

PCBKEY

PCB Design Starter Kits

ONLINE PCB Design | Signal Integrity Courses

BOOKS –Printed / CDRom 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.

CONTENTS

- About PCBkey ONLINE Training benefits - Page 2
- Training on Mentor Graphics Tool - Page 3 - 6
- CAM350 - Page 7
- Training on Hyperlynx Signal Integrity Analysis - Page 8- 9
- Books - Page 10
- FAQ - Page 11
- Price and Payment options - Page 12

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 January 8, 2013

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

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

What is an 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?

Fresh engineering graduates, Instructors, design engineers, new Cadence & Mentor-Graphics users, Board Designers, Embedded designers, project executives, project leaders, PCB Designers, HiSPEED Designers, Signal Integrity Engineers & busy executives.

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 & MENTOR Graphics TOOL ...

PADS® Logic

Duration: 9 hours

Course Overview

The PADS Logic 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 PADS 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
- Cross probing - PADS Layout integration

Learning Objectives

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

- Libraries
- The library structure
- Using the Library Manager
- Library List
- Searching Libraries
- Creating Part Types
- Using the CAE Decal Editor
- Creating a New Library
- Create an IC Decal and Part Type
- Using the CAE Decal Wizard
- 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
- OLE
- Object Linking and Embedding

The course kit comes with

1. PCB Design Software evaluation CD
2. Training Manual
3. MULTIMEDIA **DVD** with audio- video Classroom tutorial just likes live session
4. PADS reference design and LAB session files
5. Certificate after project completion
6. Support—1 month free support..**Satisfaction guaranteed**

Continued ...

PADS® Layout

Duration: 20 hours

Course Overview

The PADS Layout class will teach you the workflow of laying out a printed circuit board using the latest version of Mentor Graphics PADS Layout. 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, generating reports and Gerber information.

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 PADS 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
- Biasing objects for selection with the Filter
- Mouse/keyboard command combinations and Modeless Commands

Drafting

- Drafting Toolbar commands and Changing Drafting Preferences
- Creating and combining text and 2D-Lines
- Creating and saving frequently used objects to the Library

- Board Outlines and Cutout (holes in boards)
- Keepouts for various restrictions
- Setting global (whole board) and local (small area) height restrictions
- Dimensioning and measurement preferences

Library

- Fundamentals of PADS 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 with the: Route Editor, Dynamic Route Editor, Pin to Pin Autorouter, and Bus Route Editor
- Routing options, adding vias and layer pairs
- Protecting traces (gluing)
- Using the Trace Length Monitor
- Creating through hole, partial vias and via layer pairs
- Adding stitching vias
- Modifying trace widths

Copper Generation

- Drafting Toolbar copper tools
- Copper and copper cut outs
- Copper Pours – flooding, net assignment, and thermal generation
- Creating copper fill, patterns and solids
- Flood, Hatch command and viewing options
- Creating modifying split/mixed planes and individual thermal control

