



V9.0 Supplement

Copyright ©1987-2005 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 dated: 12/07/05 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	5
CHAPTER 1. GETTING STARTED.....	7
Installation.....	7
New Option For Checking Web Site For Patches	7
Running Easy-PC For Windows 9.0	8
CHAPTER 2. NEW FEATURES.....	9
User Interface.....	9
New GUI.....	9
Classic Look.....	9
Document Tabs	9
Customisable Menus and Toolbars	10
Sliding Preview bars	12
Add Component bar.....	12
Layers Bar	13
Tabbed Windows	14
World View.....	15
Databook Changes	16
3D View	17
Defining the 3D Packages	18
Display Dialog	22
Layer Viewing (Drawing in Order).....	23
Pad Stack Drawing.....	24
Reverse Engineer	24
Interactive Schematic Routing	25
Component Height Checking.....	26
Component Push-aside.....	28
Preferences Dialog Changes.....	29
Library Dialog Changes	30
Changes to Component Editor	30
Library Manager	34
Signal Components	34
Alphanumeric Pin Name Support.....	35
Add Circle/Add Rectangle Changes.....	36
Coordinate Edit Boxes on Dialogs	37
User Defined Coordinate Origin	37
Asymmetric Grids.....	38
Changes to Units Dialog	38
DRC Changes.....	39
Changes to Styles Dialog	40
Plotting & Printing Changes	40
Windows Setup Dialog	41
Plated/Non-plated Drill Hole Support.....	41
NC Drill Output	42
Drill Ident Drawing Output.....	43
Layer type Oversize	44
Testland Support	44
Track Fattening	46
Automatic Track Necking into SMD Pads	47
New Custom Reports Dialog.....	48

Component Rename	51
CHAPTER 3. EASY-SPICE	53
New Spice Update	53
Speed Increase	53
CircuitSim90 Benchmark Netlists	53
Easy-PC Interface Improvements	53
Random probing	57
Random Probe Types.....	58
Bias Annotation.....	58
Simulation Interface	60
New Models.....	62

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 by running the setup program through the Start and Run menu option on the Windows task bar.

The installation is the same for new and existing users alike. Existing users i.e. version N0820, V2.1, V3.0 to V8.0 can install V9.0 over an existing installation without deleting the old one first.

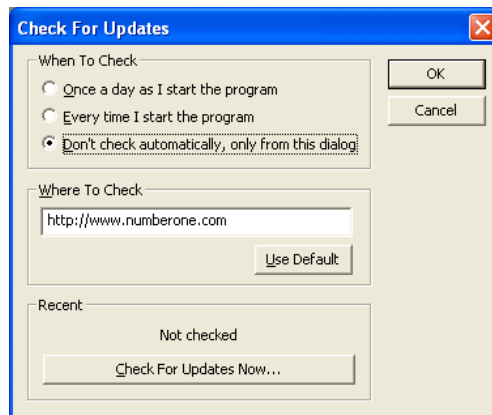
With the installer running, once the **Welcome** screen is displayed, double-click on the **Install Easy-PC For Windows - Version 9.0** option, or click then press **Run**. Following the instructions on the screen, you should use the same **Destination Folder** for the Program Files as your existing 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.

New Option For Checking Web Site For Patches

On first running version 9.0 you are prompted to choose whether you want Easy-PC to check for program updates on the web site. If you say **Yes**, a new dialog is displayed that allows you to set this up. You can also set up this option from the **Support** option on the **Help** menu and **Check for Updates**.



When checking for updates, the current version you are running is compared against the latest patch version available on the web site, and a message box is displayed if a new update is available. You can then opt to get the update, which will launch the appropriate Updates page in your web browser to allow you to download and install the update.

Running Easy-PC For Windows 9.0

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.

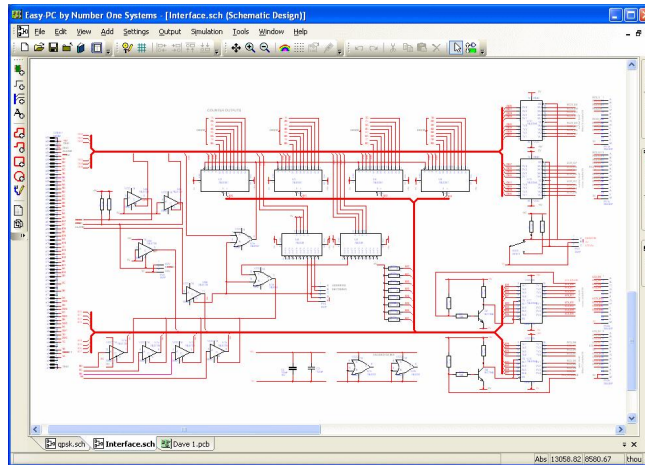
To start the program, double-click on the Easy-PC icon from the Number One Systems folder.

Chapter 2. New Features

User Interface

New GUI

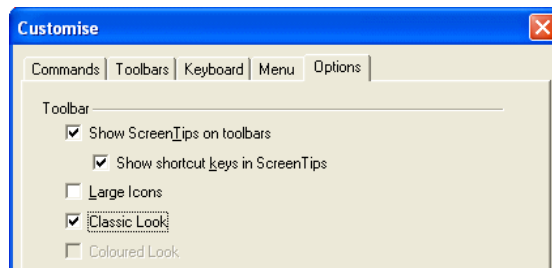
Version 9.0 has been updated to provide a Windows Office style look and feel. Menus, Toolbars, Shortcuts and Customisation reflect the new look and Menus as well as Toolbars may now be customised.



Each of the new GUI features is discussed below.

Classic Look

An option to switch back to the **Classic look** (version 8.0 and previous) is provided in the **Customise** dialog.



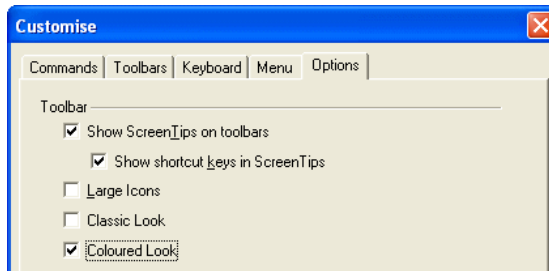
Document Tabs

There is an option, **Tabbed Designs**, on the **View** menu to have tabbed design views in the main application window. By displaying these, you can easily switch between open designs in Easy-PC by simply clicking on the required tab.

By hovering the mouse over any of these design tabs, a tooltip showing the full design path will be displayed.



There is now a **Coloured Look** checkbox on the **Options** page of the **Customise** dialog to switch on the coloured effect for the Easy-PC user interface; primarily the use of coloured tabs in tabbed windows. This option is only available if the Classic look is not being used.

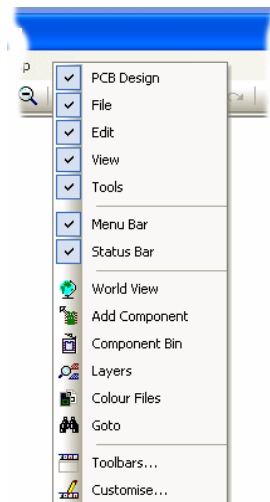


As well as sliding toolbars, the workbook mode clearly shows the coloured tabs.

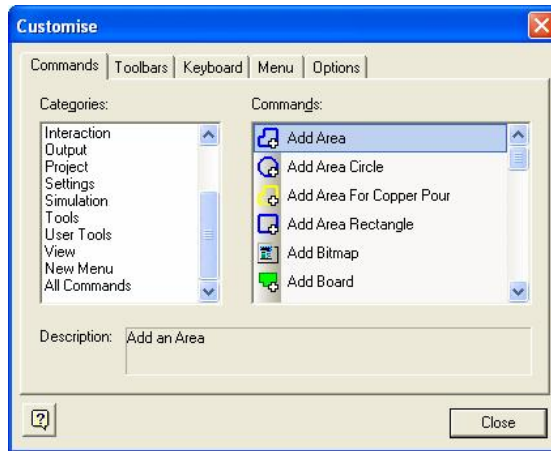


Customisable Menus and Toolbars

Customisation of the menus, toolbars and shortcut keys is done through the **Customise** dialog. This dialog is available from the **Settings** menu and by right clicking on any of the toolbars, menu bar or Easy-PC framework.



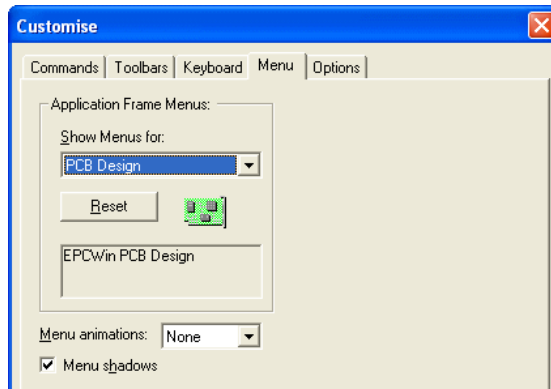
The Customise dialog has individual tabs for use with the option required as before except that **Menu** and **Options** tabs have been added.



The **Menu** tab allows menus for different design types to be customised and provides options for changing general menu behaviour and appearance.

By clicking the Reset button, the currently selected menu/application type will be reset to the default setting and your settings will be lost.

Menu animation and **Menu shadows** only affect the detailed look of the menus.



The **Options** dialog is used to customise the look of the toolbars, see previous section above under *Document Tabs*.

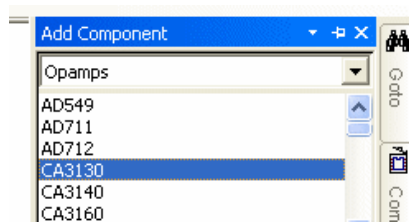
Shortcut Keys

The **Shortcut Keys** tab has now been renamed to **Keyboard** but no other functionality has been changed.

Sliding Preview bars

The **Preview Bar** has been split in to several individual dockable bars; **World View**, **Component Bin**, **Colours Files Bar**, **Layers Bar**, and a **Goto Bar**. Each of these may be displayed, hidden, docked, floated, auto-hidden (so they can 'slide') or combined together in a tabbed bar.

These bars will be automatically hidden when they go out of context and re-displayed when they come back into context.

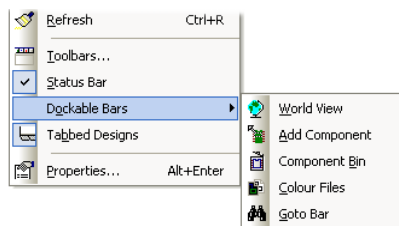


The example above is shown docked on the right side of the interface.



The three small icons (shown circled) on the top of the sliding bar indicate additional functionality. Moving left to right; these reveal the shortcut menu, a pin to enable the dialog to be sliding to conceal it or docked to be permanently displayed. A cross indicates the Close option to dismiss the dialog altogether.

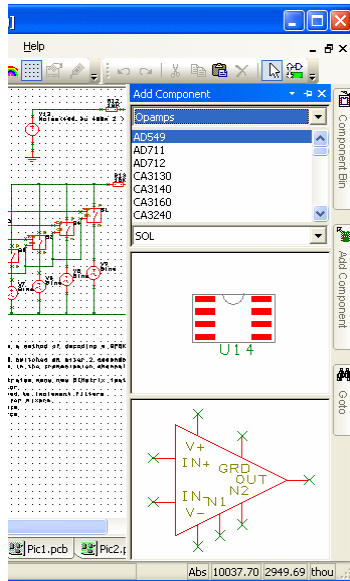
Dockable toolbars can also be selected using the **Dockable Bars** sub menu on the **View** menu.



Add Component bar

A new **Add Component** dockable bar has been added that is effectively an 'always-accessible' modeless dialog for adding components, and is available for a Schematic or PCB design.

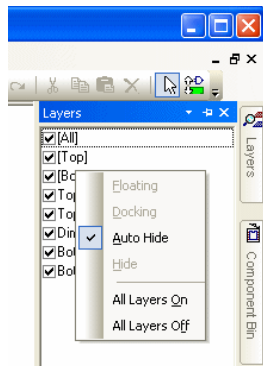
It allows a component to be 'dragged' from the selected library directly into the current design view. Alternatively, double clicking activates the normal 'button-up' add component mode and allows multiple instances of the component to be added. The component package name is also displayed allowing an alternative package to be chosen.



This window is more basic than the Add Component dialog as there is no name filtering or Find option. However, because this uses the sliding bar, then it is accessible very quickly.

Layers Bar

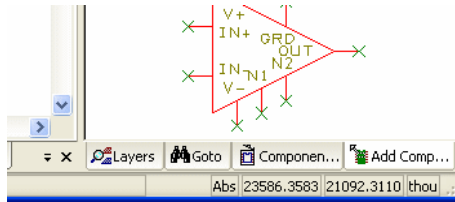
The **Layers Bar** (previously Layers On/Off in the Preview Bar) has some small changes to how it operates compared to version 8. Previously the only action available was to select/unselect layers which would show/hide that layer in the design. Now, each layer is shown with a checkbox. Clicking the checkbox toggles that layer's visibility. Single selecting the layer name merely highlights that layer in the list whereas double-click will make just that layer visible, hiding all others. There are also now context menu entries available for 'All Layers On' and 'All Layers Off'.



There are further modifications to match the changes in the **Display** dialog to show [All], [Top] and [Bottom] layers and to allow them to be toggled between visible and invisible. Single layer selection on one of these layers will only switch the other 2 off, and will not change any of the real layers. Toggling or single selection of a real layer will update these layers accordingly.

Tabbed Windows

The window can also contain tabs within any other dockable windows options. You can switch between any of these windows by clicking on the appropriate tab.



A tabbed window contains multiple dockable windows and may be manipulated like any other dockable window in that it may be resized, docked, floated or auto-hidden. Additional dockable windows may be dragged into the tabbed window and existing tabs may be dragged out to become individual windows once again. Tabs may be re-ordered simply by dragging them into the desired order.

