PSpice® - A Critical Thread in Vertical and Horizontal Curriculum Integration

Gopal Mohan, J. Michael Jacob
Purdue University, West Lafayette, Indiana

Introduction

The Electrical Engineering Technology Department at Purdue University uses PSpice® for circuit simulation. PSpice is a commercial package derived from the public domain SPICE, from the University of California, at Berkeley. PSpice until recently was a registered trademark of MicroSim™ Corporation. The evaluation version, PSpice 8.1EV, is available in the labs for students' use. It is also available for checkout from the departmental office so that students may install it on their personal computers. One lab is equipped with the professional version. The reason for choosing PSpice is that it is derived from SPICE, which is the *de facto* standard for analog circuit simulation and also because MicroSim provided its evaluation versions free of cost and encouraged free distribution to students. OrCAD, Inc. has recently acquired MicroSim Corporation.

The Electrical Engineering Technology department and its faculty are charged with the responsibility of imparting to the students a broad working knowledge of a circuit simulator. The challenge is to incorporate the capabilities of this circuit simulator across the curriculum. This is compounded by the time constraints of a rigorous curriculum and by the requirement that the students use this knowledge for the duration of their education at Purdue and beyond.

Why is learning and using a circuit simulator necessary? There are several reasons why circuit simulation is essential. Circuit simulation allows interpretation of circuit behavior (say, over a wide range of frequencies), thus the focus is broadened from a mere computation of circuit variables, which, of course, is essential as a first step. Circuit simulation allows analysis at the system level once the behavior at lower levels is understood and tested. Also, simulation is valuable in situations involving enormous computation. The post processing capabilities of circuit simulators (Probe in PSpice) serve the invaluable function of plotting the results of simulation with the capability of arithmetic operation. Here, the advantage over manual computation is that the circuit behavior is observed both as numerical and graphical data.

Background

At the Electrical Engineering Technology department at Purdue, circuit simulation is not taught in the first semester of the freshman year. In the Introduction to Circuit Analysis (EET 107) and the Digital Fundamentals (EET 196) courses students use hand calculations only. Circuit simulation is intentionally avoided to ensure that the students learn the fundamental skills of manual circuit analysis and computation.

The underlying philosophy is that simulation is one of a wide variety of tools. It is the user who is responsible for interpreting the results of a simulation in comparison with the results from

manual calculations and lab measurements. Each method has its strengths and weaknesses. Proper analysis requires a comparison of the results from all three techniques. Students are introduced to PSpice in the second semester of their freshman year in the second analog sequence course, Electronics Circuit Analysis (EET 157), and in the Digital Applications (EET 157) course.

The Electrical Engineering Technology department has instituted a policy wherein the lab sections are cross-taught. This means that a faculty member teaches a lab section of a course whose lecture is taught by another faculty member. This keeps the faculty members involved and informed of what students are learning in other courses and is also very helpful in coordinating and promoting students' skills in computer simulation software.

Students learn PSpice in a step-by-step approach. It is essential they understand the electrical properties and behavior of the components used in the circuits they simulate. They are introduced to the software with simple examples. The skills are built gradually and earlier skills are reinforced by an iterative approach. Consistently, the simulation results are compared with manual calculations and/or experimental results. Thus a strong foundation is laid which helps the student enormously when it comes to using circuit simulation in the design and implementation of group and individual projects in the Project Design and Development courses (junior and senior years).

It is emphasized to the students that PSpice does not design the circuit for them. Often, students are required to interpret netlists and construct circuits/schematics based on netlists. This gives them a better understanding and feel for the circuits they are about to analyze. Members of the Electrical Engineering Technology Industrial Advisory Board highly commend this practice of students interpreting netlists. PSpice simulation does not replace or exempt the student from doing the rigorous manual calculations, and the construction and testing of circuits in the laboratory. Rather, circuit simulation is used to support and verify the results obtained by manual calculations and those from laboratory experiments. Many courses require PSpice simulation verification of calculated and experimental results to be included in the laboratory report. The students analyze and explain any variations among the manual calculations, experimental measurements, and simulated results.