Design Verification

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

Output Generation

- Generating reports
 - Gerber files for PCB FABRICATION
-

PADS[®] Hi SPEED OPTION *Includes* PADS Hi Speed Router

Course Part# **PCBkeyPHSD709**
Pricing USD164/ INR 7049

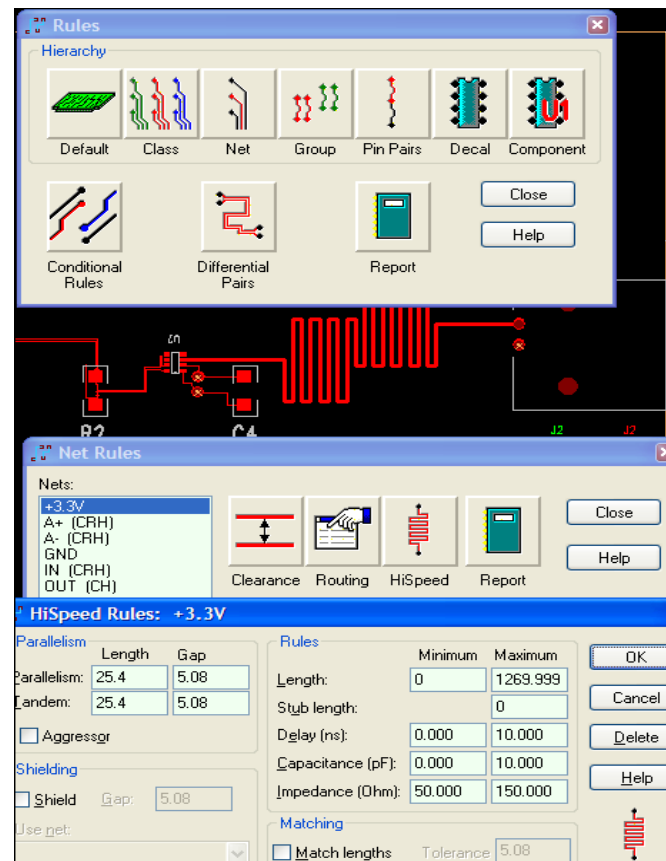
PADS[®] Hi Speed option

Duration: 9 hours

Course Overview

PADS Advanced Rule set option gives maximum control of PADS Layout extensive design rule hierarchy, necessary for today's high speed designs

- High Speed Introduction
- When High Speed
- HSD Rules
- Impedance Control Trace Implementation
- Net Length and maintaining for Net Tuning
- Differential pair
- Matched pair
- Shielding
- Shielding rules
- Routing High-Speed through PADS Router **Check next page**



PADS® Hi Speed Routing *continued*

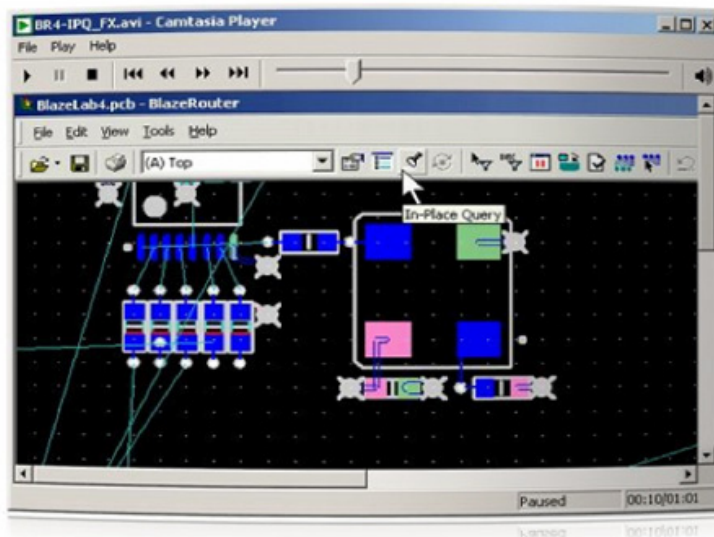
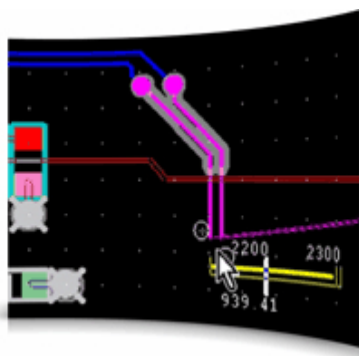
PADS® Router HSD

Duration: 9 hours

Course Overview

PADS HSD Router® Training covers:

- When you should (and shouldn't) use PADS Router®
- PADS Router® Link PADS Router® Monitor and File Management
- PADS Router® windows, menus and toolbars and interface customization
- Keypad commands, mouse operation and shortcut keys
- Object filtering, selection and query methods
- Placement methods
- Design rules and preferences and properties
- Display options
- Setting up and running the autorouter
- Generating and understanding reports
- Design verification and error checking
- Use the FIRE® option (Fast Interactive Route Editor)
- Set up Extended Rules and Properties
- Route Differential Pairs and High Speed Nets
- Use the Length Monitor and add-Accordions



