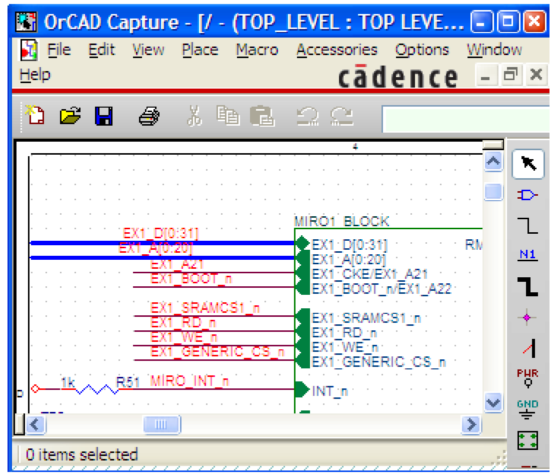
OrCAD® Capture & Allegro PCB Editor

Course Part# PCBkeyOCAPD0509

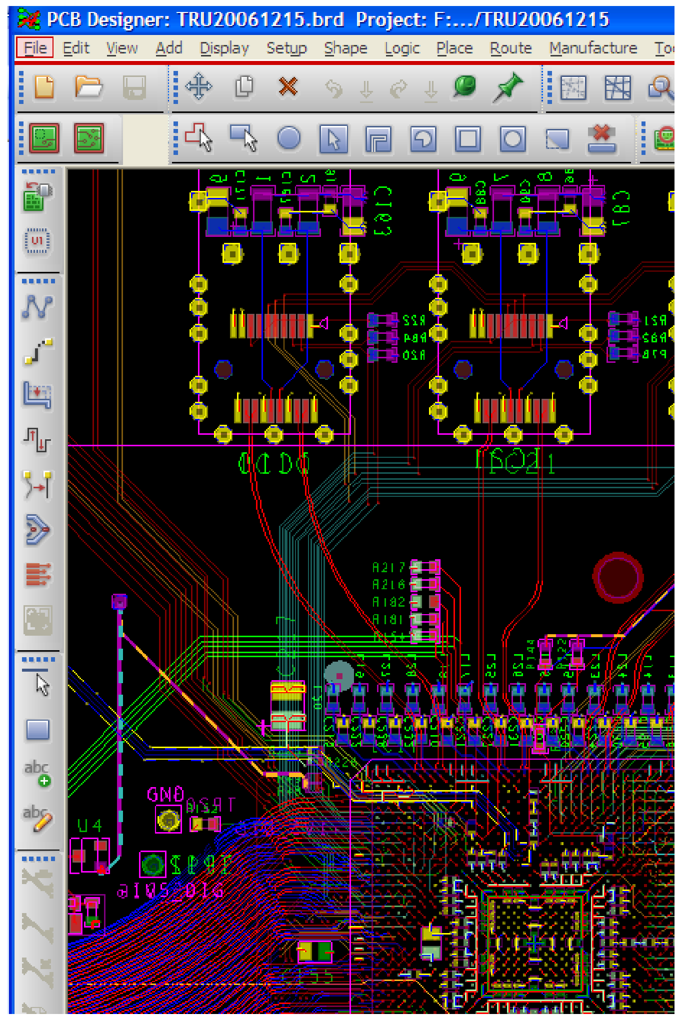
OrCAD® Capture

Duration: 10-12 hours



Course Overview

- 1. Introduction
- 2. Objective
- 3. Starting a New Schematic Project
- 4. Before you create parts library
- 5. Creating a Schematic Parts Library
- 6. Creating Schematic Symbols
- 7. Designing circuits with FPGAs just got easier
- 7.1 Generating an OrCAD Capture symboldifferent methods
- 7.2 To split a part
- 8. Schematic Entry
- 8.1 Setting up the Environment
- 8.2 Placing Parts & Making Connections
- 8.3 Connecting Pages and Naming Nets
- 9. Preparing for Layout
- 9.1 Annotation
- 9.2 Intersheet References
- 10. Before we assign Footprints
- 11. Assigning Footprints to Parts
- 12. Creating the Netlist
- 13. Creating Bill of Material (BOM)



