Impact of Simulation Software in the
Engineering Technology Curriculum

Stanley J. Pisarski
University of Pittsburgh at Johnstown

Abstract

The University of Pittsburgh at Johnstown (UPJ) offers the Bachelor of Science degree in Civil, Electrical, and Mechanical Engineering Technology Many of the courses offered in UPJ's Engineering Technology Program rely on laboratory experiments to supplement the lectures. Although there is no substitute for the experience that a laboratory environment provides, various software packages allow the user to simulate and predict with great accuracy how a system will react under actual laboratory conditions.

The Electrical Engineering Technology Department (EET) at UPJ utilizes several software simulation programs for supplementing laboratory work in the areas of circuits, electronics, digital systems, industrial electronics, digital signal processing, and control systems. Software packages such as PSpice, Logic Works, MATLAB and Simulink are heavily used to verify manual calculations and laboratory results. Additionally, these packages are used to solve theoretical problems which would be too complicated to solve by other means.

This paper provides examples of current engineering simulation software usage in the EET curriculum at UPJ, and identifies the ramifications of these experiences in the courses, student's college education and beyond.

I. Introduction

The objective of this paper is to provide examples of circuits and systems that the electrical engineering technology students simulate with various software simulation packages. Specific courses have been chosen to examine the effect that the software has on student learning of circuit analysis and design concepts.

Software simulation has been used in the electrical engineering technology program at UPJ since the early 1970's. Early circuit software was executed on an IBM 1130 mainframe computer that relied on hand-coded information for the circuit, punched computer cards, and submission of the card deck to the computer operator for the initial run. The next day the printed numerical results were obtained from the computer center. Advances in computer technology in the areas of computer software and hardware have paved the road for instant results for circuit simulations. The new computer software packages stimulate the senses with colorful, accurate circuit waveforms in graphical and tabular formats. No longer must the user of this software enter hand-coded data for circuits or systems, but a nudge of the computer mouse is all it takes to create a

professional-looking schematic of a circuit or a block diagram of a system. The simulation is fast and the results are many times better than hand calculations could provide.

II. Electrical Engineering Technology Courses at UPJ

Students enrolled in the electrical engineering technology program at UPJ take a variety of courses in mathematics, physics, chemistry, humanities, and social sciences in addition to the engineering technology courses. Currently each electrical engineering technology course that the students take has a 3 hour laboratory as a corequisite. Many of these courses stress analysis and design as the primary focus of the work completed. Since these students must assemble, test and document a variety of pre-tested and untested designs, their success in the laboratory and in the classroom relies on accurate information about the circuit or system's performance. Circuit simulation is the key to ensuring that the circuit or system will perform as expected. Limitations of the software often becomes evident when real data is compared to the software simulation after the experiment has been completed.

The list of the electrical engineering technology courses that utilize software simulation packages of various types begins with Circuits I and II, Electronics I and II, Digital Systems, Electronic Communications, and ends with Control Systems as the required courses. Elective courses in the program that use similar software simulation packages are Microcontrollers, Digital Signal Processing, Industrial Electronics, and Digital Control Systems. Commercial software simulation packages such as PSpice, MATLAB, Simulink, Logic Works and AVSIM are heavily used while software such as Electronics Workbench, and various load flow and short circuit packages are simply introduced to the students as alternatives.

III. Circuits I

Circuits I is the introductory circuits course that UPJ electrical engineering technology students take in their sophomore year. The course topics cover basic ac and dc circuit theory along with magnetic circuits, transformers and three-phase power. Main circuit analysis techniques such as Mesh, Nodal, Superposition, Thevenin's, Norton's and Maximum Power Transfer Theorem are utilized in the analysis of various circuit configurations. Transient analysis with dc sources is briefly covered as well. A laboratory course, where students assemble, test and document the measurements and calculations performed on the circuits, closely mirrors the topics covered in the lecture sections. PSpice is the main software package that is used in this course [1]. Since many authors have integrated PSpice into their text books and provided examples, it seems natural to use the software to solve both easy and complex problems.

The students begin by hand-coding various simple circuit configurations into PSpice circuit file format. A text editor is used to enter the program and PSpice is invoked to simulate the results. Mixed reactions evolve from the students. Some students feel the exercise is simply additional work while others realize the real power behind using software simulators lies not only in their ability to solve simple circuit configurations but their ability to solve large complicated circuits as well. Many students express the feeling that there must be a better way to enter the circuit information without the tedious job of hand-coding and entering the circuit information. They

soon realize that the schematic capture front-end software alleviates the problems associated with hand-coding the circuit. Students that expressed feelings about the additional work before now seem to enjoy entering and solving the circuits with the schematic capture feature.

