PCB Design Rules for Commercial Fabrication

Typical design rules

Note: these design rules are not necessarily the absolute limits of any particular manufacturer, but represent a reasonable starting point for typical PCBs. Many manufacturers can handle 6 mil tracks and 6 mil clearances but specifying 8 mil provides a safety margin. There's no point pushing the limits unless your design needs it.

Requirement	**Recommended normal limit**
Track width	8 mil
Clearance	8 mil
Hole to hole clearance	10 mil
Minimum hole size	0.3 mm (≈ 12 mil)
Recommended via dimensions	0.3 mm hole size with 30 mil diameter
Annular ring (minimum)	6 mil
Silkscreen line width	8 mil
Edge to copper clearance	>0.3 mm
Edge to hole clearance	>0.4 mm

Design requirements

- **The coordinate origin must be at the bottom left corner of your board.** This is important because all exported fabrication data will have coordinates referenced against this location.
    - Use Edit → Origin → Set to specify the origin location.
    - If your board has rounded corners, the origin must be where the bottom left corner would be if the board was expanded into a rectangle that fully encloses its shape.
- **There must be <u>absolutely no data</u> outside the board area.** Such data would collide with another board when they are panelized.
    - This applies to all layers seen by the PCB manufacturer: top layer, bottom layer, top and bottom overlays, top and bottom paste, top and bottom solder, drills, and the keepout layer.
- **The board outline must be drawn on the keepout layer.** The keepout layer defaults to outputting the correct Gerber filename ("GKO" extension) that most fabrication houses will interpret as the board outline. There are other ways to work but please follow this process for all JCU boards.
    - If you have the board shape already configured, choose Design → Board Shape → Create Primitives From Board Shape, and select the Keepout layer as the target.
    - If you would like to define a board shape with curved edges, draw the outline on the keepout layer (Place → Keepout → then select the drawing tool). Once you have the outline, select it, and choose Design → Board Shape → Define From Selected Objects.

How to validate your fabrication output

You should always export your PCB layout to Gerber files and then inspect these files to ensure that they look reasonable.

1. Select File → Fabrication Outputs → Gerber files.
2. Choose Inches and 2:4 format (unless your PCB manufacturer requests otherwise).
3. On the Layers tab, ensure that the following layers are ticked in the Plot column:
    a. Top overlay
    b. Top paste
    c. Top solder
    d. Top layer
    e. Bottom layer
    f. Bottom solder
    g. Bottom paste
    h. Bottom overlay
    i. Keep-out layer
4. Press OK. Your generated Gerber files will open in the CAM viewer. Toggle the tick boxes on the left hand side (CAMtastic panel) to view each layer. Check:
    a. **There is a board outline (GKO) that properly indicates the edges of the board. This is the #1 mistake that slows down fabrication. Make sure you have it!**
    b. Top (GTL) and bottom (GBL) copper looks reasonable.
    c. Top silk (GTO) looks reasonable.
    d. Top solder mask (GTS) covers the pads.
    e. Bottom solder mask (GBS) covers the pads on the bottom (which are probably only the through-hole parts).
5. If you are sending your board away yourself, you also need to generate an NC drill file. Go back to the PCB, then select File → Fabrication Outputs → NC Drill Files.
6. Choose the same format as before, e.g. Inches and 2:4 format.
7. Press OK. Your generated drill file will open in the CAM viewer.
8. To see the drills on the same screen as the Gerber files, return to the Gerber panel and select File → Import → Drill and then browse for the files that you just exported. They will be .TXT files in the "Project Outputs" folder of your project.
9. Ensure that the drills line up with the pads and vias.

Current carrying capability

For reference, approximate current capability of 1oz (28.35 gram) copper tracks are:

Current (amps)	Track width (mm)	Track with (inches)
1.0	0.4	0.156
1.5	0.5	0.02
2.0	0.8	0.0312
2.5	1.6	0.0625
3.5	2.5	0.1
4.5	3.2	0.125
5.0	4.0	0.156
5.5	5.0	.2
6.0	6.4	.25