

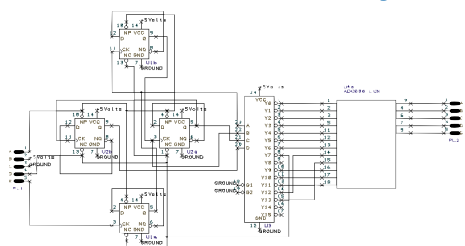
July 2005 - the amazing new Easy-PC Version 9 is released!

Number One Systems has developed a set of spectacular new features for version 9.0 to help you make your PCB design operation so much easier, enjoyable and more efficient. Once again, we have held down our upgrade prices for version 9.0 to remarkably low levels.

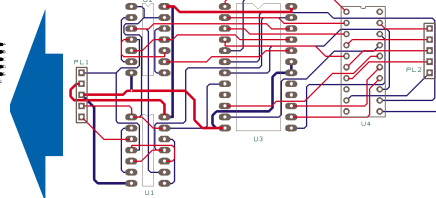
Just look at what's included when you upgrade to version 9.0...



NEW in Easy-PC V9.0 Reverse Engineer - PCB back to Schematic



The schematic is automatically created



Take the completed PCB and select "Reverse Engineer"

Automatically creates a Schematic from a PCB Layout

After laying out your PCB with components and tracks, at the press of a button you can automatically produce your Schematic with all the symbols and connections perfectly synchronised with your PCB design. Using new commands in version 9.0, you can leave Easy-PC to automatically place the symbols, or you can place them yourself, then automatically route in the connections. A little tidying to match your preferences and your Schematic is completed in a fraction of the time it would take you by hand.

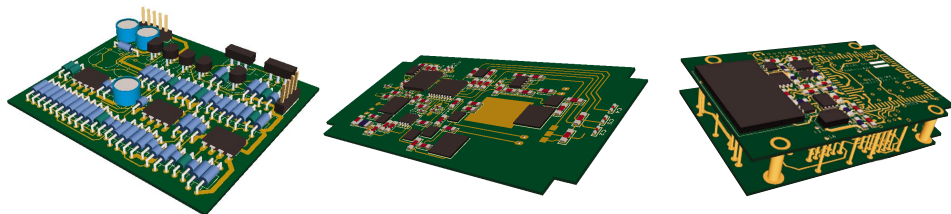
NEW in Easy-PC V9.0 3D Viewer with Height Checking

See a 3D view of your PCB and check height clearances

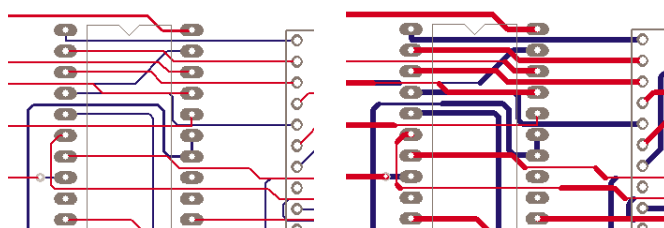
Visualise your PCB in a stunning 3D view to see just how it will look after manufacture. The image may be rotated through any axis and viewed from top, bottom, and with exaggerated perspective. With real component heights being used in the design, the 3D view will allow you to gauge clearances in the enclosure and to see new height check areas.

Component Height Checking

Component heights can be defined and checked against predefined values set up in the Spacings dialog. Height values can be set using areas within the design giving you access to 'localised' checking. A new option in the DRC dialog will check these values.



NEW in Easy-PC V9.0 Track Fattening

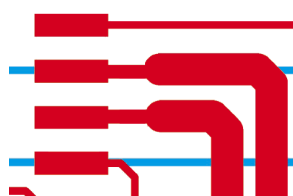


Before track fattening

After track fattening

Track Fattening automatically increases the thickness of tracks that have an alternative track width defined. Tracks are fattened where this can be achieved without violating the design spacing rules.

