

## 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.

<b>Course Title</b>	<b>Allegro PCB Editor Basic Techniques</b>
<b>Course Category</b>	<b>System Interconnect Design – Allegro &amp; OrCAD</b>
<b>Duration</b>	<b>3 Days</b>

"A very good experience, a lot of useful knowledge to take home"  
(Sebastian Eckert, Britze Elektronik, July 2014)

"This course was really good and practical, and gives super clues on some additional functions in the software which allows easier work with the tool." (Grzegorz Duszejko, Omicron Electronics, July 2014)

"I already knew the basics but I learned a lot of stuff to speed my work up and make it easier." (Johannes Woehr, Schunk GmbH & Co. KG, January 2014)

"The environment and the trainer were excellent." (David Busto, NXP)

## 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](#)