

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 Design Entry HDL Front-to-Back Flow
Course Category	System Interconnect Design – Allegro & OrCAD
Duration	3 Days

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

*"Very useful course for those using Cadence for the first time in a company environment."
(Davide Esposito, Magneti Marelli, August 2015)*

"I came here with no knowledge about this software, and now I have the basic knowledge to start using it professionally. Great." (David Alonso, Magneti Marelli, August 2015)

*„Very good presentation, well elaborated lab classes, good small classes: I would enroll again“
(Michael Kurzmann, Intel, June 2015)*

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 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. Also, schematic or footprint library development is not 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, multi-sheet 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
- Auto-route 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

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

Course Agenda

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

[AllegroPCBEditorBasicTechn.pdf](#)

[AllegroPCBLibrarian.pdf](#)