Easy-PC For Windows

Schematic Capture and PCB Layout CAD

Evaluation Guide

Number One Systems

Welcome To Easy-PC For Windows

This is a brief guided tour of the evaluation version of our latest Schematic and PCB design tool, Easy-PC For Windows. The evaluation version is a fully functional version of the program without the ability to save your designs, and with the Pen-plotter, Gerber and Excellon N.C. Drill file outputs disabled (although you can see the dialogs which are used to control the outputs. There are no time limits on this evaluation and it can be removed at any time using the Windows [Add / Remove Programs] mechanism in the [Control Panel].

The evaluation program combined with this document will allow you to use all of Easy-PC's functionality whilst being guided through the basics of the system and it's capabilities.

Easy-PC really is the latest generation of PCB CAD with no other product currently as advanced or in-line with the Microsoft operating systems - Windows 95/98 and NT. And because Easy-PC has been written using the latest software techniques, object oriented code (C++) and under Windows NT as a <u>true</u> Windows 32 bit application, it can be easily enhanced to accommodate new demands in technology and customer feedback.

Hardware Requirements

Easy-PC doesn't require special or 'high powered' hardware to run on. A fast 486 or a Pentium processor machine is suitable provided is has enough spare disk space to load Easy-PC (about 25Mb is required), and enough RAM to ensure that Easy-PC isn't being paged in and out of memory, so about 20Mb minimum* for the evaluation is ok. For the fully functional Easy-PC product you would probably need about 32Mb minimum of RAM to get the best performance out of the application. The evaluation runs happily on Windows 95[™], 98[™] and NT4.0[™] but cannot run on Windows 3.1x - sorry! A CD-ROM and mouse drive are also required to load and operate it.

* 20Mb is required to use Windows 95/98 without significant performance degradation.

Installation Of The Evaluation Software

System Setup / Installation from CD ROM

To install the evaluation version of Easy-PC For Windows, first ensure that all other applications have been closed, and relevant data saved.

Insert the CD-ROM into the CD drive and wait for a short time. The default setup of Windows 95/98 and NT enables the *autorun* facility to be automatically used. If your system has the autorun facility disabled or it simply hasn't worked, select [Run] from the [Start] menu, and type D:\setup.exe (replacing the D: with the actual CD drive letter, if necessary, then <Enter>.

The Easy-PC For Windows Setup screen will now provide you with a number of choices for installation of products and for various utilities.

A Setup screen is now displayed, select the 'Install Easy-PC For Windows Demonstration' option and follow the instructions on the preceding screens.

If at any time you click on [Cancel], the window that appears gives you the chance to Continue or Exit the installation.

When the CD has been copied, messages will briefly display indicating that registry entries have been created, and icons copied. The selected folder will appear briefly as the icons are added, then a 'Setup Complete' window appears. You can choose whether to immediately launch the program, but in any case, the final step is to click on [Finish] to complete the installation.

System Limits

What Limits ?

Apart from maximum board size of 1m x 1m, there are **NO LIMITS** to any of the major design parameters. No component limits, No pin count limits, No net count limits, No layer limits, No track width limits and No pad dimension limits, basically the system is virtually unlimited.

When creating even the very largest designs, adding more memory to your computer immediately increases the capacity but not to unrealistic levels. Some of the largest designs we have in the software development centre will load into a 'standard' Multi-media PC bought in the high street.

Easy-PC operates with **NO Dongle** or other hardware protection (although it is protected using an authorisation code).

Easy-PC is a flexible and dynamic system. Because at Number One Systems we understand that not all designers have the same budget or same design requirements, we've created a number of product variants which will appeal to all pockets. These variants (except for Schematics only) have <u>exactly</u> the same functionality but are restricted to the number of component pins* allowed in the PCB design.

Easy-PC For Windows product variants include an Unlimited, 2000, 1000 and 500 pin limited variants*. This gives you the ability to choose the right product to suit your budget or design requirements. There is also a Schematics only version of Easy-PC For Windows available which can be used to drive one of our simulators or other PCB design packages.

* Component pins means the pads on a device placed on the PCB design e.g. two 16 pin DILs placed on the design constitutes 2 x 16 pins = 32 used component pins. This limited does not include vias but does include free pads and SMT pads.

In addition to the PCB and Schematic design products and autorouter, there is also a range of analogue, digital and electromagnetic simulators available. Also complementing the electronics design portfolio are a filter design package, a software implementation of the paper Smith chart developed by Bell labs, and a stock and kitting control program.

Users Of Previous DOS Versions Of Easy-PC

The evaluation version allows you to **load, edit and print out** on Standard Windows Printers, this includes the evaluation files and files created using the original DOS programs: EASY-PC, EASY-PC Professional and EASY-PC Professional XM. Once imported, these files can be temporarily edited but not saved. They can, of course, be saved with the full version of the program.

The Full version of Easy-PC for Windows also enables you to import and use components and component libraries that have been created with EASY-PC DOS, EASY-PC Professional or EASY-PC Professional XM. Old designs can be fully supported, maintained and developed with the new Easy-PC.

