



## **V13.0 Supplement**

Copyright ©1987-2009 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](mailto:info@numberone.com)

Technical: 01480 382538

Email: [support@numberone.com](mailto:support@numberone.com)

Web site: [www.numberone.com](http://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: 13/07/09 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

<b>CONTENTS</b> .....	<b>4</b>
<b>CHAPTER 1. GETTING STARTED</b> .....	<b>7</b>
Installation .....	7
Data Files Location.....	7
Running Easy-PC For Windows 13.0 .....	7
Windows 2000 Support.....	7
<b>CHAPTER 2. NEW FEATURES IN EASY-PC V13.0</b> .....	<b>9</b>
Introduction.....	9
Rulers (Both).....	9
Dual Screen Support (Both).....	13
Part Editor Improvements (Both).....	15
Copy/ Paste Improvements in grid.....	15
Import Pin Information.....	16
Apply Increment to Selected Cells.....	18
Pan and Zoom Improvements (Both).....	19
Find using Extended Values (Both).....	20
Report in Library Manager (Both).....	20
Project Print (Both).....	20
PDF Plot (Both).....	21
Colour for moving items (Both).....	21
Status Bar (Both).....	22
Barcode Text (Both).....	22
Bitmap Import Supports JPEG and TIFF Formats (Both).....	22
Move From Bin (Both).....	23
Draw 'Empty' Value Positions (Both).....	23
Display of Component Value (Both).....	23
Highlights/Drawing (PCB).....	24
Bitmap Visibility and Picking (Both).....	24
Change Net Colour (Both).....	25
Least Used Menu Option (Both).....	25
Report File Extensions (Both).....	26
Draw Screen Grid as Lines (Both).....	27
Technology Dialog – Copy Style As (Both).....	27
Online Manuals Menu (Both).....	27
Library Dialog – Close on Edit (Both).....	28
Design Rule Check Report (Both).....	28
Flip Bus Terminal (SCM).....	28
Spice support for LT Spice (SCM).....	28
Retrieve From Bin (PCB).....	29
Swap Between Gerber RS-274-X and RS-274-D using a button (PCB).....	29
Extra check to Ignore Net Class track styles (PCB).....	30
Additional Pad Shapes (PCB).....	30
Shape Properties – Shape Area (Both).....	31
Track Length Rules on Net Classes (PCB).....	31
Composite Power Planes (PCB).....	34
Testland Vias on resist plots.....	35
3D View Changes (PCB).....	35
Named Areas (PCB).....	36
Arrange Components (PCB).....	36
Sheet Names in PCB (PCB).....	38
DXF Output Improvements (PCB).....	39

Show Nearest Node On Net (PCB).....	39
Next/Previous Layer Command (PCB).....	40
Change Layer option (PCB).....	40
DRC for 'Vias in Pads' (PCB) .....	41
DRC 'Text shapes' (PCB).....	41
DRC Silkscreen information (PCB).....	42
DRC Component-to-Component (PCB).....	42
Sign-Off Checks (PCB) .....	43
'Accepted' DRC Errors (PCB).....	45
Specetra Export menu option (PCB) (Existing Cost Option).....	48
GenCAD Output (PCB) (New Cost Option) .....	48
Micro Library Update (Updated Cost Option).....	49
Pro Library Update (Updated Cost Option).....	49
Connector Library (New Cost Option).....	49



# 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 V12.0 can install V13.0 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 13.0** 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.

## Data Files Location

There is now 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, “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 13.0

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.

## Windows 2000 Support

Support for Windows 2000 is withdrawn at the release of Easy-PC 13. Please visit the NumberOne.com web site for further information regarding our operating system support policy.



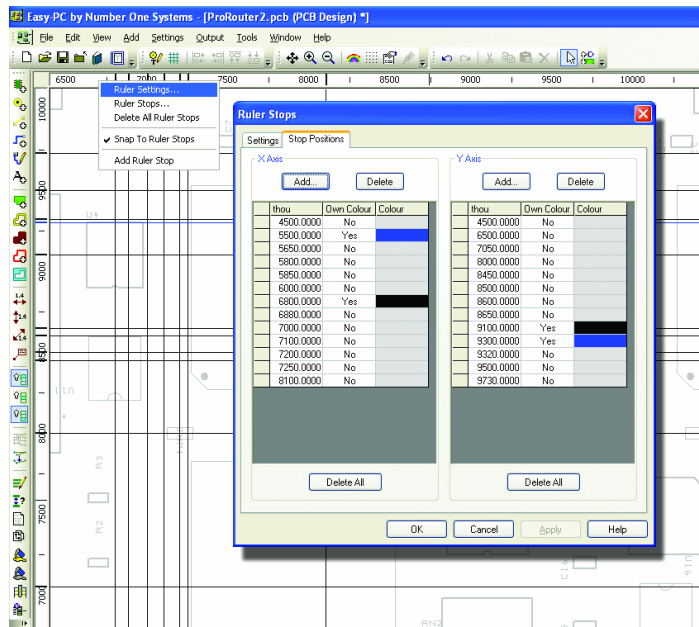
## Chapter 2. New Features in Easy-PC V13.0

### Introduction

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

### Rulers (Both)

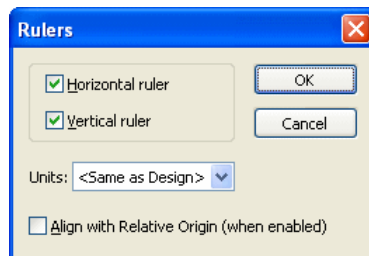
You can choose to have vertical and horizontal rulers displayed along the left and top margins of an Easy-PC design window. Rulers can be useful for sizing or aligning design items in terms of physical units such as millimetres, inches or thou.



### Changing Ruler Settings

To change the ruler settings, select **Rulers** from the **View** menu or right click on the rulers bar and use **Ruler Settings** from the context menu.

The following dialog will be displayed:



From this dialog you may choose whether you wish to display a **Horizontal ruler** and/or a **Vertical ruler** or neither. This can be different for a PCB or a Schematic design.

You can choose the **Units** to be used for the rulers scale from the dropdown list. If you select **<Same as Design>** the rulers will always shown the same units that are being used in the design, set up using the **Units** dialog.

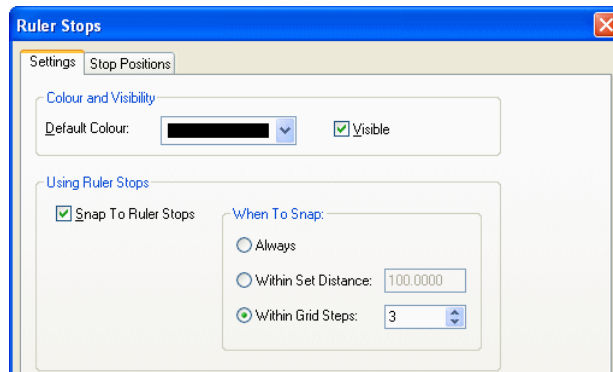
Setting the **Align with Relative Origin** option will make the rulers scale align zero with the Relative Origin rather than the Coordinate System Origin whenever **Relative Coordinates** are active.

### Ruler Stops

You can define ruler stop positions on each of the rulers, displayed as a small line in the bar. You can also optionally display a full width/height line in the design where the ruler stops are defined. These lines are like grid dots, and cannot be selected in the design. They may help you to align and size items, or may be used as an alternative to a grid to snap items onto known positions. The ruler stop positions can be added by double clicking on the ruler bar, directly typed in, or created at the origin of the selected items in the design (see below).

Ruler Stops will be saved with the design or symbol being edited.

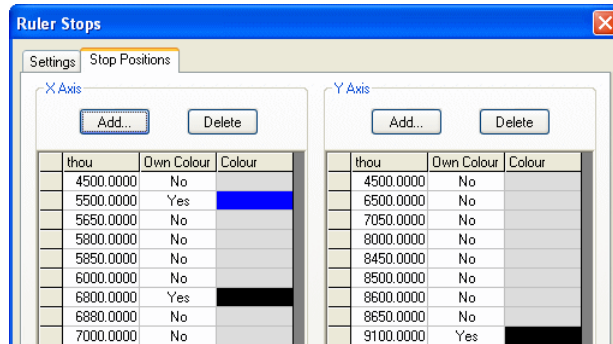
To display the Ruler Stops dialog, select **Ruler Stops** from the **Settings** menu, or right click on a ruler bar to use Ruler Stops from the shortcut menu, or alternatively, you can use a defined shortcut key or toolbar button. The following dialog will be displayed:



The **Settings** tab contains the default colour of the ruler stops, both in the ruler bars and in the design. The visible check box is used to say if the stop lines are to be displayed in the design. These values can also be changed in the Display dialog.

Check the **Snap To Ruler Stops** box if you want interactive operations on design items to snap to ruler stop positions. The **When To Snap** section defines how close the item needs to be to a stop position before it snaps onto it. The **Always** option makes ruler stops act like a grid, and the item will only move between the intersections of the stop lines.

To set up the ruler stop positions, select the **Stop Positions** tab in the dialog. The following is displayed:



The two halves of the dialog page show the stop positions along both axes. Press the appropriate **Add** button to add a new stop position at the end of the list. Don't worry if the positions are out of order, they will be sorted into order the next time you enter the dialog. For each ruler stop enter the position, whether it has its own colour, and its own colour. Pressing the return key will add another stop to the bottom of the list, enabling you to type in a list of values using the return key after each entry.

The ruler stop values will use the units defined for the ruler bars and will be relative to the zero position on the ruler bars. This will either be the relative origin (if defined and used by the rulers) or the system coordinate origin.

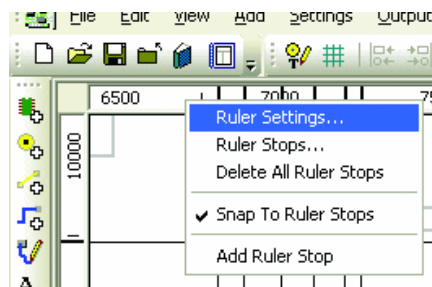
Use appropriate **Delete** button to remove the selected ruler stop. Use **Delete All** to remove all stops in the list.

You can drag ruler stops along the ruler to change their position and the **Stop Positions** table will be automatically updated with this new positional data.

You can also double click on the ruler bar to add ruler stops without having to edit the ruler stops dialog manually.

### Rule Stop Settings

You can right click on the ruler bars to use a context menu to access the dialogs mentioned above, and also to perform some other operations on ruler stops.



- Use **Delete All Ruler Stops** to quickly remove all the stop positions from both axis.
- Use **Snap To Ruler Stops** to toggle interactive snapping of design items to stop positions.
- Use **Add Ruler Stops at Selection Origin** to add a horizontal and vertical stop position at the origin of the selected items. This command is available to put on a shortcut key, and is useful for displaying a line through particular items to use to line up other items.

If a ruler stop existed at the point that you right clicked on the ruler bar and one or more items of the same type are selected in the design, two **Align To Ruler Stop** options will be available. These allow you to **Align** either the lower or upper side of each selected item to the picked ruler stop position.

If a ruler stop existed at the point that you right clicked on the ruler bar the **Edit Ruler Stop** and **Delete Ruler Stop** will be available to allow you to change the value of, or delete, the picked ruler stop. If no ruler stop was underneath the picked position, the **Add Ruler Stop** option is available. Use this to add a new stop at the position picked on the bar.

