

OrCAD® Capture

Duration: 10 hours

Course Overview

- Building parts and symbols
- Creating a new project
- Creating multi-sheet flat designs
- Copying work between projects
- Assigning reference designators
- Design checking
- Adding intersheet signal references
- Creating a Bill of Materials and other reports
- Adding part and net properties
- Creating a netlist for OrCAD Layout
- Building a hierarchical design

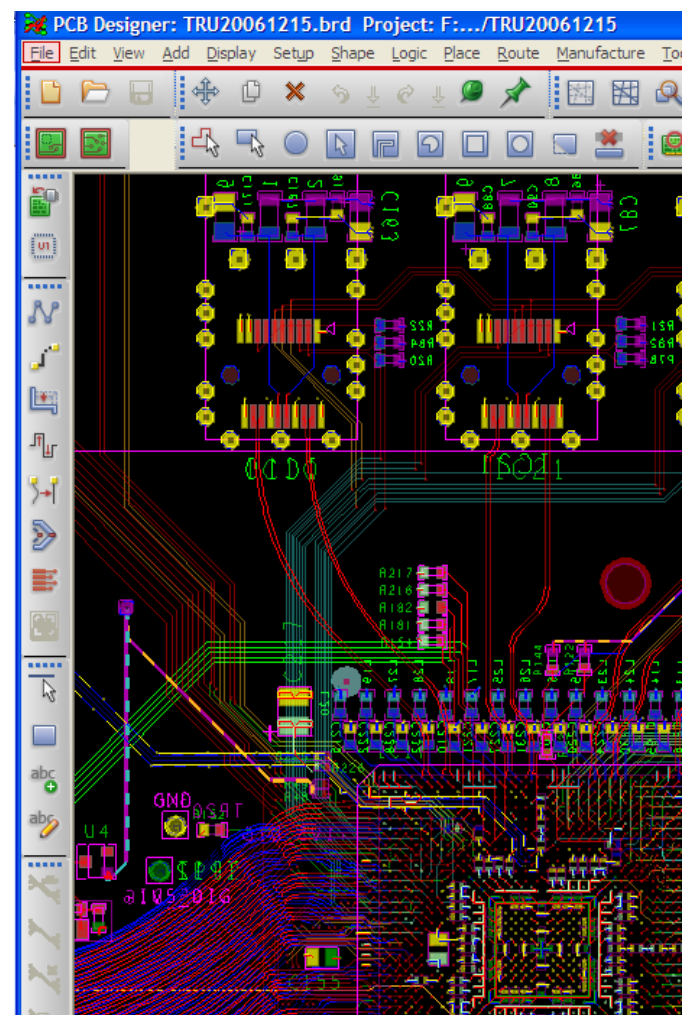
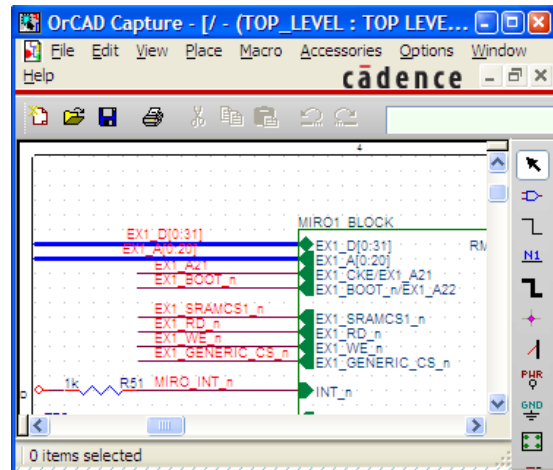
Learning Objectives

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 symbols
 - Building a hierarchical design
 - Preparing a design for Allegro PCB Layout
- LAB SESSION



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 ...

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. Allegro Design Software evaluation CD
 2. Training Manual
 3. MULTIMEDIA **USB PEN DRIVE** with audio- video Classroom tutorial just likes live session
 4. Allegro reference design and LAB session files
 5. Certificate after project completion
 6. Support – 2 months free support ***...satisfaction guaranteed***
-

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.