PSpice Training

In This Issue

Spicemodel.com eases model search

Have you ever searched for a simulation model on the internet? If so, you understand that it is a time consuming search, and you are directed to multiple sites, many times to find that the simulation model is not available.

Spicemodel.com is a free service of Interface Technologies, and contains a search engine that lists models available in various cad systems and from device manufacturers. This is a special purpose search engine for a concentrated subset of the Internet, focussing on models and modeling application notes. Try it out!

www.spicemodel.com

INTERFACE TECHNOLOGIES
ENGINEERING·SOFTWARE AND·SERVICES
111 10TH AVE · MELROSE PARK, IL 60160
TEL: 708.366.4411   FAX: 708.366.4413
EMAIL: DAN@I-T.COM   WEB: WWW.I-T.COM

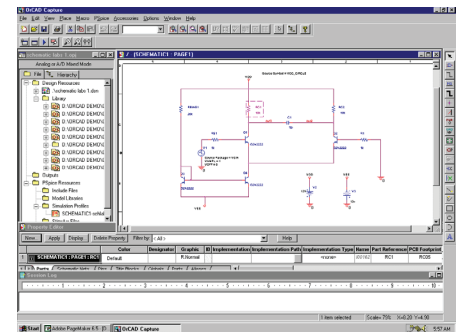## PSpice training class covers all the basic skills needed to be productive.

Interface Technologies' PSpice training now applies to users of the versions 9.x, 10.x, 15.x, 16.x and 17.x of Cadence's PSpice. We also have a version of the class for users of MicroSim Schematics!

Our class shows you how to take advantage of the new features of the OrCAD design environment by understanding its operation and keeping your design as simple as possible. PSpice training classes are conducted monthly around the United States, as well as on-site at customer facilities (including International locations).
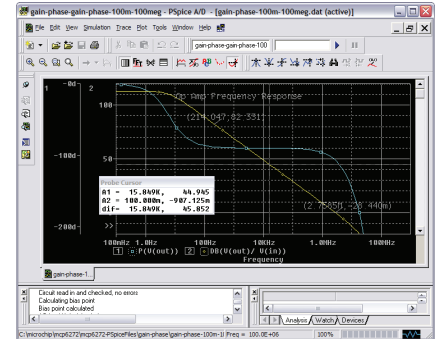
Understand the OrCAD Capture and Project Manager centered design environment.

If you enter a design and run into problems simulating it, it might be due to specifying the wrong kind of project. Changes in library structure, menus, multiple project types (some of which don't simulate) and a modal operation mode all contribute confusion to the use of Capture with PSpice. The "Introduction to PSpice A/D with Capture" class covers these subjects in detail, showing the user the "proper" method, and how to recover if your design is already entered.

Introduction to PSpice with OrCAD Capture

Who should attend

A typical student is someone that needs or wants to use Spice simulation software to become a better designer. Our typical class includes students of the following user levels:

- Beginners with little or no previous experience with spice

- Existing users that want to become more productive

- Advanced users that want to expand their command of the more sophisticated features

- Direct managers of an analog design group

- Users that want a fast learning curve on the transition from previous Versions

What you will learn in the 2 day class

The main objective of the training class is to "jump start" the student's knowlege to a level where they are productive with the software. This includes a presentation of how the software works, how to create a design, design requirements for simulation, how to run the three main types of simulation as single and multi-run simulations.

Particular attention is paid to the understanding of symbol and model libraries, which is a key element to simulating successfully. Subjects covered include how to determine if a component or library is configured properly, generating new spice models, and how to link a model to a symbol for successful use in a simulation.

Detailed Course Outline

During the 2 day class, you will learn the basic functions of PSpice and Capture. Each subject area consists of a verbal explanation with overheads, an instructor demonstration, and a hands-on lab for the students. Subjects include the following:

- Installation directories, file types and locations

- Overview of OrCAD software architecture, and implications to the user

- Overview of OrCAD Capture environment

- Definition and relationship of a project, design, schematic, libraries

- Creating a new project, and organizing your work on your computer

- Flat schematics

- Hierarchical schematics

- Sources and grounds: how, when and which one to use

- What to do when your schematic does not simulate.

