



V15.0 Supplement

Copyright ©1987-2011 by 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: 03/10/11 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 For Windows 15.0	5
CHAPTER 2. NEW FEATURES IN EASY-PC V15.0.....	7
Introduction.....	7
Library Manager Changes.....	7
Ability to export a 'package' of library data as a single file.....	7
Component Edit - Allow the same pin on different gates to have different names	9
Change a Component to Schematics Only.....	11
Library Styles Manager (Both).....	11
Source Library Report (Both).....	13
Grids (tables) (Both).....	13
New Goto (Find) Categories (Both).....	14
Measure Tool (Both).....	14
Rename Component (Both).....	15
New Align Options (Both)	16
Select Ignores Fixed Items (Both).....	16
Select Only If Completely Framed (Both).....	17
Minimum Pick Tolerance (Both).....	17
Drag Sensitivity (Both)	17
Polygon Select (Both).....	18
Export/Import Technology CSV (Both).....	19
Copy/Paste Nets across Project (SCM).....	19
Common SCM Nets across Project (SCM).....	19
Nets Technology 'Add Project Nets' (SCM).....	20
Apply SCM Colours across Project (SCM)	20
Schematic Net changes in Projects (SCM)	21
Panel Editor (PCB).....	21
Trunk Routing (PCB).....	27
Cut Track (PCB).....	29
Profiling Shapes to Excellon Output (PCB)	29
Net to Net Spacing Rules (PCB).....	30
Hollow Tracks (PCB)	31
Show Legal Completion Path (PCB).....	32
Arrange Components (PCB).....	33
Optimise to Power Planes (PCB).....	34
Define Number of Thermal Pad Spokes (PCB)	34
Delete Track does Unroute Preference (PCB)	34
Remove Attached Tracks Options	35
Lowlight Inactive Layers (PCB).....	35
3D Viewer Changes (PCB).....	35
New 3D Package Shapes.....	35
Rounded Ends.....	36
Colour-coded Bands on Components	36
3D View - Save As Image	37
Option to choose Layer Type for Component Body.....	37
New Library Contents Searcher on Web Site	38
Micro Library 4 (Updated Cost Option).....	38
Pro Library 4 (Updated Cost Option).....	39

Chapter 1. Getting Started

Installation

Installation is via the *autorun* setup. If you are not familiar with this process it is explained briefly below:

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.

The installation is the same for new and existing users alike. Existing users i.e. version V3.0 to V14.0 can install V15.0 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 For Windows - Version 15.0** 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 e.g. C:\Program Files\Number One Systems\Easy-PC

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

Remember to back-up all your libraries, not only for the upgrade but also at all times during design.

Data Files Location

There is an extra 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 Vista and Windows 7, "Documents and Settings\All Users\Shared Documents" on XP etc. if you are installing for All Users, or into your own Documents folder if installing for current user only.

Running Easy-PC For Windows 15.0

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

Chapter 2. New Features in Easy-PC V15.0

Introduction

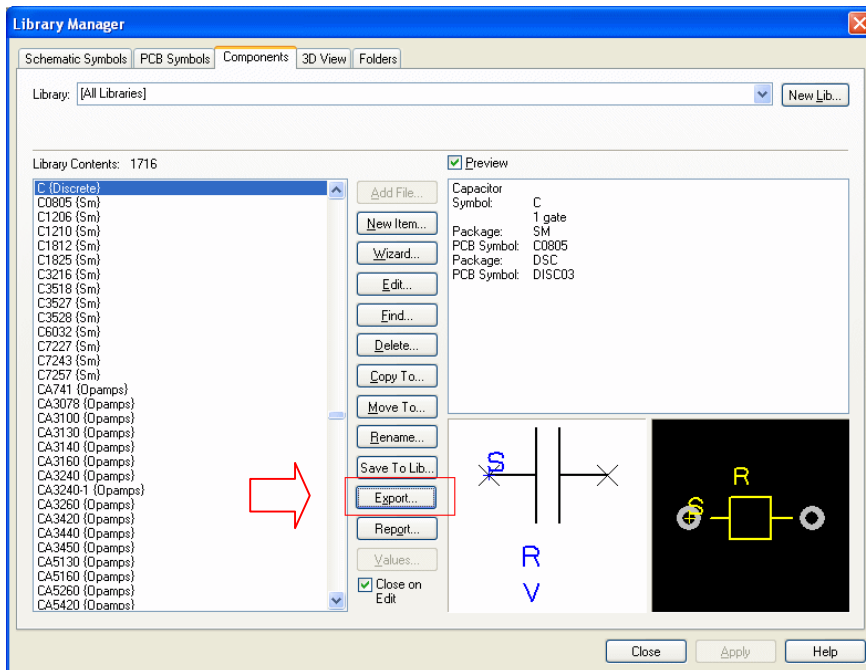
All features are categorised as being SCM specific, PCB specific or Both (relevant to both SCM and PCB design applications).

Library Manager Changes

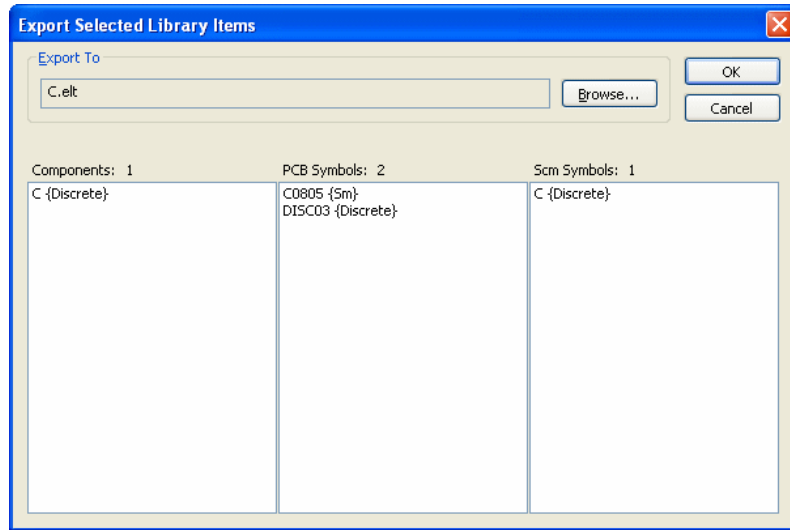
Ability to export a 'package' of library data as a single file

This option allows you to easily 'package' components into a single 'Easy-PC Library Transfer' file for moving to another system. As well as the Components, all the Symbols that are referenced by those Components will be included in the Transfer file.

A new **Export** button on the Components tab of the **Library Manager** allows you to save the selected components and all the symbols they reference to a single transfer file. You can select single components or multiple components using the normal Shift, Ctrl or combinations of these keys.



With a single component selected in the library, the **Export** button reveals the Export dialog:



The top part of the dialog shows the filename to which the library data will be written. In this example, the 'C' component which will create the c.elt package.

Underneath this are three lists, one for each type of library item. In the **Components** list will be shown all the components that were selected in the Library Manager dialog. The PCB and SCM Symbol lists will show all the symbols that are referenced by the components.

Selecting items in the lists

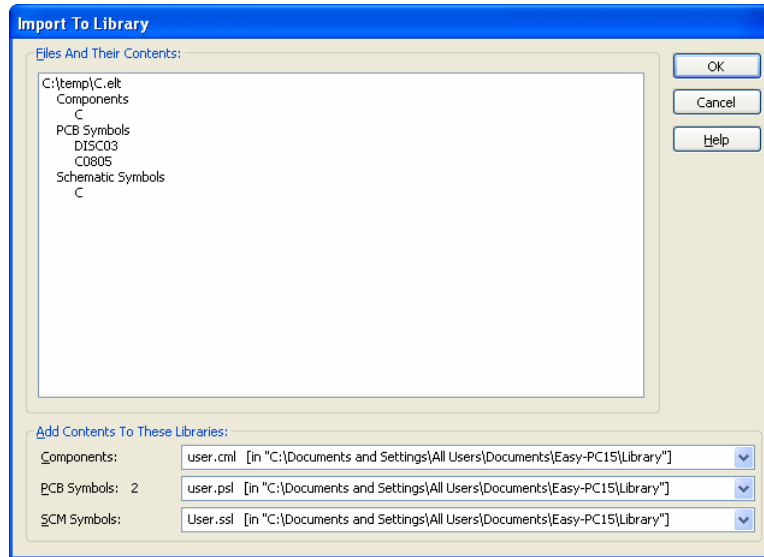
If you click on any of the components in the left-hand list, the PCB and Schematic Symbols which are referenced by the components you have selected will be shown selected. Conversely, clicking on any of the symbols will show you all the components that reference that symbol.

This feature can be used to 'explore' the relationship between the components and symbols you are going to save. This will not affect what is actually written to the target libraries, it is only done for information.

Importing 'packaged' libraries

To use the library items packaged up in a **Library Transfer** file, simply drag and drop the file onto the Components page of the **Library Manager** dialog, or use the **Add File** button to browse to where the file is stored, or double click on the filename from within a Windows Explorer browser dialog (the file extension .elt will automatically be mapped by the Easy-PC installer).

When you do this, you can choose the library file into which each of the saved 'elements' (Component, PCB Symbols and/or Schematic Symbols) are stored.

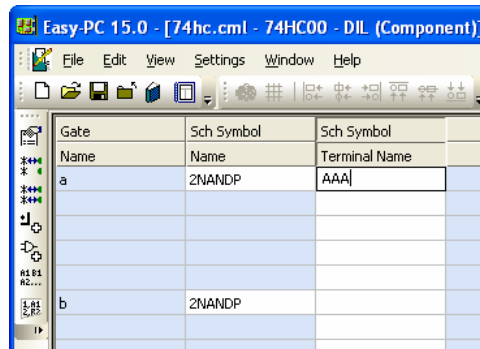


In order to select a library from the list, it must already exist. If you require a new library name, use the **New Lib** button on the **Library Manager** to do this.

