

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	Analog Simulation with PSpice-v16.3
Course Category	System Interconnect Design – Allegro & OrCAD
Duration	3 Days
Course ID	ES_5000432_16_3
Product Version	16.3

PSpice training is a three-day course that starts with the basics of entering a design for simulation and builds a solid foundation in the overall use of the tool.

Learning Objectives

The highlights of this class include the following:

- Learn how to enter a design for simulation
- Run DC bias, DC sweep, and AC sweep analysis
- Edit stimulus and run parametric analysis
- Learn how to edit models, run Monte Carlo analysis, create subcircuits and create parts for simulation from a model or subcircuit definition
- Learn how to create linear and non-linear transformers, perform temperature, worst case and noise analysis
- Learn about analog behavioral modeling, as well as running digital and mixed analog and digital simulation

Audience

- Engineers seeking maximum productivity in minimum time
- Engineers new to analog and mixed-signal simulation

Software

You need one of the following products:

- PSpice
- PSpice A/D

Prerequisites

- You must be familiar with creating a schematic using OrCAD Capture or Allegro Design Entry HDL
- You must also be proficient with using Windows and standard Windows applications

Course Agenda

Day 1:

- Building a design for simulation
- Setting up and running DC bias point analysis
- Setting up and running DC and AC sweep analysis
- Viewing simulation results in the probe window
- Setting up sources and using markers
- Creating and simulating a text netlist
- Accessing the stimulus editor using VSTIM, ISTIM, and DIGSTIM
- Running transient analysis
- Working with local and global libraries

Day 2:

- Examining common simulation errors
- Creating linear and non-linear transformers
- Setting up and running parametric analysis
- Creating a subcircuit
- Creating parts for simulation models
- Performing temperature analysis
- Configuring and running Monte Carlo analysis
- Simulating with hierarchical blocks and symbols

Day 3:

- Simulating using analog behavioral modeling
- Using digital components in a design
- Combining analog and digital components in designs
- Using performance analysis and creating goal functions
- Setting up and running worst case analysis
- Setting up and running noise analysis

Related Courses

- [Advanced PSpice for Power Users-v16.2](#)