

Easy-PC

V14.0 Supplement

Copyright ©1987-2010 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: 16/07/10 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
Data Files Location.....	5
Running Easy-PC For Windows 14.0	5
CHAPTER 2. NEW FEATURES IN EASY-PC V14.0	7
Introduction.....	7
Projects	7
Change Component (Both).....	7
Output Reports (Both).....	8
Add Component (Both).....	9
Cancel Move during Paste (Both).....	10
Design Technology and Value Names (Both).....	10
Report Destination (Both)	11
Grids (Both)	12
Multiple Grids	12
Change to the Grid Snap context menu	15
Polar (radial) Grids	16
Library Manager (Both).....	17
Component Edit Properties (Both).....	18
Save To Library (Both).....	18
Save Selected Items To Library (Both).....	19
‘Move To’ in Library Dialog (Both).....	21
Change to Rotation Angle (Both)	21
Hierarchical Schematics (SCM)	22
Getting started with Hierarchy	22
Using Hierarchy.....	29
Drill Ident Output (PCB).....	34
Dimension Text Improvements (PCB).....	34
View Board (PCB).....	35
New Easy-Router replacement – Trace-Router (PCB).....	35
Pull Tight Manual Routing Mode (PCB).....	37
Pull Tight Smoother (PCB).....	38
Double Click to Start Track Action (PCB).....	39
Design Rule Check Improvements (PCB).....	39
Design Revision Analyser (PCB)	40
Display of Item True Width (PCB).....	40
Separate Tool Numbers (PCB).....	42
Separate Slots Selection for NC Drill files (PCB).....	42
OrCAD Netlist ECO Import to PCB (PCB).....	43
Micro Library 3 Update (Updated Cost Option)	44
Pro Library 3 Update (Updated Cost Option).....	44
Connector Library 2 Update (Updated Cost Option).....	44

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 V13.0 can install V14.0 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 14.0** 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 an extra step in the **Setup** installation wizard that asks you where you want to place data files (for example, libraries, technology files, etc). The default is always to use the common documents folder, "Users\Public\Documents\Easy-PC" on Vista and Windows 7, "Documents and Settings\All Users\Shared Documents" on XP etc. if you are installing for All Users, or into your own Documents folder if installing for current user only.

Running Easy-PC For Windows 14.0

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.

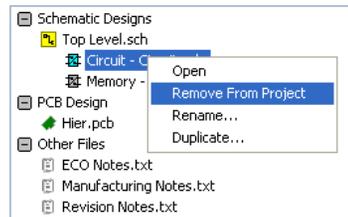
Chapter 2. New Features in Easy-PC V14.0

Introduction

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

Projects

For a selected Project, there are new items on the context menu, there are new entries for **Rename** and **Duplicate**.



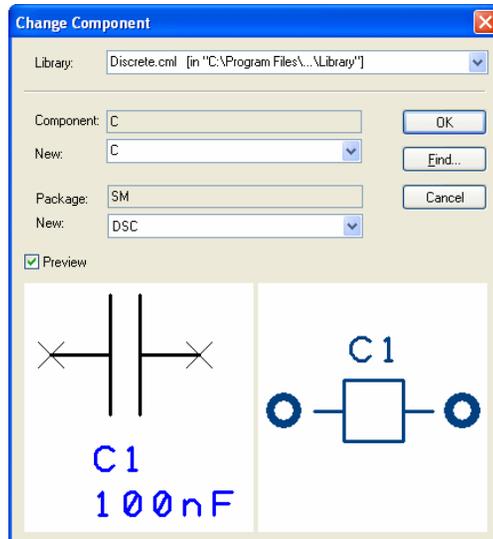
Rename : changes the name of the selected project item, then either creates a copy of the original file under the new name, or just renames the existing file.

Duplicate : makes a copy of the selected project item under a new name. This is the same as copying the underlying file in Explorer and adding the new file back into the project.

These new items are only applicable to items under **Schematic Designs** in the project list.

Change Component (Both)

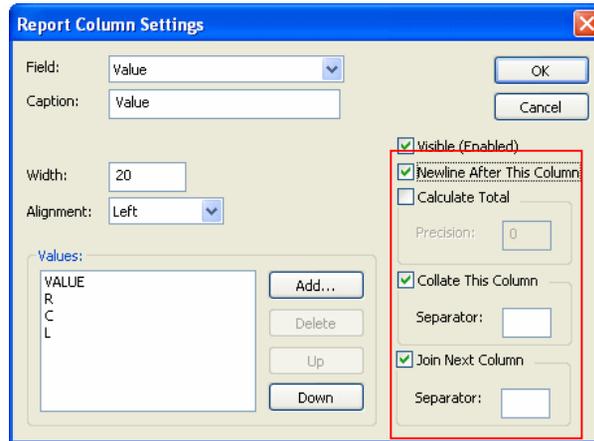
When using the **Change Component** option, the dialog now includes both SCM and PCB symbol previews of the selected component.



Output Reports (Both)

The **Reports Output** has been enhanced to provide extensions to the custom report mechanism to allow more complex reports to be generated:

- As well as components, it can now produce a lists of nets, component pads and vias. This uses an existing mechanism for components and has been extended to include the new items.
- Additional controls on fields are available for joining fields together, collating fields, and forcing newlines.



Collating Columns

When generating a parts list (Bill of Materials) the usual setup will produce one row in the report for each unique component + value combination. If you add the Name to the list as well, this would normally break the report into separate rows for every component instance because the component name changes for each row. By ticking the Collate checkbox for the Name field, this will cause the component names to be collected together to output as a single entry in the report:

Component	Value	Name	Qty
10-32-1023	6K8	R19	1
10-32-1063	68K	R18	1
10uF_16V_10%_Tant_Case_C	1uF	C93	1
10uF_16V_10%_Tant_Case_C	10uF	C69 C52 C6 C98 C64 C50 C52 C59 C51 C49	11
12CTQ035		SK2 SK12	2
12CTQ040		IR1	1
16TTS12	LM2940CT5	U9	1

Joining Columns

The Join Next Column checkbox allows you to combine adjacent fields in the report definition into a single column in the report. For example when producing a list of component pad positions, you can use this switch to combine the component name and pin name fields into one, by checking the Join box for the Name column and specifying a dot as the separator:

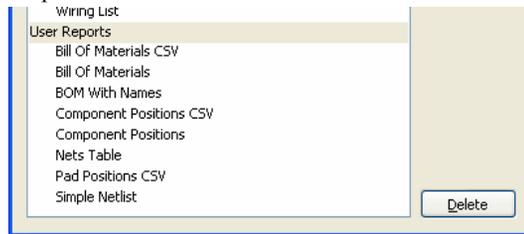
```
Name, Position X, Position Y
BR1.1, 15700.0000, 21900.0000
BR1.2, 15700.0000, 21900.0000
BR1.3, 15700.0000, 21900.0000
BR1.4, 15700.0000, 21900.0000
C1.1, 13750.0000, 22550.0000
C1.2, 13750.0000, 22550.0000
```

Newline

Although reports are generally written in a 'table' form, with columns across the report and one row per 'item', you can use the Newline switch to force a new line into the report after a particular field. This could be used for example to produce a report where each field is on a separate row, such as a netlist that outputs the name of the net followed by each pad in the net on a separate row.

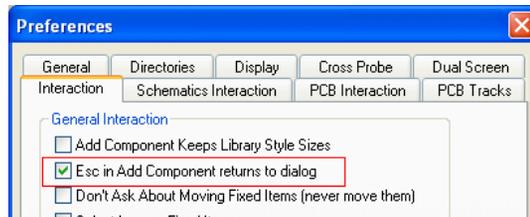
There are also new example report templates provided with the standard installation for writing:

- BOM With Names
- Pad Positions in CSV format
- A simple Netlist format

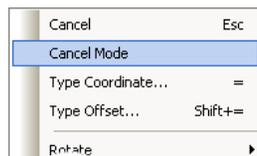


Add Component (Both)

When placing components, when pressing **Cancel**, by default, the option will go back to the **Add Component** dialog. There is a new setting on the **Preferences** dialog under **Interaction** that controls this, unchecking the box there will cause Easy-PC to revert to its original action of <Esc> meaning cancelling out of the mode completely.

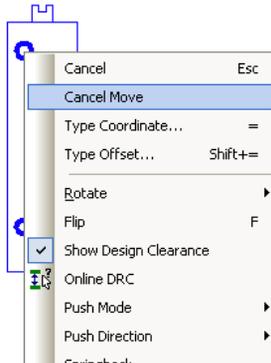


When placing components, the **Cancel** option will, by default, return to the **Add Component** dialog.



Cancel Move during Paste (Both)

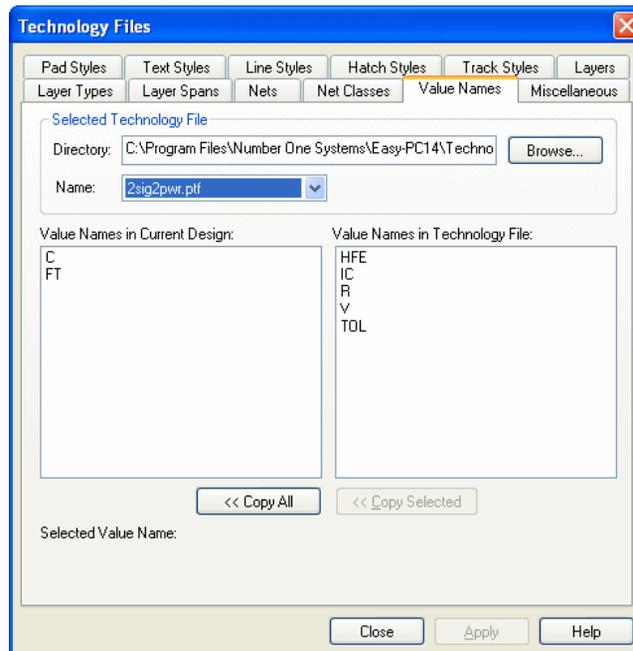
When performing a paste of design data, you will now be able to use **Cancel Move** from the context menu to paste the item in its original position. If enabled, Online DRC will be performed at this position. If the original position is outside the working area an error message will be given and you will be able to continue moving it or **Cancel** the paste completely.