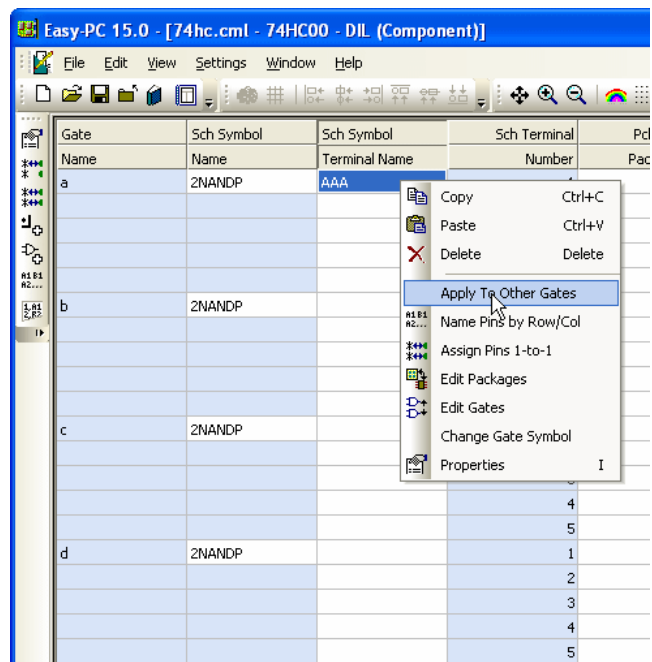
Component Edit - Allow the same pin on different gates to have different names

Previously changing the terminal name on a pin would replicate this change across all the other gates that used the same symbol. Now the default action is not to do this, but you can use the **Apply to Other Gates** command on the context menu to replicate the name to the other gates as it used to.

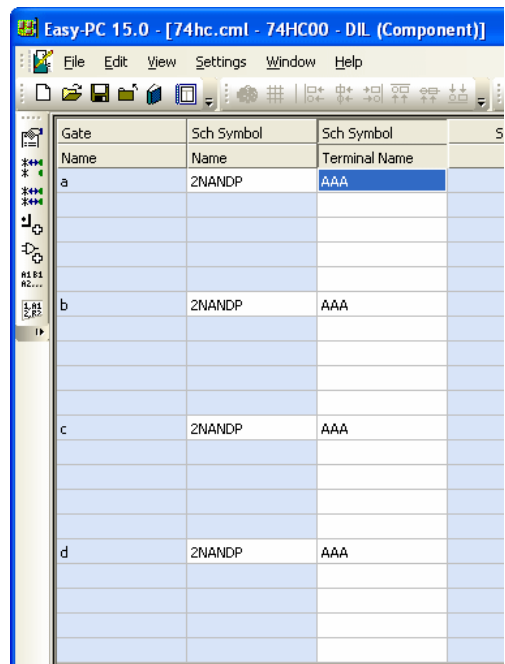
Edit the first **Terminal Name**:



Right click on the new name, the one you wish to propagate to the other cells.



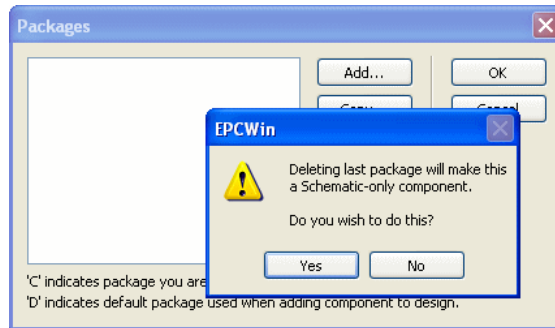
Click the **Apply To Other Gates** option from the menu.



The **Terminal Name** is now applied to the corresponding cells.

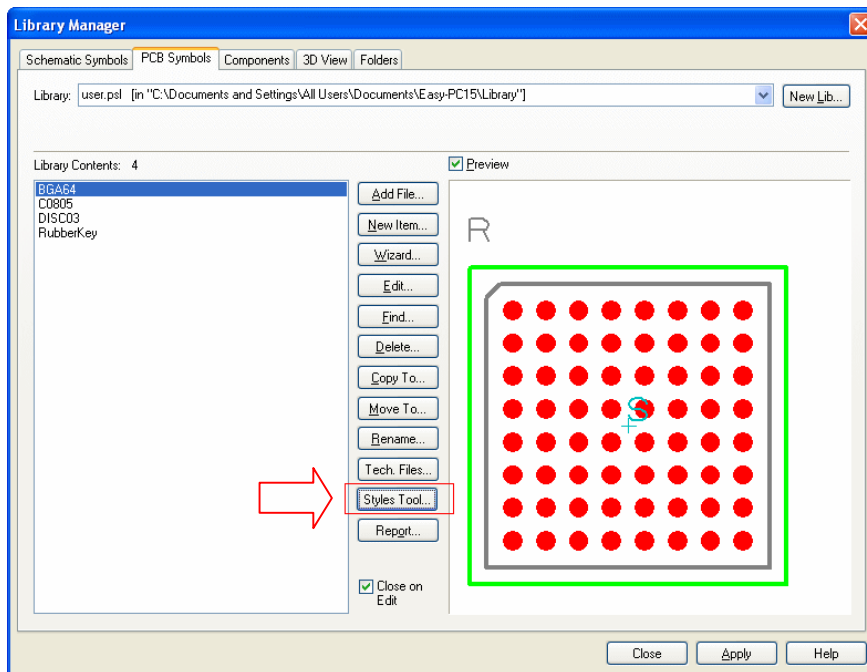
Change a Component to Schematics Only

This change now allows a component to be changed to Schematic-only by deleting the last package so that none appear in the **Packages** list of a component. Previously it would not allow you to exit the Packages dialog if you had deleted all the packages. Now it will ask you instead to confirm that you really do want to make it a schematic-only component. It will allow you to exit if you press the **Yes** button.



Library Styles Manager (Both)

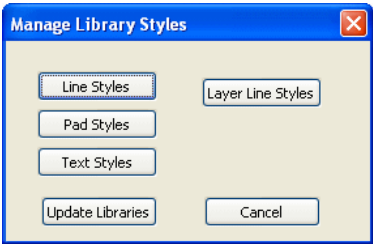
On the **Schematic Symbols** and **PCB Symbols** page of the **Library Manager** dialog, there is now a **Styles Tool** button.



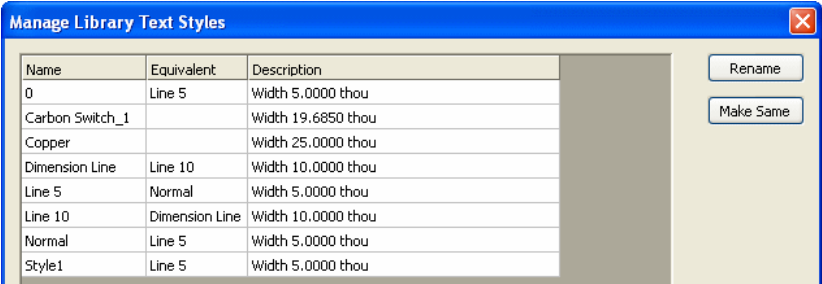
The Style Manager operates on selected library items, all in the selected library or all in enabled libraries depending on whether you have Symbols selected or [All Libraries] selected.

A **Technology** file needs to be set up in the Schematic/PCB Symbol pages of the **Library Manager** under **Tech Files** for the **Styles Tool** to be enabled.

When selected, you are presented with the **Manage Library Styles** dialog:



Buttons on this dialog allow you to manage different **Styles** and **Layer Line** styles. Each dialog will display which styles are used and the ability to rename them or merge them with other styles.

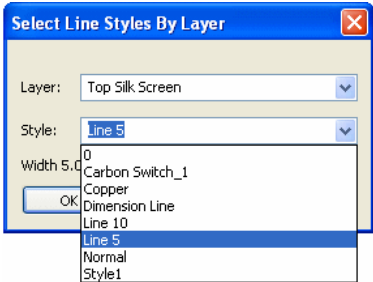


Where a style matches by its value, the style name will be displayed in the **Equivalent** cell next to it. So in the example above, style name **0**, **Line 5**, **Normal** and **Style1** all use the same width. This is perfectly acceptable if your wish to differentiate between different items using the same style value or thickness but where common items use different names, you may wish to consolidate them.

Where style names exist but the name is meaningless, for example, Style1, you can select the style name and press the **Rename** button. Type in the new name, this might be **Pins Names** for example.

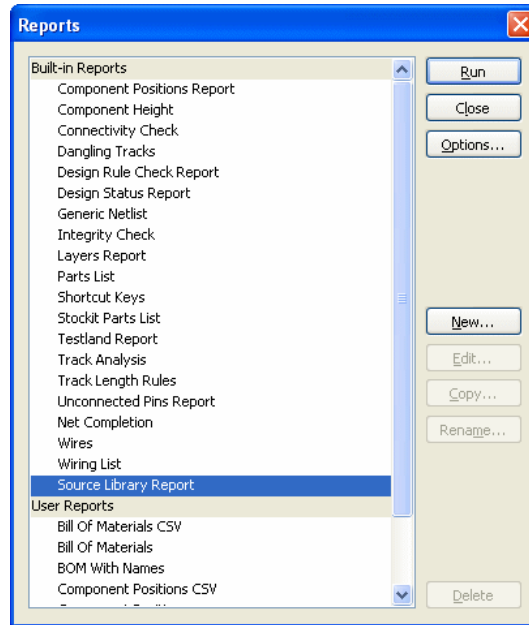
To consolidate names, you select the cell of the style name you wish to keep. If we wish to keep **Normal**, and make **Line 5** the Normal named style, you would select **Normal** from the list and press the **Make Same** button. **Style1** will also change to **Normal**.

