

# Circuit & PCB Layout Design

*“Ceda circuit design training course will help you to build the skills to complete your prototype, do modification of an existing design*

*Circuit and PCB is the building block of any Electronics device & it's a important phase of product development”.*

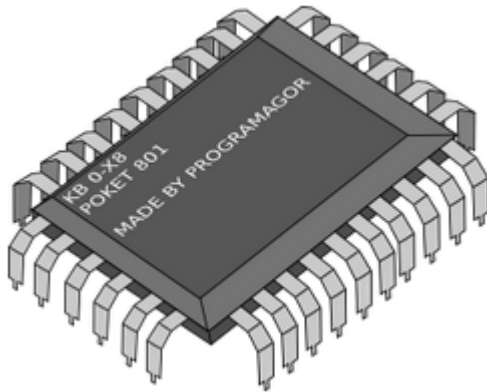
*Circuit and PCB Design course systematically designed for anyone who wants to know Electronic Design procedure from scratch to finish. At the end of the course, you will be able to make parts, building a schematic through OrCAD Capture/Altium Designer Software and build component footprint, design a PCB through OrCAD Layout/Altium software- confidently and generating Gerber files for PCB manufacturing.*

*The course will provide you a head start to understand the PCB manufacturing and design process. The PCB design skill will gradually develop as you do more projects and practice the software regularly.*

## Course Overview

### Learning Objectives

- Component - Introduction
- Getting acquainted with various component
- Identifying components through visual check
- Component Handling
- Component Functions
- Datasheet reading
- Replacing a part with equivalent part
- Part selection from Digikey
- Selection of component for your design
- Introduction to Smart designing - Online



- Few examples – DC-DC converter
- Filter design
- Amplifier design

*\*This training will help you to be a part of Electronics Project Team and those who are involved in Electronics Projects All Alone. PCB Design with OrCAD course may help you to get a better JOB opportunity and JOB switchover.*

*“Circuit & PCB Layout Design Training will help you to get JOBS as a member in R&D Team as you will know components, circuits and pcb design better than a FRESHER”*

- Protecting you circuit
- Designing Circuit protection from
- **Polarity**
- **Surge**
- **EMC**
- Interfacing Input and output device
- Driver function

### Finally

- Discussing designing a complete circuit  
LAB SESSION



**“PCB is the building block of any Electronics device & PCB DESIGN is an important phase of product development”.**

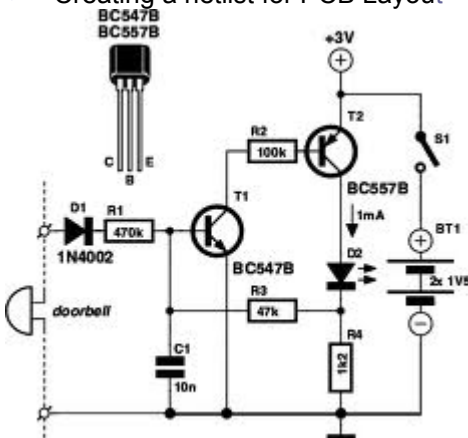
**PCB Design module** systematically designed for anyone who wants to know PCB Design procedure from scratch to finish. At the end of the course, you will be able to make parts, building a schematic through OrCAD/Altium Software and build component footprint, design a PCB through OrCAD/Altium Layout software-confidently and generating Gerber files for PCB manufacturing.

The course will provide you a head start to understand the PCB manufacturing and design process. The PCB design skill will gradually develop as you do more projects and practice the software regularly.

## Circuit Design through EDA Tool

### Course Overview

- Building parts and symbols
- Creating a new project
- Creating multi-sheet flat designs
- Copying work between projects
- Assigning reference designators
- Design checking
- Adding intersheet signal references
- Creating a Bill of Materials and other reports
- Adding part and net properties
- Creating a netlist for PCB Layout



### Learning Objectives

#### Session1

- Getting started with Schematic software
- Building a simple schematic
- Processing a design
- Building a multi-sheet schematic
- Editing properties - LAB SESSION

#### Session2

- Creating parts and symbols
  - Preparing a design for Layout
  - Creating Netlist
- LAB SESSION

## PCB Layout Design

### Course Overview

- Learn how to create footprint libraries
- Use spreadsheets to manage design data and rules
- Develop skills in part placement as well as interactive and automatic routing
- Post-process board designs to prepare them for PCB - manufacturing

### Learning Objectives

#### Session 1

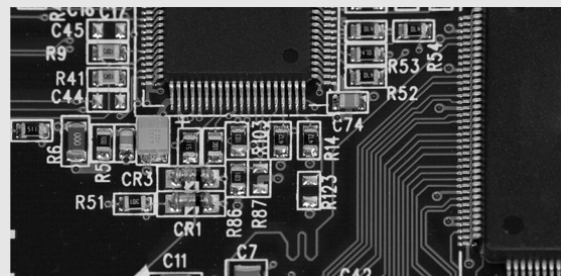
- The Layout user interface
  - Setting up your environment
  - Layer Stackups
  - Creating padstacks and footprints
  - Placing components
- LAB SESSION

#### Session 2

- optimizing placement
  - Fanout and thermal reliefs,
  - Interactive routing single-multilayer
  - Autorouting
- LAB SESSION

#### Session 3

- Copper pour
  - Back annotation
  - Post processing
- LAB SESSION



At the end of the course you should be able to kick start designing a Circuit and converting the circuit to a PCB Layout which you might give it tot a PCB-Fabricator and build a physical prototype