Depending on the application 'look', Easy-PC will now also provide you with some extra features to aid the docking of tabbed windows (these are not available in the Classic Look).

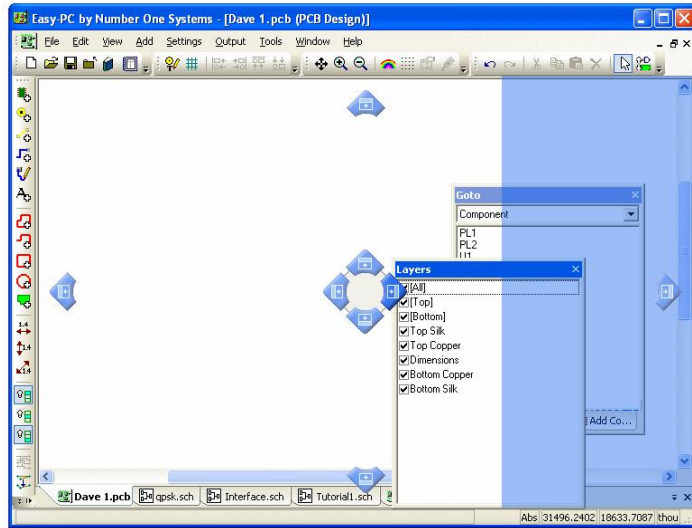
Docking Indicators

Docking indicators appear to show the potential dock positions which can then be chosen by releasing the drag with the mouse pointer over the appropriate indicator. The docking indicator is different when over an existing docked window.



To reposition a docked window, press and hold down the left-hand mouse button on the window title bar border and drag the window frame to the desired docking indicator. Any of the four 'outside' indicators will allow the window to be docked. By dragging it over the centre indicator, it will be added to the existing window as a new tab.

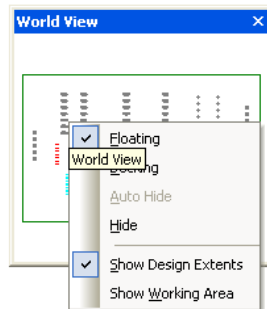
When used on a dockable window it looks like this:



Outside the main docking indicator are four other position indicators. These show that the window can be docked to the adjacent edge of the design area of main Easy-PC framework.

World View

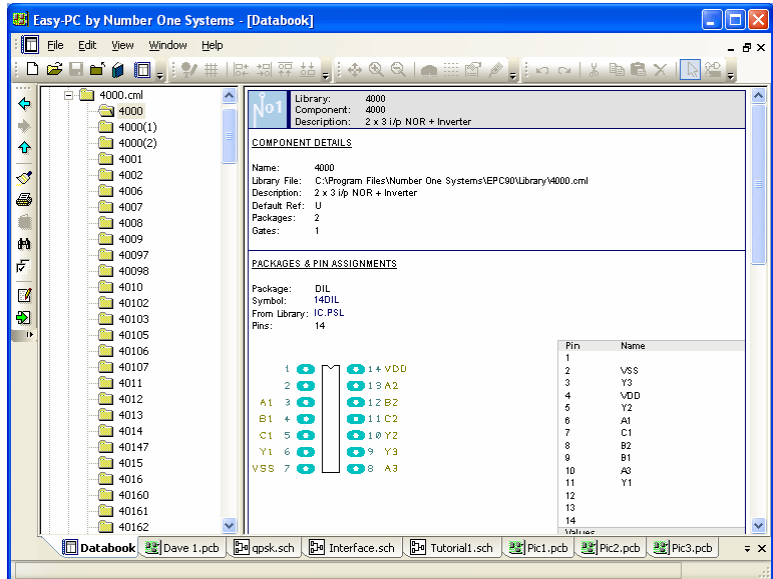
As well as now being a separate dockable bar in its own right, the **World View**, which has previously shown the entire 'working area' may now be switched to show just the 'design extents' via a context menu option.



Databook Changes

The Databook now appears as a 'child' window inside the main application window. It used to be a separate window in version 8. The Databook menu/toolbar that was embedded in the separate window is now incorporated into the application menu/toolbar in the Customise dialog when it comes into context, (like any other design type).

The **Switch To/From Databook** option will now pop the appropriate child window to the top. The **Add to Design** command will operate on the most recent used design view; i.e. usually the one from which the switch to the Databook window was made.

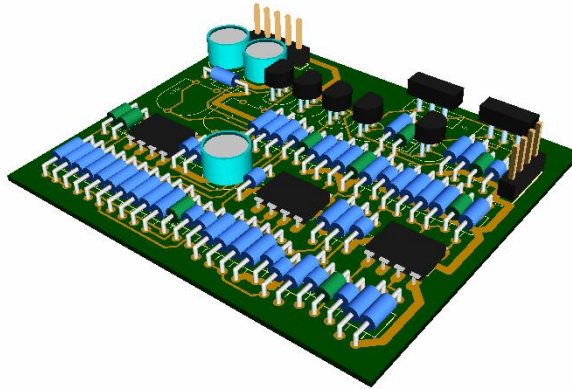


3D View

The new 3D viewer allows the 'flat' view of the PCB to be annotated so that it can be seen as it would be when populated with components after assembly.

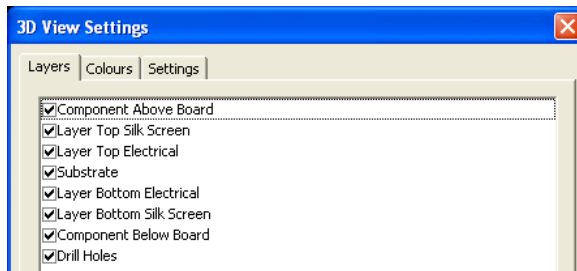
A new menu option **3D View** on the **View** menu in the PCB editor is used to send the current design into the viewer.

The appearance of components in the 3D view is controlled using a new type of library (3D packages) which can be edited as new components are added to the library.

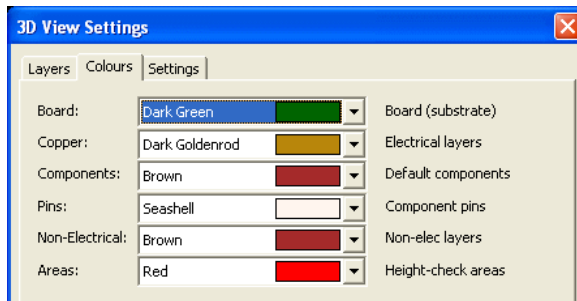


Options within the 3D Viewer

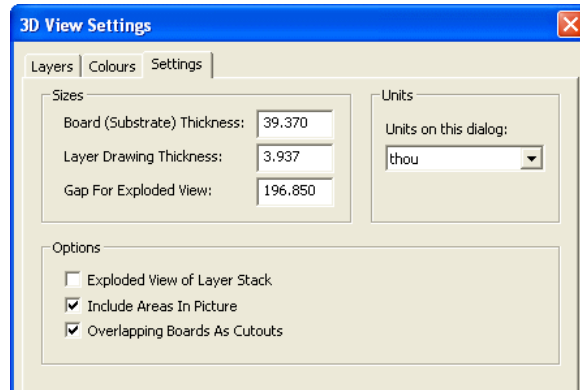
The **Options** dialog is available on the **3D** toolbar and on the **Tools** menu. The three tabs within it allow you to define how the 3D view will look.



The **Layers** tab will enable you to switch on and off the design layers within the view.

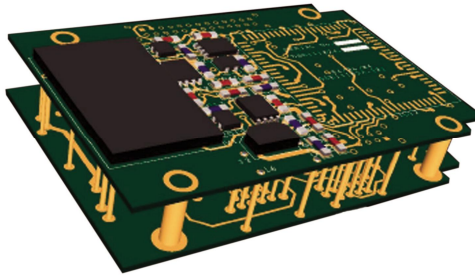


The **Colours** tab allows you to setup preferred colours for the 3D view.



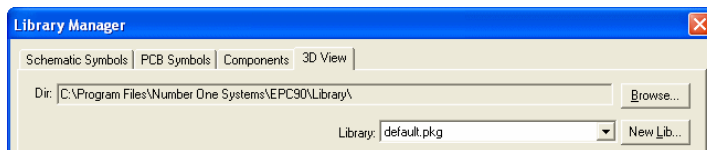
The **Settings** tab is used to define view parameters such as the thickness of the board in the 3D view.

Each of the entries for **Sizes** defines a thickness that can be created and viewed. Generally speaking, **Board** and **Copper** thicknesses would be consistent, whereas you may change the **Gap** thickness to show internal layers more clearly, perhaps to show the internal via stacks.



Defining the 3D Packages

In order to view components in the 3D viewer, you must define the package 'shape' in the library. There is a new tab in the libraries dialog called 3D View, use this to define package libraries.



The package libraries have the file extension .pkg. The contents of the package libraries have definitions which relate to a footprint or groups of footprints, for example DIP*. The supplied libraries are defined by functional groups i.e. connector, CPS, SMT, DIP

Package Overview

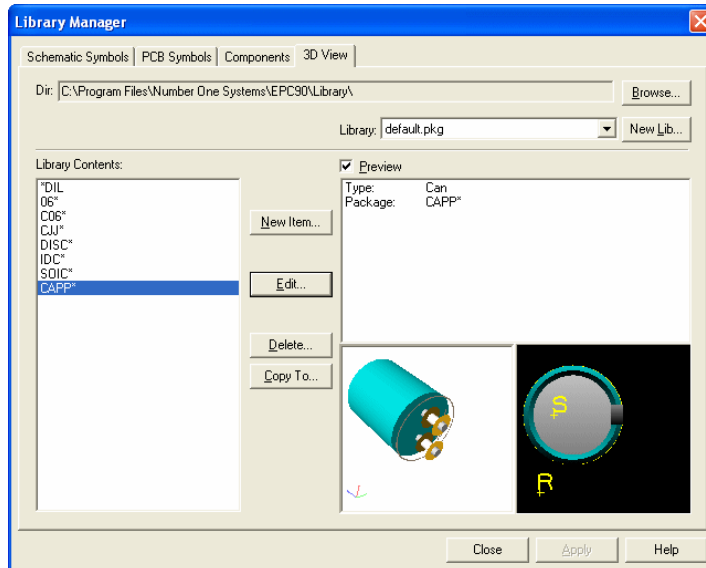
3D Packages define how PCB symbols are represented in 3D View images of your PCB designs. Each 3D Package has a 'style' (DIL, Quad, Axial, and so on), and various settings that control how the 3D image is generated. It also has one or more names that specify which PCB symbols the 3D Package applies to, allowing one package to be used to

generate the 3D image for many PCB symbols. These names can be specific symbol names, or they can include 'wildcards' to match multiple symbol names.

To generate the 3D view of the actual components on your PCB design, the set of package libraries is scanned to find the first package that matches each of the PCB symbols used in the design. Because each package can have a basic name and a set of 'aliases' both of which can include wildcards, it is possible to create packages that can auto-generate the 3D view for many PCB symbols.

3D Edit Dialog

The 3D View tab gives a summary of the selected item, and small pictures in 3D and 'normal' view of a corresponding footprint. The PCB symbol that is used for this dialog will be the first one found that matches the rules defined by the selected package.

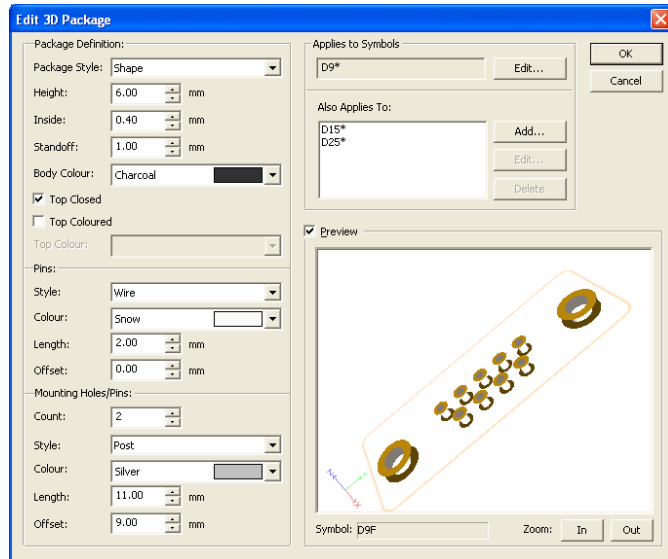


Edit 3D Package








This dialog is used to edit or create a 3D Package definition. Fields on the dialog allow you to specify which PCB symbols this package would be used for, and the style and other properties of the package.

Using the dialog

The dialog is accessed from the **3D View** tab of the Libraries dialog. Choose the package you wish to edit, and click the Edit button. The dialog is divided into several sections, each of which is described below. Depending on the type of package, and other factors like the number of pins, some sections or controls may not be available or appropriate, or may not have any effect on the resulting picture. The best way to see the effects is to try them out and see how the **Preview** picture changes.



Package Style: The main control in this section, which specifies the general type of the package. Available styles are:

- | | | |
|--------------|---|---|
| Axial | Generally two-pin through-hole components like resistors and diodes |  |
| Box | Rectangular components where the box is taken from the overall size of the component |  |
| Can | Vertical cylindrical components such as large capacitors |  |
| Chip | Surface-mount chip-type components such as chip resistors and capacitors |  |
| DIL | Dual-in-line components such as DILs, SOICs |  |
| Quad | Components (generally surface mount) with pins on all four sides |  |
| Shape | General components where the 3D body shape is derived from the 'best fit' shape in the PCB symbol |  |

Height: the overall height of the component body.

Inside: for some types, this specifies the amount by which the body should be undersized.