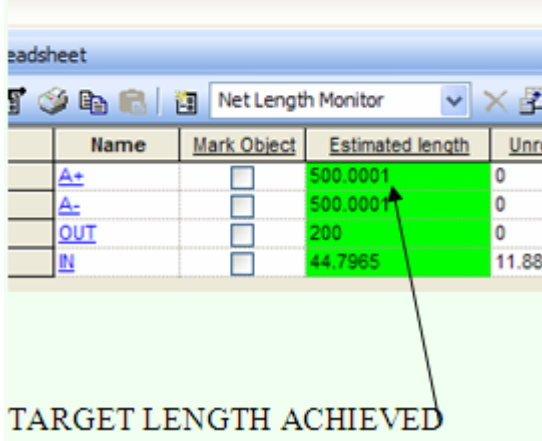
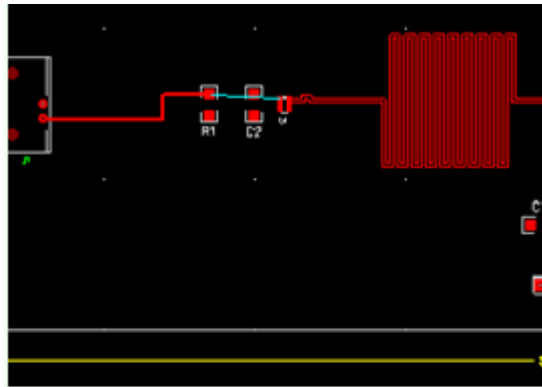
Key topics

- Basic Router Philosophies
- PADS Router Interface
- Using the Filter
- Main, Routing and Selection Toolbars
- Project Explorer Pane

Continued

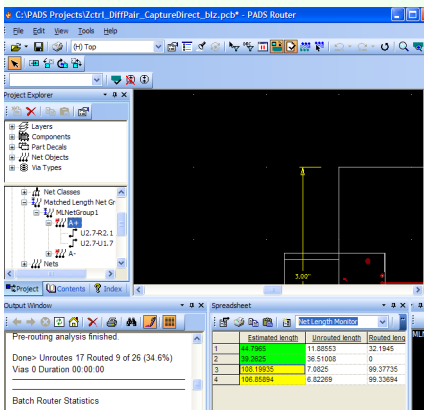
PADS Router continued

- Command and Navigation Windows
- Main Toolbar buttons
- Property Dialogs
- Customizing the Interface
- Tools Menu
- Setting General and Routing Preferences
- Setting up the Routing Strategy
- Setting Testpoint Preferences
- Other Setup Dialogs
- Net Coloration
- Assigning Properties
- By Selection Routing
- Route, Fanout, Optimize by selection
- Reviewing Router Reports
- The Routing Status Bar
- Pre-Routing Analysis
- The Route Report file
- Integrity Test
- Design Statistic, Testability and Testpoint Reports
- Autorouting Preparation
- Preparation, Routing and Fanout Order
- Layout Direction, Corner and Quality Settings
- PADS Layout Interface
- Fanout, Routing, DFT Audit, Layers and Strategy Setup



Name	Mark Object	Estimated length	Unro
A+	<input type="checkbox"/>	500.0001	0
A-	<input type="checkbox"/>	500.000	0
OUT	<input type="checkbox"/>	200	0
IN	<input type="checkbox"/>	44.7965	11.885

TARGET LENGTH ACHIEVED



Project Explorer

- Net Classes
 - Matched Length Net Gr
 - U2.7-R2.1
 - U2.7-U1.7
 - A+
 - A-

Output Window

Pre-routing analysis finished.

Done> Unroutes 17 Routed 9 of 26 (34.6%)

Vias 0 Duration 00:00:00

Batch Router Statistics

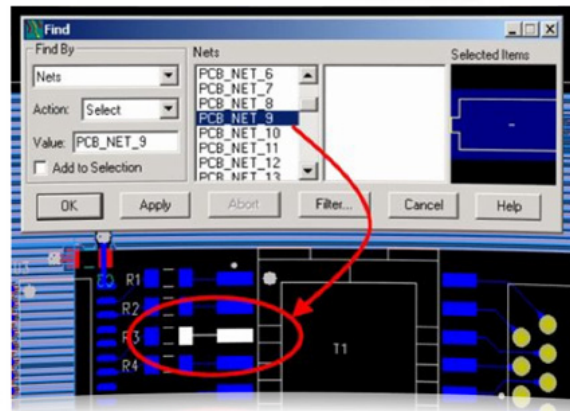
1	Estimated length	Unrouted length	Unrouted ratio
1	44.7965	11.8855	32.1445
2	200.0000	36.51000	0
3	100.0000	7.0625	99.31755
4	100.0000	6.82289	99.32694

CAM350 Training covers:

CAM350 is a multi-use tool for printed circuit board design and fabrication. It can be used to verify CAM materials that have been created by other PCB design tools such as PADS™ Layout or Cadence. It can be used to prepare those same materials for various processes such as step and repeat, panelization or adjusting feature sizes to compensate for fabrication variables. It can even be used to design a PCB board from the ground up or reverse engineer a PCB design from existing photo plots.