Remember, Number One Systems are the only CAD company who can fully read old Easy-PC designs and libraries without <u>any</u> effort at all. This feature alone saves valuable time taken creating and checking libraries, while designs retain 100% integrity in the new software, giving you confidence that your intellectual property is totally secure and maintainable.

What's New In The Latest Version Of Easy-PC For Windows ?

Like all software products, Easy-PC For Windows' development is moving at an incredibly fast pace. To make this guide more focused on your objectives - actually evaluating our product, we've put a description of all the latest changes into both the on-line help and into a file (NewInxx.wri) which can be accessed through the *autorun* Welcome screen or by locating it from the taskbar [Start] [Programs] [Number One Systems] folder. From this folder simply double click on its filename to view the file.

Getting Started

This manual is intended to take you on a short journey through the main features of Easy-PC For Windows. It is not intended as a training course but is intended to give to a 'taste' of some of the powerful features in the product and to get to started so that you can explore further features which are of particular interest to you.

We recommend that you print out this manual before starting to run the software. You will then be able to compare the printed instructions with the information on the screen without having to keep switching tasks all the time.

As this evaluation version does not allow you to save your work as you proceed, we have provided a number of example files representing different stages of the design process. These can be loaded at any point if you want to take a break and come back to the program later or if you lose your way!

Standard Installation Folder

As we explained at the beginning of this manual, and during the installation, the evaluation version of Easy-PC for Windows will install by default into C:\Program Files\Number One Systems\Easy-PC. Other folders are created underneath this for example files and libraries, ' **Examples'** and 'Libs' respectively.

Running The Evaluation Software

To start this evaluation you first need to run up the program. Click on the [Start] button on the Windows taskbar (along the bottom of your screen). Select [Programs] [Number One Systems] and [Easy-PC Demo]. The



application will now start running, wait a short time for it to fully load, this only takes about 10 seconds normally. Easy-PC is now ready to use. Incidentally, if you are familiar with the Windows 95 operation it is more efficient to create a shortcut on your desktop to Easy-PC especially if our program is one you use regularly.

Easy-PC uses the standard Windows graphical user interface conventions. The main menu items are arranged across the top of the window, with a set of toolbar buttons immediately below them. Simply placing the cursor over any menu item or toolbar button will display a brief description of its function on the status bar at the bottom of the window (called a 'tooltip').

This manual assumes that you are familiar with the main Windows operations and terminology.

It is also important to understand that Easy-PC For Windows has been strictly written using Microsoft standards which means that wherever possible menu items for 'common' commands or functions appear in the same place whether you're using Microsoft Word or Easy-PC. Some good examples are [Undo]/[Redo], [Delete], [Cut], [Copy] and [Paste] which are always located on the [Edit] menu.

Loading A Design

Load any example design file by dropping down the [File] menu and selecting [Open] from the Examples folder. We suggest the file ' div.pcb' as an uncomplicated example file.

Easy-PC Interaction - Using The Interface

Clicking, Double Clicking and Dragging

Objects are generally selected by single clicking on them, while double clicking usually selects the appropriate item and performs an operation on it. Single clicking the right mouse button displays a context sensitive menu in most situations i.e. additional operations relevant to its current mode. For example in Move you can also Flip (to the other side of the board), Rotate, change Properties etc.



Double click on a component pin starts a connection or track. Right click whilst adding or editing tracks or connections gives access to all of the properties and enables you to finish or cancel the operation. Track and connection stubs can be deleted using right click and selecting delete.

You are given a preview of, and are asked to confirm, all deletions.

Don't forget, if you delete something by accident, you can recover it by using Undo (shortcut: <Ctrl> Z)

Easy Zoom and Pan

Pressing the [A] key (All) automatically zooms to display the whole design. Place the cursor anywhere in the design and press the [Z], [U] or [P] keys. Zoom can also be selected using the numeral keys above the alpha keyboard. Try the Frame zoom option in the View menu -simply select [View], [Frame] then click and drag the cursor diagonally across an area of the design. This enables you to zoom into the area defined.

See how the [Z] oom [P]an and [U] nzoom keys work. Press the [A] Key and see how it [A] uto zooms to fill the screen with the current design. Try [View] [Frame] (or Ctrl F) then drag a box on the screen see how Easy-PC automatically zooms to make that box fill the screen.

During this you may wish to remove the Grid display from the screen. Simply press the <G> key to toggle the screen grid on and off.

Easy Move

Try "Click and Drag" on Components (click on any pad), Connections, Tracks and Text. See how easy it is to select and move them without having to set an editing mode. Pick the component, and while the mouse is pressed, drag the component on the screen. See how easily and sensibly the connections move as you move components, even the tracks move and remain electrically connected.

Easy Group Operations

Try clicking and dragging a box around a group of components and then moving the complete group together. Note how objects can be added or subtracted from a group using <Ctrl> Click on that object.

Easy Circuit Re-use

Try selecting a group of components and copying it to the clipboard, using <Ctrl> C. Paste it back to another area of the design, using <Ctrl> V. Notice how all the nets and components are automatically renumbered to avoid any conflicts. This can be done in SCM or PCB and can be done across sheets or designs making it very powerful and flexible. It also provides you an instant method for merging designs.

