

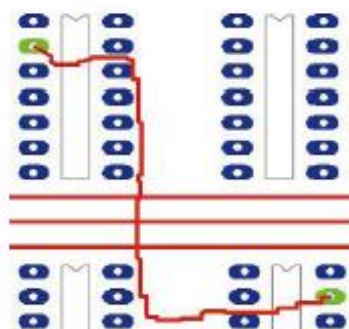


## The sensational new Easy-PC Version 8 is released!

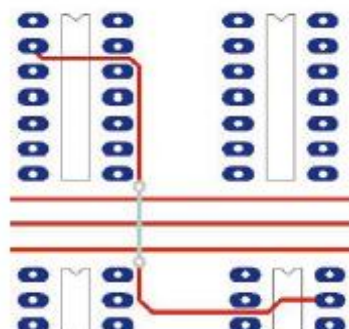
Easy-PC V8 delivers a breathtaking array of new features to help make your PCB design task that much easier. Yet again, Number One Systems has developed a realistically priced, high performance version upgrade for our much valued customers

### Sketch Mode Manual Routing

A major new feature in version 8: Sketch mode can really accelerate your manual routing. All you need do is "sketch" the approximate path you wish the track to take. Easy-PC then automatically places the individual track as close as possible to this sketched path, inserting vias and switching layers as necessary; complying at all times with your design spacing rules. Just see how quickly you can complete your manual routing with the new Sketch Mode...!



The desired path of track is identified by a rough "sketch"



The path is automatically routed as closely as possible to the original sketch. At all times adhering to the design spacing rules and changing layers as necessary

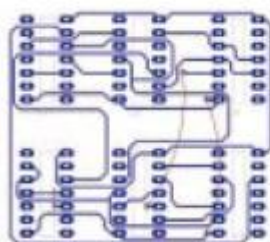


### ODB++ Format Export

Version 8 now offers ODB++ export. ODB++ is an open intelligent CAD/CAM data exchange ASCII format which helps overcome the inherent ambiguity of Gerber and Excellon. ODB++ is becoming widely acknowledged as the de-facto industry standard for PCB CAD data into manufacturing. Already, some PCB manufacturers are offering attractive PCB fabrication prices for PCB design data delivered in ODB++ format...!

### Import Bitmap

Version 8 now allows the import of bitmap and raster images into your designs. This may be your company logo or other bitmaps or symbols that need to appear on the silk screen or copper layers of your PCB designs and within the drawing of your Schematic.

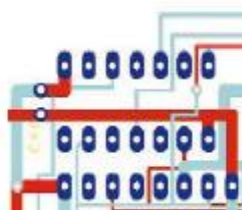


### Single-sided Autorouting

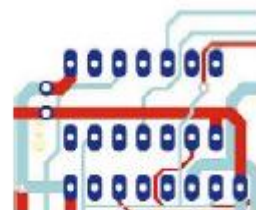
Version 8 now provides you with the ability to autoroute single-sided designs. To enhance the creation of single sided designs, the Easy-Router is now capable of using flying wires or wire links where it may have otherwise required another track layer or failed the whole track net.

### Track Mitre

In version 8 you can automatically add a 45 degree mitred corner to any 90 degree track segment. This features works on fully or partially routed boards so can be used at any stage of the design process. The option also allows you to select tracks or track segments to add the mitre.



before mitre



after mitre



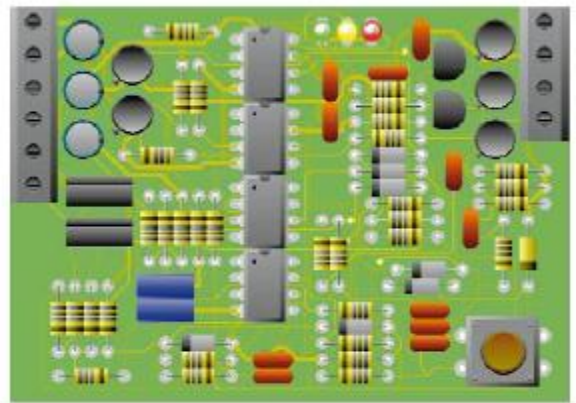
### Customisable Toolbars & Shortcut Keys

In version 8 all toolbars can be edited to include your own choice of tools. This includes the supplied tool set. Buttons on the toolbar are context sensitive which means they only work in the relevant application. Shortcut keys can be defined for all options within Easy-PC.

Even more new  
version 8 features!

## Real life board display

New for version 8, you can now view your PCB as a real life view. With gradient shading of symbols and shapes, the populated PCB can be visualised from the plan view while still in the PCB design editor. This allows you to add the detail of the PCB components and switch into this display mode for a view of how the finished board might look. With IC orientation markers and such-like added, it is much easier to spot components which are rotated in the wrong angle. This display mode can also be printed so that a 'finished' board example can be generated for assembly purposes.



## Easy-Router Improvements

Enhancements have been made to the standard Easy-Router product in version 8

The Easy-Router has been enhanced to give a better completion rate on the more complex designs. The via optimisation routine has also been significantly improved to remove redundant vias on the routed board. Easy-Router can now route multi-layer designs with unlimited layers considering all layers concurrently. A new smoothing algorithm has been added to further improve the final routed result, this removes staircases, redundant routing paths and unwanted vias.

## DXF import into Schematic

New for version 8 the DXF import option has been brought into the Easy-PC Schematics editor. This enables you to import DXF files that may have been created in AutoCAD or similar programs, existing company drawing blanks for example.

## Wires

Wires can be added to signal where a physical track cannot be inserted. A wire provides a 'jumper' over the track but continues with the same net name the other side thus retaining true electrical connectivity. Wire lengths can be pre-defined so that only specific lengths are used during insertion.

## Improved autoplacement

With more user control and an improved autoplacement algorithm, the autoplacement gives you more balanced placement with much better use of space.

## Improved copper handling

Isolated copper areas can be identified, highlighted and removed. Especially important where EMC rules are applied, and for where closer checks on manufacturing quality and yields are of significance.

## Unified QA check/report

The Design Rules Check dialog has been enhanced to enable additional checks to be carried out. These checks are for design completeness and accuracy, and also for final manufacturing consideration.



## Eagle Import

You can import Eagle files from Eagle version 4.09 and 4.11. The import handles both Schematic and PCB designs as well as the symbol libraries and components.

## New Features.....

- Repeat command
- Smooth tracks option
- Select all
- Add track using style from current selection
- Better project handling