PCB Design & Fabrication Track

Comprehensive Workflow from Concept to Functional Board

Presented by the Sapiens AI Team

Source: 352+ Thousand Circuit Board Background Royalty-Free Images, Stock ...

Program Goal & Core Curriculum

Program Goal

End-to-End PCB Workflow Proficiency

Develop comprehensive skills spanning the entire printed circuit board (PCB) lifecycle, from initial concept to a fully functional physical product.

Key Program Phases

Online Foundational

Master essential theoretical knowledge and software fundamentals (e.g., KiCad Schematic Editor, PCB Editor) through interactive online modules.

Offline Project

Apply learned concepts in hands-on, practical projects, culminating in physical board design, fabrication, and assembly.

Core Topics Covered

Design

Schematic capture, component selection, library management, PCB layout, routing, and 3D visualization.

Fabrication

Understanding manufacturing processes, design for manufacturability (DFM), and layer stack-ups.

Assembly

Component placement, soldering techniques (SMT), and testing procedures for functional circuits.

Culminating Project

Participants will consolidate learning by designing and building a **Custom Power Supply** or a **Sensor Interface**.

Phase 1: Foundational & Core Concepts (Online)

Month 1 — Online Delivery — Month 2

Building Strong Theoretical Understanding

Core Principles: Establish a robust foundation in fundamental electrical engineering concepts and circuit theory essential for effective PCB design. Understand component characteristics, signal integrity, power distribution, and EMC basics.

Design Methodologies: Learn industry-standard design methodologies and best practices, preparing for practical application in EDA tools. Focus on conceptualizing electronic systems before diving into specific tool workflows.

Practical EDA Tool Skills & Mastery

KiCad/Eagle Focus: Gain hands-on proficiency with industry-leading EDA software, primarily focusing on **KiCad**. Explore its powerful Schematic Editor for hierarchical designs, and get an introduction to Eagle as an alternative.

Mastering Schematic Capture: Develop expertise in translating circuit concepts into precise digital schematics. This includes component placement, intelligent wiring, and assigning essential component footprints.

Mastering PCB Layout: Learn to transform schematics into manufacturable layouts using **KiCad's PCB Editor**. Manage up to **32 copper layers** and leverage the **3D Viewer** to inspect mechanical fit and preview the final

Month 1: Introduction to PCB Design & Schematic Entry

Week 1 | Week 2 | Week 3 | Week 4

Week 1: PCB Workflow & KiCad Basics

- **PCB Workflow:** From concept to manufacturing.
- **KiCad Basics:** Installation, interface navigation, and project creation.
- **Component Intro:** Understanding basic electronic components and symbols.

KiCad | Interface | Components

Week 2: Schematic Entry Fundamentals

- **Schematic Purpose:** Logical representation of the circuit.
- **Capture Basics:** Placing symbols, drawing wires, nets, and power/ground.
- **Libraries:** Utilizing KiCad's extensive component libraries.

Symbols | Wires | Nets

Week 3: Advanced Schematic & Libraries

- **Hierarchical Schematics:** Organizing complex designs

Week 4: Transition to PCB Layout

- **Netlist Generation:** The bridge from schematic to layout

Month 2: Advanced Layout & Fabrication Preparation

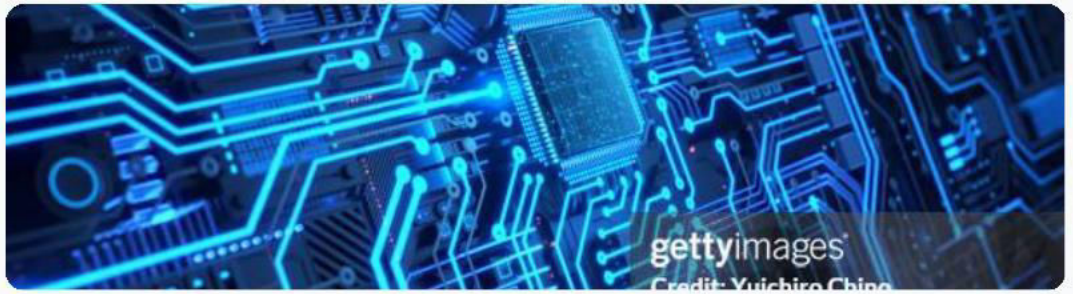
Week 5 & 6

Week 5: Component Placement & DRC

Learn systematic placement for optimal thermal management, signal flow, and mechanical fit. Implement and interpret Design Rules Checks (DRC) in KiCad to verify layout integrity and use the 3D viewer to inspect mechanical fit.

