PCB Order Checklist

This checklist is a tool to help ensure that you have the correct Gerber PCB layout guidelines before submitting your order.

Gerber PCB File Submission:

- □ My files are contained in a single compressed file (.zip) containing only the required files
- □ Once my order is put into process no additional changes to the files will be permitted.

Gerber Layer File Preparation:

- □ Preview files in a Gerber viewer prior to submission to insure that the files have exported as you expected.
- □ Verify that the board design fits into the Sunstone manufacturing Capabilities
- □ Gerber files format is either RS274X (embedded apertures) or RS274D (with aperture list separately)
- □ All external layers (top and bottom) are *positive polarity*
- □ Files do not require manual merging
- □ Power / Ground / mounting holes have sufficient clearance to the copper layers to prevent shorting.

Single Sided - 1 layer Boards: PLEASE PREVIEW ALL 1-Layer Files

- □ Preview files as if viewed looking through the top (component) layer side.
- □ Right Reading copper text is drawn on the single side (+silk screen if applicable)
- □ Single sided boards (**Limited Review** - Prototype) will contain plated through holes (pads will be added to the opposite side)

Multilayer Boards:

- □ Internal layers are individual layers (either positive or negative polarity) that do not require merging.
- □ Internal ground layers contain sufficient clearances for all holes including non-plated and mounting holes.
- □ Ground and Power layers are marked to indicate proper polarity

Board Outline:

- □ Cut shape is drawn around board perimeter using a continuous line.

Drill Setup:

- □ ASCII text file provided showing the X and Y coordinates of the holes.

- ☐ My NC drill file is NOT a Gerber layer or drawing.
- ☐ Separate tool size list included or the tool sizes are listed within the header of my Excellon drill file.

Slot / Cutouts:

- ☐ These features are indicated on the outline layer.
- ☐ The minimum width for a plated slot is 0.020"
- ☐ The minimum width for a non-plated slot is 0.031"

Solder Mask

- ☐ Top and Bottom (as required) files have been included
- ☐ Solder mask swell is at least 0.008" larger than the copper features to keep solder mask off of copper features.

Silk Screen:

- ☐ Top and Bottom (as required) files included
- ☐ Silk screen is drawn with a minimum 0.006" aperture line width to ensure legibility.

Native File Upload:

For EAGLE Users: please include a readme with your object layers defined. If these are not documented:

- ☐ Top silk layer: Sunstone will default to object layer 21 (tplace) and 25 (tnames)
- ☐ Bottom silk layer: Sunstone will default to object layer 22 (bplace) and 26 (bnames)
- ☐ All midlayer object layer selections MUST be documented in your file set.