

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 PCB Editor Intermediate Techniques v16.6
Course Category	System Interconnect Design – Allegro & OrCAD
Duration	2 Days

*"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

The Allegro® PCB Editor Intermediate Techniques course gives you a deeper understanding of the software, including 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.

This is the **second** in a two-course series. You need to complete the Allegro PCB Editor Basic Techniques course before taking this one.

This course requires the SPB16.6 QIR6 (HotFix 27) software or later

Learning Objectives

After completing this course, you will be able to:

- 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
- Use the automatic glossing routines

Software Used in This Course

- Allegro PCB Designer

Course Agenda

Day 1

- 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 2

- Advanced constraints
- Differential pairs
- Design for assembly
- Tips and tricks
- Writing extract programs
- Using SKILL programs 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:

- Board Layout and Design

You must have completed the following course:

- Allegro PCB Editor Basic Techniques

Related Courses

[AllegroPCBEditorBasicTechn.pdf](#)