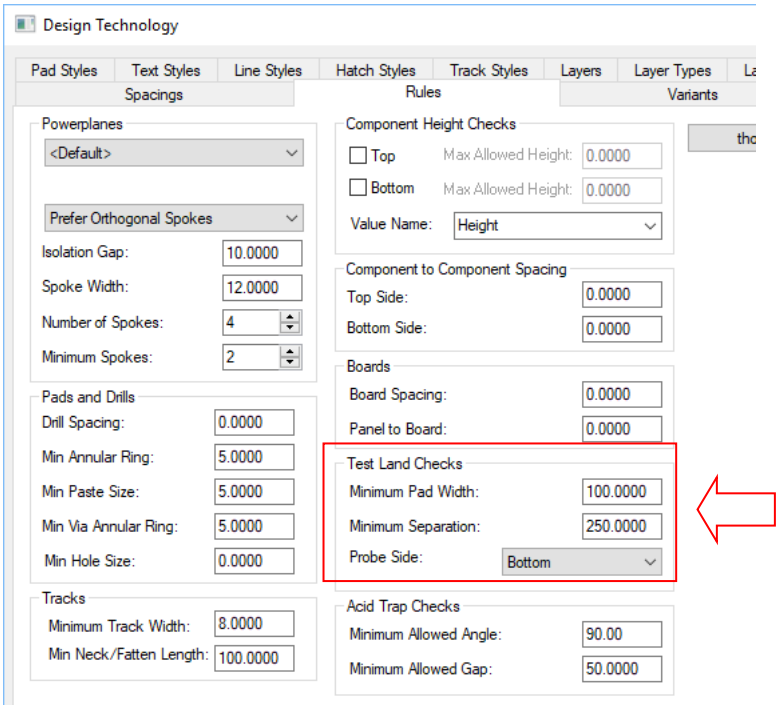
Stub Vias Check

Stub Vias: Checks for track segments which end on a via that does not appear to be connected to a Pour Area or plane. This aids the verification of the overall signal integrity to ensure no ‘gaps’ appear in the electrical path.

Test Land Check

Test Land: DRC checks are available for minimum separation, minimum land size, under components and unreachable side.

There are new settings on the **Rules** tab of the **Design Technology** page where you can set up the Minimum Size and Minimum Separation, and the Probe Side (top, bottom, either or both).



Test Land Separation - Checks to see if you have any items (pads or vias) marked as Test Lands that are closer together than the distance specified (**Minimum Separation**) in the **Rules** tab of the **Design Technology** page.

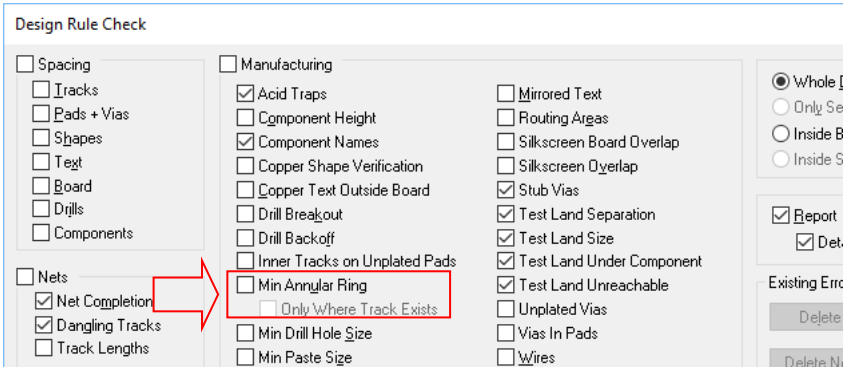
Test Land Width - Looks for items marked as Test Lands that are smaller than the defined **Minimum** (Test Land) **Pad Width** in the **Rules** tab of the **Design Technology** page.

Test Land Under Component - Looks for Test Lands that could be obscured by the body of a component and could thus be inaccessible to test probes.