A new ruler stop can also be added by double clicking on the ruler bar.

You can reposition a ruler stop by dragging it along the ruler by pressing and holding the left mouse button, release the button at the new location.

### **Aligning to guide lines**

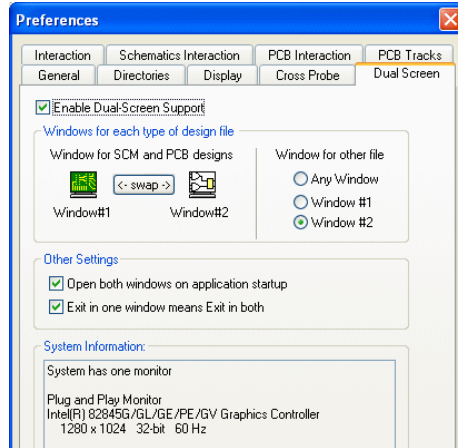
Also when selected, a ruler stop and one or more same type items are selected in the design then two of **Align Left To Ruler Stop**, **Align Right To Ruler Stop**, **Align Bottom To Ruler Stop** and **Align Top To Ruler Stop** will be available. You can use these ruler stop lines to align a single item, or multiple items of the same type, by right clicking on the ruler stop in one of the ruler bars and using one of the two Align options on the pop-up menu. This option works the same way as the Align functionality described above.

An alternative to using the Align tool is to define Ruler Stop lines and interactively snap items onto these lines when adding shapes or moving items. The distance from the stop lines where snapping occurs can be set using the ruler stop settings.

Some interactive options will snap to ruler stop positions, for example, **Adding** and **Editing** of shapes, **Move** and **Move Corner** of anything, etc. This is done by snapping to the positions if close enough, the snap distance is set up in the **Ruler Stops** dialog on the **Ruler Settings** menu.

## Dual Screen Support (Both)

Support for two monitors has been added to Easy-PC 13. Use the **Dual Screen** tab for the **Preferences** option on the **Settings** menu to access the **Dual Screen** preferences dialog. This page of the **Preferences** dialog allows you to change how the application arranges separate instances across multiple monitors or on a single large monitor.



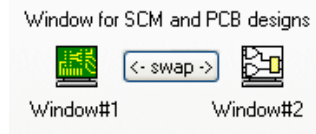
### How it works

Check the **Enable Dual-screen Support** check box to turn on multi-screen operation. With this enabled, two instances of the application will work together to arrange your files so you can easily work on Schematic designs in one, and PCB designs in the other. Each instance will have its own menus and toolbars and work independently, but will share information when they need to.

### Windows for SCM and PCB designs

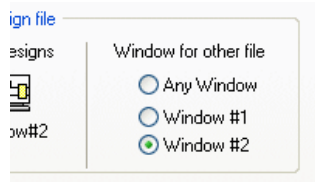
This setting shows which window is used for each design type. Although it does not really matter which way round they are, instance #1 is the one that will be launched first if you run Easy-PC with no designs open.

The small picture shows which design type will be displayed in which instance. To swap them round, simply click the 'swap' button.



### Windows for other file types

As well as Schematic and PCB designs, Easy-PC is of course used to edit all your library items (parts, footprints, etc). You can choose where these should be opened for editing. You can choose either of the two specific program instances, for example you may wish to always edit library items in the same instance as your Schematic designs, or you can allow them to be opened in the instance in which the editing 'action' takes place.

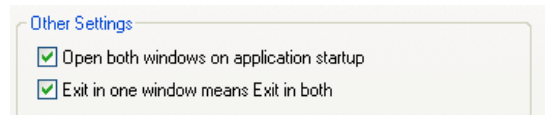


### Other Settings

#### Open both windows on application start-up

With this checked, both instances of the program will be started when you launch Easy-PC. With it unchecked, launching Easy-PC will either start instance #1 (if no designs are open), or if you are opening a particular design file then it will launch the instance for that type of file.

There is no recommended setting for this switch, it simply depends on how you prefer to work. The default setting is On.



#### Exit in one window means Exit in both

Check this box to make both instances close down if you close one of them. This makes them work more like a 'pair', but again it is simply down to how you want it to work.

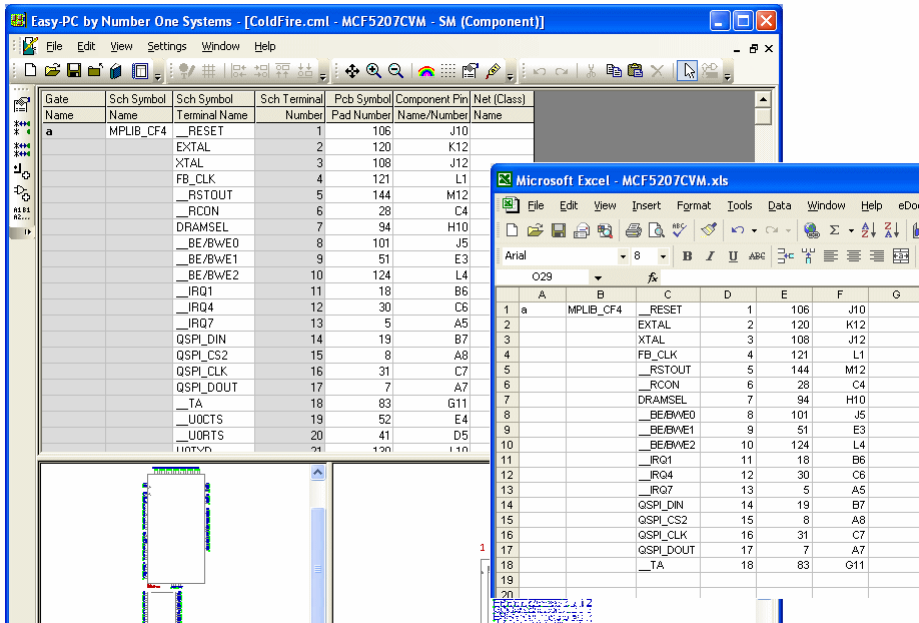
The default setting for this switch is On.

## Part Editor Improvements (Both)

### Copy/ Paste Improvements in grid

#### Component Editor dialog

It is now possible to Copy and Paste data between the Component Editor grid and external applications such as Microsoft Excel and vice versa. A single cell or a range of cells can be selected for copying or pasting. It is also possible to paste the same value in to a range of cells.



#### Using Copy and Paste To Edit Values

The values in the **Pin Assignments** grid can be manipulated using the normal Copy and Paste techniques associated with spreadsheet applications such as Microsoft Excel. This includes the ability to copy and paste multiple rows and/or columns of data. One possible use would be to copy the pin data into an external spreadsheet to take advantage of advanced features such as sorting and filtering, then paste the modified data back in to the Part Editor. When pasting from an external source it is important to ensure that the data is arranged in the same order as the cells in the grid.

The Copy and Paste commands are available from the context menu. Alternatively, the defined shortcut keys, by default Ctrl-C and Ctrl-V may be used.

#### Selecting cells to copy or paste

A single grid cell or a range of grid rows and/or columns may be selected for copy or pasting. Clicking on a single cell will select just that cell. If, rather than clicking, the mouse is dragged while the left button is held down a range of cells may be selected, indicated by being shown in reverse highlight. This range of cells can now be copied or pasted as a whole. If a single cell has already been selected by clicking, it is possible to

extend this into a range of cells by clicking with the <shift> key held down. To quickly select an entire column click in its header cell.

### Pasting Values into the Grid

When data is pasted into grid the values are subject to the normal constraints of the column into which they are being inserted and are checked in the same manner as when values are typed in. For example, a value pasted into the **Sch Symbol Name** column will be checked to ensure that it is a valid name of a symbol defined in the library.

The **Gate Name** and **Sch Symbol Terminal Name** columns are read only and as such may not have values pasted into them although they may be selected and copied.

If, when pasting a range of values, a single cell is selected in the grid, the cells affected will match the range of the paste values subject to the constraints of the rows and columns affected.

By copying a single value then selecting a range of cells, it is possible to paste the same value into multiple cells. All cells in the selected range of cells will be set to the pasted value, subject to the column constraints.

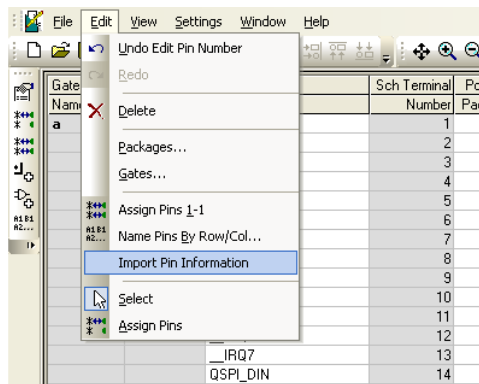
All paste operations may be reversed using the standard **Undo** facility.

### Component Values dialog

The **Component Values** dialog has also been similarly modified to allow Copy/Paste to/from Excel or similar, although the values seen in Excel will include additional 'control' tags used for the colour coding employed in the Component Values dialog.

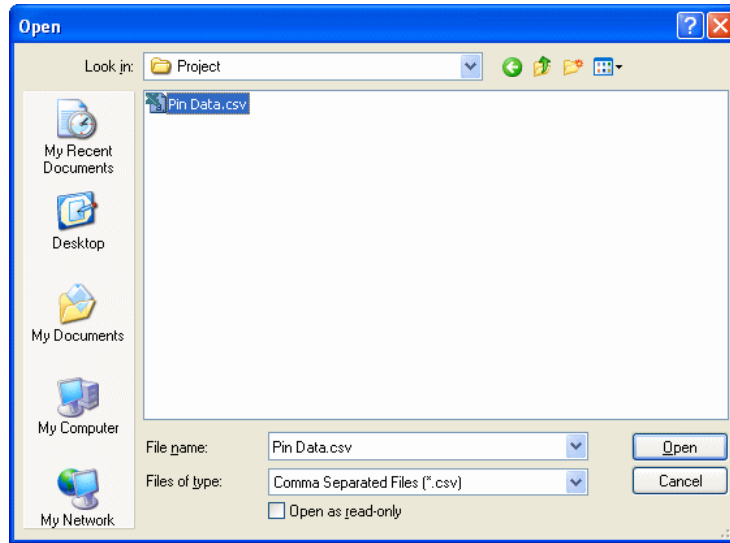
## Import Pin Information

You can import pin information into the component editor grid when editing a component. The **Import Pin Information** option is available from the **Edit** menu.



Only pin values, and not gate or symbol values may be imported using this option.

When selected from the **Edit** menu, you are presented with an **Open** dialog from which to choose the CSV file for import.



Once the required file has been selected, click the **Open** button to perform the import.

If any problems with the import are detected, a report will be displayed showing the results of the import.

### Pin Data Format (CSV)

Pin data may be imported in standard CSV format, such as can be created or manipulated in an external spreadsheet application such as Microsoft Excel.

The first row of the CSV file contains the column headings for the pin information included. The column headings have to match those shown below but do not have to be in the same order and do not all have to be included. Omitted fields will be unaffected by an import.

