

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 Design Entry Using OrCAD Capture v17.2-2016
Course Category	System Interconnect Design – Allegro & OrCAD
Duration	2 Days

“The course was optimal for beginning with OrCAD Capture. Documentation and labs are useful for further reference.” (Georg Guldán, emz-Hanauer, February 2016)

“good and competent” (Wolfgang Quehl, Loewe Technologies, July 2015)

“Very interesting and useful hints and informations.” (Andreas Bohnert, Loewe Technologies, July 2015)

Course Description

In this course, you begin with some basic schematic library development. You create and process a simple design, and then you progress into multi-sheet and hierarchical designs. You also apply part and net properties.

Learning Objectives

After completing this course, you will be able to:

- Build Capture library parts
- Create a new project
- Create multi-sheet flat and hierarchical designs
- Copy work between projects
- Assign reference designators
- Perform design checking
- Add inter-sheet signal references
- Create a Bill of Materials report
- Add part and net properties
- Create a net-list for Cadence Allegro® PCB Editor

Software Used in This Course

- PS2005 Allegro Design Entry Capture

Software Release

- SPB 17.2 or higher

Course Agenda

Day 1

- Introduction to OrCAD Capture
- Setting Up Your Environment
- Working With Libraries
- Building a Simple Schematic
- Building a Multi-sheet Schematic

Day 2

- Editing Part and Net Properties
- Building a Hierarchical Design

Audience

This is an introductory level course intended for new Cadence OrCAD Capture users.

Prerequisites

You must have experience with or knowledge of the following:

- Familiar with a PC and Windows operating system
- Experience with schematic entry and board layout components is also helpful

Related Courses

[OrCAD CISv17.2.pdf](#)

[Allegro PCB Editor Basic Techn17.2-2016.pdf](#)