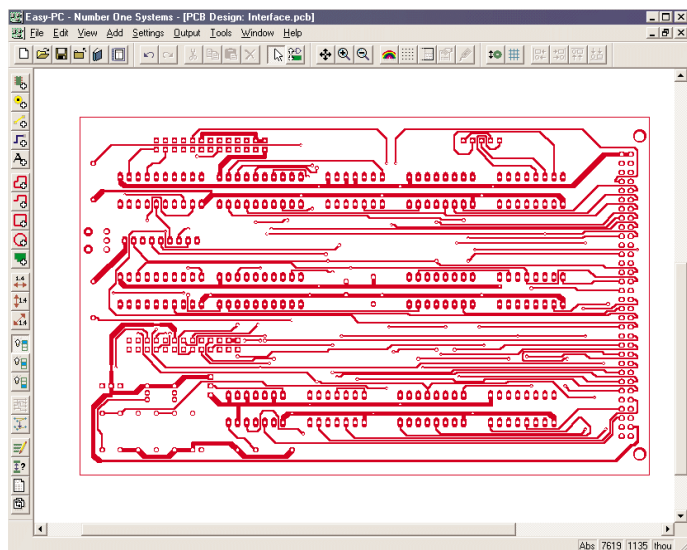


Gerber Import



With the minimum amount of effort the Gerber Import facility enables Gerber files in RS-274-D and RS-274-X format to be imported and viewed in Easy-PC For Windows version 4.0 and later.

The Gerber Import facility is available as a cost option and because it is an integrated entity within the main PCB design editor, there is no additional program or module to install. You simply 'unlock' the option using a code which we supply you.

Importing Files

The import mechanism is as easy as opening a design. You select Import and the filename and press OK. If the file detected is RS-274-X format the pad and track styles are automatically assigned. Pad styles are automatically assigned from the embedded aperture information provided in this file format. If using the RS-274-D format, the table is presented to you for manual editing of the aperture information.

Re-placement Templates

Although the Gerber import facility does not enable a full electrical rebuild of the original design, it does allow a shape template to be constructed from Gerber format files which can then be used to re-place and rebuild a design. In instances

where a design rebuild is required it is recommended that the silk screen layer(s) be imported and the board outline. This then gives you the footprint location for overlaying new components using the Add Component functionality. The electrical layers will allow you to re-trace existing tracks and vias. It may be more efficient to re-place the design first and import an existing netlist if one is available or to import the netlist and use the new silk screen template as a template for the placement.

View Before Manufacture

It also allows you view a Gerber file output from Easy-PC For Windows to verify that what is in the file is what you expect before sending the files off to be plotted. This feature alone can save money in setup charges from the plot bureaux with the added protection of not accidentally asking the plot house to plot the wrong thing!

Prerequisites For Using Gerber Import

The Gerber Import option works with and requires any Easy-PC For Windows V4.0 or a later version.

Features

- Fully integrated within Easy-PC For Windows PCB editor
- Easy to use
- Reads RS-274-D and RS-274-X formats
- Auto-detects RS-274-X format
- D-code table for RS-274-D can be rebuilt in Technology Styles
- Automatically rebuilds drill hole data from RS-274-X format files
- Provides flash to pad style conversion
- Imports electrical and non-electrical Gerber data e.g. Silk screen, and component/solder layers
- Imports board outline from Gerber format
- Import preferences can be setup e.g. Data origin, units and file format