The UPJ electrical engineering students assemble and test approximately thirty two different circuit configurations in the laboratory throughout the course of the school term. Each circuit is simulated using both hand-coded and schematic capture input outside the laboratory period before any measurements are obtained. The results of these simulations are compared with actual results obtained from the experiments. Again, mixed reactions from the students about the results occur. Nominal values of the components are used in the circuit simulations, but actual components with tolerances affect the circuit measurements. The students now see the difference between simulation results and effects due to the actual components values.

IV. Circuits II

Circuits II concentrates on transient response of circuits due to forcing function inputs and frequency response characteristics of circuit networks. Differential equation solutions and Laplace transform models are introduced early in the term. Simple and complex networks are modeled and solved in the frequency domain and are then transformed into the time domain. The students become so involved in reaching a solution to the problem through mathematical manipulations that they lose sight of the big picture--determining what is actually happening to the input signal as it passes through a circuit. Rarely do the students plot the results of the equation for the output signal in the time domain. PSpice is initially used to simulate these networks and MATLAB is introduced later in the term. By this time the students are proficient in using PSpice for circuit simulations, but they are reluctant to go back to hand-coding and writing programs in MATLAB format.

MATLAB is a software package for high-performance numerical computation and visualization. It provides an interactive environment with hundreds of built-in functions for technical computation, graphics, and animation. Special optional toolboxes contain functions written for special applications such as digital signal processing, symbolic computation and control system design [2]. The students soon realize that MATLAB is easy to use and provides high-quality graphical outputs that are now a very important factor in understanding what happened to the signal as it passes through several circuit networks. MATLAB allows the students to write their own programs to solve rather unique problems that would otherwise be difficult or impossible to solve with PSpice. Once the students gain confidence in using MATLAB software, they begin using it not only for specific assignments, but it is used to check the results of their homework and verify the results of data collected in the laboratory.

The detracting feature of MATLAB is the method of writing and entering information into the solver engine--hand-coding. Simulink is a rather new software package that integrates with the MATLAB software package. It is a graphical-user-interface to the MATLAB solver engine. It does for MATLAB what schematic capture does for PSpice; it allows the user to use the mouse to choose input devices, linear and non-linear function blocks and output devices to create complex system level block diagrams that are modeled in the frequency domain and provides

graphical outputs in the time domain. The students enjoy working with Simulink after they have had the opportunity to solve complex problems utilizing conventional Laplace and inverse Laplace transform techniques.

V. Electronics

The required electronics course sequence that the UPJ students follow is Electronics I and Electronics II with Industrial Electronics offered as a technical elective. These courses rely heavily on PSpice circuit simulation, but Electronics Workbench is introduced early in the Electronics I course as an alternative software simulation package. Since both packages offer schematic capture data input, learning Electronics Workbench is an easy transition [3]. Both are ideal packages and the students react by using whatever is available at the time.

PSpice seems to be the preferred software tool by the students only because the they are more familiar with it because of prior use. Since circuit design is stressed more than analysis in these classes compared to Circuits I and Circuits II, that deal primarily with analysis, the students use the software to experiment with the changes in circuit performance due to component changes and are now able to optimize circuit performance. The students now truly see the power of using software simulation packages since they can observe the changes in the output of their circuits as they modify component parameters. The software helps the students reinforce the lecture material learned in the classroom and link what they observe on the computer screen with the results obtained in the laboratory.

Many of the circuits that are simulated fall into categories such as zener diode power supplies, discrete transistor amplifiers, operational amplifiers, active and passive filters, three-phase rectifiers, silicon-controlled rectifier converters and inverters, buck and boost converters, and dc and ac motor drive systems. Students that simulate these types of circuits seem to have a better intuitive feeling of what is happening with the circuit than in cases where simulation was not performed. Over the past ten years the circuits that have been simulated in several courses have been rotated periodically in an attempt to minimize the possibility of plagiarized work. In cases where students simulated the circuits before taking their quizzes and exams on the subject material, there is clearly a correlation of increased knowledge and understanding of the subject material. Circuit simulation complements the laboratory environment, but cannot replace it entirely. Skills acquired in the laboratory allow the students to examine circuits from a different viewpoint.

VI. Digital Systems

Digital Systems deals with Boolean Algebra, Karnaugh Maps, logic gates, and combinational and sequential circuits. Designs of these circuits begin with a detailed description of the input states of the system and the resulting output states. Students are taught conventional techniques for circuit minimization and are quickly introduced to system truth tables. The laboratory experience for this course requires the students to design, assemble and test digital logic circuits that at times contain up to ten 14-pin integrated circuit packages. The number of wires needed

to interconnect this system is generally greater than the number of packages times the number of pins or greater than 140 individual connections for this case.