Design Technology and Value Names (Both)

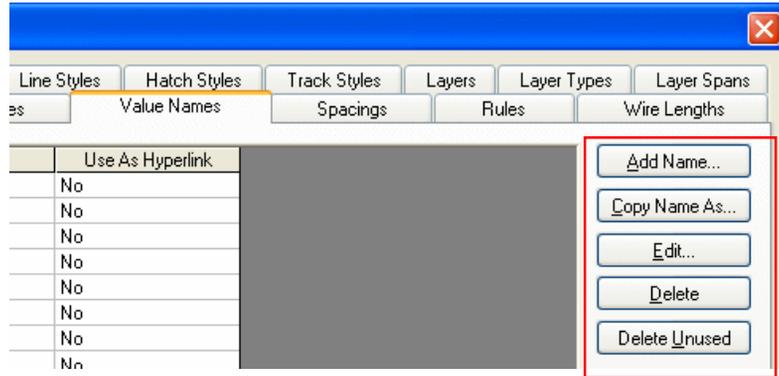
Value Names in the Technology file

Previously, it was not possible to transfer or update Value Names from a technology file to a design, the Technology Files (technology manager) dialog had no tab for **Value Names**. This has now been added to V14 and will work the same way as for **Styles**.



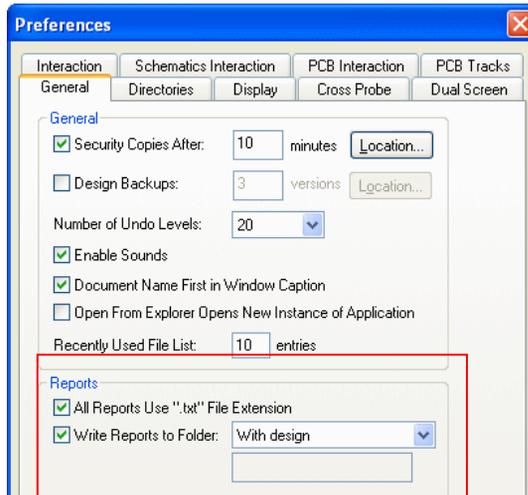
Editing Value Names in the Technology file

Previously, the **Value Names** page of **Technology** dialog had no means of changing the name of a Value, nor any ability to add, edit or delete **Value Names**. These have now been added to V14 and work the same way as for **Styles**.



Report Destination (Both)

Additional settings for report 'destination' in **Preferences**, allowing you to choose whether to write reports to the same folder as the design file (the default/original method), to a folder below the design file, or to a specific folder.

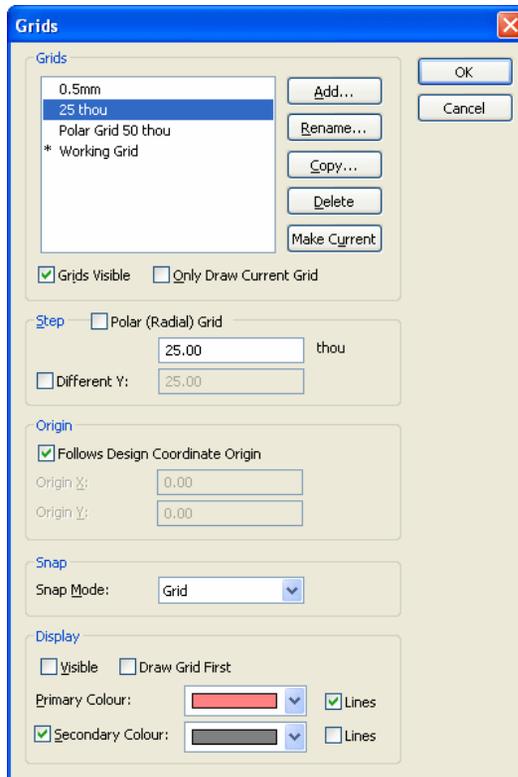


Grids (Both)

The **Grids** dialog has been significantly changed to make major improvements. In summary, you can now add and use your own named grids each with its own display settings, add and use polar (radial) grids, and have your own grids displayed in the Grid Snap context menu.

Multiple Grids

You can now add and use multiple grids, these grids can also have user defined names. Once added, the named grids are available for use on the grid snap menu (see below). These grids can be used for placing items or simply used as displayable grids for reference. To accommodate these changes, the **Grids** dialog accessible from the **Settings** menu has been redesigned and improved. This dialog is divided into several sections, each of which is described below.



Grids

The first section of the dialog shows you a list of all the Grids currently defined in the design. From here you can carry out actions such as adding new grids or changing the name of an existing grid. Apart from adding a new grid, all other actions on this dialog will act upon the grid that is currently selected in this list.

Add : add a new grid to the list. You will be prompted to supply a new name for the grid, which must be different from the names of any grids that exist already.

Rename : change the name of the selected grid.

Copy : make a copy of the selected grid under a new name, copying across all the settings from the selected grid.

Delete : remove the selected grid from the design. Note that there always needs to be at least one grid, so the application will not let you delete the last grid from the list.

Make Current : sets the selected grid as the current working grid to use for placing items. The 'current' grid is marked in the list with an asterisk. As a shortcut to changing the current grid you can simply double-click on the required grid in the list. This will make that grid the current grid, and close the dialog.

Grids Visible : this is a global switch that toggles the display of grids on and off. This is the same as using the **Toggle Screen Grid** toolbar button or command. Note that each grid also has a Visible checkbox, so that you can choose which of your grids should be drawn when grids are set as Visible.

Only Draw Current Grid : when drawing grids, normally all grids which have their Visible checkbox ticked are drawn. By checking the Only Draw Current Grid box, you can choose to draw only one grid, the grid that is set as the current working grid. Using it this way will mean that as you flip from one grid to another, the display changes to draw only the grid that is current (active).

Step

The first box is used to define the working grid step. Type the required size in the current design units.

The screenshot shows a dialog box with the following elements:

- A label "Step" with a blue underline.
- A checkbox labeled "Polar (Radial) Grid" which is currently unchecked.
- A text input field containing the value "25.00" followed by the unit "thou".
- A checkbox labeled "Different Y:" which is checked.
- A text input field containing the value "25.00".

Different Y : check this box to use different grid step in X and Y axes. By default, the X and Y steps are the same.

Polar Grid : check this box to define a polar (radial) grid. When you check this box, some different settings appear on the dialog:

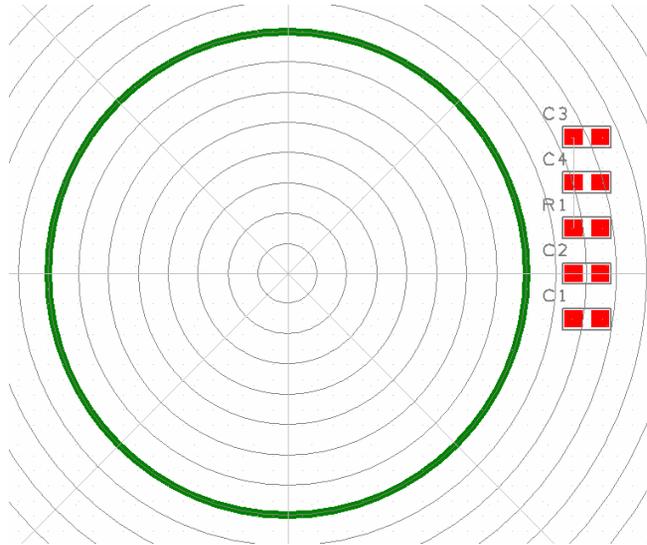
The screenshot shows the dialog box with the following elements:

- A label "Step" with a blue underline.
- A checkbox labeled "Polar (Radial) Grid" which is checked.
- A text input field labeled "Concentric Step:" containing the value "100.00" followed by the unit "thou".
- A spin box labeled "Divide Circle Into:" containing the value "8" followed by the unit "steps".

Concentric Step : this is the distance between each 'ring' of the grid.

Divide Circle Into : this is how many 'slices' the grid should be divided into.

Here you can see an example of a Polar grid, in this case divided into 8 segments:



Origin

This section of the dialog allows you to specify the origin or 'starting point' of the grid.



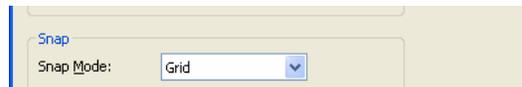
A Polar grid will be centred at the origin position, and a normal 'step' grid will have its first point on the origin and step outwards from there in each direction.

Follows Design Coordinate Origin : check this box to have the grid start from wherever the design System Coordinate Origin is placed.

Origin X and Y : here you can specify a particular location for the grid origin when it is not following the coordinate origin.

Snap

Snap Mode is used to further define the grid step that is actually used in interactive operations such as Place. Choose a value from the drop down list as follows:



If you choose a setting other than 'Grid', each step of the grid is divided according to the setting you select. Snap Mode is not available for Polar grids.

Display

This section allows you to control how each grid is displayed. As with the other settings, each grid has its own display values.



Visible : check this box to make this grid visible. Note that it will still only be drawn if the main **Grids Visible** checkbox is also checked.

Draw Grid First : check this box to draw the grid before drawing actual design items. This has the effect of drawing the grid 'underneath' everything else.