Test Land Unreachable - Looks for Test Lands which don't exist on a layer selected as a valid **Probe Side** in the **Rules** tab of the **Design Technology** page.

Min Annular Ring Check

There is a new subsidiary check box for **Min Annular Ring check** within the **DRC** dialog to check **Only Where Track Exists**. This will flag annular ring errors only on pads/vias where there is a track connected on a layer where the annular ring is too small. In other words, it will not report errors for undersized rings when there is no track trying to connect to the ring.

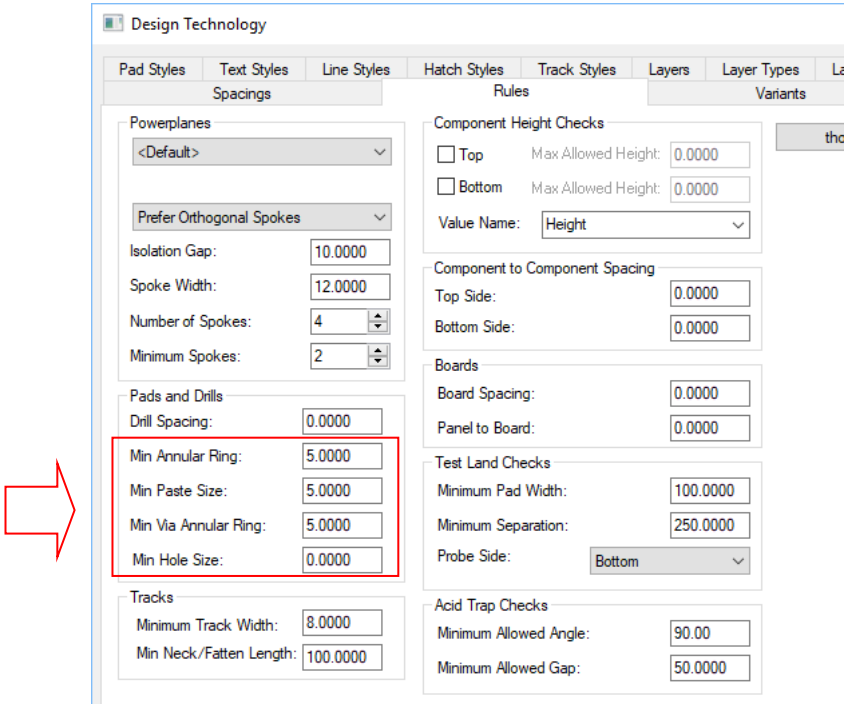


Min Hole Size and Min Via Annular Ring Checks

A new check for **Min Hole Size** has been added to the **Design Rules Check** dialog.

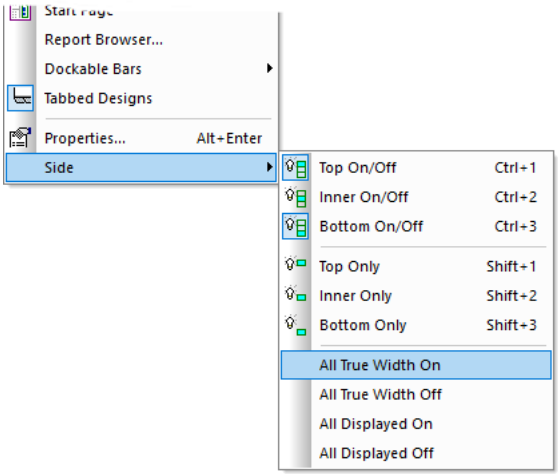
The existing check for **Min Annular Ring** has been expanded with new rules available under this check for checking the minimum ring size of **Vias.**

All the above new checks have a rule entry on the **Rules** page of the **Design Technology**.



