

CircuitPeople™

What is a Gerber File?

[Home](#)[CircuitPeople Blog](#)[Tell a Friend](#)[Register](#)[Login](#)

Introduction to Gerber Files

Gerber is a standard electronics industry file format used to communicate design information to manufacturing for many types of printed circuit boards. In many ways, Gerber is the electronics world's equivalent of PDF. This odd little format, a hybrid machine control language and image, is a core component of the electronics manufacturing supply chain. If you're involved in building electronics for fun or profit you have (or soon will) come across it. You may even come to depend on it.

I've written this article to provide a general explanation of:

1. The [Contents of a Gerber file](#)
2. The use of [Gerber Files in PCB Manufacturing](#)
3. [Gerber Files and PCB Design Considerations](#)

I hope you find this article useful, and generally accurate. In any case, I am happy to receive your comments, suggestions and corrections – just drop an email to editor@circuitpeople.com.

Contents of a Gerber File

The content of a Gerber file is ASCII text (i.e. English letters, digits and a few special characters) and looks like a human could *almost* read and understand. Here's an example:

```
G75*  
G70*  
%OFA0B0*%  
%FSLAX24Y24*%  
%IPPOS*%  
%LPD*%  
%AMOC8*5,1,8,0,0,1.08239X$1,22.5*%  
%ADD10C,0.0080*%  
D10*  
X000281Y000835D02*  
X002472Y006196D01*  
M02*
```

When the format was first created, these commands called out instructions for a special kind of machine called a [photoplotter](#) which used them to create a picture on photographic film by precisely controlling a light source. Today, however, photoplotters aren't always a part of the process and Gerbers are treated more like image files. So why doesn't the industry just use image files like postscript? It's a bit of a catch-22: The machines use Gerbers because CAD/EDA/CAM software creates Gerbers; And, the software creates Gerbers because the machines use Gerbers. Alternative formats like ODB++ exist, and are probably superior to Gerber for the

most part (except for mandating the use of more expensive hardware and software, all controlled by one company). Nonetheless, Gerber is here for the long-haul so we're well served by a little understanding of it.

About RS-274D and RS-274X

There are two "standards" for Gerber files out there: the "old" and the "new". The old standard (D) spreads information for a single layer across at least two files. The new standard (X) allows all the information for a layer to be contained in a single file. The biggest benefit of this comes down to data management (or in plain English, the fact that it's much easier to keep track of one file than two files).

The types of instructions Gerber file instructions are (roughly):

1. Configuration Parameters ("%...*%" blocks)
2. Macro and Aperture Definitions (AM and AD parameters)
3. Drawing Commands
4. X/Y Coordinate information for the locations of features

Instruction types #3 and #4 are shared between the X and D variants of Gerbers. That is, the old and new formats share the same commands and capabilities when it comes to basic drawing of lines and simple shapes. However, the new "X" version also allows the definition of some important options as well as reusable blocks of data called apertures and macros. In a version "D" file set (remember, always at least two files) the apertures are defined in an external file that can be one of many different formats. This is why an "X" file (ahem) can be viewed and rendered by CircuitPeople standalone -- all the pad shape and line widths are built into the one file -- and why "D" files are considered less desirable by PCB fabricators.

Configuration Parameters

Gerber files containing blocks of text starting and ending with percentage symbols ("%...*%") are probably the RS-274X variety. Sometimes we see files with parameters but with external aperture definitions, which is an error but our software will accommodate it. Configuration parameters control various things in the rendering process, including whether the geometry should be considered white drawn on black, or black drawn on white.

Macro and Aperture Definitions

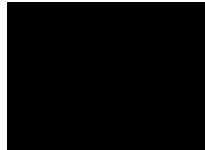
Apertures define the thickness of traces, size and shape of pads. Macros (AM parameters) define complex shapes for registration marks, logo, lettering and other special geometry that appears in a design. Macros are a "less often" used feature of Gerber, though many CAD systems employ them for filling the gaps in the original specification for features such as pad teardrops, rotated rectangles and rectangles with rounded corners. Macros support complex composition rules, and even some equation-driven parametrics. It's a shame they aren't used more often in CAM output.

It's worth noting (reiterating, maybe?) that one reason manufacturers shy away from the "D" variant gerbers (the ones with apertures defined in an external file) is that customers often forget to include them with their order. *Don't do that.* If you do, the best thing that can happen is that your order will be delayed. At worst, it won't be

delayed and you'll discover it only after assembling components only to find it non-functional.

