

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.

## Analog Simulation mit PSpice (AMS)

### Description

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.

The highlights of this class include the following:

- Learn how to enter a design for simulation; run DC bias, DC sweep, and AC sweep analyses; edit stimulus; and run transient and parametric analyses.
- 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 analyses.
- Learn about analog behavioral modeling, as well as running digital and mixed analog/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 Capture or Concept. You should be proficient with using Windows and standard Windows applications.

### Course Agenda

- Building a design for simulation
- Setting up and running DC bias point analyses
- Setting up and running DC and AC sweep analyses
- 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
- 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
- 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