CSV column headings	Corresponding Component Editor column headings
pad number	PCB Symbol Pad Number
pin name	Component Pin Name/Number
terminal name	SCH Symbol Terminal Name
net name	Net (Class) Name

Either the **pad number** or the **pin name** field must be included to map the CSV rows to the appropriate pins in the Component Editor. If both fields are included, the pad number will be used to map the pin name, and any other included fields, to the appropriate Component Editor pin.

If **Terminal Name** values are included, they will be assigned to all gates of the same type as the one to which the included pins are mapped.

The CSV file does not have to include entries for all the part pins. Those pins not present will be unaffected by the import thus allowing partial pin assignments to be made. The import report will contain a warning if not all pins are included as a precaution against accidental omissions.

An example pin information CSV file is shown below:

```
Pad Number,Pin Name,Terminal Name
1,L09,__RESET
2,H01,CLKIN
3,G01,XTAL
4,J01,CLKOUT
```

### Apply Increment to Selected Cells

The **Apply Increment to Selected Cells** option is available on the context menu when an appropriate range of cells is selected cell for Component **Pin Name/Number** cells in the **Component Editor**. This can be used to name one cell and create an incremental sequence from it to each subsequently selected cell.

Sch Symbol	Sch Symbol	Sch Terminal	Pcb Symbol	Component Pin	Net (Class)
Name	Terminal Name	Number	Pad Number	Name/Number	Name
MPLIB_CF4	_RESET	1	106	J10	
	XTAL	2	120	K12	
	XTAL	3	108	A1	
	FB_CLK	4	121	L1	
	_RSTOUT	5	144	M12	
	_RC0N	6	28	C4	
	DRAMSEL	7	94	H10	
	_BE/BWE0	8	101	J5	
	_BE/BWE1	9	51	E3	
	_BE/BWE2	10	124	L4	
	_IRQ1	11	18	B6	
	_IRQ4	12	30	C6	
	_IRQ7	13	5	A5	
	QSPL_DIN	14	19	B7	
	QSPL_CS2	15	8	A8	
	QSPL_CLK	16	31		
	QSPL_DOUT	17	7		
	_TA	18	83		
	_UOCTS	19	52		
	_UORTS	20	41		
	_UORTS	21	120		

### ► To use Apply Increment to Selected Cells

1. Name the first cell, A1 for example.
2. Select the first cell and the next 14 cells by dragging the mouse over them.

Gate	Sch Symbol	Sch Symbol	Sch Terminal	Pcb Symbol	Component Pin	Net (Class)
Name	Name	Terminal Name	Number	Pad Number	Name/Number	Name
a	MPLIB_CF4	_RESET	1	106	J10	
		XTAL	2	120	K12	
		XTAL	3	108	A1	
		FB_CLK	4	121	L1	
		_RSTOUT	5	144	M12	
		_RC0N	6	28	C4	
		DRAMSEL	7	94	H10	
		_BE/BWE0	8	101	J5	
		_BE/BWE1	9	51	E3	
		_BE/BWE2	10	124	L4	
		_IRQ1	11	18	B6	
		_IRQ4	12	30	C6	
		_IRQ7	13	5	A5	
		QSPL_DIN	14	19	B7	
		QSPL_CS2	15	8	A8	
		QSPL_CLK	16	31		
		QSPL_DOUT	17	7		
		_TA	18	83		
		_UOCTS	19	52		
		_UORTS	20	41		
		_UORTS	21	120		

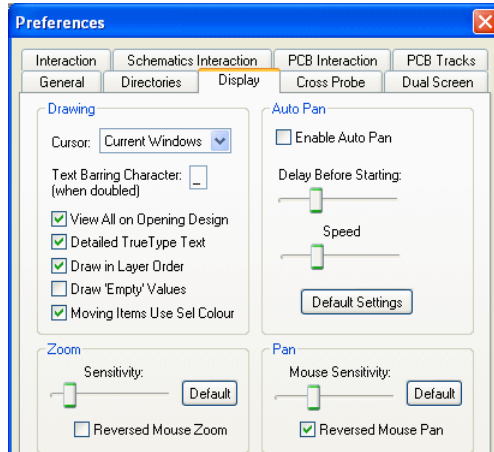
3. Select **Apply Increment to Selected Cells** option from the context menu.

Gate Name	Sch Symbol Name	Sch Symbol Terminal Name	Sch Terminal Number	Pcb Symbol Pad Number	Component Pin Name/Number	Net (Class) Name
a	MPLIB_CF4	_RESET	1	106	J10	
		EXTAL	2	120	K12	
		XTAL	3	108	A1	
		FB_CLK	4	121	A2	
		_RSTQOUT	5	144	A3	
		_RCON	6	28	A4	
		DRAMSEL	7	94	A5	
		_BE/BWE0	8	101	A6	
		_BE/BWE1	9	51	A7	
		_BE/BWE2	10	124	A8	
		_IRQ1	11	18	A9	
		_IRQ4	12	30	A10	
		_IRQ7	13	5	A11	
		QSPL_DIN	14	19	A12	
		QSPL_CS2	15	8	A13	
		QSPL_CLK	16	31	A14	
		QSPL_DOUT	17	7	A15	
		_TA	18	83	G11	
		_UOCTS	19	52	E4	
		_UQRTS	20	41	D5	
		_UQRTS	21	130	L10	

4. The **Name/Number** range will now show A1, A2, A3, A4, A5 etc. in the selected cells.

## Pan and Zoom Improvements (Both)

You can now change the **Pan** and **Zoom** sensitivity and direction of movement using sliders and check boxes on the **Settings** menu, **Preferences** dialog under **Display**.



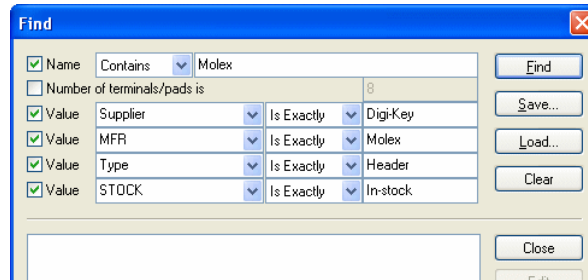
The sliders will provide you with the sensitivity required. Moving them to the left will mean they are less sensitive. When rolling the mouse backwards or forwards, the effect of this will be that the zoom is slower or faster, i.e. bigger or smaller increments.

Checking the **Reversed Mouse Zoom/Pan** check box will reverse the current direction. For **Pan**, we recommend checking this box, this will also be the new install default. The Reverse Pan direction will enable Easy-PC to work more like some other products like Adobe Reader.

## Find using Extended Values (Both)

The **Find** dialog within the **Library Manager** and **Add Component** dialog has been extended to allow up to 4 component values (attributes) to be used for searching. Values can also be compared using the same 'operators' as the Name (is, begins with, contains).

Also added is the ability to save search settings to a file and load them again, meaning commonly used searches can be kept. You can use as many or as few values as you like to refine the search.



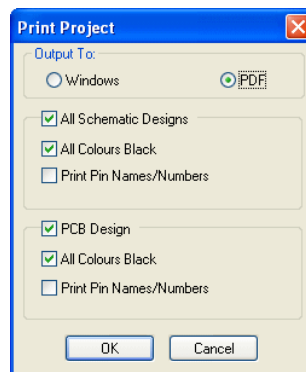
## Report in Library Manager (Both)

The **Component Library report** (run from within the Component Library Manager) now shows the name of the looked up symbol library rather than the 'originating' symbol library. That is, the name of the library from which the symbol will be loaded when the component is added to a design.

When running the Component Library report there is an additional prompt asking **Show library folders for symbols**. If answered **Yes**, the report will show the full folder path for each looked up symbol library as well as its name.

## Project Print (Both)

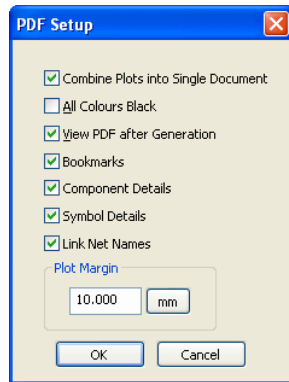
You can now choose to **print to PDF** from the project level. This output is available on the **File** menu and **Print**. The **Print Project** dialog presents you with an additional radio button to select the PDF output.



## PDF Plot (Both)

When using the **Plotting and Printing** option (from the **Output** menu), you can define a margin to be applied around a PDF plot. This is available on the PDF setup dialog under **Options** and **PDF**.

The **Plot Margin** is defined using the units shown on the dialog. These can be changed by selecting the units button.

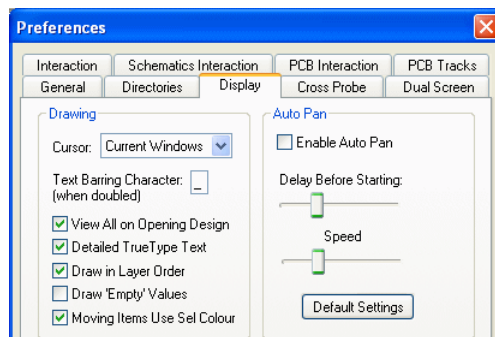


## Colour for moving items (Both)

There is a new check box, **Moving Items Use Sel Colour**, on the **Settings** menu, **Preferences** dialog under **Display**.

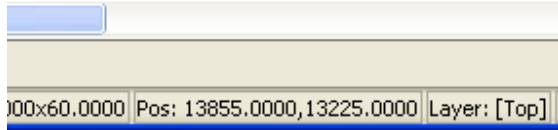
When selected, this retains the existing behaviour of drawing the selected moving item(s) in the selection colour, the default setting is On (checked).

When unchecked, dynamic selected items are instead drawn in their normal colour, so when adding tracks for example, you can see by the colour which layer the current segment is on, instead of just seeing it in the selected colour. This option has been added to aid some situations, when editing a track for example, where it might be better to actually see the normal colour of the selected segment.



### Status Bar (Both)

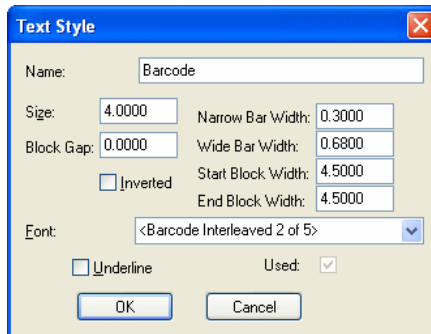
For selected items in a design, you can now show its position on the **Status Bar**. The position of **Components, Pads, Text, Vias** and **Callouts** is available under **Pos:**.



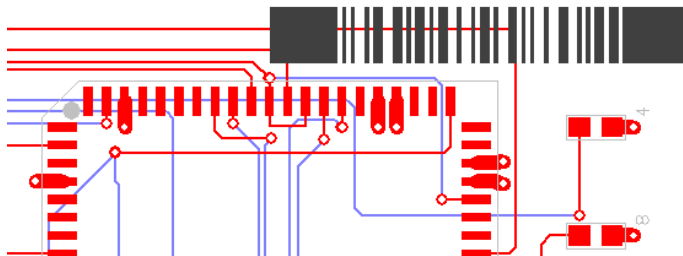
### Barcode Text (Both)

You can add barcodes into your design by adding text with a special barcode style.

To do this, can now define a barcode text style, only named barcode styles defined through the **Technology** dialog are allowed. To enable barcodes, you must use the **Barcode Interleaved 2 of 5** text font. Once this is selected, you can then define the bars and gaps used for your barcode.