Freshman Year

Purdue's Electrical Engineering Technology students are introduced to PSpice simulation in the second semester of their freshman year. In the Electronics Circuit Analysis (EET 157) course, students learn analog simulation. PSpice simulation is done to support laboratory findings. Simulation involves transistor circuits and operational amplifier circuits. Probe is used to plot and study the time domain gain and the phase relationship (inversion or non-inversion) of input and output of amplifiers.

Below are the simulations required during the Fall 1998 semester.

- Transistor common-emitter amplifier
- Differential voltage amplifier
- Non-inverting op-amp voltage amplifier

- Inverting op-amp voltage amplifier
- Summing voltage amplifier
- Waveform generator circuits

Digital and mixed signal circuit simulation are done in the Digital Applications (EET 159) course. There is an introductory lab assignment in circuit simulation, followed by homework assignments. Typically, students have weekly PSpice assignments that are individually performed outside of the lab. Half of these involve purely digital simulation and the remainder involves mixed signal circuit simulation. Probe is used to plot the output and to examine any spikes in the output. The simulations required during the Fall 1998 semester included the following:

- Design of a synchronous counter
- Analysis of a frequency divider circuit
- Frequency divider design
- Analysis of a 8-bit parallel generator
- Pulse width modulator analysis
- Design of a modulus-7, recycling, synchronous counter using JK flip-flops
- 555 monostable multivibrator
- Design of an astable multivibrator using two 555 timer chips
- Voltage window detectors using analog comparator
- Astable multivibrator with 555 timer
- Analysis of a square wave burst generator circuit
- Voltage-controlled oscillator
- 4-bit R-2R DAC circuit with an op-amp output
- Design of high-side and low-side drivers

Sophomore Year

The sophomore year is marked by even more intense PSpice simulation. In the first semester AC Electronics Circuit Analysis course (EET 207), every topic that is learned is reinforced by simulation. It is again pointed out that circuit simulation is used to support manual analysis. PSpice simulation is required for virtually every lab assignment. PSpice is done as part of an independent prelab assignment. The results of simulation should match the results of manual analysis for the prelab to be considered valid. Students rerun their simulations while in lab when actual component values are different from nominal values by more than 5%.

Students simulate in PSpice by interpreting instructor provided netlists. This requires the student to understand the input and output characteristics as well as the topology of circuits involving passive and active components. Thus, the course topics and simulation exercises are provided in conjunction and are mutually supportive. Students are encouraged to teach themselves with the instructor providing guidance. The Windows version, which provides online information in the form of dialog-boxes, graphics, and help files, is supportive of this type of instruction.

The concept of phase shifts in sinusoidal waves is introduced with AC sweep at a single frequency. Peak and effective values of sine waves are used in circuits. Simulation of responses

of series and parallel RC, RL, and RLC circuits with frequency sweep is performed. Both magnitude and phase plots are obtained. Probe is extensively used to understand the responses of these circuits. Probe plots are obtained for the responses of low pass, high pass, band pass, and resonance circuits. Amplifier frequency response plot is obtained and the concept of 3-dB frequency is studied and applied to the passband of amplifier and filter circuits. Decibel plots are made and roll-off is shown with the aid of cursors. Nonsinusoidal sources are used. Fourier analysis is done to extract the harmonics and phases of a square wave to verify the output from a filter IC used in the lab to separate the harmonics of the square wave.

Each student must individually take two lab practical exams that test his/her skills in lab. One-third of the grade in each of the two lab practicals is allocated to PSpice simulation exercises.

Below are the simulations required during the Fall 1998 semester.

- Review topics in PSpice
    RLC circuit single frequency AC analysis
    Non-inverting op-amp amplifier DC analysis
    Full-wave rectifier transient analysis
- Phase measurement with AC sweep at single frequency
- Magnitude and phase measurement of RLC circuit at single frequency
- Effect of reactance on op-amp inverting amplifier
- Low pass and high pass filters AC sweep analysis
- RLC series resonance
- Parallel tuned amplifier using BJT
- Transient response by Fourier analysis
- Harmonic extraction of a square wave

EET 257 is a required second semester sophomore course in Power and RF Electronics. Simulations are required weekly. The problems are scaled so that they can be solved with the evaluation version of MicroSim's PSpice 8.1EV, although there is a lab available with the professional version.

