Is PSPICE Applicable Across the ECE Curriculum?

William C. Dillard¹

<u>Abstract</u>

The Student Version of PSPICE and its entry tool *Schematics* form a very popular circuit simulator suite for electrical engineering educators. Originally written for simulation of analog electronics, PSPICE now supports a broader range of component and system models including digital hardware, *s*-domain blocks and analog behavior models. These features make PSPICE simulations applicable to courses in digital electronics, communications, power and controls. (It is generally conceded that PSPICE is still poorly suited to electromagnetics.) This work investigates the breadth of PSPICE and how effectively it can be used in engineering education.

Introduction

At the 1973 Midwest Symposium on Circuit Theory, Dr. Donald O. Pederson, the "godfather of SPICE", made the first public disclosure of what would become the world's most popular IC simulator. Since then, SPICE (Simulation Program with Integrated Circuits Emphasis), has been used by thousands of universities and companies in teaching and designing electronics. In 1984, Microsim Corporation introduced PSPICE, a commercial product based on the Cal-Berkeley engine. Today, over 20 companies market SPICE-derived simulators. However, because Microsim has always provided a free Student Version, PSPICE has come to dominate the educational market.

Over the years, the PSPICE has been expanded to include digital hardware and analog behavior models that greatly enhance the applicability of PSPICE to a broader academic audience. In the following sections the effectiveness of PSPICE for instruction in digital circuitry, power distribution, controls applications and communication systems is considered.

Traditional PSPICE Simulations

Most universities employ PSPICE in basic circuits and electronics classes – a natural fit to the original intent of the software. As simulations for these applications are well documented [Azemi, 1, Irwin, 10, Dillard, 5, Herniter, 9, Tuinenga, 16, Koewen, 12], they will be treated only briefly.

Introductory Circuits and the Basics

Figures 1.a and b show a simple RLC network and the equivalent *Schematics* circuit, respectively. Three basic simulation types are available in PSPICE: bias point detail (DC analysis), ac sweep (small-signal, steady-state frequency-domain analysis) and transient (large-signal, time-domain analysis). In bias point details, all capacitors are treated as open-circuits, inductors are modeled as short-circuits and all sources are set to by their dc parameters. The resulting node voltages and branch currents are available both in the output file or can be displayed directly on the *Schematics* page as shown in Figure 1.b. For simple linear circuits, computational errors caused by roundoff and imprecise convergence are very small, and pen and paper calculations will be in close agreement.

¹ 200 Broun Hall ECE Department Auburn University, AL 36849 dillard@eng.auburn.edu

In ac sweeps, capacitors and inductors are modeled as frequency domain impedances and all sources are phasors. Small-signal models are used for nonlinear devices. So, a dc bias detail is automatically performed first to establish quiescent points. Any source defined by dc or time domain parameters only is set to zero. Ac sweeps are best suited to Bode plots as seen in Figure 1.c. Transient simulations use large-signal models for all parts and all sources are set by their time-related parameters only. Waveforms of interest are plotted versus time as shown in Figure 1.d.









(d)

Figure 1. The series RLC circuit diagram in (a) is entered as a *Schematics* diagram in (b) with simulations results for a dc bias point (b), an ac sweep (c) and a transient analysis (d).

Analog Electronics and Parameter Sweeping

Figure 2 shows a popular PSPICE academic example – the differential pair with active load [Jaeger, 11, Sedra, 15]. While the three basic simulation types are available, other features, such as parameter sweeping, enhance the investigative power of PSPICE. For example, how does the threshold voltage impact the gain and bandwidth. Combining an ac sweep with a parameteric sweep of the threshold voltage, a Bode plot is created for threshold voltages from 0.2 V to 1 V in 0.1 V steps. The result, shown in Figure 2b, indicates that increase the threshold increases the gain while degrading the bandwidth. Parameter sweeping is an excellent technique for isolating these kinds of engineering tradeoffs.

As demonstrated in this example, PSPICE supports parametric sweeps of voltage/current sources, temperature and for any model parameter for any device. In addition, the user may define global parameters through the PARAM part. These global parameters can be used to create algebraic relationships for part parameters such as resistance, initial capacitor voltage, and sinewave frequency, MOSFET width and length values. Global parameters may also be swept during analysis.



(a)



(b)

Figure 2. The Schematics drawing (a) and the simulation results (b) for the active load differential amplifier.

Non-Traditional PSPICE Simulations

As stated earlier, PSPICE can be used to investigate a wide variety of applications within the electrical engineering curriculum. In the section, the Student Version of PSPICE is employed to simulate digital electronics, three-phase power applications, communication systems and controls systems block diagrams.

Digital Electronics

The Student Version of PSPICE contains 134 digital primitive components (gates, MSI modules, RAM, ROM, flipflops) in addition to various stimulus and bus options that make it an excellent tool for the digital simulations undergraduates will encounter [Cheung, 2, Giesselmann, 7, Goody, 8]. In fact, digital rather than analog simulations have been used successfully to introduce sophomores to PSPICE [Dillard, 4]. Stimuli, gate delays and setup times can edited within *Schematics* or by vector table.

