An Introduction To Gerber Extended Files

Have you ever looked at a printed circuit board and wondered what exactly it takes to make one? In this article, we take a look at Gerber and Gerber Extended Files, which are the blueprints for a printed circuit board.

What is a Gerber File?

A Gerber or Gerber Extended File is a standard file format used in the electronics industry to store and communicate design blueprints and other information to the relevant manufacturers. Gerber files are most often used to communicate design information regarding the manufacturing specifications of printed circuit boards. At their core, Gerber files can best be thought of as a PDF for the electronics manufacturing industry.

The information inside of Gerber files is written in American Standard Code for Information Interchange (ASCII) characters. Since a computer can only understand information as sets of numbers, ASCII code works as a numerical representation of non-numerical characters, such as "A" or the @ symbol.

When the Gerber file format was first created, the strings of ASCII inside of it were used as a way to call out information and instructions to a machine called a "photoplotter," which used the instructions as a means to expose a picture on a piece of film by controlling a light source.

The early photoplotters (as introduced by Gerber Scientific, Inc) used a xenon-based flash lamp to project an image from a rotating aperture wheel onto a piece of photosensitive film or a photosensitive glass plate. The imaging lamp traversed over the glass or film without touching it: this produced two different types of illumination based on how the image head traversed as well as the duration of the traverse. The two types of illumination are:

- Draws, which are vectors created by continuous illumination of the photosensitive material as the imaging head moves over it

- Flashes, which are singular, simple graphics created in one location by shining a light through an aperture of the appropriate shape at a certain location

In contrast to those early lamp-powered devices using ASCII text, modern photoplotters scan rasterized graphics or bitmaps and then use a laser, laser diode or light-emitting diode (LED) focused at multiple spots on the photosensitive material to form the image in question.

Since today's laser photoplotters treat ASCII Gerber files more like a modern image file, one question comes up again and again: why not just switch to using a modern image file format, like PostScript? The answer is a bit of a catch-22: modern photoplotters use the Gerber file format because most computer-aided design/drafting software creates files in the Gerber format and most computer-aided design/drafting software creates files in the Gerber files because modern photoplotters use it.

When it comes to actually making files using the Gerber file types, there are two formats that are generally used:

- **RS-274D.** The older of the two Gerber file format standards, RS-274D spreads information for a single layer across two separate Gerber files.

- **RS-274X.** The newer of the two file formats, RS-274X allows all of the information for a single layer to be saved as one file.

What does this mean in practice? It means that, compared to RS-274X files, RS-274D files are much more prone to errors because of the way their aperture files are packaged. Aperture files for PCB manufacture come in a wide variety of types, but almost all of them are proprietary to the software that compiled or wrote them in the first place. Because of this, the computer-aided manufacturing (CAM) engineer in charge often has to type them in by hand. Any process done by hand is going to be lengthy and time-consuming, but it becomes doubly so when it's something like a Gerber file, which is made of ASCII text that isn't easily readable by humans.

On the other hand, things get much easier for engineers when they use the RS-274X file format. In RS-274X, the aperture files are embedded as part of the file data: there's no need for a CAM engineer to do any kind of manual input. Using an RS-274X allows a PCB designer to call out and define important blocks of code, such as macros, pad shape and line width in a single file, where an RS-274D file would require these blocks to be formatted as a separate file.

There are four types of instructions that can be called out to a photoplotter in the ASCII code of a Gerber file, regardless of format:

- Configuration parameters (which begin with % symbols in the RS-274X standard) are a type of instruction/command controls various parts of the rendering process, including things such as color choice (black drawn on white or white drawn on black)

- Macro and Aperture definitions are used to define the shape of different pieces of the PCB. Macros are used to define special geometry such as logos, registration marks or other, complex shapes. Apertures are used to define the shape and thickness of the pads used on most PCBs

- Drawing commands, which can come in one of four styles: flashes, lines, polygon fills and arcs

- Coordinate information for each feature (notated in X-Y format)

Now that we've looked at what a Gerber file is and how it's made, let's take a step back and take a look at the history of the Gerber file.