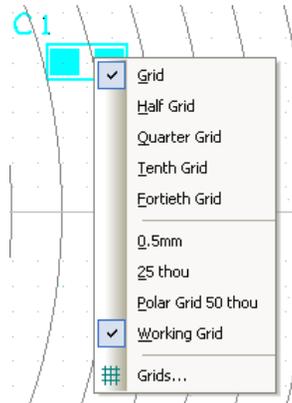
Primary Colour is used for every 10th grid point of a step grid, or for the radial 'arms' of a Polar grid. Check the **Lines** box to have the grid drawn as lines, otherwise it is drawn as dots

Secondary Colour is used for all other grid step points. As you use Unzoom to increase the amount of the design that is displayed you will reach a level where the secondary grid points are too close together and they will no longer be displayed. The same is true for the Primary points for a step grid if you carry on with zoom out.

For a Polar grid, if you uncheck **Primary Lines**, the grid is drawn with a single dot at each place where the rings and the radial arms intersect.

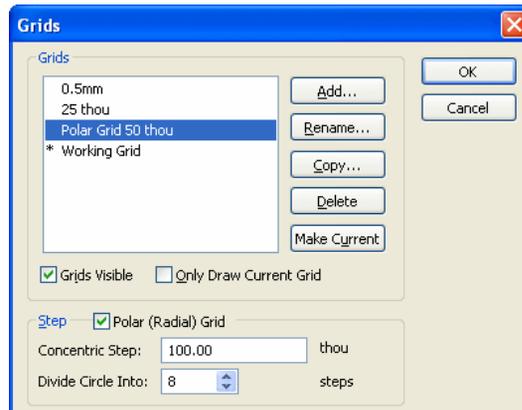
Change to the Grid Snap context menu

There are additions to 'grid snap menu' allowing you to choose current grid, and access Grids dialog. These can be changed interactively when editing or placing items in the design. The default shortcut for this is the <H> key. A small menu will be displayed which allows you to select the Snap Mode (for a step grid), to change the current working grid to a different grid (if more than one grid is defined), or to access the Grids dialog.

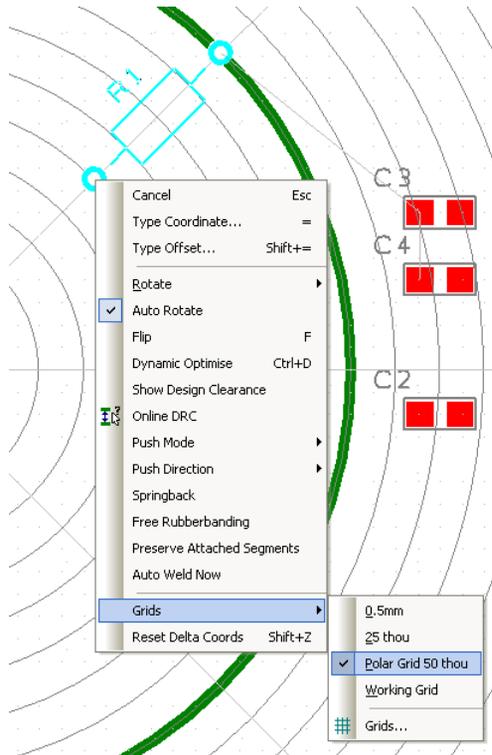


Polar (radial) Grids

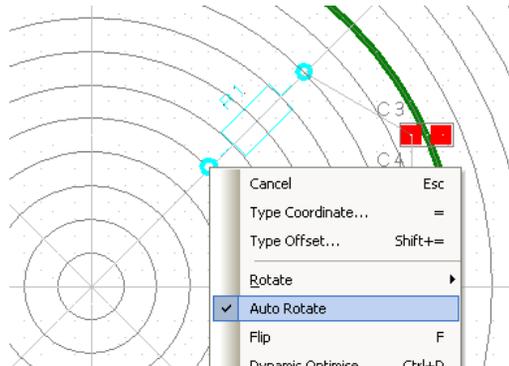
Once a Polar (radial) grid has been defined in the Grid dialog (like the one shown below), you can then place components on this grid using interactive placement options.



With the Polar grid active, you can use this during placement by selecting your Polar grid from the context menu and **Grids>**. The grid selected must be a polar grid as defined in the **Step** option of the **Grids** dialog.



Once this grid has been selected, you can then access the Auto-rotate option from the context menu. This will allow components to be rotated on-grid to a grid rotation with an accuracy of a 1/100th of a degree.



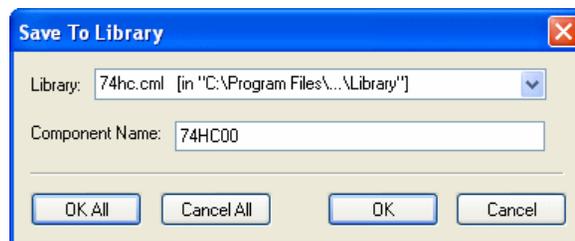
The auto-rotate settings can be predefined from the **Preferences** menu and **PCB Interaction**.



From this option, you can select which items the auto-rotate feature applies to by checking the box next to the option.

Library Manager (Both)

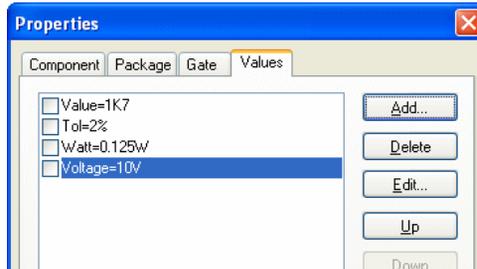
When using the **Library Manager** to copy or move a multiple selection, the **Save To Library** dialog now provides two additional buttons; **OK All** and **Cancel All**, which act on the selection as a whole.



Component Edit Properties (Both)

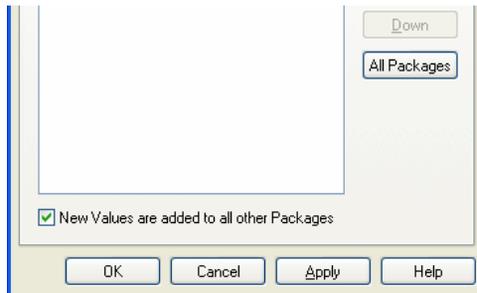
Values Dialog

The **Values** dialog in the Component editor has been changed so that it is now a page on the **Properties** dialog, this will save time swapping between the packages to get to the Values fields.



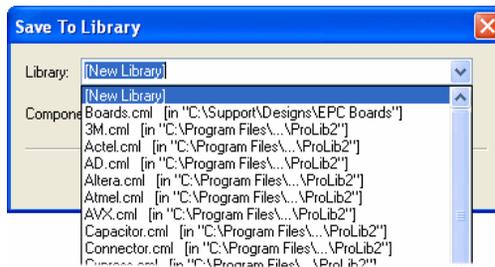
Replication of Values to all Packages

There are two new controls on the **Values** page to handle replication of Values to all packages in the component. The first is a button that causes any new Value added to be added to existing packages (the **All Packages** button). The second is a check box (**New Value added are added to all other Packages**) at the bottom of the dialog. When ticked causes any new Value added to be added to all packages within that Component.

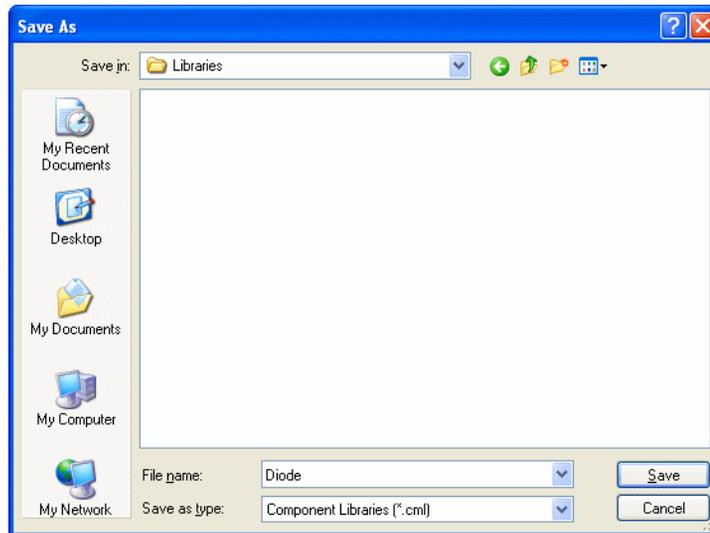


Save To Library (Both)

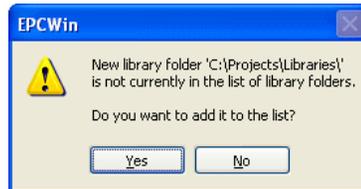
When using **Save To Library** (components or symbols) from the **File** menu, you can now select the special entry **[New Library]** from the drop down list of library files. This allows you to create a new library file to save the item into.



A **Save As** dialog is displayed, into which you type the new library name.



If the library path is a folder currently not in the list of libraries folder, Easy-PC will ask if you would like to add it. By adding it, it will then be included in the library search used when using the **Add Component** option etc.



Save Selected Items To Library (Both)

This option allows you to easily copy sets of components and/or symbols to library files. It can be used from the Components page of the Library dialog to make a self-contained copy of selected library items. For example if you want to make changes to items in our master libraries it is best to copy them to your own 'user' libraries first, to avoid any issues with overwriting your changes if our master libraries are upgraded in the future.

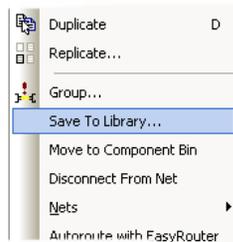
It can also be used to make a set of libraries containing some or all of the components and symbols in a PCB or Schematic design file. This can be useful for example if you have an archived design for which the libraries are no longer available or if you wish to create a project based library. A complete set of the three types of libraries can be created in one single step.

Copying Items In Library Manager

From the Components page of the Library Manager dialog, select the component or components you wish to copy, then click the **Save To Lib** button.