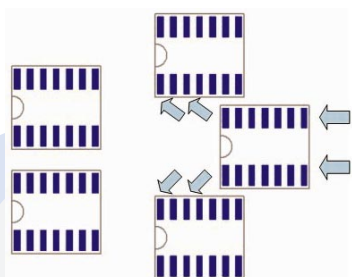
NEW in Easy-PC V9.0 Track Necking



Track Necking into SMD Pads

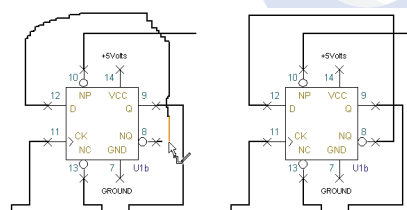
Where a 'fat' track enters a thinner surface mount pad, the necking option will automatically back off the track from the pad to create a thin track segment. When checked using the DRC option, no track-to-pad errors will be flagged.

NEW Component Push-aside



Within the PCB design Editor, using the **component pushing mode** helps you move and adjust components to fit them into the tightest of space. This aids the moving process just when you need to squeeze in that last component!

NEW Interactive Schematic Routing



Draw in the rough path required

Click on the target pin and let the router tidy it up for you

Interactive Schematic Routing enables you to guide the routing path for electrical connections within your schematic design. With point-to-point routing and 'sketch' path mode, schematic connection routing has never been so easy.

Even more new version 9.0 features inside

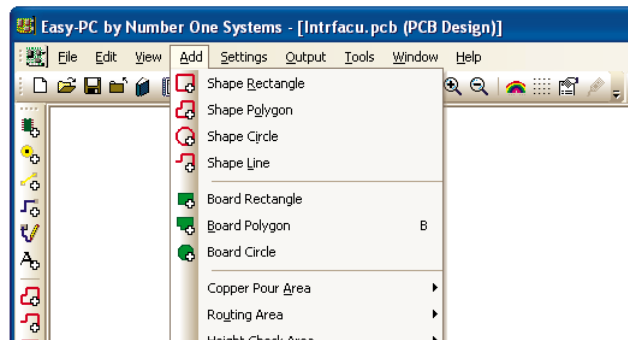


New XP Office look GUI

Even more new
version 9.0 features!

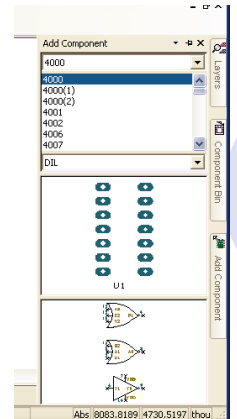
Along with a new style user interface there are now user definable menus and fully customisable toolbars. This means you can tailor the user interface to suit your own choice of tools. New separate sliding preview bars enable fast selecting by simply hovering over the semi-docked bar. A new **Add Component bar** enables component selection by dragging the component into the design.

New document tabs enable a workbook mode to clearly show open designs. Clicking on these allows easy navigation to other open designs or libraries.



Sliding Preview Bars

Once you've used sliding bars you'll wonder how you ever managed without them! Sliding bars are dockable on the side of the Easy-PC interface and when you hover your mouse over them they open up. Likewise, when you move off them after use, they slide back into the idle position ready for next time. Bars are available for the new Add Component bar; Goto bar; Layers bar; World View; Component Bin and Colours bar.



Customisable Toolbars & Menus

The user interface is fully configurable including menus and toolbars. You can add your own toolbars containing your set of tools as required. All keys on the keyboard can be defined as shortcuts for fast operation.

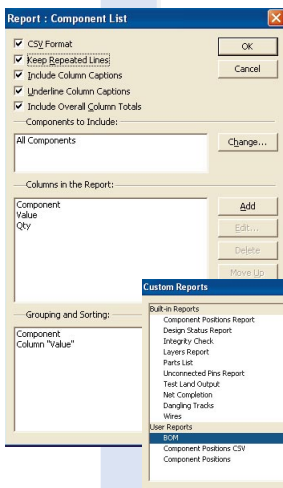
Alphanumeric Pin Names

As well as normal pin names you can now define and show your component names as alphanumeric pin names, for example A1, B1, C3 etc. This is essential for components like BGAs and connectors.