Easy Exit

If at any time you get lost or think the program has taken over there are a number of confidence checks you can make but remember the most important thing is not to press any keys unless absolutely sure! The first place to always look is at the bottom of the Easy-PC application at the Status bar Properties. This gives you information about the current mode and the item selected. Another good check is to look at the Toolbar just below the Menus. If a mode has been selected, such as [Add Track], the [Add Track] button will be selected. If you press the [Esc] key a couple of times at this point you are put back into the neutral [Select] mode (which is safe!).

Just a final note of the modes of the program. Easy-PC doesn't do anything drastic or destructive without prompting for [OK] from you before it does it, in any case Undo is always there (<Ctrl> Z).

What Should You Expect From A PCB CAD System ?

As the name suggests, computer aided design (CAD) means exactly that, designing with the aid of the computer. However, not all CAD systems are the same, and although they can provide you with CAD, their ease-of-use and operation for the 'human' aspects of the design leaves much to be desired.

This is where Easy-PC For Windows excels itself...

From the very outset of the concept for Easy-PC For Windows, the underlying factor for the development of the product was easy-of-use and usability for the user, enabling the design to be created with the least amount of effort in the shortest amount of time with an optimal tool set.

Using the product under the Windows operating environment you should expect to be using 'standard' Windows operation, such as drag and drop (without selecting a 'mode' first), double click to edit and a multiple document interface where different Schematics and PCB designs can be edited at the same time under the same application. Windows print drivers are essential for modern printers, colour printers, ink and bubble jet printers but still supporting the old faithful dot matrix printer. Windows graphics drivers are another essential prerequisite for the program to be utilising, no configuration problems and instant support for the graphics card installed in your PC. This means that if you invest in a high speed graphics card, Easy-PC For Windows will take advantage of it without any additional operations or setup.

With a modern system, the program should be a true 32 application to take advantage of the operating system, it must work to much less than .001" (1 thou/mil) to ensure true metric and imperial unit conversion. This is essential if using metric components or a mix both types. If your existing system can only work to a .001" precision then expect rounding errors, and incidentally, Easy-PC works to 1/10" Micron, so there is plenty of expansion room left and a system which can cope with all current and any predicted new technology !

Enough of the obvious, let's start the process:

The Design Process

The normal design process is to create a schematic, simulate it, correct any design errors, re-simulate the design, and then create the PCB design, checking connectivity and clearances as the design progresses. At any stage throughout the process printouts and status reports can be produced. On completion of the

PCB design, Photo plot and N.C. drill files (Numerically Controlled) can be produced and sent off to the board manufacturer. Although if your design process doesn't include Gerber plotting or NC drilling, Easy-PC is equally at home for other more DIY processes.

Schematics

Let's start the 'traditional' process with Schematics (Easy-PC doesn't need to be used with both it's own Schematics and PCB programs, you can mix-and-match applications from other vendors as well or use just Easy-PC PCB and design boards on-the-fly).

Open 'div.sch' using the [Open] icon on the toolbar, the file is located in the 'Examples' folder.

Hint: If you leave the cursor over any of the toolbar buttons for more than a second or so, the small tooltip label will appear giving you the name of the button.

Press the <A> key to display the complete schematic.

Before attempting to move a component, view the schematic and decide which one to move. Move the mouse over the component and press the <Z> key to zoom in, press it a couple of times until the component is bigger. You can now move the component by clicking anywhere inside its periphery **or** on one of its connection pins and dragging it to a new position. Note how moving a component automatically takes all of its connections and text with it - sensibly and neatly.

Multiple Windows

Easy-PC can open many designs simultaneously, including a mix of both PCBs and Schematics. The number is limited only by memory. This means many schematic design sheets for a project can be opened along with the corresponding PCB design and the library editors, all in the same application at the same time!!



Track Start: C28.2 End: U14.3 Style: nom signal Width: 10.00 Net: N0327 Layer: Top Electrical Abs 4565.76 7571.95 thou

Multi-Sheet Schematics and Projects



Adding Components to the Schematic

Components can be added by selecting the [Add Component] button on the toolbar. Then select the required component from one of the available libraries (Hit: the browse button in the Add Component dialog to see the demo libraries). Or, if a similar component already exists in the design, by selecting a similar component, right clicking and selecting duplicate from the context menu (use the <D> key).

Component Properties

While a component is selected, you can right click on it to display the popup menu and inspect or modify any of the component's properties including resistance, capacitance values etc. Note that the PCB package (Footprint) is

Properties - Comp	ponent X
Component Con	nponent Values
	[C1
Position:	9200 14325
<u>A</u> ngle:	0.0 Mirrored
Component:	Mega103 Change
F Pac <u>k</u> age:	USER
<u> </u>	Mega103
🔽 Pin Names	Pin Numbers
OK	Cancel <u>A</u> pply Help

specified at schematic stage ready for transfer into the PCB design editor. This component,

Designs can be project based and use multi-sheet schematics. This means that complex designs can be spit up into conveniently handled easily understood subsections. All the sheets in a schematic and the PCB can be open simultaneously using multiple windows as described above. The files are then all 'grouped' together during Translate To PCB automatically to create one PCB design.