In the design, use the **Add Text** to add the numbers that will define the barcode, i.e. 05505223. Then change the text style of the barcode to your barcode font and the design will now display a barcode. Until this special barcode font is used, the text will still appear as normal readable text.



### Bitmap Import Supports JPEG and TIFF Formats (Both)

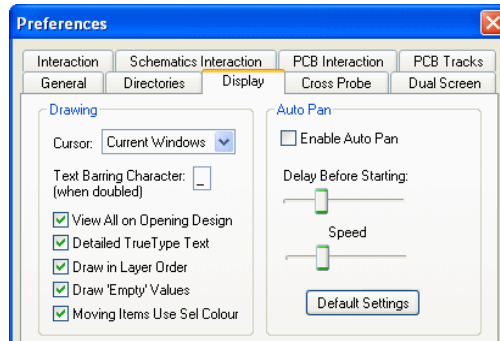
You can now import bitmaps in JPEG and TIFF formats as well as bitmaps.

## Move From Bin (Both)

When dragging a multiple selection from the **Component Bin**, it now uses the **Arrange** method of **stacking** components by Component Name instead of simply placing them in a single vertical line. This makes them easier to manage and place, and less likely to be spread off the working area or screen.

## Draw 'Empty' Value Positions (Both)

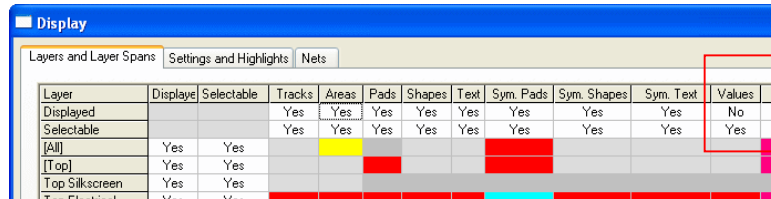
There is a new check box on the **Setting** menu, **Preferences** dialog under **Display** to **Draw Empty Value Positions**.



When checked, any value position that resolves to an empty string will be drawn as <?> instead. This is required because it is possible through the **Properties** dialog to have component values that have no text, e.g. by turning off all but **Reference Name** and then also turning off **Name** on the **Component** tab. From this you have value positions that you cannot see or pick, so they are using text styles but you cannot find them to change style, this new option will solve this.

## Display of Component Value (Both)

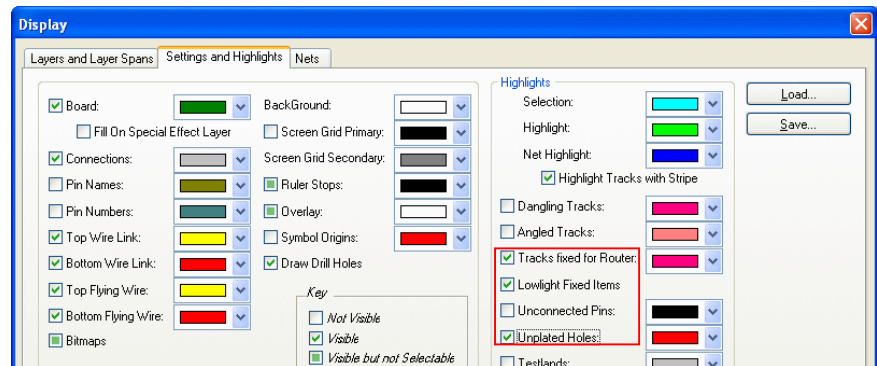
From within the **Display** dialog and **Layers and Layer Spans** tab, under **Values** you can now switch off component values in the design, values such as component name for example, and change the colour (this previously used the Symbol Text colour).



## Highlights/Drawing (PCB)

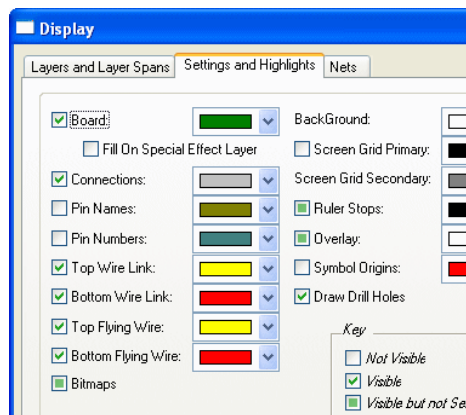
There are new highlight check boxes and colours on PCB colours (Display) dialog. Changes have been made for the following:

- **Tracks fixed for Router** (centre-stripe or whole colour as per existing 'angled' highlight).
- **Lowlight Fixed Items** (dims colour of pads and shapes of items that are fixed).
- **Unplated Holes** (draws small 'diamond' inside drill hole, only visible if **Draw Drills** is enabled in the **Preferences** dialog under **Display**).



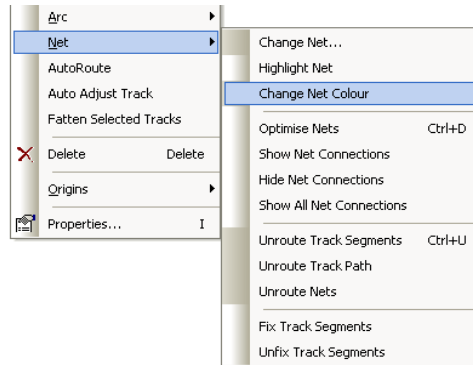
## Bitmap Visibility and Picking (Both)

The ability to set the visibility and selectability of **Bitmaps** on the **Display** dialog has been added. A check box on this dialog for **Bitmaps** will toggle through the three states.

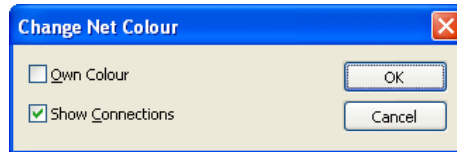


## Change Net Colour (Both)

When selecting a net or track (in PCB), on the context menu from the **Nets** > sub menu, you can now access the **Change Net Colour** option.



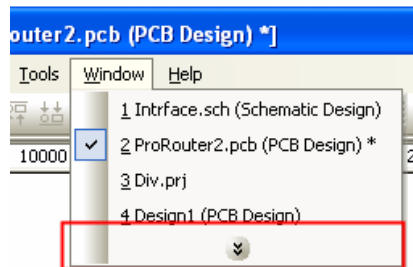
This dialog provides access to the net **own colour** and **show connections** switches and net colour. This is the same as the settings available on **Nets** page of the **Display** dialog.



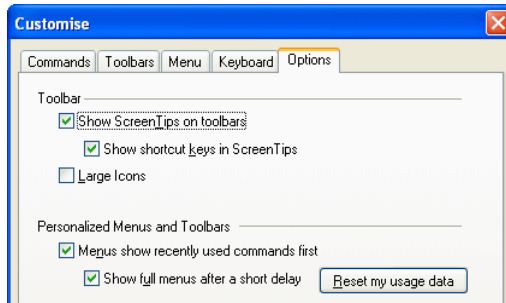
When accessing this in a Schematic design the **Change Net Colour** is directly on the context menu and there is no **Show Connections** check box.

## Least Used Menu Option (Both)

The ability to hide the least-used menu options has been added. This simplifies the menus and hides options which are less used. Items hidden can be revealed by selecting the small double-down arrows on the menus.



If you wish to have the full menus as before, the switch for this option is available under **Personalised Menus and Toolbars** from the **Settings** menu, **Customise** and **Options** page.

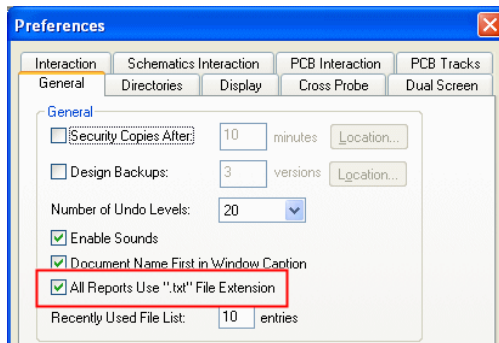


### Reset my usage data

This resets the menu commands shown on the short version of a menu to the original set. It does not change the location of toolbars, nor reset any buttons or commands moved, added or removed by using Customise.

### Report File Extensions (Both)

There is a new check box on the **Preferences** dialog and **General** page, this enables the ability to output all reports to a .TXT file extension instead of the existing drr, dcr, xyz, etc. file extensions.

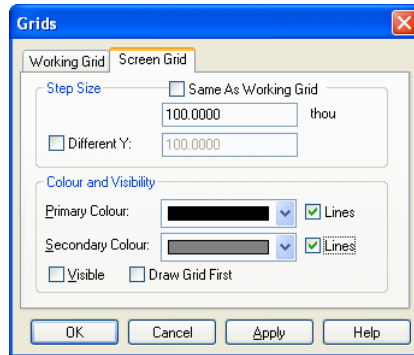


The output report will use the design type and title of report to form the pathname, for example "Design123 (PCB – Design Status Report).txt"

## Draw Screen Grid as Lines (Both)

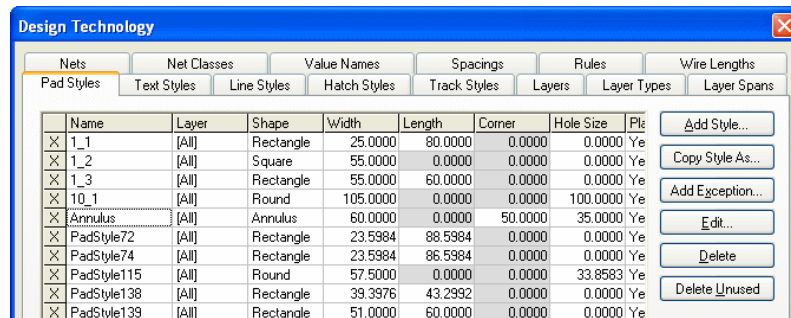
From within the **Grids** dialog, the **Primary** and/or **Secondary** screen grids can now be drawn as lines instead of dots.

You can also opt for them to be drawn *before* instead of *after* the design contents using the check box **Draw Grid First** on this dialog.



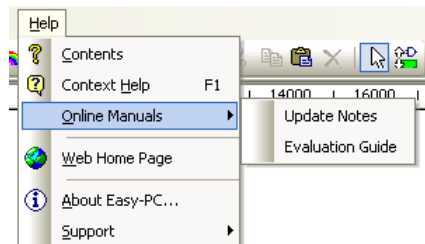
## Technology Dialog – Copy Style As (Both)

There is an extra button, **Copy Style As**, for **Styles** on the **Design Technology** dialog to add a new style based on the one currently selected in the grid.



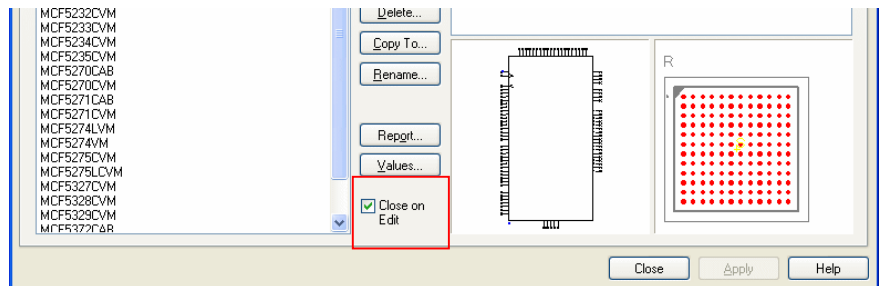
## Online Manuals Menu (Both)

There is a new sub-menu on the **Help** menu from which you can access PDF documents installed on your system.