On the **Manage Library Styles** dialog you can also select a pin style to use on all schematic symbols or select which line style to use in footprint layers.



Source Library Report (Both)

Run from the **Output** menu and **Reports**, the new **Source Library Report** will, when run from a Schematic or PCB design, report which library each different Component, Symbol and Footprint comes from. When viewing the report, be aware that the library reported may be different to the one it was actually loaded from.

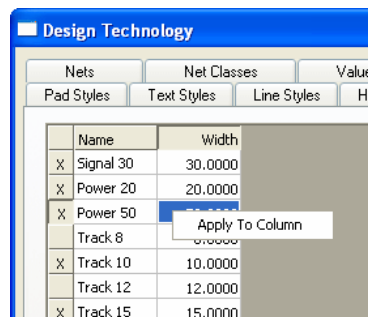


Grids (tables) (Both)

A new Grid control has been introduced for all dialogs that display information in a grid, this includes the Component Editor amongst others.

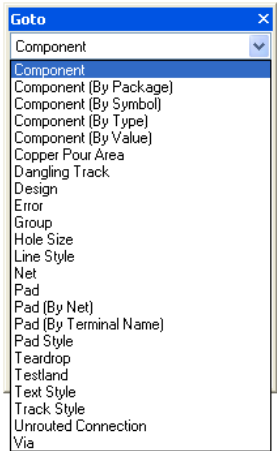
Where information within the grid extends further than the one page displayed, you can now use the wheel button of the mouse to scroll up or down the page to see this information.

Where it is applicable to take the current cell data and copy it to additional rows or columns, by right clicking the mouse in a cell, a **Apply To Row/Column/All** menu option will be available.



New Goto (Find) Categories (Both)

There are new Goto categories introduced to Version 15:



New PCB categories:

Component (By Package)

Component (By Symbol)

Copper Pour Area By net, but includes <No Net>.

Dangling Track By net

Hole Size Unplated hole sizes listed separately with **Unplated** appended.

Teardrop By net, but includes <No Net>

Testland By net, but includes <No Net>

Via By net

Plus new Schematic categories:

Component (By Package) Not project wide

Component (By Symbol) Not project wide

Unrouted Connection By net, this obeys **Show All Unfinished Connections** switch in highlight **Display** dialog

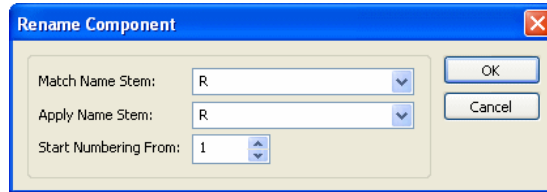
Measure Tool (Both)

The **Measure Tool** is now available in all interactive design formats, not just PCB designs. This includes the Schematic editor and PCB Footprint creation to name two specific uses.

Rename Component (Both)

You can now use an interactive point-and-click component rename in the PCB and Schematic editors.

From the **Tools** menu, select **Rename Components...**



The Rename dialog allows you to choose the component name 'stem' to look for, then click around your design in the desired order to rename components in sequence.

You can choose from the available component name 'stems' in the design. You can also choose the name stem to use when applying the renames - by default this is the same as the stem you are matching, but it can be different. For example you might wish to rename some connectors starting with **CON** to be named as **PL** instead.

You also need to enter the start number from which the component names should begin.

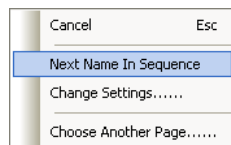
When you click **OK**, the dialog closes and the 'point and click' renaming can begin. All the components matching the chosen stem will be highlighted to help you see them easily on your design.

Each time you select a component, the current name in the sequence is assigned to that component, and the sequence number is incremented. The next name to be applied is always displayed on the Status Bar.

If the next name to be applied is already in use elsewhere in the design, the application will ensure that the names are swapped over so that no duplicate names exist.

Other Actions

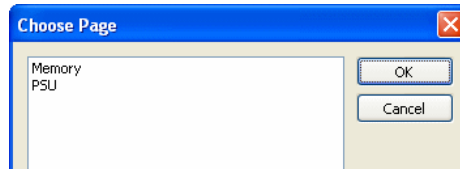
During its use, you can select additional options from the context menu by clicking the right mouse button:



Next Name In Sequence : this simply increments to the next available component name.

Change Settings : this will re-display the dialog allowing you to change settings, for example to switch the name stem to start renaming a different type of component.

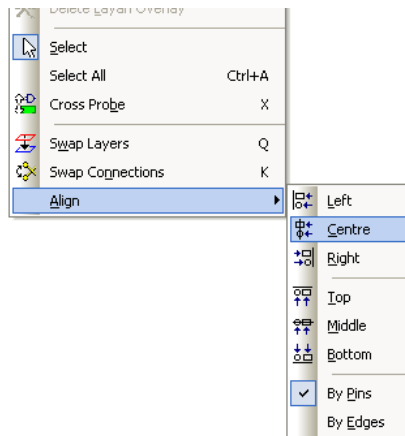
Choose Another Page : if you are currently working on a Schematic design in a Project that has more than one Schematic page, this option allows you to switch to another page in the Project. A small dialog is displayed showing the names of the other pages, on choosing one that page will be opened and the Rename command automatically invoked so that you can start renaming on that page instead.



Press the **Esc** key or choose **Cancel** from the popup menu to finish the renaming operation.

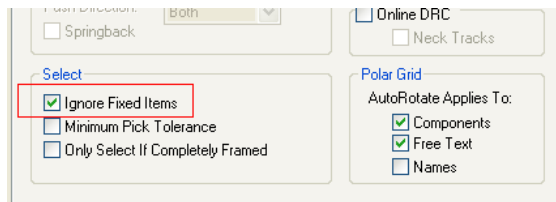
New Align Options (Both)

Additional commands for centre alignment (horizontal) and middle alignment (vertical) been added to the Align tool. All the Align functions will now also align 'unlike' items where previously you could only align items of the same type.



Select Ignores Fixed Items (Both)

Within the **Preferences** dialog, the option, **Select Ignores Fixed Items** has been moved from the **General Interaction** box on the **Interaction** page to the **Select** box on the **PCB Interaction** and **Schematic Interaction** pages. This change has been made to collect together other new 'select' type options.



Select Only If Completely Framed (Both)

Within the **Preferences** dialog, on **PCB Interaction** and **Schematics Interaction** pages there is a new selection option under the **Select** box for **Select Only If Completely Framed**. This is used by both the **Frame Select** and the new **Polygon Select** modes.

Check this option to only select whole items that are completely within the frame or polygon. The default is unchecked to behave like it did previously, that is, to Frame Select even if the item isn't completely framed.

When using this option, if the Component Reference Name is outside the selection of the component, it is still selected to ensure it is included.

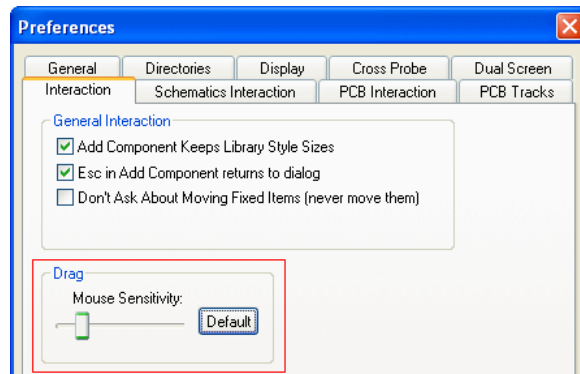
Minimum Pick Tolerance (Both)

Within the **Preferences** dialog, on **PCB Interaction** and **Schematics Interaction** pages there is a new selection option under the **Select** box for **Minimum Pick Tolerance**.

Check this option to make picking more accurate, especially useful when picking whilst zoomed out and when using a large high definition monitor. The default is unchecked, to behave like it did previously.

Drag Sensitivity (Both)

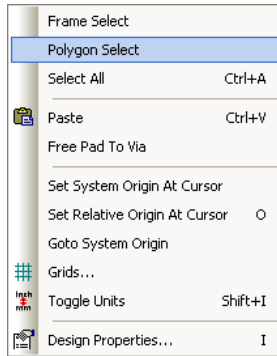
Within the **Preferences** dialog, on **Interaction** page there is a new option **Drag**, this adjusts the **Mouse Sensitivity**. This allows you to tune the drag responsiveness of the mouse when used in the design window. This is the distance the mouse has to be dragged before the application recognises the operation as a mouse drag and acts accordingly. The reason for this option is to help avoid getting mouse drags when you are trying to perform a click whilst the mouse is slightly being moved.



The **Default** button sets it to the in-built value that gives a balanced distinction between mouse clicks and drags. This is the value used for V14 and previous versions.

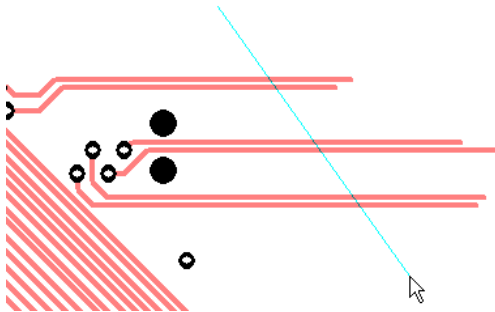
Polygon Select (Both)

In select mode you can now use **Polygon Select** from the context menu. In this mode you define a polygon by clicking at its corners. Use **backspace** to delete a corner and double click to finish. Any item crossing the polygon will be selected.