Add Compo	nent		×				
🔽 Library							
Directory:	C:\Program Files\Number One Systems Browse						
Na <u>m</u> e:	Pcb.cml	•					
<u>C</u> omponent	14DIL	T					
<u>P</u> ackage:	DIL	•					
<u>N</u> ame:	U1		Add				
Ad	d to Component Bin 🏼 🖡	🗸 C <u>o</u> unt: 2	Cancel				
✓ Preview							
	1		4				
			T				
	2 😐	• 1	3				
	3 🔴	• 1	2				
	4 😐	• 1	1				
	5 😐	• 1	0				
	6 😐	9					
	7 😐						
	U1						

Schematic Symbol and PCB Symbol relationship enables an integrated library to be created which saves both costly errors and time trying to correlate the transition between Schematics and PCB where the design moves from 'logical' to 'physical'. It's also important because of the integrated design environment with both Schematic and PCB design editors co-existing in the same application.

Schematic Connections

Connections in a schematic can be edited and moved in a similar way to components. Simply click and drag on any connection and it moves where you want it.

Editing existing connections is all done by clicking and double clicking. Single click on a line segment and move the mouse to drag it at right angles to its length. Double clicking on a line segment inserts a new corner. Single clicking on a corner selects the corner so that it can be dragged to a new position. By clicking on the right hand mouse button while editing connections you can change many entities of the connection. This includes the actual editing mode itself. What this means is that the segment editing

mode can be switched from 90 ° right angles to one of 45 ° angles as well. Click on [Segment Mode] from the right hand context menu and select another mode. You will notice the mode change when adding corners to the connection.

Adding Connections To A Schematic

Connections can be added by selecting the [Add Connection] button from the toolbar. Left click on an existing connection and take the connection over to end at the target terminal of the component clicking on arrival. Single clicking while adding puts a bend in the connection, don't worry about making it neat. Editing is so easy it can be tidied up after the connection is made. If you start your connection at the component terminal and take it to an existing connection you will be asked if you want to join the nets - and to confirm the net name. If you've added a connection and ended in the middle of another connection you will see that a junction point has been added. This large dot differentiates between a connection actually making connectivity and one just passing over the top.

Connection Properties

While a connection is selected, right click on it to display a popup menu giving access to the properties of the connection including the Net Name and Net Class (select [Change Net]). A Net Class is used to identify the net type after Translate to PCB to enable track widths and additional clearances to be applied automatically by the optional autorouter according to the requirements of the connection. It also provides the initial track width when routing manually and a method of categorising Nets into 'type' groups. Signal tracks would usually be narrow, Power tracks should be fatter and any high voltage tracks should have much greater clearances than low voltage tracks.

Properties -	Track	×					
Track Segr	ment Net						
	Choose From All Nets In Design:						
<u>N</u> ame:	GROUND						
	Change Name Of Subnet Only						
<u>C</u> lass:	Power						
<u>G</u> uard	10.00						
spacing:							
OK	Cancel <u>Apply</u> Help						

Starting a New Schematic

In Easy-PC, this begins with the simple process of selecting and placing components and joining their pins with connections. If you drop down the [File] menu, select [Open] and then select ' **intrface.sch'** from the [File] [Open] dialog.

You will see a fairly complex schematic drawing built up in just this way. (Hit the <A> key to view all of it). However, in order to concentrate on the essentials, we are going to use a much simpler example.

Schematic Technology Files

Close down the window containing ' intrface.sch' and click on the [New] button on the toolbar. Select [Schematic Design] from the [New] list and ensure that the Technology File entry is ' default.stf'. Click the [OK] Button. Note that at this stage the effect of loading the Technology file has automatically set up all of the Display colours, Maximum Resolution/ Working Area, working and screen grids, whether working in Metric or Imperial units, the number of decimal places displayed on all co-ordinates, the standard sizes of all text connection dots (Pads), line widths, the standard widths for busses and normal connections and the names to be used for the standard Net Classes. - All with one keystroke!

Have a look through the [View] and [Settings] menu entries to see what has been set up. In the full version of the program you can create different Technology Files for different requirements. A Technology file can also be created from a design. This has the benefit that it can also contain the board outline and all of the net names used in the design and the net classes that were assigned to them. These names can be selected from a pick list in subsequent schematic designs dramatically reducing keyboard entry required and also ensuring consistency throughout the design department. How many ways do you know of naming Ground? Gnd, Ground, Common, 0v, 0V, Earth, Chassis etc.

The automatic assignment of the net classes to the net names in the same way as in the originating design ensures the standardisation of track widths and clearances for clock lines, data busses, power rails, high voltage connections etc. over the product range,

Nothing is Set in Stone !

Note that all of the parameters set up by Technology files can be changed at any stage throughout the design process. Technology files are there to help you and speed the design process not manacle you or restrict you.

Selecting and Placing Components

Click on the [Add Component] button on the toolbar. A library selection dialog will appear. Drop down the list of available libraries using the [Browse] button and choose 74LS.cml. You will now see a dialog box listing all the components included in this library. Select a 74LS00. Note that the PCB Package type is set in the Add Component dialog. Hitting the arrow at the right of the Package entry gives you the choice of either DIL (Dual In Line) or SOL (Small Outline Surface Mount) PCB foot print. Press the [Preview] button to show the component which will be added, while still in the Add Component browser.