## Library Dialog – Close on Edit (Both)

There is a new check box on the **Libraries** dialog that allows you to choose whether or not the dialog should be closed after opening a library item for edit.



This enables you to open a series of components for editing without it closing each time.

## Design Rule Check Report (Both)

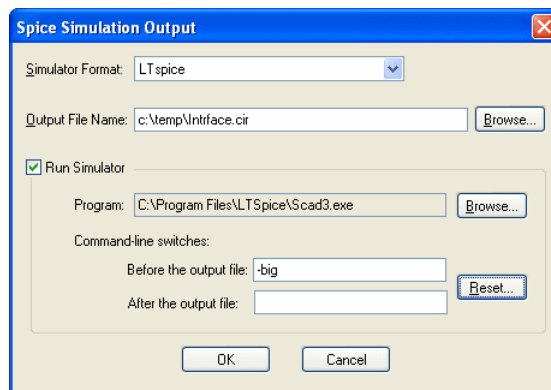
New reports are available on the Output menu and Reports for Design Rules Check report when in SCM or PCB. This reports DRC errors within the design without the need to run the DRC option.

## Flip Bus Terminal (SCM)

If bus terminals are selected, they are individually flipped so that the bus end moves and the connection end stays where it is if it can. You can also select multiple bus terminals and they are flipped individually and not as a group.

## Spice support for LT Spice (SCM)

Netlist support for the free Linear Technology (LT) Spice product has been added to Easy-PC 13.



First choose the **Simulator Format** for the simulation file. The output file will be formatted to suit the chosen simulator.

Use the **Output File Name** box to provide the name for the simulation output file.

If available you can use **Run Simulator** to specify whether to run the simulator after the netlist has been generated. Use the **Browse** button to locate the program to be run on your computer. Where possible we have added filters to this dialog to suggest a possible simulator program name.

Add any command line switches that are needed to run up the simulator, either before or after the output filename. Initially these will be set to default values for each simulator type. If you change them you can use the **Reset** button to revert back to the default values.

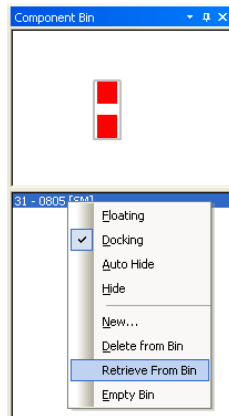
Press **OK** to continue with the simulation output process. If the output file already exists you will be asked for permission to overwrite it.

### LT Spice library

An LT Spice Component and Symbol library is available on [www.numberone.com/spicesupport.asp](http://www.numberone.com/spicesupport.asp) page under the LTSpice heading.

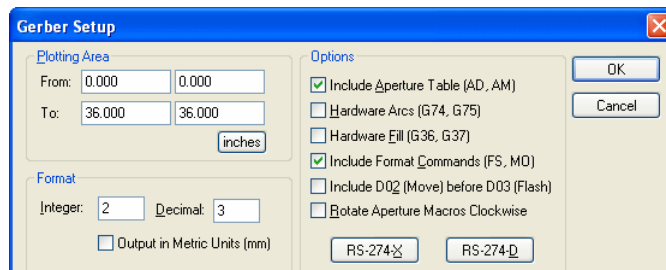
### Retrieve From Bin (PCB)

If components in PCB have been moved from the design to the Component Bin, the new **Retrieve From Bin** command is available by right clicking on one of the components in the Bin. This will return the selected item(s) to their previous location on the board.



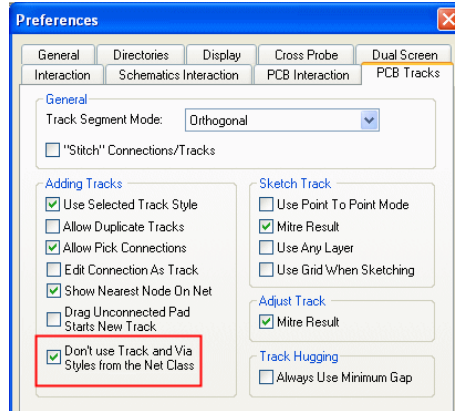
### Swap Between Gerber RS-274-X and RS-274-D using a button (PCB)

Two new buttons have been added to the **Gerber** setup dialog within the **Plotting and Printing** option to facilitate swapping between the Gerber RS-274-D and RS-274-X formats.



## Extra check to Ignore Net Class track styles (PCB)

There is a new switch in **Preferences** and **PCB Tracks** labelled **Don't use Track and Via Styles from the Net Class**. Check this box to always use the styles from the defaults dialog when interactively adding or editing tracks and vias. If this isn't checked, it will use the styles defined for the Net Class.



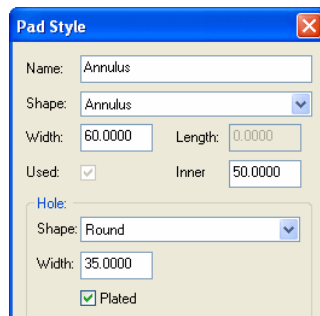
## Additional Pad Shapes (PCB)

You can now define target and annulus pad shapes for use in your designs.



For a target, you can define the line thickness used.

For annulus, you can define the **inner** ring size in the **Technology** dialog when editing the Pad style.



## Shape Properties – Shape Area (Both)

On selection of a shape, the **Properties** dialog shows the total area value in the current design units. It includes the total area with and without taking into account the line style thickness.

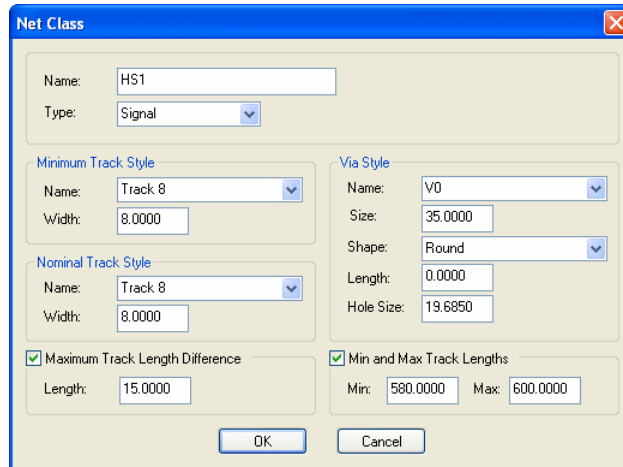


## Track Length Rules on Net Classes (PCB)

You can now add new values on a net class:

- length difference value
- min length value
- max length value

You can optionally enable the **Maximum Track Length Difference** and **Min and Max Track Lengths** options within the net class.



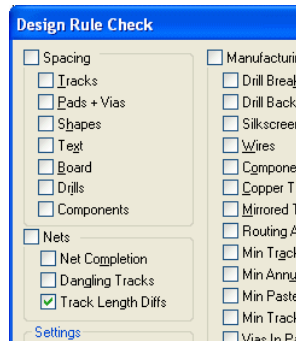
By adding these values and using the appropriate net class in your design, you can define a minimum and maximum length that the routed track must finish with. By adding a length difference value to the net class, you can then check that all the tracks using that net class will finish with their length within that tolerance. Either value can be set to zero to 'ignore' that length.

These new values are used by a new **DRC** check (Net, Track Lengths), and a new **Track Length Rules** report available from the **Output** menu and **Reports**.

### DRC Option

Within the **DRC** option, the **Track Length Diffs** calculates length of all tracks in the class, finds the median value, then shows you any net that is too far either side of this

value. The same check box enables the **Min/Max** check to simply verify track length of each net against specified values. Nets with no tracks are always flagged as errors.



### DRC Report

When the DRC option reports **Track Length Diffs** errors, the report typically might look like this:

```
Design Rule Check Report
-----
Report File      : c:\temp\ProRouter.txt
Report Written   : Saturday, July 04, 2009
Design Path     : c:\temp\ProRouter2.pcb
Design Title    :
Created        : 22/04/2004 1:28:41 PM
Last Saved     : 06/07/2009 10:47:57 AM
Editing Time   : 6470 min
Units          : thou (precision 4)

Results
-----
Track length differs from other nets of same class : Net N0009
too long, length is 602.7795 thou but needs to be no more than
600.0000 thou from rules in net class HS1

Net N0009 too long, length is 602.7795 thou but needs to be no
more than 600.0000 thou from rules in net class HS1.

Number of errors found : 1
```

### Track Length Rules

This report, available for PCB designs, analyses the tracks in the design to determine those that comply with the length rules defined in their net class.

The results produced by this report are listed by net class name, showing only those net classes which have length rules (Difference, and/or MinMax Length). For each one, it lists the rules, then lists each net that uses the net class. For each of those nets, it will show the actual length, the expected range of lengths of all nets within the class, and an indication of whether this net passes or fails the length rules.

When run from the **Output** menu and **Report** option, an example **Track Length** report might look like this:

Track Length Rules

-----

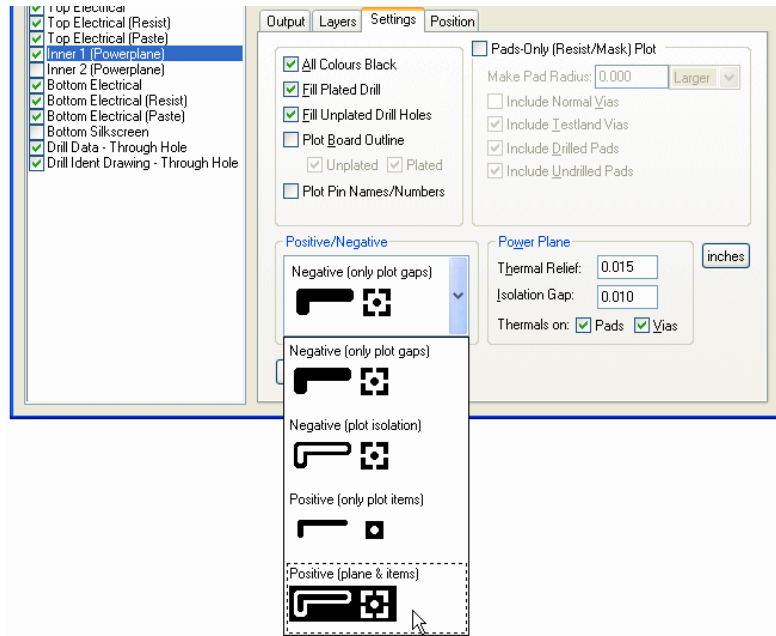
Report File : c:\temp\ProRouter.txt  
 Report Written : Saturday, July 04, 2009  
 Design Path : c:\temp\ProRouter2.pcb  
 Design Title :  
 Created : 22/04/2004 1:28:41 PM  
 Last Saved : 06/07/2009 10:47:57 AM  
 Editing Time : 473 min  
 Units : thou (precision 0)

Net Class	Rules			Net	Actual	Expected	range
Name	Diff	Min	Max	Name	Length	From	To
Fail							
HS1	15.0000	580.000	600.000	N0009	602.779	590.315	605.315
****				N0010	592.850	590.315	605.315

- Net Class** – This is the net class name that the rule applies to.
- Rules Diff** – This is the value defined by you in the rules dialog.
- Min/Max** – These are minimum and maximum lengths that you have defined for the net class.
- Net Name** – The actual net name of the net using this net class.
- Actual Length** – This is the actual length as measured in the design.
- Expected Range From/To** – The 'expected range' for the overall track length is worked out from the actual lengths of the nets in the net class. Those with no tracks are ignored, all the others are collected into a table and the median value is used as the expected length. The range shown is therefore the deviation allowed from this median value. If there are less than four nets using the net class, the average length of those nets is used instead of the median value.
- Fail** - Four \*\*\*\* will indicate clearly that the net fails the rules defined.