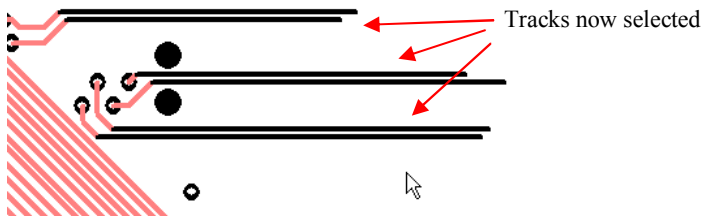


You can also use this mode as a single line for selection by dragging the cursor. This is useful for selecting angled tracks for example. The single line option ignores the **Select If Completely Framed** setting.

To do this, enter the **Polygon Select** mode, drag a line (not a polygon) across items you wish to select:



The items under the line are then selected once the cursor is released.

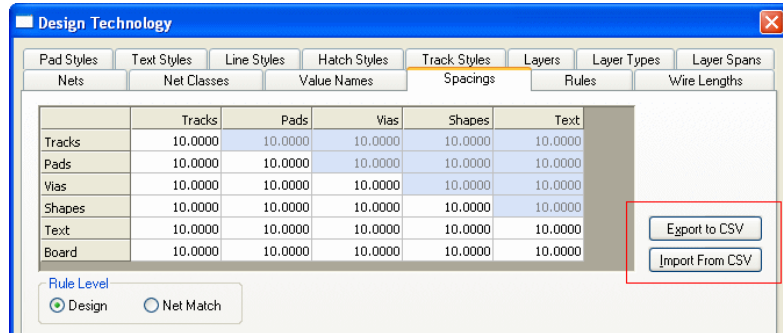


Like **Frame Select**, **Shift key down** means select whole items, and if the **Control key** is down nothing is deselected before selecting the items with the shape.

Export/Import Technology CSV (Both)

All tabs of the **Design Technology** dialog (except for **Wires** and **Rules**) now have buttons to **Export To/Import From CSV** format files. The exported files can be easily edited in Excel for example, then imported back in with the new values.

During Import, checks are made to ensure the right number of fields are imported and values are legal.



Once exported into Excel, the file looks like this:

	A	B	C	D	E	F	G
1	Object Type	Tracks	Pads	Vias	Shapes	Text	
2	Tracks	10.0000thou	10.0000thou	10.0000thou	10.0000thou	10.0000thou	
3	Pads	10.0000thou	10.0000thou	10.0000thou	10.0000thou	10.0000thou	
4	Vias	10.0000thou	10.0000thou	10.0000thou	10.0000thou	10.0000thou	
5	Shapes	10.0000thou	10.0000thou	10.0000thou	10.0000thou	10.0000thou	
6	Text	10.0000thou	10.0000thou	10.0000thou	10.0000thou	10.0000thou	
7	Board	10.0000thou	10.0000thou	10.0000thou	10.0000thou	10.0000thou	
8							

All values are written in design units. If you change the Units in the CSV file for one of the values, to 1mm say, the imported value will be converted to the current design units.

The CSV file can also be edited in a text editor but care must be taken to retain the correct file formatting.

Copy/Paste Nets across Project (SCM)

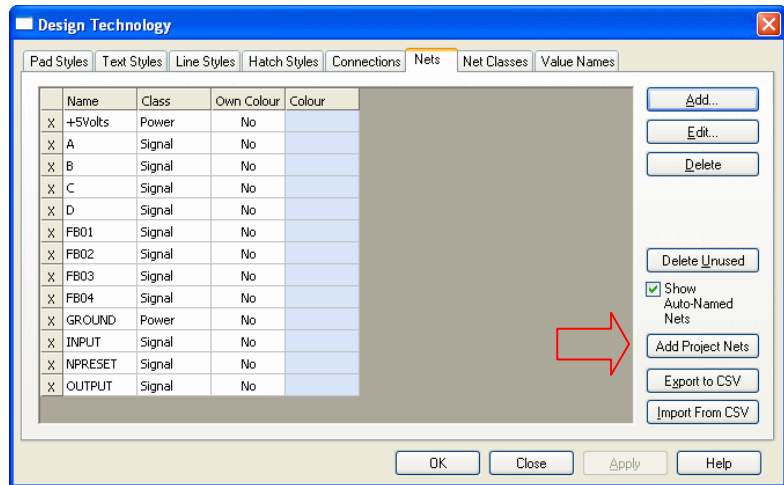
Pasting a net into a **Project Schematic** sheet now displays the **Paste Net** dialog if the net already exists in the project regardless of whether or not it is actually defined in the target sheet. Also, if merge is chosen, the net will inherit the attributes of the existing project net even if it has been copied from outside the project.

Common SCM Nets across Project (SCM)

When adding a new **Net** or **Net Class** to a schematic design that is part of a **Project**, if a Net Name or Class that already exists on another sheet is chosen its Properties will be automatically copied with the name to the current sheet.

Nets Technology 'Add Project Nets' (SCM)

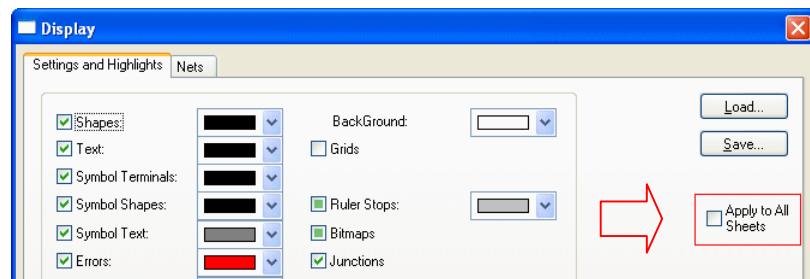
The **Nets** tab in the **Design Technology** dialog now has a **Add Project Nets** button if a schematic design belongs to a project.



When selected, it will bring user-defined nets from all other Schematic designs in the Project that are not already defined in the current design to the list of nets shown in the grid. The settings for these nets may be modified in the dialog and will not be added to the design until **OK** or **Apply** is pressed, at which point the nets in the other project designs will also be synchronised to the values defined in the grid.

Apply SCM Colours across Project (SCM)

The **Display** dialog for a schematic design that is part of a **Project** now includes an additional option **Apply to All Sheets**.



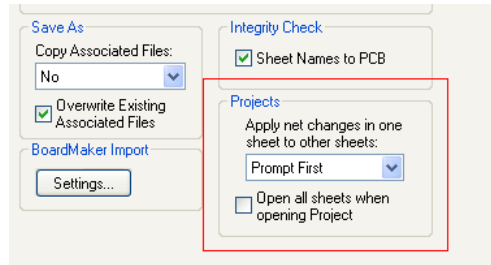
When checked, the colours displayed for the active schematic design will be applied to all other schematic designs in the project when **OK** or **Apply** are clicked. This includes nets colours for nets present in the active design.

When a **Colour File** is loaded, any net colours it contains will be applied to all schematic designs in the project according to the current net propagation method. Other colours can be applied to the other schematic designs by using the **Apply to All Sheets** option.

Schematic Net changes in Projects (SCM)

When using a **Project**, any changes to Nets and Net Classes can be propagated to the other sheets in the Project. This includes changes to net name, net class, own colour and net class name and type. Auto-named nets are not affected.

A new option on **General** tab in the **Preferences** dialog allows the method of propagating net changes to be defined using three options; **Prompt First**, **Automatically**, or **Never**. If the **Prompt First** approach is chosen, the ability to choose **Yes To All** is included on the prompt dialog when multiple nets are affected.



Any affected schematic sheet not already displayed is opened automatically. An undo item is created on each of the affected sheets for each change. The net changes are not committed for each of the schematic designs until they are saved (or a **Save All** is done).

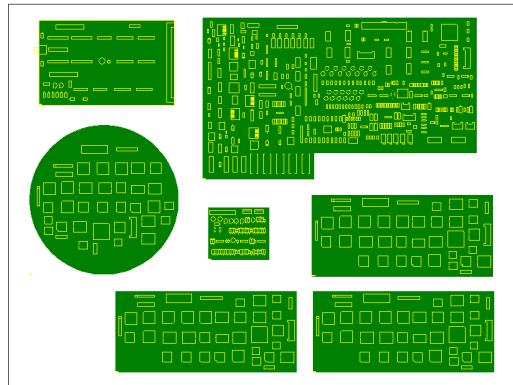
This also means that net information in **Technology Files** can now be applied to a project as a whole as even if a Net or Net Class is not present in the active design the relevant information will be copied to other schematic designs in the project that do use that net or net class.

Panel Editor (PCB)

The **Panel Editor** has been added to Easy-PC Version 15.

A panel design is a special PCB design that contains a set of panel PCB items representing instances of finished PCB designs. You would use this for collecting PCB designs together on a panel and outputting them to manufacturing outputs such as Gerber, Exellon etc.

You create a new empty panel design, or use the **Panel Wizard** to create a panel design with a set panel size and optionally include multiple instances of a PCB design.



Creating a New Panel

A new panel design can be started from the **File** menu, **New** and **Panel Design** or **Panel Wizard** from the **Wizard** tab. This is not part of the PCB design editor, it is a new design type.

A Technology file can be used, this acts the same as a PCB design Technology file.

