

Circuit to Board: PCB Design with EasyEDA

A one-day schematic-to-prototype workflow

FA-0666 · 1 day

Embedded Systems and Hardware

Introduction

A focused introduction to turning a simple electronic schematic into a PCB layout with EasyEDA.

Participants follow a prepared small-board example through components and nets, placement, routing, checks and fabrication-output review.

One-day practice concentrates on a low-voltage prototype design. Autorouting, manufacturing and four-layer considerations are demonstrated or discussed; a training layout is not automatically fabrication-ready or validated for safety-critical or mains-powered use.

Learning outcomes

- Distinguish schematic intent, component symbols, footprints and physical board layout.
- Create or adapt a simple schematic and organise a small PCB for routing.
- Apply introductory placement, trace, via, copper-pour and mechanical checks.
- Inspect available 2D/3D views and exported Gerber, drill and BOM files.
- Explain prototype ordering requirements and when multilayer design requires further study.

Prerequisites

- Basic familiarity with electronic components such as resistors, capacitors, diodes, transistors, connectors, and ICs
- Ability to read simple circuit diagrams
- General computer literacy
- A laptop or desktop with a modern web browser and internet access

- Helpful but not mandatory: prior exposure to breadboards, prototyping, or hobby electronics

Use the arranged EasyEDA edition and an account/project environment suitable for the teaching exercise; features and workflows differ between editions.

Training outline

1. 1. PCB Fundamentals and the Design Environment

- Design flow: idea, schematic, layout, fabrication and assembly; prototype versus production.
- Single-, two- and four-layer boards; substrate, copper, pads, tracks, vias, silkscreen, solder mask and board outline.
- Schematic versus layout; manufacturability constraints and common beginner mistakes.
- EasyEDA workspace, libraries, schematic/PCB editors and supported output tools; project naming, saving and revision habits.

2. 2. Schematic Capture and Component Choices

- Organise a schematic, wire nets, label signals, set power/ground symbols and annotate references.
- Choose symbols and matching footprints; verify pin mapping, values, connectors and selected devices.
- Review electrical-rule warnings and component suitability before converting the design.

3. 3. Placement and Board Planning

- Create the outline and transfer schematic connectivity; interpret ratlines.
- Select useful grids and organise components by function, orientation and routing needs.
- Account for mechanical clearances, connector/edge positions and power/signal paths.
- Use a prepared ESP8266/ESP-12F relay-control example with low-voltage demonstration loads; examine power, switching and connector sections.

4. 4. Routing and Board Details

- Route selected nets manually; compare power/signal widths and supplier-specific clearance/drill constraints.
- Use vias and layer transitions; review optional autorouting rather than trust it without inspection.
- Add mounting holes, cutouts, copper pours and readable polarity/revision markings.
- Review copper connectivity, thermal connections, board edges and placement in available 2D/3D views.

5. 5. Fabrication Outputs and Manufacturing Context

- Generate and inspect the supported Gerber/drill file set and a BOM; check exports before handoff.
- Read manufacturer requirements and compare board parameters and prototype-ordering steps without placing an order.
- Survey imaging/etching, drilling, plating, solder mask, silkscreen, surface finish and electrical testing.
- Explain the distinction between fabrication data, assembly data and tested circuit behaviour.

6. 6. Four-Layer Overview and Final Review

- Compare two- and four-layer stackups, planes, return paths, noise, vias and introductory signal-integrity considerations.
- Complete a small example or selected section of the prepared control-board layout.
- Review unresolved connectivity, footprint, mechanical and manufacturing checks; plan prototype testing and a later revision.