



V12.0 Supplement

Copyright ©1987-2008 by WestDev Ltd. All rights Reserved. E & O E

Number One Systems

Oak Lane

Bredon

Tewkesbury

Glos GL20 7LR

UK

Phone: 01684 773662

Fax: 01684 773664

Email: info@numberone.com

Technical: 01480 382538

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: 18/07/08 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 12.0	5
CHAPTER 2. NEW FEATURES IN EASY-PC V12.0	7
Introduction	7
Library Structure Overhaul (Both).....	7
Mapping of one SCM pin to multiple PCB pins (Both).....	10
Tabbed Dialogs & Improvements (Both)	11
Technology Dialog.....	11
Remembered Styles Dialog	11
Save Units used to create a symbol into the library (Both).....	11
Separate technology file settings for designs and symbols (Both).....	12
Slotted Pads (PCB)	12
Cutouts (Both).....	13
Hatched/Cross Hatched Shapes (Both).....	14
Active Status bar (Both).....	16
Active PDF Writer (Both)	16
Changes to the Plotting & Printing dialog (Both)	17
Shapes Improvements (Both)	18
New Shapes	18
Shape Editing Improvements (Both)	19
Item Positioning Improvements (Both)	20
Coordinate Origin Improvements (Both).....	21
Replicate (Both)	22
Rearrange Multiple Items (Both)	23
Preserve Attached Segments (Both).....	24
Place Horizontal and Vertical Segments (Both).....	25
Copper Pour Keepout Areas (PCB)	26
Pour Thermal Flags (PCB).....	27
Find (Both).....	27
Component Suppression Flags (Both).....	28
Measure Improvements (PCB)	28
Add SMD Pad in PCB Symbol Editor (PCB).....	29
Nets Dialog sort order (Both)	29
DRC check min annular ring (PCB).....	29
Layer Types Improvement (PCB).....	30
Layers Bar Categories (PCB)	30
Milling of board outlines (PCB)	31
Values used for Hyperlinks or Hyperlink Values (Both).....	31
Open Multiple Files at once (Both).....	32
Insert Layer Stack Preview (PCB).....	32
PCB Interaction Preferences (PCB).....	34
Auto Mitre Any Angle (PCB).....	34
Library Contents Improvements (Both).....	35

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 V11.0 can install V12.0 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 12.0** 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 completion.

Click **Finish** to complete the installation.

Running Easy-PC For Windows 12.0

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 V12.0

Introduction

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

Library Structure Overhaul (Both)

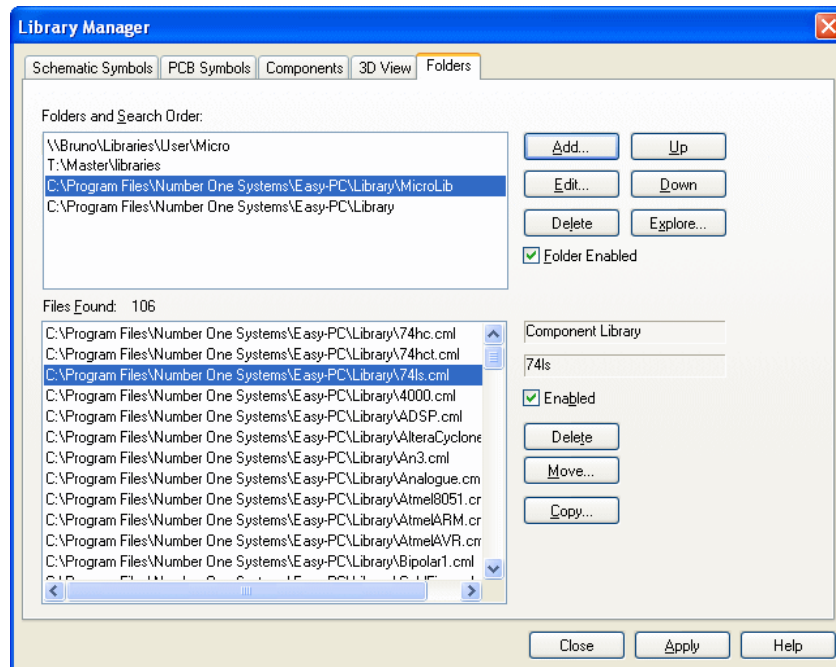
The method in which libraries are stored, organised and managed has been significantly changed.

Library symbols are no longer tied to a specific library and the library mechanism can use multiple lookup paths from which to take library items. This means network folders as well as multiple local folders can now be utilised.

Library Manager

The **Library Manager** has been altered in two areas:

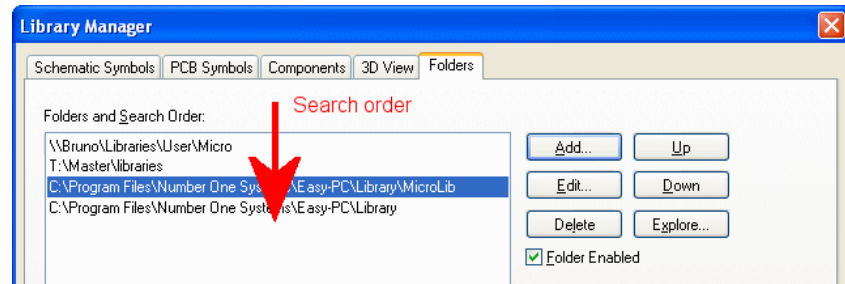
A **Folders** tab has been added to the dialog. This is used to define the folders in which libraries exist and the order in which they are searched.



The dialog is split into two logical areas of functionality.

The **Folders and Search Order** define where the libraries exist. These can be local and network paths. Multiple search paths can be defined. You are no longer restricted to one single library path.

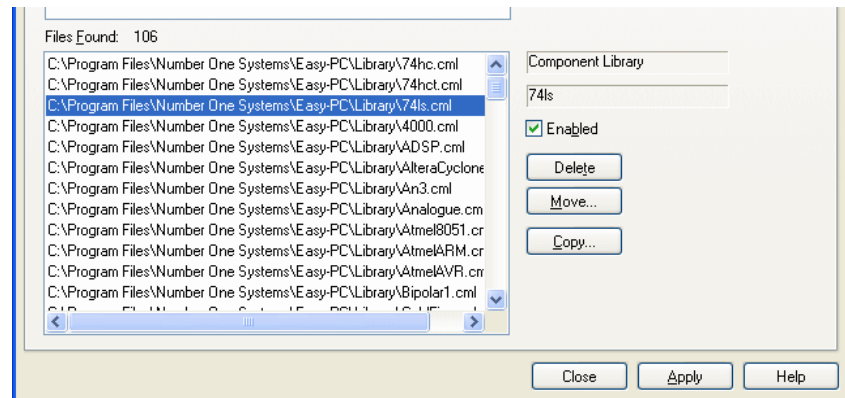
IMPORTANT The most important aspect of this area of the dialog is that the order in which the folder paths appear going down the list, is the order in which library items will be found. This is significant when adding Components to the design as this is the order in which the libraries will be searched.



Buttons on the right side of this dialog allow the creation and editing of search paths and navigation to be performed. Using the **Up** and **Down** arrows on selected folder paths allows you to change the search order. Paths at the top of this list will be searched first. Where duplicate library items exist, the one found first in the search list will be used. Libraries within a folder are alpha-numerically sorted.

The **Delete** button only deletes the search path, it does not delete the library folder.

The second area of the **Folders** dialog is the **Files Found** area. From the folder search list, each of the libraries found in the paths defined are displayed here. Again, items found within these libraries are used in the library manager or in the Add component dialog in the order they appear.

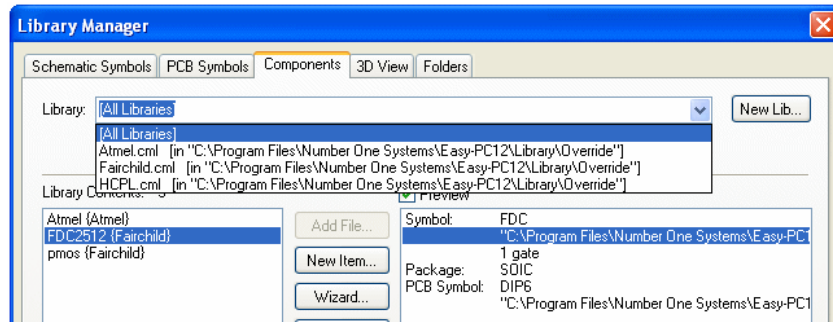


The **Delete** button here will be used to actually delete the library, so care should be taken using this.