As an example, consider the ROM-based, 4-bit squaring circuit in Figure 3a. In the PSPICE schematic shown in Figure 3b, the ROM32Kx8breakout part emulates a 28C16A 2Kx8 ROM. The corresponding assembly file, listed in Figure 3c, can be assembled and the resulting HEX file is referenced by path and name within the ROM32Kx8 model as shown in Figure 3d. Figure 4 shows the timing diagram that results from a transient simulation.



Figure 3. A squaring circuit (a) can be simulated using the ROM32KX8break part (b). ROM data is specified in a .ASM file (c) and the corresponding .HEX file is referenced in the ROM32KX8break model (d).



Figure 4. Simulation results for the squaring circuit in Figure 3a verify the design. Delay through the ROMbreak32KX8 part is discernable at roughly 0.35 µs.

Power and Custom Parts

Since PSPICE is a circuit simulator, it can successfully simulate any power distribution system that is represented by standard components (sources, resistors, inductors, capacitors, etc.) whether single or polyphase, balanced or unbalanced. While specifications such as line voltage and line impedance easily transfer to PSPICE, load specified by power and power factor do not. One option is converting loads to equivalent RLC models by calculator or software and drawing the corresponding *Schematics* diagram. A much better solution is creating custom parts in PSPICE that have line voltage and power factor as editable attributes and perform the conversion to RLC inherently. While the procedure for creating custom parts is well documented, it is recommended that faculty or teaching assistants create the parts and make them available to their students [Dillard, 5, Rashid 13].

To demonstrate, custom parts were created for balanced three-phase delta and wye sources and loads (both capacitive and inductive) with standard power specifications as editable attributes. Source attributes are converted to three voltage sources while load attributes yield an equivalent RLC load model. Figure 5 shows a 4.6kVdistribution subsystem with two wye loads. Since voltages and currents are specified as phasors at 60 Hz, an ac sweep is performed at a single frequency. The IPRINT parts collect the line current phasors and prints them in the standard PSPICE output file. (Since the system is balanced, only one IPRINT part per load is required.)



Figure 5. Using custom parts allows simulation of power systems that are specified using common power distribution parameters rather than sources, resistors, and reactances.

Communications

Most communications systems are modeled using subsystems such as filters, mixers and FM signal generators. Conveniently, these blocks are part of the Analog Behavior Model library [Wilson, 17]. (Of course, discrete component parts such as transistors and diodes may be mixed with ABM parts in the same schematic.) The VCO in Figure 6a demonstrates the use of ABM parts to create communication subsystems. The EVALUE part is an arbitrary voltage controlled voltage source whose gain is defined by an equation rather than a single value. Here the PARAM part is used to define the variables that are used in the EVALUE expression. Table 1 describes the role of each variable in creating the VCO output. Resistor R1 is needed to keep the OUT+ pin from floating. Figure 6b shows simulation results for a modulation voltage between 0 V and 1 V modulating the output frequency linearly.

While this VCO is quite simple, many other communication system blocks will require at least 10 parts (ABM and otherwise). As in power simulations, custom parts can be created for these blocks. Unfortunately the Student Version of PSPICE is limited to 50 components, whether they are in custom parts or discrete, it is very likely that the component limit will be exceeded for any simulation of consequence [Rusek ,15]. For extensive communication systems simulations, the commercial version of PSPICE is recommended.





1k

f chan



(b)

Figure 6. An ABM implementation of a VCO (a) and the simulation results (b).

TABLE 1: Variable definitions for the VCO modeled in Figure 6.

Variable	Role
f_0	output frequency at zero modulating voltage (Hz)
N	channel counter
f_chan	channel spacing (Hz)

Controls

A variety of parts in the ABM library are well suited to controls simulations, including summing and difference blocks, Laplace transfer blocks, a collection of gain elements and lookup tables for arbitrary function approximations. These parts make PSPICE attractive for analog controls simulations, however, there are no parts that can be used conveniently for digital control systems [Detjen, 3]. When compared to MATLAB/Simulink, this is a major disadvantage.

Figures 7a and 7b show a control system that models the control of a radio astronomy antenna subjected to a disturbance, T_d [Dorf, 6]. At issue is the amplifier design, particularly the choice of k_a , for low output error. Specifications state that the output shall be less than 0.2 when the input is zero. Since this a steady state issue rather than transient, a dc sweep simulation is performed where k_a is the swept variable and plot the output versus k_a . The *Schematics* diagram and the simulation results are shown in Figures 8b and 9, respectively. Displaying the output on the log-log plot in Figure 8, the required value is easily found to be 74.3.





Figure 7. The block diagram (a) and *Schematics* diagram (b) for a radio astronomy antenna control system under disturbance T_d .



Figure 8. Simulation results for the antenna system in Figure 8a show that the parameter k_a must exceed 74.325 if the output error at zero input is to be less than 0.2.

