LIBRARY · REFERENCE GUIDE

GERBERS AND THE FAB PACKAGE

Gerbers are the universal files a factory reads to make your board, one per copper, mask, and silk layer. Add the drill, BOM, and placement file and you have the complete fab package.

ONE THOUSAND DRONES ENGINEERING TEAM · VERIFIED 2026-07

Gerbers are the universal files a factory reads to build your board, one file per copper, mask, and silk layer. Add the drill file, the bill of materials, and the placement file, and you have the complete package a fab needs. Export those, check them in a viewer, and you are ready to order.

WHAT A GERBER IS

A gerber file describes one layer of the board as vector shapes: where copper is, where the mask opens, where silk prints. There is one gerber per layer, so a two-layer board has a top copper gerber, a bottom copper gerber, two mask gerbers, two silk gerbers, and a board-outline gerber. Gerber is an ASCII format, human-readable and unambiguous, which is why it has been the fabrication backbone for decades (Ucamco).

- Ucamco. The Gerber format: the de-facto standard for PCB fabrication data (format specification and drill/route data).- ucamco.com

THE MODERN FORMAT

The current format is Gerber X2, which extends the older RS-274X by embedding attributes: which layer each file is, what a pad's function is, even net names. That metadata lets the fab set up your board with less guesswork. When your tool offers X2, use it; a fab that wants plain RS-274X can still read the copper.

THE DRILL FILE

Holes are not in the copper gerbers. They ship as a separate drill file, historically in Excellon format, listing every hole's position and diameter. Without it the fab knows your copper but not where to drill, so the drill file is as essential as the gerbers themselves (KiCad).

- KiCad. PCB Editor (Pcbnew) documentation: plotting gerbers and generating drill files. docs.kicad.org

THE REST OF THE PACKAGE

Two more files turn a bare board into an assembled one. The bill of materials lists every part by reference designator and part number. The placement file, also called the centroid or pick-and-place, gives each part's position and rotation so an assembly machine can place it. Zip the gerbers, drill, BOM, and placement together, and that archive is what you upload.

THE COMPLETE FAB PACKAGE: ONE GERBER PER LAYER, PLUS THE DRILL FILE, THE BOM, AND THE PLACEMENT FILE.

> **PREVIEW BEFORE YOU ORDER**
>
> Load your gerbers into a gerber viewer and look at every layer before you send them. A missing ground pour, a silk layer that did not export, a board outline plotted on the wrong layer: all of these are obvious in a viewer and invisible in the raw files. Five minutes here saves a wrong board and a lead time.

CHECKPOINT

1. What does a single gerber file describe?

a. The whole board as a 3D model

b. One layer of the board as vector shapes

c. The list of parts to buy

ANSWER · B

There is one gerber per layer: top copper, bottom copper, mask, silk, and outline.

2. Which file tells the fab where the holes go?

a. The drill file

b. The top copper gerber

c. The bill of materials

ANSWER · A

Holes ship separately as a drill file (Excellon), listing each hole's position and size.

3. The placement (centroid) file is used for what?

a. Setting the soldermask color

b. Pricing the finished board

c. Telling an assembly machine each part's position and rotation

ANSWER · C

The pick-and-place file positions parts for automated assembly.

- Prerequisite: silkscreen, soldermask, and polarity marks
- See it on a real board: the L1.01 build
- Next: DFM and ordering a board