The **Library Manager Folders** dialog now provides a single point of control for all library paths within Easy-PC, **Translate to PCB** for example. This dialog is the only location you can change them. The previous **Library Directory** path in **Settings, Preferences** has been removed and is replaced using the **Folders** dialog.

The Library Manager dialog

In addition to the new Folders tab, each of the existing dialogs has been modified to cater for the expanded functionality.



The separate Library name path controls have been replaced with a single **Library**: drop down list. This list is based on the libraries found in the Folders search path. You can choose individual libraries to select items from or you can now choose [All Libraries].

The **Library Contents**: list now displays the library item found and the library name from which it was read only if <All Libraries> is selected.

Checking for Duplicate Library Items

A report is included in Easy-PC V12 for checking your existing libraries for duplicate items. There is no automatic method for correcting these, you must manually locate the duplicates and make the changes necessary.

The **Report** button from each of the library manager dialogs (Schematic Symbols, PCB Symbols and Components), as well as listing the library contents (as it did previously), will also now list any duplicate items it finds when reporting all libraries.

The master library contents supplied by Number One Systems have been checked and corrected for duplicates, none will now exist. If you elect not to install the supplied libraries, it is recommended that you run the Report to check for duplicates in your own libraries.

Important: When checking for duplicate library items, be careful which item you delete. Be aware that schematic symbol pin number sequences will have an effect on the Component pin numbering and may not be consistent. Check Components carefully after deleting duplicates.

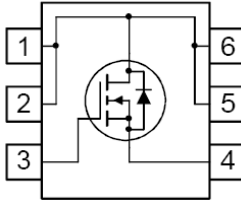
LibNames.txt

If you have been using the libnames.txt file for library ordering, this feature is now redundant. However, changes within new Library Manager dialog mean that you can now use the library search order option to force particular library folders and libraries to be located and used first. By moving search folders further up the list (in the Library Manager, Folders), these will take precedence. Files within these folders are search alphanumerically, renaming library files means they will be more or less significant in that folder.

Mapping of one SCM pin to multiple PCB pins (Both)

Using the Component editor within the Library Manager, you can now map one Schematic symbol pin to multiple PCB footprint pins.

This is best demonstrated with a real example. The example used here is the Fairchild FDC2512 part. The datasheet looks like this:



The internal mapping of the PCB footprint means that pins 1, 2, 5 and 6 are common to the one schematic symbol pin. The schematic symbol contains 3 pins overall.

In the Easy-PC Library Manager the mapping is done using the **PCB Symbol Pad Number** column. Use a comma character to indicate multiple PCB pin numbers. In the example below, the Schematic symbol pin, pin 1, is mapped to PCB footprint pins 1,2,5 and 6. When this option is used, you must have a letter in the **Component Pin Name/Number** column for these corresponding pins.

Gate	Sch Symbol	Sch Symbol	Sch Terminal	Pcb Symbol	Component Pin	Net (Class)
Name	Name	Terminal Name	Number	Pad Number	Name/Number	Name
a	hcpl		1	1,2,5,6	1,2,5,6	
			2	3	3	
			3	4	4	

The graphical view shows the schematic symbol on the left with terminals X3 and X4, and the PCB footprint on the right with pins 1, 2, 3, 4, 5, and 6 highlighted in green. The component is labeled U1.

Pin Mapping and Connections

There are two formats which can be used for denoting multiple mapped pins; you can use commas and the plus (+) sign.

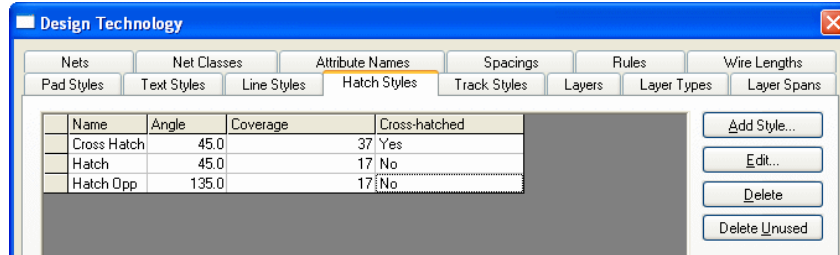
If a comma is used, the PCB pins will be mapped externally and Optimise will show connections to each of these pins.

If a + sign is used, the connections are made internally and only the first pin in the list will be connected to.

Tabbed Dialogs & Improvements (Both)

Technology Dialog

Technology style dialogs on the **Settings** menu have been consolidated to one easy-to-use Technology dialog.



The dialogs available in V11 and previous Easy-PC versions which have been effected are: **Nets**, **Net Classes**, **Styles**, **Spacings**, **Values**, **Wires**, **Layer Spans**, **Layers** and **Layer Types**.

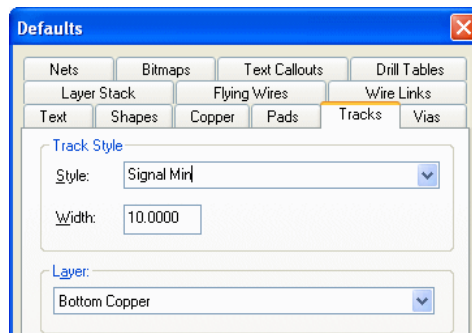
A new page exists for **Hatch Styles** and **Value Names**.

Remembered Styles Dialog

The Remembered Styles dialog is now called Defaults and contains a tabbed dialog for all the existing Remembered Styles.

Existing Remembered Styles dialog options now moved are: pads, text, shapes, copper, vias, nets, dimensions, text callouts, flying wires, wire links, drill tables, layer stacks.

In the PCB editor, there is also a new tab for remembered **Tracks**.



Save Units used to create a symbol into the library (Both)

When you use the **Save to Library** feature to save Symbols and footprints to the library, V12 now saves additional information. It now also saves the **Grid**, the **Units** and the **Origins (Relative and System)**, so they may be reused if the symbol is re-edited.

Separate technology file settings for designs and symbols (Both)

Default technology files for designs and symbols are now separate, this allows design technologies to include items (board, shapes, etc) and have colour settings (e.g. pin names) which need to be different to symbols.

Slotted Pads (PCB)

The ability to have slotted pad styles has been added to Easy-PC V12.

From the **Design Technology** dialog and **Pad Styles**, you can create the slotted pad the same as you would any other pad style.

	Name	Layer	Shape	Width	Length	Corner	Hole Size	Plated
X	Large Via	[All]	Round	56.00	0.00	0.00	32.00	Yes
	Pad	[All]	Round	60.00	0.00	0.00	35.00	Yes
X	Rec 56	[All]	Rectangle	56.00	100.00	0.00	0.00	Yes
X	Slotted	[All]	Rectangle	60.00	110.00	0.00	Slot	Yes
	Small Via	[All]	Round	32.00	0.00	0.00	16.00	Yes
X	Via	[All]	Round	48.00	0.00	0.00	24.00	Yes

When creating or editing pad styles, you can now select a **Hole Shape**.

Pad Style

Name: Slotted

Shape: Rectangle

Width: 60.00 Length: 110.00

Used: Corner: 0.00

Hole:

Shape: Rounded Rectangle

Width: 25.00 Length: 60.00

Plated Corner: 5.00

Rotated

OK Cancel

The shapes available to you are listed in the drop down list box.

Hole:

Shape: Rounded Rectangle

- Chamfered Rectangle
- Oval
- Rectangle
- Round
- Rounded Rectangle

Rotated

When using shapes with rounded corners, you are also able to enter the **Corner**: radius of the shape.



