



Checklist for Gerber Submission

Section 1 : Required Files

- 1. Drill File in Excellon Format
- 2. Bottom Soldermask
- 3. Bottom Copper Layer
- 4. Top Copper Layer
- 5. Top Soldermask Layer
- 6. Top Silkscreen Layer
- 7. Board Outline & Milling Layer
- 8. Bottom Silkscreen (optional)
- 9. Readme.txt (optional, recommended)

Section 1b : Multilayer Files

the below is only required for multilayer PCB orders Skip this section if ordering for 1 or 2 layer boards

- 1. Internal Layers
- 2. Layers Stackup (Layers.txt)

For Standard Service, all the above layers is required. If any layer is missing from the file set, we will not apply it during fabrication For example, if Silkscreen layer is missing among the files, we will assume that the designer does not want silkscreen applied. For Full Panel Service, please submit all the required layers only. Always include silkscreen layer whenever possible. It helps us orientate the layers correctly. If the submissions is for RS-274-D Gerber format please do not forget to include the aperture file.

Section 2 : Design Considerations

- 1. Pased Design Rule Check against Silver Circuits design rules.
- 2. All layers are not mirrored.
- 3. There is no traces or planes 40 mils (1.0mm) from the cutout edge.
- 4. All cutout and milling is under a single Gerber layer.