Gerber file

1. Introduction

A Gerber file is a file format that includes information used to guide the manufacturing process of a PCB.

Gerber files have been in use since the 1960s and were named after their inventor, H. Joseph Gerber.

The Gerber format is a type of file format utilized to depict the picture of a PCB.

2. Usage

Gerber files store all of the shape and location data for every element in a printed circuit board layout. In general, each layer in your PCB layout data will be placed into its own Gerber file.

It is essentially a 2D coordinate file that characterizes each layer of the board. In addition to copper traces, drill holes, and silkscreen layers, Gerber files also cover knowledge about solder mask layers, paste layers, and surface mount pads.

The following are the main uses of Gerber files in the production of printed circuit boards (PCBs):

- Design validation: Gerber files facilitate a pre-production DFM review of the design layout of the board before manufacturing to correct any errors or issues that could result in costly mistakes during production.

- Photolithography: Gerber files are utilized to generate photolithographic image films that are used to produce physical PCBs. The Gerber files direct the UV-sensitive material's precise exposure on the film, which is developed to etch the copper traces and other features onto the board.

- Drilling: Gerber files are also used to guide the drilling, ensuring that the drill head is accurately positioned to create precise drill holes.

- Solder mask application: Gerber files contain information about the solder mask layer that protects the board from unwanted solder bridges forming between adjacent leads and prevents damage to the copper traces during the soldering process.

- Silk screen printing: Gerber files include data regarding the silk screen layer, which enables the printing of additional information.

3. Gerber file data

Gerber file data is stored in an ASCII text file and it includes the following four elements:

1. Configuration parameters

2. Aperture definitions

3. XY coordinate locations for draw and flash commands

4. Draw and flash command codes

The Gerber viewer's role is to recreate a view of the PCB layout. This is the view manufacturers will use to initially inspect the board before providing a quote.
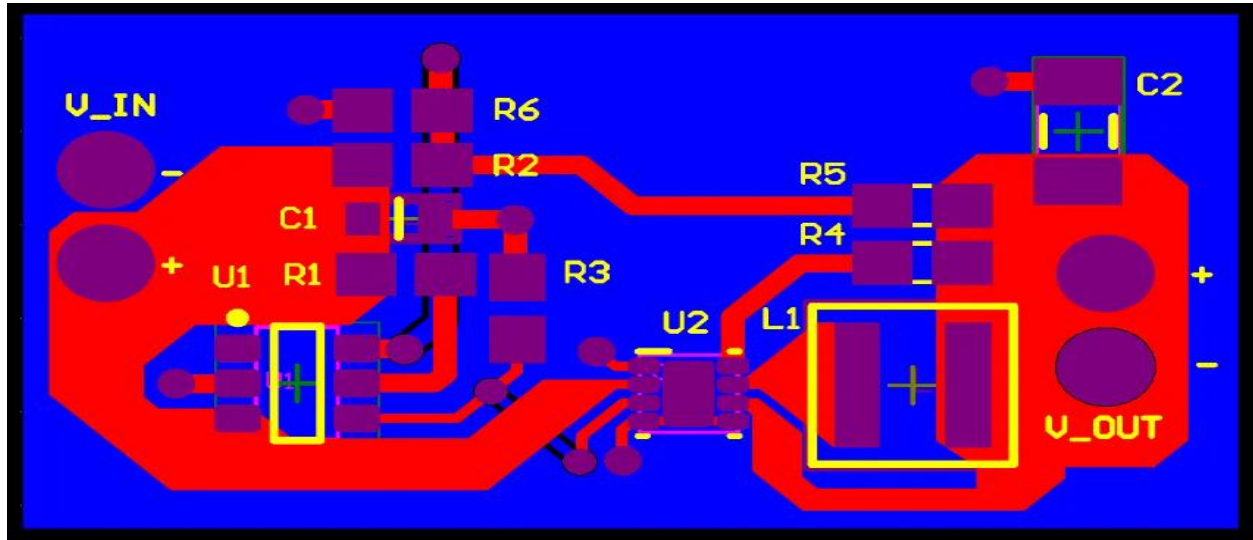

Figure 3.1