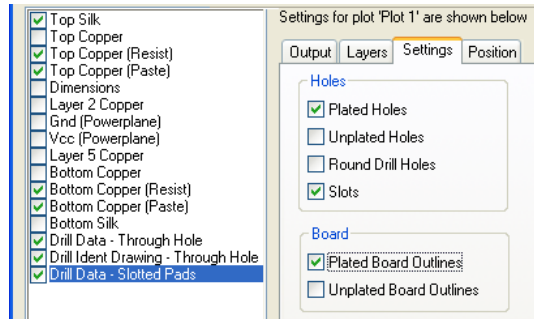
Slotted pads styles cannot be local, they must always be global and defined in the Technology dialog.

Existing Designs

Pad Styles in existing designs can be modified to include the new slotted pad style if required. You must create new plot types to process these new styles for manufacturing.

Plotting & Printing O/P (Excellon)

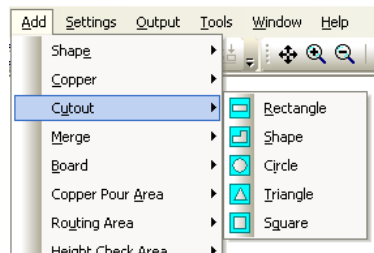
The **Plotting & Printing** dialog has been modified to enable slotted pads to be output. When an Excellon format file is requested, the Settings tab now has check boxes for **Holes**. You can check the Plated status required for output as well as **Round Holes** or **Slotted** holes.



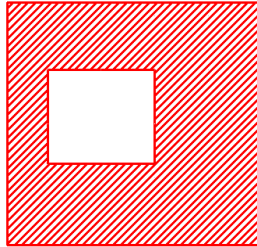
The ODB++ output has now been changed to accommodate slotted pads. Plated Slots are output to the special layer 'plated slots', along with any plated board outlines. Unplated slots will be added to the 'boarddata' layer.

Cutouts (Both)

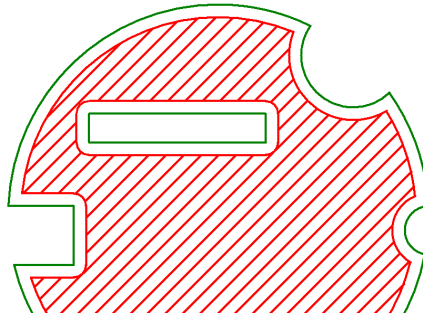
Shape can now contain true cutouts. Cutouts can be added using the new **Add Cutouts** options from the **Add** menu.



To use this option, select the shape required from the **Add Cutouts** menu. Then select the shape in which to add the cutout to. Draw in the cutout and release. The cutout will be drawn in the shape.



Cutouts can intersect with the shape edge or other existing cutouts. The effect of this is to remove the cutout from the shape edge. The cutout may even break the shape into several pieces.

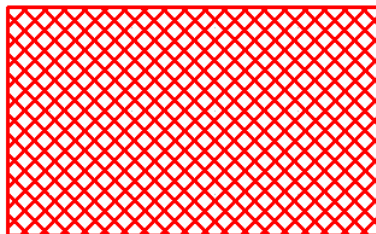


Add Merge

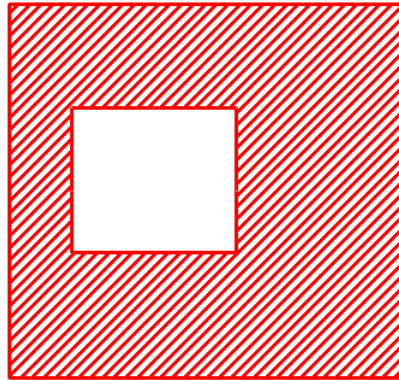
You can add a shape to an existing shape. The resulting outline is the combination of the two shapes.

Hatched/Cross Hatched Shapes (Both)

Shapes can now be **Hatched**. The hatch style can be **Hatch** or **Cross Hatch**. Cross Hatching is a double crossed line, whereas Hatching is not crossed. The example below shows a **Cross Hatch** style.

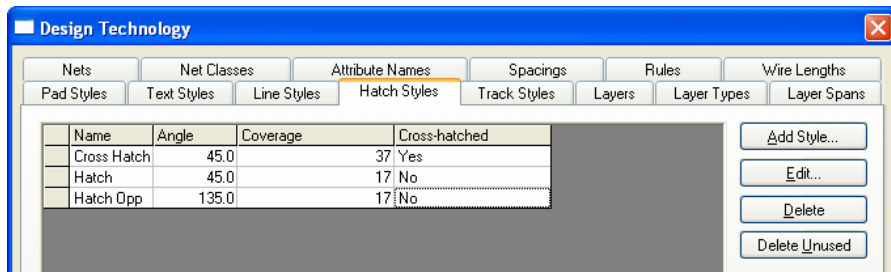


The example below shows a **Hatch** style.



Hatch Styles

Hatching is defined on the **Settings** menu and **Design Technology** dialog under the **Hatch Styles** tab.



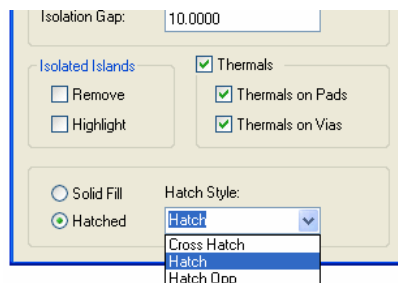
Angle is the angle of the hatch shape line or lines. For Cross Hatch, the second line is at 90 degrees to the first line.

Coverage specifies how much copper as a percentage of the shape area will be covered with shape. For example, 50% would be half shape coverage and half empty.

Cross-hatched allows you to specify hatching or Cross Hatching.

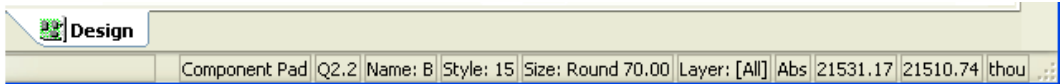
Copper Pour

You can specify the hatch style on the copper pour dialog. Check the **Hatched** radio button to select this option.



Active Status bar (Both)

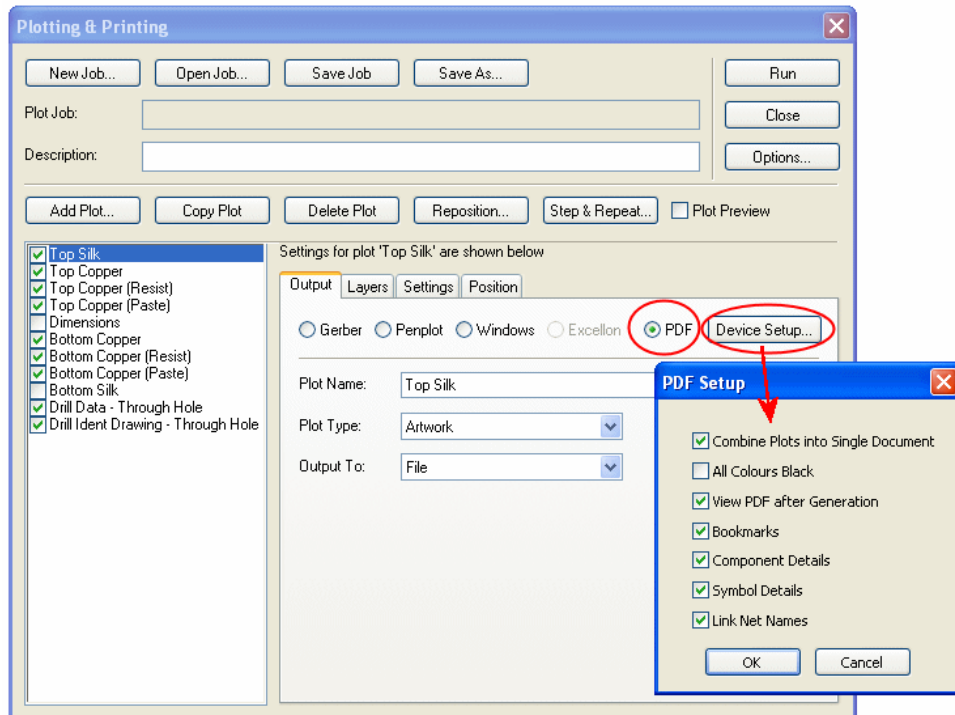
A subtle change has been made to the status bar at the bottom of the Easy-PC window. By double clicking on the words displayed, **Abs** for example, you are able to change the appropriate setting. You can change **Styles**, **Layers**, **Units**, **Coordinates** and switch between **Abs/Rel** modes.