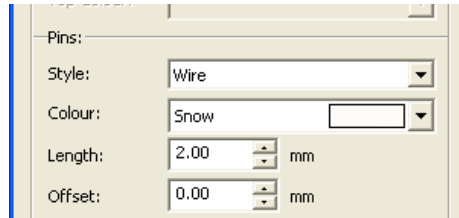
Standoff: the distance between the board surface and the underside of the component body.

Body Colour: the colour of the component body.

Top Closed/Coloured/Colour: Some packages styles allow the top of the body to be open or closed, and/or set to a different colour to the rest of the body.

Pins

This section specifies how the pins should look.



The screenshot shows a dialog box titled "Pins:" with the following settings:

- Style: Wire
- Colour: Show
- Length: 2.00 mm
- Offset: 0.00 mm

Style: Type of pin. Some of the pin styles are sized and positioned to fit particular styles of package. For example, J-Lead, Gull-wing and DIL pins will only work for package styles such as DIL and Quad where the pins are around the edge of the component.

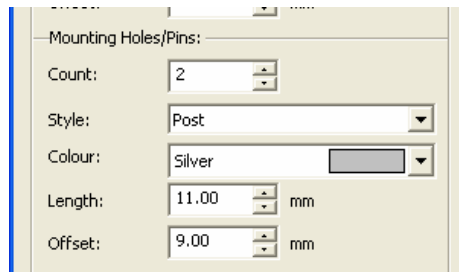
Colour: Pin colour

Length: Overall length of pin. This does not have any effect for some types such as J-Lead and DIL which measure the pin length automatically from the midline of the component body.

Offset: Allows the pin to be offset up or down the component.

Mounting Holes/Pins

Certain styles allow some of the pins to be designated as mounting holes/pins. If Count is set non-zero, the last pins on the PCB symbol are drawn using the specified mounting style instead of the pin style.

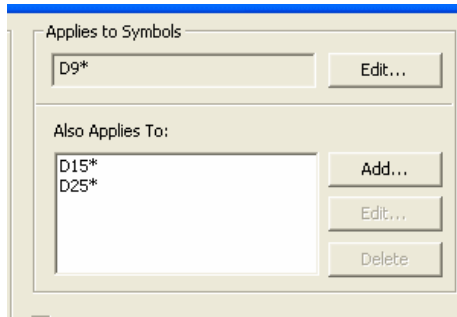


The screenshot shows a dialog box titled "Mounting Holes/Pins:" with the following settings:

- Count: 2
- Style: Post
- Colour: Silver
- Length: 11.00 mm
- Offset: 9.00 mm

Applies To Symbols

This section specifies the PCB symbol names to which this package will apply. The first name is the name by which the package will be listed in the Libraries dialog. The other list shows additional names to which this package will match. Any of these names may contain wildcards, as shown in the example below.



Preview

This section shows a preview of the 3D package you are editing, as long as an appropriate PCB symbol can be located in the available libraries. Just below the preview is shown the name of the PCB symbol that is being used.

You can see the effect of changing any of the controls on the dialog straight away. The picture can be rotated in any direction (in the same way as for the main 3D view itself) by moving the cursor into the preview window, pressing and holding down the left mouse button and then moving the mouse. Buttons are also provided to allow you to zoom In and Out of the preview picture.

Display Dialog

Pad Colours

Pad colours in the PCB Display dialog can now set on individual layers to allow exceptions and oversizing to be distinctive from other items on that layer.

Display						
Layered Items		Other Items				
Layer	Displayed	Selectable	Tracks	Areas	Pads	SI
Displayed			Yes	Yes	Yes	
Selectable			Yes	Yes	Yes	
[All]	Yes	Yes				
[Top]	Yes	Yes				
[Bottom]	Yes	Yes				
Top Silk	Yes	Yes				
Top Copper	Yes	Yes				
Dimensions	Yes	Yes				
Bottom Copper	Yes	Yes				
Bottom Silk	Yes	Yes				

Layer Sets

You can now make layers [All], [Top] and [Bottom] selectable or pickable in their own right.

These ‘special’ layers should not be confused with a ‘special effect’ layer. Setting a special layer off/on is a way of hiding/showing through-hole or surface-mount pads on one side of the board without affecting other pads, and without changing the visibility settings of the individual ‘real’ layers. They should be seen as an override to the individual layer settings. For example, not-displaying [Top] will hide all surface mounted pads on the top side of the board and re-displaying [Top] will make visible only the real layers on the top side that were visible before [Top] was not-displayed. Although switching the visibility of a ‘special’ layer doesn’t affect the real layers that comprise that set, changing the visibility of one of the constituent layers will reset the appropriate

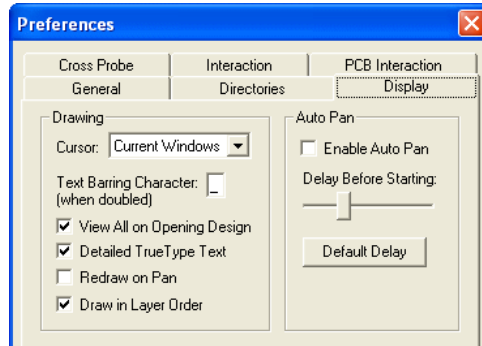
‘special’ layer if necessary. So, for example, if [Bottom] was set not-displayed, subsequently switching Bottom Copper from invisible to visible would force [Bottom] to be reset to visible also. Doing the opposite and switching Bottom Copper to invisible will only set [Bottom] invisible if no other real layers remain visible on the bottom side of the board.

Layer Viewing (Drawing in Order)

Layer Order Drawing

Now in PCB, when a design is repainted all layer-based design items will be redrawn in correct layer order such that all items on upper layers will appear on top of items on lower layers; for example, tracks and copper on the top layer will appear on top of tracks and copper on an inner layer, which in turn will appear on top of tracks and copper on the bottom layer.

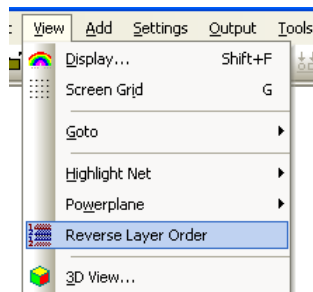
In Easy-PC V9.0 this option will be enabled by default, but if you do not wish to use this feature, it may be switched off by un-checking the **Layer Order Drawing** option in the **Display** tab of the **Preferences** dialog.



For the best effect when using the Layer Order Drawing it is best to switch off the **Merge Track/Shape Colours** options in the **Display** dialog.

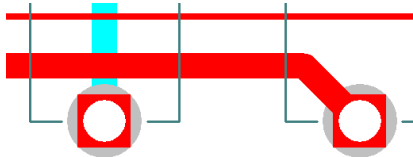
Reverse Layer Order

There is a new command, **Reverse Layer Order**, available on the **View** menu for a PCB design. This allows the normal **Layer Drawing Order** to be reversed so that lower layers draw on top of upper layers. It may also be added to a toolbar or assigned to a shortcut key by using **Customise**.



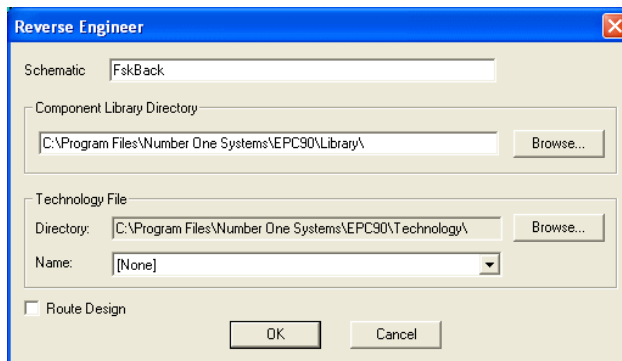
Pad Stack Drawing

In a PCB or PCB Symbol design, a pad will now be displayed on screen with any defined oversizing or exceptions visible. The oversizes and exceptions will be displayed if the particular layers on which they are defined are enabled and will appear in the appropriate layer order as viewed from above, this is reversed if reversed layer order is active.



Reverse Engineer

The **Reverse Engineer** tool is designed for use where there is no corresponding Schematic design for the existing PCB design. It is not intended for use where a PCB design is modified and the changes are to be reflected in the Schematic (the Schematic is always considered the 'master' design in this situation).



Where a PCB design has been created, and if the PCB design is used to create 'local' libraries with the Schematic symbols added to the components in the new library, you must update the components into the PCB design first before using the **Reverse Engineer** option. Use the **Update Component** option from the **Tools** menu.

If you opt to use PCB only components, the Schematic Symbols/Components will be auto-generated from you when the Reverse Engineer option is selected. This method will work fine but will create dummy Symbols only, symbols that are shown as boxes with a representative number of terminals.

Using the Reverse Engineer dialog will enable you to select the new **Schematic** name, which library **folder** to use and the **Schematic Technology File** name. You can also elect to **Route the Design** automatically once autoplaced. Once back in the Schematic design, all Symbols are automatically placed.

Autorouting selected Symbols

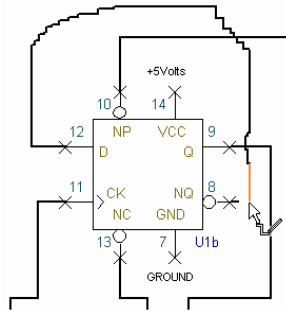
If you choose not to autoroute the design, you can selectively route them afterwards by using shortcut menu options on selected pins or components.

It does 95% of the work in creating a Schematic. A bit of tidying-up is required of course but it is a big time saver for those that choose to work this way.

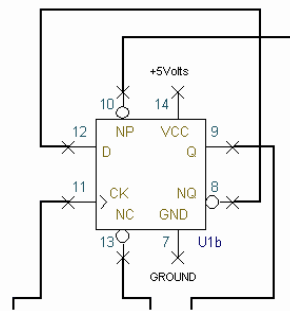
Interactive Schematic Routing

The Sketch connection path mode has been introduced into the Schematic design editor. This allows you to interactively make a connection between two points using a basic 'sketching' mode, or by using a point-to-point mode.

Using the **Sketch Connection** mode from the **Tools** menu, you can add connections to the design. Once in this mode, this is verified by the display of a modal cursor, a cursor which shows a mode.



This picture shows the approximate path that you 'sketch' in. The modal cursor is also shown.

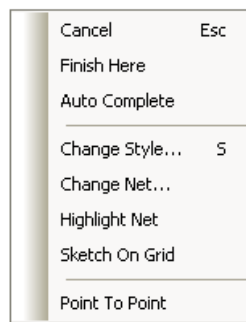


This picture shows the final result after completion and tidy-up.

Immediately after using sketch mode, the tidy routine will snap the connection onto the grid and remove all the rough edges for you. Once added, the connection path can be tidied up using normal connection editing options.

Shortcut menu options

While using the sketch mode, you can use options from the shortcut menu.



Point to Point mode

On the shortcut menu, you can select **Point To Point** mode. This allows you to point and click to add a connection path. After the mouse click, the path will be tidied back to the last click, this gives you a 'guided' routing path which is automatically tidied as it goes along. This may be a preferred mode of operation to the normal sketch mode.

Component Height Checking

Height Checking has been added to the Easy-PC PCB design editor. This allows you to define a special height area on the design with a maximum allowed height. The maximum height allowed can be assigned on the whole board (using the Spacings dialog) or on an area basis (using Height Checking Areas).

Using Height Checking

The process for using this option is this:

1. Define the Height **Value** to be used in the **Spacings** dialog.
2. Optionally, define the **Side** to be checked and the **Max Allowed Height** for that layer in the Spacings dialog.
3. Add a **Value of Height** to the components to be checked in the design. Do this in the component library or in the design itself (using **Properties** or **Component Values** editor) on a component basis (it is more efficient to do this once in the component library for each component type).
4. Run the **Design Rules Checking** option and select **Height** under **Manufacturing** rules.

Spacings dialog

On the **Spacings** dialog in the PCB design editor, there are new controls for setting up the maximum component heights for Top and/or Bottom layers, and choosing the default Value name for components to define their height.

	Tracks	Pads	Vias	Shapes	Text
Tracks	10.0000				
Pads	10.0000	10.0000			
Vias	10.0000	10.0000	10.0000		
Shapes	10.0000	10.0000	10.0000	10.0000	
Text	10.0000	10.0000	10.0000	10.0000	10.0000
Board	20.0000	100.0000	20.0000	20.0000	20.0000

Powerplanes:
 Isolation Gap: 10.0000
 Thermal Relief: 10.0000

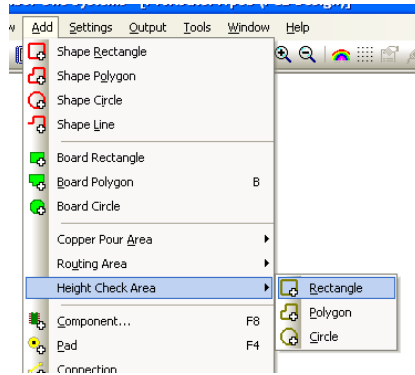
Pads and Drills:
 Drill Spacing: 0.0000
 Min Annular Ring: 15.0000
 Min Paste Size: 3.0000

Component Height Checks:
 Top Max Allowed Height: 400.0000
 Bottom Max Allowed Height: 200.0000
 Value Name: Height

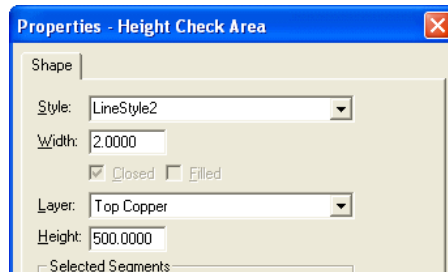
Tracks:
 Minimum Track Width: 2.0000
 Min Neck/Fatten Length: 250.0000

New Add menu option

On PCB **Add** menu, there is a new option to add **Height Check Areas**. These areas are layered and have a maximum component height setting available in their **Properties**.

