

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 PCB Router Basics
Course Category	System Interconnect Design – Allegro & OrCAD
Duration	2 Days

In this course, you learn how to use the Allegro® PCB Router to perform automatic and interactive routing. You run autorouting, modify design rules, run interactive routing, analyze router log files, fan out a design, and improve manufacturability. You also restrict routing to specific areas of the board and keep signals away from other areas. Some of the introductory material assumes that you are using the router in conjunction with the Allegro PCB Editor tool for board layout.

Learning Objectives

After completing this course, you will be able to:

- o Understand some of the underlying data and processes involved when you route an Allegro PCB Editor design using the Allegro PCB Router
- o Create do (command) files to control the autorouting process or modify design rules
- o Perform batch mode routing to test different strategies
- o Analyze router log files to predict success or failure as early as possible
- o Perform interactive routing
- o Autoroute only specific nets and parts
- o Restrict routing to specific areas of the board, and learn how to keep signals away from other areas
- o Understand the fanout router and fan out a design
- o Examine strategies for routing difficult designs
- o Clean up autorouted connections, diagonalize corners, and reduce via counts to improve manufacturability

Software Used in This Course

- o Allegro PCB Designer

Software Release(s)

- o SPB 16.5

Course Agenda

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

Day 1

- o Process Overview
- o Basic Commands and Controls
- o Analyzing Router Results

Day 2

- o Interactive Routing
- o Fanout Router
- o Strategies for Difficult Designs
- o Manufacturing Preparation

Audience

- o This course is for new users who need a basic understanding of the PCB Router. The course assumes that you are using Allegro PCB Router in conjunction with Allegro PCB Editor.

Prerequisites

You must have experience with or knowledge of the following:

- o PCB design
- o Windows, Linux, or UNIX

Related Courses

- [Allegro GRE Interconnect Flow Planning v16.5](#)