

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.

Advanced PSpice for Power Users

Description

This is an Engineer Explorer class that is designed around more advanced topics and exploration of the tool. This training material is intended to introduce users to the advanced analysis features of PSpice Simulator.

The «Advanced PSpice for Power User» class is a two-day course that focuses on library, models, convergence and design analysis-related issues of concern to the advanced user.

In this course you will learn to:

- Use techniques of creating PSpice Designs
- Create, edit and implement models, create subcircuits and create parts for simulation
- Analyse temperature behavior
- Resolve convergence problems
- Create linear and non-linear transformers
- Create magnetic core models
- Use the Magnetic Parts Editor

Audience

- Engineers seeking maximum productivity in minimum time
- Analog Designers
- Electrical Engineers

Software

- You need one of the following products:
- PSpice
- PSpice A/D

Prerequisites

- You must be familiar with creating a schematic using OrCAD Capture
- You must also be proficient with using Windows and standard Windows applications

Related Courses

Analog Simulation with PSpice

Course Agenda

Day 1:

- Useful techniques of creating PSpice Designs

- Overview of the libraries
- Model abstraction - Structure of the Models)
- Working with the Breakout Library
- Analog Behavioral Modeling
- Creating PSpice Models (different implementations)
- Modelling of Temperature Behavior

Day 2:

- Theory of PSpice Convergence
- Troubleshooting Convergence Failure
- Magnetic Core Modeling
- Linear Transformer Model
- Non-linear Transformer Model
- Core Model Level 2 (Jiles-Atherton) and Core Model Level 3 (Tabrizi)
- Creating a B-H Core Model using Model Editor (from Datasheet)
- Using Magnetic Parts Editor