Active PDF Writer (Both)

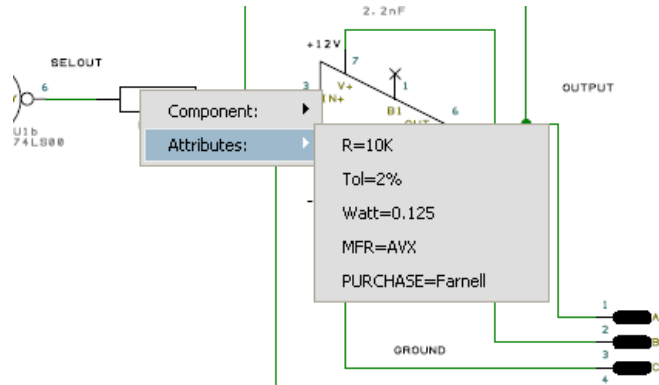
Available in both the **Schematic** and **PCB** design editors, you can now opt to output your plots to **PDF** format. For this, Easy-PC has a built-in PDF writer. This can be selected from the **Plotting & Printing** dialog on the **Output** page.

If installed, you can still use your own Windows PDF driver but using the mechanism means the PDF file is also active. You can see design information displayed within it once plotted.



The output format of the PDF file can be tailored to suit your requirements. By selecting the **Device Setup** button, different parameters can be changed.

When plotted, the output is now active and looks like this:



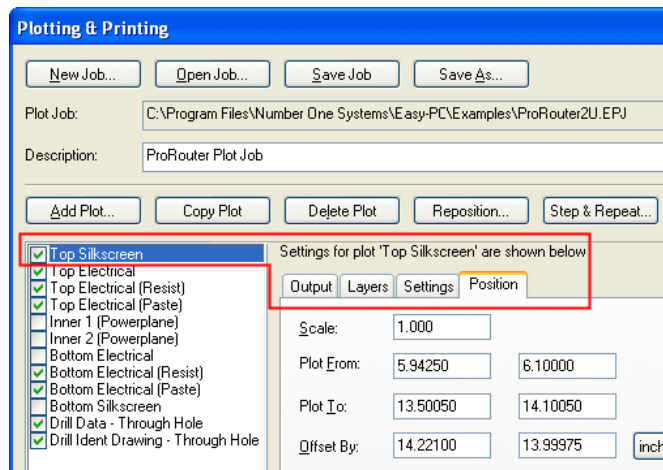
By hovering the mouse on a component (or net), a small dialog is displayed. From this you can select **Component**> or **Attributes**> and view the values for each. Attributes are only displayed if values have been assigned to component in the design.

Changes to the Plotting & Printing dialog (Both)

The Plotting & Printing dialog has been improved to facilitate your plotting.

Dialog Declutter

A small change has been made to rearrange the main dialog to declutter it and ensure you know that the four tabs are not global for 'all' plots but specific to each.



Pre-Plot Checks

The pre-plot checks that are done when you click **Run** will now make sure that at least one plot has board outline enabled. This check – and the warning message you get if no board is selected is turned on/off by the **Warn if no plots include the board outline** checkbox on the **Options** dialog.

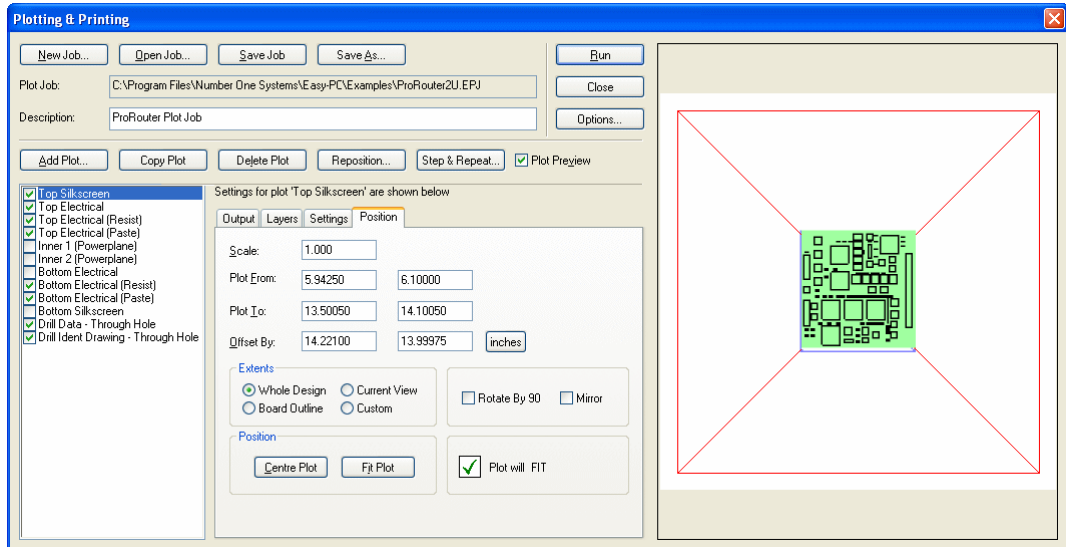
Plots which reference layers that are empty (and hence would produce nothing if plotted) are automatically unchecked.

The check box to **Use all Spacings** on the **Plotting & Printing** dialog under **Options** is now in **Design Technology** dialog and on **Rules** dialog.

Preview of plot

By selecting the **Plot Preview** checkbox on the plotting dialog, you can now display a preview of the plot, plotter 'bed' and design extents.

In the Preview window, the bigger red area shows the plotter bed size, this would be the paper size if a Windows printer or the size defined for the plot bed if Gerber. The smaller inner blue area is the design extents. This would include drawing borders or items outside the board outline if defined. The green area is the board outline. Black items within this are details of the plot that will get printed. This is not an exhaustive display but is designed to give you an idea of the plot output, not every minute detail of that plot.



When using the **Position** dialog, changing values and using the **Position** buttons will have an instant impact on the preview.

Shapes Improvements (Both)

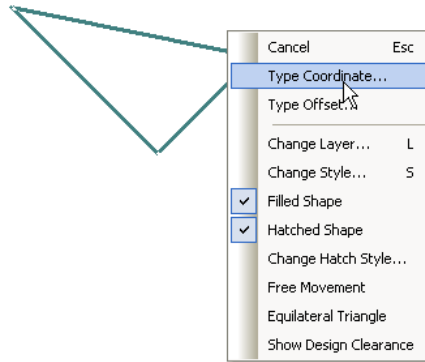
New Shapes

Three new shapes have been added to the **Add** menu for Shapes and Copper: **Square**, **Triangle** and **Single Line**.

To accommodate the line function the old Polygon and line options are renamed to Closed and Open Shapes. Square has been added to all other shape types, and Square & Triangle have been added to **Add Cutout** and **Add Merged** options.

When adding Square, Rectangle or Circle shapes, from the context menu you can now select **Define From Centre**. Instead of the shape being added from the edge, it will now be added from the centre of the shape.

When adding a Triangle shape, there is a new option available on the context menu - **Equilateral Triangle**. This is available in the last phase of adding a Triangle, i.e. once you have defined one side, and it forces the triangle to be an equilateral shape.

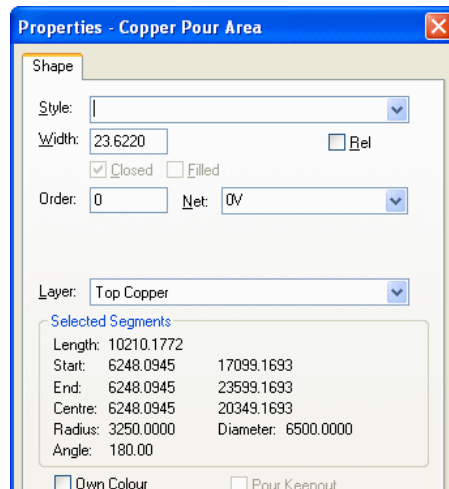


Shape Editing Improvements (Both)

A range of shape editing improvements have been made to version 12 to aid your design process.