All PCB design tools have DRC (design rule check) utilities to verify connectivity and spacing rules. But these utilities only check the rules against the Design Database. CAM350 has the ability to check the design database against the actual materials that will be used to manufacture the PCB. This adds an extra layer of protection from errors that can be introduced either inadvertently by incorrect settings in the utility used to generate the fabrication materials, or the potential inability of the design software to detect certain errors.

Also explained are the output capabilities of CAM350 to create check plots, Gerber and NC files, and a “smart” ODB++** database that is compatible with Valor-Genesis software. The necessary steps will typically be explained with text and graphics, but most operations will also be demonstrated with “live action” using on-screen digital video media. The real-time video clips demonstrate their respective subject areas.

**How It Works**

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



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.

Signal integrity (SI) analysis is an essential part of modern electronic design. Increasingly fast edge rates in today's integrated circuits (ICs) cause detrimental high-speed effects, even in PCB designs running at low operating frequencies. As driver ICs switch faster, a growing volume of boards suffers from signal degradation, including over/undershoot, ringing, glitching, crosstalk, and timing problems. When degradation becomes serious enough, the logic on a board can fail.

Hardware engineers, PCB designers and signal integrity specialists alike can use HyperLynx as a team; getting simulation results without requiring weeks of software training. The emphasis is on getting designs right the first time, avoiding costly overdesign, and saving recurrent layout, prototype and test cycles in the lab.

Hyperlynx Training covers:

The HyperLynx Signal Integrity Analysis course was developed to help you understand basic signal integrity, crosstalk, EMI concepts and gain proficiency in signal integrity, crosstalk, and EMI analyses, pre and post layout.

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

You will learn how to

- Create and simulate LineSim cell-based and free-form schematics
- Investigate termination strategies
- Investigate stack-up strategies
- Translate design databases
- Identify and debug SI and Crosstalk issues
- Read eye diagrams and how to specify a mask
- Simulate EMC and evaluate the results
- Assign models and component values
- Evaluate SI and Crosstalk issues
- Translate PCB layout databases
- Run BoardSim simulations interactively and in batch mode
- Run BoardSim for both single and multi-board projects
- Evaluate batch mode simulation reports to identify potential problems

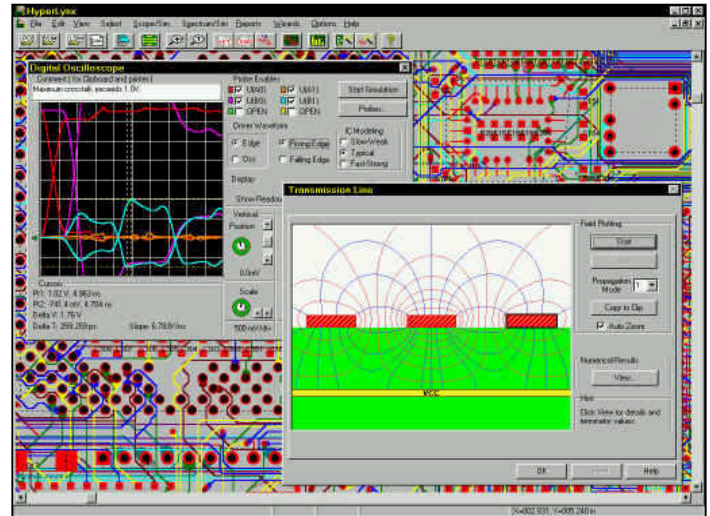
The course kit comes with

1. HyperLynx Software evaluation CD
2. Training Manual
3. MULTIMEDIA DVD with audio- video Classroom tutorial just like Live session
4. Hyperlynx reference design and LAB session files
5. Certificate after project completion
6. Support- one month free support **satisfaction-guaranteed**

Audience: Design and CAD Engineers who will use HyperLynx to analyze signal integrity

Prerequisites

- Familiarity with High-speed PCB concepts
- Slight Familiarity with Hyperlynx user interface is recommended though not mandatory

