

TRAINING

Bei dem hier beschriebenen Training handelt es sich um ein Cadence Standard Training. Sie erhalten eine Dokumentation in englischer Sprache. Die Trainingsprache ist deutsch, falls nicht anders angekündigt.

Unter <http://www.FlowCAD.de/TrainingKontakt.php> können Sie sich zum Training anmelden.

Course Title	Allegro PCB Editor Advanced Methodologies
Course Category	System Interconnect Design – Allegro & OrCAD
Duration	1 Day

Course Description

This class is designed around more advanced topics and exploration of the tool. This course does not teach basic tool operations. It is required that designers who are not actively using the tool first complete the Allegro® PCB Editor Basic Techniques and the Allegro® PCB Editor Intermediate Techniques courses.

In this course, you will explore the use of advanced methodologies available from within the PCB Editor software. You will start with creating multiple cross sections in your design. This is usually required with Flex designs but can also be used in standard PCB designs. You will then explore creating Inter Layer checks. Again, this is a requirement for Flex designs but can also be used in standard PCB designs. Next, you will create via structures. Via structures consists of a series of vias, usually blind and/or buried vias, connected by routing to create a single element that can be used much like a library element. Finally, you will set up your design and run the Backdrilling routine. Backdrilling is a manufacturing technique that can be used to remove stubs created by vias.

Learning Objectives

After completing this course, you will be able to:

- Create multiple cross sections in a design
- View and create Inter Layer checks
- Create and work with Via Structures
- Use backdrill to remove stubs
-

Software Used in This Course

- Allegro PCB Designer
- High-Speed Option (only needed for High-Speed Via Structures)

Software Release(s)

SPB17.2-2016

Modules in this Course

- Multi-Cross Section Support
- Inter Layer Checks
- Via Structures
- Backdrilling

Audience

- This course is for PCB Designers and others who wish to learn more about the Allegro PCB Editor tool.

Prerequisites

You must have completed the following courses or have the equivalent experience:

- [Allegro PCB Editor Basic Techniques](#)
- [Allegro PCB Editor Intermediate Techniques](#)