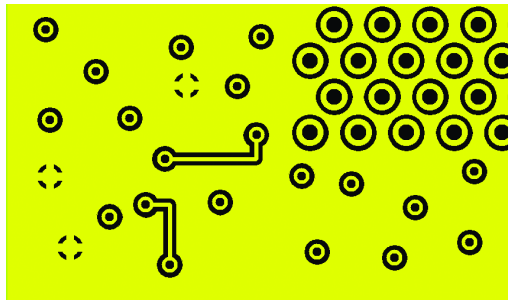
## Composite Power Planes (PCB)

From within the **Plotting and Printing** option, you can now plot **Positive (plane & items)** plots when outputting power planes with embedded items (such as tracks or text).



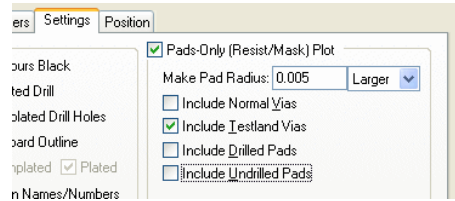
The net effect of a **plane & item** plot is that no additional composite of the plane is required and only one plot is supplied to your manufacturer. Previously, two plots were required which would then be combined and a composite produced.

A final composite plot might look something like this:



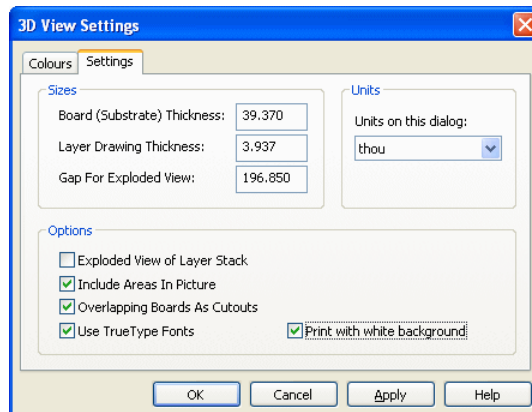
## Testland Vias on resist plots

On the **Settings** page of **Plotting and Printing** dialog for **resist/paste** plots (with **Pads-Only** selected) is a separate check box for enabling **Testland vias**. This allows 'normal' vias to be covered with resist whilst leaving Testland vias clear.



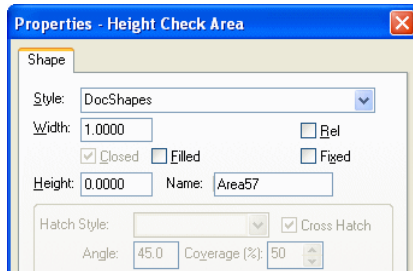
## 3D View Changes (PCB)

The background colour on printing can now be forced to white using the **Print with white background** check box enabled. Enable this from the **3D Settings** dialog on the **View** menu and under the **Settings** tab.

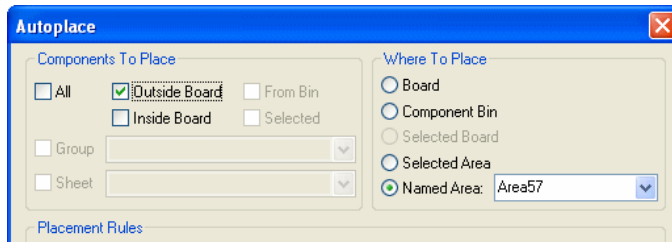


## Named Areas (PCB)

You now have the ability to **name** any **area** using the **Properties** dialog.

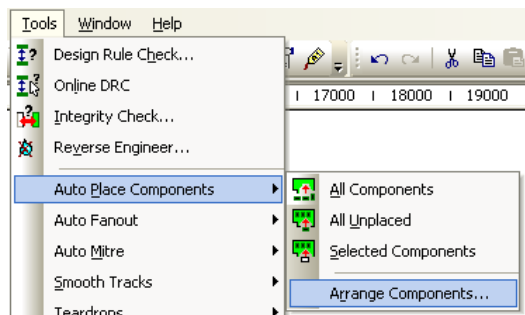


Naming an area allows it to be referenced by name in the **Arrange Components** and **Autoplace** options.

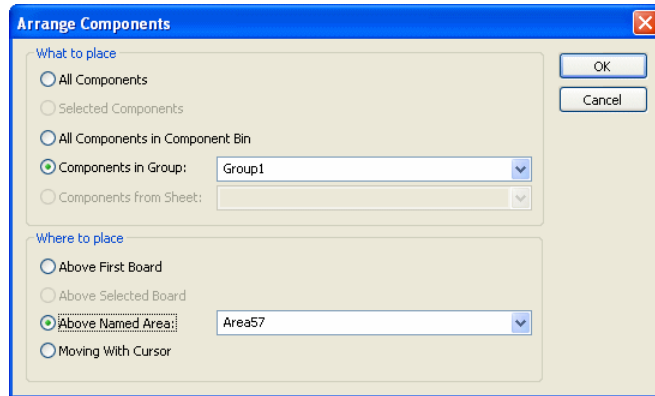


## Arrange Components (PCB)

There is a new command on **Tools** menu under **Auto Place** called **Arrange Components**.



This can be used to help you group together 'sets' of components in your PCB design as an aid to the initial placement on the board. These components are arranged in 'stacks' by component and package to make it easier for you to manage.



### What to place

Choose which components you would like to arrange. Some of the options may be greyed out, in the above example there are no components selected in the design and no groups defined.

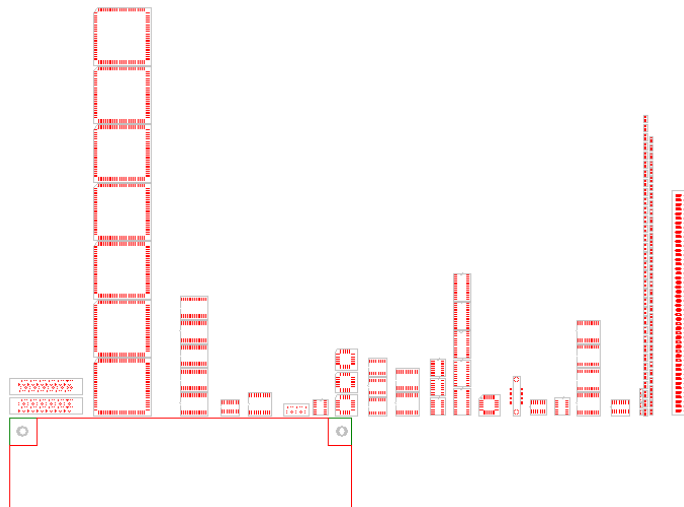
You can choose from **All** components, **Selected** components, from the component **Bin**, from components in a **Group** or from a schematic **Sheet**.

### Where to place

Choose where to put the chosen components. As this command is intended as an aid to manual component placement, the components will be placed 'above' the chosen location. For example if you choose 'Above First Board', then the start point for the arrangement is the top left corner of the bounding box of the first board outline in the design. The exception to this is Moving With Cursor when the arranged components will be moving around for you to place interactively.

You can place **Above first board** outline, above a **Selected Board** outline, above a **Named Area** and **Moving with Cursor**.

When run, this option stacks up the components by Component Name (part) and Package, (the same as **Translate to PCB** does if you don't use the **Component Bin**).

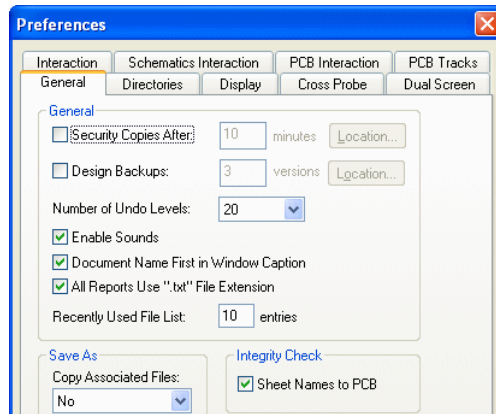


## Sheet Names in PCB (PCB)

Components in the PCB design can know which Schematic sheet they come from. This is useful for a project that has multiple sheets.

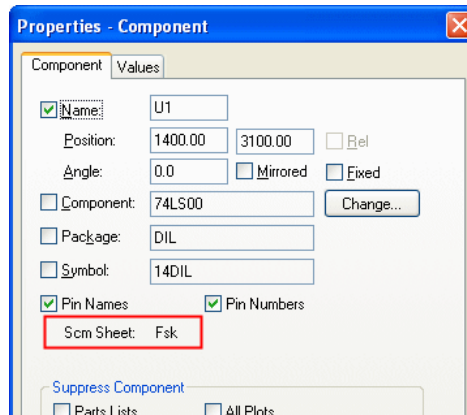
### Setting the Sheet Names to PCB option

From the **Settings** menu and **Preferences**, there is a new check box, **Sheet Names to PCB**, under the **General** tab and **Integrity Check**. When selected, this enables **Translate to PCB** to set initial sheet names on components as they are added to the PCB, and enables **Integrity Check** to synchronise sheet names from Schematic to PCB.



### Properties

The **Properties** dialog for a selected component shows the sheet name.



Once the PCB has sheet names on the components, you can use these in the following options:

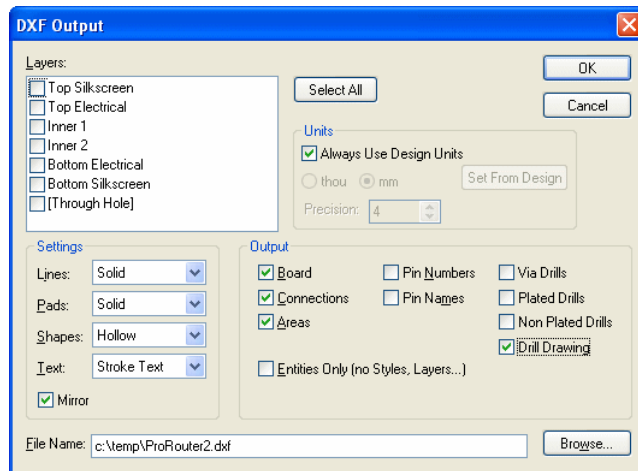
**Component Bin:** from the context menu and using **Select All On Same Sheet** by choosing from the dialog showing a list of sheets, or **Select All On Sheet xxx**, if the component already selected that is on a sheet, to select all the components that originate from the same schematic sheet.

In **Select Mode:** use **Select All From Sheet from the context menu**, this is the same as **Select All On Same Sheet**, above.

In **Autoplace** using **Match to Area Names**: by choosing to place only those components originating from a particular sheet. Use the **Sheet** button to enable this. However, it will consider components originating from the named Schematic sheet. This option is only enabled if the PCB design has sheet information from the Schematic sheets in the Project. The named area must use the same name as the sheet for this to work.