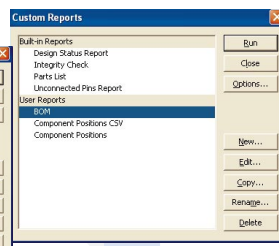
User Defined Coordinate Origin

Place your system origin anywhere on the design. This means you can move it anywhere on the design, for example to the centre or lower left hand corner of the board outline if required.

Custom Parts List Output



A new configurable Parts list option enables you to easily create lists of Parts using a simple format. Components in the report can be grouped and sorted on any field including Values; columns can be totalled, and the report can contain multiple component lists each selecting components with different properties.

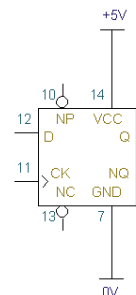


Testland Support

You can define pads and vias in the design as being testlands. A report of the testland positions, their name and net name can be output for use with automatic test equipment (ATE).

Signal Components

Schematic Symbols can have pre-allocated net names or net class names. When used in the design, the connection is automatically assigned the net name. This saves time naming the net yourself and is especially useful for power & ground symbols.



Drill Ident Drawings

Plot the drilling ids using the plotting option. This drawing is provided to your manufacturer to identify where individual drills sizes are used. Idents are defined and identified by symbols and letters assigned to drill sizes for easy visualisation when plotted.

NC Drill output

In support of the definition of plated and non-plated drill hole styles, the NC Drill option can now select either drill hole type. The NC Drill report also shows the two drill hole types, this is important when critical plating definition is required.

Other New Features in Version 9.0....

- Support for plated and non-plated drill holes
- Asymmetric gridding - different X and Y grid steps
- Layer viewing (drawing in order)
- Support for Solder Layers and over/under sizing
- Additional DRC checks
- Enhanced World View shows Working Area and Design Area
- Automatic web site checking for the latest fixes
- New spiral bound style Users Guide

Remember - you can upgrade from any version of Easy-PC For Windows to Version 9.0

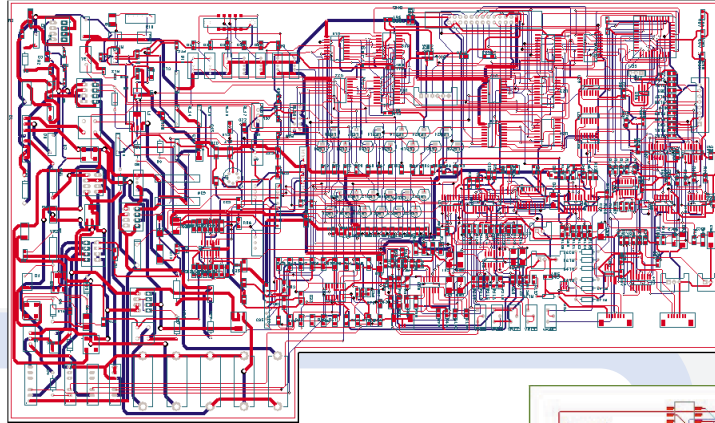
Pro Router

The high performance routing option for Easy-PC

Pro-Router delivers a new dimension of auto-routing performance for Easy-PC users with fast, high completion rate auto-routing on the most densely packed, multi-layer PCB designs. A high performance router delivering minimal overall track lengths and via counts; Pro-Router provides the ideal routing solution for large or complex designs. Pro-Router is a new generation of auto-routing software using a multi-pass cost-based conflict reduction algorithm to find a routing solution adapting to the natural flow of the nets. Adaptive routing algorithm is the only proven approach to reach high completion rates on the modern generation of designs.

