



Sunstone Quick-turn Prototype File Submission Checklist

Please read all related technical file setup explanations on our website or print this checklist and pass to the person setting up your pcb files. Your board will be built as-sent without design review.

File Submission; (Submitted during or directly after order placement):

- I am sending 1 zip file containing only the required files for fastest processing.
- I understand order will be manufactured when file is received; no changes can be made.

Gerber Layer File Preparation:

- I have previewed my design using Viewmate from www.pentalogix.com
- My design requirements fall within the standard guidelines -- see capabilities
- Files are Gerber RS274X (embedded apertures) or RS274D (includes separate aperture list).
- All spaces/traces are a minimum of .006 (6 mil) or larger.
- All outer layers (top and/or bottom) are positive and do not require manual merging.
- All holes have proper pads for through-hole connections from top to bottom layer.
- Power and ground layers have sufficient clearances for all holes, including mounting holes to prevent shorting.
- My board is viewed as looking through from top (component) layer side.

Single Sided - 1 layer Boards: YOU MUST PREVIEW 1-Layer Files!

- My board is viewed as looking through from top (component) layer side.
- Right reading copper text is located somewhere on the single side (+ silkscreen if applicable)
- I understand all boards are through-hole plated; Sunstone will add copper pads to bare side of 1-sided pcbs.

4-6 layer order gerber files; (PCBexpress E4 Service Only):

- Inner layers are individual layers (either positive or negative) that do not require manual merging.
- Internal ground plane layers have recommended clearance sizes for all holes, including mounting holes.
- Ground and power layers are clearly marked to indicate proper polarity.

Board Shape Outline:

- Board outline is drawn around perimeter only using a non-breaking trace with no internal cut-outs.

Drill Setup; ASCII text file showing X&Y coordinates for holes: -- see Drill Setup

- I have verified NC drill file is NOT a gerber layer or drawing.
- I understand if my tool sizes do not match the preset drill sizes, they will be placed into our pre-set finish drill sizes: .008, .014, .020, .025, .029, .033, .036, .040, .043, .046, .053, .061, .067, .080, .087, .093, .100, .110, .125, .141, .151, .167, .193, .251 (.008 available for 4-6 layers only).
- I have reviewed the recommended pads & clearances for preset drill sizes.
- I understand all holes are treated as through-hole plated.
- My design does not contain slots, connected holes, blind or buried vias.
- Hole quantity greater than 35 holes per sq-inch may be subject to 10% drill fee for additional machine time.
- I have a separate tool size list or the tool sizes are listed within the header of my excellon drill file.

Solder Mask (if applicable):

- I have included both top and bottom solder mask files.
- Solder mask swell is at least .008 larger than copper surfaces to keep mask off pads.

Silkscreen (if applicable):

- I included a top silkscreen file and optional bottom silkscreen file (needed if added to order).
- Silkscreen does not cover soldering surfaces. No clip performed on this service.
- Silkscreen traces are drawn with .007 (7mil) or larger aperture line width (not height) to ensure legibility.