

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	Allegro High-Speed Constraint Management v16.5
Course Category	System Interconnect Design – Allegro & OrCAD, Advance with Engineer Explorer Series
Duration	2 Days
Course ID	ES_86081_16.5
Product Version	16.5

This is an *Engineer Explorer* course that 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.

This course uses a series of lectures, examples, and hands-on labs so you can 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, match the propagation delay of driver and receiver pairs, and more.

Learning Objectives

In this course, you

- 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

Allegro Design Entry HDL XL
Allegro PCB Design HDL XL

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 already have knowledge of the following:

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

Related Courses

- [Allegro Design Entry HDL Front-to-Back Flow v16.5](#)
- [Allegro PCB Editor v16.5](#)
- [Allegro Package Designer v16.5](#)