Features

- Gridless routing of up to 256 layers
- Adaptive routing strategy
- Via size by net class
- SMD escape fanout control
- Routes SMDs on both sides
- Memory routing pass
- Unlimited Power Plane layers
- Split Plane/Ground Planes support
- Track and Via grid control
- Customisable cost factors
- Post-route cleanup optimisation
- Runs under Win98, ME, NT4.0, 2000, XP
- Available with Easy-PC version 7 onwards



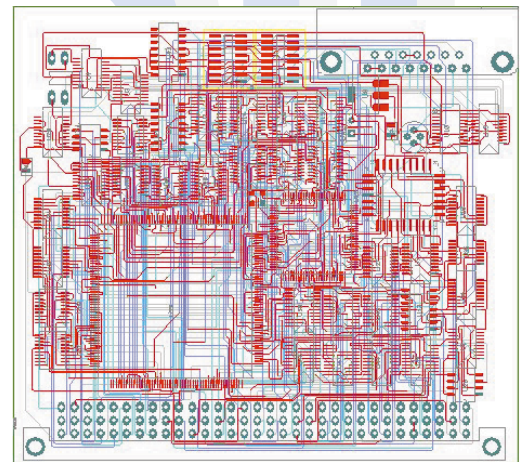
Design with 422 components, 1042 connections, 2 layers, power & ground tracks routed, mixed technology components - routed 100% using Pro-Router in 1 minute, 43 seconds on 3.0 Ghz Pentium

Pro-Router gives you the quality of routing results frequently associated with manually routed designs. Plus, speed, quality and completion rates only normally associated with autorouters at 20x to 40x the price of Pro-Router. Pro-Router is integrated into the Easy-PC PCB design environment, so, no exporting of designs or importing results. Just click 'ProRouter' and 'Route'.

7 Models of Pro-Router are available:

- 1000 pins, 2 routing layers
- 1000 pins, unlimited routing layers
- 2000 pins, unlimited routing layers
- Unlimited pins, 2 routing layers
- Unlimited pins, 4 routing layers
- Unlimited pins, 6 routing layers
- Unlimited pins, unlimited routing layers

* must have Easy-Router; and if not upgrading to version 9.0, be running Easy-PC For Windows version 7 minimum
All products have unlimited power plane layers.



53 components, 1287 component pins, routed 100% using Pro-Router in 6 minutes on a 2Ghz Pentium

You can try Pro-Router on your own designs with a demo download from www.numberone.com/downloads

NEW Easy-Spice V2.0 Update

Easy-Spice

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. No other product under £10K can do this!

Easy-Spice Product

A new Simulation menu makes the definition and selection of parameters easier using a tabbed dialog. New models and new Symbols/Parts have also been added to this update.

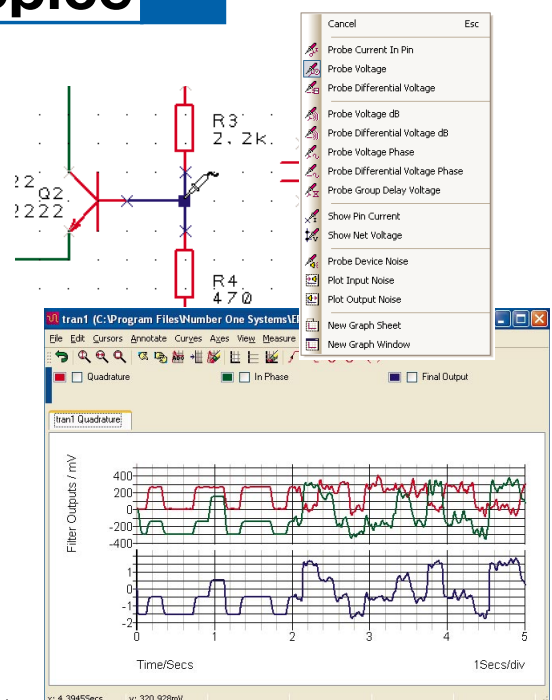
Random probing is a new option that allows you to add additional probe types to the design once the initial simulation has been run. Without rerunning the simulator, you can 'probe' around the design selecting nets and view instant feedback in the simulator graph window. Numerous probe types are instantly available from the shortcut menu.