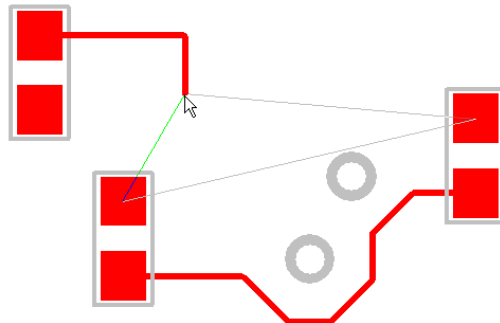
## DXF Output Improvements (PCB)

From the **DXF** output option on the Outputs menu, you can now **Mirror** the output or output the **Drill Drawing** ident's. These are both selected from the dialog using check box selections.



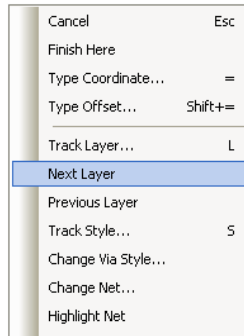
## Show Nearest Node On Net (PCB)

There is a new PCB option on context menu when adding a track or sketching, called **Show Nearest Node on Net**. Use this option to show a line to the nearest item on the net to the end of the track being added. The option is also available on the **PCB Tracks** tab in the **Preferences** dialog. Autoroute will route this line (if not editing a connection).



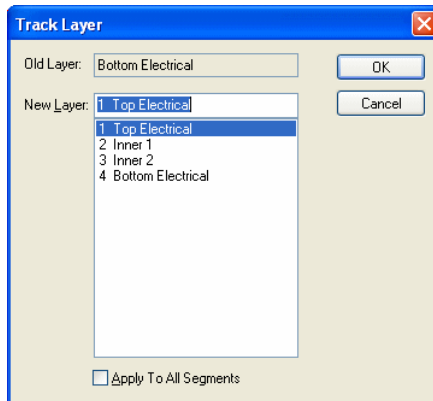
## Next/Previous Layer Command (PCB)

Two new commands are available for cycling through available layers during most interactive editing of shapes and tracks. These are available on the context menu as **Next Layer** and **Previous Layer**. These commands can also be assigned to shortcut keys using the **Customise** dialog.



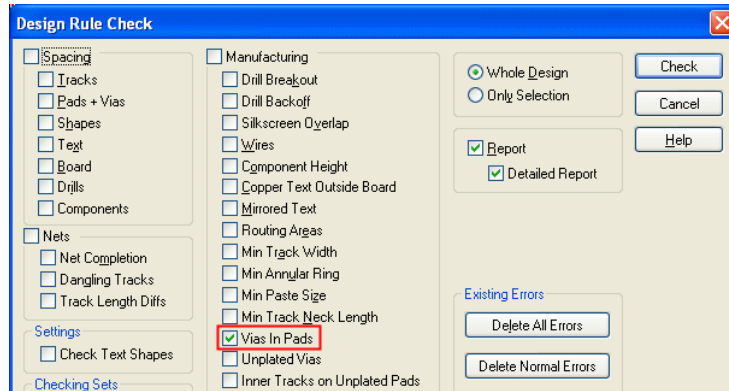
## Change Layer option (PCB)

The **Layer** drop-down list, is now 'pre-dropped' like the **Change Style** dialog. This new style dialog also auto-numbers the electrical layers to make it easy to access by key. If there is only one possible layer swap, e.g. from top to bottom layers, it will not display this dialog. So typing '4', will select the 4<sup>th</sup> electrical layer.



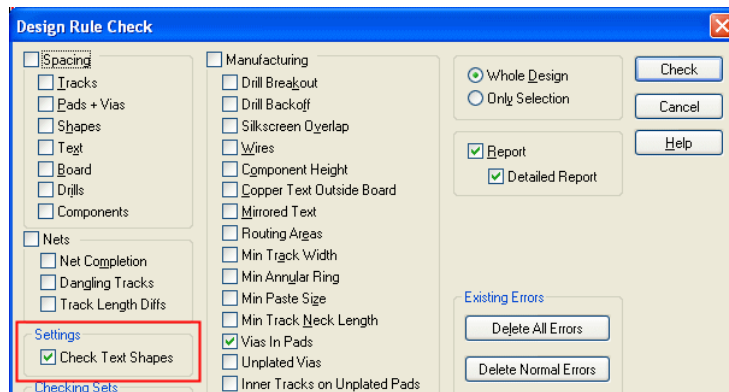
## DRC for 'Vias in Pads' (PCB)

An additional manufacturing check on the **DRC** dialog will flag same-net pad-to-via errors if a via touches a pad (no spacing is applied, the via must touch pad to cause an error).



## DRC 'Text shapes' (PCB)

When checking text (either with **Spacing Text** check or **Manufacturing Silkscreen** check selected), with the **Check Text Shapes** option selected, it will force the check of the actual shapes/strokes of the text characters instead of just the bounding box. With it left unchecked, the bounding box is used.



### DRC Silkscreen information (PCB)

When an error for silkscreen text/shape over pad/via, more information is now included in the error description, such as text or shape and pad name.

For example:

```
Via to Silkscreen Error (V-S) at (10135 11350) on Layer "Top Silkscreen". Silkscreen text 'IC2' overlaps via
```

```
Pad to Silkscreen Error (P-S) at (6325.0000 8250.0000) on Layer "Top Silkscreen". Silkscreen text 'R1R 0.063W...' overlaps pad.
```

```
Pad to Silkscreen Error (P-S) at (13777.5000 12697.5000) on Layer "Top Silkscreen". Silkscreen text 'U32' overlaps pad.
```

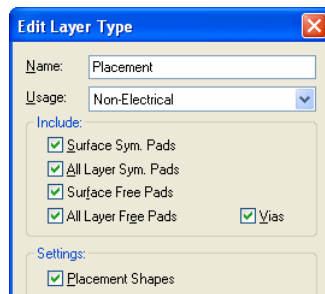
### DRC Component-to-Component (PCB)

It is now possible to define and check the space between components. You can check (using the **Spacing Components** option in **DRC**) the space between components using the placement extents of a component and its bounding box (the bounding box includes all shapes and pads of a component). The required minimum distance between components is defined in the **Design Technology** and **Rules**.

There are two methods to define the 'bounding' box. The first is to let Easy-PC define it for you using the maximum extents of the shapes and pads of the component, or to define the bounding box yourself using a placement area.

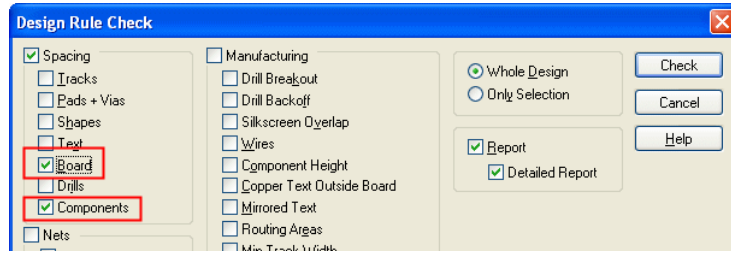
#### Defining a placement area

Any shape on the top or bottom layer can be defined as a **Placement** area by putting it on a placement layer. You can set a layer to be a placement layer by using the **Layer Types** dialog. A new check box under **Layer Types** for **Placement Shapes** should be checked to enable this feature.



Once checked, when the **DRC** option for **Spacing** and **Components** is selected, the shapes defined on that specific layer will then be checked. If a component does not have a placement area defined on one of the placement layers (top or bottom), the bounding box of the component extents will be used.

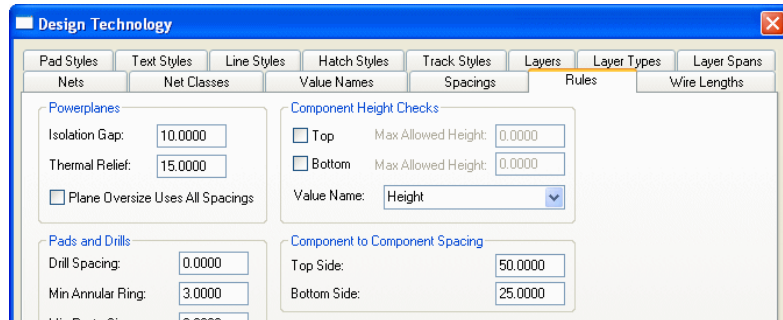
## DRC dialog



If you also check **Board** under **Spacings** on **DRC** dialog, it will also check for components outside the board outline.

## Spacing dialog

The Component Spacings are defined on the **Rules** page of the **Design Technology** dialog. You can specify different **Component to Component Spacing** values for the **Top Side** and **Bottom Side**.



## Sign-Off Checks (PCB)

Two sets of DRC settings are now stored in the design (these were stored in your system registry). One of these two sets is the **Sign-off** check set, and is used at the start of the **Plotting** process to check your design before producing outputs.

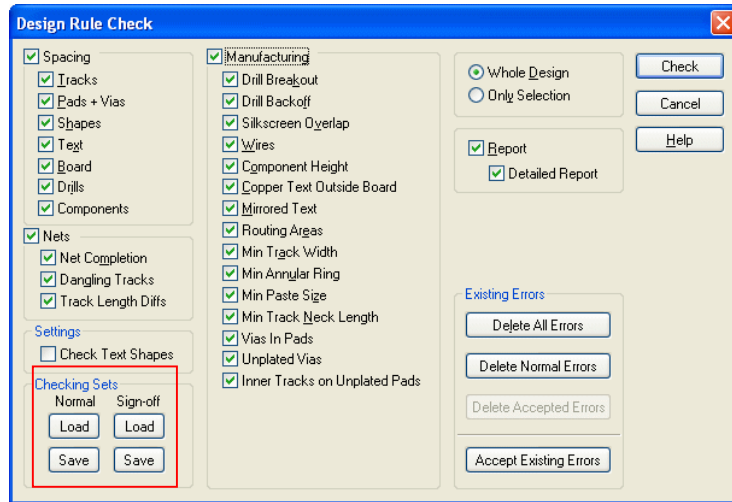
The Sign-Off checks are used as a final verification that your design conforms to the rules you've specified. It highlights important information, errors and warnings the moment before you send off your plots to be manufactured. It will ensure you remember to run the DRC checks and will highlight the errors to you at a critical time in the design process.

## DRC option

There are two sets of DRC settings stored in the Design. You can use the **Load** and **Save** buttons in the lower left corner of the dialog to load either of the two sets into the dialog, and save the current dialog settings away to either set.

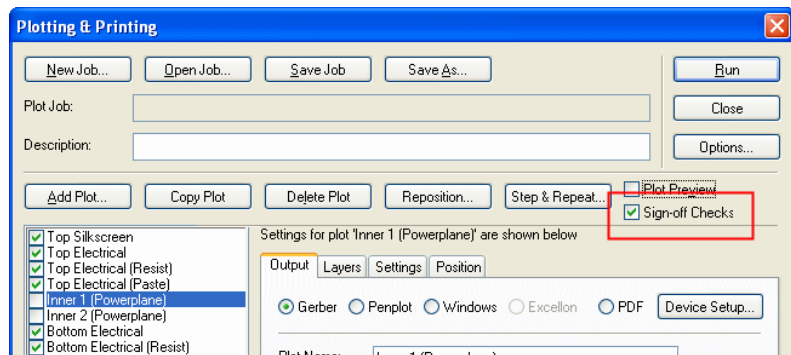
- **Normal** - The first is the 'normal' set that is used when running **Design Rule Check**.
- **Sign-off** - The second is a 'sign-off' set that is used when running the Sign-off Checks prior to producing plots.