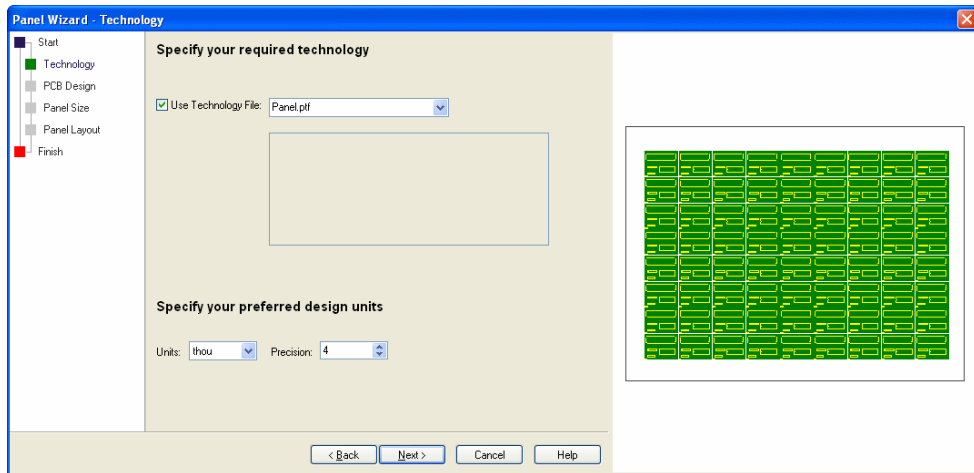
Once the panel design has been created, you can add a **Panel** outline from the **Add** menu, similar to using **Add Board**. You can also add a Panel Outline using the **Panel Wizard**, or by importing a DXF file.

PCBs can be added to the panel using the **Add PCB** option from the **Add** menu. Single or multiple PCBs can also be added using the **Panel Wizard**. It should be stressed that only Easy-PC PCBs can be added to the Easy-PC Panel Editor.

Panel Wizard

The **Panel Wizard** allows you to create a new panel with a panel outline and a single or multiple PCB designs added in an array. Run this from the **File** menu and **New Wizard** page and **Panel Design**.

The **Technology** page allows you to define your **Technology** file up-front and to specify the **Design Units**. This is a PCB design Technology file.



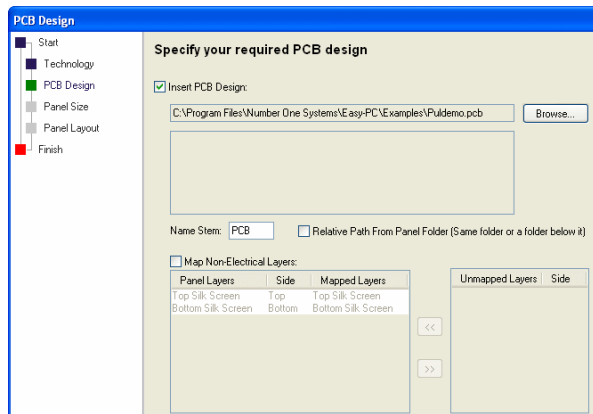
The preview window displays the panel as it will appear in the design once the Finish button has been pressed.

Use the **<Back** and **Next>** buttons to navigate the Wizard. Alternatively, click on the words in the left hand panel on the page required.

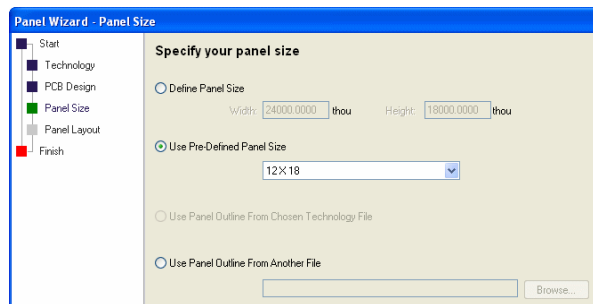
The **PCB Design** page enables you to add a design to the panel. This is not mandatory if you only wish to create a panel outline at this stage. If you don't wish to add a PCB, uncheck the **Insert PCB Design** check box.

Where inserted PCB designs contain different layer names of non-electrical layers to that of the Technology file, you can use the **Map Non-Electrical Layers** option to map them to the correct layer name. In order to add PCB designs to the panel, their number of electrical layers and layer spans must match the layer counts in the panel.

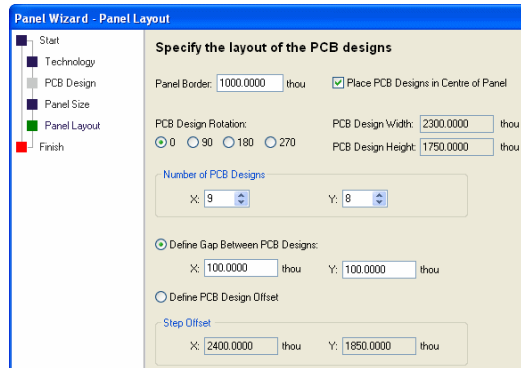
You can choose to reference the PCB design at its absolute file path, or have it remember the **Relative Path** to the file from the panel design. This allows you to save the panel design with the PCB design, and if they are both moved to different folders, the PCB design will still be found.



The **Panel Size** page allows you to define how big the panel will be. You can choose from a user defined **Panel Size**, a **Pre-defined Panel Size** or to use an existing Panel from the **Chosen Technology File** or another **Panel Design**.

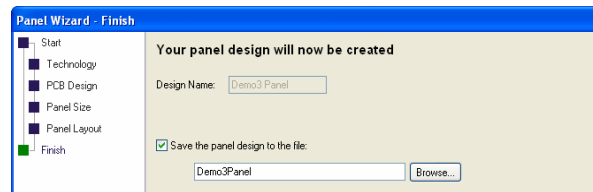


Use the **Panel Layout** page to lay out and position the PCB design within the panel. You can also use this page to define and array of PCBs to create a step and repeat panel.



If you try to leave the page and the PCB designs don't fit the panel or they overlap, you will be warned.

The **Finish** page is used to name the Panel and Save it to file for reuse.



Interactively Positioning the PCBs

PCB design instances can be added, aligned, moved, rotated, deleted and reloaded in the panel design. Each instance shows its board outline and shapes from each of their components to aid identification of the board.

A new **Zoom** option on the **View** menu is available to zoom to **Panel Outline**.

Checking the Panel

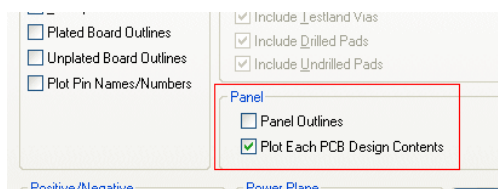
Check board separation, also allow boards to butt up against each other for 'V' cutter cutting.

Design Rules Checking - Design Rule Check will check that PCB designs are not overlapping.

Plotting the Panel

The panel can be output to ODB++, Gerber, Excellon, Windows, PDF, Pen etc. as you would any normal PCB design. You can also output reports such as assembly machine reports.

When plotting panel designs you can choose to include the full contents of the PCB designs, or just plot the simple displayed instance. These options are available on the **Settings** page within the **Plotting & Printing** dialog.



Adding Extra Detail

Extra detail can be added to a panel ready for manufacture: Copper, Dimensions, Callouts, Text, Free Pads, Bitmaps (although they cannot be plotted in Gerber), Drill Tables and Layer Stack preview. Inserted drill tables in the panel will show drill count totals for all PCB designs in the panel.

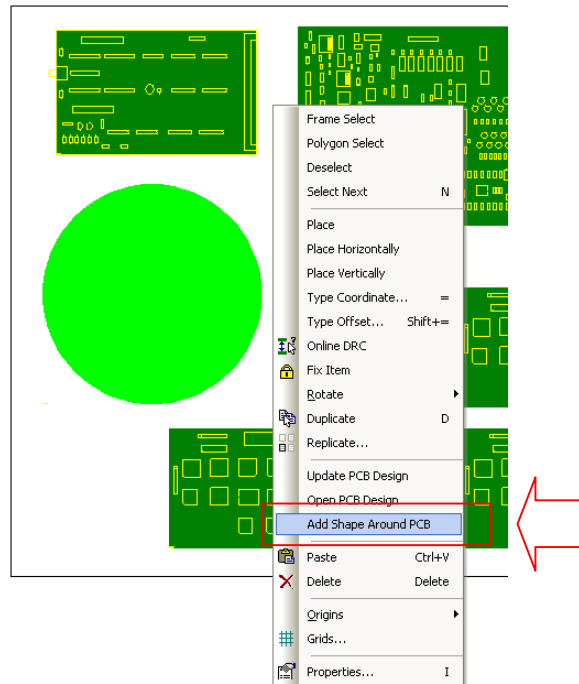
Components, such as Fiducial markers, can be added. These are used for assembly machine optical alignment or for mounting the panel onto the machine.

Non-electrical shapes can also be added, for example routing profile indicators which can then be output to Excellon or Gerber.

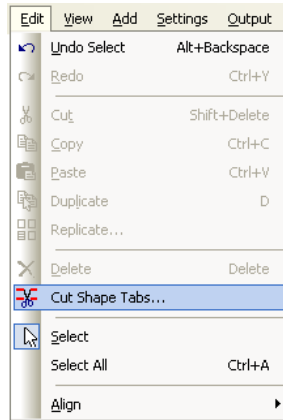
Routing Profile Shapes

There are two new features to aid the creation of routing profile shapes:

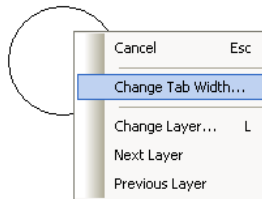
Add Shape Around PCB – Select one or more PCB instances and use this option from the context menu to add an outline using the **Default Shape Style** and **Layer** that fits tightly around each board. It is recommended that you create a new **Layer** and **Layer Type** to handle profiling (routing), for example, a layer named **Profile Layer**. The **Shape** size used should be your normal profiling tool size, consult your manufacturer for this information.



