



OVERVIEW: Hands-On-Training on PCB Design Using Kicad (Open Source) Software

Why is PCB designing important to students?

Professionals in PCB design are needed by almost every electronics manufacturer, be it a small-scale industry or an MNC. PCB design is very critical to the performance of the hardware. The mechanical strength and aesthetics of a product also depend on how well the PCB has been designed. With the rapid growth of the embedded systems industry, the demand for good PCB designers is also increasing rapidly.

Module 1: Introduction

Introduction to different types of PCBs
Familiarisation with electronics components

Module 2: Schematic creation

Understanding schematics and symbols
Searching components footprints and symbols
Choosing the right components
Schematic drawing
Editing symbol libraries
Running ERC

Module 3: Board creation

Manual routing
Component-placing
Practice via manual routing on PCB
Design verification
Designing of single-sided PCB

Using of packaged libraries

Auto routing

Editing and creation of components

Module 4: Report generation

Net list

Bills of Material (BOM)

Gerber file creation

Drill legend generation

Module 5: Fabrication

Printing of design on copper clad sheet using toner transfer method

Etching process using FeCl₃

Drilling of PCB

The aim is to make a Pin Through Hole (PTH) PCB below 5MHz using various electronics components, such as ICs, transistors, diodes, passive components, switches and connectors.

Note:

1. Course fee is Rs 4,000 + 10.3% service tax = Rs 4,412 (including registration, training, reference CD and take away project)
2. Timings for week days are 10:30 AM to 5:30 PM.
3. Timings for weekend classes are:
Saturday 1.30 PM - 5:30 PM; Sunday 10 PM - 5 PM
4. Students will receive certificates from EFY on successful completion of the course.
5. A CD, which contains the course content, tutorial notes and Kicad software, will be provided at the end of the course.