History of the Gerber File

The Gerber file was the brainchild of Austrian-American H. Joseph Gerber. Gerber first got his start in engineering a t Rennsselaer Polytechnic Institute in New York. While working on a degree in aeronautical engineering, Gerber invented a variable scale as a way to help with an assignment.

The first variable scale was built out of the waistband of Gerber's pajama pants, which, in the European style of the time, could detach the elastic waistband. Gerber marked out a scale on the elastic and used that to help him with the tedious measuring, calculating, and replotting that was required to finish his assignment.

Gerber was encouraged to patent the idea by his professors: he did so, and, after graduation, incorporated the Gerber Scientific Instrument Company to sell the devices that he was designing. As the Gerber Scientific Instruments Company became more and more successful, Gerber traveled around the US, talking to his customers and learning about on-the-job problems that they faced.

All of this interaction with other engineers gave Gerber an interest in plotting. He realized that the analog plotters that were currently in use at the time simply weren't accurate enough for what was being plotted. This led to him working to develop the first digital plotter. During the 1950s, he developed a digital X/Y coordinate table, which he later used as the basis for the first digital drafting machine.

While on a trip to RCA headquarters in the late 1960s, Gerber discovered another use for the X/Y coordinate table he'd developed years earlier: as the basis for a photoplotter to be used to create the photographs that would be used as the blueprint for a printed circuit board. The patent for the first photoplotter, dubbed the "Variable Aperture Photoexposure Device" in the patent paperwork, was filed in September of 1970 and granted in October of 1972.

The first photoplotter worked worked by moving the head, which contained a xenon-based lamp (in the first iterations) to the right spot over the film. A wheel with different holes along its surface (known as apertures) was then rotated so that the right hole was lined up under the lamp

While the invention of the first photoplotter was incredibly important to the PCB industry, the team at Gerber

Scientific was just as important to the field of computer-aided design. Led by senior engineer Ron Webster (who invented the photoplotter that had been patented in 1972) and chief engineer David Logan, the team at Gerber Scientific began to develop a whole system of fabricating printed circuit boards. The early Gerber system included some CAD functionality, automated optical inspection systems and networking capabilities.

Gerber Scientific's proprietary file format (.gbr) was derived from (and later incorporated as part of) an already-existing file format, RS-274D, mentioned as one of the two Gerber file format standards earlier in this article. RS-274D was approved as a standard file format by the Electronic Industries Association (EIA) in 1980 and used to drive numerically controlled plotting machines, including those built by Gerber Scientific.

How Gerber Files Are Used Today

With the introduction of RS-274D as a standardized file format in 1980, the use of CAD as a system was becoming more prevalent as the first and only thing PCB manufacturers used to put together blueprints, to the point that they began to replace the 2-to-1 hand-taped blueprints that had previously been in use. Assisted by CAD software, a photoplotter could be used to generate the photographic blueprints by outputting the drive data straight to the plotter itself.

By the time RS-274D was introduced as a standard, other photoplotter manufacturers began to enter the market. However, the Gerber variant of the RS-274D format became the de facto standard because Gerber released the full specifications of their variant on the format when it was released in 1980.

As mentioned above, the RS-274D file format has one major limitation when it comes to using it as a way to transfer the different layer images for PCBs: the size of the apertures, the shape of the apertures and amount of apertures that could be used by an RS-274D file at any one time was limited by the space available on the aperture wheel itself.

While this could (and did) work for designs that used regular through-hole components, the use of an aperture wheel meant that the plotter could not handle the newer surface-mount components that were gradually coming into use: these surface-mount components used a several different types of rectangular pads in different sizes, which were hard for the photoplotters to draw.