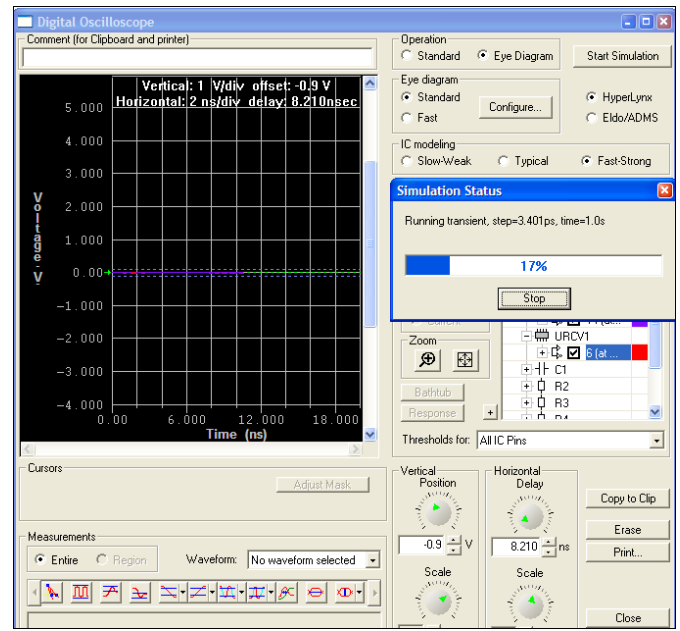


Throughout this course, extensive hands-on lab exercises provide you with practical experience using HyperLynx software. Hands-on lab topics include:

- LineSim free-form and cell-based schematic creation, editing stack-up, assignment of models and component values
- Compare theoretical values with simulation results
- Manual and automatic waveform measurements of overshoot, flight-time and crosstalk.
- Evaluating the effects of different impedance matching strategies, topologies, technologies, and stack-ups in LineSim
- Create, test, and debug IBIS models
- Add model libraries using different methods in BoardSim
- Perform Batch Mode Quick Analysis in BoardSim and review the report
- Perform Batch Mode Detailed Analyses in BoardSim and analyze the reports
- Interactively simulate different scenarios to fix problems in BoardSim
- Document and forward annotate changes made in BoardSim
- Analyze nets using eye diagrams and SPICE simulations
- Set up and simulate nets between boards using multi-board projects

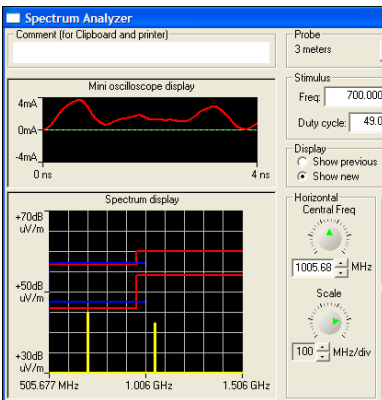
Key Topics

- Pre Layout Activities
- Post Layout Activities
- Fundamental Concepts of SI and Crosstalk
- LineSim's Free-form and Cell-based User Interface
- Using the Stackup Editor and the Schematic Editor
- Assigning Models and Component Values
- Setting Simulation Power Supply Voltages
- Simulation Analysis via the Scope & Spectrum Analyzer
- Using the Terminator Wizard
- Technology, Topology, Termination, and Transmission Line Solutions
- Design Flow
- Multi-board Analysis Pre Layout
- Constraint Generation
- Passing Constraints to Layout
- Creating and Testing MOD Models (Databook)
- Testing IBIS Models
- Modeling Vias in LineSim and BoardSim

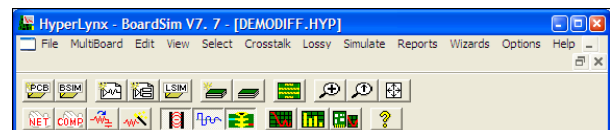


Throughout this course, extensive hands-on lab exercises provide you with practical experience using HyperLynx software. Hands-on lab topics include:

