

Your First PCB: From Schematic to Manufacturing

End-to-end walkthrough of designing your first PCB in KiCad, schematic, layout, manufacturing files, ordering. Targets a simple sensor breakout board.

Intermediate

45
min

PCB
Design

Jun
26,
2026

blitztechelectronics.co.zw/learn/your-first-pcb-from-schematic-to-manufacturing/

Designing your own PCB feels intimidating until you've shipped one. This guide walks you through every step using KiCad (free, open-source) and EasyEDA (browser-based, free). We'll design a simple DHT22 + LED breakout, then send it to JLCPCB to come back as a real board.

1 Pick a small first project

Resist the urge to design your dream board on attempt #1. Pick something with ≤ 5 components, 2 layers, no high-speed signals, no controlled impedance. A DHT22 breakout with a power LED, a pull-up resistor, and a 4-pin header is perfect.

2 Draw the schematic

Install KiCad 8 from kicad.org. Open the schematic editor. Place: DHT22 symbol, 10k Ω resistor, 220 Ω resistor, LED, 1 \times 4-pin pin header. Wire them: VCC \rightarrow DHT22 pin 1, DHT22 pin 2 \rightarrow header pin 'DATA' AND through 10k Ω to VCC, LED + 220 Ω from VCC to DATA for visual feedback. Run Tools \rightarrow Annotate Schematic, then Inspect \rightarrow Electrical Rules Check. Fix any errors before moving on.

3 Assign footprints

Tools \rightarrow Assign Footprints. Pick through-hole footprints for everything for your first board (you can hand-solder TH; SMD requires more practice). DHT22 \rightarrow Sensor_AM2302; resistors \rightarrow R_THT_Axial_DIN0207; LED \rightarrow LED_THT_D5.0mm; header \rightarrow PinHeader_1 \times 04_P2.54mm_Vertical.

4 Lay out the board

Tools \rightarrow Switch to PCB Editor. Import the netlist. Drag the parts into a roughly 30 \times 30 mm rectangle. Set design rules: 0.3 mm trace width, 0.2 mm clearance, 1.6 mm board thickness. Route the traces (manually, autoroute makes ugly boards on your first try). Add a copper pour on the bottom layer connected to GND. Define the board outline by drawing a closed rectangle on the Edge.Cuts layer.

5 Run DRC and export gerbers

Tools \rightarrow DRC (Design Rule Check). Fix any errors. File \rightarrow Plot, choose Gerber format, select all the standard layers (F.Cu, B.Cu, F.Mask, B.Mask, F.Silkscreen, B.Silkscreen, Edge.Cuts). Also generate the drill file. Zip all the gerbers together, that's your manufacturing package.

6 Order from JLCPCB

Upload the zip to JLCPCB.com. 5 boards in green soldermask, 2 layers, 1.6 mm thick = about \$5 + shipping. Shipping to Zimbabwe via DHL is roughly \$20-30 and arrives in 7-14 working days.

BlitzTech consolidates shipments for clients in Zimbabwe, we order in batches monthly and split shipping costs. Contact us if you want to piggyback on our next run.