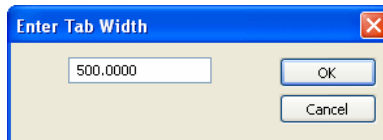
Cut Shape Tabs – From the **Edit** menu, this mode allows you to draw lines across shapes and have ‘tabs’ remove from them.



You can define the width of the tabs using the **Change Tab Width** option from the context menu once in this mode.

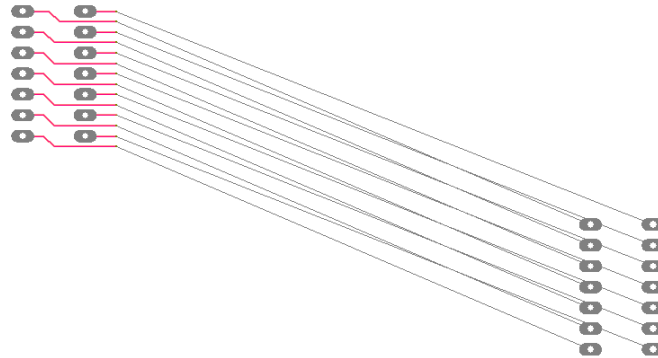


Type the value required for your tabs.

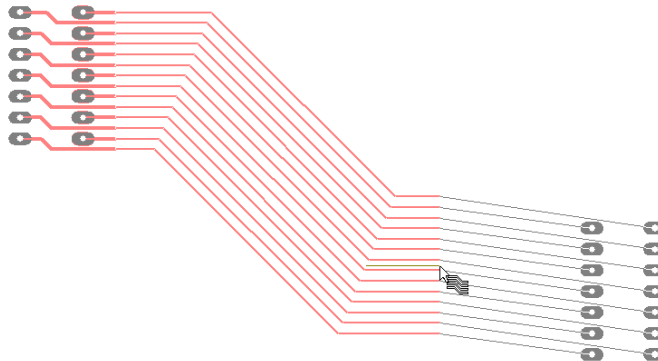


Trunk Routing (PCB)

Version 15 introduces a new PCB interactive function to add multiple parallel tracks at the same time, the **Trunk Router**. This has three phases, first select the items you want to start from, then position the line of junctions at the start of the Trunk Route, and lastly add the segments of the Trunk Route.



You can preselect the start items before entering **Add Trunk Route** on the **Add** menu, or you can enter the first phase of the mode and add or remove start items using left click, or select multiple items in one go by dragging using Frame Select. Clicking in space with no start items selected will prompt you for how many parallel tracks you want to add.

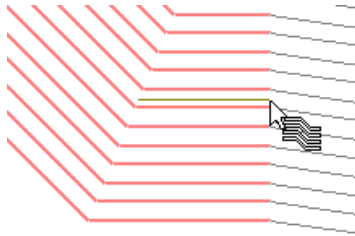


Click to add corners as you would when manually routing a normal track.

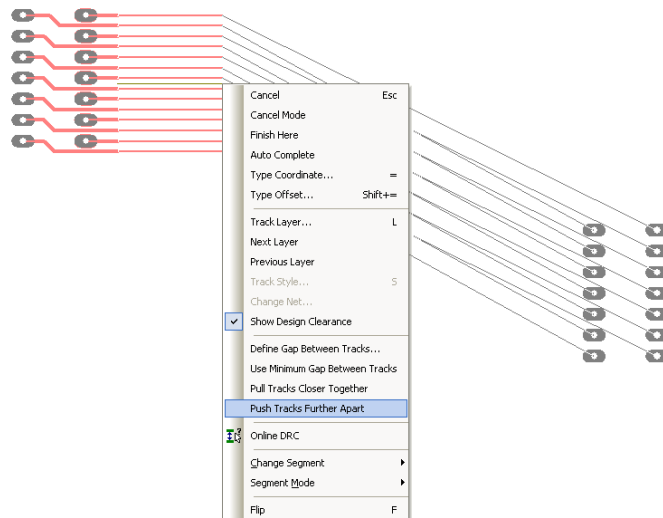
The new tracks will be separated by the appropriate track to track spacing for their nets, unless the selected start items are junctions at the end of parallel tracks, in which case the track separation will be kept.

The ends of the dynamic tracks being added can also be welded to the ends of the same number of parallel tracks already in the design.

During Trunk Routing, a model cursor is displayed indicating you are working in this mode. Also displayed is the centre line of path of the trunk routing. This line will always be on grid, and the tracks within the Trunk spaced out from this according to their **Track to Track Spacing** rules or the rule defined.



Whilst adding the track segments you can change the gap between tracks using various functions from the context menu; **Track Spacing** between the tracks; Define Gap Between Tracks, Use **Minimum Gap Between Tracks**, **Pull Tracks Closer Together** and **Push Tracks Further Apart**.



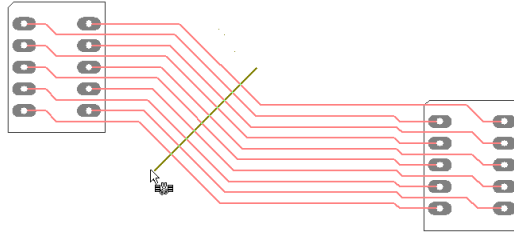
Post-Trunk Routing

After creating the trunk route you cannot edit it as a single item, but the **Cut Track** function (see below) has been added to make it easy to remove and add new parallel segments in it.

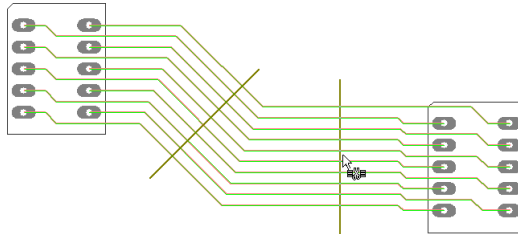
Cut Track (PCB)

There is a new interactive option to cut multiple PCB tracks in two places and remove the track segments between the two cuts. From the **Edit** menu, select **Cut Tracks**.

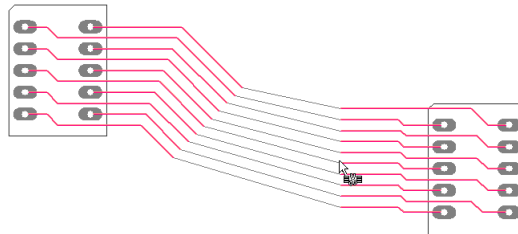
Define the start and end points of the first cut and all tracks crossing it will be highlighted. A model cursor indicates you are in the **Cut Track** mode.



Then position a second cut over the highlighted tracks you want to cut a section from.

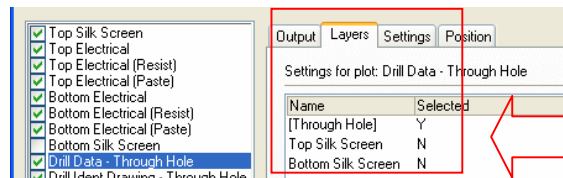


This second cut can be rotated (from context menu or by using a shortcut key), or be redefined by dragging the cursor. The tracks will be split in two by having the section between the two cuts removed.



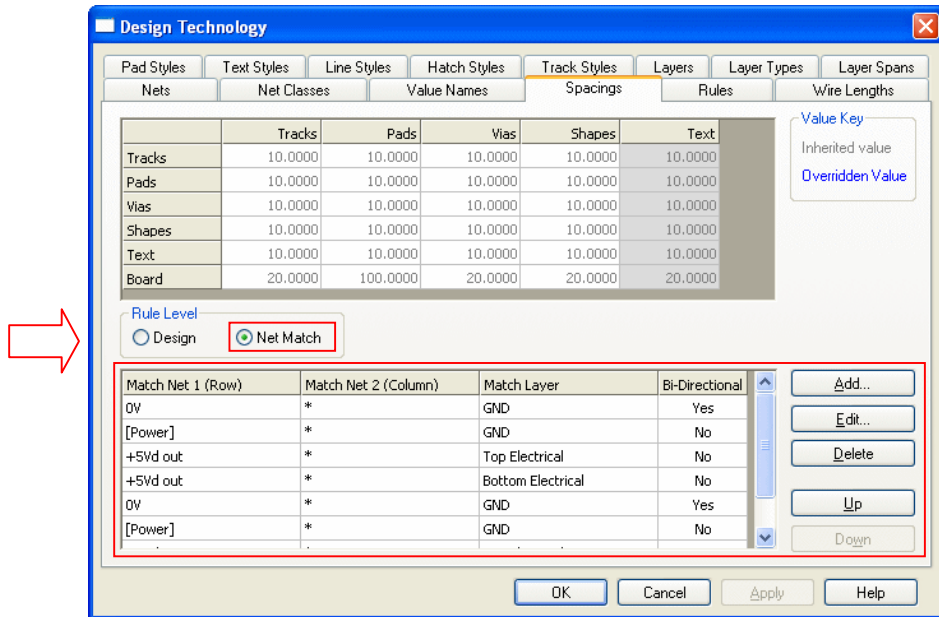
Profiling Shapes to Excellon Output (PCB)

When using the **Plotting & Printing** dialog, you can now select a non-electrical layer(s) when outputting an **Excellon** drill file, this is in addition to [Through Hole] for normal drilling. This means any shapes on the selected layer, will be output as a 'routing' profile. This can be used for board profiling or for outputting slotted pads or shapes.



Net to Net Spacing Rules (PCB)

You can now define spacing rules which match by Net or Net Class name and Layer Name. These are used to produce different spacing rules between items on matching nets.



