

Easy-PC

V18.0 Supplement

Copyright © 1998-2014 WestDev Ltd. All rights Reserved. E & O E

Number One Systems

20 Miller Court
Severn Drive
Tewkesbury Business Park
Tewkesbury
Glos GL20 8DN
United Kingdom

Phone: 01684 296 501
Fax: 01684 296 515
Email: info@numberone.com

Technical: 01480 382 538
Email: support@numberone.com

Web site: www.numberone.com

The product is licensed by WestDev Ltd to the original purchaser of the product for use only on the terms set forth in the WestDev Licence agreement. Copying, duplicating, selling or using the product contrary to these terms is strictly forbidden.

This Manual Copyright to WestDev Ltd. All Rights reserved. No part of it may be copied, photocopied, reproduced, translated, or reduced to any electronic medium or machine readable form without WestDev Ltd. prior written permission.

Information in this manual is subject to change without notice and does not represent a commitment on the part of the vendor.

Number One Systems, Easy-PC, Easy-Spice, Easy-Router, ProRouter, Pulsar, Layan, Analyser, MultiRouter, Filtech, Z-Match, StockIt and all variants thereof are Trademarks of WestDev Ltd.

All other trademarks acknowledged to their rightful owners.

Number One Systems, a trading division of WestDev Ltd.

While every care has been taken in the preparation of this manual WestDev Ltd cannot be held responsible for any errors or omissions within it. If informed we will add comments and features which you may like to see written which may help others using this manual. Please send your comments through the technical support desk.

Manual date: 21-July-2014 Issue 1

Warranty and Software Licence Agreement

WestDev Ltd and Number One Systems

This document contains the WESTDEV LTD SOFTWARE LICENSE AGREEMENT which will govern the use of the Westdev and/or Number One Systems' products supplied with it.

You agree to the terms of this agreement by the act of opening the sealed package which contains the storage media on which the software is recorded. Do not open the sealed package without first reading, understanding and agreeing the terms and conditions of this agreement. If you do not agree to the license conditions you may return the SOFTWARE for a full refund before opening the sealed package.

1. LICENSE AGREEMENT

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

2. OWNERSHIP OF SOFTWARE

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

3. COPY RESTRICTIONS

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

4. TRANSFER OF LICENSED SOFTWARE

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

5. MODIFICATION

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

6. TERMINATION

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

7. WARRANTIES

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

8. OTHER SOFTWARE

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

9. GOVERNING LAW

The Governing Law of this Agreement shall be that of England.

Westdev Ltd will reward anyone giving information leading to successful prosecution regarding breach of copyright.

Contents

CONTENTS	4
CHAPTER 1. GETTING STARTED	5
Installation	5
Running Easy-PC 18.0.....	6
CHAPTER 2. NEW FEATURES IN EASY-PC V18	7
Introduction.....	7
BOM (Bill Of Materials) Composer (Both).....	7
Library Reports (Both).....	9
Report Header settings (Both)	9
Settings – Folders (Both).....	11
Styles – Merge (Both).....	12
Select From List (Both).....	13
Temporary Free Segment Mode (Both).....	13
Colour by Net Class (Both).....	13
Fetch Component (Both).....	14
Automatic Fill Symbol Shapes in Schematics (SCM).....	14
Radial and Angular Dimensions (PCB).....	15
Dimension Properties	17
Dimensioning of Drill Holes (PCB).....	18
Snapping Ruler Stops to Drill Holes (PCB).....	19
Area Colours (PCB).....	20
Thermal Rules (PCB).....	20
Pour Copper Changes (PCB).....	21
Changes to Area Properties (PCB).....	21
Additional Pad Style Shapes (PCB).....	22
Change Layer in Edit Track (PCB).....	23
Component Properties – Suppress from DRC (PCB).....	23
Snapping Improvements (PCB).....	24
Double Click To End Track (PCB).....	25
Display Names in Pads and Tracks (PCB)	25
Reverse Mirrored Component Coordinates (PCB).....	26
Rotation Correction Value in Report output (PCB).....	26
Board size in Design Status Report (PCB)	27
Use Board Centreline for Spacing Rules (PCB).....	28
Testland Output (PCB).....	28
Full Extended IPC-D-356A Output.....	28
New Testland Settings.....	29
Shift Select for Cutouts (PCB).....	29
CHAPTER 3. UPDATED LIBRARIES	30
Pro Library 7 (Updated Cost Option).....	30
Micro Library 7 (Updated Cost Option).....	30
Find Library Parts using the Library Searcher	30

Chapter 1. Getting Started

Installation

Backing up your files

If you already have Easy-PC installed, please remember to back up all your libraries, technology files and any other data files before proceeding with the installation of the new version. The installer should not overwrite any of your own named files, but it can re-install new copies of our standard data files so if you have changed any of those files it is important to back them up first. If you are uncertain, check the time/date stamp on the file.

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

Installation

Installation is via the *autorun* setup. Insert the CD-ROM into your CD-ROM drive and wait a short time. The CD-ROM will run up to speed and an Easy-PC Welcome screen will appear. If *autorun* has been disabled on your computer you must execute the 'setup.exe' program using the **Start** menu and **Run** command from the Windows task bar.

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

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

With the installer running, once the **Welcome** screen is displayed, double-click on the **Install Easy-PC - Version 18.0** 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 and later, “Documents and Settings\All Users\Shared Documents” on Windows XP (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 18.0

As with all **Easy-PC** programs, an icon will appear in the **Number One Systems** folder, or the Start pane if using Windows 8, you may also wish to create an **Easy-PC** Shortcut icon that sits on your desktop.

To start the program, double-click on the **Easy-PC** icon from the **Number One Systems** folder or the Start pane if using Windows 8.

Chapter 2. New Features in Easy-PC V18

Introduction

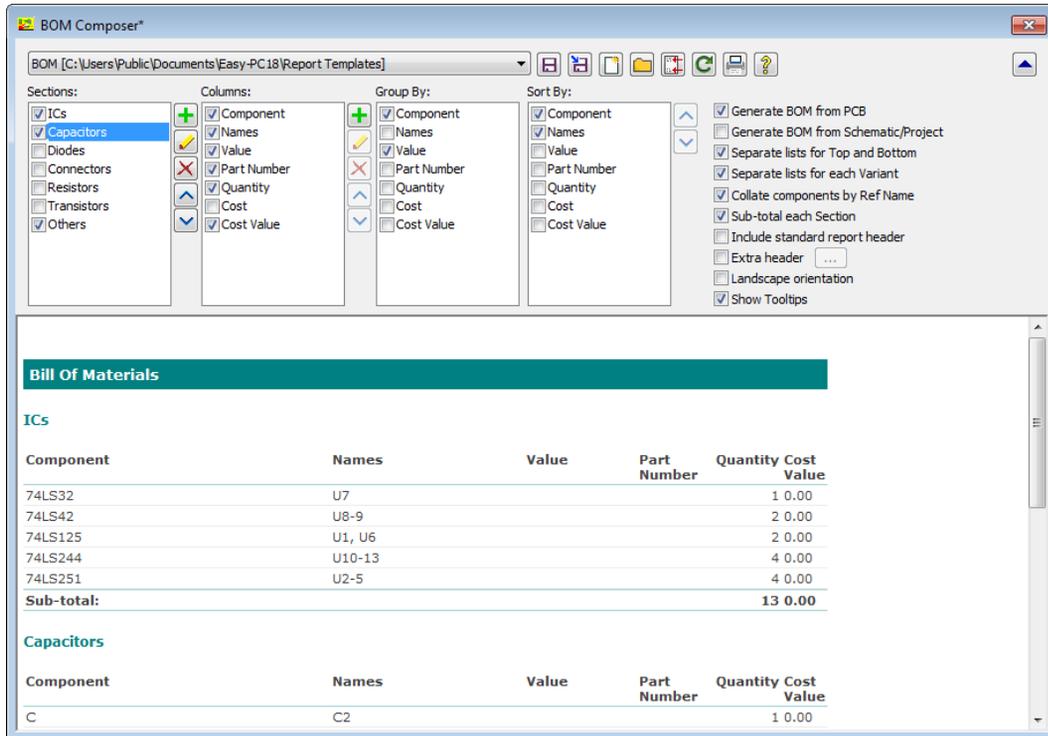
All features are categorised as being SCM specific, PCB specific or Both (relevant to both SCM and PCB design applications).