Conclusions

A consideration of the Student Version of PSPICE as a simulation tool across the electrical engineering curriculum reveals that PSPICE is an excellent simulator for basic circuits, analog and digital electronics courses. A wide collection of commercial and generic parts are available without the need to create subcircuits or custom parts. If the commercial version is readily available, entire computer systems based on the 80X86 processor family can be simulated. This is an attractive alternative for departments whose curricula structure has no room for traditional wire-and-board laboratories. Also, a variety of parameteric sweep options are available that facilitate isolating trends and tradeoffs.

For power distribution systems, one may always convert the system specifications to primitives (sources, resistors, inductors and capacitors). However, creating custom parts is a better option. The power system specifications are the editable part attributes that are converted to equivalent RLC and source values in a subcircuit definition. Since the *Schematics* diagram can contain both primitive and custom parts, the system model is not limited to or by the custom parts.

Communication and control systems are best served by the Analog Behavioral Model library whose parts include general block such as gain, summing, integrators, Laplace transfer functions and arbitrary dependent sources. The primitives in the ABM library are well matched to standard analog control blocks so that subcircuits and custom parts are generally not required. There are however two significant limitations when using PSPICE for controls simulations. First, PSPICE can display frequency domain data in either a table or a Bode plot - there is no inherent Nyquist plot capability. Second, although digital control algorithms can be embedded into PSPICE [Detjen, 3], the process is complex and inappropriate to undergraduate study. This lack of built-in system blocks for discrete time control gives the Matlab/Simulink suite a clear advantage.

While a small collection of amplifiers and filters are available in the ABM, robust modeling of communication subsystems is best done through subciruits and custom parts. When simulating realistic scenarios, it is rather likely that the number of primitives will exceed the Student Version limitation of 50 parts per schematic. However, migrating to the commercial version will alleviate this problem.

References

- 1. Azemi, A. and Yaz, E.E.(1994) "PSPICE and MATLAB in Undergraduate and Graduate Electrical Engineering Courses," *Frontiers in Education Conference*, pp. 456–459, IEEE, New York.
- 2. Cheung W. N., (1999) "Digital Simulation of Electrical Systems to Reinforce Laboratory Teaching," *Transactions on Education*, vol. 42, no. 1, pp. 22 32, IEEE, New York.
- 3. Detjen, Dirk, Schroder, Stefan and De Doncker, Rik, (2003) "Embedding DSP Control Algorithms in PSpice," *IEEE Transactions on Education*, vol. 18, no. 1, pp. 294 300, IEEE, New York.
- 4. Dillard, William, (2001) "Effectively Incorporating a Hardware Experience into a Digital Electronics Service Course," *ASEE Annual Meeting*, ASEE, Montreal, CA.
- 5. Dillard, William and Irwin, J. David, (2002) "Basic Engineering Circuit Analysis Study Guide," J. Wiley and Sons, New York.
- 6. Dorf, Richard, (1967) "Modern Control Systems," Addison Wesley, New York.
- Giesselmann, M. G., (1999) "Using PSpice 8.0 to Teach Digital Logic," *IEEE Transactions on Education*, vol. 42, no. 4, pp. 356 356, IEEE, New York.
- 8. Goody, Roy, (2002) "OrCAD PSpice for Windows Volume III: Digital and Data Communications," Prentice Hall, New York.
- 9. Herniter, Marc, (1994) "Schematic Capture with PSpice," MacMillan Publishing, New York.
- 10. Irwin, J. David, "Basic Engineering Circuit Analysis." J. Wiley and Sons, New York.
- 11. Jaeger, Richard C., (2003) "Microelectronics Circuit Design," MacGraw Hill Publishing, New York.
- 12. Koewen, John, (1991) PSpice and Circuit Analysis," MacMillan Publishing, New York.
- 13. Rashid, M. H. and Al-Biyat, S. A., (1996) "Power electronics Laboratory Using PSpice," *Frontiers in Education Conference*, pp. 534 537, IEEE, New York.
- 14. Sedra, Adel and Smith, Kenneth (2003), "Microelectronic Circuits", Oxford Press, New York.
- 15. Rusek, Anthony and Oakley, Barbara, (2001) "PSpice Applications in the Teaching of Communications Electronics," *ASEE Annual Meeting*, New York.
- 16. Tuinenega, Paul, (1992) "Spice A Guide to Circuit Simulation and Analysis Using PSpice," Prentice Hall, New York.
- 17. Wilson, I. M., (1989) "Analog Behavioral Modeling Using PSpice," *Proceeds of the 32nd Midwest Symposium on Circuits and Systems*, pp. 981 984, IEEE, New York.

William C. Dillard

Mr. Dillard is a Ph.D. candidate in the Electrical and Computer Engineering Department at Auburn University. He holds M.S and B.S.E.E degrees from Auburn University as well. His present interests in educational research are teaching strategies that promote professionalism and career development, learning styles and innovative laboratories that crosscut the curriculum. His technical interests are in the areas of power electronics and the emergence of silicon integration in power electronics control systems.