

OrCAD/Allegro PCB-Editor Introduction

Duration: 1 day
Time: 9:00 – 17:00

Description: This one day course will cover all the necessary steps for designing a printed circuit board, from loading logic/netlist data through producing manufacturing/NC output. Learn about the user interface for the Orcad/Allegro PCB Editor

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:

- User Interface and Configuration
- Rule Manager from Capture to PCB Editor
- Set up rules
- PCB templates
- Location of components
- Routing
- Copper Fields
- Postprocessing including reports, silkscreen, Gerber / ODB ++, drill, etc..