BOM (Bill Of Materials) Composer (Both)

The new **BOM Composer** allows you to create BOM reports using a simple point-and-click WYSIWYG (what you see is what you get) dialog.

Available on the **Output** menu the **BOM Composer** option opens up a new set of dialogs to control the BOM (Bill Of Materials) creation with various levels of user control over formatting.

Using example templates, a built-in default setup, or building your own template from scratch, the whole process is quick and easy. What's more, you can immediately see the effect of each change you make to the template, right in front of you on the screen.



When the BOM Composer is first run, the dialog will look something like the picture shown above.

Along the top are some controls that are used to load, save and generate BOM templates and reports. Below this are some lists and buttons which allow you to control what appears in the BOM and how it is built and formatted.

Finally, the main body of the dialog shows you the BOM as it is generated based on the current settings. At the top right is a button that lets you 'fold up' the dialog to hide the lists and other editing buttons. Once you have got your BOM formatted how you wish, you can hide away the formatting controls to give you more space to view the actual BOM.

Templates

BOM Composer uses 'templates' to control how the BOM should look. The templates are simple files (.BCT) that are stored in the same folder as your other Report Format templates (.RFT), the ones that are used by the formatted report manager. You don't need to edit the template files yourself, they will be loaded and saved using the buttons and controls on this dialog. You can set up as many templates as you wish, and choose which one to use from the drop-down list at the top left of the dialog. You may for example have one template to produce a detailed list for pre-production inspection, one list including prices and part numbers for your purchasing department, and so on. The report file will take the name of the template file as its heading, so you can distinguish between the different types of reports.

Main controls

Template

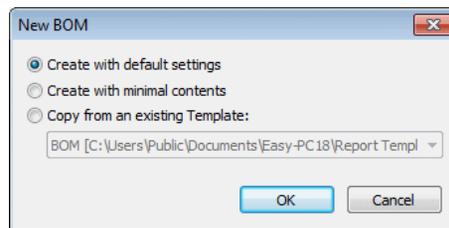
The drop-down list at the top left of the dialog is where you select which template you wish to use. This list is populated with all the currently enabled template (.BCT) files that the application can find in the currently enabled Report Files folders.

Save, Save As and New

Three buttons are provided to allow you to save the current template to the current name or to a new file, and to create a new template.

If you have modified the template you will be prompted to save it as you close the dialog anyway, but the Save button is there to use if you want to save your changes more often.

When creating a new template, you will be asked if you want to start with a default setup or with a 'blank' template, or copy from an existing template.



Controlling the BOM contents

Sections

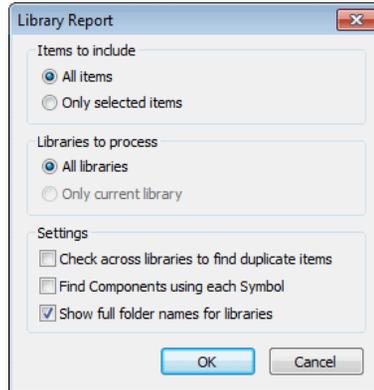
Each BOM is divided into sections, using the component reference name stem to distinguish between different 'types' of components. You can create as many or as few sections as you wish, specify one or more name stems to match for each section, and the order in which the sections appear. To assist in making quick - and reversible - changes to the report, you can enable/disable sections using the checkbox rather than adding and deleting sections.

The final section in the list is a 'catch all' section which will always appear last (you can't use the button to move it up the list), and has a pre-defined name stem of '*' to match any

component name that has not been matched by any of the preceding sections. This ensures that the BOM will always be complete and no components are missed out. Note that as well as being 'locked' at the foot of the list, you cannot uncheck the catch all section by unchecking its check box either. Again that is to make sure every component will find a matching section somewhere in the report.

Library Reports (Both)

From within the **Library Manager**, reports generated for **Symbol** and **Component libraries** now have user-selectable options to control what goes into the report.



On selection of the **Report** button on the **Library Manager** dialog, a dialog is displayed allowing you to choose whether to process **All** (library) **Items**, or only the ones currently **selected** in the contents list and **All libraries** or only the **current library**.

There are also additional **Settings** selections to check across libraries for **duplicate items across libraries**. The **Duplicates** report is the same one as it was previously. It always works across all libraries.

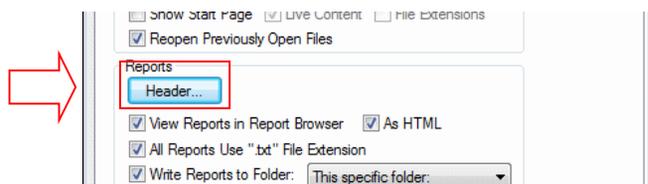
The new **'where used'** feature for symbols allows you to find all the components that use each symbol. This is also useful for identifying where libraries might be affected after symbol edits etc.

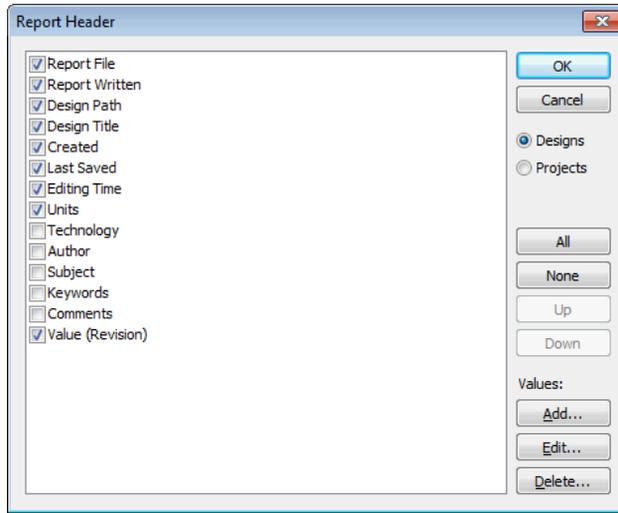
The **Show full folder names for libraries** will report the full library path when selected.

Report Header settings (Both)

The **Report Header** dialog is used to manage the items you wish to include in the 'standard report header' section that appears at the top of most reports. It also allows you to include Values (design attributes) of your choosing in the header.

There is a new **Header** button on the **General** page of the **Preferences** dialog





The list box shows all the possible items that can appear on report headers. The list includes standard 'inbuilt' ones such as the report file path, design or project path, and last saved time.

It also includes any **Values** you have added to the list of report header items (see below).

The check box on each entry can be used to enable/disable the report header item. There is a separate list for design reports and project reports.

Using Values

As well as the inbuilt items, you can add your own Values to the report header, by adding the required Value Name to the list.

Click **Add** to choose from the list of Value Names collected from all currently open design files, or to type in a Value of your own choosing.

The **Edit** and **Delete** buttons can be used to edit and delete Value items in the list.

