

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 GRE Interconnect Flow Planning v16.5
Course Category	System Interconnect Design – Allegro & OrCAD
Duration	2 Days
Course ID	ES_84486_16.5
Product Version	16.5

Course Description

The *Global Route Environment* (GRE) provides the technology and methodology to capture as well as adhere to a designer's intent. In the Allegro® GRE Interconnect Flow Planning course, you apply this technology and methodology to convert your design intent into the final board design.

You develop the first stage of flow planning by creating bundles manually, applying constraints to them, and editing the bundles. After this initial stage, you develop and define the flow of the bundles. After defining the flow, you plan (or route) it. You explore the three phases of planning, spatial, topological, and accurate, in detail.

Learning Objectives

After completing this course, you will be able to:

- Understand GRE terminology and technology
- Create, edit and use bundles
- Create and modify the flow of a bundle
- Understand the entire planning process
- Identify the three stages of planning, spatial, topological, and accurate
- Determine how to increase cycle time by having multiple designers work on the same design at the same time

Software Used in This Course

- Allegro PCB Global Route Environment Option - XL

Software Release(s)

- SPB16.5

Course Agenda

Note that this course can be tailored to better meet your needs – [contact the Cadence training staff](#) for specifics.

Day 1

- Introduction to the Interconnect Flow Designer (IFD) and GRE
- Creating and Using Bundles
- Editing Bundles
- Flow Editing

Day 2

- Planning Overview
- Spatial Planning
- Topological and Accurate Planning
- Increasing Throughput

Audience

- PCB Designers

Prerequisites

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

- Allegro PCB Design HDL L or Allegro PCB Design CIS L
or
- Allegro PCB Design HDL XL or Allegro PCB Design CIS XL

You must have completed the following courses:

- Allegro PCB Editor Intermediate Techniques v16.5

Related Courses

- [Allegro High-Speed Constraint Management v16.5](#)