PSpice is *not* taught in EET 257. Instead it is used as a tool for two purposes. First it is used to predict and verify the manual calculations of the performance of a circuit *prior* to testing it in lab. Secondly it is used to investigate parameters (i.e. power, start-up transients) that are difficult to measure in lab or conditions that potentially may destroy the circuit without leaving good data. Efforts are made to point out the strengths and *weaknesses* of simulation, continuing to foster the attitude that this is just another tool that can provide information which is difficult to obtain any other way. But, simulation has its limitations and the results must be viewed in the context of design parameters, manual calculations, model definition, and hardware verification.

Below are the simulations required during the Fall 1998 semester.

- Power calculations
- Worst case analysis
- Op-amp review

- Class AB amplifiers
- Amplifier protection
- Reactive loads
- SCR firing
- MOSFET drivers
- H bridge
- Buck regulator
- Boost regulator
- High speed op-amp compensation
- Common emitter amplifier design verification
- LC oscillators

Students use circuit simulation to design and layout printed circuit boards to fabricate in the Electronic System Fabrication (EET 296) course in the second semester of the sophomore year. This course introduces project planning and basic concepts in electronic design automation (EDA). The students develop a project from an engineering rough sketch to a printed circuit by utilization of EDA (PSpice). MicroSim "PCBoards" is used. The project training involves analog, digital, and mixed signal circuit ideas. Circuit performance is verified by mixed signal simulation before the students link to the PCB layout program. This course is precursor to the Project Design and Development courses in the junior and senior years.

Incidentally, it is noted that the Purdue EET-AS graduate will have acquired wide simulation knowledge in analog, digital, and project design. For the continuing bachelor degree student, this rigorous training in circuit simulation helps tremendously in the upper level courses.

Junior and Senior Years

The junior and senior years are comprised of advanced, elective, and project design courses. Circuit simulation is used in many of these upper division classes. For example, simulation involving advanced concepts is done in Analog Network Signal Processing (EET 307) course.

The following simulation projects were performed in the EET 307 course during the Fall 1998 semester:

- PSpice review
    A review of frequency response and transient analyses. Students simulate a x10 oscilloscope probe to illustrate probe compensation.
- Fourier analysis – Fourier coefficients
    Transient analysis simulations to find the Fourier coefficients of periodic waveforms.
- Fourier analysis – Filter spectral analysis
    Spectral components are determined using PSpice and the amplitude and phase of each component is filtered to determine the spectral components of the filter output. The output components are summed to determine the output waveform.

- Fourier analysis – Harmonic distortion

  Transient analysis is performed on an overdriven op-amp amplifier to determine the amount of harmonic distortion.
- First order networks – Poles and zeros

  Frequency response analyses are run on first order networks containing a pole and a zero to illustrate the roll-off characteristics between break points.
- First order networks – Op-amp characteristics

  Frequency response analyses are run on internally and externally compensated op-amp amplifiers to illustrate gain-bandwidth and slew rate characteristics.
- Second order networks – Gain characteristics

  Frequency response analyses are run on underdamped, overdamped, and critically damped second order networks.
- Second order networks – Time domain response

  Transient analyses are run on the same second order networks to examine the time domain responses.
- Second order networks – Complex poles

  Transient analysis is used to determine the step response characteristics and stability of second order, underdamped networks. A Sallen-Key second order section of an active filter is used to control the damping of the network.
- Butterworth and Chebyshev active filter designs

  PSpice is used to simulate the operation of several active filter designs. Low pass, high pass, and band pass designs are simulated using Sallen-Key and State variable configurations. Both frequency response and transient response analyses are performed.
- Step analysis based on Laplace techniques

  A PSpice part (Laplace) is introduced to illustrate how analyses can be performed on the transfer function of a network rather than circuit itself. Both analysis forms are performed and the results compared.