Show Diameter in Properties for arcs

The Properties dialog now shows the **Diameter** for arcs or circles.



Type Radius/Diameter

When editing **Mitres**, **Type Radius** has been added to the context menu, and, if fillet, **Edit Diameter** has been added. When the edit has been completed an online DRC check has been added.

When editing **Arcs**, **Type Radius**, **Type Diameter** and **Type Angle** have been added to the context menu. When the edit has been completed an online DRC check has been added.

When editing **Circles**, **Type Radius** and **Type Diameter** have been added to the context menu.

When in **Select Mode**, you can now select an arc segment and from the arc submenu, you can use **Type Radius**, **Type Diameter** and **Type Angle**.

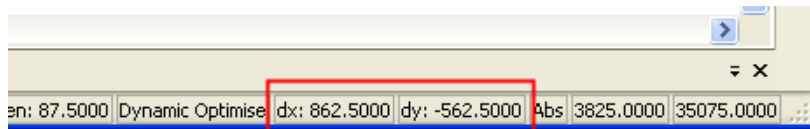
Add Corner and Delete Corner

The **Add Corner** and **Delete Corner** options have been added for use with shapes. These options are available on the context menu for selected line segments.

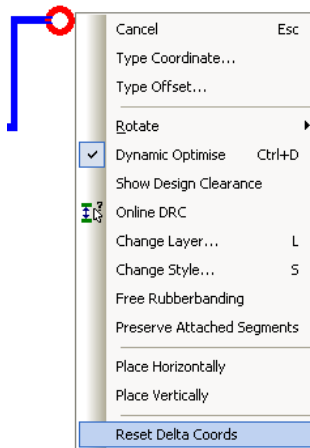
Item Positioning Improvements (Both)

Show Delta Coords on Status Bar

When moving an item in the design, the X,Y coordinates of where the item came from and is currently are shown on the status bar. This is displayed as the Delta coordinates dx: and dy:

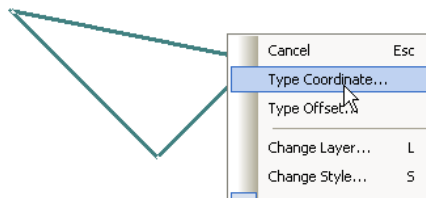


During move, you can also use the **Reset Delta Coords** option from the context menu to reset these coordinates.



Type Offset and Type Coordinates

When moving items or for selected items, you now have the general **Type Coordinate** and **Type Offset** commands. Additionally, in **Add PCB Connection** and **Swap Connections**, you have the **Type Pin Name** commands.



Typing Shape Coordinates

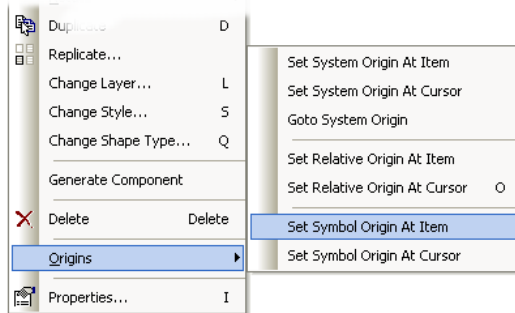
When using the **Type Coordinates** dialog, you can type a whole shape in by typing a space after each coordinate instead of pressing OK. The shape segment is added and you are returned to the Type Coordinate dialog to continue. To finish the shape, use OK or use '=' after the last coordinate. In this mode the first two coordinates are absolute start X and Y, and the rest are offsets.

When using the **Type Coordinates** dialog, you can also type in **U1.6** (for example) as the 'target' component/pad in the X coordinate box to move to the position of that pad, or start a track on that pad.

Coordinate Origin Improvements (Both)

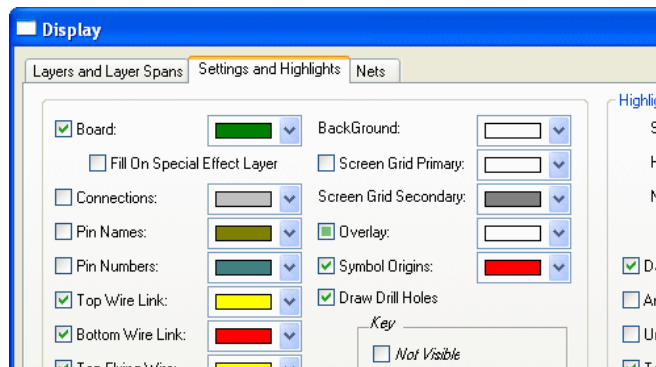
Set Symbol Origin

The **Set Symbol Origin** command has been added to the symbol editors. This is available on the context menu along with the existing Set Origin commands.



View Origins

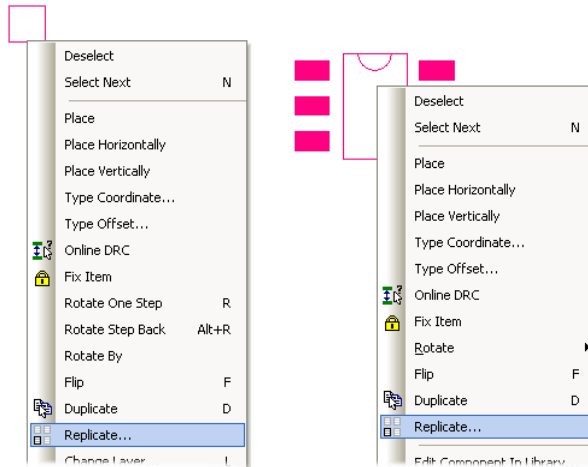
You can now view the **Symbol** origins in the Schematic or PCB designs and change their colour. Colours are changed in the **Display** option on the **Settings and Highlight** dialog under **Symbol Origins**.



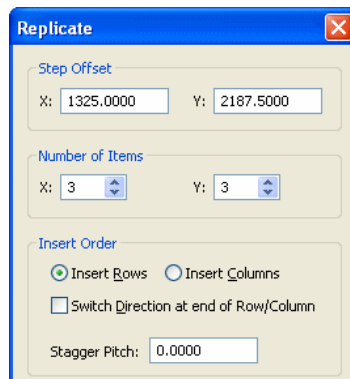
Replicate (Both)

The **Replicate** option has been added to Easy-PC V12. This is the ability to duplicate an item and then specify the number of copies and their position.

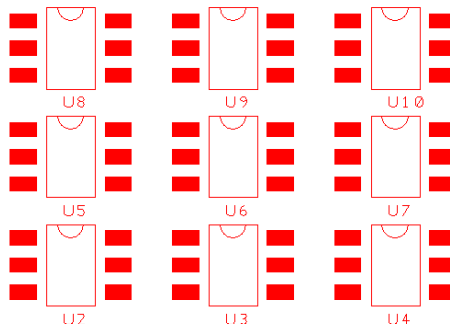
From a selected item, a component or shape, from the context menu, select **Replicate**.



The Replicate dialog is displayed.



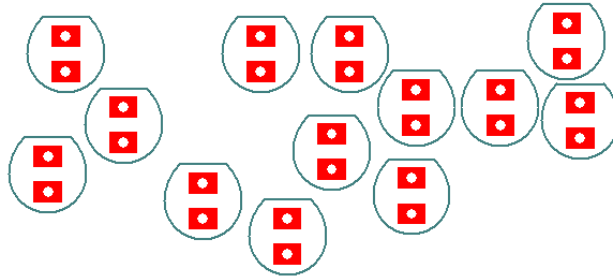
From this dialog set your required parameters and click **OK**.



