Workshop Report 1: **PCB Designing and Fabrication**

Event Overview

The "PCB Designing and Fabrication" workshop was conducted from 06th October 2025 to 08th October 2025 and was organized by the Department of Electrical & Electronics Engineering at Annamacharya University (formerly Annamacharya Institute of Technology and Sciences, Rajampet). The workshop was held in collaboration with Technotran.

The main goal of the workshop was to introduce participants to the essential concepts of Printed Circuit Board (PCB) design and fabrication techniques, as well as provide hands-on experience in designing and assembling PCBs. It was tailored to cater to both theoretical understanding and practical knowledge, preparing students for real-world PCB design and manufacturing processes.

Objectives of the Workshop

- Understanding PCB Design: To provide participants with foundational knowledge of how to design effective PCBs, considering factors like component placement, signal integrity, and circuit layout.
- Hands-On Fabrication: To offer hands-on experience in the PCB fabrication process, from designing the schematic to producing the final circuit board.
- Tools and Techniques: To introduce participants to industry-standard tools used for PCB design and fabrication.
- Practical Applications: To equip students with the ability to design functional PCBs for various electronic systems and understand how they can be used in real-world applications.

Key Focus Areas

1. Introduction to PCB Design
    - Overview of PCB design and its importance in modern electronics.
    - Types of PCBs: Single-sided, double-sided, and multilayer PCBs.
    - Basic PCB terminology, including traces, vias, pads, and components.
    - Design Considerations: Signal integrity, power distribution, and grounding techniques for high-performance circuits.
2. PCB Design Process
    - Schematic Design: Introduction to schematic capture tools and circuit design.

- Layout Design: Using PCB design software (such as Eagle, KiCad, or Altium Designer) for creating PCB layouts. Focus on correct component placement and routing.
- Design Rule Checks (DRC): Ensuring that the design adheres to industry standards and is free of design errors that could affect functionality.
- Gerber Files: The creation and significance of Gerber files for PCB manufacturing.

3. PCB Fabrication Techniques
   - PCB Manufacturing Process: Overview of the PCB fabrication process, including copper etching, drilling, and layering.
   - Etching Process: Introduction to the chemical etching process used to create traces on the PCB.
   - Soldering and Assembly: Practical demonstration of soldering electronic components onto the PCB. Introduction to both manual soldering and automated soldering methods.
   - Testing and Debugging: How to test and debug a PCB after fabrication to ensure it works as intended.
4. Hands-On PCB Design and Fabrication
   - Participants were introduced to design tools and software used in the industry, with a hands-on session that guided them in designing their own simple PCBs.
   - After designing the PCBs, participants fabricated their own PCBs using the standard etching process and assembled components onto the boards.
   - A troubleshooting session helped participants understand how to detect common mistakes during fabrication and assembly.

Training and Hands-On Sessions

The workshop was structured to balance theoretical learning with hands-on activities:

- Introduction to PCB Design Software: Participants were taught how to use PCB design software to create schematics and layout for their PCB designs.
- PCB Fabrication Demonstration: A step-by-step demonstration was provided for the process of fabricating a PCB, including etching and drilling.
- Assembly and Testing: Students assembled components onto their fabricated PCBs, followed by testing to ensure that the circuit functioned properly.

- Real-World Projects: The participants had the opportunity to design and fabricate a basic functional PCB, applying everything they learned throughout the workshop.