Press the [Add & Place] button and the dialog boxes disappear and the four gates comprising the component appear in the drawing area. Move the cursor to a convenient position and notice how the four gates move with it. Now, click the left mouse button to fix one of the gates in position. There are now just three gates following the cursor around the screen. Place each of these four gates in turn. Easy-PC assumes that you want to continue placing more of the same type of component, so another four "mobile" gates will appear at this point. Simply press the right mouse button and select [Cancel].

Each of the gates can now be selected and moved individually. Temporarily unused gates should be deposited anywhere out of the current working area for use later on in the design. If



subsequently not used they should have their unused inputs tied appropriately to one of the power rails for safety.

Repeat this process to add a couple more components from other libraries. If you put down discrete components like a resistors or capacitors (R or C in the discrete.cml library), you can right click and select [Values] [Add] to add and set the component's value(s).

Making Connections

The components are connected together by connections joining the component pins. Click on the [Add Connection] toolbar button, then simply click on a component pin to begin a connection. Click anywhere on the drawing to put down a corner, and click on another component pin to end a connection. Use the <F> (Flip) key to toggle the XY to YX slope of the connection. Connections can also begin and end on existing connections, forming "T" junctions.

Each connection can be given a Net Name and have a Net Class allocated to it. In Select mode, single Right click on a connection or part of a connection drops down the context sensitive menu which has a [Properties] entry dialog. Selecting [Change Net] drops down the [Net Name] and [Net Class] entry dialog. Type in the required

Properties	- Track	<
Track Seg	ment Net	
	Choose From All Nets In Design:	l
<u>N</u> ame:	GROUND	I
	Change Name Of Subnet Only	I
<u>C</u> lass:	Power	l
<u>G</u> uard Spacing:	10.00	l
-,		l
		l
		l
OK	Cancel Apply Help	

net name (Or select an existing one from the Pick List) and either accept the Net Class entry or select an alternative from the Net Class pick list.

Note that allocating net names and net classes is <u>not</u> mandatory, but if the schematic is subsequently to be used with a simulator or the board layout is to be carried out by another person or autorouted then the allocation of net names and Net classes at schematic stage will dramatically speed up the design process.

Try dropping a Resistor on to a connection. See how the net is automatically split and that you are reminded to re-name one of the split nets. (Select the net and use [Properties] [Change Net] [Net Name] and check [Change Name of Subnet] only).



Becomes...



Busses

Busses can be created, combined and split with impressive simplicity. Both Open and Closed busses are supported with automatic net grouping, and bus terminal insertion on the busses, as well as the display of net names. Busses can be drawn easily and shaped to meet most design standards.

Busses have not been specifically detailed here but are available for you to try in the evaluation version.

Printing Your Schematic

At any stage you can output a print by selecting [Output], followed by [Windows]. A typical Windows printer dialog opens in which you can set up your Windows default printer. Uncheck Plot Pins and Pin Names/ Numbers to stop these from printing, if required. Essentially, Easy-PC For Windows supports all Windows printer drivers even colour printers, so we've provided an [All Colours Black] if a black and white only plot is required.

Output Windows	×
Plot Type: Artwork	Print <u>S</u> etup
Output To: HP LaserJet 6	Position Plot
🔽 All Colours Black	 ✓ Plot Pins ✓ Pin Names/Numbers
Plot Close	Cancel

The [Position Plot] button opens up a menu giving you full control over the printing process including centring it on the paper and scaling large schematics to fit the paper size.

[Current View] just prints the current screen. [Fit Plot] scales the drawing to make maximum use of the Printer capability - rotating the print as appropriate.

Translate to PCB

PCB Design: Design3

Component Library Directory

Translating a Schematic to a PCB

Having designed the schematic OR, if you wish, you can load any of the supplied schematic examples here. Select [File], [Open], and ' **fsk.sch'**.

Now, drop down the [Tools] menu and select [Translate to PCB].

The Translate to PCB menu gives you the choice of using different Technology files for the translation and hence the resulting Layout. These are displayed by hitting the Pick List button immediately to the right of the Technology file dialog. Note that the second browse button is to enable the selection of the technology file directory. The Library Directory must point to the directory containing the supplied library files.

2SIG2PWR.ptf is a Technology file as an example for Four Layer Boards with signal traces on the two outside layers and two internal power and ground plane layers.

Technology File Directory: C:\Program Files\Number One Syste Browse Name: Default.ptf 2sig2pwr.ptf 4sig2pwr.ptf Default.ptf Fineline.ptf Metric.ptf Ssided.ptf [None]

C:\Program Files\Number One Systems\Easy-P

X

Browse.

4SIG2PWR.ptf is a Technology file as an example for

Six Layer Boards with signal traces on the two outside layers and two internal power and ground plane layers plus an additional two inner signal layers.

DEFAULT.ptf is a fairly standard high-production-yield Technology file for relatively low cost Double Sided Through Hole Plated Boards. This includes generous track widths and clearances.

