

## 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.

<b>Course Title</b>	<b>Allegro PCB Editor</b>
<b>Course Category</b>	<b>System Interconnect Design – Allegro &amp; OrCAD</b>
<b>Duration</b>	<b>5 Days</b>

*"The course was well prepared and I think that giving us time to do the labs after explanations is a really good way to directly practice our learning.  
(...) the contents were clear and easy to understand. The teacher was nice and took time for us." (Ruben Guerreiro, Meggitt Sensing Systems, July 2016)*

*"The training was great and satisfying my needs. I especially like the lab and the support of the instructor in doing this lab." (Eslam Sawaby, Valeo, July 2016)*

### Course Description

This course combines our three-day *Allegro® PCB Editor Basic Techniques*, followed by two days of *Allegro® PCB Editor Intermediate Techniques*.

In the first three days, you learn all the necessary steps for designing a PCB, from loading logic and netlist data through producing manufacturing/NC output. The task-oriented labs show you the combined use of interactive and automatic tools. You import logic design data, learn about design rules, run interactive placement, route nets interactively, and run the autorouter. You also create copper areas for positive and negative planes, and prepare the board for manufacturing.

The last 2 days give you a deeper understanding of the software, and include features and tips. You use the constraint manager, autoroute high-speed designs, and work with differential pairs. This course also includes exploring high-speed design rules, creating areas in your design that require different routing rules, using the glossing routines, and generating testpoints. In the task-oriented labs, you use a combination of interactive and automatic tools

### Learning Objectives

After completing this course, you will be able to:

- Efficiently navigate the User Interface
- Create library parts which are used throughout the entire process of board layout
- Import logic design data
- Identify the different design rules and set them in your design
- Place your design interactively
- Route nets interactively
- Run the automatic router
- Create copper areas for positive and negative planes
- Produce manufacturing output and documentation
- Use the online help system successfully
- Use the constraint manager efficiently
- Autoroute high-speed designs
- Create rules and route differential pairs
- Perform interactive bus routing
- Perform advanced interactive and automatic routing of critical nets
- Generate testpoints

- Place parts based upon design-for-assembly rules
- Write extract programs
- Use existing SKILL programs in the PCB Editor
- Create designs with split and complex power planes

## Software Used in This Course

- Allegro PCB Designer.

## Course Agenda

### Day 1

- User interface
- PCB Editor initialization
- PCB Editor library features
- Padstacks
- Component symbols

### Day 2

- Board template
- Import logic
- Design rules
- Properties and constraints
- Interactive placement
- Advanced placement

### Day 3

- Interactive routing
- Automatic routing
- Copper areas (planes)
- Manufacturing output and documentation

### Day 4

- Reuse of design constraints
- Split and complex planes
- Unused pad suppression
- Testpoint generation
- Interactive routing tips and techniques
- High-speed etch editing
- High-speed constraints using the Constraint Manager

### Day 5

- Advanced constraints
- Differential pairs
- Design for assembly
- Tips and Tricks
- Writing extract programs
- Using SKILL in the PCB Editor
- Glossing the design

## Audience

- CAD Engineers
- Designers
- Electrical Engineers
- Layout Designers
- PCB Designers
- PCB Layout Designers

## Prerequisites

You must have experience with or knowledge of the following:

- Layout Design