Placement DRC 3D Viewer

Week 6: Routing Techniques



Master routing on single and double-layer boards. Explore advanced strategies for critical signals like differential pairs to minimize integrity issues, leveraging the powerful features of KiCad's PCB Editor.

Single-Layer Double-Layer Signal Integrity

Phase 2: Industry Immersion & Capstone Project (Offline)

Month 3

Intensive Offline Experience

Industry Immersion & Direct Mentorship

Transition into a dedicated, state-of-the-art lab environment for a month-long, intensive offline experience, simulating real-world engineering workflows.

Benefit from personalized, direct mentorship by experienced industry professionals. Engage in collaborative problem-solving and receive immediate feedback to accelerate practical skill development.

Applying Learned Concepts & Capstone Goal

This phase is the culmination of all prior learning. Participants will apply every concept from schematic design and PCB layout to fabrication preparation and assembly.

The core objective is to independently **Design, Fabricate, Assemble, & Test a Functional PCB**. This project is the ultimate test of integrated skills and problem-solving.

Offline Immersion **Direct Mentorship**

Hands-on Assembly **Functional PCB**

Capstone Mini Project: Design & Fabrication Prep

Capstone Project: Design & Prep Phase

Weeks 9 & 10: Project Initiation & Detailed Design

Week 9 — **Week 10**

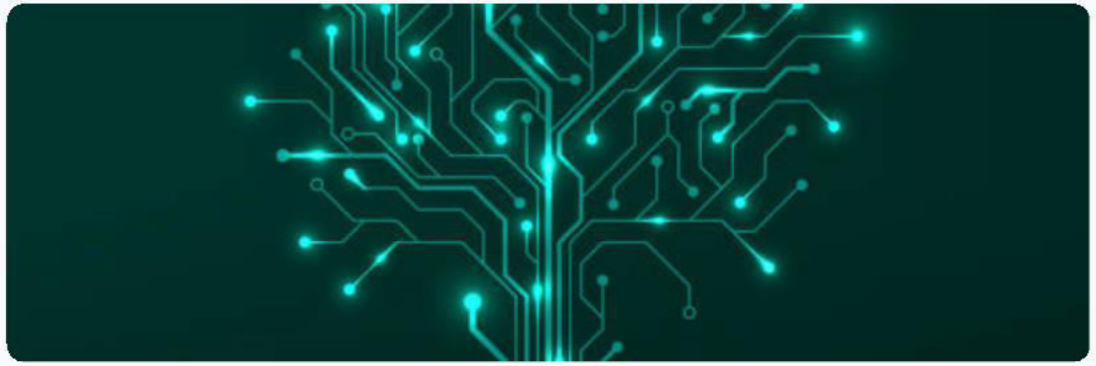
Detailed Design & Design Rule Compliance

Schematic & Sourcing: Finalize the circuit schematic, ensuring component values are optimized. Complete component sourcing, considering lead times and costs.

PCB Layout: Execute the complete PCB layout in KiCad, focusing on strategic component placement and precise routing for signal integrity and thermal management.

Verification: Perform rigorous Design Rule Checks (DRC) and Electrical Rule Checks (ERC) to identify and resolve any layout or

Manufacturability Review & File Generation



DFM/DFA Review: Conduct a thorough Design for Manufacturability and Assembly review to ensure the board can be reliably and cost-effectively produced.

File Generation: Generate all necessary manufacturing files, including Gerber (RS-274X), Excellon drill files, and a comprehensive Bill of Materials (BOM).

