

ALTIUM DESIGNER BASIC ELVEL TRAINING

Course Summary

Duration: 3 Days (9:30am – 4:30pm)	Provider: ACAD Solutions Sdn Bhd
Venue: Malaysia	Course Type: Instructor In-led Classroom

Course Description



Altium Designer Basic Level Training (3-Day Course)

This 3-day course is aimed for students who are new to Altium Designer Software. The main objective is to enable participants to learn how to create PCB Board from scratch, starting from Schematic and PCB to fabrication outputs and documentation outputs, using Altium Designer. In Addition, students will also get to learn topics such as Creation of libraries and Multi-Sheet Design.

A recognized training certification will be presented after completion of the course as proof of their attendance to an accredited Altium Designer training.

Upon completion of this course, participants will:

- Understand Altium Designer workspace and user interface.
- Create PCB Project from scratch
- Create Schematic and PCB Library
- Understand the usage of Multiple schematic sheets designs

ALTIUM DESIGNER BASIC LEVEL TRAINING CONTENTS

Chapter 1: Design Environment

Chapter 2: Schematic Libraries and Components

Chapter 3: Schematic Capture

Chapter 4: Circuit Simulation

Chapter 5: Defining PCB Board Shape

PCB Board Wizard, Define Board Shape manually, Define Board Shape using Coordinates, 2D Mechanical (*.dxf, *.dwg) or 3D Step model (*.step) file

Chapter 6: Design Rules

Chapter 7: Routing Essentials

Interactive routing | Interactive differential pair routing | Interactive multi-routing

Chapter 8: Polygon Pours & Copper Regions

Chapter 9: Fabrication Outputs

Gerber Files | NC Drill Files

Chapter 10: Smart PDF and Bill of Materials

Chapter 11: Understanding and Using Output Jobs

Chapter 12: PCB Library

PCB Library Footprint According to Specifications | PCB Library Using Component Wizard | Adding 3D Bodies to a Footprint.

Chapter 13: Schematic Library

Creating schematic components with single & multiple parts

Chapter 14: Multi-Sheet Design

Building a Hierarchical structure by defining different ways of net connectivity.

Chapter 15: PCB Layer Stack Management

Chapter 16: Editing Multiple Objects