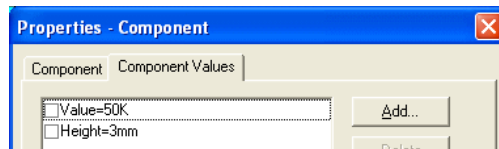


When the area is selected, its **Properties** will allow you to change the area **Height** for checking. That is, components within this area will have their Value checked against the maximum defined for the Area.



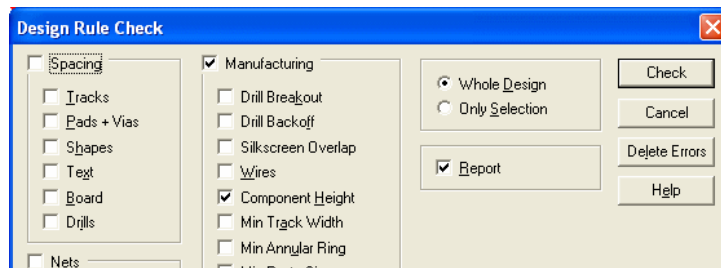
Component Properties

For **Properties** of components, you should add a Height value (Height is the default value but you can define and use any Value you wish). This value can be assign locally within the design on a component basis but it is more efficient to define this in the component library.



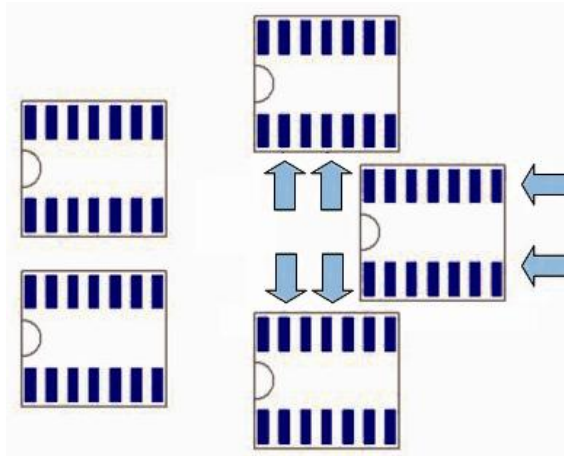
Design Rules Checking

On DRC dialog, a new checkbox for **Component Height** under **Manufacturing** has been added. Use this for checking the Values and Spacing Heights of components.

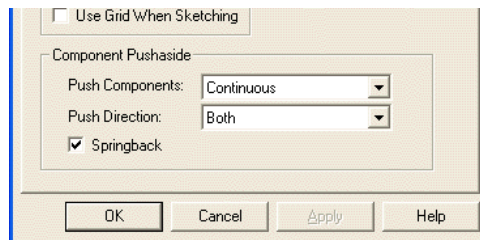


Component Push-aside

Components can now be 'pushed' while interactively moving them. This feature is enabled through the **Preferences** dialog and **PCB Interaction**, or while actually moving the component using options on the shortcut menu. Options are available for pushing to select the push mode as **Never**, **Continuous** and **On-drop**. The push direction can also be set to **Both**, **Vertical** and **Horizontal**. You can also set whether components use Springback (with springback set the component being pushed will continually attempt to return to its original position before the pushing started. This is particularly useful on a tightly packed board where space is at a premium).

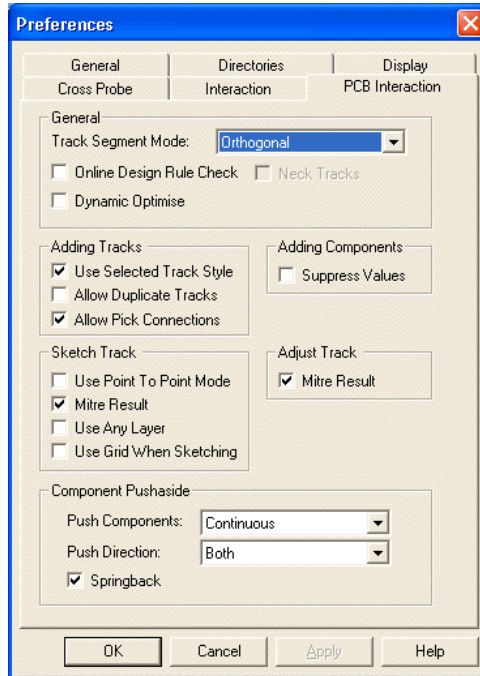


There are corresponding settings on **Preferences** dialog and **PCB Interaction**.



Preferences Dialog Changes

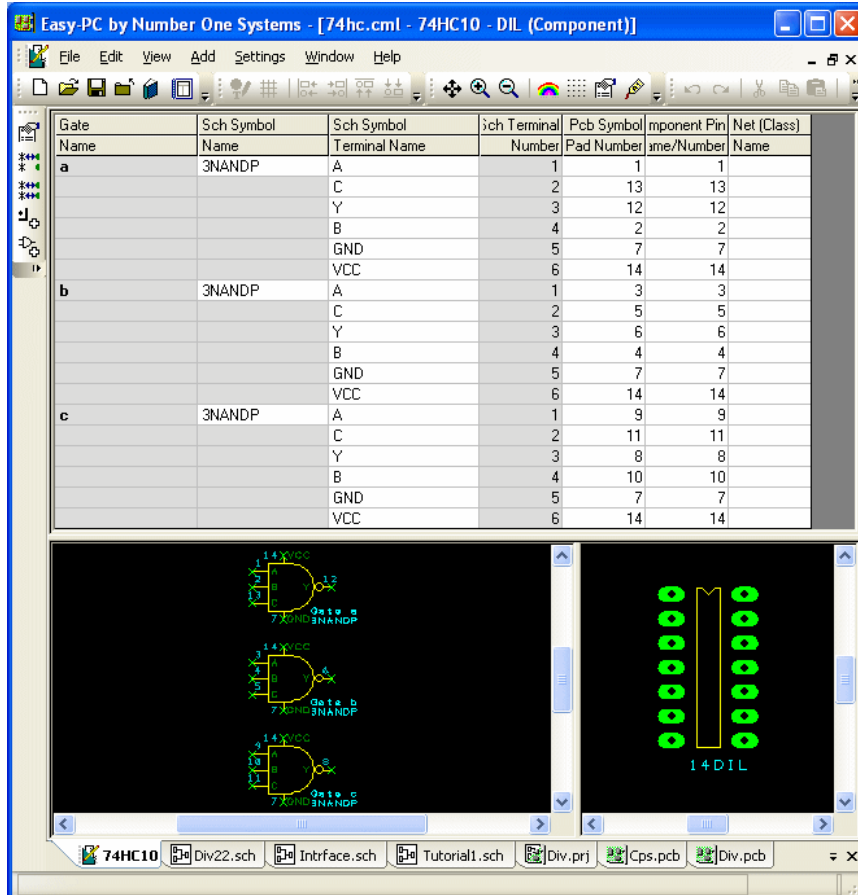
The **Preference** dialog has been split into more tabs to make it easier to read and use. Changes and additions to the Preferences dialog are discussed elsewhere.



Library Dialog Changes

Changes to Component Editor

The Component Editor dialog has undergone a significant change, not enough to be completely different so that you have to relearn it but enough to improve the functionality.



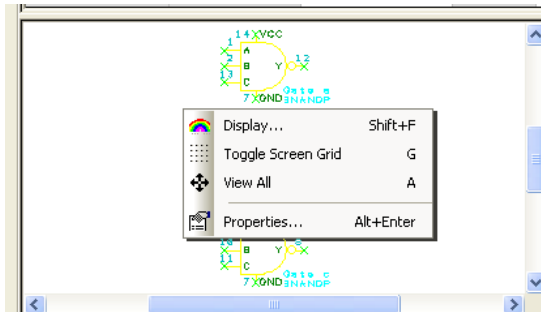
Gates Listing

You will initially notice that the grid has been rotated so that each gate now appears down the list rather than being across the page, gates are numbered a, b, c etc. All cells not greyed out are editable fields.

Gate Name	Sch Symbol Name	Sch Symbol Terminal Name	Sch Terminal Number	Pcb Symbol Pad Number	Component Pin Name/Number	Net (Class) Name
a	3NANDP	A	1	1	1	1
		C	2	13	13	
		Y	3	12	12	
		B	4	2	2	
		GND	5	7	7	
		VCC	6	14	14	
b	3NANDP	A	1	3	3	

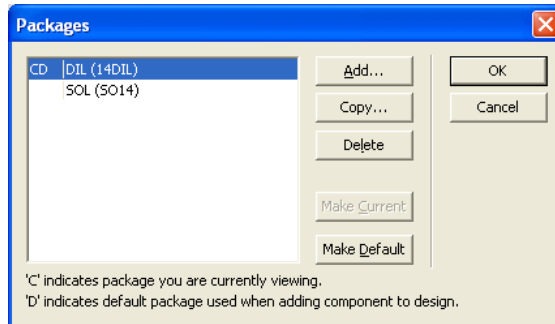
Shortcut Menu Usage

Within both the Schematic and PCB Symbol Preview windows, you can use functionality available on the shortcut menu.



New Edit Package Dialog

This dialog is used to add or edit packages to the component. Alternative PCB Symbols can be added here as well. The order in which they appear is how they will be added in the Schematic editor when transferring from Schematic to PCB designs. Other packages under the top most item will be available as alternatives using the Properties dialog.

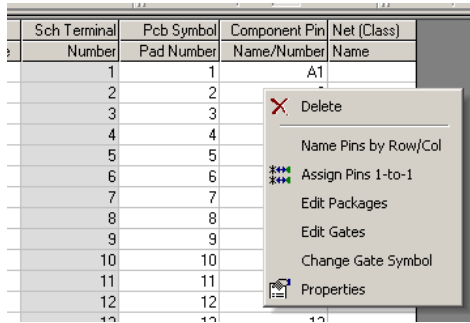


New Net (Class) Names Column

A new Net (Class) Names column has been added to the right side of the grid. This is used to predefine net names or net class names where a known signal name is required. This may be useful for pre-assigning power and ground signal names to gates or for use with specific power and ground symbols, such as +5V and GND for example.

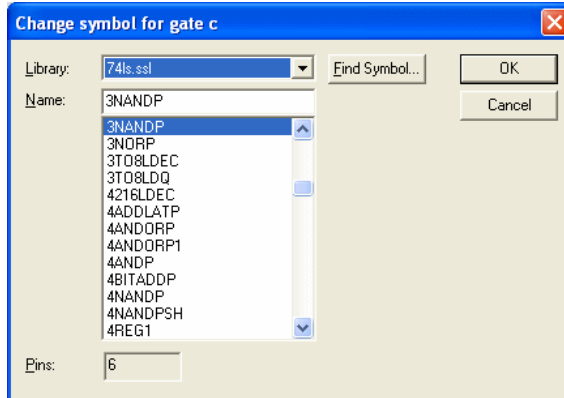
New Shortcut Menu in Component Grid

A new shortcut menu in the Component Grid allows you access to additional functionality.



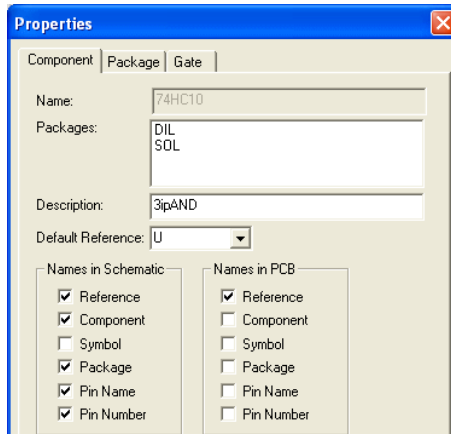
You can also use this dialog to **Assign Pins 1:1** (PCB symbol pad numbers – Component Pin numbers). The **Name Pins by Row/Col** option is used to automatically number pins which require alphanumeric names/numbers (see be in the section on *Alphanumeric Pin Name Support*).

The Change Symbol Gate option enables you to add more Schematic Gates to the Component. This is also available from the Component Editor toolbar.



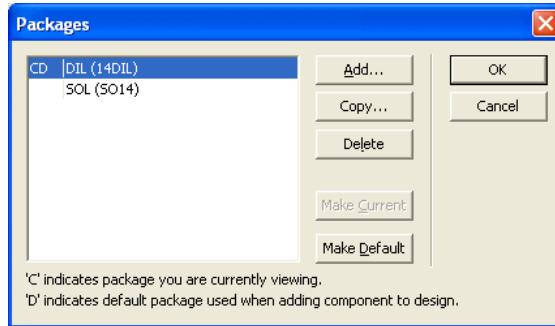
Properties

The **Properties** option opens the Properties dialog for this component.



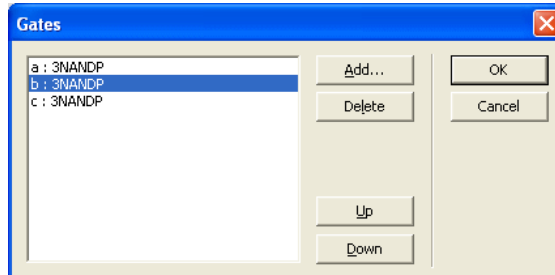
Edit Package Option

The **Edit Packages** option allows you to define package alternatives for this component and to set the package default, that is, the one that is used initially when you use Add Component in your design.



