

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 Design Entry HDL Front-to-Back Flow v16.5
Course Category	System Interconnect Design – Allegro & OrCAD
Duration	3 Days
Course ID	ES_86015_16.5
Product Version	16.5

Course Description

In this course, you create board-level schematic designs with Design Entry HDL. You explore the integration between Design Entry HDL and other tools in the design flow, including the Allegro® PCB Editor. You follow the actual design flow by creating a schematic and taking it all the way through board layout.

Although board layout is introduced as part of the front-to-back flow, this is not a board layout course. Note also that no schematic or footprint library development is included in this course. See the Related Courses below.

Learning Objectives

After completing this course, you will be able to:

- Set up new projects
- Create a flat, multisheet design
- Check the design
- Use part tables
- Package a design
- Create and customize a bill of materials
- Build a hierarchical design
- Use schematic properties to control part placement
- Use the Constraint Manager to define high-speed routing requirements in Design Entry HDL
- Transfer the design to the PCB Editor
- Place parts manually
- Autoroute with the PCB Router
- Compare the schematic and layout
- Synchronize design differences
- Incorporate engineering changes
- Link projects together in a team-design scenario
- Reuse a hierarchical block and associated layout in another design

Software Used in This Course

- Allegro PCB Design HDL - XL

Software Release(s)

- SPB 16.5

Course Agenda

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

Day 1

- Getting Started
- Project Setup
- Design Entry and Packaging

Day 2

- Hierarchical and Team Design
- Design Rules

Day 3

- Rules-Driven Layout
- Engineering Changes

Audience

- Design Engineers
- PCB Layout Designers

Prerequisites

- This course has no prerequisites.

Related Courses

- [Allegro High-Speed Constraint Management v16.5](#)
- [Allegro PCB Editor Basic Techniques v16.5](#)
- [Allegro PCB Librarian v16.5](#)