Final Verification: Meticulously inspect all generated files using a Gerber viewer to confirm accuracy and prevent costly

Week 11: Hands-on Soldering & Assembly Techniques

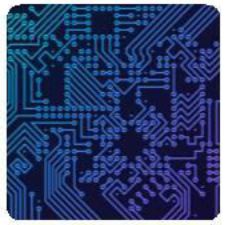
Module 3: Assembly & Testing

Week 11

Foundational Skills: Safety, Tools & Component Prep

Soldering Safety & Tool Proficiency: Prioritize safety with proper ventilation, eye protection, and a stable workstation. Master soldering iron temperature control, tip care, and the effective use of solder, flux, wick, desoldering pump, and basic hand tools.

Component Identification & Polarity: Develop skills in accurately identifying components (resistor



Through-Hole (TH) Soldering Best Practices

Learn the 'heat the pad and lead, apply solder to the junction' method. Focus on creating shiny, conical solder fillets, avoiding cold joints, solder bridges, and excessive solder.

Concave Fillet | Proper Wetting | No Cold Joints

Basic Surface Mount Device (SMD) Soldering

Explore techniques for applying solder paste, precise component placement with tweezers, and carefully reflowing using a fine-tip soldering iron or hot air station.

Fine-Tip Iron | Precision Tweezers | Hot Air Basics

Desoldering & Rework Techniques

Master essential desoldering tools like solder wick and solder suckers. Learn systematic rework procedures for correcting errors like lifted pads or solder bridges.

Week 12: Project Assembly, Testing & Troubleshooting

Capstone Project Final Stage

Week 12

Final PCB Assembly & 'Smoke Test'

Meticulous Component Assembly: Execute the final population of the PCB, ensuring correct orientation, secure connections, and clean solder joints for both Through-Hole and SMD parts.

Initial Power-up & 'Smoke Test': Conduct the crucial first power-up with caution. Visually inspect for immediate failures and monitor initial current draw to prevent damage.

Pre-Power Checks: Prior to power-up, meticulously check for shorts between power and ground, continuity of critical traces, and correct component polarity using a multimeter.

Secure Assembly | Visual Inspection | Power-up Protocol | Short Check

Systematic Troubleshooting & Debugging

Structured Approach: Implement a methodical approach to fault isolation, from system-level checks to component-level issues.

Common Pitfalls: Identify and resolve issues like cold joints, solder bridges, misaligned parts, and power shorts.

Utilizing Test Equipment: Leverage tools effectively to pinpoint fault locations and understand failure modes.

Root Cause Analysis | Iterative Debugging | Safety First

Functional Testing with Industry Tools

Multimeter Applications: Use for precise voltage, current, and resistance measurements to verify power integrity and circuit operation.

Oscilloscope Diagnostics: Employ to analyze dynamic signals, confirming waveform shapes,

Documentation & Showcasing

Project Documentation: Compile design files (schematics, Gerbers), BOM, test procedures, and logs.

Mastering PCB Design & Fabrication

Comprehensive Workflow Mastery

Gain a holistic view of the PCB development lifecycle, from initial schematic capture to fabrication and assembly considerations. Understand each stage's critical role and interdependencies, leveraging KiCad's extensive capabilities with up to **32 copper layers** and **14 technical layers**.

Full Lifecycle | KiCad Proficiency | Multi-Layer Design

Hands-On Practical Experience

Develop practical proficiency through direct engagement with real-world design tasks. Master creating circuits in KiCad's Schematic Editor, designing PCBs in its Layout Editor, and physically assembling components to produce functional boards.

Schematic to Assembly | Physical Prototyping | Direct Application

Professional Development Readiness

Equip yourself with advanced skills and problem-solving methodologies necessary to tackle complex modern PCB designs, including rigorous Design Rules Check (DRC/ERC), Design for Manufacturability (DFM)/Assembly (DFA)

Real-World Application Skills

Master the application of theoretical knowledge to create reliable electronic circuits, encompassing effective component selection, signal