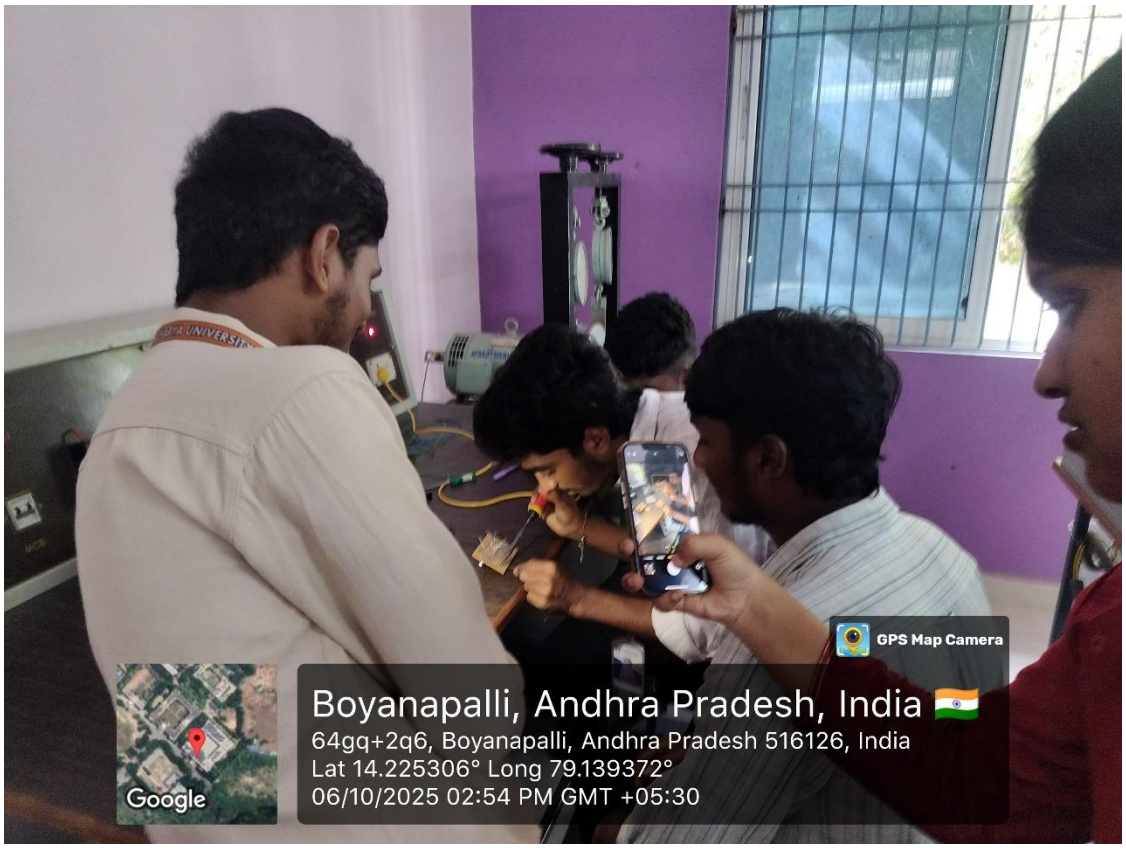
Outcome of the Workshop

- Knowledge and Skills Acquired: Participants learned the core principles of PCB design, including designing circuits, understanding layout considerations, and using industry-standard tools for design and fabrication.
- Hands-On Experience: Students gained direct experience in PCB fabrication, from designing the schematic to assembling the components and testing the final product.
- Improved Understanding of the PCB Manufacturing Process: The workshop enabled participants to understand the practical challenges and processesinvolved in turning a design concept into a functional PCB.

Number of Students Participated

The workshop had active participation from 104 students, primarily from electrical, electronics, and related engineering disciplines, who gained valuable knowledge and practical skills in PCB design and fabrication.




GPS Map Camera
Boyanapalli, Andhra Pradesh, India
64gq+2q6, Boyanapalli, Andhra Pradesh 516126, India
Lat 14.225306° Long 79.139372°
06/10/2025 02:54 PM GMT +05:30
Google


GPS Map Camera
Boyanapalli, Andhra Pradesh, India
64gq+2q6, Boyanapalli, Andhra Pradesh 516126, India
Lat 14.225307° Long 79.139366°
06/10/2025 02:54 PM GMT +05:30
Google


GPS Map Camera
Boyanapalli, Andhra Pradesh, India
64gq+2q6, Boyanapalli, Andhra Pradesh 516126, India
Lat 14.225323° Long 79.139367°
06/10/2025 02:53 PM GMT +05:30
Google


GPS Map Camera
Boyanapalli, Andhra Pradesh, India
64gq+2q6, Boyanapalli, Andhra Pradesh 516126, India
Lat 14.225293° Long 79.139416°
06/10/2025 02:55 PM GMT +05:30
Google