

TRAINING

Bei dem hier beschriebenen Training handelt es sich um ein Cadence Standard Training. Sie erhalten eine Dokumentation in englischer Sprache. Die Trainingssprache ist deutsch, falls nicht anders angekündigt.

Unter <http://www.FlowCAD.de/TrainingKontakt.php> können Sie sich zum Training anmelden.

Course Title	Allegro PCB Editor Basic Techniques
Course Category	System Interconnect Design – Allegro & OrCAD
Duration	3 Days

"Very good training. I already knew a lot about this tool but could learn a lot of tricks anyway!"
(Marc Schiricke, Ferchau Engineering, April 2016)

"A well-organized compact course. The instructor had a lot of experience."
(Satisfied Cadence customer from GrassValley, January 2016)

"Very interesting and complex course made simple by good teaching. I liked the labs, they are very useful to speed up learning of the PCB designer tool. (Davide Esposito, Magneti Marelli, August 2015)

"Very good course material, excellent instructor, good environment. Course material is very structured, right amount of labs to practice the commands." (Thomas Zieger, Magneti Marelli, August 2015)

Course Description

In this three-day Allegro® PCB Editor Basic Techniques course, you learn all the necessary steps for designing a PCB, from loading logic and netlist data through producing manufacturing/NC output. The task-oriented labs show you the combined use of interactive and automatic tools. You import logic design data, learn design rules, run interactive placement, route nets interactively, and run the autorouter. You also create copper areas for positive and negative planes, and prepare the board for manufacturing. This is the first in a two-course series. You should attend the second course, Allegro PCB Editor Intermediate Techniques, after attending this course.

Learning Objectives

After completing this course, you will be able to:

- Efficiently navigate the user interface
- Create library parts which are used throughout the entire process of board layout
- Import logic design data
- Identify the different design rules and set them in your design
- Place your design interactively
- Route nets interactively
- Run the automatic router
- Create copper areas for positive and negative planes
- Produce manufacturing output and documentation

Software Used in This Course

- This course uses the base license of Allegro PCB Designer.

Course Agenda

Day 1

- User interface
- PCB Editor initialization
- PCB Editor library features
- Padstacks
- Component symbols

Day 2

- Board template
- Import logic
- Design rules
- Properties and constraints
- Interactive placement
- Advanced placement

Day 3

- Interactive routing
- Automatic routing
- Copper areas (planes)
- Manufacturing output and documentation

Audience

- CAD Engineers
- Designers
- Electrical Engineers
- Layout Designers
- PCB Designers
- PCB Layout Designers

Prerequisites

You must have experience with or knowledge of the following:

- Layout Design

Related Courses

[AllegroPCBEditorIntermediateTechniques.pdf](#)