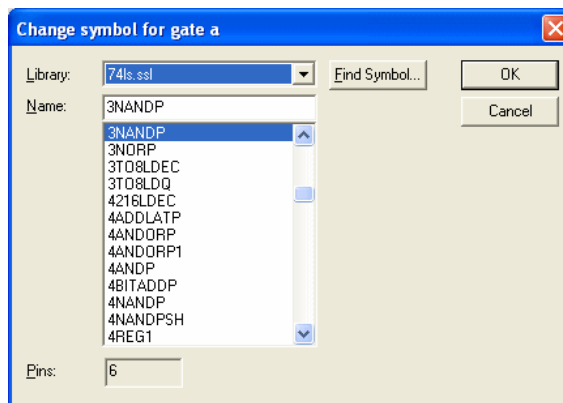
Edit Gate Option

The **Edit Gates** option enables you to add or remove gates from this component and to change the order in which they appear in the component grid.



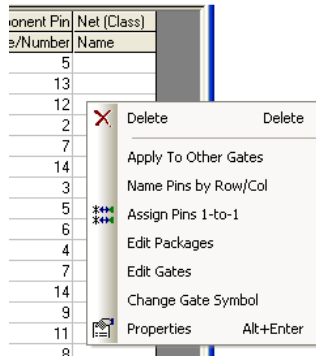
Change Gate Symbol

The **Change Gate Symbol** option is used when you need to add or replace a gate symbol for another one.



Apply To Other Gates

This option is used to apply the name in the selected cell to all other cells where the gates are the same.

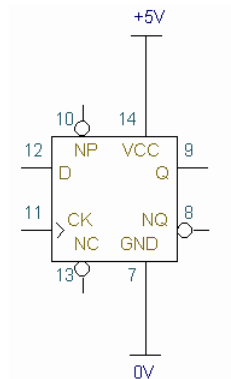


Library Manager

The **Library Manager** now remembers last selected item and 'topmost' item in the contents list, this means it will return to the same items next time you open the dialog.

Signal Components

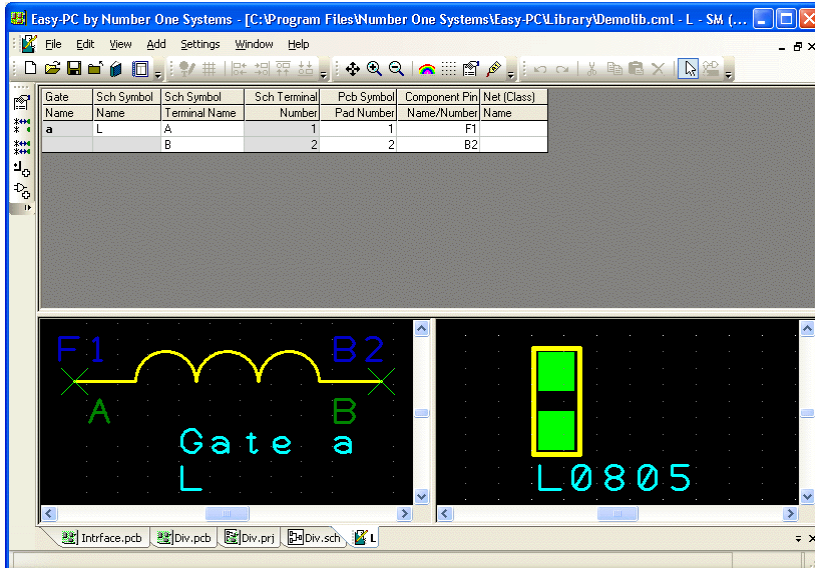
Components which represent pre-defined signal connections such as power and ground, can be defined more easily in the component library.



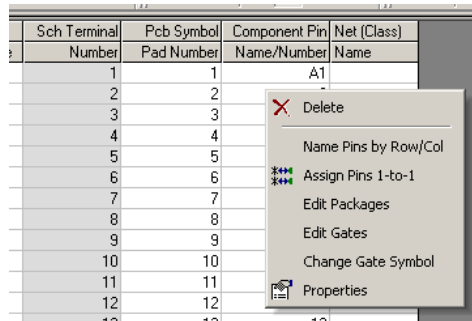
Within the Component Editor, the component grid now has an additional field for defining the **Net (Class) Name** for each pin on a component. This can be a Net name, or enclose the name in parentheses to make it a net class name (hence the column title). This is used to auto-connect this pin to the given net / net class when the component is added to a design. This makes the addition of these components much quicker when used.

Alphanumeric Pin Name Support

Alphanumeric pin names are often used to identify pad numbers in the PCB design rather than pad numbers. Typically, pads on devices such as BGAs (ball grid arrays), or some connector types they would use pad names, such as A1, A2, A3, B1, B2, B3 etc. On BGAs these also represent columns and rows of the pads to make pad location easier.

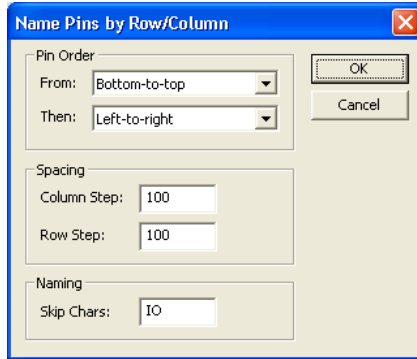


Within the modified **Component Editor** dialog, you can now specify the **Component Pin Name/Number**. On normal components, this would be same number/name as the PCB Symbol number.



By selecting in the Component Pin grid, new shortcut menu options enable you to define the column.

The **Name Pins by Row/Column** option is used to assign pin numbers to grid based components such as BGAs when using the **Component Editor**.



Pin Order

These controls allow you to choose the direction in which the pins will be numbered. The settings you need to choose here will depend on the order in which you have numbered the pins in your PCB symbol. With the settings shown above, the pins would be named starting at the top left, then stepping rightwards across each column to the end of the first row, then moving down the rows.

Spacing

These controls allow you to specify the step between the pads that constitute each row and column of your PCB symbol. The initial settings will be automatically determined by examining the arrangement of pads on your symbol, but depending on the order in which the pads were created you may need to adjust these values.

Naming

This control allows you to specify particular characters that will be skipped over when creating the pin numbers.

The results

Once you have set up the dialog controls and clicked OK, the pin numbers will be automatically created and added to the corresponding cells on the pin assignment grid of the component editor.

The first pin will be numbered A1, the second A2, and so on until the end of the row or column is reached. The next row or column will start with B1, and so on until all the pins are numbered.

Add Circle/Add Rectangle Changes

Behaviour of the **Add Circle** and **Add Rectangle** options is now consistent. Starting a shape with no keyboard keys pressed down still does corner-to-corner shape.

Starting a shape with the **Ctrl** key down produces a centre-to-outside shape.

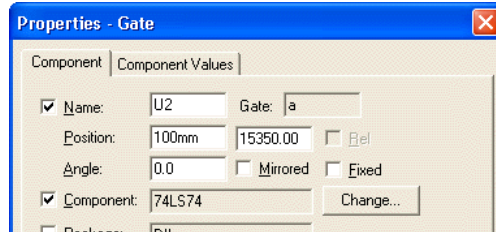
Starting with the **Shift** key pressed down, the **Rectangle** shape is constrained to be **square**.

Unfortunately this reverses the default behaviour of the **Add Circle** option, but was thought it better to be consistent.

Ctrl and **Shift** together can be used to constrain the Rectangle shape to a Square and starting from the centre.

Coordinate Edit Boxes on Dialogs

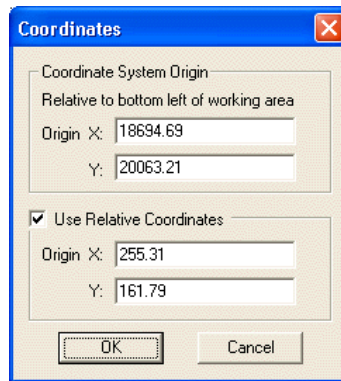
All coordinate edit boxes will now accept unit name after the number. Thus you can enter 16.5mm even if the current units are thou. Typed value is converted into equivalent current units and then applied.



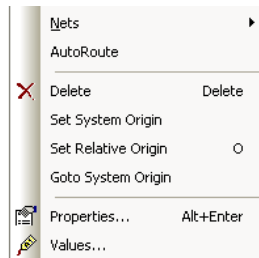
User Defined Coordinate Origin

You can now move the system coordinate origin to anywhere in the design. So if you need to, you can define it to the lower left hand side of the board outline or board centre for example.

The origin can be moved using the **Relative Coords** dialog on the **Settings** menu or interactively using commands on the shortcut menu.



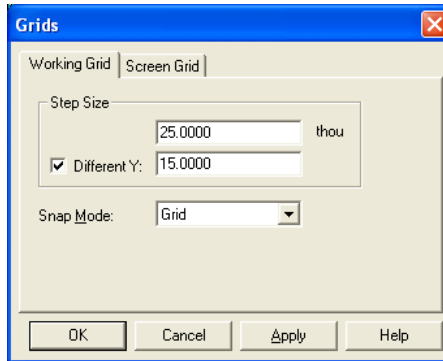
Shortcut menus for selected design items allow you to set the and go to the **System Origin** and go to the **Relative Origin**.



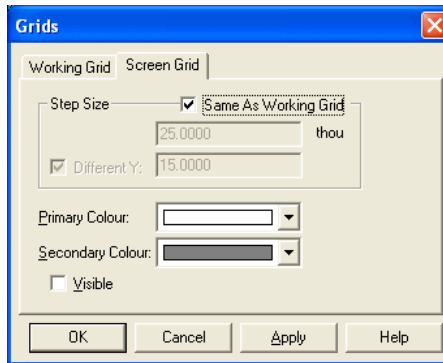
Set Origin with an item selected now sets the origin at the position of the item.

Asymmetric Grids

Grids can be set so that the X and Y grid positions can be different. From within the Grids dialog on the **Settings** menu, there is a **Different Y** check box to change the Y grid settings.

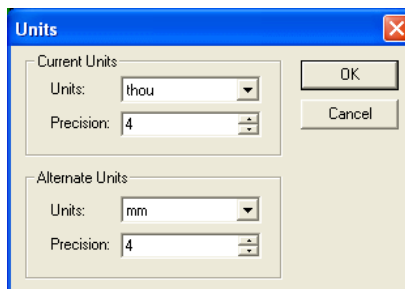


On the **Screen Grid** page of the Grids dialog, you can now make the screen different or the same as the Working Grid. Like the Working Grid, you can also make the Screen Grid use a Different Y grid.



Changes to Units Dialog

The **Units** dialog now has two unit settings, so you can choose the pair of units (and their precision) instead of setting just **thou** and **mm**. There are also additional unit types available for **cm**, **Mils** and **Inch**.

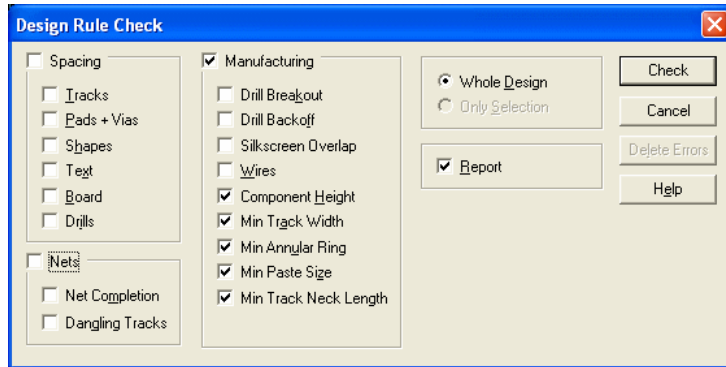


Care must also be taken when switching units to ensure that the **Precision** of the units is changed accordingly. For example, Imperial units in **thou** would normally be defined to 0 (Zero) decimal places, while Metric units in **millimetres** would be defined to 2 or 3 decimal places because of the rounding and accuracy required. This is especially important for the definition of drill sizes.

One final important point for unit precision, be careful if changing between the unit types on the same design. It is possible to sometimes define an item as say 12.99 and to round the unit precision up to 0 decimal places which then shows 13 as the displayed size. But because the value has not been changed, the value is still 12.99 as the stored value and the value which is then used throughout the design for all operations.

DRC Changes

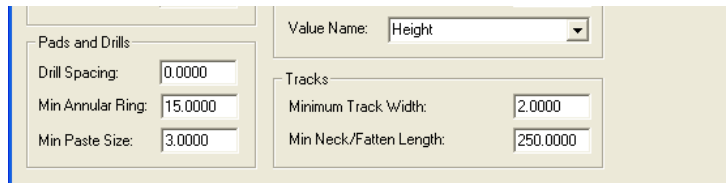
There are additional checks under **Manufacturing** on the **Design Rules Checking** dialog to check.



New Checks

- Minimum annular ring (drill getting too close to outer edge of pad)
- Minimum paste size (pad undersize causing paste pad to be too small)
- Minimum track width (track segment too narrow)
- Min neck length (necked/fattened track segment too short)

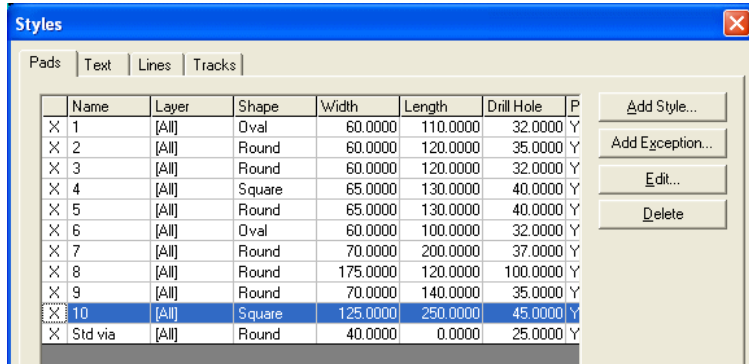
There are additional boxes on the **Spacings** dialog to set these values.