With the RS-274D format, the CAD engineer's only option was to use a series of draws to make tiny lines, "painting" on the pads. Similarly, simple plane layers could be drawn in reverse by the plotter — the holes used for clearance in the plane were plotted in black: whoever was manufacturing the board would then reverse the polarity to white using the front-end CAM system or doing so physically by contact printing on the board. However, this didn't work for layers of mixed planes or signal layer planes — instead, those layers had to be painted in using draws, much like pads were. Using this style with the plotters of the time, a large image could take more than a day to plot.

An obvious problem presented itself: the amount of time that a photoplotter using the RS-274D file format needed to build a PCB blueprint. The solution to this was twofold: a new photoplotter was designed, with a new file format to go with it. The new type of photoplotter was the raster photo-plotter, which used a light, like a laser diode or LED, to scan the film in a continuous pattern. The lasers on raster plotters were fed a series of on/off commands to build the image of the PCB blueprint up in sequence.

With the advent of raster plotters, any shape of printed circuit board could be built, made up as it was of rasterized pixels. Today, the raster photoplotter has become the standard tool when it comes to making images for PCBs. Today's photo-plotters use more than 40 beams, all independently-switched and running simultaneously. This allows PCB designers to plot their blueprints at resolutions as high as 50,000 dots per inch or more.

With the advent of laser/raster photoplotters, PCB manufacturers and designers were able make the Gerber format more flexible so that it could better suit the requirements of continued PCB design. To bring the file format in line with the advances in photoplotters, EIA launched the RS-274X file format in 1991. The RS-274X file format allowed the designer to image any shape in one of three ways: as a full pad, as a long track, or on a plane or polygon. Since aperture definitions did not need a physical wheel to be printed on the photo-image, they could be

immediately pulled from the CAD file when and as needed. Aperture and macro definitions are now included in the Gerber file as a standard output: under the RS-274D file format, they were included as a separate file.

Since the introduction of RS-274X in 1991, it has become the standard file format for the transfer of PCB layer data. Even still, the old RS-274D file format is still in use by some PCB designers, especially when they have to design to an older style of PCB or copy an old PCB job.

Gerber Files & PCB Manufacturing Include: any special considerations when putting together the file

So how are Gerber files actually used in PCB manufacturing? In the subtractive method of PCB manufacture (the most commonly-used method), the PCB starts as a sheet of substrate material that has been encased in copper on either both sides or just a single side, depending on the type of PCB being made. Gerber files are used to provide a picture of where on the PCB copper should remain at the end of the process.

The picture is then used to guide the creation of channels to remove copper from the board, with the intention being to leave behind only the conductive traces used by the actual PCB. The standard convention when it comes to marking out copper traces is to use clear markings for areas with no copper and black markings for the actual copper traces. However, the use of Gerber files in PCB manufacture doesn't just stop at marking at copper traces. Once the copper traces are marked out, other layers, such as the solder mask and the silkscreen (lettering, nomenclature, etc) go over top of them. The Gerber file will also have pictures for these other layers. In addition to all of these layers, a Gerber file can also provide a PCB manufacturer with a picture of locations of the board's drill holes and even the actual size of the board if the designer so chooses.

Since Gerber files can contain so much information, a PCB manufacturer can get up to nine separate files — even if all they're making is a two-layer PCB. Examples of the files a manufacturer might receive include:

- The copper traces for the top of the circuit board

- The copper traces for the underside of the circuit board

- A picture of the solder mask for the top layer

- A picture of the solder mask for the bottom layer

- Silkscreen imaging for the top layer

- Silkscreen imaging for the bottom layer

- The coordinates of any and all drill holes, as well as their properties, such as depth

- An outline of the physical board with all the machining operations marked out, such as scoring, slots and internal cutouts

- A simple text README file explaining the uses for all of the files listed above

Gerber files can be designed in a number of software applications, including Cadence and Altium Designer. Once the Gerber file has been designed, it transferred to the PCB through the use of a photoplotter. The photoplotter prints out each of the files in the list above (e.g. the top layer of the circuit board, the bottom layer of the circuit board, the top layer solder mask, etc) on one piece of film. So the top layer of the circuit board is one piece of film and so is each subsequent layer in line.