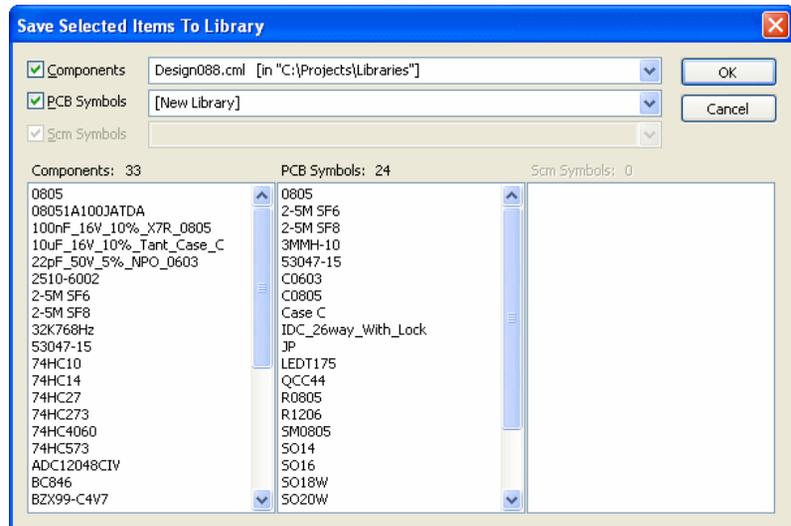
Copying Items From A Design

To copy library items out of a PCB or Schematic design, simply select the component or components you wish to copy to the library, then right-click and choose **Save To Library** from the context menu.



Using the dialog

The top part of the dialog shows the three types of library, and a dropdown list where you can choose the target library to which the relevant library items should be written. The checkbox alongside each of the library types controls whether or not that type of library item will be written. In the dropdown list of libraries, there is also an entry [New Library] which if selected will prompt you to create a new library before copying starts.



Underneath this are three lists, one for each type of library item. In the Components list will be shown all the components that were selected in the design or on the Library Manager dialog, depending on where you started from. The PCB and SCM symbol lists will show all the symbols that are referenced by the components. If you are saving from a design file, only one of the two symbol lists will be filled as a design only contains one type of symbol.

Selecting items in the lists

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

This feature can be used to 'explore' the relationship between the components and symbols you are going to save. This will not affect what is actually written to the target libraries, that is only controlled by the checkboxes at the top.

Starting the copy

To initiate the actual copying, click the OK button. If you choose to write to a new library for any or all of the library item types, you will be prompted to specify a folder and filename for the new library.

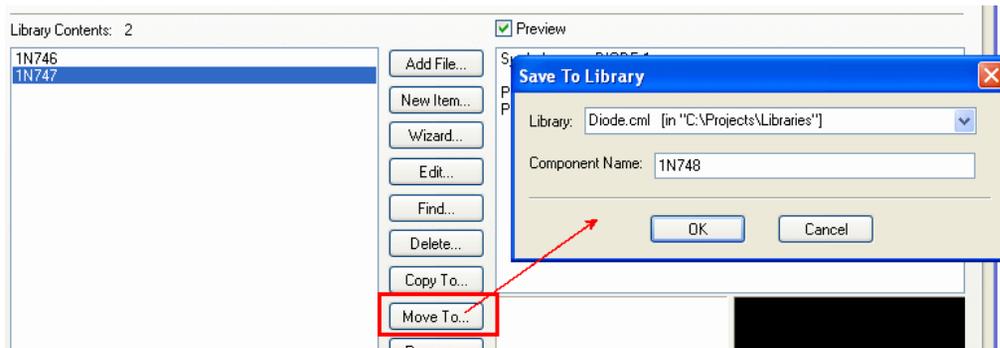
If the target libraries already exist, then they will be checked to see if any of the items to be copied already exist, and you will be given the choice of overwriting the existing items, skipping those that exist, or cancelling the whole copy operation.

'Move To' in Library Dialog (Both)

There is a new button in **Library Manager** dialog **Move To**. This does same as **Copy To** but deletes the items from the source library afterwards. When run, a confirmation dialog is displayed to warn you about the deletion afterwards.



Pressing the **Move To** button gives you the **Save To Library** dialog.



Change to Rotation Angle (Both)

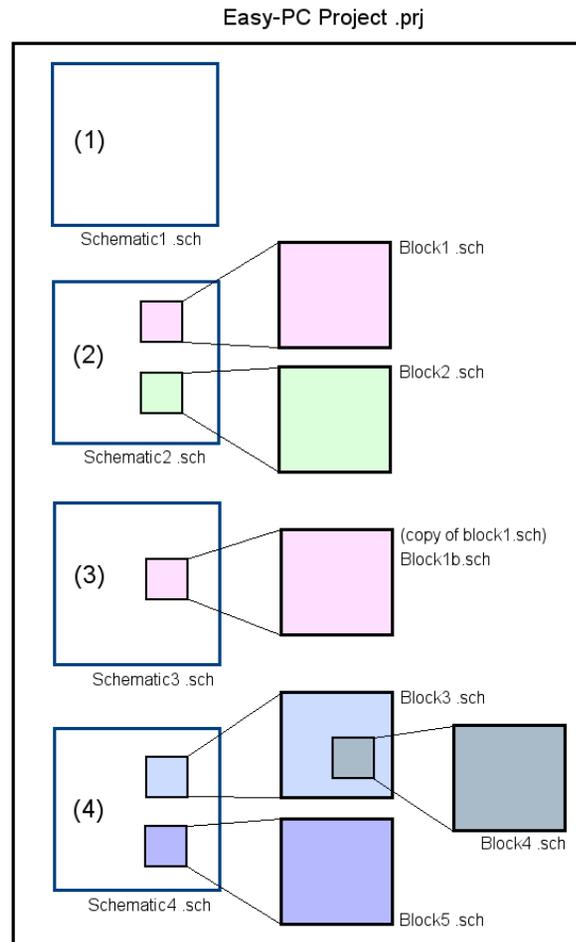
Angular resolution (precision) has increased from 1 to 2. This means items like components can now be placed to an accuracy of 0.01 degree (1/100th degree). When using this precision, care must be taken to ensure your later processes, like assembly machines, can handle it.

Hierarchical Schematics (SCM)

Getting started with Hierarchy

Version 14 introduces the ability to add multiple hierarchical schematic sheets to a project. From within a Project file, you can now have multiple sheets and blocks, these blocks link to lower level circuitry.

To use hierarchy, the designs must be contained within a **Project** file.

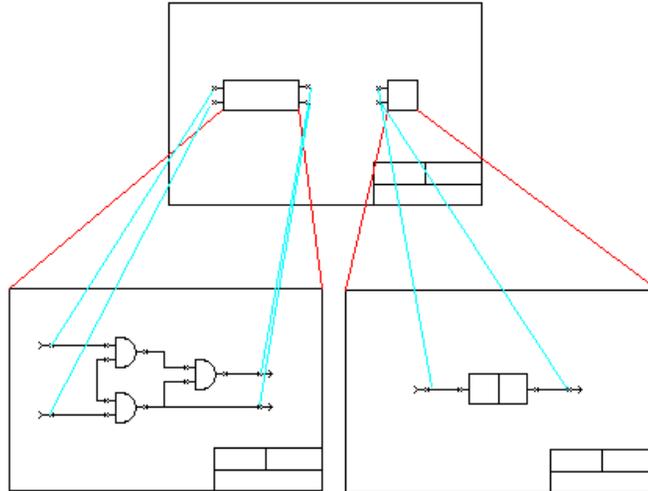


Hierarchical Building Blocks

The diagram above illustrates how various scenarios are possible when using hierarchy, all within the same Project file. A design sheet can contain multiple blocks (2) and multiple hierarchical sheets. In (3) an existing block can be copied and effectively used again, it is renamed though. Schematic (4) shows multiple blocks and a block inside a block (a nested block).

Overview

Schematic design sheets in a project can have a hierarchical structure. This is different to using multiple sheets in a project at the same level in that you can add a block on a schematic sheet and push into that block to reveal the block contents on another schematic design. For example, this could be detail that can be covered by an overview block, say PSU or MEMORY etc. but for which you can show the lower level detail on a separate sheet. To use hierarchy, the designs must be contained within a Project file.



The above shows a top level schematic design with two blocks. Under this, each block's contents is a separate schematic design within the Project.

There are two reasons for using a hierarchical schematic design structure.

- To allow a schematic design to be structured, so that it can be broken down into its functional elements. This promotes "Top Down" design; break down a problem into functional elements, then define the detail of each element.
- To facilitate the re-use of commonly used circuit elements. This promotes "Bottom Up" design; use predefined elements to build a solution.

In practice, you may want to use a combination of both "Top Down" and "Bottom Up" design.

What is a block?

In Easy-PC a block consists of two parts. The first part is a block symbol, inserted into a schematic design to represent the block at the higher level. The second part is a complete schematic design containing the design elements that define the contents of the block, and this represents the implementation of the block at the lower level.

The two halves of the block can be defined separately, but are eventually linked within a project by using a common block name. The block symbol has a name, and a schematic design representing a block's contents can be allocated a name using the **Design Properties** dialog.

The block symbol can contain block pins that can have schematic connections attached to them to pass signals to a corresponding block port in the block's contents design. The block ports are single pin symbols that can be separately placed anywhere in the lower level design.

The link between a block symbol pin and its corresponding block port is set by both having the same port name.

Blocks are not copied to the PCB design, but exist to place a hierarchical structure on the schematic representation of the design. When translating the schematic into a PCB design a check will be made to ensure each block symbol pin and its corresponding block port are on the same net.

A hierarchical design is easier to navigate, because you can **Push** into the design representing the block contents, from it's block symbol, and **Pop** out of the block design, back to the block symbol.

In a project tree view, schematic designs that represent the contents of blocks will have a blue icon and display their block name. If their block symbol is also defined, they are shown as an indented branch below the design that contains the block symbol. This way the hierarchical structure of the design is clearly shown.