Changes to Styles Dialog

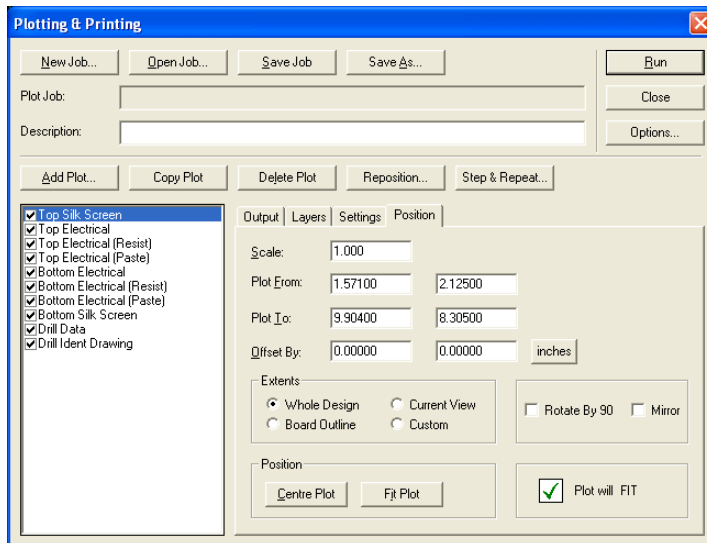
The **Style** dialog has changed, entries in the various styles grids are now sorted by name.

The dialog will also pre-select the style corresponding to the first appropriate selected item in the current selection (first line-style for selected components, etc).



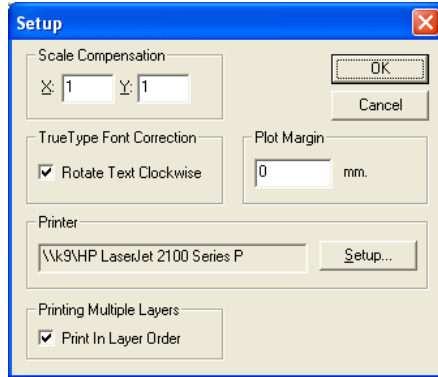
Plotting & Printing Changes

The plot **Position** tab (**Plotting & Printing** option) has changed, the 'extents' buttons are now radio buttons to show which part of the design you have selected for plotting, and there is a new button for 'board outline' which sets the plot extents to the extents of the first board outline in the design.



Windows Setup Dialog

An extra checkbox has been added to the **Windows Setup** dialog in **Printing & Plotting**, this allows you print multi-layer plots in layer order.



Plated/Non-plated Drill Hole Support

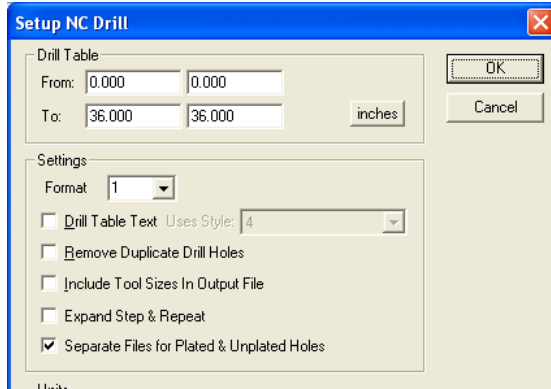
The Pad Styles dialog on the Settings menu can define both **Plated** and **Non-Plated** through hole types. There is a check box to define that a hole is **Plated**.



This plating status is used when outputting an NC Drill output file and a Drill Ident Drawing.

NC Drill Output

To further support the use of plated and non-plated through holes, the **NC Drill** output from within the **Plotting & Printing** option now supports both plated hole types.



A new checkbox has been added to the NC Drill setup dialog for **Separate Files for Plated and Unplated Holes**. With this checked, a separate drill file for un-plated drill holes is generated with **(Unplated)** appended to the file name. The report also lists the two sets of drills separately.

An example report is shown below:

NC Drill Output

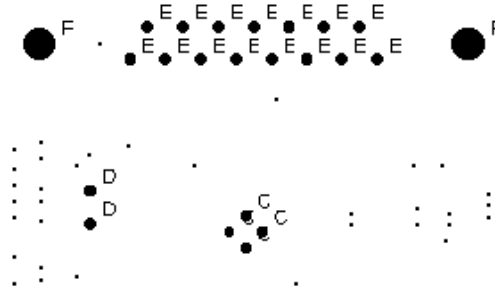
```
-----
Plated drill holes
-----
"C:\Designs\Fsk - Drill Data.drl"
Tool          Size          Count
-----
T000          00.032 in          6
T001          00.035 in          10
-----
All                          16
-----
```

NC Drill Output

```
-----
Unplated drill holes
-----
"C:\Designs\Fsk - Drill Data (Unplated).drl"
Tool          Size          Count
-----
T000          00.032 in          22
-----
All                          22
-----
```

Drill Ident Drawing Output

Drill ident drawings or drill drawings as they are also known, are an optional output file that can be produced. These are supplied to your board manufacturer and are used to indicate the position and type of drill holes within the design. Typically, where you have used mixed drill sizes, you would need to indicate these to the manufacturer so that the correct sizes are used.



A new Drill ID Setup dialog enables you to setup the drill hole and letter/shape association. When the drill ident drawing is produced, holes and sizes are indicated by shapes and letters, this makes the hole types easier to identify. The setup table enables this association to be defined.

Drill ID Setup

Drill ID Text

Use Text Style: 4

Offset Text: X: 50.0000 Y: 0.0000

Generating Entries For Drills Not In Table

Symbol Shape: Round

Symbol Size: 100.0000

Start Text From: A

Drill ID Table

Drill	Plated	ID	Shape	Size
50.0000	Yes	A	Round	50.0000
30.0000	Yes	B	Round	50.0000
32.0000	No	A	Round	50.0000

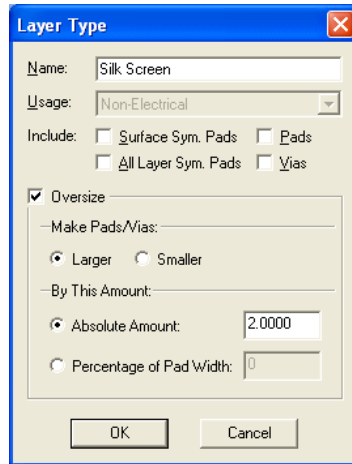
Add... Edit... Delete Open... Save As...

File: C:\Program Files\Number One Systems\EPC90\DrillIdent.did

Other options within this dialog allow you to define how and where the ids will be positioned once the plot is produced. It is normal to produce this plot as either a Windows printed plot, or as a pen plot (if you use a pen plotter). It can also be supplied as a Gerber file along with the other manufacturing plots and sent as part of the plot 'set'.

Layer type Oversize

From within the Layer Type dialog, you can now define the over and undersize required for when producing solder resist or solder paste mask plots. These are additional plots used during the manufacture process. There are additional controls on the **Edit Layer Type** dialog allow you to specify the under/oversize values to be used.



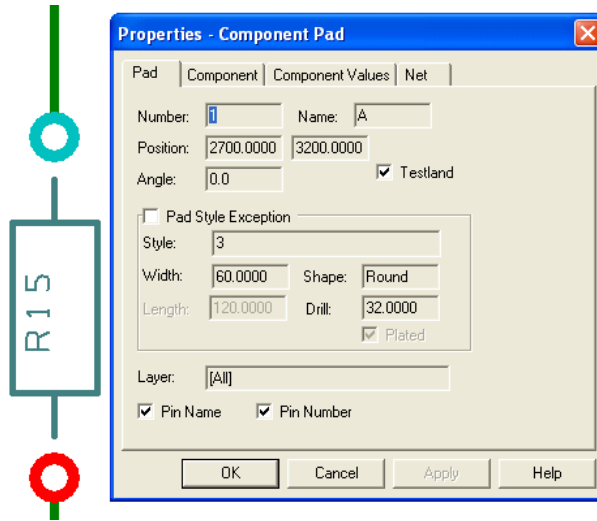
With the **Oversize** check box selected, you can make **pads** and **vias bigger** or **smaller** by the defined amount. The amount defined can be an absolute amount (using the current design units) or a percentage of the pad width.

Typically, for a Solder Resist plot, the pads in the plot would be specified slightly bigger (to allow for any mask movement or shrinkage), and a Solder Paste Mask would be specified to be slightly smaller to allow for the melting of the solder paste when the surface mount device 'leg' is soldered to the board. Before adjusting these values, check with your board manufacturer to see what he recommends.

Testland Support

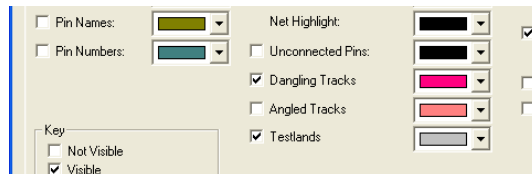
Testlands are used to define 'probe' points along a net within the PCB design. These probe points are used in automatic test equipment as 'landing' sites for test probes. These probe points can be any pad or via in the design.

Any pad or via can be selected and made into a **Testland** from within the **Properties** dialog.

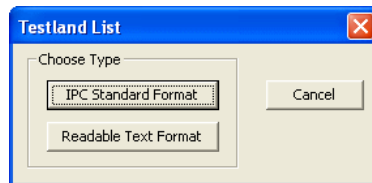


Other than pads and vias within the design, if you require ‘floating’ pads to be added to the design, then add these using the **Add Pad** option. You must also connect these to the required net by using the **Add Connection** option.

From within the **Display** dialog, the colour of the Testlands can be changed to highlight them in the design.



When you choose the **Testland List** menu option from the **Output** menu, it will open a dialog where you can choose between IPC format and 'readable' format.



The **IPC Standard Format** report will output an industry standard Testland netlist that manufacturers will use for automatic test equipment.

An example of the **Readable Text Report** is shown below:

```

Testland Report
-----

Report Written : Thursday, July 07, 2005
Design Path   : C:\Number One Systems\Designs\Laser.pcb
Design Title  :
Created       : 07/07/2005 09:22:30
Last Saved    : 07/07/2005 09:22:41
    
```

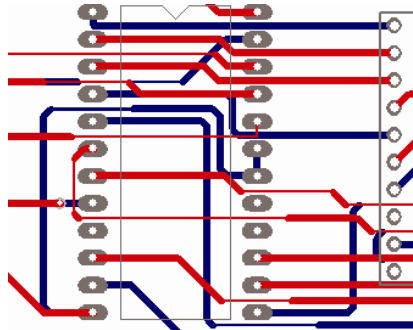
Editing Time : 28 min
 Units : thou (precision 4)

Type	Name	X	Y	Side	Net
Comp Pad	PL2.24	2850.0000	4050.0000	All	0V
Comp Pad	PL2.25	2750.0000	4150.0000	All	0V
Comp Pad	PL2.26	2750.0000	4050.0000	All	0V
Via	-	2850.0000	3925.0000	Both	0V

Counts	
Comp Pads	3
Free Pads	0
Vias	1
Total	4

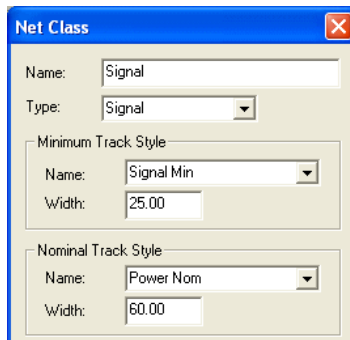
Track Fattening

There are new options to allow tracks to be ‘fattened’ up allowing more track copper to be laid down on the PCB.

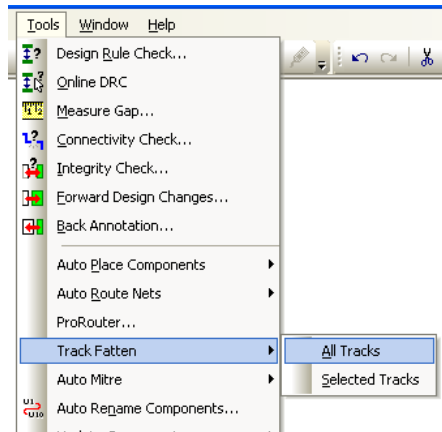


The **Net Class** track settings on the **Settings** menu are used to define the **Nominal** and **Minimum** widths.

Where the Nominal width used causes a DRC violation and with the Neck Track feature enabled, the option will ‘neck’ down to the **Minimum Track Width** defined.



New options **Fatten All Tracks** and **Fatten Selected** on the **Tools** menu and shortcut menu while editing tracks enable this.

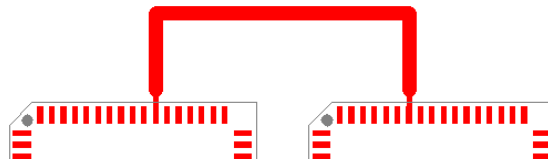


The minimum length setting can be defined in the **Spacing** dialog under the **Min Neck/Fatten Length** rule.

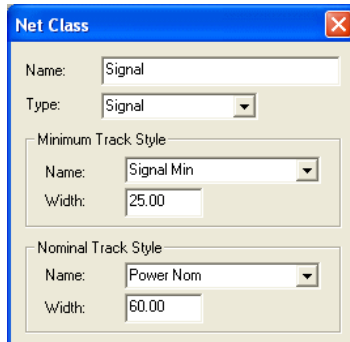


Automatic Track Necking into SMD Pads

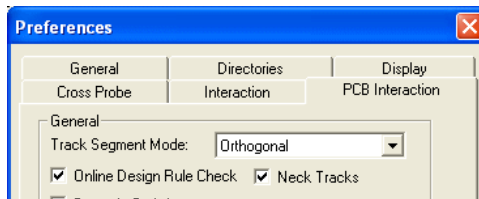
While editing a track, it is possible to enable automatic track necking if the 'fat' track is too wide to enter a pad without causing a design rule violation.