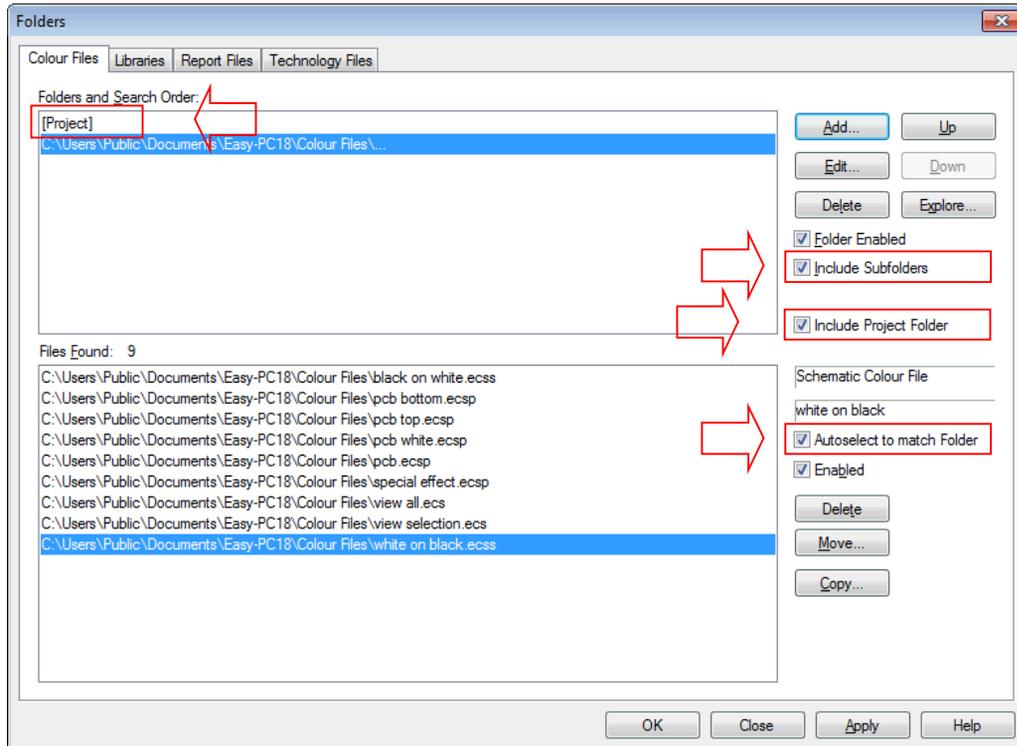
Using a Value with **no Value Name** will insert Value () in the list but will add a blank line into the header.

Saving the settings

Click **OK** to close the dialog and save the settings. The settings are also saved if you use the radio button to change from **Designs** to **Projects**.

Settings – Folders (Both)

There is a new check box on **Folder** pages (Colour Files, Libraries, Report Files and Technology Files) to **Include Subfolders**. Checking this box for a folder name in the **Folders and Search Order** list will append a suffix “\...” to the end of the folder name, to indicate that it will include matching files from all sub-folders of the stated folder path, as well as files in that specific folder.

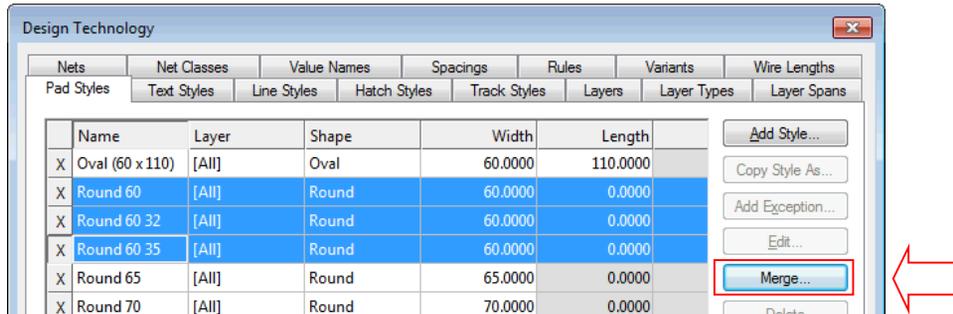


There is also a new check box on **Folder** pages to **Include Project Folder**. This will add/remove a special value “[Project]” in the **Folders and Search Order** that will automatically track the currently active project or design file, and thus automatically add the folder where that file lives into the search path. So, if you have project-specific libraries, you can enable the [Project] folder, optionally edit that entry to name the sub-folder where the libraries live if they are below the design files, and the application will ‘see’ the relevant library files as you open/close project or design files without having to change folder settings.

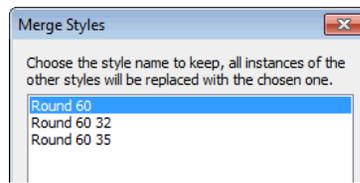
There is another new check box on this dialog, **Autoselect to match folder**, which will select all the entries in the **Files Found** list that belong to the folder selected in the **Folders and Search Order**. This will make it easier to find the relevant libraries in a long list of files when you have them scattered across several folders.

Styles – Merge (Both)

There is a new **Merge** button on the five **Styles** pages of the **Design Technology** dialog.



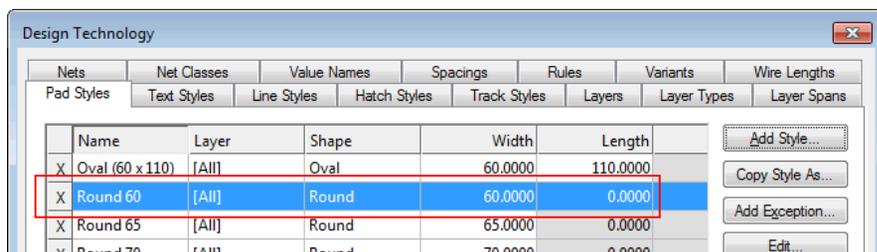
This allows you to select two or more rows in the table and merge them together. This will update the design to replace all instances of the ‘obsolete’ styles with the one style you have opted to keep.



From the dialog, choose the style that you wish to keep and click OK. All the other styles will then be merged with the selected style, including updating all references to that style in the current design. Note that you cannot remove built-in styles (those with names surrounded by square brackets), so if you select any of those you may not be able to perform the Merge action.

You may find the Merge facility useful if you have combined sections of data (components, connections, etc.) together from several different sources, where the styles used in those sections may not be consistent. Note that merging styles will only affect the current design, it does not go back to the library and rationalise style settings there.

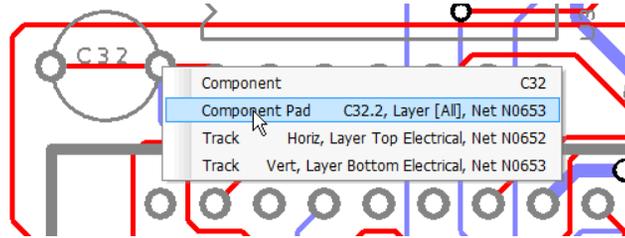
The resultant Pad styles dialog now looks like this after merging:



*Note: The ability to select multiple rows required for **Merge** means that you can now use **Delete** on several entries at once.*

Select From List (Both)

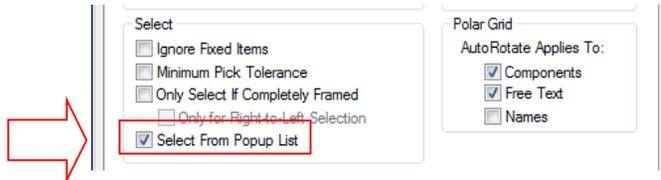
There is a new option on Select, to select from a popup list.



Where multiple items reside within the picking tolerance under the cursor, as an alternative to using select next <N>, you can now press the **Alt** key and select to display the selection menu. Once this is displayed, pick from the list using the item required.

This option is triggered if there is more than one item under the cursor. In this instance, you will get the popup menu of items to choose from.

By default this option is switched on. If you wish to disable it, this can be done from the **PCB Interaction** page of the **Preferences** dialog. At the bottom of the page, uncheck the **Select From Popup List** option under **Select**.



Temporary Free Segment Mode (Both)

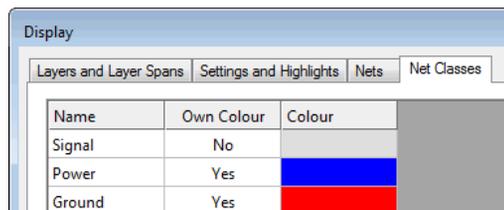
Holding the **Shift** key down whilst adding Shape, Bus, Schematic Connection or Track segments will temporarily switch to **free-segment mode**. On release of the mouse and a move, the option switches back to the previous segment mode.