- Schematic and simulator intereactions, and how to interpret error messages

- The development of Spice, and how that affects what you do today.

- Simulation types: DC, AC, Transient

- Mixed A/D analysis

- Single and Multi-run Simulations

- Parameter sweeping

- Monte Carlo and Sensitivity analysis

- Simulation and convergence problems, how to overcome the problems.

- Viewing and Measuring waveforms

- Using markers to take measurements

- Automatic waveform measurements

- Waveform mathematics

- Spice models: what they are, where to get them, how to use them.

- Downloading models from the internet, and using them in a schematic/simulation

- Creating your own spice models

- Function blocks and ABMs: Analog Behavioral Models

- Overview of online documentation and application notes.

- Putting it all together: schematics, simulation, measurements, engineering tasks
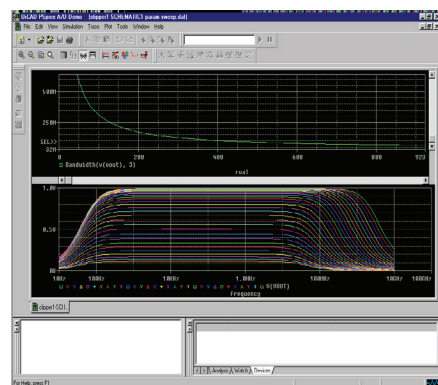
The PSpice window was updated in version 16. Find out how to use the new enhancements to be more productive. Single and multi-run simulations, and automatic measurements are covered in detail.

The PSpice Masters Class

Dan Waterloo giving Bill Nye, 'The Science Guy', some tips on how to use spice.

About Your Instructor

Daniel P. Waterloo received a BSEE from the University of Illinois in 1976, an MBA and a Masters of Project Management from Keller Graduate School of Management in 1987 and 1993, respectively. Mr. Waterloo has been involved with many aspects of spice and spice modeling since 1985. After founding Interface Technologies in 1993, he became the 4rth largest (of 56 worldwide) distributor of MicroSim PSpice. Based on many customer requests, he created a hands-on PSpice training class, which he has delivered since 1996. Interface Technologies distributes design software to the Electrical Engineering community throughout the world. Interface Technologies has customers on 5 of the 7 continents of the world: North America, South America, Europe, Asia and Austrailia. Interface Technologies hosts the "Spicemodel.com" website which distributes spice models, provides tech support for Spice simulators, and generates simulation models for 20 semiconductor companies.

INTERFACE TECHNOLOGIES
ENGINEERING·SOFTWARE AND·SERVICES
111 10TH AVENUE · MELROSE PARK, IL 60160
TEL: 708.366.4411   FAX: 708.366.4413
EMAIL: DAN@I-T.COM   WEB: WWW.I-T.COM

Partial List of Advanced Subject Areas

Our PSpice Masters class starts with all of the materials listed in our 'Introduction to PSpice with OrCAD Capture', adds one day of instruction on Layout (introductory level), then expands into advanced subject modules listed below for detailed instruction on specific uses of PSpice and the OrCAD design suite.

- Analog Behavioral Modeling: in depth instruction
     Table Models
     Laplace Model
     Frequency Model
     Evalue and GValue (mathematical equation transfer functions)
- Probe:
     Syntax for waveform searches
     Creating your own Goal Functions
     Recording and Replaying Macros
- Temperature Analysis:
     Verifying that model is temperature dependent
     How to Specify Temperature with Spice
     Sweeping temperature, analyzing results
- Component Advanced Modeling:
     Defining your own tolerance distribution curve
     Adding temperature effects to your passive devices
     Adding parasitics and frequency varying characteristics to R, L, Cs
- Transformer modeling:
     Linear transformers
     Multiple winding transformers
     Non-Linear transformers: (saturating cores)
     The Non-Linear core model
     Specifying core parameters
- Moving data into and out of your design:
     Moving your schematic into MSWord
     Moving your waveforms into MSExcel
     Importing oscilloscope waveforms for use in Spice simulations
     Using Probe data as an input to simulation
- Noise Analysis
     Overview of noise components
     Discussion of model parameters that affect noise
     Determine if noise is modelled in a device
     Running a noise simulation
     Viewing and using noise calculations
- Switch-mode power supply simulation
     Modeling the PWM with Behavioral models
     Overview of various circuit topologies
     Setting the Spice options for better convergence
     Using Intial Conditions
- Introduction to Op-Amp modeling
     Creating useful op-amp models based on the Boyle model

