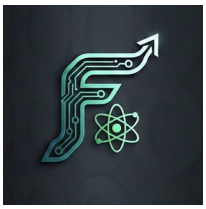

PCB Design Rules

Fabrication & Assembly Guidelines

Rev 1.0 | April 2026

Overview

This document defines the design rules for PCBs fabricated by Foundry. Following these guidelines ensures your boards are compatible with our fabrication and assembly processes and helps achieve the fastest possible turnaround time.

Board Specifications

Parameter	Requirement
Maximum Board Size	150 mm x 100 mm (5.91 in x 3.94 in)
Board Thickness	1.6 mm
Layer Count	2 layers or fewer
Minimum Trace Width	0.250 mm (9.84 mil)
Minimum Trace Spacing	0.250 mm (9.84 mil)
Minimum Copper-to-Edge Clearance	0.5 mm (19.7 mil)
Silkscreen Line Width	0.25 mm minimum
Minimum Silkscreen Text Height	1.0 mm
Recommended Silkscreen Letter Size	2.0 mm x 2.0 mm or greater

Smallest Passive Footprint	0402 (0603 preferred)
Via-in-Pad	Not supported

Via Specifications

Foundry supports two standard via sizes. Designs must use one or both of the options listed below.

Drill Hole Diameter	Via Diameter
1.5 mm (59.1 mil)	2.6 mm (102.4 mil)
0.7 mm (27.6 mil)	1.3 mm (51.2 mil)

Solder Mask

Parameter	Requirement
Color	Green
Solder Mask Expansion	0.2 mm (7.9 mil) per side
Minimum Dam Width	0.5 mm (19.7 mil)

Surface Finish

Boards are shipped with bare copper pads. No surface finish (HASL, ENIG, OSP, etc.) is applied. For best solderability, it is recommended to assemble boards promptly after fabrication to minimize copper oxidation.

Board Edge & Outline

Parameter	Requirement
Minimum Copper-to-Edge Clearance	0.5 mm (19.7 mil)
Board Outline Tolerance	±0.2 mm (±7.9 mil)
Minimum Inside Corner Radius	0.5 mm (19.7 mil)
Routing Method	CNC milling (1 mm end mill)

All internal cutouts and routed slots are subject to the 0.5 mm minimum inside corner radius. Design board outlines and cutouts with filleted corners where possible.

File Format Requirements

Foundry accepts standard Gerber and drill files as exported by major EDA tools. The following formats are supported:

File Type	Accepted Formats
Gerber Files	RS-274X, Gerber X2
Drill Files	Excellon / NC Drill

Tested EDA Exports

- **KiCad** Default Gerber export (Gerber X2 format with Excellon drill files).
- **Altium Designer** Default Gerber export (RS-274X format with Excellon drill files).

Turnaround Time

Single-layer boards with only through-hole components will provide the fastest turnaround time.

If your design can be routed on a single layer using only through-hole components, this is the recommended approach for the quickest delivery. Single-layer boards with SMD components and two-layer boards are fully supported but may require additional processing time.

Design Tips

- Use 0603 passive footprints where possible for improved manufacturability and yield.
- Avoid via-in-pad designs; route vias away from component pads.
- Keep trace widths and spacing at or above the 0.250 mm minimum to ensure reliable etching.
- Prefer single-layer routing with through-hole components when feasible for the fastest turnaround.
- Use silkscreen letter sizes of 2.0 mm x 2.0 mm or larger for best results. Smaller text down to 1.0 mm is supported but may suffer from resolution issues below 1.0 mm x 1.0 mm. Maintain a minimum line width of 0.25 mm.
- Design board outlines with filleted inside corners (0.5 mm radius minimum) to match the CNC routing process.
- Assemble boards promptly after receiving them to minimize copper oxidation on bare pads.

Questions?

For design rule clarifications or fabrication inquiries, contact us at:
hello@foundrypcb.com