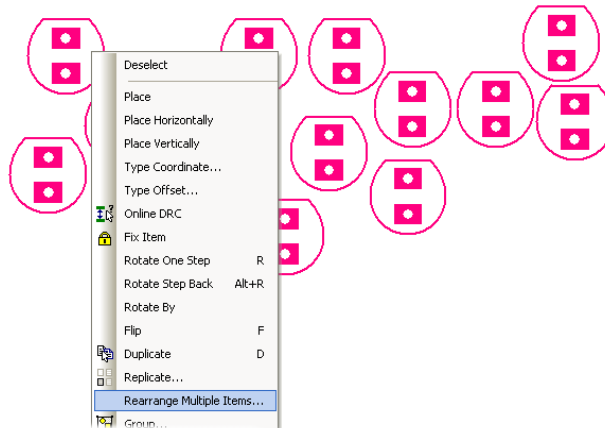
Rearrange Multiple Items (Both)

A new option similar to Replicate has been added to V12. The **Rearrange Multiple Items** option allows you to select an existing collection of like items and place them into an array. This can be used on shapes, components, pads or Vias.

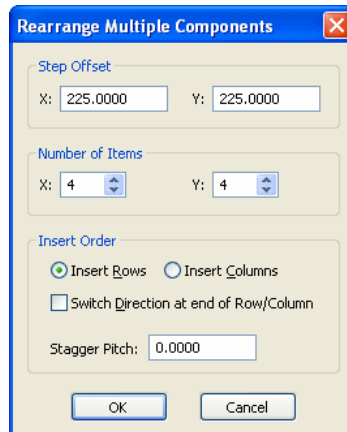
A collection of similar items can be selected:



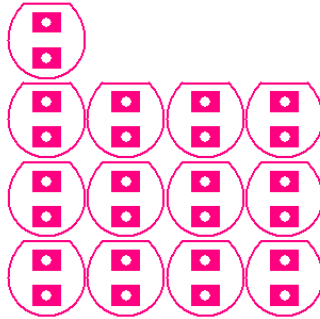
From the context menu select **Rearrange Multiple Items**



Change the parameters required:



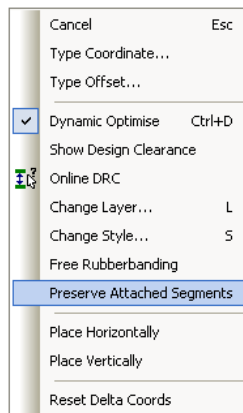
The result will match the parameters. Where there are not enough items to satisfy the parameters, the option will complete it as best it can. When there are more items than the Number Of Items specified in the X, Y directions, in our example below if we chose 2 by 2 say, the remaining items would be placed in the X direction.



When placing components, they do not have to be same symbol, and pads do not have to be the same style.

Preserve Attached Segments (Both)

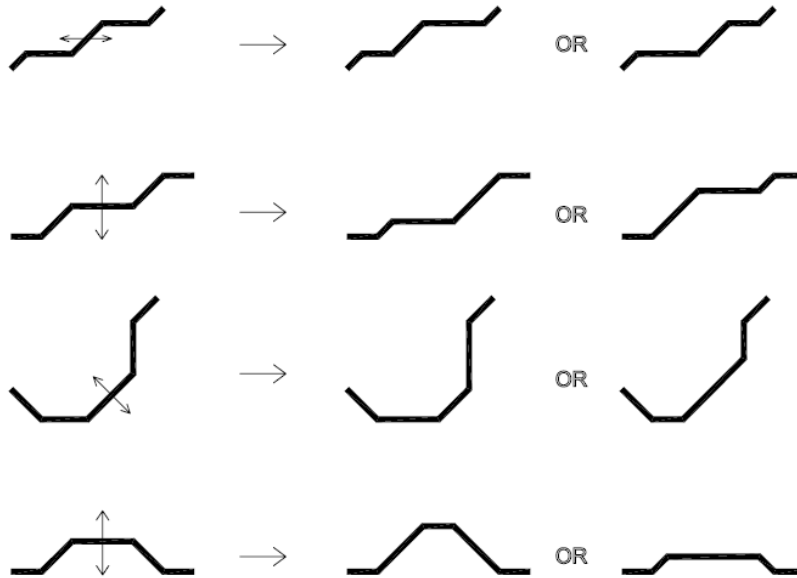
During move (Place), the new **Preserve Attached Segments** option is available to describe how unselected segments attached to moving selected segments are altered.



If you leave this option unchecked, it will work as before, i.e. the attached segments are matched to a segment mode and altered thus.

Select it to preserve the angle and far end point of the unselected attached segments. This is usually achieved by changing the length of the selected segment.

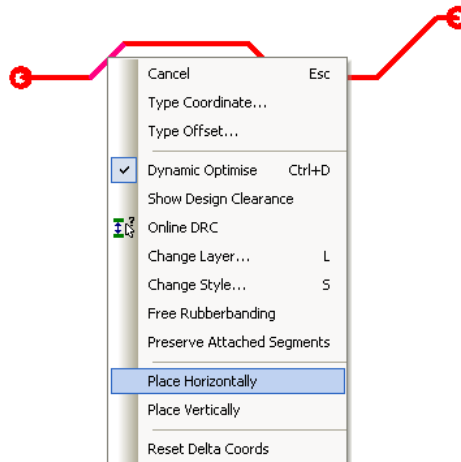
This is demonstrated during track editing.



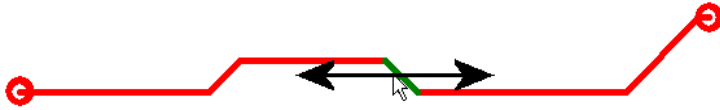
Place Horizontal and Vertical Segments (Both)

During **Place**, two new commands **Place Horizontally** and **Place Vertically** are available on the context menu. Selecting one of these restricts the item movement to that direction only.

These options can be added to a shortcut key or can be used from the **Place** context menu. The commands can also be used directly on a selected item (so that you don't get the jump to grid when it is first moved).



From the context menu, select **Place Horizontally** or **Place Vertically**.



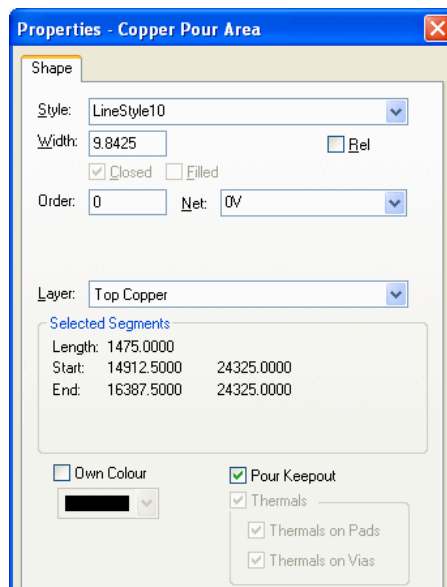
Another small change is the ability to use the three Place commands (**Place Horizontally**, **Place Vertically**, **Preserve Attached Segments**) with shortcut keys in select mode when nothing is selected. In this case it will pick select the item under the cursor. This enables us to use **Place Horizontally** etc. with a corner without getting a small move first.

Copper Pour Keepout Areas (PCB)

The ability to make **Copper Pour** areas **Keepout** areas have been added to PCB design. When you add a **Copper Pour Area**, you can now select the **Pour Keepout** check box on the **Copper Pour Area Properties** dialog.

Copper Pour Keepout areas would normally be added inside a Copper Pour area to create a void in the copper.

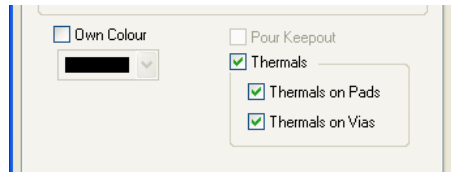
If the Net name is assigned to the Copper Pour Keepout Area matches the net being poured, then it is avoided.



Pour Thermal Flags (PCB)

On the **Copper Pour Areas**, there are now two **Thermal** check boxes, **Thermal Pads** and **Thermal Vias**. This allows thermals to be specified separately for pads and vias. Previously you could only include/exclude thermals on vias, they would always be done on pads.

Thermal flags are also now stored and retained on each Area, rather than being set at the time of Pouring.

