

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.

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

Course Title	Analog Simulation with PSpice
Course Category	System Interconnect Design – Allegro & OrCAD
Duration	3 Days

“I liked and learned very much about the practice and resolving simulation. The trainer has done a very good job on my learning experience.” (Fei Fan, ebm-papst Mulfingen, February 2014)

Course Description

The Analog Simulation with PSpice® course starts with the basics of entering a design for simulation and builds a solid foundation in the overall use of the software. You run DC Bias simulations, transient analysis simulations, and sweep simulations, allowing you to sweep component values, operating frequencies, or global parameters. You also have the opportunity to simulate several types of analog circuits, transformers, digital circuits, and mixed analog and digital circuits

Learning Objectives

After completing this course, you will be able to:

- Enter a design for simulation
- Run DC bias, DC sweep, and AC sweep analyses
- Edit a stimulus and run a parametric analysis
- Edit models, run a Monte Carlo analysis, create subcircuits, and create parts for simulation from a model or subcircuit definition
- Create linear and nonlinear transformers, and perform temperature, worst-case, and noise analysis
- Apply analog behavioral modeling and run digital and mixed analog and digital simulation

Software Used in This Course

- OrCAD® Capture
- PSpice A/D

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 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

Day 2

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

Day 3

- Running simulations 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

Audience

- Analog Designers
- Electrical Engineers