

# Introduction to PCB Design

**Course Description:** This course provides a comprehensive introduction to PCB (Printed Circuit Board) design, covering fundamental concepts, essential terminology, and industry-standard workflows. Learners will explore CAD tools, schematic creation, PCB layout, and manufacturing processes. Through hands-on exercises, students will gain practical experience in designing, documenting, and preparing PCB projects for fabrication and assembly.

**Audience Profile:**

- Engineering students and professionals interested in PCB design
- Electronics hobbyists looking to enhance their design skills
- Individuals transitioning into PCB design roles
- Anyone interested in learning Altium Designer and related PCB design tools

**Prerequisites:**

- Basic understanding of electronics and circuit components
- Familiarity with schematic diagrams
- No prior experience in PCB design required

**Course Objectives:** By the end of this course, learners will be able to:

- Understand the fundamentals of PCB design and its significance in electronics
- Navigate and utilize PCB design software effectively
- Create schematic diagrams and design PCBs using CAD tools
- Select and place components appropriately within a schematic
- Develop and optimize PCB layouts for manufacturing
- Generate fabrication and assembly files required for PCB production
- Understand the PCB manufacturing and assembly process

## **Table of Contents:**

### **Unit 1: Introduction to PCB Design**

- Introduction and Important Terminology
  - What is Printed Circuit Board Design?
  - PCB Design Software
  - Basic Terms to Get Started
- The Role of CAD Tools
  - Schematic Sheets
  - PCB Layout
  - Bill of Materials (BOM)
  - PCB Libraries
  - Summary
- Your First PCB Project
  - How to Create a New Project
  - Make Your Project Shareable

### **Unit 2: Understanding Schematics**

- Anatomy of a Schematic
  - What is a Net?
  - Power and Ground Ports
  - Notes in a Schematic
  - How Components Are Used in a Schematic
  - Multi-Part Symbols
  - Inputs, Outputs, and Bi-directional Pins
  - Ports for Connections Between Schematics
  - Summary
- Types of Schematics

- Hierarchical Schematics
  - Flat Schematics
  - What to Do Before Building Schematics
  - Summary
- Selecting and Placing Components
  - Where to Find Components
  - Finding Components in Altium Designer
  - The Components Panel
  - Other Resources to Find Components
  - Where to Place Components in a Schematic
  - Organizing Schematics
  - Understanding the Electronics Supply Chain
  - Exercise: Open Your Project and Recreate a Schematic
  - Summary

### **Unit 3: PCB Layout**

- What is a Printed Circuit Board?
  - Anatomy of a PCB
- The PCB Design and Manufacturing Process
  - The Role of a PCB Designer
  - Design Tasks Outside Your ECAD Software
  - Preparing for Manufacturing
- The PCB Design Workflow
  - What About Solo Designers?
  - Next Steps

### **Unit 5: PCB Manufacturing**

- Preparing for Manufacturing

- PCB Fabrication Files
  - PCB Assembly Files
  - Generating PCB Manufacturing Files
  - DFM Evaluation and Analysis
  - DFA Inspection
- Panelization
  - An Example Panel
  - Standard Panel Sizes and Orientation
  - Depanelization
  - Creating a Panel in Altium Designer
- The PCB Fabrication Process
  - Inner Layer Processing
  - Outer Layer Processing
  - Final Electrical Testing and Inspection
  - Summary