Drawing Commands

Gerber has three drawing primitives: flashes, lines and arcs with thickness, and polygon fills. It can be a challenge to represent complex geometry using only these shapes. Luckily, RS-274X also adds "Step and Repeat" which allows us to create patterns of features. CAM systems are clever in the way they create Gerber files. For example, the system may "pour" a large area of black (copper) then draw over it with a "scratch" layer of wide geometry, then finally draw a copper trace down inside the clear. Only three commands are issued, but the result seems more complex:



1) Polygon Fill



2) Scratch Setback



3) Draw Trace

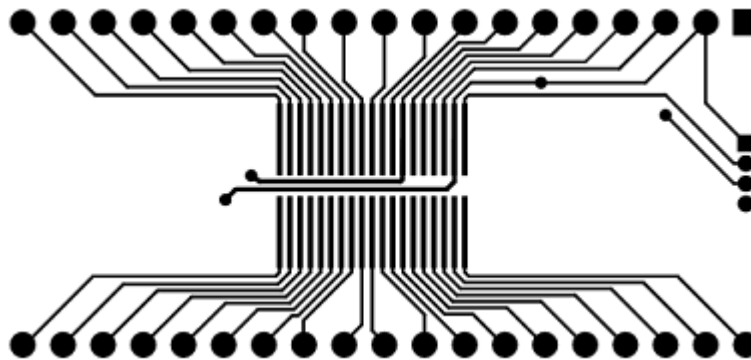
X/Y Coordinate Information

..placeholder...

Gerber files in PCB Manufacturing

The purpose of the PCB is to mechanically support components as well as move the proper number of electrons to where they are need. That is, the PCB is made with channels (traces) of conductive material along which electricity flows (if you will) from component to component.

A "bare" PCB is one that has yet to have components attached, and is generally constructed of alternating layers of conductive material and non-conductive insulating material. Copper is the most commonly used conductor with various forms of fiberglass, FR-4 being the most common, as the insulator. However, due to requirements for high thermal conductivity, strength or dielectric many other materials are used as well. There are a lot of ways to make a PCB, but in this paper I'm only going to talk about one common method. A *subtractive* PCB manufacturing process is one that starts with sheets of composite material with a uniform layer of copper plated of the entirety of one or both sides. Channels (traces) are formed by selectively removing portions of the copper conductor, leaving behind the traces we see on finished boards. In this process, Gerber files provide the "picture" of where copper should remain:



I literally mean a picture of where the copper should exist on each layer – imagine clear for no copper, black for copper. But there is more to a PCB than just copper, which when exposed to air oxidizes making it difficult to adhere to with solder. Over the copper will be a layer of solder mask (commonly green) and probably some white letter referred to as *silkscreen*, *legend* or *nomenclature*. As with the copper, Gerber files provide pictures of where the solder mask paint, silkscreen paint, or cut-outs in a stencil will go. In addition, it isn't uncommon to supply locations and for holes in a Gerber file – each size of hole represented by a different symbol – as well as the shape of the board.

You can probably sympathize with the poor manufacturer. For a "simple" two layer PCB, they can receive up to nine files:

1. An image of the copper conductor for the top side of the circuit board.
2. An image of the copper conductor for the bottom side of the circuit board.
3. An image of where solder mask is to be applied on the top layer.
4. An image of where solder mask is to be applied on the bottom layer.
5. An image of where silkscreen is to be applied on the top layer.
6. An image of where silkscreen is to be applied on the bottom layer.
7. The locations of the drilled holes, and descriptions of the properties of the holes. Often, PTH and NPTH (see below) are separated into different files.
8. The outline of the board, including internal cutouts and other machining operations (scoring, slots, etc.)
9. A README file with the name and intended use of each of the above files.

In addition to these files, a manufacturer will need other information that generally isn't included (and shouldn't be included, IMHO) in the Gerber data:

1. Are the images for the solder mask and stencil layers supplied in "negative" (holes drawn black) or "positive" (holes drawn white) convention?
2. Are the bottom layers (copper, mask and silk) shown as viewed from the top (mirrored) or as viewed from the bottom?
3. How thick does the copper need to be on the outer layers, inner layers and in the barrels of holes and vias?
4. If the holes are supplied as a "drill drawing" in a gerber or DXF file, the table of sizes has to be manually translated to NC (Excellon) file(s).
5. If the holes are supplied in Excellon files (yet another numerical control language), which holes are plated, non-plated, through, blind, or buried (and if blind or buried, on which layers do they start/stop?).
6. What colors should be used for the solder mask and silkscreen?
7. What material should be used for the core?
8. How thick does the finished PCB need to be?

