

# Formats we accept

We accept the most common PCB files: RS-274X Gerber files. Please be sure they have the correct file extensions shown below and put them in a ZIP archive.

We now also accept Cadsoft Eagle .brd files. Upload a .brd file directly or put it in a ZIP archive. Our CAM exports boards with the following layers enabled:

- Silk: dimension tplace tnames
- Soldermask: dimension tstop
- Copper: dimension top pads vias
- Routing: dimension milling
- Drill: dimension drills holes

Use our gerber generator at your own risk!

## Ordering

Upload one or more PCBs on the order page. Gerber files should be inside a .zip archive with standard file extensions:

Gerber file extension	Layer
GTO	Top Silkscreen (text)
GTS	Top Soldermask (the 'green' stuff)
GTL	Top Copper (conducting layer)
GBL	Bottom Copper
GBS	Bottom Soldermask
GBO	Bottom Silkscreen
GML/GKO/GBR*	Board Outline*
TXT	Routing and Drill (the holes and slots)

### **\*Required**

These layers/files are typically needed to manufacturer a complete PCB. **Only the Board Outline (GML/GKO/GBR) is required.**

**Multiple outlines:** if you include multiple board outlines in your file, the industry standard dictates that the **SMALLEST OUTLINE** will be used.