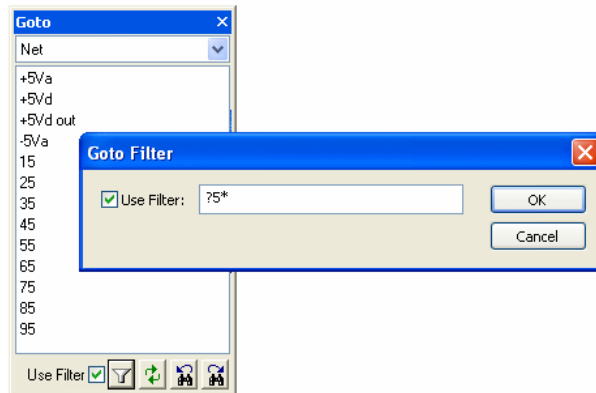


Find (Both)

Find Filter

You can filter found items in the list by using the filter on the **Goto** dialog.

Check the **Use Filter** box to activate the filter. A dialog will be displayed into which you can type the name you wish to find. You can use standard wildcards (* and ?) in this field.

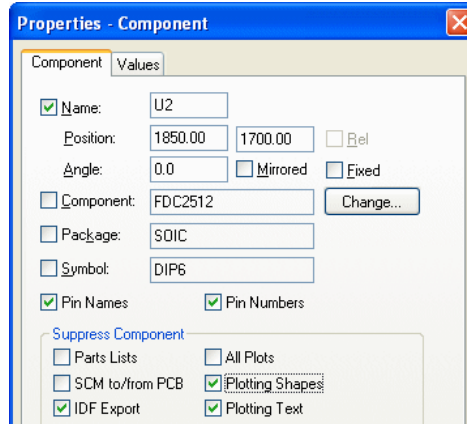


Find Pad [By Terminal Name]

The Goto list has also been expanded to include the item **Pad [By Terminal Name]**.

Component Suppression Flags (Both)

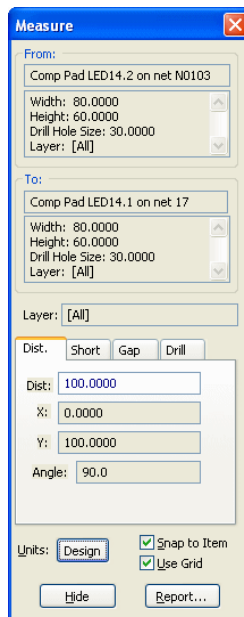
On the **Component Properties**, there are several new check boxes that allow you to suppress the component (or portions thereof) from various aspects of the program (plotting, parts lists, SCM to/from PCB).



You also now have the ability to save and load suppression settings using simple text files. These are described in detail in the online help under the **Index** heading **Suppressing Components**.

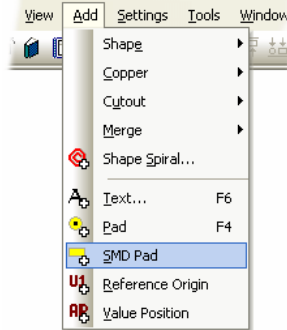
Measure Improvements (PCB)

The **Measure Tool** has been enhanced to more comprehensive allowing you to measure the distance between any two items, and the distance between two selected points. It now gives you more size information about the selected items.

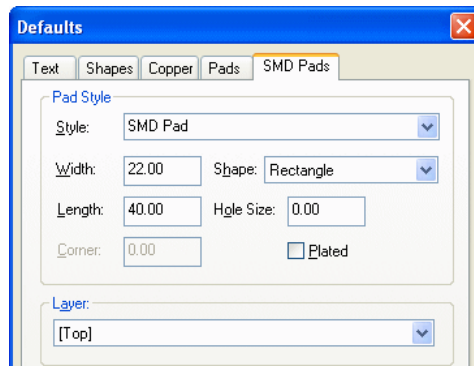


Add SMD Pad in PCB Symbol Editor (PCB)

When in the **PCB Symbol editor**, you can now select **Add SMD Pad** from the menu or **Add** toolbar. This is used to add a pad which lives on the Top side of the PCB and has no drill hole.



The defaults used for when adding this pad are taken from the new **Defaults** dialog on the **SMD Pads** tab.



Nets Dialog sort order (Both)

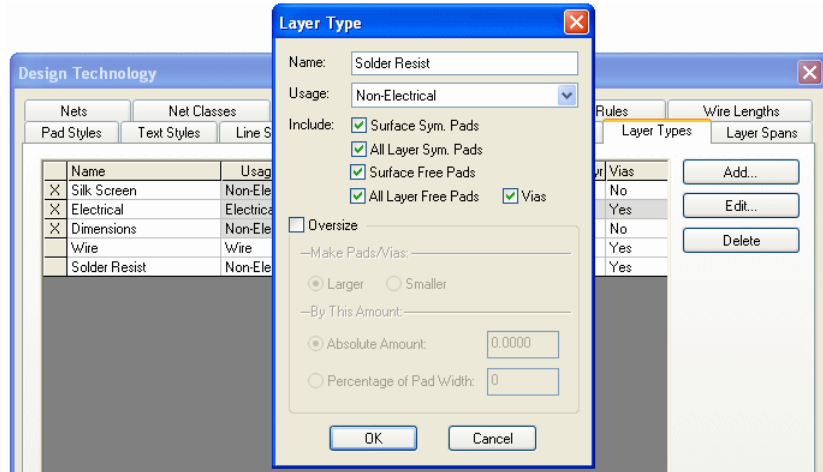
On the **Nets** page of the **Design Technology** dialog, this now sorts default (unnamed/auto-named) nets to the end of the list, the user-named nets appear first in the list. It also has a check box to show/hide auto-named (default) nets.

DRC check min annular ring (PCB)

The existing **DRC** check under **Manufacturing** for **Min Annular Ring** no longer checks min annular ring for unplated holes. This allows you to create bare holes without getting DRC error.

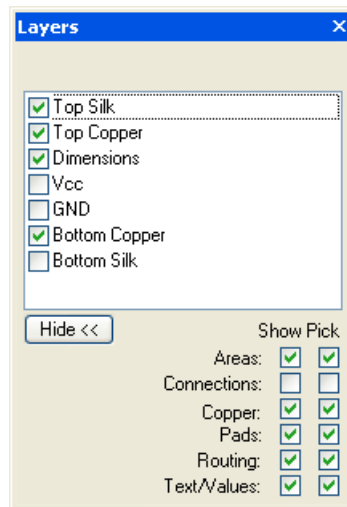
Layer Types Improvement (PCB)

On the new **Design Technology**, **Layer Types** has been changed with the inclusion of a new check box for **Surface Mount Free Pads**. The old **Pads** check box has been replaced with the **All Layer Free Pads** check box.



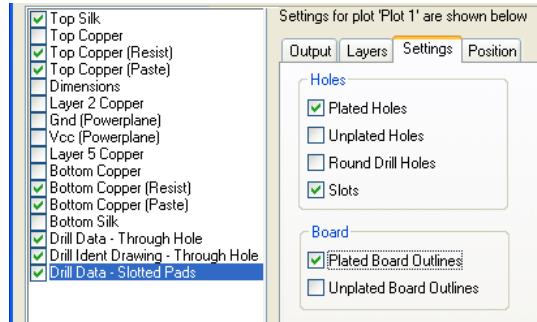
Layers Bar Categories (PCB)

Within the **Layers Bar**, a 'Categories' on/off selections has been added. A new button **Show>>** and **<<Hide**, has been added. Once revealed, this provides you with a quick mechanism for switching on or off specific categories of display items.