A variety of problems plague the students every step of the way. These problems include incorrect circuit designs, defective integrated circuits, and missing, poor or incorrect circuit connections. In addition, the time constraint imposed on the student to provide correctly operating circuits many times leads to student frustration. Software simulation provides an alternative, but not a complete replacement, for physical assembly and testing of digital circuitry. Students can concentrate on understanding the design, operation, and timing problems associated digital circuits when they are simulated, without having to worry about many of the problems identified above.

Although PSpice and Electronics Workbench contain models of digital components and work well for digital circuit simulations, the electrical engineering technology students at UPJ use the Logic Works digital circuit simulation software package for all of their digital circuit simulation needs. Students enjoy simulating digital circuits that allow multi-channel timing diagrams to be easily displayed on one screen. The same amount of information can only come about by using expensive logic analyzers on physical circuits. The ability to connect light emitting diodes and 7-segment displays to counter circuits is a tremendous benefit that allows the students to clearly observe sequential circuit operations. There is definitely a savings in time required to assemble, test and document laboratory results since these activities are performed on the computer screen instead of on a hand-wired breadboard.

VII. Control Systems

Control Systems involves the study of simple and complex networks of interconnected circuits or systems in open or closed loop configurations. Analysis techniques include Laplace and inverse Laplace transforms, State Variable, and Root-Locus. System block diagrams are generally represented in the frequency domain but the resulting overall system transfer function and output signals are represented graphically as responses to the inputs in the time domain. Most of the analysis and design techniques are mathematically complicated and manually performed hand calculations are both tedious and error-prone. Software simulations such as MATLAB and Simulink are the perfect combination of tools for this kind of analysis and the graphical input of system parameters allows the user freedom to experiment with poles, zeros and system gain to optimize system performance. Linear and non-linear transfer function blocks can easily be added to the system to account for non-linearities and the results can be instantly observed. Simulink provides the graphical output serious control systems engineers require to solve complex control systems networks.

The electrical engineering technology students at UPJ are given various control system configurations to solve manually, solve again with MATLAB by writing MATLAB code and then solve again with Simulink. Even after the system was solved once manually and again with MATLAB, the students were still missing the concept of system stability with closed loop feedback due to system gain. After the students simulated the system the third time with Simulink, which provided graphical output of the response, were they able to understand the

concepts and could easily modify the system to experiment with different system gains. The students agree that Simulink is the best tool for use in control system design and analysis. Response of complex systems is difficult to predict and correspondingly difficult to explain and understand, but Simulink makes that job easy. Simulink is especially useful in designing compensation networks to improve system stability while increasing overall transient response. Different compensation networks such as phase-lead, phase lag, lead-lag, and pole-zero can be inserted into the network to observe their effect on system stability, system gain and transient response.

VIII. Conclusion

Circuit simulation is a very powerful method for solving many types of simple and complicated circuits and electronic systems. Today's software simulation packages offer easy-to-use graphical-user-interfaces and provides fantastic graphical outputs that rival the best electronic bench equipment. Virtual instruments are cheap to own and operate and they never need calibration and service. Students enjoy simulating circuits without having to put up with the hassles of the physical networks and the vast number of interconnecting conductors that link everything together. This encourages students to experiment with circuit components, identify the component's function and link each function to the characteristics of the circuit's response. The more time spent studying a topic from different viewpoints with various techniques, the better the topic will be understood. The electrical engineering technology students at UPJ look to the future when virtual reality will be combined with circuit simulation allowing more of the human senses to experience the joy of simulation.

References

1. Tuinenga, Paul W., "Spice A Guide to Circuit Simulation & Analysis Using Spice", 1992, Prentice- Hall, Inc., Englewood Cliffs, New Jersey

2. Pratap, Rudra, "Getting Started with MATLAB", 1996, Harcourt Brace & Company, Orlando, Florida

3. Rashid, Muhammad H., "Electronics Circuit Design Using ELECTRONICS WORKBENCH", 1998, PWS Publishing Company, Boston, Massachusetts

STANLEY J. PISARSKI
Stanley J. Pisarski has been an instructor of EET at the University of Pittsburgh at Johnstown for 19 years. He received a BSEET in 1977 and has worked as a project engineer for Robicon Corporation in Pittsburgh, PA, consulting engineer for Ocenco Inc., in Blairsville, PA, and a research and design engineer for K. H. Controls Inc., in Blairsville, PA. He is a Licensed Professional Engineer in Pennsylvania.