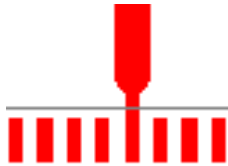
Like the Fatten Track option, the **Net Class** track settings are used to define the **Nominal** and **Minimum** widths. Where the Nominal width used causes a DRC violation and with the Neck Track feature enabled, the option will 'neck' down to the **Minimum Track Width** defined.



To operate in this mode you need two features enabled. You need **Online DRC** enabled and **Neck Tracks** enabled. Both of these features are enabled in the **Preferences** dialog under **PCB Interaction**.



The 'stub' distance can also be defined. This is the distance between the pad and the start of the 'fat' track segment.



This setting is defined in the **Spacing** dialog under the **Min Neck/Fatten Length** rule.

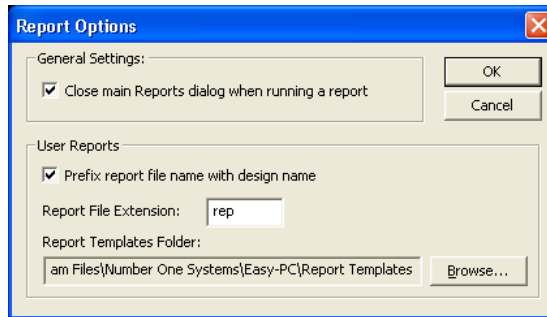


New Custom Reports Dialog

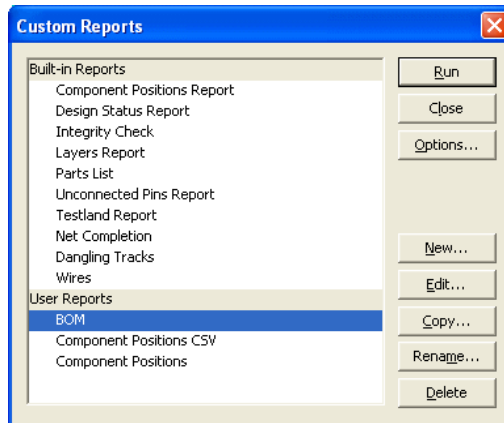
A new menu option (on the **Output** menu) in Schematic and PCB editors opens the **Custom Reports** dialog. From here you can run the supplied inbuilt reports, plus you can create user-formatted parts lists using a series of dialogs that allow you to choose which columns to include in the report and how the components should be grouped and sorted. This includes the ability to calculate totals of selected columns (e.g. Qty, or perhaps Cost attribute).

New default files are supplied as examples and are installed into a **\Report Templates** folder under the main Easy-PC installation folder.

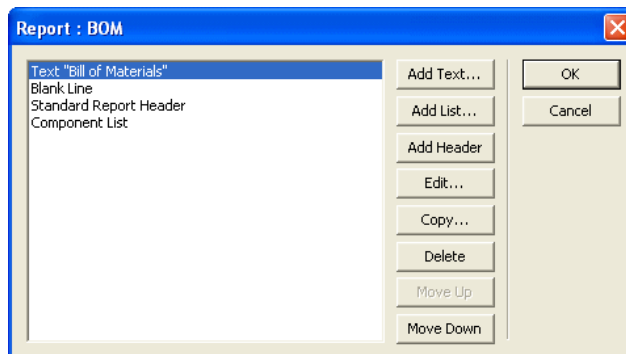
The **Options** button allows you to customise how the reports will operate and where they are located.



The main dialog displays a list of the **Built-in** reports and a list of **User Reports**.



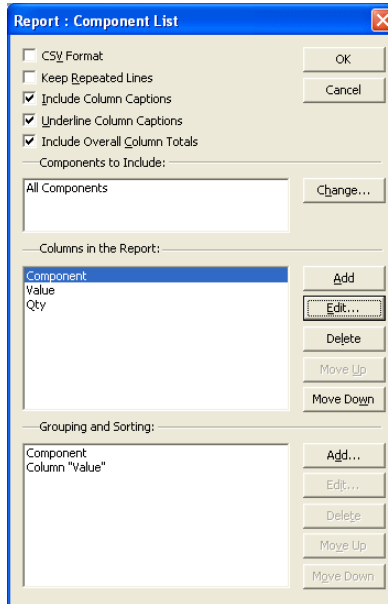
By selecting one of the **User** reports in the list and pressing **Edit**, or by clicking on the **New** button you enter the initial report editing dialog. A custom report can contain a number of elements, each of which adds to the report. The elements currently in the report are listed in this dialog, and you can use the buttons down the side of the list to change the contents of the report.



This dialog enables the format of the report to be defined. Each line in the list represents an action that will be taken when the report is run. The action is converted into a formatted line in the report.

A list of buttons enables you to edit the contents of a user-defined parts list.

This dialog allows you to edit the contents of a component list contained in a user-defined parts list. The settings on this dialog control how the output will be formatted, which components should be included, and in which order they are listed.

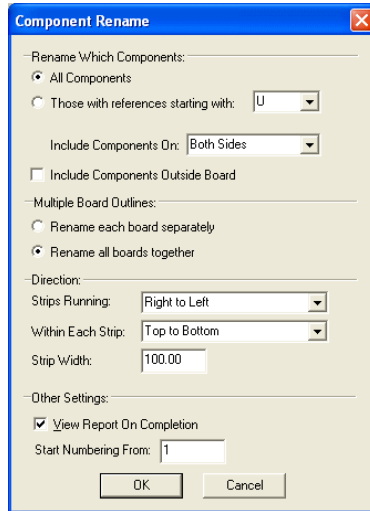


Each of the options listed in this dialog are fully described in the online help which is installed with the Easy-PC product.

Component Rename

From within the Component Rename option, if the **Include Components On: Both Sides** selected, it renames top side first then bottom side, instead of mixing the sequence of names on both sides.

The **Start Numbering From:** entry allows you to define the start number for Components rather than it always starting from 1.



The screenshot shows the 'Component Rename' dialog box with the following settings:

- Rename Which Components:**
 - All Components
 - Those with references starting with: U
- Include Components On:** Both Sides
- Include Components Outside Board
- Multiple Board Outlines:**
 - Rename each board separately
 - Rename all boards together
- Direction:**
 - Strips Running: Right to Left
 - Within Each Strip: Top to Bottom
 - Strip Width: 100.00
- Other Settings:**
 - View Report On Completion
 - Start Numbering From: 1

Buttons: OK, Cancel

Chapter 3. Easy-Spice

New Spice Update

The features detailed below are available in Easy-Spice version 2.0. Version 2.0 is like any other update and is a chargeable option.

Speed Increase

The Spice 'engine' itself has been updated to be significantly faster (between 50-150% simulation speed improvement depending on the circuit) and more accurate with convergence improvements.

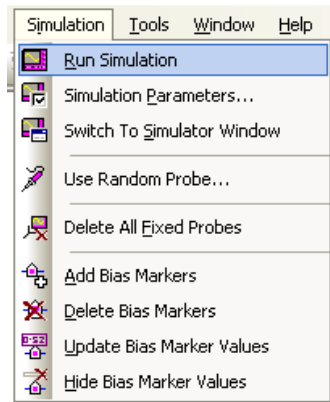
CircuitSim90 Benchmark Netlists

Of the industry standard CircuitSim90 benchmark netlists (57 in all), Easy-Spice now runs all 57 to the point at which they converge and produce an accurate result.

Easy-PC Interface Improvements

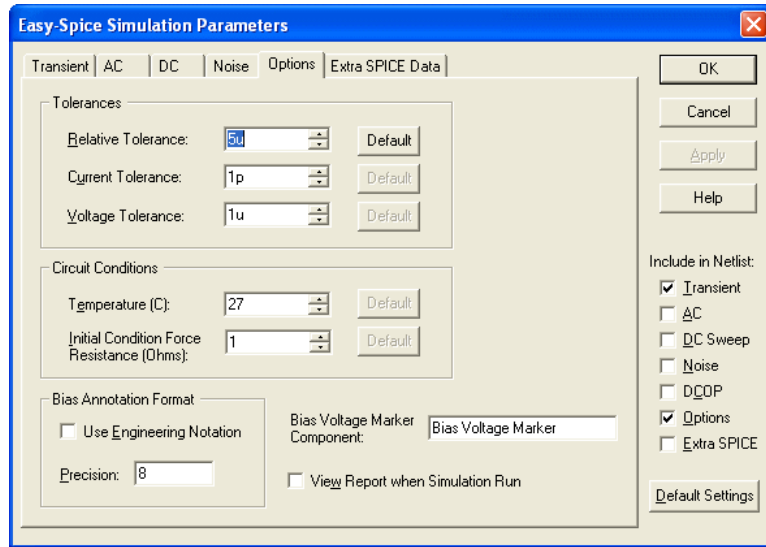
New Simulation menu

Easy-Spice version 2.0 now has its own menu – **Simulation**. On this menu is the **Run Simulation** command and **Simulation Parameters**, along with other options.



Once the design has been set up, the **Run Simulation** options is used to run the simulator.

During creation of the design before simulation, you can use the **Simulation Parameters** dialog to define which mode the simulator will run in.

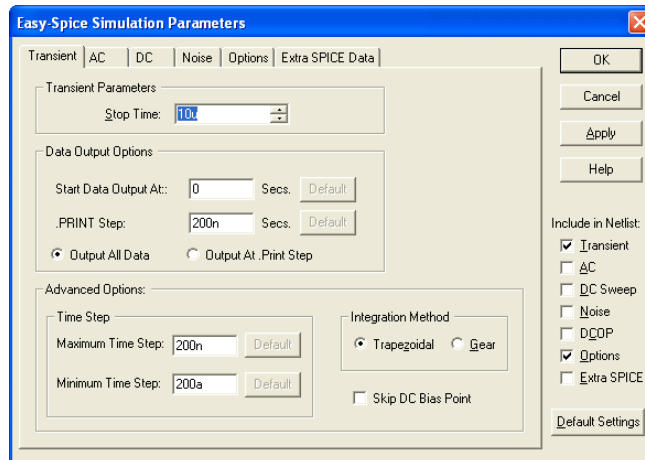


The **Simulation Parameters** dialog is now a tabbed dialog rather than being separate menu items. This dialog allows parameters to be defined without running simulator.

This dialog also allows multiple analysis modes to be used at the same time, i.e. AC, DC, Transient.

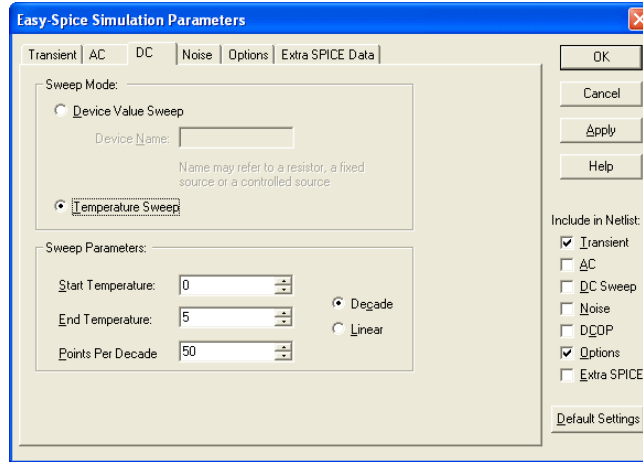
Extra Transient Analysis Parameters

The Transient Analysis dialog has been separated out and expanded. New options for Data Output Options and Advanced Options can be used.



DC Analysis

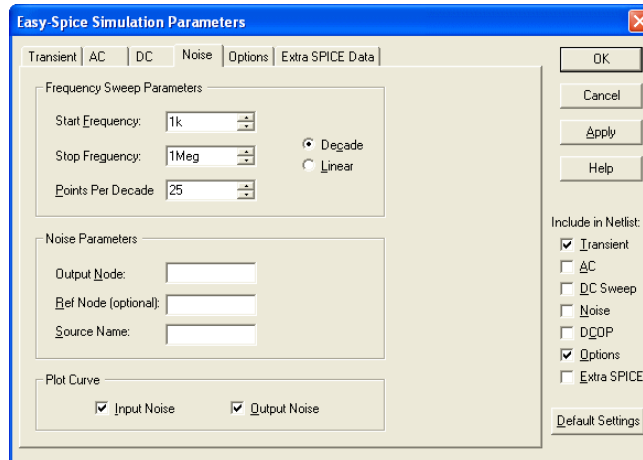
DC Analysis can now do a temperature sweep.



Select the radio button for the type of item you want to perform the sweep on. Choose between sweeping a device value or the global circuit temperature (using **Temperature Sweep**). The value edit boxes will change to accommodate the chosen sweep type.

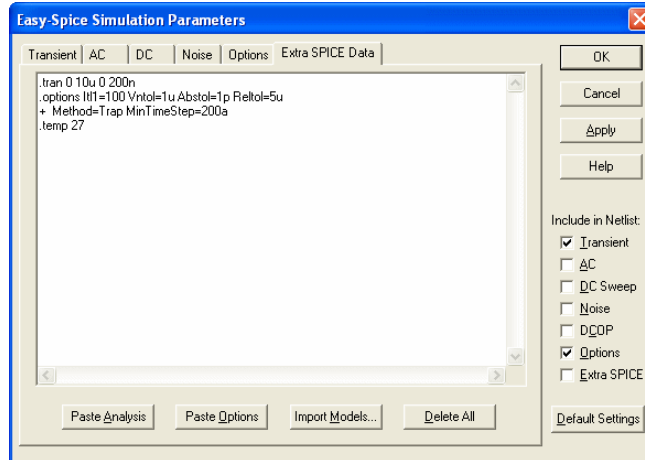
New Noise Analysis mode

Like AC analysis, **AC Noise** analysis is a small signal mode. The circuit is treated as linear about it's DC operating point and the contribution of all noisy devices to a designated output is computed. The total noise at that output is also calculated and optionally the noise referred back to an input source may also be computed.



Extra Spice Data

An additional menu option, **Extra Spice Data** now appears as a tab in the dialog.



