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	**OrCAD Capture CIS-v16.5**
Course Category	**System Interconnect Design – Allegro & OrCAD**
Duration	**2+1 Days**
Course ID	**ES_5000411_16_3**
Product Version	**16.5**

This three-day course consists of two parts which can also be booked separately.

The first two days on OrCAD® Capture cover the full range of front-end design processes, from setting up design templates to creating a netlist for board layout, as well as part management. You are encouraged to bring specific real-world design questions with you to class.

The third day covers all the necessary steps for administrating and working with CIS (Component Information System).

Learning Objectives

The highlights of this class include the following:

OrCAD Capture

- Learn how to use design templates, create a new design, build parts, and draw schematics.
- Use advanced tools and procedures and transfer designs from Orcad Capture to other Orcad products.

OrCAD Capture CIS

- CIS and it's benefit
- CIS – Administration
- Find and Place Parts from CIS
- Update and manage the components in your project
- CIS Bill of Material
- Variant Design
- ICA

Audience

- Engineers, designers and technicians engaged in schematic design who are seeking maximum productivity in a minimum amount of time.
- The third day is for database administrators, printed circuit designers and design engineers who have an understanding of Schematic Entry and PCB layout.

Software

You either need one the following series product(s):

- Design Entry CIS
- OrCAD Capture CIS

OR you need the following legacy product:

- Allegro PCB Designer
- OrCAD PCB Designer CIS

Prerequisites

- Proficiency with Windows and standard Windows applications.
- If you would like to only attend the third training day, you should have a working knowledge of Capture.

Course Agenda

Day 1 – OrCAD Capture Part 1

- Getting Started with Capture
- Setting up Your Environment
- Creating Part Libraries
- Creating Homogeneous Parts
- Creating Heterogeneous Parts
- Creating Parts from a Spreadsheet
- Creating Symbols
- Building a Simple Schematic
- Processing a Schematic Design

Day 2 – OrCAD Capture Part 2

- Building a Multi-Sheet Schematic
- Processing a Multi-Sheet Schematic
- Other Tools in OrCAD Capture
- Editing Properties
- Building a Hierarchical Design
- Creating a Hierarchical Block
- Processing a Hierarchical Design
- Processing the Schematic for OrCAD or Allegro® PCB Editor

Day 3 – OrCAD Capture CIS

- Learn to use a CIS database, add parts to the schematic, and modify part properties
- Check the status of database parts, place parts in a schematic, and link them to the database
- Creating design variants for different product assemblies or functions
- Creating a CIS BOM and a BOM for variants
- How to access ActiveParts; locate, download, and place parts on schematic page

Related Courses

- Allegro PCB Editor V16.3
- Analog Simulation with PSpice-v16.3