# Gerber Export Guide (EasyEDA)

This board uses a standard 2-layer stackup with a \*solid GND plane\* on the bottom. Use these settings to export Gerbers and a drill file your fab (e.g., JLCPCB/PCBWay) will accept.

## EasyEDA

1. \*Fabrication → Generate Fabrication File (Gerber)\*.

2. Ensure these layers are enabled: \*TopLayer, BottomLayer, TopSilkLayer, BottomSilkLayer, TopSolderMaskLayer, BottomSolderMaskLayer, BoardOutline\*.

3. \*Drill: Excellon format; units \*\*mm\*.

4. Download the ZIP from EasyEDA.

---

## Fabrication Notes to Include

- \*Material:\* FR-4, Tg ≥ 135 °C

- \*Layers:\* 2

- \*Thickness:\* 1.6 mm

- \*Copper:\* 1 oz (35 µm) both sides

- \*Finish:\* ENIG or HASL (lead-free)

- \*Solder mask:\* any color; silkscreen white

- \*Impedance control:\* not strictly required; keep D+/D− as short, parallel tracks

- \*Panelization:\* let the fab do it unless you need multiples

---

## Pre-Flight Checklist

- [ ] DRC clean (no unconnected nets, no mask slivers, no overlaps)

- [ ] Text readable (no mirrored references unless intended)

- [ ] Fiducials added if planning assembly

- [ ] Mounting holes correct

- [ ] USB shell connected to GND via ESD path

- [ ] All test points labeled