FINELINE.ptf is an example Technology file for fairly high density fine line work as used surface mount PCB's for example.

METRIC.ptf is a Technology file as an example for two Layer Boards but with units defined in Metric (all other technology files are based on Imperial units).

SSIDED .ptf is a Technology file for low cost Single Sided boards with very generous track and gap widths.

We will finish the Schematic section here in 'mid-air' but the design process is continued now into PCB design.

Designing In The PCB Design Editor

PCB Technology Files

Styles and Technology files work in many ways similar to Style Sheets in some Word Processors. Once set up for a particular manufacturing process or group of similar end products, they can ensure consistency of track widths, pad shapes and sizes, layer definitions and design clearances and, if required, even net names.

The PCB Technology file has set up the layers used, the display colour allocated to those layers, the autorouting bias for each electrical layer (for use with the autorouter), the maximum working area (and hence the maximum resolution available), the display grid, the working grid, whether working in Metric or Imperial units and the Precision (N $^{\circ}$ of Decimal Places), the Styles available for Pads, Text, Lines and Tracks, The design rule spacings, and the Track Width and Via Pad Styles allocated to the Net Classes. **All with one key stroke!** This produces dramatic improvements in productivity for professional users and minimises the learning curve for new users and light users.

Named Styles allow the user to set up sensibly recognisable Net Classes, track widths, pad types etc., for use throughout a design. For Example track styles can be given names like Clock, Signal, 50 Ohm, 75

Ohm, Low power, High voltage, etc., and the appropriate widths and any additional clearances set up for them. Technology files and the Net Classes included in them allow sets of these pre-defined parameters to be transferred between schematic and PCB and help ensure that the companies' standard board, track and clearance requirements are used by the layout engineer.

Sample Technology Files are supplied for:-

Single Sided designs Double sided Through Hole Plated designs Fine Line Surface Mount Four layer two power plane designs

On the full version of the program these can be edited and added to, as required.

Technology files can be created as new files, but can also be extracted from existing designs. These can also include the net names and net classes from that design to ensure consistency across a range of similar boards or products.

Continuing from the end of the Schematic section, where we left the Translate to PCB dialog waiting for a technology selection, select ' **default.ptf'**, the standard Double Sided Through Hole Plated PCB technology file and press the [OK] button. You may have already previously pressed the [OK] button, in which case the display will look like this:



A new PCB window will open containing all the components in your design – now in their PCB forms – connected together by direct connections. Components of similar types are stacked up in piles and all are connected by the connections or nets defined in the schematic. It's easy to see why designs at this stage are often referred to as "Rat's Nests".

The components can be picked up and moved by clicking and dragging on any of their Pins (or Pads in the case of surface mount components).

Zoom out a little and adjust the scroll bars until there is a clear area above the group of components, a quicker way is to use the [View All] icon on the toolbar or simply press the <A> key.

Adding A Board Profile

To add a board outline drop down the [Add] menu and select [Board] and [Rectangle], alternatively it is quicker to select the [Add Board] icon from the toolbar. Position the cursor in the top left of the clear area above the components. Hold down the left mouse button and drag the opposite corner until the rectangle seems big enough to contain the components. Don't worry about being too precise – it's very easy to make it larger or smaller later and to include corners, mitres or curves. Release the mouse button and a board outline will be left on the screen.

Now, select each component in turn by clicking on any pad and drag it into the board outline. While a component is selected, you can change its properties, including its angle and layer (for surface mount devices), and which of its labels are displayed, by clicking the right mouse button to drop down a popup menu. Drag all the components onto the board and adjust their orientations and positions until you are

happy with the result. The grid is set to 0.050" (50 Thou) by default. All components placed will currently 'snap' to this grid. You may wish to switch the display of the grid off, press the <G> key to toggle it off (<G> will also toggle it back on). Use the [Settings] [Grids] option to change the placement (Working) grid to your own value if required.

Your design could look like this example on the right >

Note how all of the connections or nets move with the components ensuring that the connections defined in the schematic remain true in the PCB. Easy-PC contains a powerful Net Optimiser to tidy up all these connections before we start to route them properly. The Net Optimiser is called simply by dropping down the [Tools] menu and selecting [Optimise Nets] and [All Nets].

The design is now ready for routing.

If you wish you can load **'firstu.pcb'** at this point, this is an unrouted example PCB file.

Converting Nets to Tracks

Zoom in as appropriate and choose a rats nest connection that is relatively short and double click on it near to one of the pads. This inserts a corner in the connection at the point where you clicked on it. Move drag the corner (by dragging it) to a suitable position and single click to fix it in position. Continue clicking to fix the next corner, and so on until the track is in the correct position. Use the right hand mouse button to display the popup menu which will enable you to change the characteristics of the track segments being added e.g. the track style and layer etc. If you change the layer, notice how the system automatically inserts a via for you. Finally, double click to release the track.

If you accidentally start another track from the end of one of those previously routed, press the <ESC> key on the keyboard.



