

Introduction to PSPICE: software package and capabilities

Lecture 9

Special Topics:
Device Modeling

Outline

- Basic concepts in PSpice
- Input and output files
- Capture and PSpice
- Editing circuits and models
- Hands-on session: working with example files

Introduction

- SPICE package family for analog and mixed-signal circuit simulation
- Parts and circuits modeling
- Device modeling in 2-D and 3-D through equivalent circuits
- Will use Lite version (free) of OrCAD PSpice and Capture CIS (package by Cadence)

SPICE: history

- SPICE program (Simulation Program with Integrated Circuit Emphasis) is dedicated to process modeling in electrical circuits
- First SPICE program was developed at the Electronics Research Laboratory of the University of California, Berkeley (SPICE1 1973, SPICE2 1975, coded in Fortran), was distributed for a nominal charge to cover cost of magnetic tapes
- SPICE inspired and served as a basis for many other circuit simulation programs

SPICE: general capabilities

- AC/DC analysis
- DC transfer curve analysis (a sequence of nonlinear operating points calculated while sweeping an input voltage or current)
- Noise analysis
- Transfer function analysis (a small-signal input/output gain and impedance calculation)
- Transient analysis (time-domain large-signal solution of nonlinear differential algebraic equations)

SPICE: available packages

- XSPICE, academic spinoff, by Georgia Tech
- Cider (previously CODECS, from UC Berkeley/Oregon State Univ.), added semiconductor device simulation
- HSPICE, commercial product by Synopsys, Inc.
- PSPICE, commercial product, presently owned by Cadence, distributed under OrCad
- Several SPICE-based circuit simulation programs by various IC manufacturers

Cadence OrCAD Lite package

- Capture CIS Lite
- PSpice AD Lite
- PSpice Advanced Analysis Lite
- PSpice Accessories:
 - Magnetic Parts Editor
 - Model Editor
 - Simulation Manager
 - Stimulus Editor

Request on-line free download or a disk

Capture CIS

- Capture CIS (component information system) - used to create schematics and produce connectivity and simulation information for printed circuit boards (PCB) and programmable logic designs
 - Choose to set up a PSpice project, PCB project, or programmable logic project when start a new project
- Provides standard libraries of parts that can be used to design schematics; use the property editor to add, change, or delete properties, create your own custom parts and library

Capture CIS

- All circuit designs can be entered as text files
- Capture provides user interface for creating a circuit, which is converted to a netlist text file
 - Netlist (default filename Schematic.net) – text description of all the components and their respective connections to other components
- This file can be parsed, properties of components modified before running through PSpice

PSpice Netlist file format

- Flat SPICE format netlists have the following general syntax (dependent on the part)
 - P<name>< + node>< - node> <parameter><value>
- For the voltage source and resistor (name is unique for each part):
 - V<name>< + node>< - node> DC <voltage value>
 - R<name>< + node>< - node> <resistance value>
- Capture uses the node names placed on the schematic page where available

Files needed for simulation

- To simulate a design, PSpice needs to know about:
 - Parts in your circuit and how they are connected
 - The simulation models corresponding to the circuit parts
 - What analysis to run
 - The stimulus definitions with which to test
- This information is provided in various data files: some are generated by the design entry program such as Capture, others come from libraries, and still others are user-defined

Files needed for simulation

- Capture (or other design program) generates the netlist file (*.net) and the circuit files (*.cir) that PSpice reads before doing anything else
- Each part has an associated model library (to determine its response to electrical inputs)
- The simulation profile contains references to the other user-configurable files that PSpice needs to read (for example, include file with PSpice commands, etc.)

