Gerber Labs

Gerber Labs Design Rules:

Board Features	
Feature	**Specification**
max. board size	500 * 500 (mm)
min. trace width	0.15 (mm)
min. trace spacing	0.15 (mm)
min. trace pitch	0.2 (mm)
min. BGA-pad pitch	0.1(mm)
min. spacing trace / via-pad	0.15 (mm)
min. spacing trace / ground plane	0.15 (mm)
min. spacing trace / BGA-pad	0.1 (mm)
min. spacing via-pad / via-pad	0.2 (mm)
min. spacing via-pad / ground plane	0.2 (mm)
min. spacing via-pad / BGA-pad	0.25 (mm)
min. spacing BGA-pad / BGA-pad	0.15 (mm)
min. spacing SMD-pad / SMD-pad	0.15 (mm)
min. spacing ground plane / ground plane	0.15 (mm)
min. spacing PCB edge to trace	0.4 (mm)
min. spacing PCB edge to pad	0.5 (mm)
min. spacing PCB edge to ground plane	0.5 (mm)
min. annular ring	0.15 (mm)
via plating thickness	18 (μm)

Gerber Labs

Drill Specification	
Feature	**Specification**
min. / max. hole size	0.3/6.5 (mm)
positional tolerance	+/- 0.075 (mm)

Solder Mask and Silkscreen Specification	
Feature	**Specification**
min. soldermask web	0.35 (mm)
min. silkscreen	0.2 (mm)

Note: Standards and tolerance per IPC Class II

File Extension Convention:

Top Copper Layer: *.gtl

Bottom Copper Layer: *gbl

Mid-Layer 1: *.g1

Mid-Layer 2: *.g2

Top Soldermask: *.gts

Bottom Soldermask: *.gbs

Top Silkscreen: *.gto

Bottom Silkscreen: *.gbo

Board Outline: *.gko

Top Solder Paste Mask: *.gtp

Bottom Solder Paste Mask: *.gbp

Drill: *.xln or *.drl

Gerber Labs

Note:

1. Replace “ * ” with your file name.

2. All image layers and the drill file should be placed in the same folder before being compressed into a .zip file, other compression file formats are not supported at this time.

List of Supported CAD Software:

We support gerber output from the following softwares:

KiCad

Eagle (.cam file for Gerber export)

Altium

Orcad

gEDA PCB

DipTrace

Revision: 2020/11/10