Bias annotation enables you to add voltage and current markers to each net in the design. This can be done interactively, automatically (to every net) and to show/hide, update and delete values in the design as required.

Simulation Interface Improvements

Significantly improved simulator interface with the ability to drag and drop models into the library, much better graph dialog editing and the ability to save graphs once plotted.

There are numerous detailed improvements to the interface, such as the support for the definition of Fourier plots. The .PARAM can now be used in lib models and sub-circuits, Easy-spice now also supports the .FUNC function.



New Random Probe function allows 'probing' within the design with results instantly displayed on the graph

Datasheets are available from www.numberone.com



Deliveries from **24** Hours

PTH PCBs with solder resist and legend from just £30.00 lot price.

Visit www.pcbtrain.com for full details.

(price example is for a 100 x 100 mm PTH pcb on 1.6 mm FR4 laminate with 2 x green solder resist, 2 x white legends manufactured in 10 w/days PCBTrain scheduled delivery).

- ▲ NEWBURY ELECTRONICS, the home of PCB Train, the UK market leader for low-cost prototype PCBs
- ▲ Exclusive UK representatives for Circuit Mission Ltd (China) - manufacturers of low-cost high quality rigid PCBs
- ▲ Exclusive UK representatives for Sunflex Ltd (Taiwan & China) - manufacturers of flexible circuits



t +44 (0) 1635 40347 >>>
>>> f +44 (0) 1635 36143

NEWBURY ELECTRONICS LTD

Faraday Road Newbury Berkshire RG14 2AD UK

Pro Library

Over 14,500 library parts for Easy-PC

The Pro-Library contains over 14,500 Components, Symbols and Footprints ready-to-use in your design selected from the following popular manufacturers parts lists - plus others!



You can get a full listing of parts from www.numberone.com/downloads/prolibrary.pdf

IPC-SM Library

The NEW optional IPC library contains over 950 supplementary surface mount PCB footprint libraries for use with any variant of Easy-PC



The IPC PCB symbol library contains surface mount devices in a range of package types. These include chip capacitors, flat packs and chip carriers, in a range of sizes and pin counts, as defined in the standard. The component library contains the same number of items as the PCB symbol library, with a one-to-one correspondence in naming to save you time creating them.

Datasheets are available from www.numberone.com

Discrete Devices Flat Pack Devices Chip Carrier Devices Chip Capacitors Ceramic Flat Packs (CFP) Leadless Ceramic Chip Carriers (LCC) Aluminum Electrolytic Capacitors Bumper Quad Flat Packs (BQFP) Plastic Leaded Chip Carriers (PLCC) Moulded Body Capacitors (Polarised) Ceramic Quad Flat packs (CQFP) IC Packages MELF Diodes Plastic Quad Flat Packs (PQFP) Small Outline ICs (SOIC) Moulded Body Diodes Quad Flat Packs (QFP) Small Outline J-Lead ICs (SOJ) Chip Inductors Shrink Quad Flat Pack (SQFP) Small Outline Packages (SOP) Molded Chip Inductors Thin Quad Flat Pack (TQFP) Shrink Small Outline ICs (SSOIC) Wire-Wound Chip Inductors Thin Shrink Quad Flat Pack (SQFP) Thin Small Outline Packages (TSOP) Small Outline Transistors (SOT) and Diodes (SOD) Thin Shrink Small Outline Packages (TSSOP) MELF Resistors Ball Grid Arrays (BGA)

For information on pricing contact:
Number One Systems, Oak Lane, Bredon,
Tewkesbury, Glos. GL20 7LR, UK

Tel: +44 (0)1684 773662
Fax: +44 (0)1684 773664
e-mail: sales@numberone.com

Further information on all
products and prices available at:

www.numberone.com