Note: This does not affect Add Pull Tight Track option.

Colour by Net Class (Both)

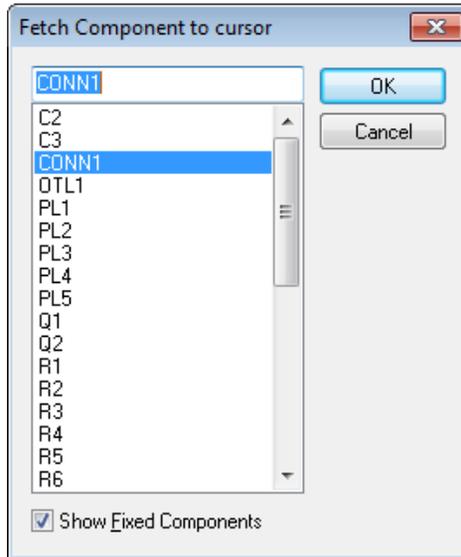
Net Class now has an **own colour** status, the same as a Net. This can be defined in the **Display** dialog under **Net Classes** or **Design Settings** dialog and **Net Classes** page.

Own colour on a **Net** takes precedence over own colour of a net class, so you could set net class Power as dark red to get all those nets dark red, then net 24V as bright red to make that net override the net class colour.



Fetch Component (Both)

There is a new command (**Fetch Component**) on the context menu in **Select mode** and when **picking in free space** with nothing else selected. When selected, it presents you with a dialog showing all the component names in the design. Type or select the desired name to bring that component to the cursor ready for interactive placement.

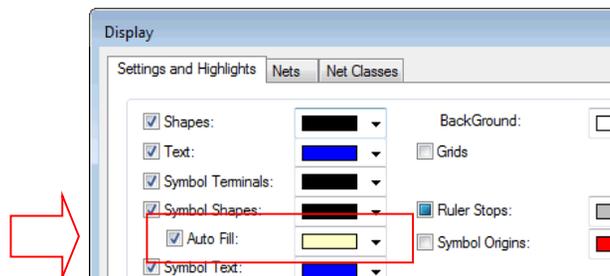


The **Fetch Component** command can be assigned to a shortcut key for convenient use. Using this method, you can simply press the shortcut key, type the name and press Enter. The component is then there on your cursor to be placed. This command is not assigned to a shortcut key as standard.

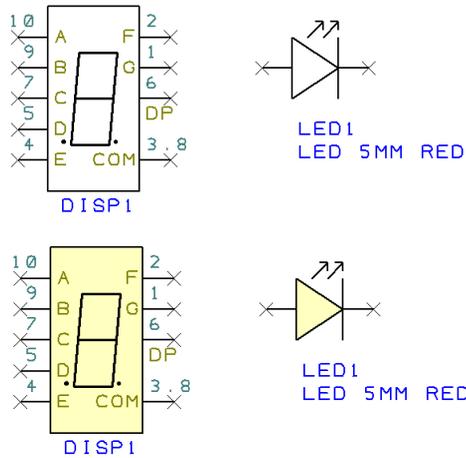
Automatic Fill Symbol Shapes in Schematics (SCM)

Closed unfilled symbol shapes in Schematic designs can now be auto-filled with a user-defined colour. This helps improve the 'look' of the overall Schematic drawing without having to modify all of your symbols.

There is a new check box for **Auto Fill** and the colour selection in the **Display** dialog on the **Settings and Highlights** page.



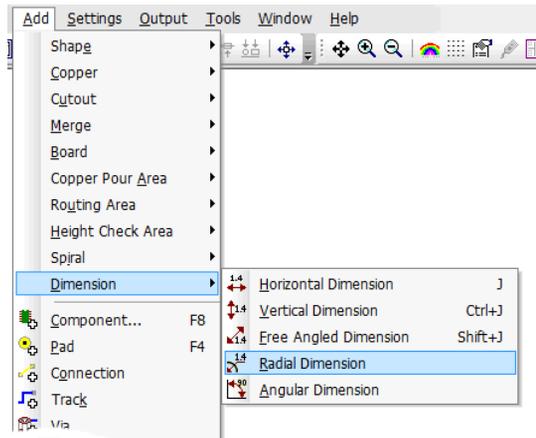
The pictures below illustrate the 'before' and 'after' effect of using the Auto Fill switch.



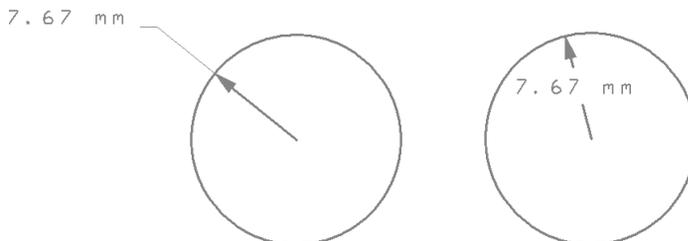
Note: Unclosed shapes will not be filled

Radial and Angular Dimensions (PCB)

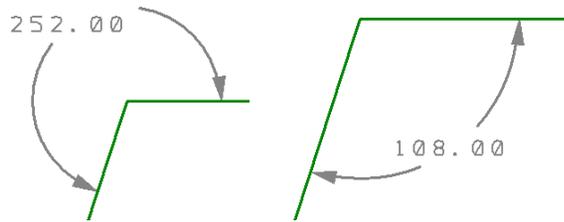
You can now add **Radial** and **Angular** dimensions to a PCB designs. There are new options on **Add** menu and on **PCB toolbar**.



Examples of Radial Dimensions (outside and inside shape):

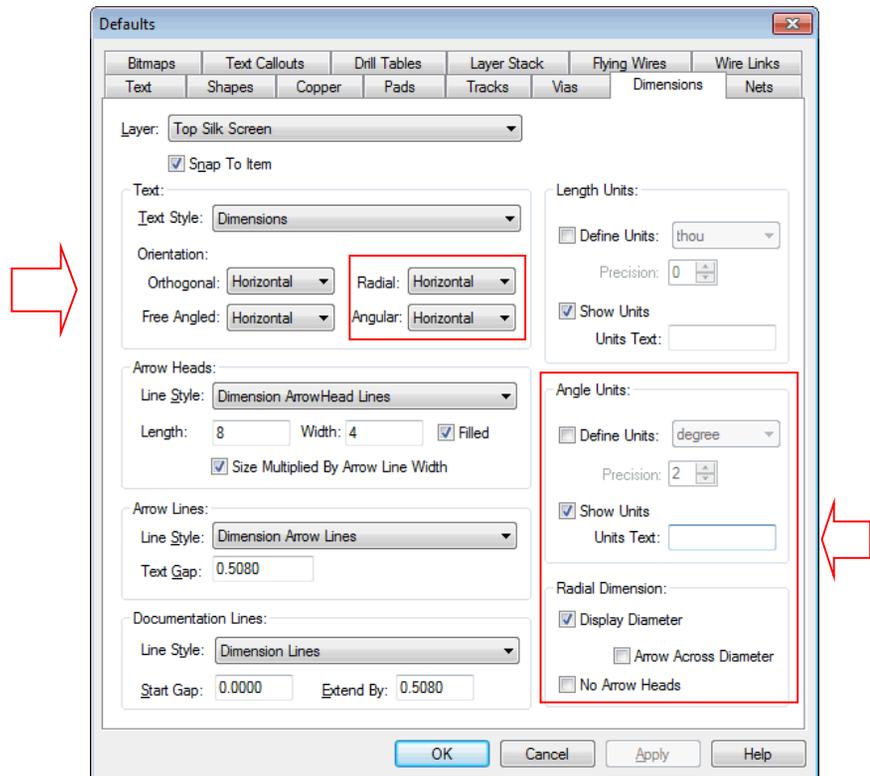


Examples of Angular Dimensions (outside and inside shape):



Dimension Defaults

There are changes to the **Defaults** dialog and **Dimensions** to cater for Radial and Angular dimensions.