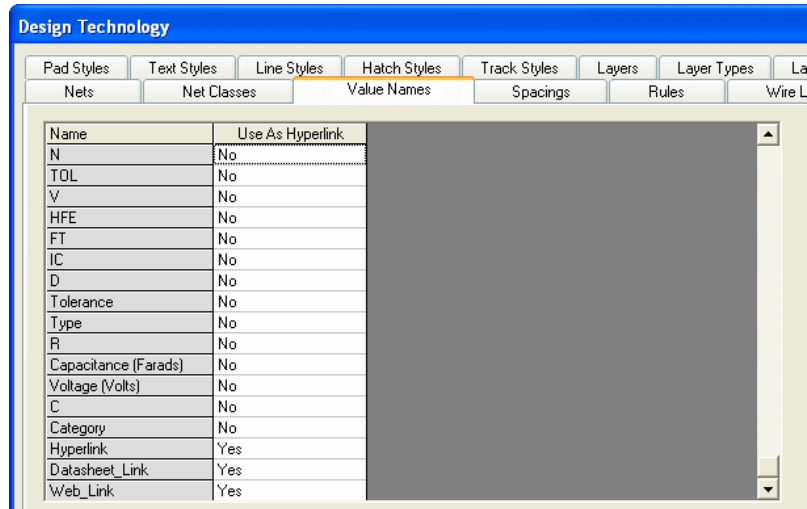
Milling of board outlines (PCB)

You can now output board outlines as routs in a Drill Data type of plot. There are now two board settings under **Board** on the **Settings** tab for this type of plot. You can choose to output plated or unplated outlines. The outlines will be milled with the width of the thickness of the board line style. If this is too small, it will use the minimum routing tool size defined in the NC Drill setup dialog.



Values used for Hyperlinks or Hyperlink Values (Both)

The new **Value Names** tab has been added to the new **Design Technology** dialog. From this, you can specify whether a Value name can be considered as a **Hyperlink** or not.

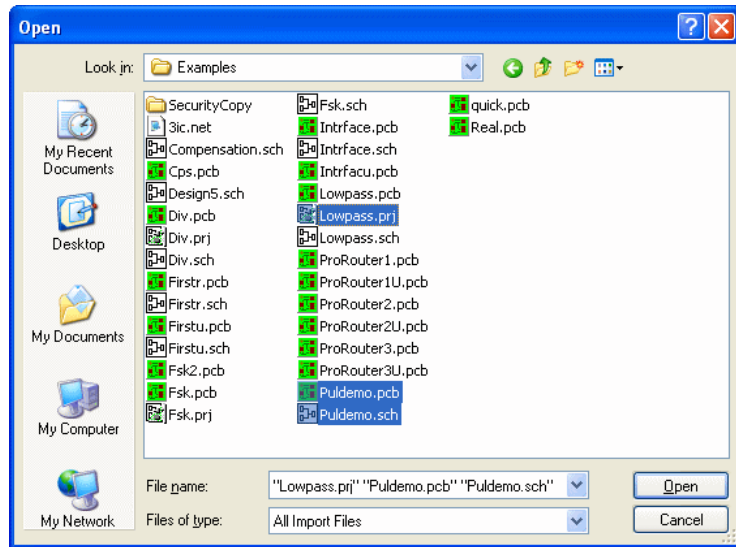


Selecting components that have any of the above attributes will have the **Execute Hyperlink** command on the context menu. Also, free text and callouts that look like URL's or filenames have the **Execute Hyperlink** command on the context menu.

Open Multiple Files at once (Both)

From the **File** menu, you can now multi-select designs to open at the same time using the **Open** dialog. They are still opened in their own design editors separately.

Multi-select files can be done using frame select, Ctrl-pick or Shift-pick.



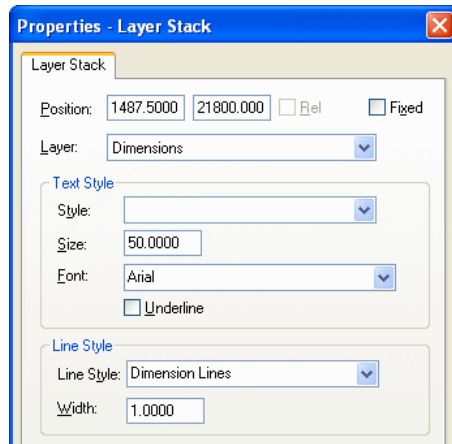
Insert Layer Stack Preview (PCB)

Insert Layer Stack Preview

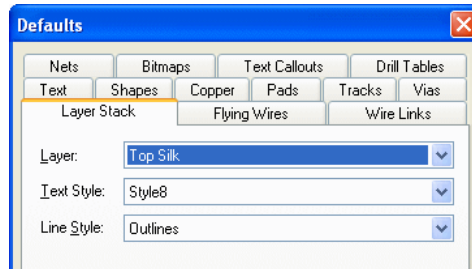
The **Layer Stack** preview can now be added to the design. Use the **Add Layer Stack** command from the **Add** menu. Once added to the design, it can be printed or plotted as required. The Layer Stack preview can be added to any layer in the design, it can also be moved to another layer using the **Change Layer** command from the context menu.

Once added to the design, if your layer stack changes, uses the **Update Layer Stack** command from the context menu to update the preview.

You can view and change **Properties** of the Layer Stack Preview by selecting it in the design. **Text Styles** and **Lines Styles** used for the preview can be changed.

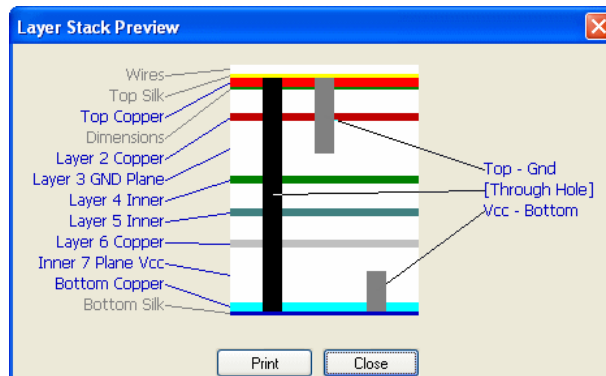


Default Properties of the **Layer Stack** preview being added can be defined in the **Defaults** dialog on the **Settings** menu and **Layer Stack**.



Print Layer Stack Preview

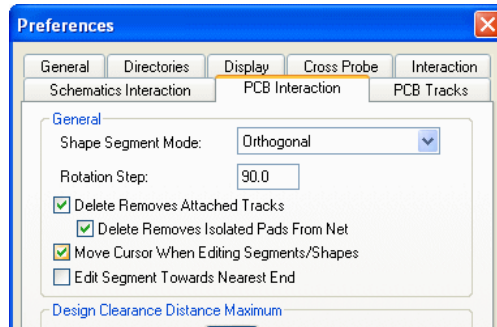
You can now print the **Layer Stack Preview** using the **Print** button on the dialog.



PCB Interaction Preferences (PCB)

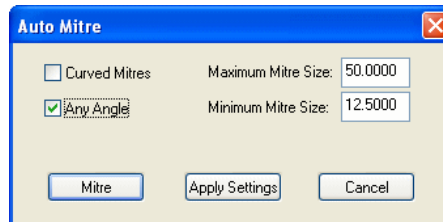
A new check box for **Delete Removes Isolated Pads From Net** has been added the **Preferences** dialog under **PCB Interaction**.

When checked (the default setting), this does what the program has always done when deleting items – if after delete you end up with pads/nodes that are ‘isolated’ (not connected by track to anything else on the net) they are removed from the net. When unchecked, Delete will no longer take those pads out of the net, you would need to use the **Remove From Net** command manually if you want them disconnected.



Auto Mitre Any Angle (PCB)

From within the **Auto Mitre** option, you can now select the **Any Angle** check box. This enables the auto mitre option to mitre tracks with non-orthogonal angles, normally only 90 degree corners are mitred.



Library Contents Improvements (Both)

Inconsistencies have been removed from the standalone set of libraries supplied with the product and some items have been moved into more appropriate libraries.

The component libraries (.cml) that have been removed are: **AN3, Analogue, Bipolar1, Demolib, Fet1, Fet2, Layan, Opamp1** and **Pulsar**.

The Symbol libraries (.ssl) that have been removed are: **Demolib** and **Layan**.

The PCB Symbol libraries (.psl) that have been removed are: **Demolib** and **Layan**.

The library **Transistr** has been renamed to **Transistor**.

Other Library Changes

Connector.cml - was Conn_std. Some connectors have also been moved from Discrete.cml

Opamps.cml – now contains Opamp1, and op-amps from Analogue.cml

Transistor.cml - was Trnsistr and now contains previous component libraries Bipolar1, FET1, FET2, and transistors from Analogue.cml