

## Analog simulation with Pspice AD / AMS Simulator

Duration: 2 days  
Time: 9:00 – 17:00

Description: The PSpice training is a two-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

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

Prerequisites: Proficiency with 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