

Training

Course Title	OrCAD Capture Constraint Manager PCB Flow
Course Category	Front End Schematic
Duration	1 Day

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.

[Anmeldung zum Training](#)

Course Description

This course shows you how to use Constraint Manager to add routing rules to an OrCAD Capture schematic. You transfer the design to Allegro where you quickly place and route the board, and recheck the design rules with Constraint Manager. This course also shows you how to backannotate OrCAD Capture schematic, and implement an engineering change in schematic and board layout.

Audience

- Design engineers
- PCB and CAD support

Learning Objectives

After completing this course, you will be able to:

- Enable use of Constraint Manager on an OrCAD Capture schematic
- Work with electrical constraints
- Attach properties
- Start a new board layout, place parts and route signals
- Rename reference designators on the board, backannotate schematic, and archive project
- Copy and rename project for engineering changes
- Modify schematic and board layout
- Analyze routing, modify constraints, and synchronize schematic and layout

Modules in this Course

- Design Rules
- Rules Driven Layout
- Engineering Changes

Software Used in This Course

OrCAD PCB Designer Professional, Release SPB174

Prerequisites

You must have experience with or knowledge of the following:

- OrCAD Capture
- Allegro PCB Editor