# 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** |
| **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 Guldan, 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)

*"It was a very good training. We were able to discuss our problems and issues with the trainer and she was also very cooperative to explain our queries and questions." (Shamsuddin Ahmed, Freescale Semiconductors, July 2014)*

*"The training was very good and informative. The lecturer was able to explain all objects in a simple way and answered all our questions." (Soran Majied, Freescale Semiconductors, July 2014)*

## 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



www.FlowCAD.com

FlowCAD Confidential

## 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

OrCADCIS.pdf
AllegroPCBEditorBasicTechn.pdf



www.FlowCAD.com

FlowCAD Confidential