Orientation

There are two new drop-down list boxes under **Orientation** for **Radial** and **Angular** Dimensions. Use these to set the direction of the text item for new dimensions. Choose between having the text **Horizontal**, **Vertical** or **In Line** with the dimension arrow lines.

Angle Units

Use this to define the **units** that new angular dimensions will use. Leave the box unchecked to allow these new dimensions to use the current design units at the time they are created.

The **Precision** specifies how many digits (if any) to display after the decimal point. Values that do not exactly "fit" using the defined precision are rounded to the nearest applicable value before they are displayed.

Show Units In Text - Use this to specify whether the angular dimension text includes the units name. If checked, you can specify your own **Units Text** characters, or leave it blank to use the default name for the type of units specified.

Radial Dimensions

These controls effect the appearance of radial dimensions. These dimensions display the radius or diameter of an arc or circle.

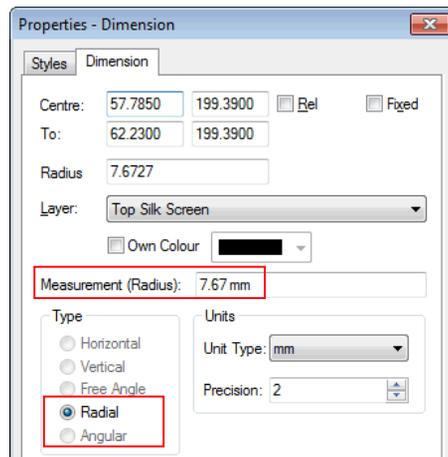
Display Diameter - Check this box if you want the dimension text to display the diameter of the circle or arc instead of the radius.

Arrow Across Diameter - If displaying diameter text, check this box if you want the arrow to cross the whole diameter of the circle or arc instead of just to the centre.

No Arrow Heads - Check this if you do not want arrow heads shown in your radial dimensions.

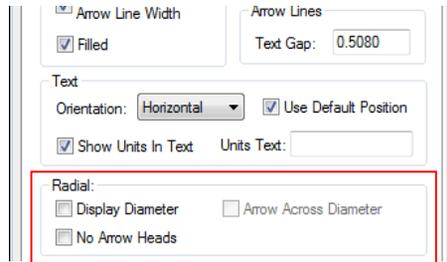
Dimension Properties

Properties for **Dimensions** has changed to cater for the new Radial and Angular dimensions.



Measurement - This shows the actual value the dimension represents. For linear and radial dimensions this is a length or radius using the defined length units. For an angular dimension this is an angle using the defined angle units.

Type - This shows the type of dimensions selected. If angular or radial dimensions are selected their type cannot be changed.



Radial - These controls effect the appearance of radial dimensions. These dimensions display the radius or diameter of an arc or circle.

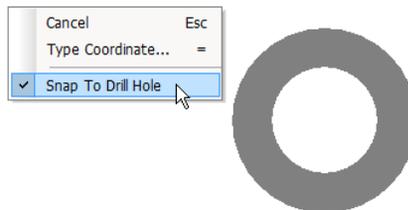
Display Diameter - Check this box if you want the dimension text to display the diameter of the circle or arc instead of the radius.

Arrow Across Diameter - If displaying diameter text, check this box if you want the arrow to cross the whole diameter of the circle or arc instead of just to the centre.

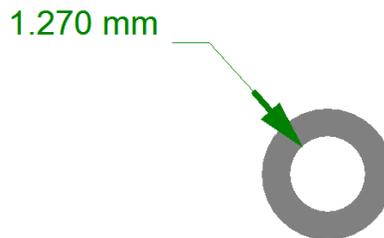
No Arrow Heads - Check this if you do not want arrow heads shown in your radial dimensions.

Dimensioning of Drill Holes (PCB)

When adding **Linear** or **Radial Dimensions** there is a new **Snap To Drill Hole** option on the context menu.

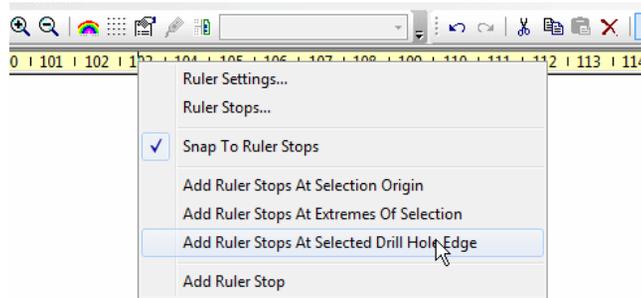


Enable this option to dimension the diameter or radius of the selected drill hole or slot.

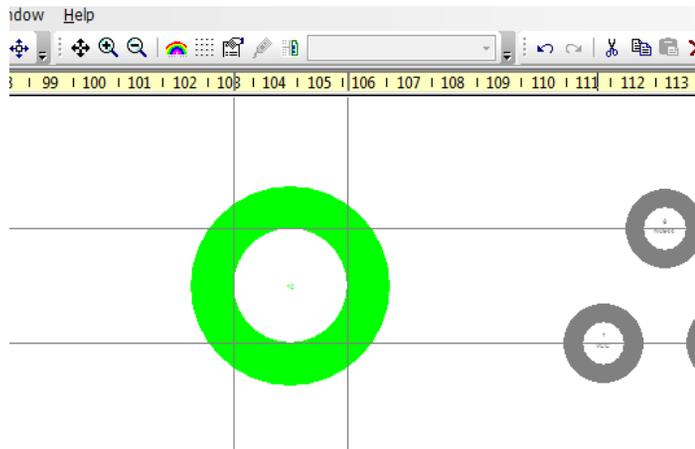


Snapping Ruler Stops to Drill Holes (PCB)

If you want to add a horizontal or vertical dimension from the edge of a drill hole to another item, this can now be done using a new function on the rulers bar. Select a pad with a drill hole and right click on the rulers bar and use the new **Add Ruler Stops At Selected Drill Hole Edge** option.



Ruler stop lines will appear on the edges of the drill hole. You can then add a dimension and snap to these ruler stop lines.



Area Colours (PCB)

There are now separate colours for the three different types of Area (Routing, Copper Pour and Height Check).

Available from the **Display** dialog and **Layers and Layer Spans** tab.

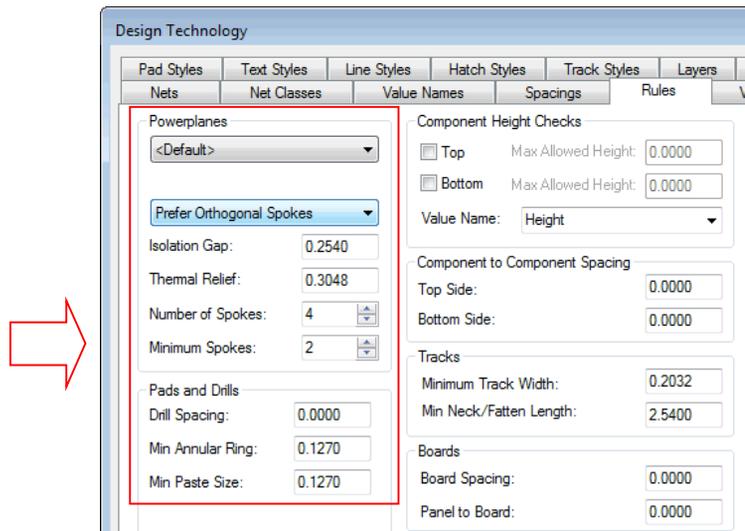
Layer	Display	Selecta	True Width	Tracks	Routing Areas	Pour Areas	Height Areas	Pads
Displayed				Yes	Yes	Yes	Yes	Yes
Selectable				Yes	Yes	Yes	Yes	Yes
True Width				Yes	Yes	Yes	Yes	Yes
[All]	Yes	Yes	Yes					
[Top]	Yes	Yes	Yes					
Top Silk Screen	No	Yes	Yes					
Top Electrical	Yes	Yes	Yes					

Thermal Rules (PCB)

In Easy-PC V18 there is an expanded definition of rules for thermal relief in Pour Copper and Powerplanes.