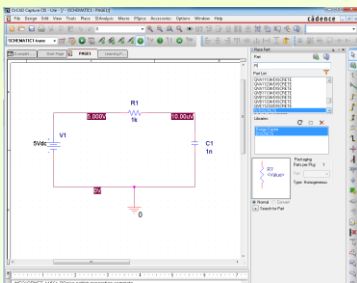
Files that PSpice generates

- After reading all required input files, the simulation starts
- Simulation results are saved to two files:
 - the probe data file (*.prb) with waveform results to be displayed graphically (add traces or cross-probes)
 - the output text file (*.out), that contains the netlist, PSpice command syntax for commands and options (e.g., enabled analyses), simulation results (will depend on simulation control parts like VPRINT1 and VPLOT1), warning and error messages

Capture CIS limits in Lite version

- Cannot save designs that have more than 75 nets, more than 60 parts, including the hierarchical blocks in the design. You can still view or create larger designs
- Cannot have more than 1000 parts in the Capture CIS database
- Cannot create parts with more than 100 pins
- Some other limitations

Capture CIS interface



- Create circuits by placing standard library parts
- Edit parts
- Run simulations through PSpice interface

PSpice

- Pspice is a simulation tool that performs analysis of analog and mixed-signal circuits
- Includes model libraries that feature over 15,000 analog and 1,600 digital models of devices manufactured world-wide
- Uses schematics from Capture, including edited standard and custom parts
- Types of analyses: transient, AC, DC, parametric sweep, and worst-case

PSpice limits in Lite version

- Circuit simulation limited to circuits with up to 75 nodes, 20 transistors, no sub-circuit limits but 65 digital primitive devices, and 10 transmission lines (ideal or non-ideal) with not more than four pairwise coupled lines
- Device characterization and parameterized part creation using the PSpice Model Editor is limited to diodes

PSpice limits in Lite version

- Sample model library named eval.lib (containing analog and digital parts) and evalp.lib (containing parameterized parts) are provided
 - This library consists of parameterized resistor, source, and diode
- Magnetic Parts Editor allows you to design power transformers only
- Some additional limitations

PSpice advanced analysis

- Advanced Analysis tool (used in conjunction with PSpice A/D) is available to analyze
 - Sensitivity – identifies critical circuit components
 - Optimizer – optimizes key circuit components
 - Monte Carlo – analyzes statistical circuit behavior and yield
 - Smoke – detects component stress
 - Parametric Plotter – examines solution space through nested sweeps
- Limitations in Lite version

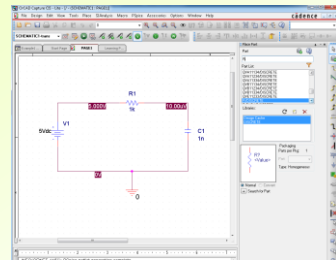
PSpice: Model editor

- Use the Model Editor to determine the model parameters for many standard device types, such as bipolar transistors, and to create the subcircuit models for more complex device types, such as operational amplifiers
- Simpler devices, such as resistors, may only need the resistance value to have a complete model

PSpice DC analysis

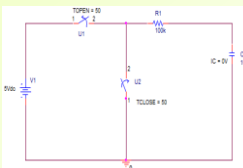
- DC bias point analysis calculates node voltages and currents through devices in the circuit
- It takes into account any voltage sources applied to the circuit and any initial conditions set on devices or nodes in the circuit
- The output file provides a list of the entire analog and digital node voltages, currents, and other relevant parameters
- Obtain IV curve for a PV device

Model for hands-on session

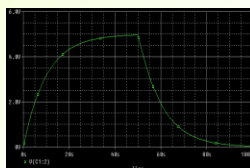


PSpice transient analysis

Sample RC circuits for transient analysis



Waveform for voltage across capacitor



Summary

- Lite version of OrCAD commercial package is available for evaluation in limited capacity, but for unlimited time
- Most basic functionality of the licensed package
- Limitations are primarily on the size of the model (number of parts)

References

- L. Castañer and S. Silvestre, Modelling Photovoltaic Systems using Pspice, Wiley 2002 (Example files can be downloaded from <http://www.esf.upc.es/esf/>)
- OrCAD Capture user manual
- OrCAD PSpice user manual