

PCB DESIGN AND FABRICATION

Date: 13 – 14 October 2023

Platform: Offline

Venue: College of Engineering Trivandrum (CET)

Host: IET On Campus CET

Speaker: Mr. Sriram M, Department of Electrical Engineering, CET

Attendees: 40 students

The *PCB Design and Fabrication* workshop was a two-day offline training program aimed at giving students end-to-end exposure to the theory, design, and practical fabrication of printed circuit boards (PCBs). Conducted by Mr. Sriram M from the Department of Electrical Engineering, the event combined structured technical instruction with extensive hands-on practice.

Day 1 – Fundamentals of PCB Design and KiCAD Training

The first day laid the groundwork by introducing participants to the essential concepts and tools required for PCB design.

Session Highlights:

1. Different Methods of Circuit Making – An overview of various approaches such as breadboarding, stripboard prototyping, and PCB-based solutions, highlighting their advantages and limitations.
2. Types of PCBs – Explanation of single-layer, double-layer, and multilayer PCBs, their structure, applications, and manufacturing considerations.
3. PCB Designing Software – Introduction to popular design software, with a focus on open-source solutions.
4. PCB Etching – Theoretical discussion on chemical etching techniques, materials involved, and safety protocols.

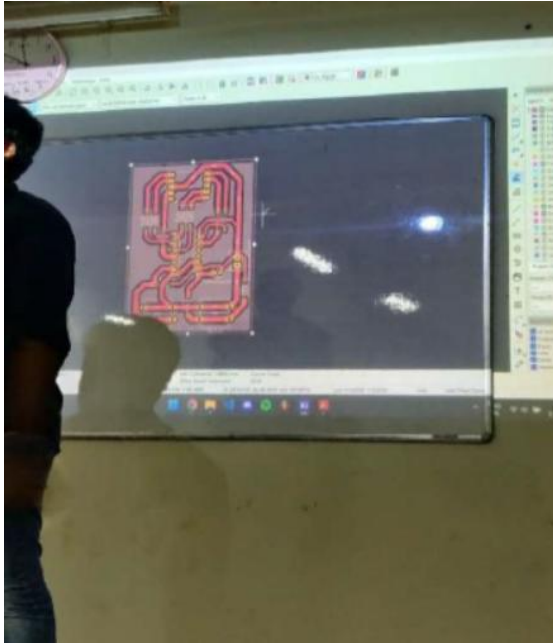
KiCAD Tutorial:

The latter part of Day 1 was an in-depth tutorial on KiCAD, a professional-grade open-source PCB design suite. The instructor demonstrated:

- Setting up a new project and schematic capture.
- Adding components from the library and assigning footprints.

- Creating electrical connections and verifying the schematic.
- Converting the schematic into a PCB layout.
- Routing single-layer boards efficiently.

Participants then applied these techniques to design their own single-layer PCB layouts, gaining familiarity with KiCAD's interface and workflow.



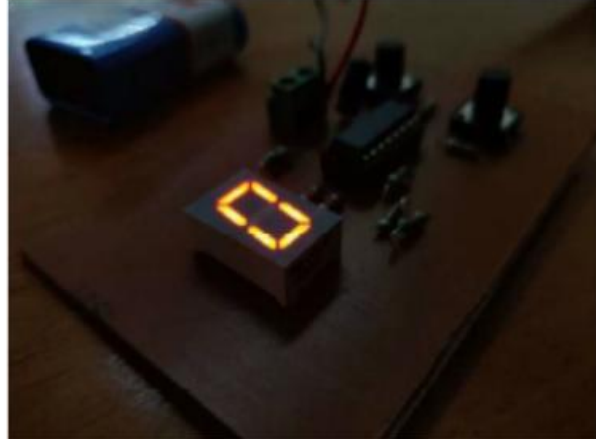
Day 2 – PCB Fabrication Hands-On

Day 2 transitioned from virtual design to physical creation, allowing students to fabricate and assemble the PCBs they had designed the previous day. Participants were grouped into teams of three, encouraging collaboration and peer learning.

Fabrication Process:

1. Toner Transfer – Using printed PCB layouts, participants transferred the design onto copper-clad boards.
2. Etching with Ferric Chloride – Guided through safe handling procedures, teams etched away excess copper to reveal the circuit traces.
3. Drilling Component Holes – Precision drilling of through-holes using PCB drilling machines.
4. Soldering Components – Placement and soldering of electronic components to complete functional circuits.

By the end of the session, all teams successfully powered up their boards, witnessing their designs come to life. The tangible outcome, combined with the satisfaction of building a functional circuit from scratch, made the experience both educational and motivating.



Outcome and Feedback:

The workshop not only improved participants' technical understanding of PCB design and fabrication but also enhanced their practical skills in using industry-relevant tools. Students expressed that the real-world application of concepts, coupled with immediate results, made the learning highly impactful.