Although PSpice simulation is not employed in the Advanced Digital Systems (EET 345) course, simulation skills and knowledge acquired using PSpice for both digital and mixed signal simulation in the Digital Applications (EET 157) course are very beneficial when students do their work with CPLD.

In the elective Linear Integrated Circuits (EET 368) course, students investigate the operation of analog circuits at the extremes of low level DC inputs and outputs, single supply biasing, and high frequency performance. Students compare the results of PSpice simulation to lab measurements as well as to the results of another circuit simulator, the Electronics Workbench®. Efforts are made to point out the strengths and limitations of simulation.

Below are the simulations required during the Spring 1998 semester.

- DC characteristics
- Op-amp speed
- Composite amplifier
- Switched capacitor voltage inverter

- Switched power supply computer exercise
- Sallen-Key
- State variable
- XR2206 function generator part 1
- XR2206 function generator part 2

Concluding Remarks

Knowledge and understanding of the proper application of simulation is of critical value to the practicing engineer or technologist. PSpice, the circuit simulation software, provides this foundation. Our graduates are often faced with situations that need informed decision making and economics demands that the cost to this decision making process be kept to a minimum. Simulation plays a vital role in trying out the various options the technologist might consider without expending costly resources. More could be achieved on a circuit simulator including simulated measurements as compared to actually doing it on the lab station. The computer with its capacity for iterative calculation also gives a global perspective as compared to solution around a single operating point in a circuit. Electrical Engineering Technology education at Purdue provides this critical training in simulation. Circuit simulation using PSpice provides the vertical threads that run through the course sequences. The elective and project design and development courses are intertwined in the fabric of learning that is fostered in the Electrical Engineering Technology Department at Purdue University.

Bibliography
1. Herrick, Robert J. Electronics Circuit Analysis Laboratory Manual. Learning Systems, Inc., West Lafayette, IN. (1998).
2. Boylestad, Robert L. Introductory Circuit Analysis. Prentice-Hall, Inc., Upper Saddle River, NJ. (1997).
3. Moss, Gregory L. EET 159 Digital Applications Simulation Exercises.   Purdue University (1998).
4. Jacob, J. Michael. AC Electronics Circuit Analysis Laboratory Manual. Learning Systems, Inc., West Lafayette, IN. (1998).
5. Herniter, Marc E. Schematic Capture with MicroSim PSpice®. Prentice-Hall, Inc., Upper Saddle River, NJ. (1996).
6. Tuinenga, Paul W. SPICE: A Guide to Circuit Simulation and Analysis Using PSpice®. Prentice-Hall, Inc., Englewood Cliffs, NJ. (1995).
7. Boylestad, Robert L. and Louis Nashelsky. Electronic Devices and Circuit Theory. Prentice-Hall, Inc., Upper Saddle River, NJ. (1996).
8. Jacob, J. Michael. Power and RF Electronics Laboratory Manual. Learning Systems, Inc., West Lafayette, IN. (1998).
9. Polus, John, D. Analog Network Signal Processing Laboratory Manual. Learning Systems, Inc., West Lafayette, IN. (1998).

GOPAL MOHAN
Gopal Mohan is an Assistant Professor of Electrical Engineering Technology at Purdue University, West Lafayette, Indiana. He has previously taught at Savannah State University, GA, West Virginia State College, and Bluefield State College, WV. He has, also, worked as an engineer for Philips Corp., and Intel Corp.

J. MICHAEL JACOB
Mike Jacob, the McNelly Distinguished Professor, is an outstanding educator. Professor Jacob has won the Purdue Electrical Engineering Technology Department teaching award seven times, Purdue School of Technology teaching award three times as well as the Amoco Undergraduate Teaching Award (the top teaching award at Purdue University). He has published several internationally popular textbooks, and a variety of papers and conference presentations on the art of teaching. Mike has led over thirty teaching workshops. He has six years of industrial experience as a test engineer in the automotive and aerospace industries.