

TRAINING

The training described here is a Cadence Standard training. You will receive the documentation in English. The training language is English, unless otherwise announced.

You can sign up for this training at <https://www.flowcad.com/en/training.htm>

Course Title	Allegro System Capture v17.4
Course Category	Engineering Managers, Design Engineers, PCB Layout Designers
Duration	1 Day

Course Description

This course shows you how to create a schematic design using Allegro System Capture. Allegro System Capture helps you capture the design of a PCB in a schematic form and add constraints to it. The schematic design is passed to the layout tool, Allegro PCB Editor, to place and route the board-level design. Any changes made in the layout are then brought back to the schematic to complete the round trip.

Learning Objectives

After completing this course, you will be able to:

- Create a System Capture project
- Create flat and hierarchical System Capture designs
- Copy System Capture schematic pages
- Import DE-HDL and System Capture blocks
- Generate a BOM report
- Edit properties and define routing rules
- Export the design to PCB Editor for placement and routing

Software Used in This Course

Allegro System Capture

Software Release

SPB 17.4 or later

Course Agenda

- Introduction to Allegro System Capture
- Project Setup
- Creating a Schematic
- Creating a Hierarchical Design
- Design Rules
- Rules-Driven Layout

Audience

This course is intended for design engineers.

Prerequisites

- You must have a working knowledge of the front-to-back PCB design flow

Related Courses

Allegro PCB Editor Basic Techniques