

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.

Allegro PCB Editor SKILL Programming Language v16.01

Description

The Allegro® PCB Editor SKILL Programming Language course provides you with the basic knowledge required to begin writing useful commands and functions to customize and extend the functionality of the base PCB Editor and Advanced Package Designer (APD) tool set. The first part introduces you to the core SKILL programming language as it exists in the PCB Editor and APD design tools. The second and third parts teach you how to use the SKILL programming language to write interactive and batch functions that access and manipulate the PCB Editor and APD user interface and database.

Learning Objectives

In this course, you learn to

- Use the SKILL interpreter in the PCB Editor environment
- Control program flow
- Work with lists
- Access the PCB Editor database
- Interact with the user through the PCB Editor user interface
- Build menus and forms with Allegro PCB Editor SKILL
- Register new PCB Editor commands

Software

Allegro PCB Design XL
Cadence® Advanced Package Designer

Audience

This course is for CAD programmers, tool integrators, system administrators, and PCB Editor or APD users who wish to customize the PCB Editor or APD environments.

Prerequisites

A working knowledge of PCB design.
Some programming experience is helpful, but not required.

Course Agenda

Day 1

- Summarize SKILL syntax and data types
- Build and manipulate lists
- Create user-defined functions
- Control program flow
- Read and write text files in the operating system

Day 2

- Read data from the Allegro PCB Editor database
- Create new elements in the Allegro PCB Editor database

Day 3

- Request keyboard input from the user
- Request mouse input from the user
- Create forms and menus
- Register new PCB Editor commands