

Orcad/Allegro PCB-Editor

Duration: 2 days
Time: 9:00 – 17:00

Description: PCB Editor is a central part of the OrCAD PCB Designer and Allegro PCB design solutions. This two day course will cover all the necessary steps for designing a printed circuit board, from loading logic/netlist data through producing manufacturing/NC output

Audience: This course is for printed circuit designers and design engineers who have an understanding of PCB layout and manufacturing considerations.

Prerequisites: You should have a working knowledge of PCB design and a basic understanding of PCB manufacturing processes. You must also be proficient with using Windows and standard Windows applications.

Course Agenda:

- PCB Editor User Interface
 - Identify the user interface components
 - Navigate within the PCB Editor window and access UI features to tailor the tool for individual needs
- Managing the PCB Editor Work Environment
 - Create and use scripts
 - Use the Control Panel to locate board database objects and report information about them
- Padstacks
 - Create a flash symbol user for thermal relief
 - Use the Pad Designer to create padstacks for a number of typical pins, such as through-hole and surface-mount pads
- Component Symbols
 - Use the Package Symbol Wizard to create a Package Symbol
 - Use the Symbol Editor to create a surface-mount package
- Board Design Files
 - Use the Mechanical Symbol Editor to create Board Mechanical Symbols
 - Use the PCB Design Editor to create a Master Board Design
- Importing Logic Information into PCB Editor
 - The Logic information can come from Concept, Capture, Third party
- Setting Design Constraints
 - Assign Standard & Extended Design Rules for Spacing & Physical Dimensions
 - Add, Change & Delete Properties of Components & Nets

- Component Placements
 - Use Floorplanning to organize the placement of components with the same ROOM property
 - Assign Reference Designators for preplaced parts
 - Interactively place components
- Advanced Placement
 - Turn rsnets on and off to selectively place components
 - Use interactive and auto swapping for pins and gates
 - Perform cross placement between PCB Editor and DE HDL or DE CI
- Routing and Glossing
 - Define and Display etch grids used for Routing
 - Add and Delete connect lines and vias
 - Prepare for Autorouting by creating preliminary embedded planes
 - Route net connections with PCB Router
 - Use Slide and Replace Etch to improve routing
 - Use the Cut option in conjunction with other etch editing commands
 - Use Gloss to automatically clean up the routed etch in the Design
- Copper Areas and Positive or Negative Plans
 - Learn how to generate Positive and Negative Plans
- Preparing for Post Processing
 - Rename Reference Designators and backannotate changes made in the PCB Editor to the Schematic Environment
- Preparing the Board Design for Manufacturing
 - Generate and Edit Silkscreen layers
 - Use Reports available in PCB Editor
 - Setup the Design File for Artwork - Preview before plotting
 - Generate Drill Symbols and legend for fabrication drawing
 - Create check plots
 - Output a Drill file used for drilling the board holes in manufacturing