Adding Blocks

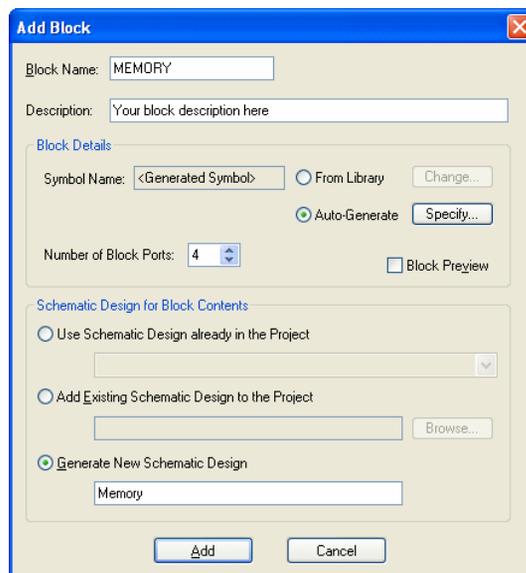
There are three ways of adding a block into a design.

1. Add Block

Use this option from the Add menu to insert a new block into the design. A dialog will be displayed for you to define the block name, set the number of block ports, choose the block symbol and supply the name of the block contents design.

Define what the block symbol will look like by either using a schematic symbol from the library, or generating a standard rectangular shape using user specified parameters.

The block contents design can be a new design or you can hook up to an existing design either already in the project or a design file in the project folder. The block symbol can be placed in the current design and the block design will be automatically added to the current project. Block ports will not be initially added to a new block design, but you can add them when editing the design by using **Add Block Port** when the port is needed.



2. **Cut To Block**

Use this if you want to take an existing schematic design and cut a section of it into a block. This option prompts you to draw an outline around the items to be pushed into a block, and across any connections that are to be passed down into the block. The connections are cut at the outline and terminated on a block symbol pin. The other half is pushed to the new block design and terminated on a corresponding block port.

The drawn outline and new block pins become the block symbol, and all data within it is moved into the new block design. The block symbol can now be replaced with a more suitable shaped symbol using the **Change Block Symbol** option.

The new block design will take a default file name based on a name stem defined with the **Block Defaults** dialog. You can change this name in the project view by selecting the schematic design and using the Rename option from the context menu.

3. **Duplicate Block**

Once you have added a block using one of the two methods above, you can create multiple copies of it using the **Duplicate** command. This will allow you to copy the block symbol and the designs that it uses to define its block contents.

All component and block names used in the original block will be incremented in the duplicated block designs. User named nets will be merged with the original nets if they are used at a higher level outside of the cut block, or will become new nets with incremented names if they are used just within the cut block.

Navigating A Hierarchical Design

To navigate the hierarchy of a project, you can either use the project view to switch to the design representing a block, or you can use the **Push** and **Pop** commands within a design.

To **Push** into a block, select any part of the block symbol and use the **Push Into Block** command from the context menu. The blocks design will be opened and if you had selected a block symbol pin the corresponding block port will be selected. You can use double-click on the block symbol outline, or on a connected block symbol pin to quickly perform the same operation.

To **Pop** out of a block contents design, back to its block symbol, select the **Pop Out Of Block** option from the context menu. You do not need to select anything in the block design to do this, but if you select a block port the corresponding block symbol pin will be selected instead of the whole block symbol. You can use double-click on the block port outline to quickly perform the same operation.

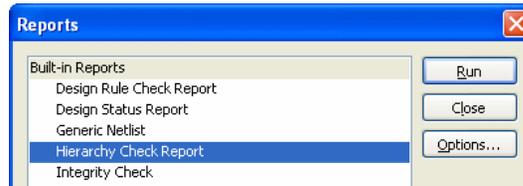
Hierarchical Nets

The span of nets in a project containing blocks is no different to a project with separate schematic sheets in it. If the net in a block design has a user defined name, it may also be used anywhere in the entire design. So the user named net can be used inside and outside of a block, and all items on the net in both designs will be connected together when a PCB design is created. Therefore all user defined net names are always global in a project. Auto named nets are only unique within a single schematic sheet, and so these cannot be used to connect from a block symbol pin in one design to its corresponding block port in another design, as they would end up as different nets in the PCB design.

Each block symbol pin and its corresponding block port must therefore be on the same user defined net. If two different named nets are used, an error will be given when running

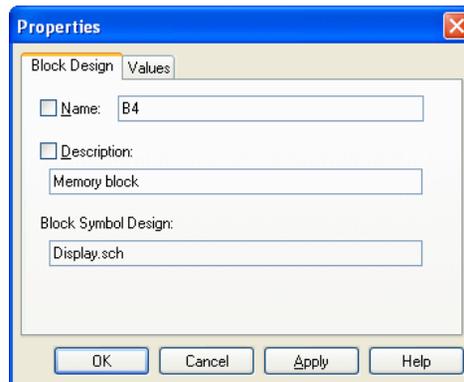
the hierarchy report, or when attempting to create a PCB design or adding forward design changes to a PCB design.

To help keep these nets the same, when you start a new connection from a block port, its corresponding block symbol pin from the higher level design will be accessed to retrieve the name of the net attached to it. This net will then be automatically used for the new connection. Thereafter, though, the nets will not be kept in line automatically, but can be checked at any stage using **Design Rule Check** or the **Hierarchy Check Report**.



Creating Commonly Used Blocks

You may want to create a set of separate schematic designs representing commonly used blocks, such as power supplies or signal converters. These designs can contain block ports to terminate signals to be passed in and out of the block. It is then possible to copy these existing block designs into a project and link them to block symbols added to a higher level design in the project. This is done by using **Design Properties** (**View** menu, **Properties** with nothing selected in the design) to change the block name for the design. The block port names will have to be resolved at this stage to ensure the correct ports are linked to their corresponding block symbol pins.

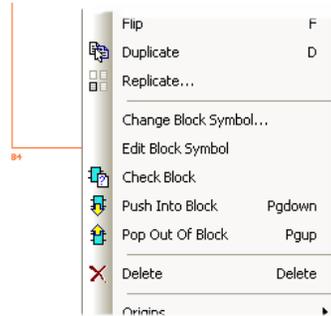


Editing Block Symbols

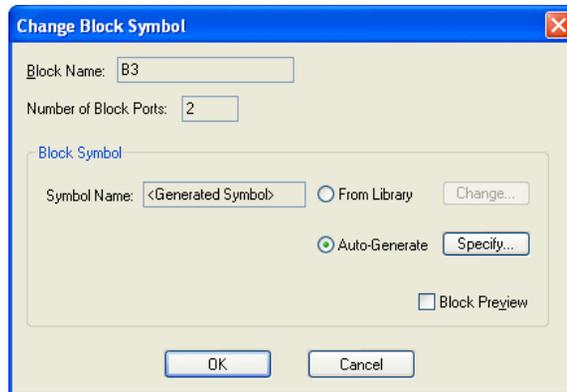
Blocks and block ports resemble schematic symbols, but are implemented as an internal "group" of shapes, text and pins representing the symbol. This enables them to be easily altered within the design, for example to have extra pins, a wider box, or descriptive text added. You can use **Edit Block Symbol** on a selected block symbol or block port symbol to open a new window to edit its contents, add shapes, pins and text etc. After edit, use **Save** to replace it in the design and it will again act like a single item. **Edit Block Symbol** is available on the context menu for a selected block outline.

When adding block symbols or block ports to a design, you can choose to auto generate them using some simple parameters, or use a schematic symbol from a library. The symbol from the library itself will not be used, but a copy of its elements will from the block symbol or port. This enables you to predefine standard block and block port

symbols in your libraries to ensure a consistent interface. For example you could define separate input and output block port symbols.



A block port created from a library symbol will remember its symbol name and can be replaced by a changed symbol from the library at a later stage using the **Change Block Symbol** function from the context menu, this is available for a selected block outline.



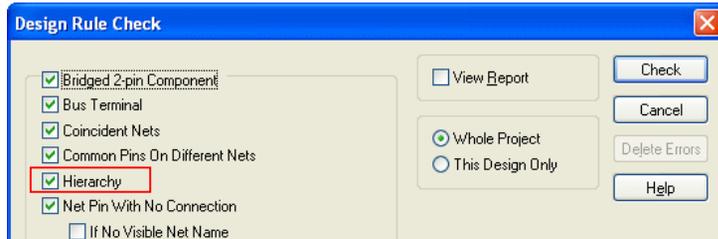
You can set up a highlight colour to force all Block Symbols and Block Ports to be the same colour, thus differentiating them from components in the design. Block symbols can also be individually changed to have their own colour using the **Properties** dialog.

Adding A New Connection To A Block

If you have a new signal that you wish to physically take down to an existing block design, you can start or end the connection directly on a block symbol outline. A new block symbol pin will be added to the outline to terminate the connection on, taking an automatically generated port name. You can then double click on the block symbol to push down into the block design and use **Add Block Port** to add the port, from which you can start a connection to continue the signal in the block design. The net will be automatically set to the net used in the upper level.

Checking The Hierarchy

You can use the **Design Rule Check** option to check the consistency of the two halves of each block in a design or project. An error marker will be placed on a block symbol pin if it has no corresponding block port, or if its block port is connected to a different net.

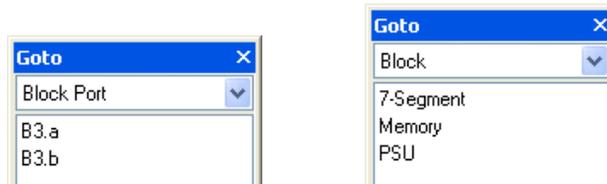


Alternatively, if you don't want to produce error markers, you can run the **Hierarchy Check Report** from the **Reports** dialog to get a report of any problems in the hierarchy. You can run this report on the whole project (from the project view), on a specific schematic design, or just on the selected block symbols from the context menu.

