

PCBKEY

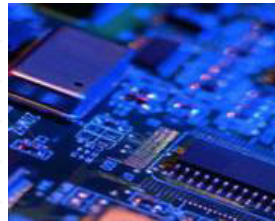
PCB Design & Embedded System Starter Kits

ONLINE PCB Design | Signal Integrity Courses | Embedded System Design

Course | Digital Design with Verilog Course

BOOKS –Printed / DVD format

Product Catalog



PCB Design Training course on demand at your desk without leaving your workspace -

No-campus attendance No commuting No travel cost No deadline pressure. All exams are open-book/open notes Learn at your own pace; complete in as little as 25 days! Anyone with Electronics background - Enroll any time and enjoy affordable fee.

The modules start with solid basics. **Download FREE PCB design tutorial from www.ceda.in before you begin the course.** You will begin by learning the important principles that are the fundamental rules for just about, whatever you may do in PCB design. Then, you will get detailed instruction in techniques and procedures. Gradually, we will guide you through exciting hands-on projects that you can complete at your kitchen table.

Academia, amateurs, professionals and research labs across the India and around the world would appreciate the kinds of technical skills we teach. There are no educational prerequisites to enroll. However, you should be familiar with basic electronics, and comfortable with fundamentals.

YOUR PROFESSIONAL TUTOR

Online tutor



Training on demand at your desk without leaving your workspace, saves travel time and cost

Learn the skills you need to succeed as a PCB Design Engineer
- at home, at your own pace

Disclaimer: Due to CEDA's policy of continuous improvement, specific course content is liable to change without notice. Therefore, this agenda should be taken as a guide only and does not form part of any agreement between CEDA and any other party

Updated November 27, 2013

PCBKEY

KEY to your ONLINE PCB DESIGN & SIGNAL INTEGRITY TRAINING needs



E / DISTANCE LEARNING

Keeping your design skills up to date requires continually learning and exploring the latest tools, techniques, and methodologies. However, it can be hard to fit training and travel time into busy project schedules. PCBkey offers Online training as a flexible and cost-effective alternative to site-based training. PCBkey Training is a part of CEDA located at New Delhi, INDIA. For more details about CEDA visit www.ceda.in

About Online/Distance Certificate course?

Online Training is a Virtual Classroom audio-video environment that allows you to feel in live training events without the need to travel. You listen to lectures, participate in lab exercises, ask questions, and receive feedback just as you would do in a conventional classroom— except you do it from the **convenience of your desktop**, or anywhere you have an Internet and phone connection. It saves the hassle, expense, and travel time to a training site.

Why choose Online/Distance course?

Save 75% training cost to improve your productivity by learning the very latest tools. Virtual classes consist of multiple three-hour sessions spread over several days, leaving the rest of each day free for you to concentrate on your design projects.

What I need at my end?

A Windows based PC preferably with Internet connection or a Phone

Who will be benefited?

Engineering graduates, Instructors, design engineers, new Altium users, Board Designers, Embedded designers, project team members.

What is inside the Course material Package?

1. PCB Design Software
2. Training Manual
3. MULTIMEDIA DVD with audio-video Classroom tutorial just likes live session.
4. Certificate after project completion.
5. Support – one month free support through phone & email.



Course Material

How it works? How I practice? What is the support level?

Install the PCB Design Software package from the DVD inside the course material.

Training consists of conventional written discussions (in Adobe Acrobat®) of subjects, including links to multimedia demonstrations.

Show me now

A click on the "Show me now" opens an audio-video clip showing exactly how to perform a specific action. User controls are available to pause or repeat the clip until the action is understood. Additionally, the same PCB design files used to create the demonstrations are included, allowing further practice for LAB session

Practice the LAB session on your own – WERE STUCK? A Click on the "Show me now" opens an audio-video clip showing exactly how to perform a specific action. ...**Still not clear?** Phone or email us – Your Specialized Tutor will be happy to help you. At this end **almost 24x7**, one-specialist tutor will be present to **support** - clarify your doubts through either email or phone.

Feedback

"The course provided me with a better understanding of PCB manufacture and will assist in resolving PCB failures. Also, I will now be able to design PCB to a standard to create ease of manufacture and reliability".

