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 in either Gerber RS274X or ODB++ format

□ 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

□ Files supplied are either Gerber RS274X (embedded apertures) or ODB++

□ All external layers (top and bottom) as well as soldermask and silkscreen layers 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 should be clearly identified which order they should be in

□ 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. (for more info on these requirements see https://www.sunstone.com/cam-resources/inner-layer-clearances)

□ 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:

□ NC or Excellon format drill file provided showing the X and Y coordinates of the holes.

□ 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 or mechanical 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 .006” diameter 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