This is used to include simulator controls, local SPICE models and other user supplied SPICE statements at the end of the SPICE netlist.

The data entered is held in a file with the same name as the schematic design or project but with a ".spi" file extension.

Type the simulator commands you wish to include at the end of the netlist. There are several buttons below the data box to aid you in this process. Use the **Delete All** to clear the data box. Note that All text pasted into the data will replace any currently selected text in the edit box.

Import Models

Use this button to paste a copy of all models and sub-circuits used in the design into the extra SPICE data.

The advantage of including all used models with the schematic design is so they can have their parameters altered locally without changing the model library, and to make the schematic design files complete. If the design files are passed to another designer, or brought out of archive, it should still simulate.

Clicking on the **Import Models** button will bring up the following dialog to allow you to choose to paste the used models directly into the data, or to paste a reference to a file containing the models.

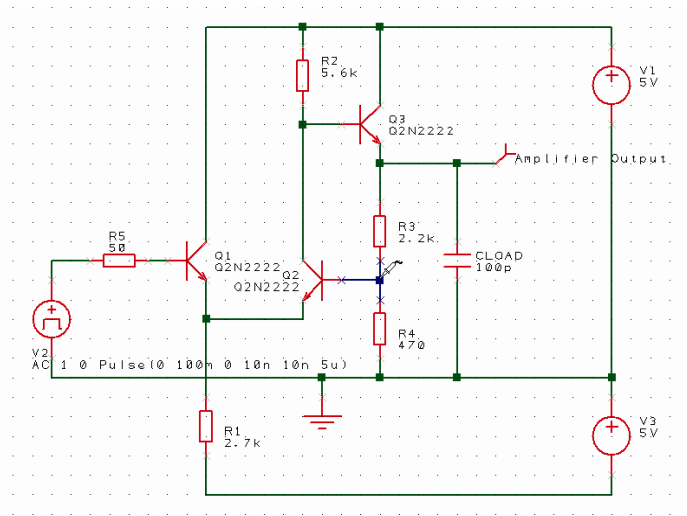


When **OK** is pressed the models are extracted from the Simulator model libraries and saved in a file with same name as the schematics design appended with a **.mod** extension.

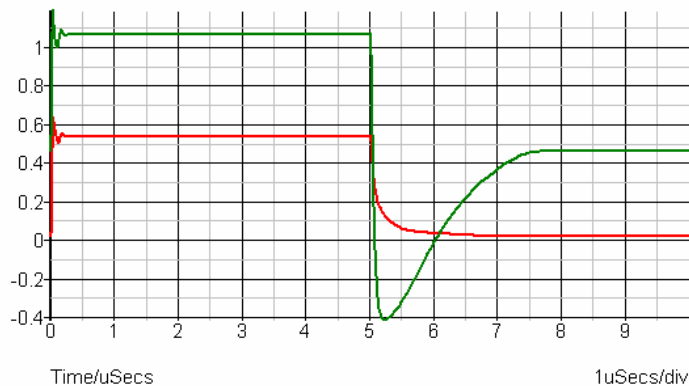
Random probing

Random probing is a mode that allows you to 'probe' around the design without rerunning the simulator each time. Use this interactive tool to plot the voltage and current on selected nets and device pins in the design.

This is a simple alternative to adding fixed voltage or current probes to the schematic design before simulation. The Easy-Spice Simulator will then generate graphs of the probed voltages and currents automatically when simulation is run.

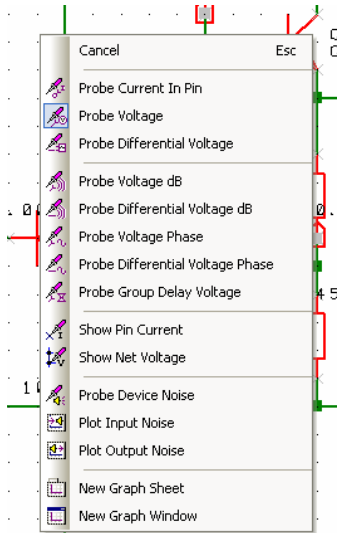


With **Random Probe** selected from the **Simulation** menu, a modal cursor with the probe image is displayed. You must have already run a simulation for the design for this mode to operate. On selection of a net, an extra response line is drawn in the graph window overlaying the existing graph.



Random Probe Types

Different probe types are available on the shortcut menu when in the Random Probe mode.



The first eight options are the standard probe types that match the fixed probes available in the SPICE component library.

Pick nets when using the **Voltage** probes and component pins when using the **Current** probes. The **Differential Voltage** probes asks for you to select two nets before plotting the voltage difference. The first is the "signal node" and the second the "reference node."

The new curve will be added to any existing graph if the X-axis has the same units. Otherwise, a new graph sheet will be created.

If plotting the dB or voltage phase and the AC Analysis used a logarithmic sweep, the x-axis will be logarithmic. You can change this to linear using the **Edit Axis** function on the simulator's graph window **Axes** menu.

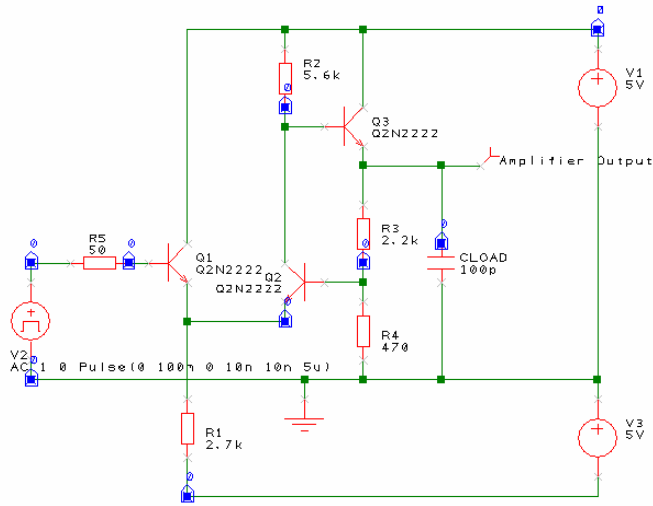
An additional option allows you to **Delete all fixed probes** from the design.

Note: You can plot the current into a subcircuit pin as well as those of primitive components. This means you can plot the terminal currents (and power) of power MOSFET and IGBT devices directly. Most other SPICE products require you to place a voltage source in series with the device pin to plot current in a MOSFET.

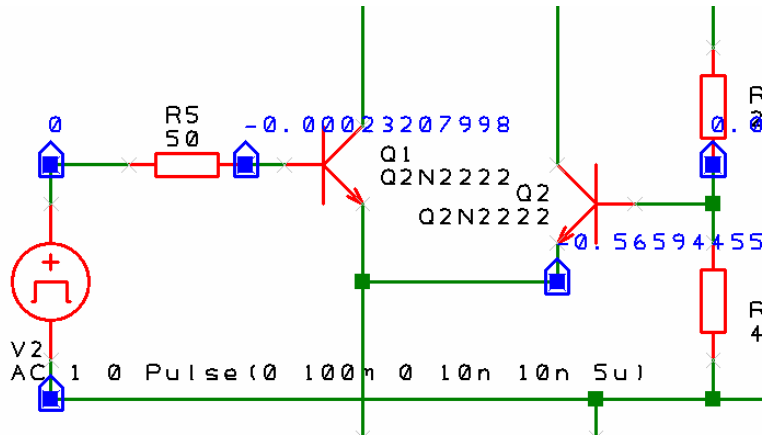
Bias Annotation

Bias Voltage and Current Markers are components used to annotate the schematic with the results of a **DC Operating Point** analysis, displaying the voltage on the net, or current on the pin, they are placed on. The values shown on the markers are updated each time you simulate.

The voltage and current markers can be added interactively and automatically to the design.



With the markers added, if the simulator is run, the results can be annotated into the design using **Update Values** option on the **Simulation** menu.



There are menu options to **Show/Hide Values** in design, **Update Values** and **Delete Values**. You can change the formats of the values using the **Options** page in the **Simulation Parameters** dialog.

Bias Marker Components

Bias Marker components should have a single pin. The voltage marker component should have the component value **SpiceBiasMarker = Voltage**, and the current marker component should have the value **SpiceBiasMarker = Current**. Both should also have the component value **SpiceValue = 0**, ready for receiving and displaying the analysis result.

The name of the component to be used for Voltage Markers should be set up in the **Simulation Options** dialog. This is to enable the automatic addition of voltage markers to know which component to use.

There are special components **Bias Voltage Marker** and **Bias Current Marker** provided in library SPICE.cml.

Adding Bias Markers

Individual bias markers can be added from the SPICE library using **Add Component**. Bias Voltage Markers should have their pin attached to the net you wish to display the value for, and Bias Current Markers should have a connection from their pin to the device pin you wish to display the value for.

Use the **Add Bias Markers** option from the **Simulation** menu to automatically add a **voltage** marker component to each net in the schematic design.

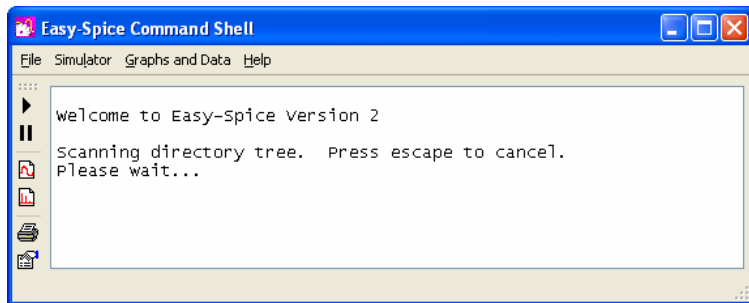
First all existing Markers on the Schematic Page will be removed. Then a Bias Marker will be added to all nets on the page.

If you have already run a **Simulation**, you may now update the values displayed on the markers by using the **Update Bias Marker Values** option.

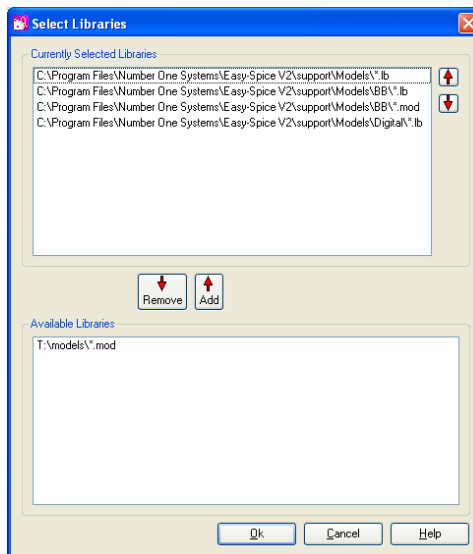
Simulation Interface

Drag n Drop models

You can now drag models or library folders onto the **Command Shell** window to add new libraries to Easy-Spice.



Dropping the folder from the Windows Explorer dialog onto the **Command Shell** window displays the **Select Library** dialog.



Much better Graph dialog editing

Below is a summary of the new features found within the new simulator interface:

- Can now resize all dialogs used
- Windows XP look when running on XP
- Toolbar on command shell
- Toolbars and graph legend panel are dockable

Graph Annotation:

- Curve marker, free text, boxed text and captions
- Add legend into graph for printing

Better Cursors:

- New cross hair cursor
- Position displayed on graph
- Improved cursor positioning

New features in the graph windows

- Now have fixed menus at top of program.
- Auto-vertical scrolling
- Highlight curves

Axes:

- Switch between log and linear after graph has been created
- Can adjust the height of digital axes
- Y Zoom improvement
- Re-order grids in Y direction (separate digital and analogue)

Saving:

- Graph to binary and recover later
- Graph to metafile

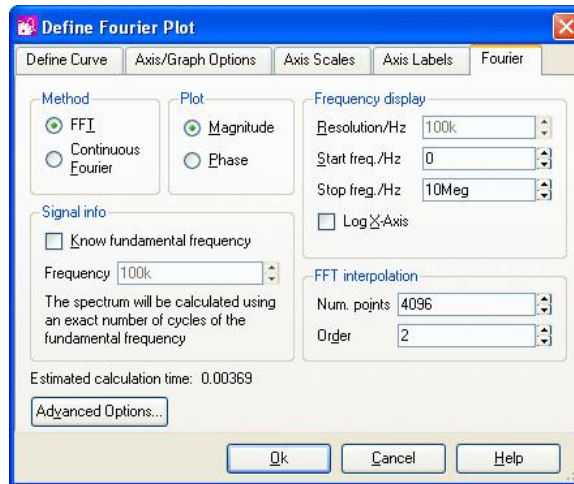
Define Fourier plot

A Fourier spectrum of a signal can be obtained in a number of ways. You have a choice of using the default settings for the calculation of the Fourier spectrum or you can customise the settings for each plot. The following menus use the default settings:

Graph menu and Measure Plot Fourier of Curve

Graph menu and Measure Plot Fourier of Curve (Cursor span)

The **Define Fourier** option on the **Graphs and Data** menu displays the following dialog:



You will see a dialog similar to that shown for **Plotting arbitrary expressions** will be displayed but will include a **Fourier** tab. Click on this tab to display the **Fourier** analysis options.

.PARAM Command

The PARAM command can now be used in library models and sub-circuits. It defines a simulation variable **for use in an expression**. Expressions may be used to define device parameters, to define model parameters, for arbitrary sources and to define variables themselves.

Support of .FUNC Function

Easy-Spice now supports the .FUNC function. .FUNC defines a function that can be used in a model or device parameter expression, a parameter defined using .PARAM or in an arbitrary source expression.

New Models

New models have been added to the Easy-Spice product libraries.

- 313 new models have been added to the IR, LTC and Zetex libraries.

New libraries have been added for the following:

- Infineon library containing 450+ models
- Magnetic Core libraries containing 50+ models

New Components/Symbols have been added to support the new models supplied.