Vinod Sharma, Max Remmer - Samsung

Dr. M. Narayanan – American University of Sharjah

Victor Naobia – Nigerian College of Aviation Technology, Zaria

Pankaj Verma – Southampton, UK



A practical ONLINE tutorial session to help you, design PCBs through CADENCE & ALTIUM TOOL ...

Altium® Schematic

Duration: 9 hours

Course Overview

The Altium designer Training class will teach you the workflow of creating a design. You will be guided through the steps necessary to capture schematics. Course highlights include generating electrical components, adding/updating parts, generating netlists, generating reports (BOM), and interfacing with Altium Layout.

Hands-On Labs

Throughout this course, extensive hands-on lab exercises example will help you understand better. Hands-on lab topics include:

- Navigating the Interface
- Creating and Modifying Drafting Items
- Creating Libraries and Library Objects
- Designing a one page Schematic
- Setting Board Properties
- Outputting Netlists and Plots

Learning Objectives

- User Interface
- Mouse Functions and the Main Toolbar
- The Status Window and Toolbars
- Design Preferences and Display Colors
- File Management

- Libraries
- The library structure
- Using the Library Manager
- Library List
- Searching Libraries
- Creating Part Types
- Creating a New Library
- Create an IC Decal and Part Type
- Drafting
- The Drafting Toolbar
- Adding Text
- Creating 2D Lines
- Design Preparation
- Design rules - Hierarchy structure, rules interface, Default rules and extended rules option, Clearance, Routing and High Speed rule categories, Conditional rules and printing rule reports
- Board layer preparation - number of layers, naming, routing and plane types, assigning nets to planes, routing direction, increasing and decreasing layers and documenting physical board make-up
- Designing a Schematic
- Printing and Documentation
- Print/Plot Schematics
- Report Files
- Bill of Materials Setup
- Creating a Netlist

The course kit comes with

1. Altium Designer Software evaluation CD
2. Training Manual in DVD
3. MULTIMEDIA **DVD** with audio- video Classroom tutorial just like live session
4. ALTIUM reference design and LAB session files
5. Exercise
6. Support – one month free support **...satisfaction guaranteed**

Continued ...

Altium® Layout

Duration: 20 hours

Course Overview

The Altium Layout class will teach you the workflow of laying out a printed circuit board using the latest version of Altium Designer. You will be guided through the steps necessary to design a printed circuit board. Course highlights include creating physical components, importing netlists, adding/updating parts, routing connections and Gerber file generation for pcb fabrication.

Detailed lab exercises help reinforce the discussion during the lectures and provide you with extensive tool usage experience under the guidance of our industry expert instructors.

You will learn how to

- Customize and work with the Altium Layout interface
- Create Board Outlines, Keepouts and 2D Line items
- Create Parts Types, Decals and Pad Stacks
- Import netlists, create board stackup and set up design rules
- Place parts manually and use cross probing capabilities
- Route a design using manual/interactive routing and editing tools
- Create copper shapes and plane areas
- Verify the design
- Generating Gerber Data
- Work with ECO files
- The Decal Wizard for instant creation of common component footprints
- Using Pad Stacks to change pin pad shapes on board and documentation layers

Learning Objectives

User Interface

- Graphical interface - toolbars
- File management - importing/exporting file types
- Changing tool defaults/creating and using start-up files
- Setting layer and object colors
- Altering settings through Preference and Status interfaces
- Mouse/keyboard command combinations
- Board Outlines and Cutout (holes in boards)
- Setting global (whole board) and local (small area) height restrictions
- Dimensioning and measurement preferences

Library

- Fundamentals of Altium Layout library structure
- Accessing libraries and changing library settings
- Part Types – assigning decals, gate and pin swaps, refdes prefix, alphanumeric pin numbers, implicit pins and attributes
- Decals – creating component footprints by adding pins, silkscreen outlines, polarization indicators and copper

Placement

- Grid and Design Preferences
- Design Toolbar placement tools

Routing

- Design Toolbar routing tools
- Routing
- Routing options, adding vias and layer pairs
- Creating through hole, partial vias and via layer pairs
- Modifying trace widths

Copper Generation

- Drafting Toolbar copper tools
- Copper and copper cut outs
- Copper Pours – net assignment

Design Verification

- Measuring clearances with the View Clearance tool
- Using the Verify Design utility to check the design against design rules
- Working with Error Markers

Output Generation

- Generating reports
 - Gerber files for PCB FABRICATION
-