File format accepted for PCB production:

Only RS-274X Gerber files can be accepted. Please compress all the Gerber files related with one PCB design to one .zip or .rar file.

How to create RS-274X Gerber files:

- **How to convert your Eagle files to Gerber files?**

Download the method
Download Eagle RS-274X Cam Processor

- **How to export Gerber files with PCB-Elegance?**

Download the method

Important info:

• Board outline must be included at least in one layer.
• The pcb dimension is the smallest rectangle that can accommodate the pcb design.
• Milling info must be added in the board outline layer.
• Large holes or shapes that need to be cut out of the pcb must be mentioned in the board outline layer.
• The holes for 2-layer or multi-layer PCBs will have through hole plating.
• Do not panelize many different designs into one panel.
• Non-standard PCBs can be produced on request.

Check your Gerber files:

You can check if your Gerber files are created correctly by using the software called GC-prevue. This is a free Gerber viewer that can be downloaded from the following website:
http://www.graphicode.com/GC-Prevue

Gerber files requirement (the following layers are needed):

Extended Gerber RS-274X (single-layer)

- Silkscreen top
- Layout solder side
- Solder mask solder side
- Drill info
- Drill info (txt)
- Silkscreen bottom (optional)
- Mill info (optional)
- Peel off info (optional)

Extended Gerber RS-274X (double-layer)

- Silkscreen top
- Layout top side
- Solder mask top side
- Layout solder side
- Solder mask solder side
- Drill info
- Drill info (txt)
- Silkscreen bottom (optional)
- Mill info (optional)
- Peel off info (optional)

Extended Gerber RS-274X (4-layer)

- Silkscreen top
- Layout top side
- Solder mask top side
- Layout inner layer 1
- Layout inner layer 2
- Layout solder side
- Solder mask solder side
- Drill info
- Drill info (txt)
- Silkscreen bottom (optional)
- Mill info (optional)
- Peel off info (optional)