Press the **Add** button to add a rule. You need to define three wildcard strings which will match two sets of nets (or net classes) and layers. (Note: if they are not wildcard strings (containing '*' or '?'), then they will explicitly match a single net or layer). You can match a set of net class names (instead of net names) by enclosing the Match Net string in /square brackets/.

The **Bi-Directional** column indicates that the rule applies to the design types of either net (so the Pads to Tracks spacing will always be the same as the Tracks to Pads spacing). This does not have to be the case. For example [Power] nets might need a large Pads spacing against Tracks on all other nets (*), but a smaller value is needed for the reciprocal Tracks on [Power] to Pads on the other nets.

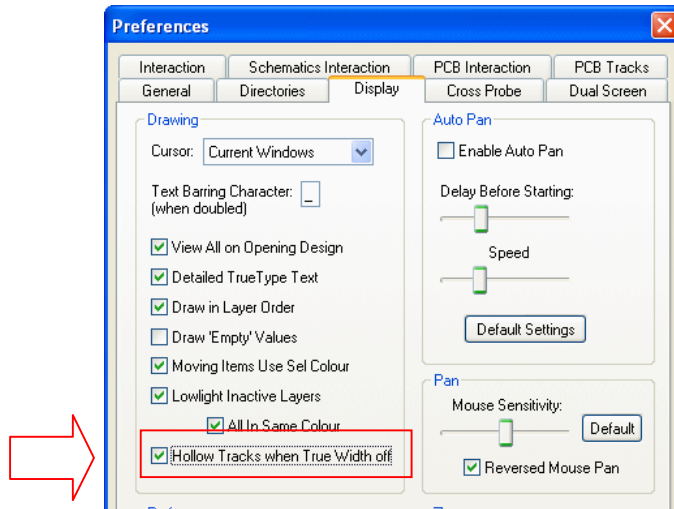
Once a rule has been defined, it is added to the **Net Match** grid. You can move a rule **Up** or **Down** the list. The rules are applied strictly in the order that they are defined in the grid, until the first one which matches. So if you define the first rule as * to * on layer *, which matches everything, then none of the subsequent rules would ever be applied.

If you select a row in the Net Match grid, then the spacing grid above will show the values which apply. Initially, all values are inherited from the design level (indicated by grey text). Typing values into the grid will set an override value for this rule (indicated by blue text). You can also delete an override value to turn it back to the inherited value.

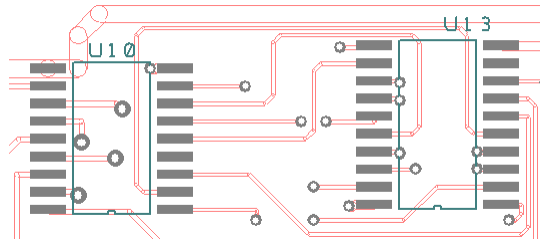
If you have defined the rule as Bi-Directional, then the top right half of the grid is greyed, because those values are the same as in the lower half of the grid. In the case when *Bi-Directional* is disabled and the Match Net criteria would match either way round, then the larger value is used. Note that the net string * also matches with items not on a net. If you use this string, then the Text and Board values become available. So, for example, you can define a specific value between net Gnd and the Board.

Hollow Tracks (PCB)

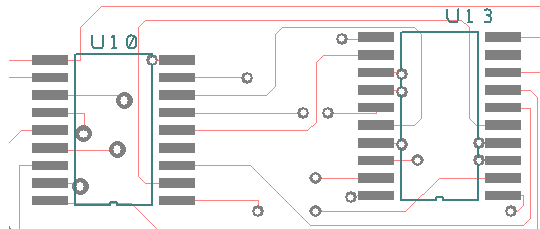
With **True Width Off** in the **Display** dialog, you can now draw PCB tracks as outlined hollow segments instead of just centre-line. To enable this feature, check the **Hollow Tracks when True Width off** box in **Preferences** and **Display**.



Tracks appear as 'hollow' when enabled:

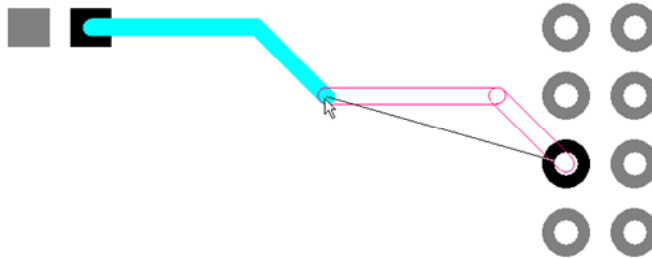


In comparison, without this option selected, tracks appear as centre-line:

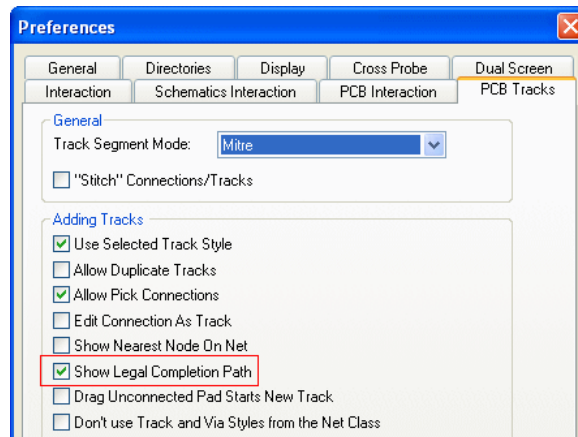


Show Legal Completion Path (PCB)

There is a new option in **Add PCB Track** and **Add Pull Tight Track** to display the legal completion path from the dynamic track end to the target item at the end of the unrouted connection.



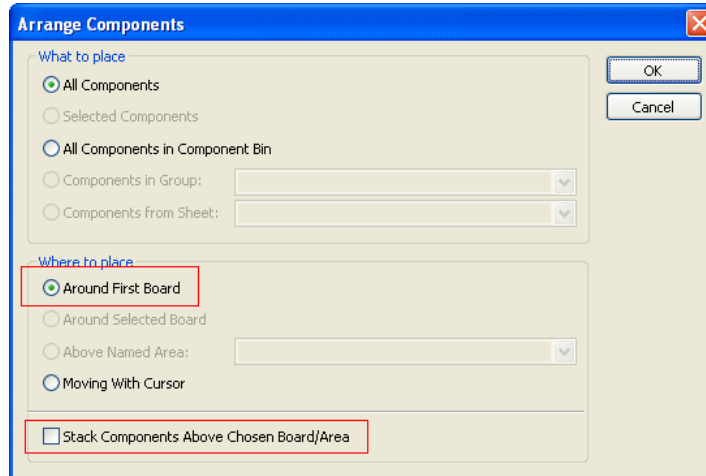
This uses hollow segments in the unfinished track highlight colour. If it is displayed, the **Auto Complete** function will use it. Is also a switch in the **PCB Tracks** page of the **Preferences** dialog.



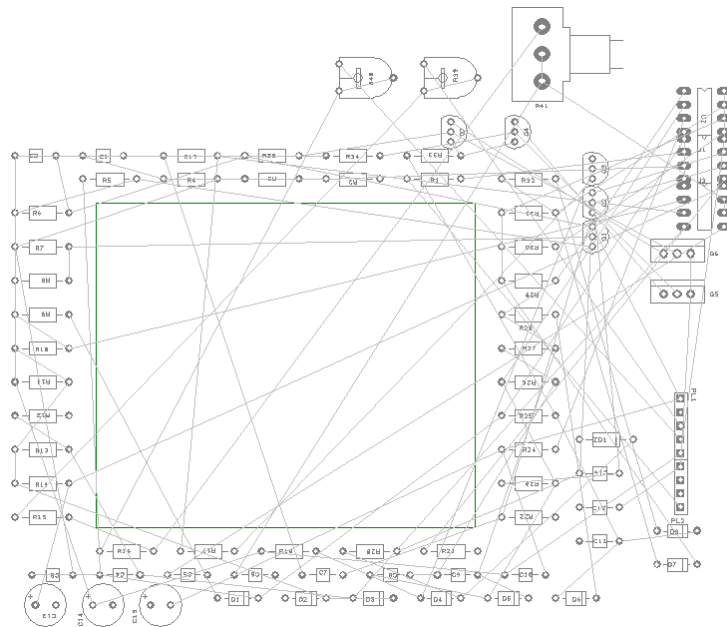
*Note: In **Add PCB Track** the **Auto Complete** option at top of context menu replaces the **Auto Route** option that used to appear half way down the context menu. This change is so that it is more obvious as to what it does, i.e. not to indicate that it is using the autorouter, and is now consistent with **Pull Tight** and **Sketch Route** options.*

Arrange Components (PCB)

On the **Tools** menu, a new check box, **Stack Above Chosen Board/Area** has been added to the **Arrange Components** dialog. The dialog has also been changed so that the **Above First Board** option has been changed to be **Arrange Around First Board**.



Arrange **Around First Board** will enable components from the dialog choice to be placed around the board outline ready for placement. This placement makes final component selection and manual moving easier.



Choosing **Stack Above Chosen Board/Area** will place the components above the first board outline in a stacked placement as it did previously did.

