

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 PCB Librarian |
| Course Category | System Interconnect Design – Allegro & OrCAD |
| Duration | 2 Days |

Course Description

In this course, you learn to create schematic libraries for Allegro® Design Entry HDL and footprint libraries for use with the 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. 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

Course Agenda

Day 1

- Explore a 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 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