

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 V16.5
Course Category	System Interconnect Design – Allegro & OrCAD
Duration	5 Days
Course ID	ES_86098_16.5
Product Version	16.5

Course Description

This course combines our three-day *Allegro® PCB Editor Basic Techniques*, followed by two days of *Allegro® PCB Editor Intermediate Techniques*.

The first three days cover 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.

The last two days give you a deeper understanding of the software, including features and tips. You use the constraint manager, autoroute high-speed designs, and apply advanced features of differential pairs. This part also includes exploring high-speed design rules, creating areas in your design that require different routing rules, and generating testpoints. In the task-oriented labs, you use a combination of interactive and automatic tools. Half of these modules require either the Allegro PCB Performance Option L license or the 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
- Use the online help system successfully
- Use the constraint manager efficiently
- Autoroute high-speed designs
- Apply the advanced features of differential pairs
- Perform interactive bus routing
- Perform advanced interactive and automatic routing of critical nets
- Use the advanced glossing functionality
- Generate testpoints
- Place parts based upon design-for-assembly rules
- Create designs with split and complex power planes

Software Used in This Course

- Allegro PCB Design HDL – L or Allegro PCB Design CIS – L; or
- Allegro PCB Performance Option – L; or
- Allegro PCB Design HDL – XL or Allegro PCB Design CIS - XL

Software Release(s)

- SPB16.5

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

Day 4

- Reuse of design constraints
- Split and complex planes
- Unused pad suppression
- Testpoint generation
- Interactive routing tips and techniques
- High-speed etch editing

Day 5

- High-speed constraints using the Constraint Manager
- Advanced constraints
- Differential pairs
- Design for assembly
- Automatic glossing
- Automatic placement

Audience

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

Prerequisites

You must have experience with or knowledge of the following:

- Board layout and design