What Are Gerber Files for Printed Circuit Boards, and Who Needs Them?

Jim Usery
Electronic Manufacturing Service
Corinth, MS

When an electronics design engineer has completed their circuit design for an application, the next step towards completing the product design is to enter the schematic details into a computer based schematic capture program. The schematic capture program, which is usually part of an Electronic Design Automation (EDA) or Computer Automated Design (PCB CAD) software design package, will create a net list from the completed schematic that details every electrical connection between each electronic component. This net list is used by the printed circuit board or PCB designer in the process of designing the printed circuit board with the EDA or PCB CAD software. The finished printed circuit board will provide the physical assembly and interconnection platform for the various electronic components required by the schematic.

The printed circuit board is made up of one or more conductive layers of copper plating that is etched to form the component pads and interconnection traces and one or more layers of insulating material such as epoxy-filled fiberglass to separate the conductive copper layers and to provide the mechanical strength for the board. A single layer board would have components on the top side of the board and connecting traces on the bottom side of the board. A double layer board could have components on the top side only or have components on both the top and bottom sides of the board along with connecting traces on both sides of the board. A multilayer board would have both top and bottom sides with components and traces along with a number of internal layers used for interconnections and for voltage and ground plane layers.

The EDA or PCB CAD program provides the detailed information about the completed board design in a series of data files for each conductive layer (top, bottom, and any internal layers). The Gerber File format, named after the Gerber Scientific Instruments Company, a pioneer in photoplotter manufacturing, is the standard format for these data files. The original Gerber format conformed to the EIA RS-274D standard and consisted of a command file for each conductive layer and a tool description file. The command file consisted of a series of short commands, each followed by a set of X and Y coordinates, which would provide a photoplotter with the information to create a graphic representation. These command files became known as the Gerber files. The tool desciption file, or aperture file, defined the trace line widths and dimensional data for all of the pads and geometric shapes on the layer.

These data files of computer generated information for the printed circuit board design are then sent to a printed circuit board fabrication company to have the physical boards manufactured. The Gerber files contain all of the information necessary for the computer controlled machines at the printed circuit board (PCB) fabrication houses to etch the copper layers to create the component pads and connection traces, drill all required holes, and cut the board to the required size.

Since a PCB may have from one to many conductive layers, the older Gerber format EIA RS-274D always assumed a set of command files (one for each PCB layer) and one "tool description" file, or aperture file. A standard for the "aperture files" was never established so every EDA or PCB CAD software product had its own version of the aperture file format. If the printed circuit board fabrication house could not read the aperture file format as sent, then the aperture information would have to be reentered manually.

The newer Gerber format conforms to EIA RS-274X and this format includes the aperture information in the file headers as embedded information for each command or Gerber file. This newer format is often called X-Gerber. With all of the aperture information included within the header fo the file, each X-Gerber file provides all of the information required to fabricate the related portion of a PCB layer.

The file names for the Gerber files should be descriptive enough for the pcb fabricator to understand which board and board layer that each file applies to, such as membdtop.gbr as a file name. The standard process is to include with each set of files for a board design a special readme.txt type text file that defines each file name and its application for the board design. The board vendor will use this readme.txt text file as the starting point for the board manufacturing process. Gerber file extensions are often .GBR, .GBX, or .ART. Sometimes extensions such as .TOP and .BOT or .SMT and .SMB are used instead of the .GB_ type extensions. Often the file extension for a type of file (top, bottom, silkscreen, paste, inner layer) is controlled by the EDA or PCB CAD software package or is selectable within the package. This variation in the extensions makes the inclusion of the readme.txt file as a requirement in the overall file package for the board vendor.

The list of files for a board design will include the silkscreen for the top and sometimes the bottom layers if components are mounted on both sides, component placements for the top and sometimes the bottom layers, solder screen paste files for surface mount applications, drill drawings, solder mask files, panel drawings, pad master top and pad master bottom, etc.

For instance, for a double sided (2 layer) pcb, the Gerber files will consist of two positive Gerber layers (top and bottom), aperture file (if not in the RS-274X format), NC Excellon drill file, Drill Tool List file, Silkscreen file for each side with components, soldermask files for top and bottom, and top and bottom screen paste files for surface mount boards where applicable. A four layer board would have all of these files plus two inner layer files and a six layer board would have all of these files plus four inner layer files.
At Innovative Circuits Inc., we interface with PCB fabrication houses for our own board designs as well as board designs provided by our customers for whom we are providing board assembly services. Thus we are very familiar with Gerber files and their purposes and functions. But we also realize that countless other people in other organizations who are involved in ordering raw boards or board assembly services will see or hear the term Gerber files without having any knowledge of the term, and this article if for them.

Jim Usery
Sales and Marketing Director
Innovative Circuits Inc.
311A S Parkway St
Corinth, MS 38834
office 662-287-2007
toll free 866-887-7381
fax 662-665-9275
email jusery@icimfg.com

Visit our website at www.icimfg.com for more information about Innovative Circuits Inc. and the services and expertise that we provide to our customers, or for additional technical articles.