

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.

Allegro Design Entry HDL Front-to-Back

Description

The main goal of this course is to teach how to use Concept® HDL to create board-level schematic designs. A second goal is to demonstrate the integration between the ConceptHDL and other tools in the design flow, including Allegro®. You will follow the actual design flow by creating a schematic and taking it all the way through board layout.

The highlights of this class include the following:

- Overview of the mainstream board design process
- Project setup
- Creating flat, multi-sheet designs
- Design checking
- Using part tables during design entry
- Packaging
- Creating and customizing bill of materials
- Building a hierarchical design
- Using schematic properties to control part placement and routing in Allegro
- Using the Constraint Manager to define high-speed routing requirements in Concept
- How to transfer the design to Allegro
- How to place parts manually, and perform a basic autoroute using SPECCTRA
- How to use the Design Differences tool to compare the schematic and layout
- Synchronizing design differences
- Engineering changes

Audience

- Electrical Engineers
- PCB Layout Designers
- Design Engineers
- PCB Designers
- Managers
- Logic designers using the ConceptHDL product as front-end tool for board design
- SCALD-based Concept users interested in learning about the new HDL architecture
- Physical layout designers wanting to know more about mainstream board design processes using the ConceptHDL product

Software

Software requirements include:

- You need either the PCB design Expert with ConceptHDL or the PCB Designer with ConceptHDL Series product(s)

OR you need the following legacy products:

- Concept HDL Expert
- SPECCTRA Expert AutoRouter
- Allegro Expert or Allegro Designer

Prerequisites

There are no prerequisites for this course. It is recommended that you take this course before taking the PCB Librarian course

- Some familiarity with Windows or Unix is helpful

Related Courses

- Allegro PCB Layout System
- SpecctraQuest Foundations
- PCB Librarian

Course Agenda

- Getting Started
- Design Entry
- Part Tables
- Packaging
- Intro to Board Layout
- Hierarchical Design
- Design Rules
- Transfer to Allegro
- Engineering Changes