SMPS Modeling and Simulation

Christophe Basso and Dan Waterloo discuss spice simulations at APEC 2019

About Your Instructor

Daniel P. Waterloo received a BSEE from the University of Illinois in 1976, an MBA and a Masters of Project Management from Keller Graduate School of Management in 1987 and 1993, respectively. Mr. Waterloo has been involved with many aspects of spice and spice modeling since 1985. After founding Interface Technologies in 1993, he became the 4rth largest (of 56 worldwide) distributor of MicroSim PSpice. Based on many customer requests, he created a hands-on PSpice training class, which he has delivered since 1996. Interface Technologies distributes design software to the Electrical Engineering community throughout the world. Interface Technologies has customers on 5 of the 7 continents of the world: North America, South America, Europe, Asia and Austrailia. Interface Technologies hosts the "Spicemodel.com" website which distributes spice models, provides tech support for Spice simulators, and generates simulation models for 20 semiconductor companies.

INTERFACE TECHNOLOGIES
Engineering·Software and·Services
111 10th Avenue · Melrose Park, IL 60160
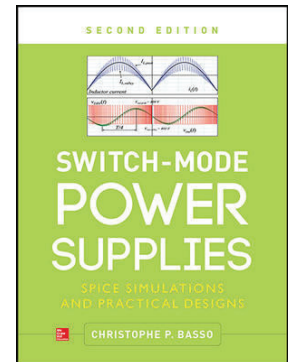Tel: 708.366.4411   Fax: 708.366.4413
Email: dan@i-t.com   Web: www.i-t.com

This advanced modeling and simulation class focuses on the topics needed to successfully model and simulate switch mode power supplies.

The class content parallels the material presented in Christophe Basso's excellent book: basic power conversion concepts, building generic modeling blocks and combining into controllers, and finishing with practical simulation techniques of SMPS designs.

More details on the book can be found here:
https://www.mhprofessional.com/9780071823463-usa-switch-mode-power-supplies-second-edition-group

SMPS Modeling and Simulation Class Content

Day 1: Spice Skills review

  - Schematic Entry
  - Simulation Overview, Settings
  - Waveform Viewing
  - Configuring Schematic Symbols
     and Simulation Models

Day 2 and 3: Power Conversion Overview, Modeling basics

Chapter 1 : Power Conversion Overview
 - Resistors
 - Switches
 - Duty Cycles
 - Topology Overview
Chapter 2: Small Signal Modeling
 - SSA
 - Average Models
Chapter 3: Feedback and Control Loop Overview
Chapter 4: Basic Blocks and Generic Switching Models

Day 4: Power Supply Simulations

Chapter 5: Non-Isolated Converters
Chapter 6: Offline Converters
Chapter 7: Flyback Converters
Chapter 8: Forward Converters

All attendees of this 4 day class will receive a hard copy of the book 'Switch-Mode Power Supplies' by Christophe Basso, a demo copy of the simulation software, example files from the book and developed by the instructor, hard copy and electronic copy of the overhead presentation, and a Certificate of Completion.

The class is scheduled by request. It is also delivered as an on-site classe, minimum of 3 students. The class costs $3995.00 per student. For on-site classes, the hosting company will need to provide computers and a conference room, and travel expenses for the teacher.

For inquiries, please contact Dan Waterloo at 'dan@i-t.com'.

