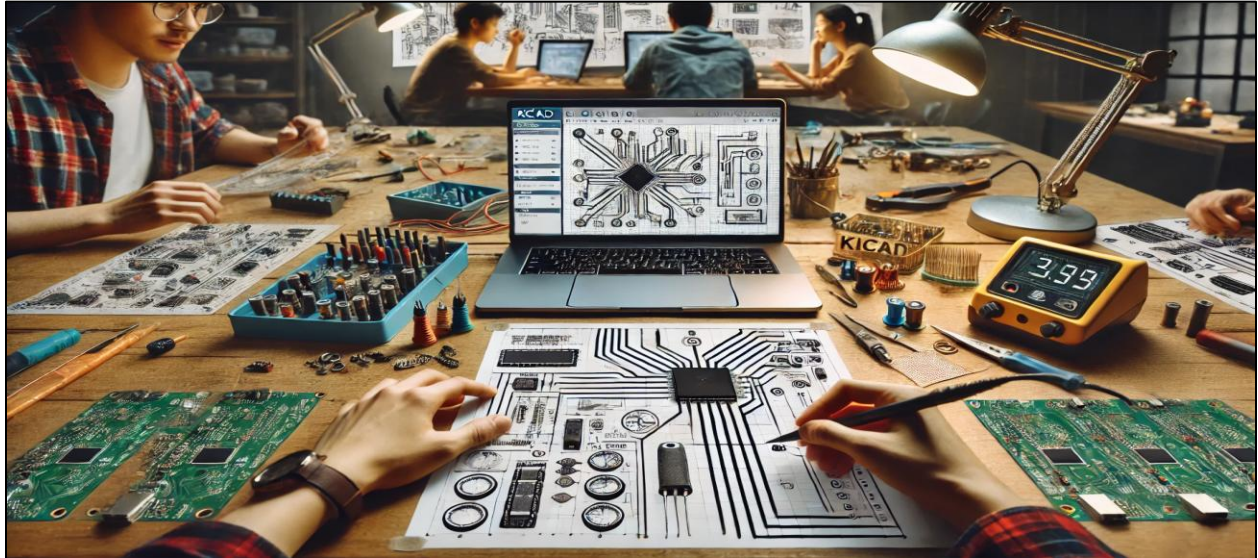


PCB Design Workshop



INTRODUCTION:

Printed Circuit Boards (PCBs) are the backbone of modern electronics, providing a compact, reliable, and cost-effective solution for connecting electronic components. PCBs are essential in industries ranging from consumer electronics to aerospace, making PCB design a critical skill for engineers and hobbyists alike. KiCAD, an open-source software, offers a user-friendly platform for designing PCBs, from simple circuits to complex multi-layer boards.

ABSTRACT:

This workshop introduces participants to the fundamentals of PCB design using KiCAD. Participants will learn to create schematics, design layouts, perform rule checks, and generate Gerber files for manufacturing. A practical session on etching will complement the software training, providing a complete understanding of PCB prototyping. By the end of the workshop, attendees will have hands-on experience designing and fabricating a basic PCB circuit.

DISCUSSION IN WORKSHOP

Total Duration: 120 Minutes

Software Required: KiCAD

TOPIC	DETAILS
1. Introduction	Basics of PCBs, applications, software, and why KiCAD.
2. Designing Process	Overview of PCB design steps explained with a simple LED circuit example.
3. Hands-on PCB Design	Design a 555 Timer-based circuit for blinking an LED using KiCAD.
4. Etching Process	Overview and Hands-on demonstration of the etching process

PREREQUISITES:

This workshop is designed for individuals from various backgrounds and is beginner-friendly. No prior experience is required, but a basic understanding of electronic components such as resistors, capacitors, LEDs, and fundamental circuit design principles will be helpful. Enthusiasm to learn and experiment with PCB design is the only essential prerequisite!

Participants are encouraged to **bring their own laptops** with KiCAD pre-installed to ensure seamless participation in the hands-on sessions.