Hierarchical problems can be serious and difficult to fix once the PCB design has been created, so the **Translate To PCB**, **Integrity Check** and **Forward Design Changes** options will run the Hierarchy Checker prior to their operations and will not continue if there are problems.

Finding Blocks

You can use the **Goto Bar** to find any **block symbols** and **block ports** in the entire project.



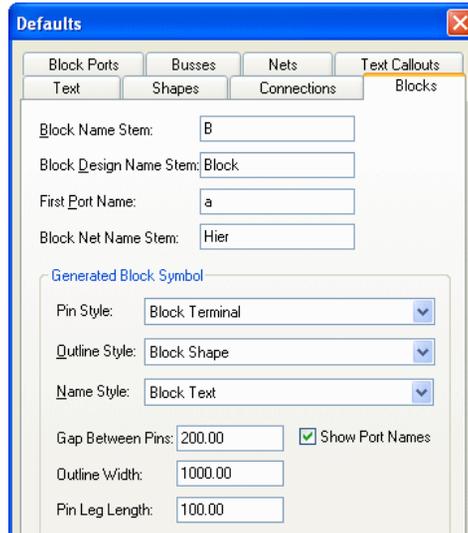
Deleting Blocks

When block symbols are included in a selection that is being deleted you will be asked if you want to delete entire hierarchical blocks by removing all of their block designs from the project. At this stage you can also decide to just delete the block symbols and leave the block designs in the project. These designs can be re-introduced into the hierarchy at a later stage by linking by name to free block symbols, or can be deleted from the project using the project view.

Removal of entire blocks, including their contents, with the **Delete** option does not delete the actual design files from disk. The files can be restored to the project at a later stage or moved to another folder to be used in a different project.

Block Default Settings

The **Defaults** dialog has sections for setting the parameters that define what a generated **Block Symbol** and **Block Port** will look like. Here you can also set up the default name stems for block names, block design names, port names and default block net names.

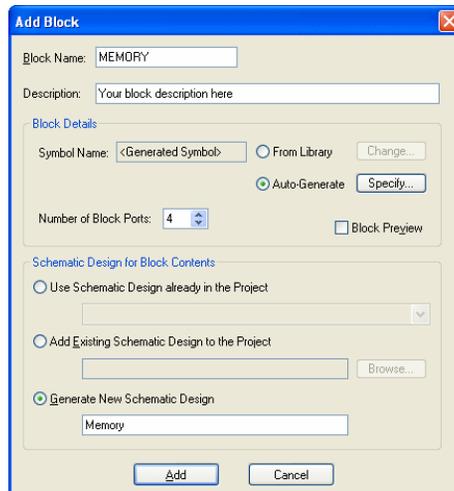


Using Hierarchy

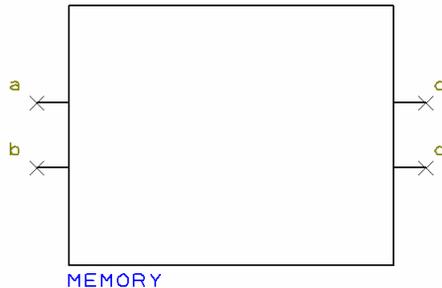
Starting with Top Down Design

► To create a top-down hierarchical design

1. Start with an open schematic from within a Project file.
2. We will first use a new block with no circuitry, we will assume the circuit will be added afterwards.
3. Select **Add Block** from the **Add** menu.



- The dialog is split into three sections; **Block Name** and Description, **Block Details** for the block 'outline' and the number of 'ports', and the bottom box containing the block schematic **contents**.
- For **Block Name**: this identifies the block. It can be a system allocated name, like B1, or a user defined name, like MEMORY. A name to reference the actual contents of the block would be more useful.
- For **Block Details**: we will select **Auto-Generate**. Once you have established some block shapes, you can save these to the library and reuse them using the **From Library**: option. The definition of the auto-generated block shape can altered by selecting the **Specify** box. A dialog will allow you to create the auto-generated block using your own criteria.
- For this example, make the **Number of Block Ports: 4**
- We will create a new schematic design for the **Block Contents**.
- Select the **Generate New Schematic Design**: radio button.
- A default name is entered for you (Block1), change this to be more meaningful, we've typed **Memory**. This will be the new name for your lower level schematic design. It also matches the Block name but this isn't essential.
- Click the **Add** button. This will add your new block symbol to the current design. Don't forget, it will also add you a new lower level schematic design named Memory.



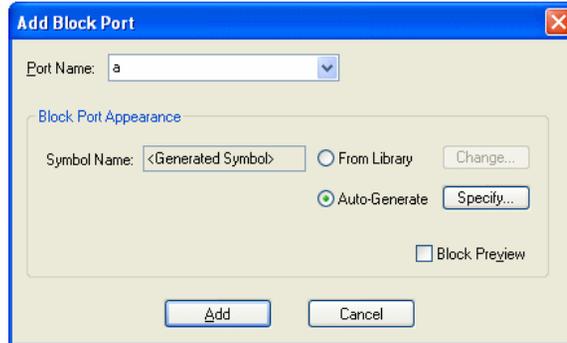
- Once added, the **Add Block** dialog is displayed again to allow you to add another.
- Click the **Cancel** button on the dialog to close it, we only need one block.
- In the design, double-click on the block outline to push down into it. You could also select the outline and then select **Push Into Block** from the context menu. If you have a shortcut key allocated to this option, selecting the block and pressing the shortcut key is also a quick way to push into the block.
- You'll notice the design name appears on a workbook tab at the bottom of the design area, if the **Tabbed Designs** option is enabled on the **Views** menu:



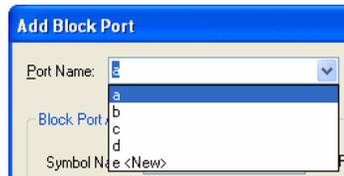
- The lower level schematic will now come to the front and be editable. You now need to add corresponding Block Ports (pins) to the schematic.

► **To add Block Ports to the lower level schematic**

1. From the **Add** menu, select **Add Block Port**.



2. A list of **Port Names**: is shown in a drop down list. This will be a list relating to the number of block ports shown on the top level block symbol.
3. You'll see that the list contains the same number of Block Ports plus one extra, shown as <New>. The list knows how many should be added to match the top level block.



4. The Block Port Appearance is the shape of the block port itself. This can be a symbol from the symbol library or an Auto-Generated one.
5. If using an **Auto-Generated** symbol, the **Port Style** and **Port Outline** can be defined using the **Specify** button and dialog box.
6. For now, we will keep things simple and leave the default Auto-Generated values.
7. Press the **Add** button to add a block port to the design.



8. Each block port is added to the design in sequence. Once one port has been released, you are returned to the **Add Block Port** dialog to add the next.



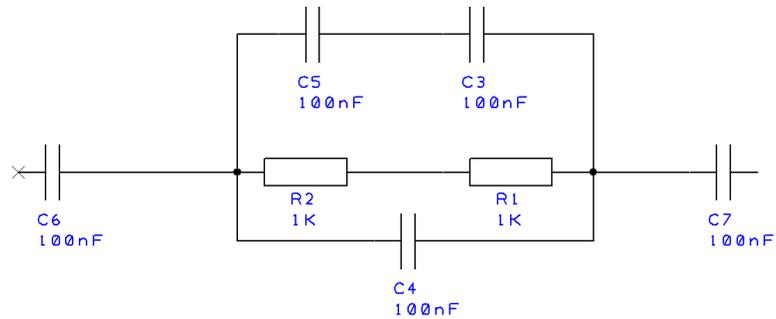
9. When the option gets to the first new port (the one marked as e <New> in our example, and the fifth port to be added, a warning dialog is displayed to inform you that there is a mismatch in numbers of ports between the top and lower levels.
10. Press the **Cancel** button to close the dialog.
11. Your block is complete with both upper and lower levels.

Cutting your design to block

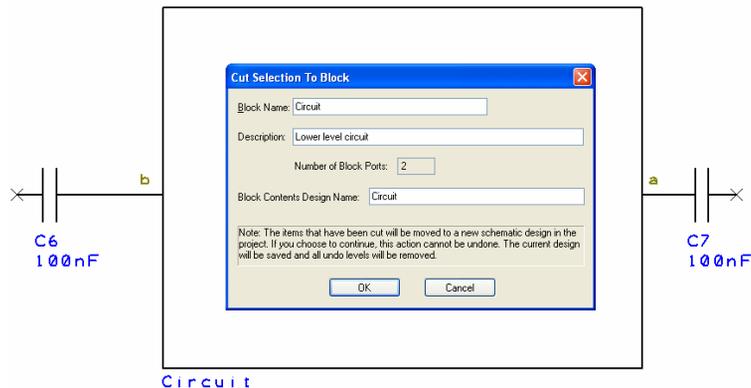
If you have an existing non-hierarchical design, you may wish to divide the design into blocks. The **Cut To Block** command allows you to select a piece of circuitry and create a block from it. Cutting a piece of circuitry to a block does not change the connectivity of the design.

► To cut a design to block

1. Select the **Cut To Block** option from the **Edit** menu.



2. Draw around the circuitry you wish to cut into a block. By default, this is a rectangular frame, you can also use a polygon by selecting **Polygon Select** from the context menu.

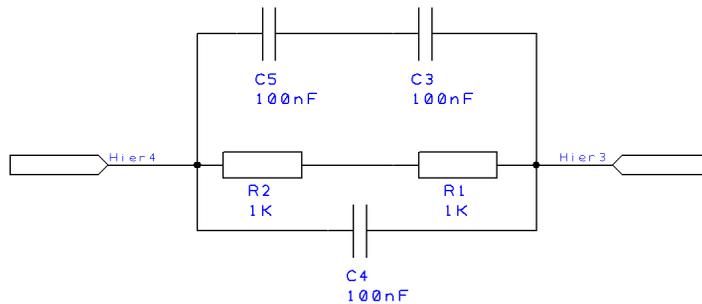


3. Once the selection is completed, a block is created which contains all the items that were fully selected.

4. Each of the connections and busses are cut at the point they cross the selection shape. Connections are terminated on the upper level at a block symbol pin, and on the lower level by a block port.