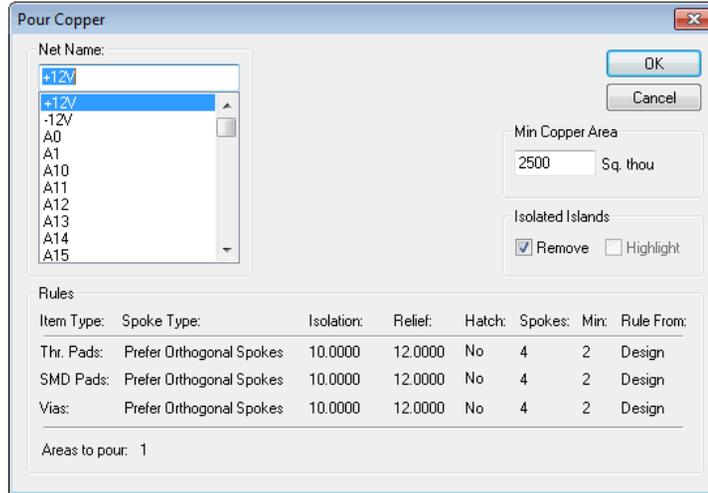
The extra settings at design level (defined in **Design Technology** and **Rules**) allow the number of spokes, minimum spokes, spoke direction, isolation gap and thermal relief width to be set for all items, and optionally over-ridden for plated through hole pads, SMD pads, and vias.

The same set of thermal rules is available per Area if required. These design-level rules can also be overridden in the Copper Pour Area Properties (see below).



Pour Copper Changes (PCB)

When using the **Pour Copper** dialog, the parameter selection dialog has been changed. It now shows you the rules that the pouring will use.



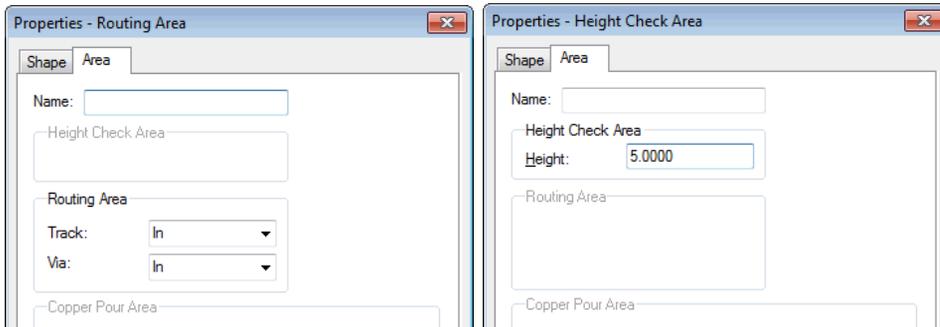
Properties for copper pouring will be taken from the **Design Technology** dialog and **Rules** page.

Changes to Area Properties (PCB)

Area Properties are now on a separate tab (they were previously on the **Shape** tab). This has been implemented to allow for the expanded controls for thermal rules per area.

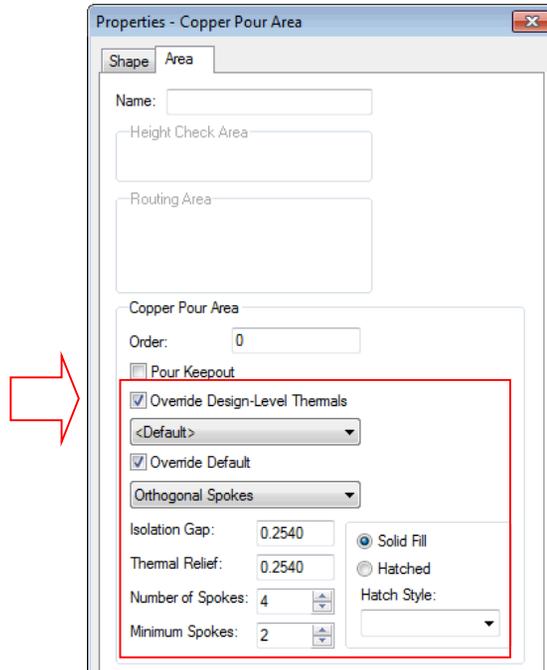
Routing and Height Check Areas

Below are shown **Properties** of **Routing Areas** and **Height Check Areas**:



Copper Pour Areas

Below shows the **Properties** dialog for **Copper Pour Areas**:

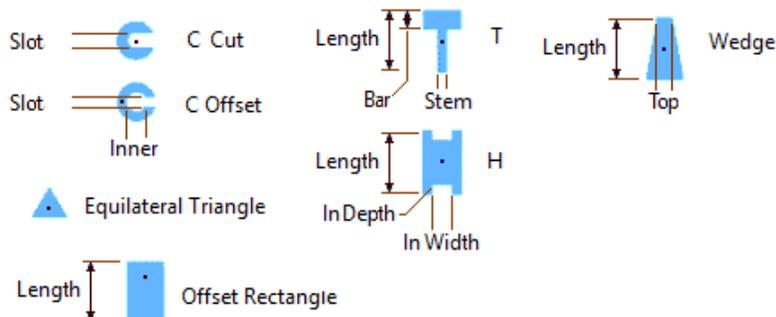


Checking the **Override Design-Level Thermals** will allow you to add specific thermal properties to individual areas. This overrides the settings defined in the **Design Technology** on the **Rules** page.

Additional Pad Style Shapes (PCB)

There are additional pad shapes available in V18:

C Cut, C Offset, H, T, Wedge, Offset Rectangle, and Equilateral Triangle



These are available in the **Design Technology** dialog under **Pad Styles**.

The **Equilateral Triangle** shape is also available as a new **Pin Style** in Schematics.

Change Layer in Edit Track (PCB)

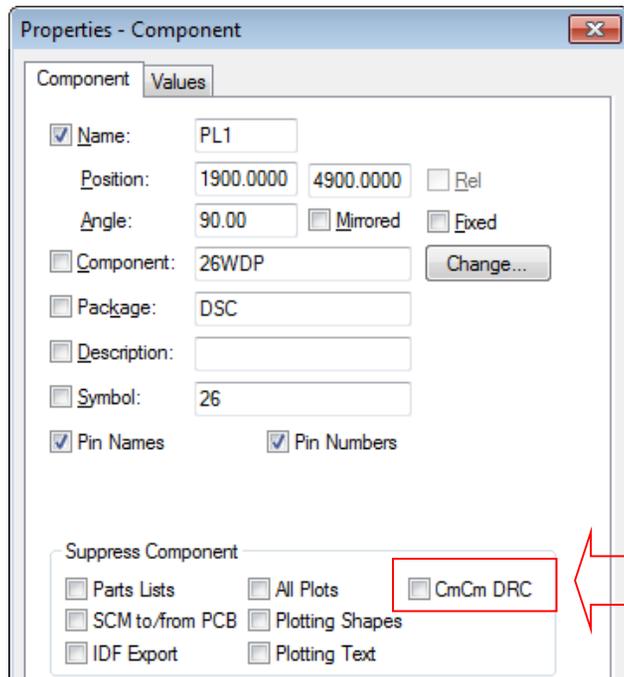
When adding or editing a track, holding the **Ctrl** key down while clicking with left mouse button will insert a corner as normal but it will now then change layer to the 'other' layer. This is a shortcut to the same action as you would get by choosing the command **Change Layer** and clicking **OK** but it avoids having to display the dialog. Top layer will toggle to bottom and vice versa, inner layer will toggle to top.

