

KiCad PCB Engineering from Circuit Concept to Production

Target Audience

This course is suitable for electronics learners, graduate engineers, technicians, circuit developers, and working professionals who want to turn an electronic circuit idea into a verified, production-ready printed circuit board using KiCad.

Course Objective

The course aims to develop an end-to-end PCB engineering workflow. Participants move from circuit and board fundamentals to library setup, schematic capture, design verification, physical layout, performance improvement, and final manufacturing release.

Course Outcome

- **Select an appropriate board structure** and establish a well-organized KiCad project for the intended circuit.
- **Develop connected, multi-sheet circuit drawings** and verify their electrical logic before layout.
- **Translate a verified schematic into a practical board arrangement** with suitable placement, layers, planes, and routing.
- **Improve board reliability** by considering heat, high-speed behavior, signal and power quality, and electromagnetic compatibility.
- **Release a documented manufacturing package** containing fabrication data, component information, and assembly guidance.

Course Outline

The course comprises **32 hours** of theory and practical labs and is divided into **7 progressive chapters** that follow the complete workflow from circuit planning and KiCad project setup to verification, physical board development, performance improvement, and production release. Guided practical exercises are included throughout the course to help participants apply and evaluate the concepts covered.

Module 1. From Electronic Circuit to Printed Board

- Relate basic electronic principles to the role of a printed circuit board
- Distinguish between single-sided, double-sided, and multilayer board construction
- Identify the components and graphical symbols commonly used in PCB projects

Lab 1: Classify board types and interpret the parts of a sample circuit

Module 2. Building a KiCad Design Workspace

- Navigate the KiCad tools used for schematic and PCB development
- Create and organize a new PCB project with clear file and library management
- Bring external symbols and footprints into the project library

Lab 2: Prepare a reusable KiCad project and add an external library item

Module 3. Capturing and Organizing the Circuit

- Draw the electrical circuit as a structured schematic
- Choose, place, and adjust component properties
- Create logical connections with nets, aliases, and off-page links
- Divide larger designs into hierarchical and multi-sheet sections

Lab 3: Produce a connected multi-sheet schematic

Module 4. Checking the Design Before Layout

- Use electrical checks to locate schematic connection problems
- Simulate circuit operation and review the expected response
- Resolve identified issues before beginning the physical board design

Lab 4: Check and simulate a circuit before transferring it to the board

Module 5. Shaping, Placing, and Routing the Board

- Set up the PCB Editor and define the physical board boundary
- Arrange components according to practical placement principles
- Choose between manual and automatic routing approaches
- Distribute connections across one, two, or multiple copper layers
- Create and manage ground and power plane regions

- Run board design checks and correct layout violations

Lab 5: Build, route, and check a multilayer PCB layout

Module 6. Improving Electrical and Thermal Performance

- Plan board geometry and routing for high-speed signals
- Evaluate signal quality and power-delivery behavior
- Reduce electromagnetic interference and improve EMC performance
- Control component temperature and provide effective heat dissipation

Lab 6: Review a board for high-speed, thermal, and EMI/EMC risks

Module 7. Production Release and Board Assembly

- Inspect the completed board in 3D before production release
- Export Gerber fabrication data for the PCB manufacturer
- Compile the Bill of Materials required for component sourcing
- Review the main processes used for PCB assembly and soldering

Lab 7: Assemble and verify a complete PCB manufacturing package