And the list goes on. Obviously, this process leaves plenty of room for errors, which brings up the topics of automation and DFM. But before going there, we need to take a side-track into how the PCB manufacturing process works.

While the construction of a PCB is conceptually simple, correctly manufacturing the circuit board can be complex. A general outline of the traditional PCB manufacturing process for a simple two layer board (meaning a board with two conducting layers) may look like the following:

1. Drill all the holes in the panel that will be plated with copper. This step includes preparation of an NC drill program for the automatic drill machines most shops use. Excellon is a very common format for these NC programs, because Excellon drills are very commonly found in shops around the world.
2. Carefully clean the copper surfaces and carefully remove any "burrs" on the edges of the holes caused by drilling.
3. Coat the entire surface of the copper with photo resist. Resist is a material that is soluble unless "cured" with heat or a strong light source, and that is chemically inert during the following process.
4. Attach the appropriate films to each copper side, and expose the "clear" sections to light to cure them.
5. Remove the uncured resist, leaving the cured resist to protect the copper from being plated with tin in the next step.
6. Put the panel in a chemical bath (various formulations are used) to add a very thin layer of tin to the surfaces no longer protected by resist. This coating will protect the copper from being removed in the etching step, below.
7. Remove the cured etch resist using a different solvent, or mechanical means.
8. Put the panel in a chemical bath (various formulations are used) to remove the unprotected copper from the panel.
9. Put the panel in another chemical solution that will deposit a thin (or thick, depending on the design specifications) layer of copper on all remaining copper and hole barrels. This layer of copper increases the thickness of the traces, and provides the copper inside the holes to carry electrons between layers.
10. At this point the board is electrically functional, but with bare copper exposed it will oxidize very quickly rendering it difficult for solder to adhere. So, the copper will be protected with solder mask (commonly green) and/or plating with an inert material such as gold, silver, OSP or tin.
11. At this point the entire panel is coated with solder mask paint. Like the resist, the solder mask can be cured with strong light.
12. Using artwork for the solder mask layer expose the parts of the solder mask that are to remain on the board to UV light.
13. Remove the unexposed solder mask.
14. Cure the remaining solder mask using high heat (probably in an IR oven)
15. Using the artwork for the legend layer, a silkscreen technique (yep, just like T-shirts) is used to apply the white legend/nomenclature.
16. Cure the solder mask and silkscreen ink in an oven.

Viola! Your board is ready to have components attached (aka. PCB "assembly") and start service in your project or product.

From the above you might have noticed the part where Gerber files were made into "films" sounds a little antique. Well, the truth is that it *is* antique. Although these steps are commonly in use, many fabricators use a technique akin to inkjet technology for applying silkscreen (aka. "nomenclature," or "legend") to the board. Similarly, solder mask and etch resist can be applied in a computer-controlled "spray painter". Hence,

films may not actually ever be used, and for PCB designs involving very small features or high layer counts almost certainly won't be.

Every shop has a unique variation on the above process, and new technologies are available almost weekly. So, if you visit a fabrication shop, expect to find some other technologies and processes to be in play. But the overall process should follow the general outline above.

Gerber files and PCB Design Considerations

What's all that have to do with Gerber files? Well, lots of things. I'm going to break them down into two areas: Legacy Issues and DFM.

Legacy Issues

As mentioned earlier, the Gerber format was originally used as NC to drive mechanical systems that selectively exposed photographic film to light in a darkroom environment. To make a pad, these machines would "flash" light through an appropriate shaped aperture. To make a trace the machine would keep the light on while moving to create a "drawn" line or arc. As you can probably guess, that isn't the way it's done today. Those old photo plotters were functionally limited, notoriously unreliable and expensive. Remember those "aperture definitions" in the Gerber format? Yep, those used to indicate physical holes of different sizes that would be used to control the amount of light exposed onto the film.

Today, the machines have evolved into something much more like a laser printer, using a beam of light bounced-off rapidly moving mirrors to expose the film. And, as I mentioned above, in some cases similar machines expose the etch resist and solder mask directly rather than use films as well. In its role as a vector image format, Gerber is not unlike Adobe PostScript, Windows Meta files, or SVG. But, because it was not intended for this purpose, some nuisance issues come into play when Gerbers are used for PCB specification:

1. Uninterrupted Data. The Gerber format includes many commands for how to treat film. These are largely meaningless when used as an image format. However, it's entirely possible that the CAD system used to generate the Gerber file didn't know this, and functional differences can result.
2. Absence of hole data. Because photo plotters of old didn't drill holes, the format didn't include hole data. Excellon NC files are often used to augment the Gerber data with hole drilling information, but even that is incomplete since it doesn't differentiate between through, buried and blind holes. Worse perhaps, is the lack of assurance that the scale and offset of the hole data matches that in the Gerbers. CAM technicians at PCB manufactures often have to rescale and offset hole data to make it match the true hole locations.
3. Absence of functional mapping for files. Imagine the files sent to a manufacture were named A.gbr, B.gbr, etc. Nothing inside the files tells which part of the PCB they're intended to define, if they are "positive" or "negative", or if they are mirrored. Hence the need for a README.
4. Inclusion of "out of bounds" data is common with Gerber files as a way to communicate design intent. I often see files with "TOP SIDE" or "THIS SHOULD BE READABLE FROM BOTTOM SIDE" notes to clue-in manufactures about important notes. This is great, but all that data must be manually clipped from the files before fabrication to avoid using excessive amounts of core material. It's best to put those notes elsewhere.

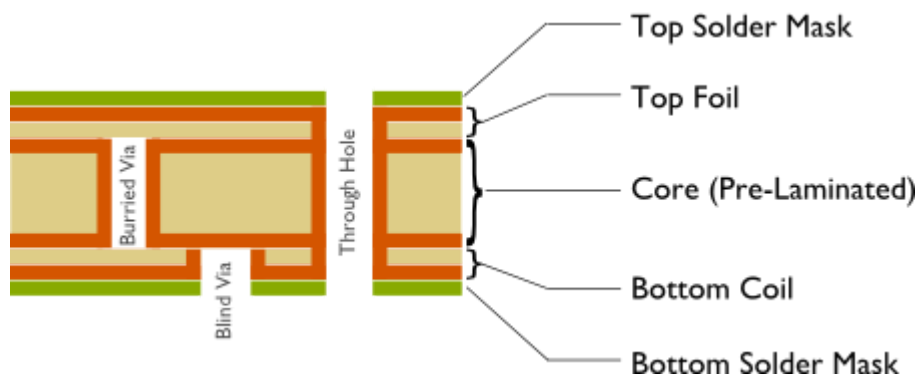
DFM Issues

Whether you view it as an image format or NC data, Gerber doesn't include any sort of "intent" about the design. Worse, it contains no protections from creating designs that are very difficult, if not impossible to manufacture. Here are some common errors to avoid with your PCB designs:

1. Missing files
2. Minimum feature size too small (you'll certainly pay in one way or another for anything under 6 mil traces)
3. Minimum gap size (likewise, spaces less than 10 mils lower yields and increase costs)
4. Size of annular rings (good as they are, automated drills miss by a couple mils sometimes)
5. Acid traps (acute corners in copper features can get eaten into during etching)
6. Slivers (very small, isolated islands or peninsulas of copper can actually flake-off during processing and land somewhere else, potentially causing a short circuit)
7. Silkscreen on pads (it's pretty difficult to solder over the top of white paint)
8. Features too near, or even outside the outline (like the drill, the machine that cuts out the shape of the board can wander and destroy features too close to edges or slots)
9. Drilling into the same hole twice, also known as "double hits" (holes end-up bigger than they should)

Holes

Holes in PCB can be plated with conductive material to carry electrons from one "layer" to another (holes used in this way are referred to as vias) or non-plated. Depending on which set of layers are connected by the hole, vias may be referred to as "buried", "blind" or "through holes":



The above example 4-layer PCB cross-section shows a rigid core of material prelaminate with conductor on each side and two "foil" layers separated from the core by a thin layer of insulating material. The different types of plated holes are labeled. A non-plated through hole (NPTH) is drilled after the board has been fabricated, so it doesn't contain any plating on the internal wall and therefore doesn't electrically connect traces on one layer to another. These types of holes are generally used for mounting hardware (stand-off, push-tabs, etc.) or other non-electrical uses. Plated through holes (PTH) penetrate all layers of the PCB and have interior walls plated with copper. Such holes can be used for mounting "through hole" components, or as mentioned above exist only to carry electrons between conductive layers (vias).

Conclusion

The goal of this article was to provide readers with a little background on the Gerber file format(s) as well as provide some hints on common pitfalls when communicating with manufacturing. I hope I have achieved that goal. If I have, please let me know. If I haven't, definitely let me know. Thanks, and enjoy!

-Ed editor@circuitpeople.com

Copyright © 2008-2019 CircuitPeople LLC, All rights reserved. CircuitPeople™, Got Gerbers?™ and TrueGerber™ are trademarks of CircuitPeople LLC.