Component Properties – Suppress from DRC (PCB)

The ability to suppress components from a **Component to Component** check in the **Design Rules Checks (DRC)** has been added to the **Component Properties** dialog.

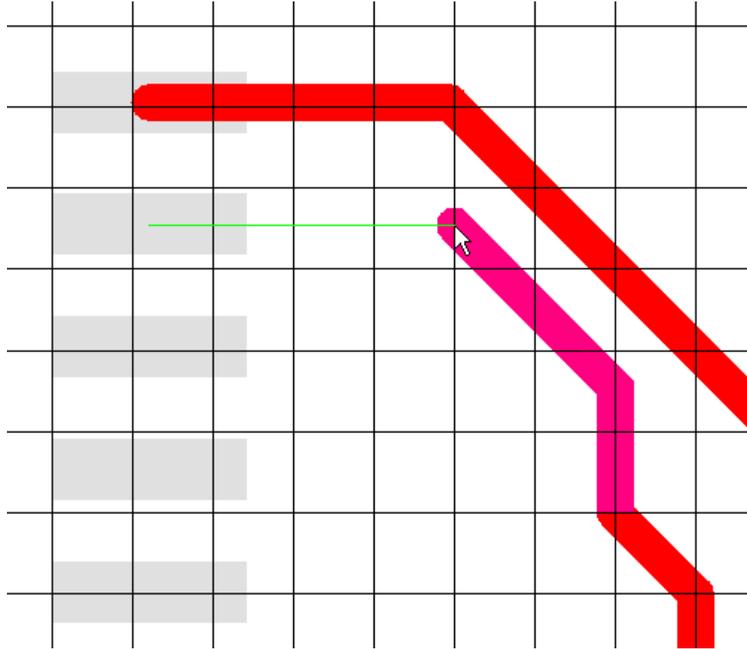
A new check box has been added in the **Suppress Components** settings on the **Component Properties** dialog.

Checking this box will tell the **DRC** option to ignore this component when doing Component-to-Component checking. Sometimes there is an instance where you know two physical components can sit together perfectly fine without DRC errors but there is no ideal mapping of this situation in the Easy-PC program. This option allows for this.



Snapping Improvements (PCB)

In the **Add/Edit Shape**, **Schematic Connection** or **Track** options, Easy-PC will now always snap to any position orthogonal to the start point of the dynamic segments being added, even if this point is off-grid.



Editing any existing segment will now always snap to any position orthogonal to the target point, i.e. the end of the free angled trailing segment, even if this point is off-grid.

Add Track will now always snap to any position orthogonal to the target node, i.e. the node at the far end of the unrouted connection attached to the end of the track being added, or the nearest node (if this is being displayed).

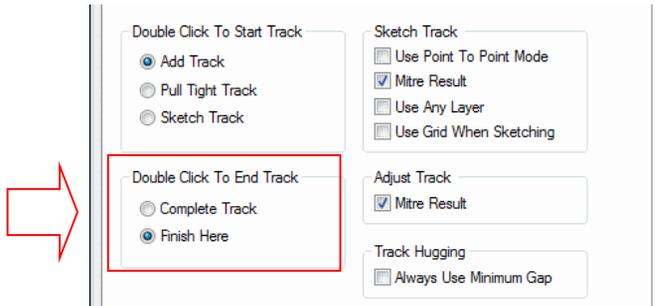
Move and Move Corner

Move and **Move Corner** will now always snap to any position orthogonal to the original position of the selected items or corner.

All of these improvements make it easier to work with off-grid items, and keep segments orthogonal when using a small grid.

Double Click To End Track (PCB)

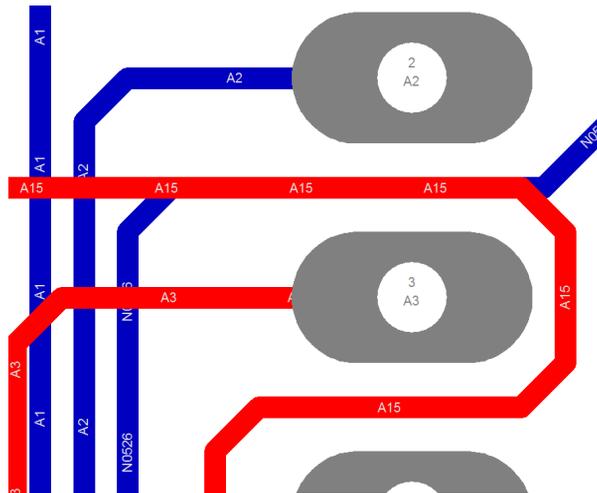
There is now a new option in the **PCB Tracks** dialog in **Preferences** for you to choose what **double click** does when finishing tracks.



You can choose between **Complete Track** or **Finish Here** (like release 17). If Complete Track is not available at the time (not on context menu) then double click with this option will do a Finish Here instead.

Display Names in Pads and Tracks (PCB)

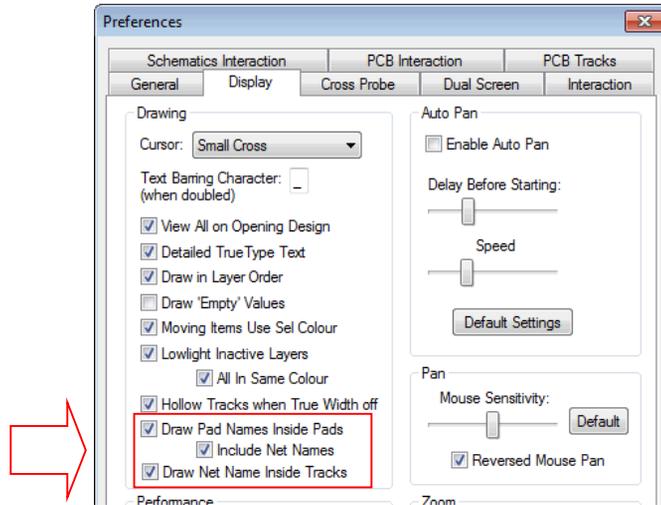
In a PCB design, the net names can be drawn inside track segments and pad names (optionally plus net name) drawn inside pads.



The names are drawn in complimentary colours so they are visible. The text style is proportional to the track thickness so that it fits. Note, this may be very small text if the tracks are particularly small, like 4 thou (mils) / .100mm (100 Microns).

When zooming out, when the text becomes too small, it is not drawn in.

This is controlled by new check boxes on **Display** page of **Preferences** dialog.



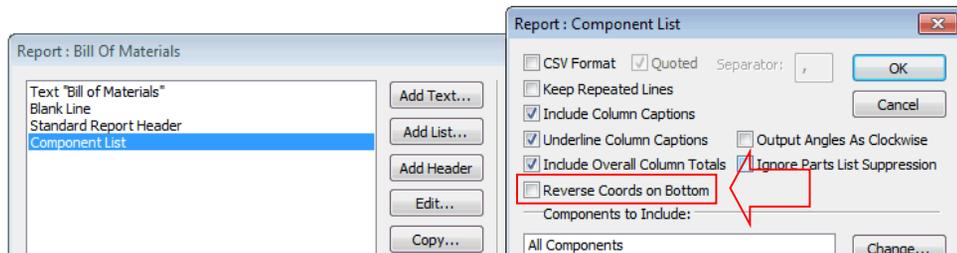
Draw Pad Names Inside Pads - with this enabled, Component Pads in a PCB design will show their pad name at the centre of the pad.

Include Net Names - in addition to the above, the net name can also be displayed. For names longer than 6 characters, the name will be abbreviated like "NAMING..." to avoid it overhanging the edge of the pad by too much.

