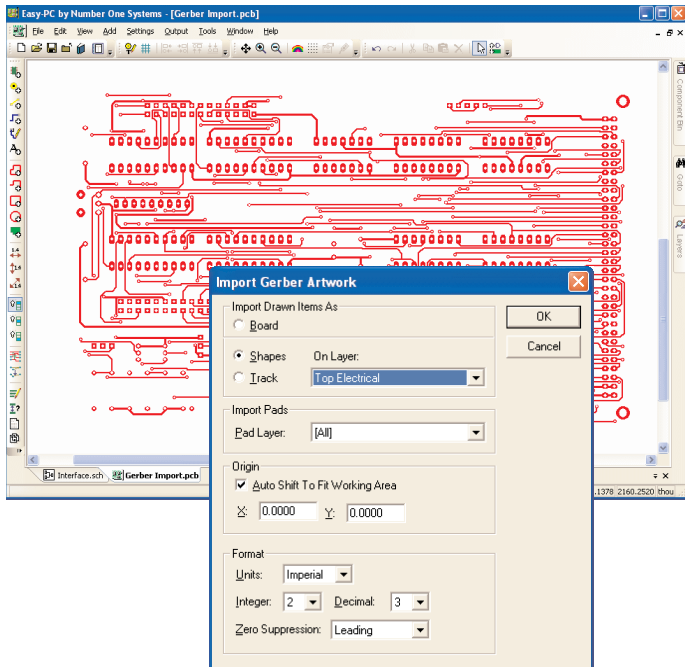


Gerber Import



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 8.0 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 reconstruct a design. In instances where a full design rebuild is required it is recommended the Intelligent Gerber Import option is used.

View Before Manufacture

It also allows you view a Gerber file output from Easy-PC For Windows to verify 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 bureau 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 Easy-PC For Windows V8.0 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 set up e.g. Data origin, units and file format
- Also see Intelligent Gerber Import Datasheet for design rebuild option (requires Easy-PC V10)