Each PCB layer and the accompanying solder mask gets its own sheet, so a simple two-layer PCB needs, at minimum, four sheets of film — one each for the top and bottom trace layers and one each for the accompanying layers of solder mask.

After the film sheets are printed, they're lined up and a hole, known as a registration hole, is punched through them. The registration hole is used as guide to align the pieces of film later on in the process. The registration hole is used to align the pieces of film onto the copper and substrate layers so that the substrate panel is covered by the film.

The PCB blueprint film is then covered up by a type of photo-sensitive film called the resist. This photosensitive film is made of a layer of chemicals that harden after they're exposed to ultraviolet light. By using a resist, the PCB manufacturer can get a perfect match for the photos on the Gerber file.

Once the resist and the layer of substrate have been lined up using the registration holes from earlier, they receive a blast of ultraviolet light. The ultraviolet light passes through the translucent parts of the film, hardening the photo resist. This indicates areas of copper trace that are meant to be kept as pathways. In contrast, the black ink prevents any light from getting to the areas that aren't meant to harden so that they can later be removed.
This process is repeated for each layer, including the solder mask layers and the silkscreen layers.
When it comes to designing a Gerber file and transferring to to a PCB, there are two big considerations that designers need to take into account:

- **Legacy issues.** The Gerber file format was originally used to drive numerically controlled photoplotters. Today's PCB printing machines are much more in line with a modern laser printer — something that the Gerber file format wasn't built to handle. This causes some issues when Gerber files are used with modern raster plotters to print PCB specifications.

  ○ Hole data can be absent because the original photo plotters used for PCB manufacture didn't drill holes. Excellon numerical control files can be used to augment the Gerber data, but even this is incomplete — it doesn't differentiate between through, blind or buried holes. Technicians reading the data also have no idea whether the data they're reading for the holes they need to drill has been given the correct scale or offset properly.

  ○ No functional definition or mapping of the files. Nothing in the Gerber file itself tells the CAM technician whether it's a top file or a bottom file, or if it's been mirrored. This is why most modern Gerber files include a README.

- **Design for manufacture (DFM) issues.** Gerber files contain no way to read the "intent" of the PCB being manufactured. There are also no protections against a PCB designer building a PCB that is difficult or even functionally impossible to create. One of the most common errors that PCB technicians need to watch out for is missing files, but other errors include:

  ○ Putting silkscreen onto the PCB pads

  ○ Putting features of the PCB too close to the outline of the PCB

  ○ Drilling holes twice, making them end up larger than they should be

As long as designers and technicians keep a lookout for these and other issues, the Gerber file format can and will be used for PCB design and manufacture well into the future.

Pros and Cons of the Gerber File

Much like every other file format in existence, the Gerber file format has its pros and its cons, namely:

Pros	Cons
- Simplicity: one of the easier file formats to build and use - Ubiquity: is one of the most common file formats in use today - Is supported by free file viewing software	- Multiple files needed, depending on the format - All files need to be examined prior to fabrication

There are other file formats in use, as well, with the main competitor to the Gerber file format being the ODB++ format. ODB++ has several advantages over the Gerber file format, with possibly the largest one being its single-file nature: a single ODB++ file contains all the pieces needed for stacking up the PCB layers, drilling the requisite holes, and performing any masking.

ODB++ has its own pros and cons, too. Pros of ODB++ include the fact that, like Gerber files, the ODB++ file is widely used. It also is checked for quality and has a DFM check as part of the fabrication routine. However, one of the things that might make a manufacturer pick ODB++ over Gerber is its ease of import to the chosen fabricator, especially when compared to Gerber's ASCII output.

The main faults with ODB++ plus lie in the fact that it's more complex than Gerber, making it possibly more prone to errors when encoding, and that it is designed as a different image format, meaning that some fabricators that can use the Gerber .gbr file may not be able to use the ODB++ file format.