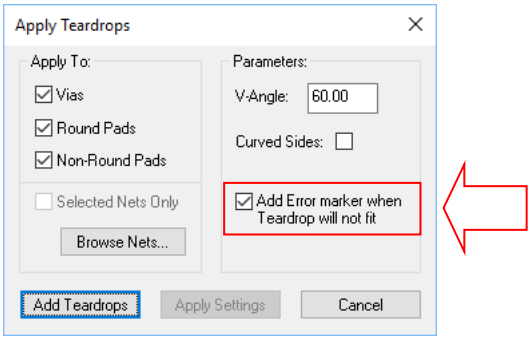
View Menu - True Width and Visibility On/Off

There are four new commands available from the **Side** submenu of the **View** menu to toggle all **True Width** and **Displayed** (Visibility) **On** or **Off**.



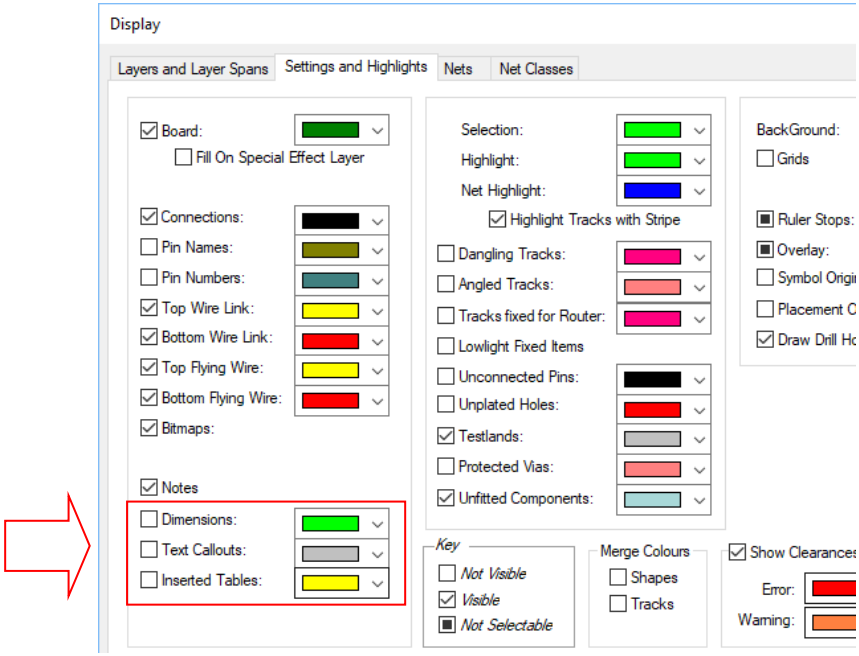
Add Teardrops – Add DRC Error Markers When Cannot Be Added

There is a new check box on **Apply Teardrops** dialog to **Add DRC markers when Teardrop will not fit**. This flags up places where the application cannot insert the required teardrop because it is not legal (due to shortness of track segment, pad too small, too close to other items, etc.).



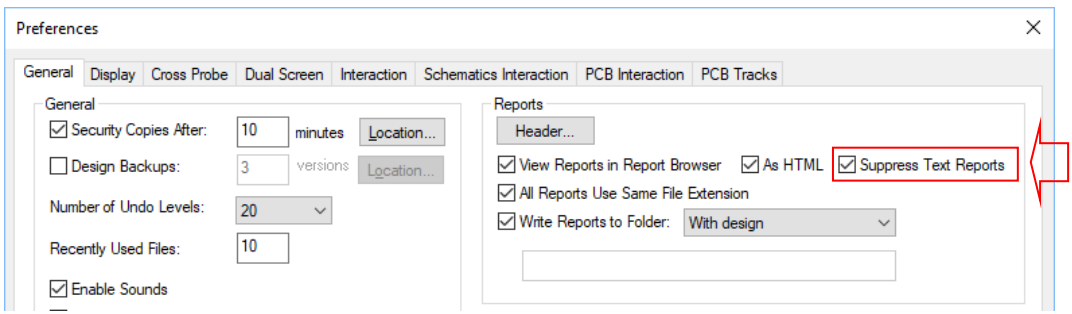
Display (Colours) for 'Composite' Items

Within the Display (colours) dialog under Settings and Highlights, the ability has been added to actually show or hide 'composite' items, such as Dimensions, Text Callouts, and Inserted Tables (Drill table, Layer Stacks, etc.), rather than simply altering their colour by highlighting them.



