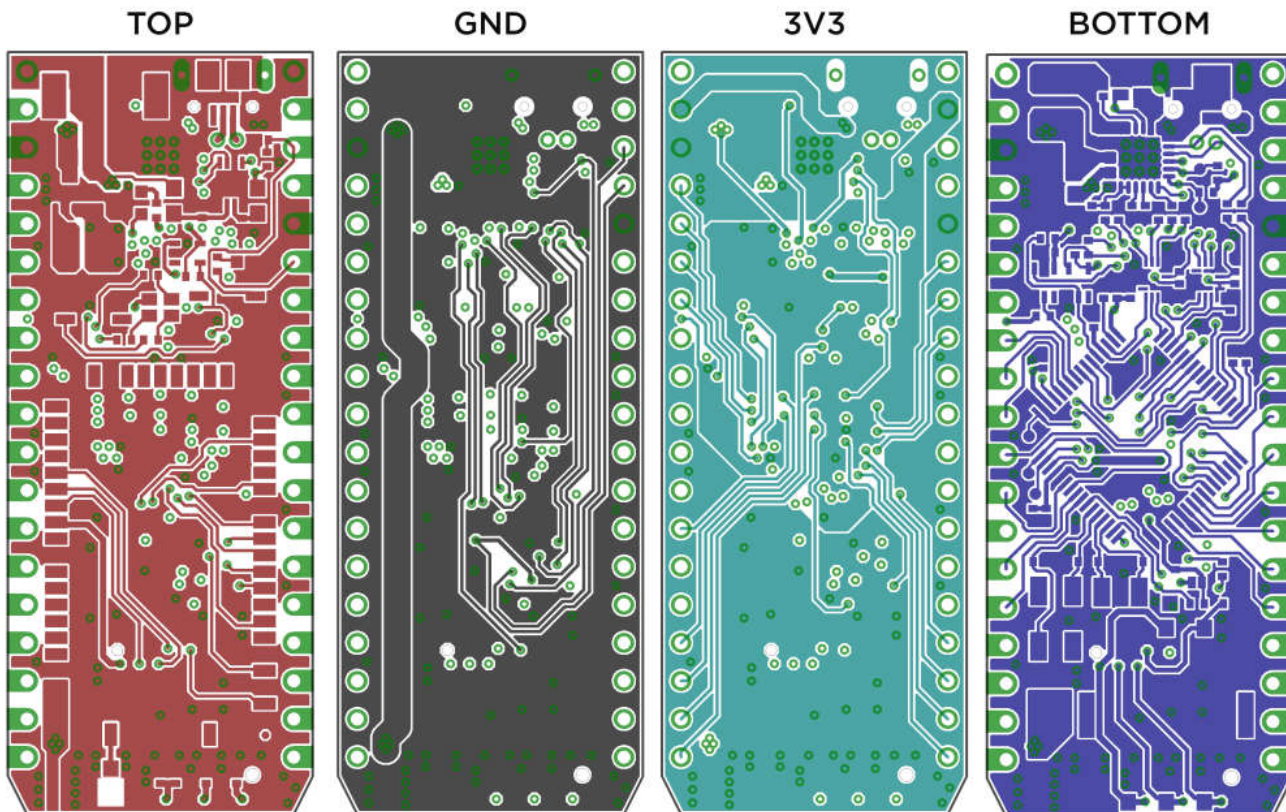# ─RAKO STUDIOS─

Rako Studios » Media » Tech » Electronics » PCB files for your fab house

PCB files for your fab house

Required files for your printed circuit board fabrication.

I asked Oscar at Proto Express what files they need to fabricate a circuit board. He sent the following list. While I usually don't send the assembly drawing, I always make a fab drawing so the specifications are clear.

I once sent out for a board at Advanced Circuits and they had to call back to tell me their inside layers were 1/2 oz (at that time). This saved me some hassle, since I needed the thicker copper for heat dissipation. I don't always make a pdf of the fab, I just send it in Gerber format.

1. Gerber Files (RS-274-X or RS-274-D format)

   - All copper layers (inner & outer)
   - Solder mask layers
   - Silkscreen/Legend layers
   - Via plugging layers (if applicable)
   - Solder paste layers (for board assembly)
   - Aperture list if apertures are not embedded in the Gerber data (i.e. not using RS-274-X)

2. Drill file with tool codes and X-Y coordinates for all holes (ASCII or EIA format).
3. IPC-D-356 netlist (used for continuity testing).
4. Readme File containing Engineering contact information and any special instructions.
5. Fab drawing

   - Board outline, dimensions, including cutouts, chamfers, radii, bevels, scores, etc.
   - Dimensions from a reference hole in the board to a corner or to two sides of the board outline. - A drill chart with the hole symbols on the drawing and the finished hole sizes
   - Material requirements
   - Finished board thickness and tolerance
   - Layer stack-up order
   - Controlled impedance requirements (if applicable)
   - Dimensioned array drawing if the design is to be shipped as a multiple-up array
   - Notes defining any other requirements or specifications pertinent to the design

Optional:

1. Valor ODB++ file - preferred by PCB vendors instead of Gerber format.
2. Check plots in Adobe PDF form. Incoming QC uses this to check PCB.