Draw Net Name Inside Tracks - with this enabled, any straight (non-arc) track segments in a PCB design will have their net name displayed at intervals within the track.

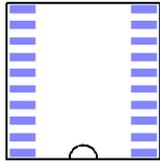
Reverse Mirrored Component Coordinates (PCB)

If you use mirrored components in your design, you can now reverse the coordinates on outputs for the bottom side in reports. There is an extra check box on the **Component List** sub-dialog of the main **Report** editing dialog.

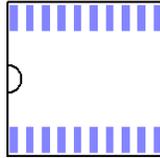


Rotation Correction Value in Report output (PCB)

Where footprints have been created and saved in the library, using different default rotations, perhaps from different sources or from different designers, you can now correct the default rotation reported in **Properties** or in an output report by adding a new **Value** attribute called **Rotation**.



Footprint created at zero rotation but at 90 degrees to the true zero



Footprint created at 'true' zero rotation

Use the component value attribute called **Rotation** to hold a correction value to be applied to the standard component angle.

The rotation value can be added to the Component definition in the library or locally on each component that requires it. Be aware though, if added to components in the design, the value must be added to every component in the design that requires it, not just one instance. If adding it to the Component definition in the library, it will only need to be added once.

Using the Value Rotation

Add a **Value** attribute of **Rotation** to the component. Give it a value of the desired 'correction' angle in degrees. When the Rotation is output to formatted reports, this value will be added to the Angle of the component as shown in **Properties**. The value can be positive or negative to indicate counter-clockwise or clockwise correction, for example Value Rotation = -90 or 180 etc.

Board size in Design Status Report (PCB)

When writing out the standard **Design Status** report from the **Reports** option, the **Board maximum Extents (X & Y)** are now included in the report. Sizes displayed are consistent with those shown on **Status Bar** and in **Properties** when selecting a board outline.

```

Design Status Report
Report File: c:\temp\CPP (PCB - Design Status Report).htm
Report Written: Thursday, July 03, 2014
Design Path: C:\Documents\Easy-PC\Examples\CPP.pcb
Design Title: CPP Example Board
Created: 02/07/2014 09:20:04 AM
Last Saved: 03/07/2014 10:50:08 AM
Editing Time: 2756 min
Units:

```

```

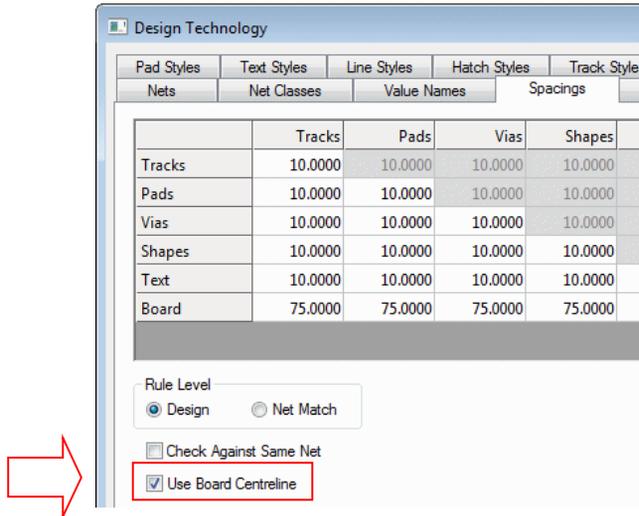
Board Area
37537500.0000 thou squared
Board Extents
X:6500.0000 thou Y:5775.0000 thou

```



Use Board Centreline for Spacing Rules (PCB)

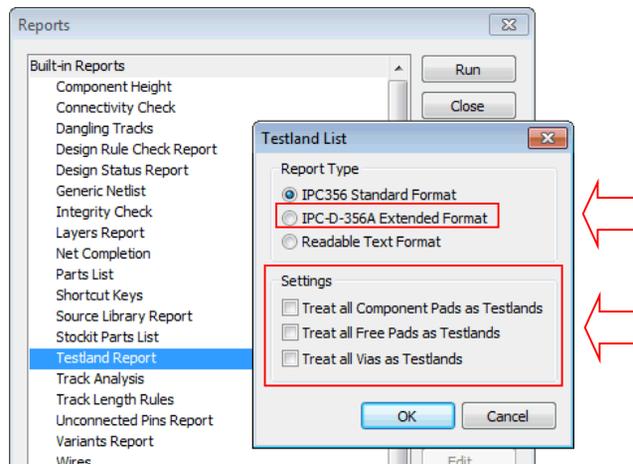
Check the **Use Board Centreline** box to ignore the width of all board outlines during DRC. Checking would then be done up to the centreline of the board segments and not the outer line width. This could be used in the panel editor to allow PCB boards to be touching for use in 'V' Cut production.



Testland Output (PCB)

Full Extended IPC-D-356A Output

In Easy-PC 18, the ability to generate a full **extended IPC-D-356A** format netlist has been added. This is available from the existing **Reports** dialog using the existing **Testland Report** output.



There are additional buttons on the Testland List report.

IPC-D-356A Extended Format

This report details all net nodes in the design, formatted using the IPC standard format for testland netlists (IPC-D-356A), like this:

```
327uP_TO_FPGA    U1    -F17    A01X-026535Y-007451X0177Y0177
327uP_TO_FPGA    U3    -J16    A01X-024291Y-008199X0197Y0197
317uP_VDDA       VIA    -NPI    NMD0079PA00X-030787Y-007923
327uP_VDDA       U1    -G4     A01X-030630Y-007766X0177Y0177
```

Note that because of the way the IPC format is defined, this output will only include testland information on connected items. Any item that is not connected to a net will not appear in the output.

New Testland Settings

Settings

New check boxes in the **Settings** section can be used to force all items of a particular type to be output. Choose whether to include **All Component Pads**, **Free Pads** and/or **Vias** in the report regardless of whether or not they have the Testland property enabled

Note that this does not apply to full IPC-D-356A because that is a defined netlist format which needs to include net names.

Shift Select for Cutouts (PCB)

When editing a **Cutout** shape, you can now use the **Shift pick** to select the whole shape in one. Previously, you could not do this. Once selected, the shape can be moved.

Chapter 3. Updated Libraries

Pro Library 7 (Updated Cost Option)

The Pro-Library has been updated to include over 6,600 additional Parts and their associated Symbols and Footprints. This takes the contents of the Pro-Library to just over 89,200 Parts! If you have purchased any previous versions of this library already, we can supply this update to you for a small upgrade charge when you upgrade to Easy-PC 18.

Note: for all updated library contents, the exact quantity may change slightly from the numbers above at the time of release. Please use the Library Finder page on our web site to check for any required parts. E&OE

Micro Library 7 (Updated Cost Option)

The Micro-Library has been updated to include over 1,640 additional Parts and their associated Symbols and Footprints. That's over 10% extra! This takes the contents of the Micro-Library to just over 11,300 Parts! If you have purchased any previous versions of this library already, we can supply this update to you for a small upgrade charge when you upgrade to Easy-PC 18.

The Micro Library 7 update is made up of 543 Renesas micro-controllers, 992 Atmel devices (ATxx various, ATMEGA, ATTINY) and 114 Xilinx (Spartan 3, CoolRunner-II).

Note: for all updated library contents, the exact quantity may change slightly from the numbers above at the time of release. Please use the Library Finder page on our web site to check for any required parts. E&OE

Find Library Parts using the Library Searcher

A list of all the updated contents and current library contents can be located on our web site in the online library searcher www.numberone.com/libfinder.asp

Finding New Contents for ProLib 7

Select **ProLib** or **MicroLib** in the **Library:** drop down list, then type **7** into the **Version:** box.

Update to the Library Finder

You can now use the **Library Finder** to search for contents of the IPC SM Library. On the drop down list, under **Library:** select **IPC SM** and it will display the entire contents. Note: Bear in-mind that this is a PCB footprint library and so will not have manufacturers component names.