

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 Librarian v16.5
Course Category	System Interconnect Design – Allegro & OrCAD
Duration	2 Days
Course ID	ES_86022_16.5
Product Version	16.5

Course Description

In this course, you learn to create schematic libraries for Allegro® Design Entry HDL and footprint libraries for use with Allegro PCB Editor. About 75% of the course time is focused on front-end schematic library development, and the remaining 25% is spent on back-end footprint creation. You create a project area for building schematic symbols, pin maps, part tables, and package symbols. You also test these part definitions in a front-to-back flow. Allegro PCB Editor footprint creation is also covered in the Allegro PCB Editor course. Please see the list of Related Courses below.

Learning Objectives

After completing this course, you will be able to:

- Set up for library development
- Create symbol, package, part table, and simulation views
- Build padstacks and package symbols
- Test parts
- Create asymmetrical and split parts, and build new parts from existing ones

Software Used in This Course

- Allegro PCB Librarian - XL

Software Release(s)

- SPB16.5

Course Agenda

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

Day 1

- Explore an existing library using the library development tools
- Create a directory structure for building and testing Design Entry HDL and PCB Editor parts
- Create a symbol, chips, part table and simulation view
- Test parts by simulating its use in a design flow
- Release the new parts for general use
- Create a CMOS octal transparent latch
- Use Design Entry HDL to modify symbol graphics

- Create a CMOS quad buffer
- Explore Part Developer setup options
- Create a resistor pack

Day 2

- Create a CMOS octal transceiver
- Make a new Design Entry HDL part from an existing part
- Create an asymmetrical symbol
- Create a symbol from an Excel file
- Create PCB Editor padstacks and flash symbols
- Create PCB Editor package symbols manually
- Create PCB Editor package symbols using the Symbol Wizard
- Create a PCB Editor package symbol that uses a pad shape symbol
- Create a PCB Editor board symbol

Audience

- Library Developers

Prerequisites

- There are no prerequisites for this course.

Related Courses

- [Allegro Design Entry HDL Front-to-Back Flow v16.5](#)
- [Allegro PCB Editor Basic Techniques v16.5](#)