



Understanding Gerber PCB Layers

Usually there are layers below in a Gerber:

- Top silkscreen
- Top solder mask
- Top copper
- Bottom copper
- Bottom solder mask
- Bottom silkscreen
- Outline / mechanical
- Drill
- Inner copper

Top Silkscreen (GTO)

The top silkscreen Gerber file contains data to be printed, typically the white or black text on the PCB board, such as part designators, component outlines, logos, polarity indicators, etc. This layer is optional. It is totally OK to have a PCB without silkscreen if that is what you want.

Top Solder Mask (GTS)

Contains data for the protective solder mask which protects the copper. This is typically a green film but can come in a variety of colors. Solder mask layers are negative—meaning filled areas correspond to openings in the solder mask where no solder mask will be applied. These filled areas in the Gerber solder mask layer will leave copper exposed in the final product. This exposed copper is used for component attach and test access points.

Top Copper Layer (GTL)

Contains data for all areas requiring copper, including traces, pads, copper pour, vias, plated through holes, etc. All filled areas correspond to areas with copper plating.

Bottom Copper Layer (GBL)

Identical to top copper, but flipped horizontally around the y-axis. All filled areas correspond to areas with copper plating.

Bottom Solder Mask (GBS)

Identical to the top solder mask, but flipped horizontally around the y-axis. Solder mask layers are negative—meaning filled areas correspond to openings in the solder mask where no solder mask will be applied. These filled areas in the Gerber solder mask layer will leave copper exposed in the final product. This exposed copper is used for component attach and test access points.

Bottom Silkscreen (GBO)

Identical to the top silkscreen mask, but flipped horizontally around the y-axis. This layer is optional. Many single-sided PCBs will not have any silkscreen on the bottom side.

Outline (GKO) and Mechanical Layer (GML)

Contains the board outline, any cut-outs, v-cuts, panel outline, tabs, etc. Without this layer, the manufacturer does not know how to cut out the individual PCBs from the larger panels typically used in manufacturing. Mechanical elements should be included in a single outline layer to prevent some features from being missed unless it makes the outline unclear.

Drill Layer (DRL)

Contains all the drill locations and hole and via sizes. Drill files have numerous file extensions, including .txt or .drl. This layer file must not be confused with the drill drawing or drill map files. Technically not a Gerber file. Some EDA software will export separate files for plated and non-plated holes (vias). This has little meaning, and whether the holes are plated or not will depend on the copper layers.

Inner Copper Layer (GT#)

Multilayer boards may contain additional copper layers for signals or planes (ground or power) between the top and bottom copper layers. Inner plane layers may be in negative format, meaning the filled areas are areas of no copper.

PCB Stencil Gerber Files

Gerber files are also used in the manufacturing of the stencils that are used during the application of solder paste prior to component attachment. Figure 11 highlights a portion of a solder stencil. The solder stencil is aligned to the PCB and placed in direct contact. The solder stencil openings are aligned to exposed copper on the surface of the PCB. The solder paste is forced through the openings by a squeegee blade on the stencil printer.

Top Paste (GTP) and Bottom Paste (GBP)

Contains data to make PCB stencils for the boards. Similar to solder mask layers, the data typically contains openings for surface mount parts only. Openings may be slightly smaller than the solder mask openings or partitioned to reduce the volume of solder paste that is applied to the pads. The bottom paste layer is identical in function to the top paste layer but is flipped horizontally around the y-axis.

Understanding Gerber layers doesn't have to be confusing. Familiarizing yourself with the names and functions of each Gerber layer in PCB manufacturing can enhance your design process and streamline communication with your board house. Hitech has professional engineers who can help you design Gerber files. Just share your idea, and we'll take care of the rest. If you have Gerber, you can send it to us for quotation. We will provide you with the best price. Just email sales3@hitechcircuits.com For more information, please enter <https://hitechcircuits.com/>

