

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.

Course Title	Allegro PCB Editor Basic Techniques v16.5
Course Category	System Interconnect Design – Allegro & OrCAD
Duration	3 Days
Course ID	ES_86097_16.5
Product Version	16.5

Course Description

The three-day *Allegro® PCB Editor Basic Techniques* course covers 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.

This is the **first** in a two-course series. In the second course, *Allegro PCB Editor Intermediate Techniques*, fifty percent of the material is applicable to all users, and the other fifty percent is applicable to users with the Allegro PCB Performance Option L or Allegro PCB Design XL license.

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 and differential-pair 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 has no requirement for a specific software license.

Software Release(s)

- SPB16.5

Course Agenda

Note that this course can be tailored to better meet your needs – [contact the Cadence training staff](#) for specifics.

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
- Differential pairs
- 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

- [Allegro PCB Editor Intermediate Techniques v16.5](#)