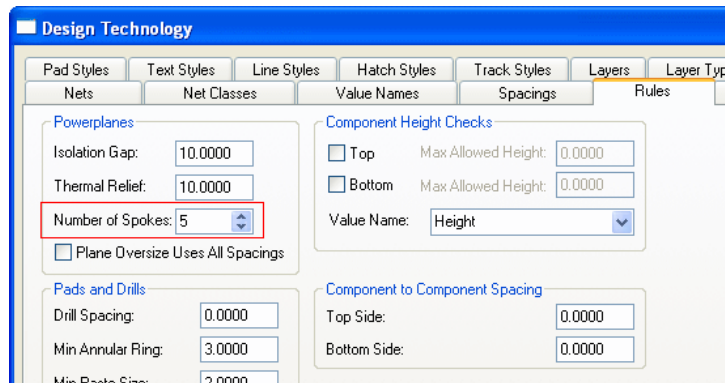
Optimise to Power Planes (PCB)

Optimise will now put a zero length connection to a power plane. All zero length connections are now drawn in a butterfly shape.

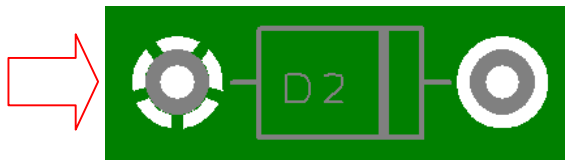


Define Number of Thermal Pad Spokes (PCB)

Thermal spokes on pads and vias can now have any number of spokes. Change the **Number of Spokes:** from the **Design Technology** and **Rules** dialog. Normally, you would set this to 4 or 5 for an optimal number. This will be the maximum number of spokes inserted if the spokes can all be added.

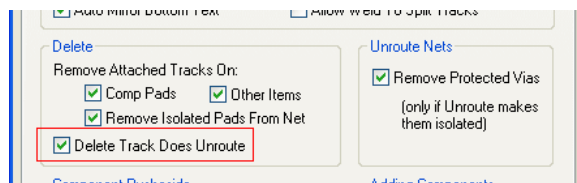


If the number of spokes was defined as 5, the pad would look like this:



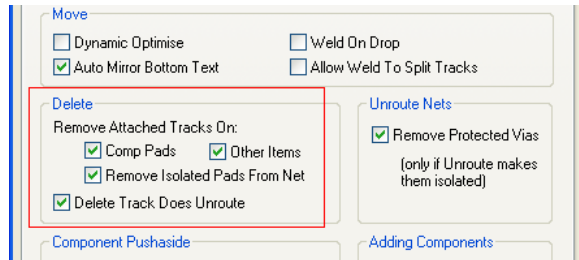
Delete Track does Unroute Preference (PCB)

There is a new check box on **PCB Interaction** page of **Preferences** dialog, **Delete Track Does Unroute**. With this option checked, when you delete an item in a PCB, any selected tracks or track segments are unrouted first (just as if you had used **Unroute Track Segments** on them) before the remaining selection is deleted.



Remove Attached Tracks Options

There are new check boxes on **PCB Interaction** page of **Preferences** dialog, to **Remove Attached Tracks On:** This option is for use when deleting items other than **components**. With this left unchecked, tracks are deleted along with the deleted item.



Lowlight Inactive Layers (PCB)

During editing or move, you can control layers other than the one currently being edited to be dimmed in a lowlight colour. This is controlled by new switch on the **Display** page of the **Preferences** dialog.

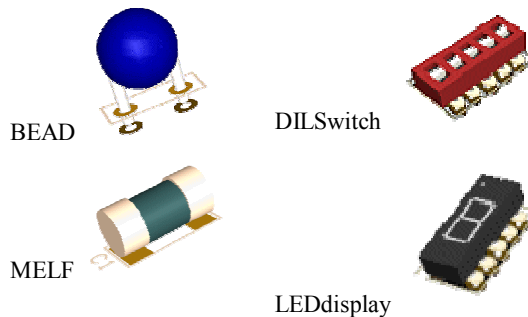


There is a secondary switch **All In Same Colour** that will draw the inactive layers in all the same 'near-background' colour instead of dimming the actual layer or item colour.

3D Viewer Changes (PCB)

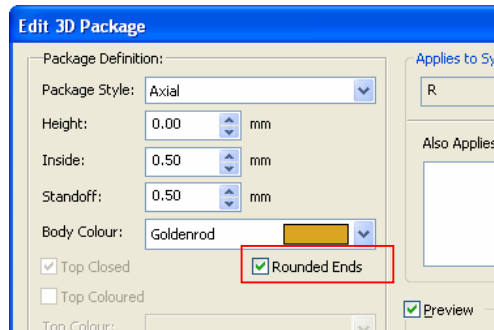
New 3D Package Shapes

New 3D package shapes have been added for BEAD, MELF, DILSwitch and LEDdisplay.



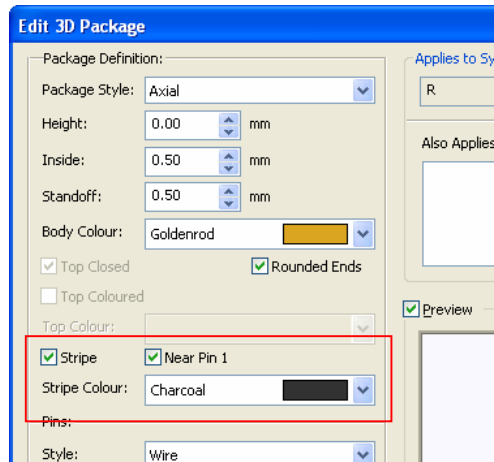
Rounded Ends

For **Axial** components you can choose to round off the ends of the components using the **Rounded Ends** check box.



Colour-coded Bands on Components

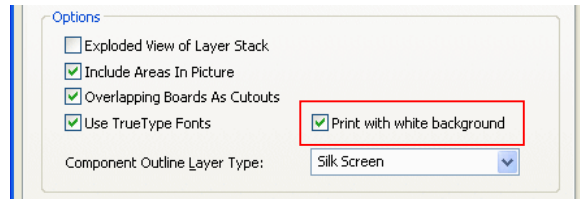
Can and **Axial** components can now be given dynamically colour-code stripes (bands) on the component body. The colour-code bands are based on the value of the component.



CAN shapes will get a stripe up the side of the body near to pin 1 or pin 2 depending on the pin 1 checkbox. **Axial** components get a single stripe near one end or the other (Capacitors, Diodes etc.), except for **Resistors (R)** and **Inductors (L)** which will get the correct set of coloured stripes depending on the value of each component. The **Value** attribute is read and converted to a standard colour coding. Values in formats such as 2.2K or 2K2 are accepted and converted.

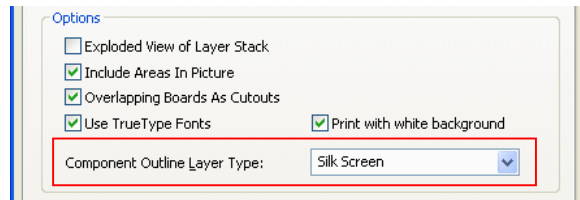
3D View - Save As Image

From the **File** menu and **Save As Image**, you can now write current 3D view out as an image file. **Save As Image** supports Bitmap Files (*.bmp), JPEG Files (*.jpg) and TIFF Files (*.tif). The background of the image will be white if the setting for **Print with white background** is enabled in the **3D Settings** dialog.



Option to choose Layer Type for Component Body

There is a new selection the **Settings** page of **3D Settings** dialog to choose the **Component Outline Layer Type**.



This specifies the name of a **Layer Type** to be used when determining the 'size' of the body of the component. If this Layer Type name is specified, only shapes in the footprint that live on a layer of that type will be considered. The default value is to have no Layer Type name defined, which then allows any shapes to be matched and used for generating the 3D component bodies. To remove a previously specified Layer Type name, simply delete the text from this box.

New Library Contents Searcher on Web Site

Not a new product feature but you can now actively search our web site (www.numberone.com/libraries.asp) for the contents of the Easy-PC optional libraries. Choose between different search criteria or simply choose the current version of a particular library to check the added contents.

Number One Systems
The Electronics CAD Software Specialists

Home Easy-PC Stockit Downloads Sales Technical Admin

Library Finder

Use this form to search for components in our optional Easy-PC add-on Libraries. You can also use this form to see what components were added at each library update.

Find Components

Manufacturer: Choose from the list if you know the manufacturer of the component.

Mfr Part Number: Use this to look for a particular manufacturer part name/number.

Footprint: Use this if you know all or part of a footprint name.

Pincount: Use this if you know something about the pincount of the component.

Library: Use this to search only in one of our add-on libraries.

Version: Use this to find parts added in a particular version of a library.

Latest Library Versions

Listed below are the latest versions of each of the libraries to assist you in searching for components in a particular library version. More information on the current add-on libraries for Easy-PC is available on the [Libraries](#) page.

Connector	2
JPC	1
Micro	4
ProLib	4

Micro Library 4 (Updated Cost Option)

An additional 1,300 new micro controllers have been added to the existing Micro-controller library from Freescale, Silicon Labs plus additional Microchip 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 buy Easy-PC 15.

A list of the updated contents can be located on our web site in the new online library searcher www.numberone.com/libraries.asp. Select **Micro** in the **Library:** drop down list, then type **4** into the **Version:** box. Select the **Search** button to start the search.

Find Components

Manufacturer:

Mfr Part Number:

Footprint:

Pincount:

Library:

Version:

Pro Library 4 (Updated Cost Option)

The Pro-Library has been updated to include over 7,100 additional Parts and their associated Symbols and Footprints. This takes the contents of the Pro-Library to just over 58,000 Parts! Components have been added for National, StMicro, TI, Tyco and Zilog. 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 buy Easy-PC 15.

A list of the updated contents can be located on our web site in the new online library searcher www.numberone.com/libraries.asp Select **ProLib** in the **Library:** drop down list, then type **4** into the **Version:** box. Select the **Search** button to start the search.

Note: for both updated library contents, the exact quantity may change slightly from the numbers above at the time of release.E&OE