Editing Tracks

Editing existing tracks is all done by click and drag and double click and move. Click on a track segment and move the mouse to drag it at right angles to its length. Double clicking on a track segment inserts a new corner as we have already seen. Single clicking on a track corner selects the corner so that it can be dragged to a new position. Double clicking on a corner converts in into a mitre (a 45° segment which cuts off the corner) that can also be dragged into and out of the corner.

Track layers are changed by selecting a track segment and right clicking to display the popup menu. Select layers and choose the new layer. Vias are added automatically

After a little practice you should be able to completely route a board like ' firstu.pcb' in around an hour.



Easy Editing

The track, component and connection editing has been optimised to provide maximum productivity and be very easy to learn. The ability to click and drag or double click and edit anywhere along a track, connection, or on any object is smooth, intuitive and fast.

Note that PCB components are best selected by clicking on any of their pads (as nets and tracks are always picked up preferentially).

Move a component and all of the tracks or connections move with it - sensibly.

Easy Mitres

Put a mitre in a right angle corner of a PCB Track simply by double clicking on the corner and moving the cursor. Edit a mitre by double clicking on the mitre and dragging it as required. Right angle corners can be mitred in a flash to shorten total track lengths and reduce reflections and radiation at high frequencies.

Easy Shapes

Circles, Rectangles and Polygons, filled or unfilled can be added on any layer and added to any net instantly. Shielding, Grounding and heat sink areas can be added in a flash. Circular and complex board outlines and components can be created quickly.

Easy Angles

Components can be rotated to any angle down to 0.1 degree either in fixed, user-defined steps or by a specified angle.

Easy Library Component Creation

The component creation methods have been greatly simplified to minimise keyboard entry and maximise ease of use.

Adding Copper Areas

Copper shapes for Ground plane or heat sink areas can be added on any layer using [Add] [Shape] from the menus. Complex shapes can be created using the Polygon option. The circle and rectangle options produce fast results. These can be added on any layer, shapes on electrical layers can be added to nets and are included in Design Rule and Connectivity checks. Shapes can be filled or unfilled.

Power Planes

Internal Power and Ground planes can be added either at the Translate to PCB stage by using an appropriate PCB Technology file (e.g. 2sig2pwr.ptf) or by adding the layers directly to the PCB, by using the [Settings] [Layers] [Add] menu. Give the layer a name, set the Type to Electrical, the Side to Inner, Bias to Power Plane, Usage to Electrical, and Net to the name of the net that you wish to connect to the plane. This can be selected from the pick list. Power planes are not shown on the screen. Tracks can be placed on power plane layers and they will be ploughed by the set clearance. Tracks on Power planes dramatically increase the impedance of the plane and should be used only when absolutely necessary. Thermal relief pads with set width and isolation gaps are produced at the post processing output stage of the power and ground plane layers.

Design Rule Check

You can check whether you have broken any design rule spacings using [Tools], [Design Rule Check]. This produces a printable report and marks the layout with the violations. After running DRC, pressing [T] (for nex[T]) and [V] (for pre[V] ious) will take you forward and backward through the errors on the screen. After correcting all of the errors (or altering the spacings), re running the DRC will clear the errors.

Integrity Check

[Tools] [Integrity Check] extracts the connectivity of the layout and compares it with the net list of the schematic. Thereby ensuring that the PCB matches the schematic. It warns of any differences and advises the changes required to bring the two into line. This should be done using a corresponding schematic and PCB design.

Connectivity Check

[Tools] [Connectivity Check] provides a check that all of the nets are 100% routed and warns if a net is split into two or more unconnected segments. [View], [Highlight Net] enables you to inspect appropriate nets and make corrections.

Design Status Report

[Output] [Status Report] provides a summary of the whole board including Components used number of drill holes etc. Have a look at the reports of some of the samples provided.

PCB-only Designs

Whilst most PCBs start life as a schematic design that has been translated, there are many occasion where it is useful to work from a sketched schematic or even no schematic at all. This is no problem for Easy-PC For windows - PCB designs can be created on-the-fly easily.

Simply select [File] [New], select an appropriate Technology file (if required), and place your components and tracks as before. Track widths can be changed on the fly using right click [Style]. Track Layer can be changed using right click [Layer]. Vias are inserted as necessary during routing.

Again, the board outline can be added using [Add Board] as can routing areas and additional free text.

Import From Other Programs

In addition to the standard import facilities for the DOS Versions of EASY-PC imports from other programs will be available soon. - If you have any specific requests please contact us. We can currently load netlist from OrCAD Schematics and IMT EDA Workbench, again from a netlist.

Printing Layouts

This evaluation can print out schematics and layouts to any printer whose driver is installed in Windows. There are no special drivers or setup, just use what is already available I your operating system. The full version of the program can plot to HPGL pen plotters and generate Gerber and Excellon manufacturing files.

