

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 High-Speed Constraint Management
Course Category	Advance with Engineer Explorer Series; System Interconnect Design – Allegro & OrCAD
Duration	2 Days

*"Enough time to solve all questions and problems. Well done teaching speed, all in all positive. Great introduction examples prior to book explanations."
(Sinisa Rubil, Automotive Lighting, January 2016)*

*"A well-organized compact course. The instructor had a lot of experience."
(Satisfied Cadence customer from GrassValley, January 2016)*

Course Description

This *Engineer Explorer* course is designed around advanced topics and exploration of the software. This course does not cover basic operations. If you are not actively using the software, then you need to complete the Allegro® PCB Editor, the Allegro Package Designer, or the Allegro Design Entry HDL Front-to-Back Flow course.

In this course, you apply and verify high-speed constraints across a design process. You learn to schedule nets, control impedance on nets, control the propagation delay from your drivers to receivers, and match the propagation delay of driver and receiver pairs.

Learning Objectives

After completing this course, you will be able to:

- Define specific net scheduling of high-speed nets
- Match the propagation delay of nets and connections
- Define minimum and maximum propagation delays for nets and connections
- Identify high-speed constraint violations
- Identify all the high-speed constraints that you can apply to the nets in your designs
- Create spacing and physical constraints as well as area constraints and class-to-class rules
- Customize worksheets
- Create formula-based constraints
- Create customized constraints using the SKILL® programming language

Software Used in This Course

- Allegro PCB Designer, High Speed Option
- Allegro Design Entry HDL

Course Agenda

Day 1

- Database setup
- User-defined net scheduling
- Propagation delay
- Relative propagation delay
- Impedance constraints

Day 2

- Total etch-length constraints
- System constraints
- Physical and spacing constraints
- Formula-based constraints
- Custom constraints

Audience

- Logic Designers
- PCB Designers

Prerequisites

You must have experience with or knowledge of the following tools:

- The Allegro PCB Editor, Allegro Package Designer, or Allegro Design Entry software

Related Courses

[AllegroPCBEditorIntermediateTechniques.pdf](#)

[AllegroDesignEntryHDLFronttoback.pdf](#)