- LineSim free-form and cell-based schematic creation, editing stack-up, assignment of models and component values
- Compare theoretical values with simulation results
- Manual and automatic waveform measurements of overshoot, flight-time and crosstalk.
- Evaluating the effects of different impedance matching strategies, topologies, technologies, and stack-ups in LineSim
- Create, test, and debug IBIS models
- Add model libraries using in BoardSim
- Perform Batch Mode Quick Analysis in BoardSim and review the report
- Perform Batch Mode Detailed Analyses in BoardSim and analyze the reports
- Interactively simulate different scenarios to fix problems in BoardSim
- Document and forward annotate changes made in BoardSim
- Analyze nets using eye diagrams and SPICE simulations
- Set up and simulate nets between boards using multi-board projects



- BoardSim User Interface
- BoardSim Interactive SI, Crosstalk, and EMI simulations
- Translating PCB Layouts to HyperLynx
- Batch Mode (Board Wizard) in BoardSim
- Quick Analysis in BoardSim
- Back Annotation
- Termination Strategies Using Quick Terminators
- Finding Possible Solutions by Comparing Potential Solutions
- Setting up differential pairs
- Analyzing with eye diagrams
- Lossy Analysis
- Simulating with SPICE and S-parameter models
- Multiboard Analysis - Create/edit Multiboard Projects and Caveats
- Multiple Instances (e.g. DIMMS, SIMMS)
- Adding Terminators in Multi-board Projects





Frequently Asked Questions

PCB Design ONLINE/e-learning Training FAQs



Q: Who will be benefited from PCBkey ONLINE TRAINING-KITS?

A: Engineering Students, Planning to switch over to Designing, Project leaders, Embedded designers, Board designers, Layout Engineers, PCB Designers, New Cadence and Mentor Graphics users, Hi-SPEED Designers, Signal Integrity Engineers.

Q: What exactly is e-learning / ONLINE Training?

A: E-learning is the teaching concept for the 21st century, a learning environment on the Internet developed especially for you, which guides you through the material using explanations, animations and questions, and which includes a test at the end. If you have any questions, you can use email or phone to communicate with your own specialist tutor. The result is a modern, pleasant and flexible way of learning.

Q: If I need more support to understand the PCB tutorials?

A: We can voice and video chat through goggle or we can set up Microsoft net meeting. The specialist tutor will browse your desktop and present a demo to clarify your doubts remotely. It is like having your own Live Tutor in front of you... at the hours that you need them most !

Q: Are you struggling to work on your Cadence® OrCAD®, Cadence® Allegro® PCB Editor and PADS layout project?

A: After you buy PCBkey Training KIT, you get the support right away through out the project. The easy payment procedure will help you to complete the payment.

Q: I am not in a position to move out from my work place for 3-4 days training. Do you have training facility at our location?

Is it possible to provide me training on Weekends?

Is it possible to provide training at my location?

A: 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. Start the training instantly NOW

Q: I am doing MSc. in Microelectronics from Bolton University - checked up many PCB Design Training service providers in UK and US. The training fees start from \$1500 - \$3000 which is excluding travel cost, hotel cost.

A: CEDA's PCB worldwide training is an extended service of CEDA PCB Designing services. Cost cutting continues to be the mantra at most organizations. CEDA's training fees is almost 75% less than the prevailing fees, which are in the tune of \$2000. Further, CEDA's PADS/Allegro PCB online training programme will help companies and individual to downsize travel budget

As per our study no other training company can offer this much value for your training investment

Training on demand at your desk without leaving your workspace -

No campus attendance, no commuting, no deadline pressure. All exams are open-book/open notes Learn at your own pace; complete in as little as 25 days! Anyone familiar with basic electronics- 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.

Buy now



[*ADD 50 \\$ for Shipping & Handling charges through FedEx](#)

Need more information? Interested to Buy? Drop a line about the product-name of your interest. We will send you the payment a/c details



009810338939



pcbkey@gmail.com

Make a Payment



Or Wire-Transfer

Payments from INDIA accepted through Cheque /DD in favor of CEDA payable at Delhi. If you are in a hurry, you may deposit the amount in your nearest branch of ICICI bank (mail us we will provide the ICICI a/c details)

International payments accepted through Paypal or through secure credit card option.

*PCBkey TRAINING products brought to you by **CEDA**. For more details visit www.ceda.in*

PCBKEY

PCB Design Training kit & Books

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

5G- DEEPA PLOT #10 IP EXTN. PATPARGANJ, DELHI 110092 INDIA
PHONE: 91 11 22780394 MOBILE 0091 9810338939