

PSPICE ESSENTIALS

TRAINING SYLLABUS



SYLLABUS DETAILS (2-DAY):

Day One

LESSON 1- Building a Design for Simulation

Creating a circuit in Capture that is PSpice ready and assigning various elements (reference designator, voltage values, net names, etc.)

LESSON 2- Bias Point Analysis

Executing the simulation created in Lesson 2 with a Bias point simulation and navigating the results.

LESSON 3- DC Sweep Analysis

Building a signal clipping circuit and executing an AC Sweep on that circuit and navigating the results for a "complex" or advanced trace result. This explores various advanced functions and markers (in Capture).

LESSON 4- AC Sweep Analysis

Understand how to generate a footprint project, import a footprint library, navigate the footprint project views, check in and out footprints for versioning, and how to "Promote" the footprint once completed and approved. Labs are included.

LESSON 5- Transient Analysis

Returning to the original RC design (Lesson 2), we add an amplifier to the circuit and then setup and run a transient analysis, examining the transient related options for controlling the simulation execution and resolution of the results.

LESSON 6- Modelling Applications

This explores the newly added capability of PSpice for modeling applications, providing quick access and quick creation of realistic (i.e. non-ideal) components and stimulus sources such as capacitors, inductors, piece-wise-linear sources and noise sources.

Day One (con't)

LESSON 7: Resolving Simulation Errors

Explores the various typical errors commonly encountered such as syntax, netlisting, and convergence issues.

Day TWO

LESSON 8: Transformers

Covers "linear" (mathematical based or electrically coupled only) and "non-linear" (magnetic core based) transformer models.

LESSON 9: Parametric Analysis

Covers creating parameterized variables of sweeping values, model parameters, temperature, and more.

LESSON 10: Linking PSpice Models to Capture

This is the first of the 3 modeling lessons, where you are given a model file from a vendor as if you had downloaded a new model from a vendor to include in your simulations. Covers obtaining the model, creating a symbol for use in Capture, linking to the newly created symbol library, and linking PSpice to the model file.

LESSON 11: Editing a Model

Second modeling lesson. Covers editing an existing model using the PSpice Model Editor to a new performance characteristic, as well as creating a new model from a data sheet specification.

LESSON 12: Monte Carlo Analysis

Covers what Monte Carlo analysis is, how to prepare a circuit for it, and how to configure the simulator to perform Monte Carlo.

LESSON 13: Worst Case Analysis

Covers what Worst Case analysis is, how it's configured for simulation, and some to the limitations of Worst Case as applied to the PSpice way of analysis.

LESSON 14: Analog Behavioral Modeling

Covers the a limited selection of the vast amount of Analog Behavioral Models within PSpice where you can create simulated model behavior without having to build an actual transistor based modle.

LESSON 15: Digital and Mixed Circuit Analysis

Covers working with mixed signal design (analog/digital), how to setup/control the simulations and discusses the digital model for control of timing and I/O behavior.

APP I: Adding New Parts

The third modeling lesson where you can setup a schematic for conversion into a PSpice model. Also covers creating a symbol from scratch using the Capture Part Editor, and setting that symbol up to link to a PSpice model.