

Bei dem hier beschriebenen Training handelt es sich um ein Cadence Standard Training. Sie erhalten eine Dokumentation in englischer Sprache. Die Trainingsprache ist deutsch, falls nicht anders angekündigt.

## Allegro PCB Editor

### Overview

#### Description

The first part of this course cover all the necessary steps for designing a printed circuit board, from loading logic/netlist data through producing manufacturing/NC output. The last part of this course introduce you to ways to build on the basics of the tool and address board design for high-speed purposes, while touching on ways to increase productivity. The task-oriented labs show you the combined use of interactive and automatic tools.

**The highlights of this class include the following:**

- Introduction to the user interface for the PCB Design Studio and Allegro Design Expert products
- Design preparation (libraries)
- Importing logic design data
- Design rule definition
- Constraint Manager
- Technology files
- Interactive and automatic placement
- Pin and gate swapping
- Interactive and automatic routing
- Complex design strategies
- Automatic glossing
- Creating copper areas
- Manufacturing output and documentation
- Customizing Allegro
- Differential Pairs
- Test Prep
- Split Planes
- Advanced constraints

#### Audience

- This course is for printed circuit designers and design engineers who have an understanding of PCB Layout and manufacturing considerations

## Software

You need the PCB Design Expert series product(s)--either:

- PCB Design Expert with Concept HDL or
- PCB Design Expert with Capture CIS

OR you need the following legacy products:

- Allegro Expert
- SPECCTRA Expert AutoRouter v15.0
- A schematic capture tool (Concept HDL, Capture, or Capture CIS)

## Prerequisites

- You should have a working knowledge of PCB design and a basic understanding of PCB manufacturing processes.
  - Some familiarity with Windows or UNIX is helpful.

## Related Courses

- [Concept® HDL Front-to-Back Design Flow](#)
- [SPECCTRA AutoRoute Basics](#)
- [Advanced SPECCTRA® AutoRouting Techniques](#)
- [SPECCTRAQuest® Foundations](#)
- [Cadence® Advanced Package Designer](#)

## Course Agenda

- User Interface
- Allegro Librarian Features
- Creating Padstacks
- Building Component Symbols
- Creating a Board Template
- Importing Logic
- Defining Design Rules
- Using Properties and Constraints
- Interactive Placement
- Advanced Placement
- Interactive Routing

- Automatic Routing
- Creating Copper Areas (Planes)
- Manufacturing Output and Documentation
- Customizing Allegro
- Allegro Productivity
- Constraint Manager
- Advanced Constraints
- Differential Pairs
- Automatic Placement
- Testpoint Generation
- Routing Tips and Techniques
- Automatic Glossing
- Split and Complex Planes