Learning Objectives

Session1

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

Session2

- Creating parts and Schematic symbols
- Building a hierarchical design
- Preparing a design for Allegro PCB Layout
  LAB SESSION

CEDA-Labz

Allegro® PCB Editor

Allegro® PCB Editor

Duration: 40 hours

Course Overview

The Allegro PCB Editor consists of nine modules, which include all the fundamental steps for designing a PCB, from loading logic/netlist data through producing. Manufacturing/NC output. The last four modules of this course introduce you to ways to build on the basics of the tool and address board design for high-speed purposes, while touching on ways to increase productivity. The task-oriented labs show you the combined use of interactive and automatic tools.

Modules

- Allegro PCB Editor: Module 1: Getting Started
- Allegro PCB Editor: Module 2: Library Development
- Allegro PCB Editor: Module 3: Logic Import, Design Rules and Component Placement
- Allegro PCB Editor: Module 4: Routing, Glossing and Copper Areas
- Allegro PCB Editor: Module 5: Post Processing and Manufacturing Output
- Allegro PCB Editor: Module 6: Being Productive using the PCB Editor
- Allegro PCB Editor: Module 7: Rules, Constraints, and High-Speed Routing
- Allegro PCB Editor: Module 8: Autoplace, Interactive Routing, and PCB Router Tips
- Allegro PCB Editor: Module 9: Glossing, Copper Planes, and Tools to Produce Designs

Learning Objectives

Session1

- Learn about the user interface.
- Learn about design preparation (libraries).
- Padstacks
- Component Symbols

  LAB SESSION

Session2

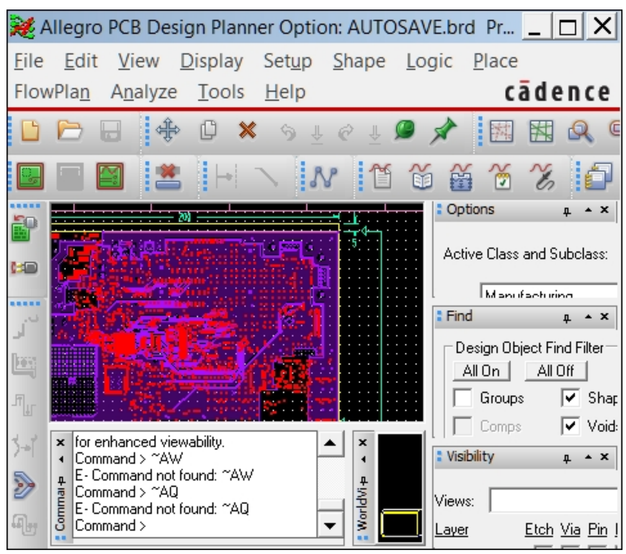
- Import logic design data
- Learn about design rule definition
- Perform interactive placement
- Perform pin and gate swapping

  LAB SESSION

Session3

- Perform interactive routing
- Perform automatic routing
- Create copper areas for positive and negative planes.
- Learn about manufacturing output and documentation.

  LAB SESSION



Continued …

CEDA-Labz

Allegro PCB Editor continued

Session4

- Customize PCB Editor to increase productivity.
- Learn about the different files.
- Use the Constraint Manager.
- Learn about advanced constraints.
- Autoroute high-speed designs.
- Learn about differential pairs.
  LAB SESSION

Session5

- Perform interactive bus routing.
- Perform advanced interactive and automatic routing of critical nets.
- Learn complex design strategies.
- Design to interface to PCB Router.
- Perform advanced glossing.
- Generate test points.
- Learn about split and complex power planes.
  LAB SESSION

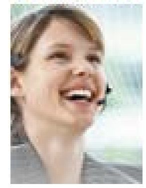
The course kit comes with

1. Training Manual
2. MULTIMEDIA *USB PEN DRIVE* with audio- video Classroom tutorial just likes live session
3. Reference design and LAB session files
4. Support – 2 months free support

CEDA-Labz

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.