⚡ BlitzTech Learn

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, 10kΩ resistor, 220Ω resistor, LED, 1× 4-pin pin header. Wire them: VCC → DHT22 pin 1, DHT22 pin 2 → header pin 'DATA' AND through 10kΩ to VCC, LED + 220Ω from VCC to DATA for visual feedback. Run Tools → Annotate Schematic, then Inspect → Electrical Rules Check. Fix any errors before moving on.

3 Assign footprints

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

4 Lay out the board

Tools → Switch to PCB Editor. Import the netlist. Drag the parts into a roughly 30 × 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 → DRC (Design Rule Check). Fix any errors. File → 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.