Checklist for Gerber Submmision

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 Silkscren Layer
- [ ] 7. Board Outline & Milling Layer
- [ ] 8. Bottom Silkscreen (optional)
- [ ] 9. Readme.txt (optional, recommended)

Section 1b : Multilayer Files

Skip this section if ordering for 1 or 2 layer boards

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

If the Gerber layer is missing from the file set, we will not apply it during fabrication. For example, if Silkscreen layer is missing , we will assume that the designer does not want silkscreen applied. 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. Passed Design Rule Check against Silver Circuits design rules.
- [ ] 2. No layers are mirrored.
- [ ] 3. There are no traces or planes 40 mils (1.0mm) from the cutout edge.
- [ ] 4. All cutout and milling is under a single Gerber layer.