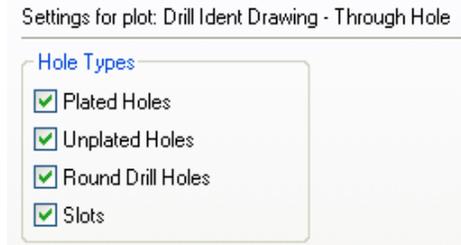
5. If a cut connection does not have a user defined net name, its net is changed to a name generated using the name stem taken from the block defaults.
6. The block symbol outline on the upper level is created using the selection shape. The symbol line style and block pin style for this symbol are taken from the **Block Defaults**.
7. You can later edit this block symbol shape using **Edit Block Symbol** from the context menu, or generate a standard box shape, or use an existing schematic symbol, by using the **Change Block Symbol** option from the context menu.
8. Pushing into the block by double clicking on its outline will reveal all the items removed from the upper level.



9. Connections which were cut will end on corresponding **Block Ports**.

Drill Ident Output (PCB)

You can now choose to output plated/unplated, round or slotted drills separately on the **drill ident** (drill drawing) output in the **Plotting & Printing** option.



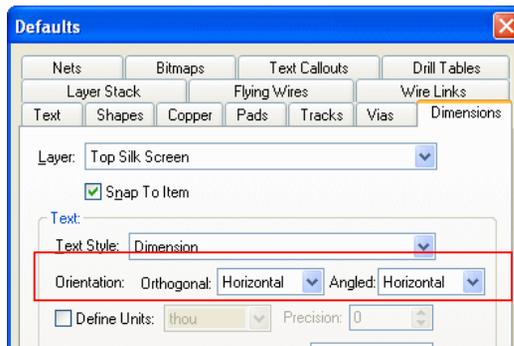
Dimension Text Improvements (PCB)

There are many small changes to the Dimension option to improve its use:

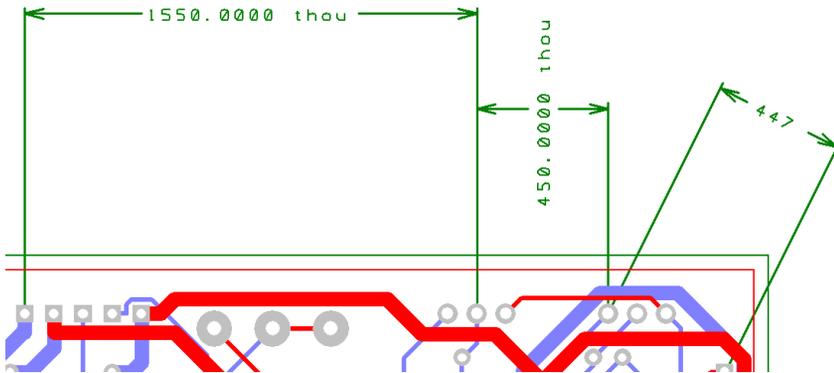
Alignment of Dimension Text with the Dimension

Dimension text can now be defined as Horizontal, Vertical or In-Line with the dimension, previously they were always Horizontal.

This can be predefined in the **Defaults** dialog on the **Settings** menu.



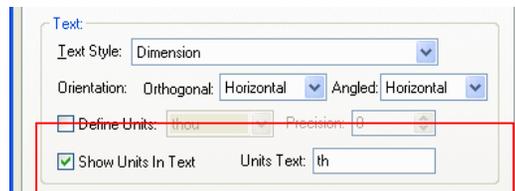
Use this to set the direction of the text item for new dimensions. Choose between having the text **Horizontal**, **Vertical** or **In Line** with the dimension arrow lines. You can set the default separately for **Orthogonal** dimensions and **Free Angle** dimensions.



These settings are also available for individual dimensions by using **Properties**.

Optionally showing the Dimension Text Units

Dimension text can now optionally show the units, previously, it always showed the units.



With the **Show Units In Text** box selected, it will either show the current units or your own **Units Text**: With it not selected, just the dimension value will be shown.

Using the Default Text Position

You can now select a dimension and toggle **Use Default Text Position** from the context menu to change the dimension text from always being put in its default position. If switched off, you can move the text to any position without having to 'explode' the dimension.

Setting up defaults for Dimensions and Properties page

The Dimension text position and text units can be set as **Defaults**, and all new features can also be changed on individual dimensions using the **Properties** dialog for selected Dimensions.

View Board (PCB)

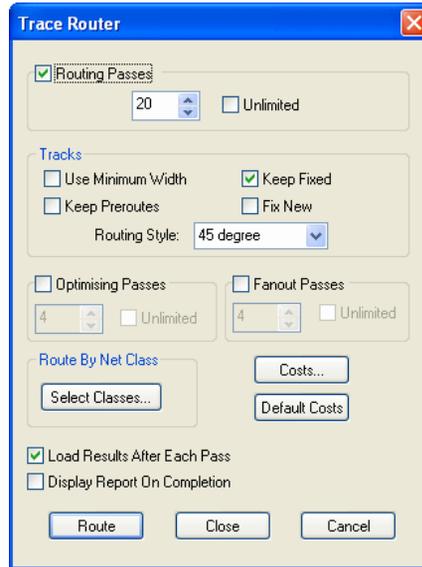
There is a new option on the View menu for zooming into the board outline. Whereas View All will zoom to the maximum extents of all items in the design, the View board option will only zoom to the board outline. It will ignore all other design items outside the board.

New Easy-Router replacement – Trace-Router (PCB)

The Easy-Router product has been replaced with the new **Trace-Router** autorouter from Number One Systems. This will be a direct replacement and will require a new auth-code to license it. It is licensed the same as all cost options using the **Settings** menu and **Optional Features**.

This router has been developed to combat the ever increasing rise in fine pitched devices available today.

The **Trace-Router** is available on the **Tools** menu and when selected the following dialog is displayed:



Routing Passes

Routing This invokes a number of routing passes which will route either all nets in the design or the specified nets, depending on which option started this dialog. You can specify the number of passes that you want. The more your design the larger number of passes that you will want to give it. If it routes to 100% before the number of specified passes then no more passes will be attempted.

Optimising This invokes re-routing passes which attempt to reduce the overall track length and via count once the Routing passes are complete. You can specify the number of passes that you want.

Fanout This invokes a number of passes that routes short escape tracks to vias from SMD pads. This is done before the Routing passes. Fanout passes are not necessary in order for the router to function, and is more useful as a initial phase before manual routing. You can specify the number of passes that you want.

Any of the above pass options can be disabled by unchecking the appropriate box. You can also specify Unlimited passes, in which case, the router will continue until the board is complete, or until it is cancelled.

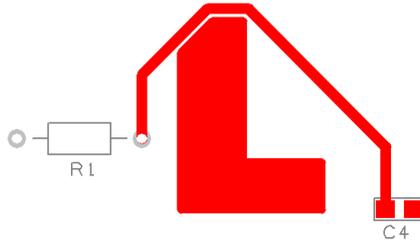
Tracks

Use **Minimum Width** means that if a net has a net class defined the router will use the minimum track style as opposed to the nominal track style.

Keep Preroutes means that existing routing is protected against being ripped up by the autorouter.

Keep Fixed means that only routes that are already fixed will be protected against being ripped up by the autorouter.

And after Pull tight Smoothing. You can clearly see that the track path now ‘hugs’ the obstacle to ensure the track is pulled as tight as it can be between the source and target pads.



Double Click to Start Track Action (PCB)

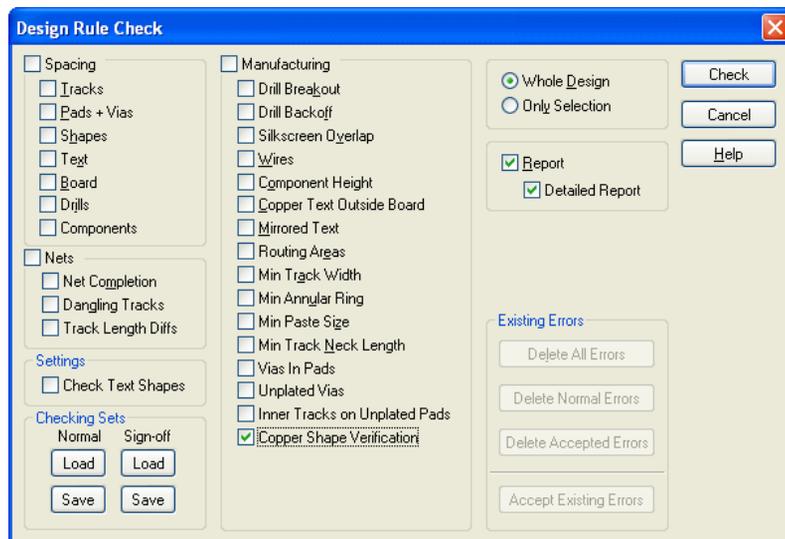
On the **Settings** menu and **Preferences, PCB Tracks**, you can now decide how the double-click on a connection action will behave. You can choose between **Add Track**, **Pull Tight** and **Sketch** modes.



Design Rule Check Improvements (PCB)

Copper Shape Verification

An additional check **Copper Shape Verification** has been added to **Manufacturing**. This examines the shapes that would be generated during Gerber hardware fill and verifies that they won't break out or foul other items.