Enrollment Form and On-Site Classes

Month (2019)	Location	PSpice Intro*	PSpice Master's *
January	Chicago, IL	January 14-15	
February	Phoenix, AZ	February 19-20	
March	Albuquerque, NM	March 11-12	
April	Denver, CO	April 15-16	
May	Chicago, IL	May 5-6	May 5-9
June	Las Vegas, NV	June 3-4	
July	NA		
August	Seattle, WA area	Aug 12-13	
September	Austin, TX	September 15-16	September 15-19
October	Atlanta, GA	October 7-8	
November	Chicago Area, IL	November 4-6	
December	Phoenix, AZ	December 19-20	

* On-site classes for 4 or more students can be customized and are available by special arrangement with Interface Technologies.
Call Dan Waterloo at (708) 366-4411 (dan@i-t.com) to discuss your requirements.

On-site Classes

Special On-site Class Offer!

Train up to 15 students for the price of 5!!!
Order your onsite class on or before Sept. 30, 2020, and receive one two day session for $8.975 (plus travel expenses).

Add an optional 3rd day to cover advanced subjects that are critical to your engineering efforts. Many companies add a day of selected subjects from our PSpice Masters class after the Introduction to PSpice class.

Call us or send an email with your requirements to schedule a 2 or 3 day onsite class!

INTERFACE TECHNOLOGIES
ENGINEERING·SOFTWARE AND·SERVICES
111 10TH AVENUE · MELROSE PARK, IL 60160
TEL: 708.366.4411   FAX: 708.366.4413
EMAIL: dan@i-t.com   WEB: WWW.I-T.COM

How to enroll:

Call Interface Technologies at +1 (708) 366-4411 with the class name and dates/location above, and the student information listed below. Or circle the class dates you are interested in, fill in the enrollment form below, and fax it to 708-366-4413. Your reservation will be confirmed within 24 hours, and you will receive additional information regarding travel arrangements by fax or email.

FAXBACK REGISTRATION FORM

I understand that the course costs $1,795/person, and includes two days of training, an evaluation copy of the OrCAD software and a certificate of achievement. (If reserving a space in the PSpice Masters Class, cost is $3,895 for 4 days of training.)

I prefer to pay for the class by _____ Credit Card _____ Check _____ Purchase order.

Name: _____

Company: _____

Address: _____

Mail Stop: _____

City, State, Zip_____

Telephone: (_____)_____

Email: _____

5

In This Issue

⇨ *PSpice Training Class dates and locations for 2020*

⇨ *On-Site class Special Offer*

⇨ *'PSpice Masters Class' in Austin TX*

The 'PSpice Master's Class' contains all of the advanced subject areas that have been requested by our customers over the last ten years.

Enjoy the Austin TX location while learning OrCAD Capture, PSpice and advanced techniques!

Visit www.i-t.com for detailed information!

'PSpice Master's Class' scheduled for September, in Austin TX!

Interface Technologies is please to announce the creation of the 'PSpice Masters Class', which will be given in Austin in September, 2020. The Masters Class is a consolidation of all our training material, delivered in a 7 hour/day, 4 day format.

It includes detailed instruction of the following subjects:

- 2 days: Introduction to PSpice with OrCAD Capture
- 2 days: Detailed Instruction on a selection of Advanced PSpice use:

- Simulating at different temperatures
- Assigning tolerances, and tolerance distributions
- Creating automatic waveform measurements by writing Goal Functions
- Model real world resistors, capacitors and inductors with parasitics
- Two approaches to creating multiple winding transformer models
- Discussion of model syntax, conversion of models between simulators
- Basic op-amp modeling using subcircuits and the Boyle model
- Initializing circuits with nodesets and initial conditions
- Creating models from datasheets, creating symbols for models
- Performing a noise simulation, modeling noise in devices
- Pulse width modulator chip modeling basics
- Modeling and simulating Power Supplies

For up-to-date scheduling information, please visit www.i-t.com

6