Printing is started by dropping down the [Output] menu and selecting [Windows]. The first dialog displayed is used to select the layers to be printed. [Print Setup] is used to adjust the printer settings, while [Position Plot] sets up the positioning and scaling of the design on the printed page. Checking the [Fit Plot] checkbox will ensure that the design is scaled and, if necessary rotated, to the maximum size that will fit on the page. This gives you maximum flexibility to print the area of the design or

Display										×
 ✓ Board: ✓ Connections: ✓ Vias: ✓ Overlay:] I Sc Sc] Hig] Ne	reen Grid F reen Grid S phlight: t Highlight	Primary Secon	r: Constant	· ·	BackGrou Selection Pin N.	und:		OK Cancel
Layer	Displayed	Tracks	Areas	Pads	Shapes	Text	Sym. Pads	Sym. Shapes	Sym. Text	Errors
Displayed		Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes
[All]										
[Top]										
[Bottom]										
Top Silk	Yes									
Top Copper	Yes									
Bottom Copper	Yes									
Bottom Silk	Yes									

position the print exactly as you require it on the paper.

If you are sending the output to a colour printer, the colours of all elements will match those displayed on the screen. To change the colours use the [View] [Display] option, or for check prints you can also choose to make all the colour to black during printing by selecting the [All Colours To Back] option.

Generating Manufacturing Files

This is inhibited on the evaluation version of the software. Easy-PC For Windows supports formats for the following:

HP-GL pen plotters Gerber RS-274-D photo plotters Gerber RS-274-X photo plotters which includes the embedded aperture data Excellon NC Drill machine for drilling the boards Windows printers drivers DXF (AutoCAD format) files for connection to mechanical CAD packages

Output files can be generated for all aspects of the manufacturing process including, check plots, artworks, drill drawings, photo plots, NC drill files, silk screen, assembly drawings and board build drawings.

Please contact Number One Systems for sample files, if required or for more detailed information.

Easy-PC Help

If you need more detailed information on any of the functions offered by Easy-PC, simply select [Help] from the main menu, or press the <F1> key on the keyboard. You can either select a topic by double clicking it in the [Contents] tab, or search for a topic by typing the first few letters of a keyword in the [Index] tab.

Explore for Yourself

Now that we have taken you through the design process from schematic to PCB, it is time for you to carry on exploring the facilities of this program on your own. As save is inhibited, you will not change permanently any of the sample files provided or your own designs if you are a previous user of the DOS versions of EASY-PC.



Technical Support

For more detailed reference information on menu entries, etc. we recommend that you use the on-line Help system accessible via the [Help] menu on the main menu bar, or by pressing the <F1> key on the keyboard.

If you have any problems installing or running this program, you can contact your dealer or our Technical Support department by telephone on +44 (0)1480 382538, by fax on +44 (0)1684 773 664, or by e-mail to support@numberone.com.

How To Buy The Full Programs

Now that you've decided you like the program and want to buy it, how do you do this?

To obtain one of the fully enabled versions of Easy-PC For Windows, complete with a printed manual, full libraries and the ability to generate industry-standard manufacturing files, contact us at our International Office on +44 (0)1684 773 662 or by fax on +44 (0)1684 773 664

We can provide you with a price quote or Proforma invoice if one is required. Check out our new prices on the Number One Systems web site <u>www.numberone.com</u>, prices are the lowest they've been for years

including upgrades and competitive cross-grades. These new prices combined with new product variants make Easy-PC For Windows an affordable professional choice.

Competitive Upgrades

Special competitive upgrade discounts are available if you are transferring from the use of a product from another supplier. Please contact us for an attractive price quotation.

Over 45,000 Licences Sold!

Number One Systems has sold in access of 45,000 licences over the last 20 years into over 100 countries. The list of companies using one of our products is increasing all the time. With major company representation world-wide owning at least one licence from Number One Systems, our product portfolio is an integral part of the EDA and electronics CAD community in all facets of the electronics business be they large or small, professional or occasional users.

Web Site Access

www.numberone.com

Our web site keeps you fully up-to-date with current developments and new product information. The site is fully accessible and not restricted in any way, please feel free to visit and browse at any time. Also remember, that an off-line version of the web site is supplied on the evaluation CD-ROM of the software but will only be current on the day of the cut of our CD-ROM, there may be more current information on the live web site. Our web address is: <u>http://www.numberone.com</u>

Evaluation versions of our simulators and other products can be downloaded from our Web site at the address above or obtained direct from our sales offices or dealers.

Who Are WestDev Ltd?

WestDev are a software development company based at Bredon near Tewkesbury, Gloucestershire, in the UK. The team at WestDev boasts over 140 man years software development experience. Our aim at WestDev is to bring you outstanding value for money and realistically priced, power-packed products long into the future.

Although WestDev Ltd. wrote and owned the Easy-PC For Windows product they purchased the intellectual property rights to all of Number One Systems product at the end of 1998 and have since been marketing, selling, supporting and developing the complete product range from their own premises under the Number One Systems trading name.

We can be reached at the following address:

Number One Systems Oak Lane Bredon Tewkesbury Glos GL20 7LR UK

Tel: +44 (0)1684 773 662 Fax: +44 (0)1684 773 664 Email: info@numberone.com web site: www.numberone.com

Copyright (C) 2001 WestDev Ltd. All rights reserved.

Number One Systems, EASY-PC, Easy-PC, Pulsar, Layan, Analyser, Multirouter, Filtech, Z-Match, StockIt, StockWin, Easy-Router and all variants thereof are Trademarks of WestDev Ltd.

All other trademarks acknowledged to their rightful owners. E&OE