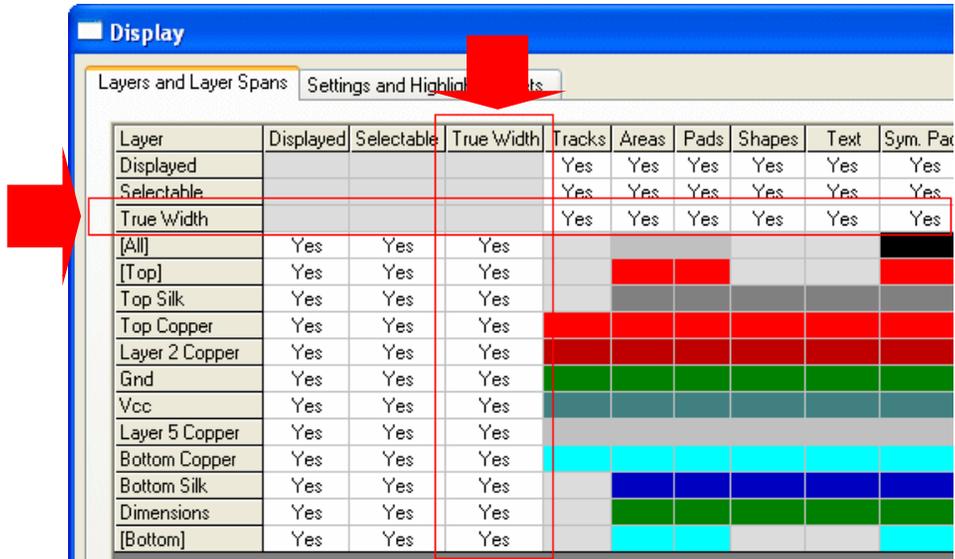
Design Revision Analyser (PCB)

When using the **Design Revision Analyser** from the **Tools** menu, it now checks for changes in **Testland** state on pads and vias. If you have added or removed the Testland status of a pad or via, it will be reported.

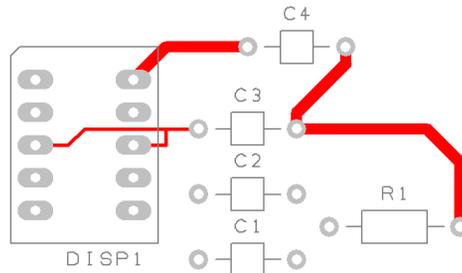
Display of Item True Width (PCB)

To aid the de-cluttering of PCB designs, Easy-PC 14 introduces the ability in **Display** to switch on and off the true width of individual items. When switched off, the item is shown as a centreline object. The exception to this are pads and vias which are displayed as their outer edge (as an outline) and the pad diameter inside.

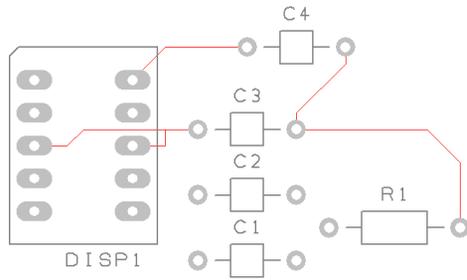
From the **View** menu, this feature is available in **Display**. You will see two extra fields next to Display and Selected called True Width. You can switch the true width of items on and off in columns for each layer or in rows for each item. So for example, you could switch the true width off of all items on the top layer to make the lower layers more prominent. Likewise, you could switch the true width off of all pads in the design, making the tracks more prominent while routing.



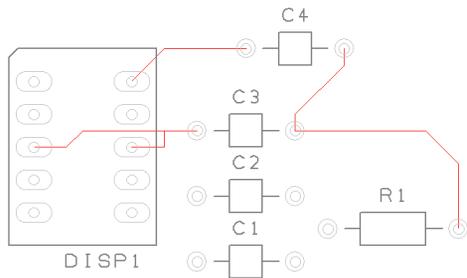
Below are illustrations showing a small PCB design with various combinations of the True Width mode enabled and disabled.



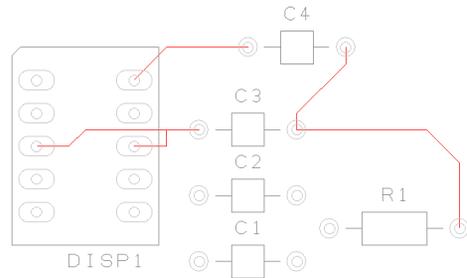
A PCB showing the 'normal' state where all widths are shown as true thickness.



This PCB is shown with tracks with true width set to off.



This PCB shows both tracks and pads shown with their true width off.

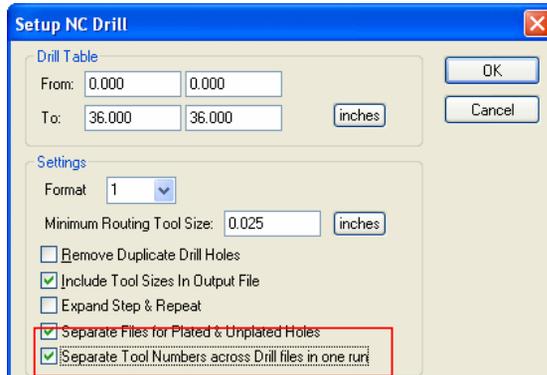


This PCB shows all items with their true width off.

Separate Tool Numbers (PCB)

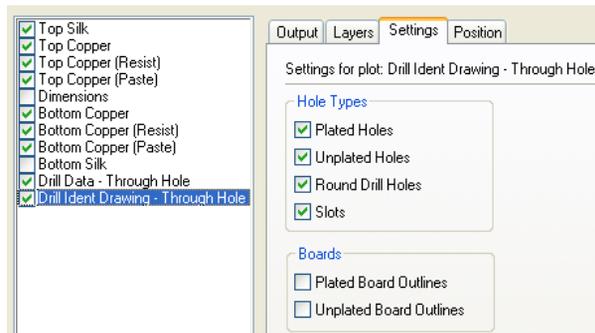
Within the **Plotting & Printing** option, there is a new check box on **Setup NC Drill** (Excellon) dialog; **Separate Tool Numbers across Drill files in one run**.

This option allows you to force tool numbers to be unique across all the drill files generated in one 'run'. So if you have several layer spans and some plated and unplated holes, and are outputting all those different types of hole to different drill files, checking this box will ensure that the tool numbers in each file follow on from each other. With this box unchecked, or for files produced in separate runs, the tool numbering will start from 1 in each drill file.



Separate Slots Selection for NC Drill files (PCB)

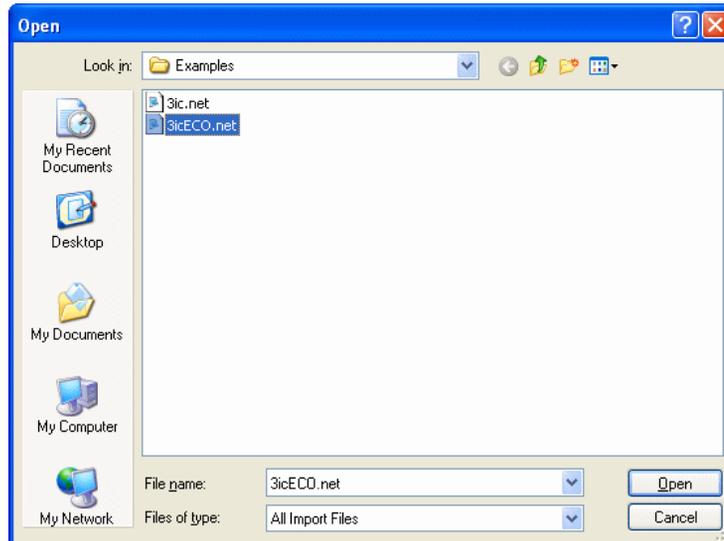
In the **Plotting & Printing** dialog, you are now able to select **Slots** as a new **Hole Type** output for NC Drill and plotting options, for **Drill Drawings** for example.



OrCAD Netlist ECO Import to PCB (PCB)

After running **Import** and using an **OrCAD** netlist, you can now use the **ECO From Netlist** option from the **File** menu to import the updated netlist file. This would be an updated netlist file generated from OrCAD. It is not an ECO file or such like.

When the **ECO from Netlist** option is run, it requests the name of the ECO file.



The net list is read and passed through a comparer to check for differences. The **Forward Design Changes** dialog is displayed:



If changes are found, a confirmation dialog is displayed:



Once the changes have been made, a report is shown with the changes made:

```
Added net "ECO_NET".
Connected pin "U2.12" to net "ECO_NET".
Connected pin "U1.4" to net "ECO_NET".
```

Micro Library 3 Update (Updated Cost Option)

Note: for all updated library contents, the exact quantity may change slightly from the numbers below.

An additional 750 new micro controllers have been added to the existing Micro-controller library from the Altera SX range. If you have purchased this library already, we will supply this update to you free-of-charge when you buy Easy-PC 14. If you decide you would like the update without buying V14, there is a small upgrade charge.

A full list is supplied online in the Micro-Controller datasheet.
www.numberone.com/downloads/datasheets/MicroLibraryUpdate.pdf

Pro Library 3 Update (Updated Cost Option)

The Pro-Library has been significantly updated to include over 20,000 additional Parts and their associated Symbols and Footprints. This takes the Pro-Library from 29,000 to just under 50,000 Parts! This is the second update with an increase of over 50% for you to use instantly. Components have been added for Altera, LT, Maxim, Microchip, National and Xilinx. This upgrade is supplied free of charge when you upgrade to version 14 or as a chargeable option if you decide not to upgrade to version 14.

A full list is supplied online in the Pro-Library datasheet.
www.numberone.com/downloads/datasheets/ProLibraryUpdate.pdf

Connector Library 2 Update (Updated Cost Option)

With it's second update, another 210 components have been added to the connector library giving you a total of over 22,710. These are ready to use and to simply drop into your design. New connectors are included for the Hirose library. This upgrade is supplied free of charge when you upgrade to version 14 or as a chargeable option if you decide not to upgrade to version 14.

A full list is supplied online in the Connector Library datasheet.
www.numberone.com/downloads/datasheets/ConnectorLibraryUpdate.pdf