ROLE OF SIMULATION SOFTWARE IN ENHANCING THE QUALITY OF ELECTRONICS TEACHING

G. S. Shahane and P. A. Lohar
Department of Electronics, DBF Dayanand College of Arts and Science, Solapur, MS, India

Abstract:
Computer aided simulation is an integral part of science and engineering technology. Simulation provides verification of the basic theory, understanding the basic principles, greater attention to the theoretical limitations, and application of logical analysis to solve real-world problems. The objective of this paper is to discuss simulation software packages used in Electronics for better understanding of the subject, with special emphasis on PSPICE for analog and digital circuits. Studies showed that students who used simulation prior to conducting actual experiments performed better than the students who conducted the laboratory experiments without conducting simulation first.

Keywords: Simulations, Interactive learning, Electronics education

Introduction
In recent years, the field of science and engineering have become very dynamic due to recent advances in computer technology. These advances have resulted in number of computer programs to solve traditional and novel problems. These programs use the computer’s increased computational capabilities and assist in the design, development, and control of complex systems. Computer-based visualized simulation learning has covered almost all subjects of science education (Gordin and Pea 1995; Jensen et al. 2002; Chen et al. 2011). Review of these research findings revealed that learning performance can be enhanced if a visualized learning environment promotes learner interactions and gives them opportunities for manipulation (Gordin and Pea 1995; Jensen et al. 2002). A common problem faced by learners of electronics is being unable to fully understand the basic concepts and system responses predicted by theoretical models. This often results in learners being unable to see the link between models and actual circuits. The electronics graduates must be proficient in the use of engineering and scientific equipment, conducting experiments, collecting data, and effectively presenting the results. Furthermore, these graduates must be well-trained in designing and developing analog electronic circuits, digital systems, microprocessors and microcontrollers, communication and networking, power systems, PLC and virtual instrumentation, etc. One cost-effective way of achieving this is through the use of simulation software programs. The use of computer simulation by learners allows them to readily manipulate parameters and observe the resulting changes in a given phenomenon, which helps the process of higher-level reasoning.

Computer Aided Simulation
Electrical and electronic circuit design requires cost-effective and accurate methods for evaluating circuit performance. It is found that computer aided simulation has great potential as a supplementary tool, with simulation as a medium helping learners to get missing links between theories and actual electronics processes (Ronen and Eliahu 2000). Electronic Simulation software utilizes mathematical models to replicate the behavior of an actual electronic device or circuit. Essentially, it is a software program that converts your computer into a fully functioning virtual electronics laboratory. A number of simulation software packages are available for these purposes. These software packages can be used for number of applications, starting from simulation of simple
electric circuits to complex tasks such as networking and computer circuits with the ultimate objective of providing illustrations of concepts that are not easily visualized and difficult to understand. Simulators are also used as an adjunct to and, in some cases such as distance learning courses, as a substitute for actual laboratory experiments. For simulation of electronic circuits (analog and digital), packages like PSPICE, Proteus, MultiSim, VisSim, Logic Works, Design Works, MatLab are widely used. Softwares like Keil, Pinnacle, Prog-Studio, MPLAB, AVR studio for microcontrollers and LABVIEW for virtual Instrumentation are widely used by engineering and technology programs at different institutions (Swain et al. 2008).

**About PSPICE**

PSPICE (PC Simulation Program with Integrated Circuit Emphasis) is an exceptional simulation software package that can be used to efficiently solve simple as well as complicated circuit designs. PSPICE is the version of the SPICE software developed by MicroSim Company. The software allows you to construct a virtual circuit using a schematic, simulate it using various solving techniques and analyze the results. The process of simulating a circuit has five major steps. First, a circuit is drawn by inserting various components from the directory, arranging the parts and connecting them with wires. Next, the names, values and other attributes of the parts are modified. The schematic file is then saved and checked for the errors. The outputs are then viewed and analyzed.

To get an idea of PSPICE, consider an example of transistor CE amplifier circuit as shown in Figure 1. This circuit consists of transistor, resistors, capacitors, dc voltage source and the sinusoidal voltage source.

![Figure 1: Circuit diagram of transistor CE amplifier](image1)

To design this circuit first a new project is created. Different parts can be selected from the part browser window and placed by clicking on the “Select Part” icon. Figure 2 shows a part browser window.

![Figure 2: Part Browser Dialog Box](image2)

Various components are placed and interconnected according to the circuit diagram. Names, values and other attributes of the components are modified as per requirement. Figure 3 shows the schematic of the circuit after connecting all parts.

![Figure 3: Schematic of transistor CE amplifier](image3)

The schematic is then saved and compiled for errors. Finally, outputs are displayed. Figure 4 shows the frequency response of the CE amplifier. Further, the effect of coupling capacitors, load resistance, biasing network can be very easily studied just by changing the values of the respective components.
It is observed that students who used simulation prior to conducting actual experiments performed better than the students who conducted the laboratory experiments without conducting simulation first. This visual and interactive mode of operation is very helpful in understanding the subject in detail. In short, a computer-aided simulation allows the designed system to be simulated so that the expected circuit behavior can be verified under specific operating conditions, any design errors can be identified and the system performance can be modified by fine-tuning relevant parts of the design. Hence, mistakes can be avoided well before the final hardware implementation of the circuit. It is seen that though the simulation cannot replace the physical hands-on experience, but it can enhance the teaching and learning experience.

**Conclusion:**

Computer-aided simulation allows the designed system to be simulated so that the expected circuit behavior can be verified under specific operating conditions, any design errors can be identified and the system performance can be modified by fine-tuning relevant parts of the design. This visual and interactive mode of operation is very helpful in understanding the subject in detail. Also, mistakes can be avoided well before the final hardware implementation of the circuit.

**References:**


