FlowCAD

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**
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

Software Release(s)

- SPB 16.5 (or higher)

Course Agenda

Note that this course can be tailored to better meet your needs – contact the Cadence training staff *for specifics.*

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

Prerequisites

There are no prerequisites for this course.

Related Courses

Allegro Design Entry HDL Front-to-Back Flow