HTML Report Setting in Preferences

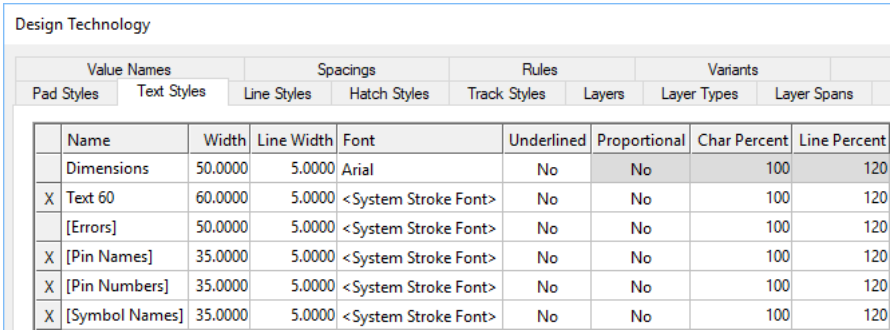
An extra check box has been added to the **Preferences** dialog and **General** page, allowing you to suppress the text version of HTML reports – suppress Text Reports. Previously, if you enabled HTML reports, you would also get the same report in TXT format alongside every HTM file.



Proportional Text Styles for System Font

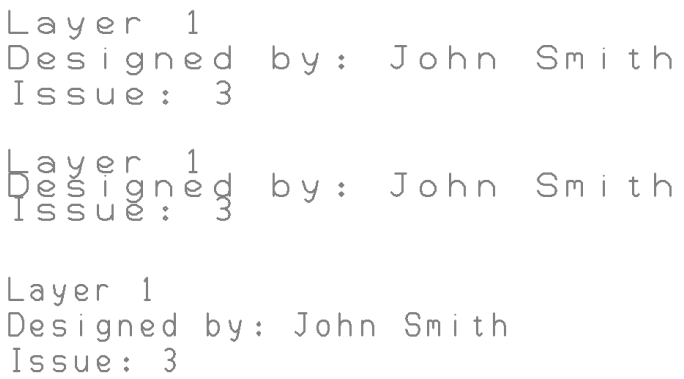
From within the **Design Technology** dialog and **Text Styles** page, text styles that use the <**System Stroke Font**> can now be set as **Proportional**. In addition, you can modify the **percentage character** width (to create 'compressed' text) and the **line spacing** (to squeeze up or spread out multi-line text).

As well as the **Text Styles** page of the **Design Technology** dialog, these changes also affect **Change Text Style** and the **Text Properties** dialog.



	Name	Width	Line Width	Font	Underlined	Proportional	Char Percent	Line Percent
	Dimensions	50.0000	5.0000	Arial	No	No	100	120
X	Text 60	60.0000	5.0000	<System Stroke Font>	No	No	100	120
	[Errors]	50.0000	5.0000	<System Stroke Font>	No	No	100	120
X	[Pin Names]	35.0000	5.0000	<System Stroke Font>	No	No	100	120
X	[Pin Numbers]	35.0000	5.0000	<System Stroke Font>	No	No	100	120
X	[Symbol Names]	35.0000	5.0000	<System Stroke Font>	No	No	100	120

The three examples below show how the new settings affect the character spacing and line spacing. All three examples have the same Width and Line Width:



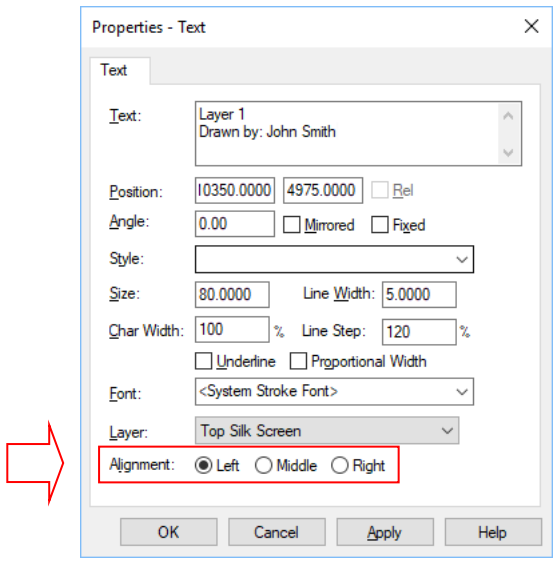
Normal:
Char % 100
Line % 120

Char % 100
Line % 80

Char % 75
Line % 120

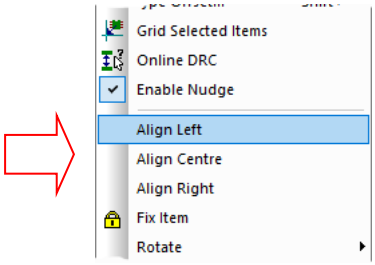
Properties - Text Alignment

Text Properties alignment is now 3 radio buttons in place of the drop-down list for quicker access to the alignment setting.



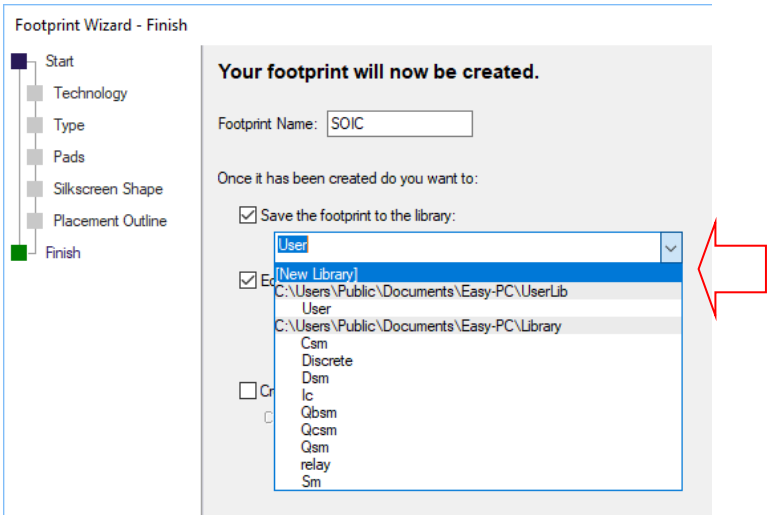
Text Alignment Changes

Three new commands have been added to the context menu for selected text. This enables you to set all text items of current selection as either **Left**, **Centre** or **Right** justified.



New Library option on Library Wizards

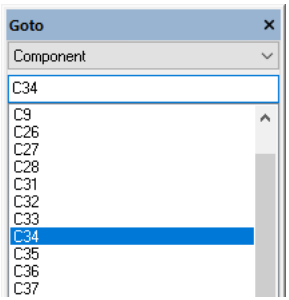
On the **Finish** page of the three library item Wizards (Schematic Symbol Wizard, PCB Symbol Wizard and Component Wizard), you can now select the **[New Library]** option from the drop down list as a choice of libraries to specify a new library file in which your new item is to be saved. When run, this will prompt you for the name and location of the new library.



Goto Bar Changes

Typed Names in Edit Box

The **Goto Bar** now has edit box where you can type in the name of the item you are looking for, rather than being purely a list box of items of the chosen type. The speeds up your search process in long lists where you know all or part of the name of the item you need to find and where it is at the bottom of the list, X1 or U22 for example!



Copper Pour Areas Details

When listing **Copper Pour** areas in the **Goto bar**, they are now listed with details of their **Net Name**, **Layer** N**ame** and **Pour Order** (previously they were all listed/grouped only by Net). If you have the Goto Bar displayed (locked out) and you make changes to the **Copper Pour** areas items in the design, you must use the **Refresh Find List** from the context menu (right clicking in the Goto Bar) to update it.