**Note:** When you click the **Run** button on the **Design Rule Check** dialog to start the checking, whatever settings are present in the dialog at that time will be saved to the 'normal' set.

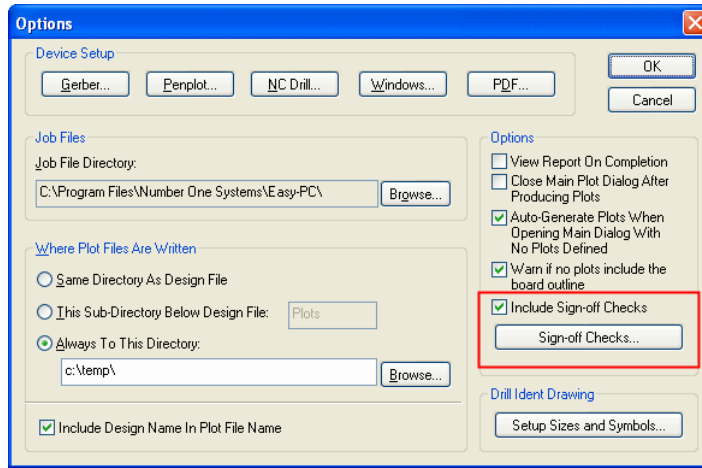


### Plotting dialog

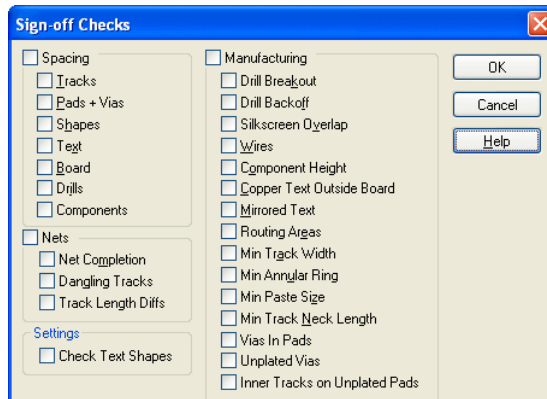
An additional check box on the **Plotting Options** dialog is used to enable the Sign-Off Checks when the plots are generated.



There is an additional button on the **Options** dialog to setup the **Sign-Off** checks. This is a reduced version of DRC dialog showing only settings, and main DRC dialog has controls for saving and loading both sets of settings.

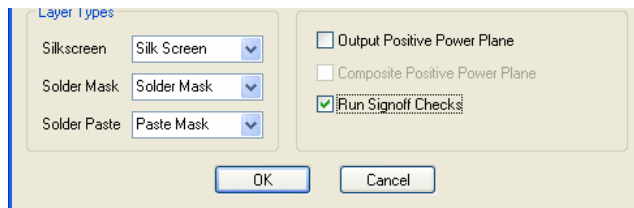


The **Sign-Off Checks** button displays a reduced set of DRC options to enable you to select the rules to check.



### ODB++ Dialog

The ODB++ dialog also has the Sign-off checks check button to enable to run the rules.



### 'Accepted' DRC Errors (PCB)

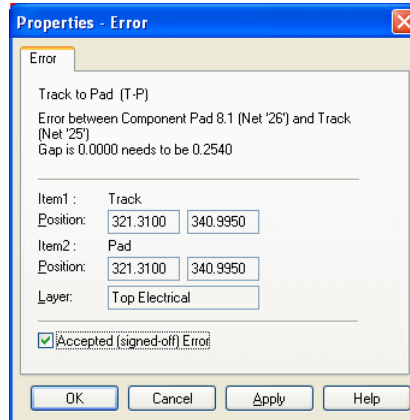
Error markers in Easy-PC can now be 'accepted' (as 'signed-off'). You may find that DRC finds errors which you are prepared to accept (or 'sign off'). This might happen for example if you have a set of 'finger' pads touching the edge of the board. DRC will flag these with 'Pad to Board' errors, which you can then set as 'Accepted' so that there is a positive indication that you have examined these errors and agree that they are not faults that need to be fixed. Once you have resolved all the faults in the design except those that

you are prepared to sign off, you can use the **Accept Existing Errors** button to mark all remaining errors as accepted. Alternatively an error marker can have its Accepted property toggled using the **Properties** dialog or by right clicking an error and selecting the option from the context menu.

Accepted error markers are only deleted by you, either by deleting them interactively in the design, or using the **Delete Accepted Errors** button on the DRC dialog. When you run Design Rule Check, any existing accepted errors are left alone.

### Accepting errors

Individual errors can be checked as **Accepted** on **Properties**.



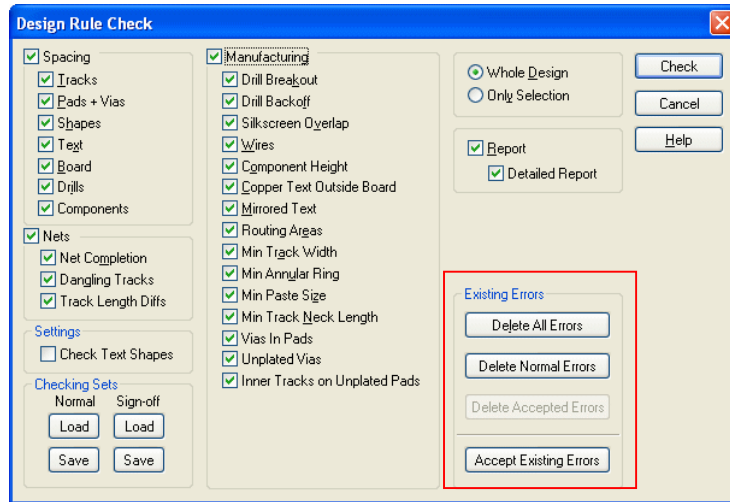
### DRC dialog

The **DRC** dialog has some new buttons to help with clearing and accepting existing errors. These are listed under the **Existing Errors** section:

The buttons in this section are used to deal with existing error markers in the design.

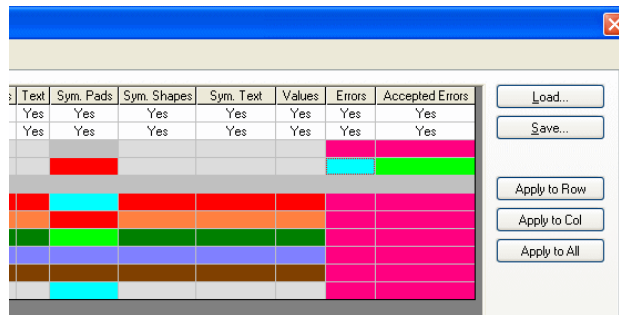
- **Delete All Errors** : removes all existing error markers from the design.
- **Delete Normal Errors** - removes all existing error markers that are not set as 'Accepted' errors.
- **Delete Accepted Errors** - removes only existing error markers that are set as 'Accepted' errors.
- **Accept Existing Errors** - use this button to mark all existing error markers as 'Accepted'.

On running **DRC**, the option will only delete 'normal' errors and will leave **Accepted** errors in-place.



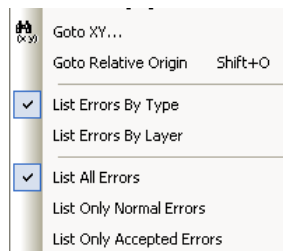
### Display dialog

There are different colours and visibility settings on the **View** menu and **Display** dialog under **Layers and Layer Spans** and **Accepted Errors**.



### Find Error types

From the **Goto** bar, on selection of Errors, you can now list errors by type. From the context menu within the Goto bar, you can select **List All Errors**, **List Only Normal Errors** and **List Only Accepted Errors**.

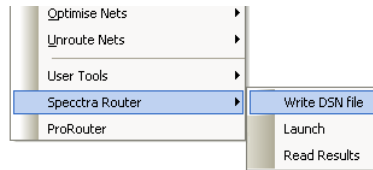


When showing all errors, they are separated into two lists, normal and Accepted so they are easily visible.

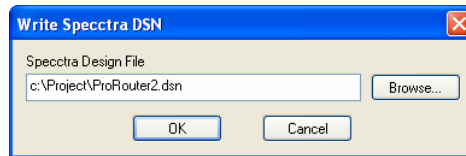
## Specetra Export menu option (PCB) (Existing Cost Option)

There is a new menu option to simply write DSN file without attempting to launch the Specetra product. This is only available for users who have purchased the Specetra interface cost option.

From the **Tools** menu, select **Specetra Router** > and **Write DSN File** from the sub menu.



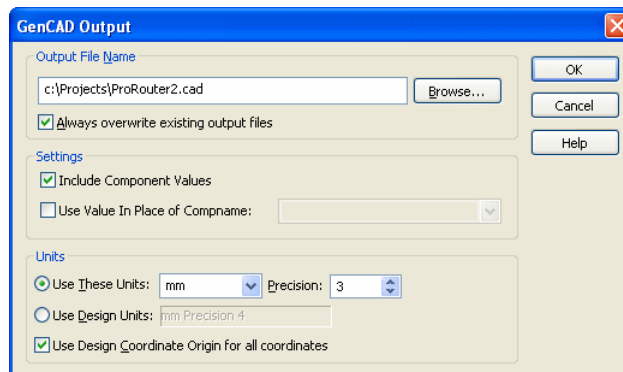
A small dialog is displayed in which to enter the folder where the DSN file is to be written.



## GenCAD Output (PCB) (New Cost Option)

A new cost option is available in Easy-PC 13 to output GenCAD format files.

The GenCAD format is used to transfer physical design information to other software, usually for more advanced test or manufacturing purposes. The Easy-PC GenCAD output option conforms to the GenCAD V1.4 specification.



Use the dialog to select the output features you need. The options selected will depend on the requirements of the software into which you are going to read the resulting GenCAD file. Please check with the person requesting this format file before outputting it.

### **Micro Library Update (Updated Cost Option)**

An additional 500 micro controllers have been added to the existing Micro-controller library. This upgrade is supplied free of charge when you upgrade to version 13 or as a chargeable option if you decide not to upgrade to version 13..

A full list is supplied online in the Micro-Controller datasheet.  
[www.numberone.com/downloads/datasheets/MicroLibrary2.pdf](http://www.numberone.com/downloads/datasheets/MicroLibrary2.pdf)

### **Pro Library Update (Updated Cost Option)**

Over 13,000 new Components and associated symbols/footprints for this existing library have been added to version 13. Users who have also purchased this library can purchase the upgrade from our sales office.

A full list is supplied online in the Pro-Library datasheet.  
[www.numberone.com/downloads/datasheets/ProLibrary2.pdf](http://www.numberone.com/downloads/datasheets/ProLibrary2.pdf)

### **Connector Library (New Cost Option)**

A new library of over 22,000 new Connector components has been added to version 13.

A full list is supplied online in the Connector Library datasheet.  
[www.numberone.com/downloads/datasheets/ConnectorLibrary.pdf](http://www.numberone.com/downloads/datasheets/ConnectorLibrary.pdf)