

Technical specifications for the supply of Gerber files

The files for the production of printed circuits must be in the following standards:

- Gerber 274 X
- Gerber 274 D + openings tables
- ODB++
- DPF
- NcDrill in metric or imperial format

Any technical specifications contained in the files will not be considered. Gerber files not adhering to the specifications listed on our website will be returned to sender.

In case of layouts with scoring / witnesses in the panel, have not be present different patterns.

Every single Gerber files should contain only one PCB model.

If the panel layout is not indicated in Gerber or in the order notes, the PCB will be delivered single.

The sizes of the PCB to be quoted have to include additional area for bands, milling witnesses, etc.

The outline of the PCB should be indicated with the center of a track present on a file, preferably on the Solder Resist layer.

Diameter of holes have to be listed in the drill file or in a table automatically generated by CAD software.

For the orientation of the PCB will be followed the writings on copper that have to be readable on completed PCB.