AC 2007-1224: BIOMEDICAL ENGINEERING VIRTUAL CIRCUIT SIMULATION LABORATORIES

Robert Szlavik, California Polytechnic State University

Biomedical Engineering Virtual Circuit Simulation Laboratories

Abstract

Circuit simulators, such as SPICE (Simulation Program with Integrated Circuit Emphasis) are useful tools that can enhance the educational experience of students in many subject areas within a biomedical engineering curriculum. Courses on biomedical instrumentation are venues for which virtual laboratory experiments, using circuit simulators, can be readily developed. The instructor can use the circuit simulation platform to illustrate relatively complex concepts, such as differential amplification, which have wide applicability to biomedical instrumentation.

More advanced courses that focus on the physiology of excitable cells or neural modeling and simulation are also venues for which circuit simulators may be applied to study the dynamics of related physiological models such as the Hodgkin-Huxley model. The equivalent circuit paradigm provides the student with an alternative to developing an understanding of complex physiological models. Application of SPICE based circuit simulators for neural modeling and simulation require the development of excitable membrane equivalent circuit models. Such models have been implemented by the author using SPICE primitive circuit elements in the form of a netlist sub-circuit. More advanced approaches have involved the implementation of neuron models using the SPICE code model paradigm. This alternative approach facilitates the implementation of the neuron model whereby it can be referenced from within a SPICE netlist program in the same way as any other device model would be referenced.

At California Polytechnic State University (CalPoly), students at both the undergraduate and graduate levels are exposed to circuit simulation tools that are integrated into the course content as virtual labs in the biomedical engineering instrumentation course. More advanced courses at CalPoly in neural modeling and simulation also make use of the SPICE circuit simulation platform.

SPICE as a Generic Virtual Laboratory Platform

SPICE is an acronym that stands for Simulation Program with Integrated Circuit Emphasis. The initial version of the SPICE program was developed in the late 1960s for applications related to integrated circuit design. Several contemporary versions exist, in both the commercial and public domains, with different graphical based programs for developing circuit simulations. A wide range of graphical post processor programs are implemented with the different distributions for visualizing simulation results.

Although the original intent of the designers was to develop a CAD tool for circuit simulation, the program, and its more modern incarnations, may be viewed as a highly efficient nonlinear differential equation solver that can be applied to simulate a wide variety of physical systems. The only practical constraint is that the system under study must have an analogous equivalent circuit representation. For a physical system that is not electrical in nature, the current and voltage terminal variables in the equivalent circuit model are interpreted in the context of the appropriate units for the model. Many physical systems have equivalent circuit analogs such as

mechanical systems or control systems[1]. Simulations utilizing SPICE have been developed to study the interaction between conduction band electron density and photon concentration in semiconductor diode lasers[2].

From the perspective of biomedical engineering education, SPICE may be used to simulate common analog electronic instrumentation blocks, such as filters or differential amplifiers. More involved simulations have been developed that utilize SPICE primitive circuit elements to implement Hodgkin-Huxley active membrane models[3,4]. These models allow for simulation of hybrid biological and synthetic electronic circuits under a single unified simulation platform. An example of the type of simulation that is possible using a SPICE platform is the filtering effects associated with a micropipette electrode used to detect intracellular action potentials from an electrically small neuron. Students can utilize the simulator to investigate the use of a negative input capacitance amplifier to offset this effect[5]. More recently, the implementation of the code model paradigm has allowed for development of biological neuron models that are implemented within a SPICE netlist in an analogous manner to other analog electronic devices such as bipolar junction transistors or diodes.

SPICE Based Virtual Biomedical Instrumentation Laboratories

In the undergraduate biomedical engineering curriculum at CalPoly, a course in biomedical instrumentation is available to upper division undergraduate students and graduate students. The course (BMED 440) consists of a lecture component as well as a laboratory component.

In the laboratory component, extensive use is made of SPICE based circuit simulators to illustrate the behavior of specific analog electronic circuit building blocks. While simplified representations of the circuits are utilized in the simulation studies, the concepts that are inherent to the behavior of the building blocks are widely applicable in biomedical electronic instrumentation.

Concepts such as differential amplification and the effect that the amplifier has on the common mode versus the differential mode components of a signal can be easily studied using SPICE. In one of the virtual laboratory experiments in the BMED 440 course, students implement the simulation of a simple differential amplifier that is constructed from discrete operational amplifier models.

SPICE Based Virtual Electrophysiology Laboratories

More advanced virtual lab experiments that utilize Hodgkin-Huxley neuron models are carried out by students in the BMED 440 course as well as an upper division physiological simulation and modeling course (BMED 430).

Extensive use of the SPICE sub-circuit definition capabilities are made to develop virtual experiments such as is illustrated in Figure 1., involving the injection of a stimulus current pulse train into an electrically small biological neuron such as a leech Retzius Cell[4].
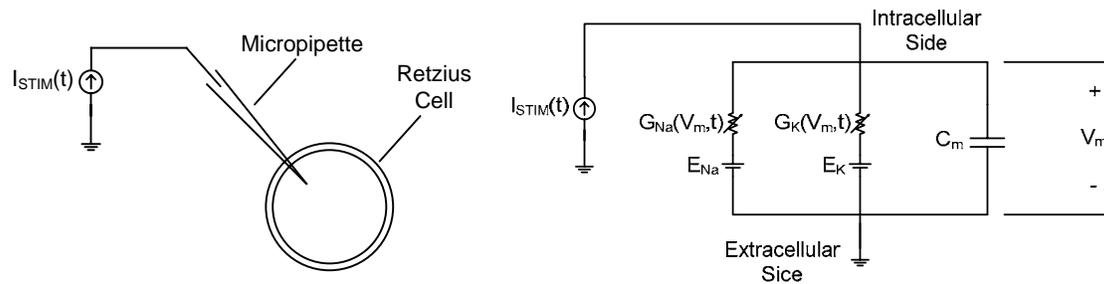
Figure 1. Simple SPICE equivalent circuit model of a Retzius Cell that is excited with a stimulus current pulse train $I_{STIM}(t)$. The nonlinear conductances in the circuit are implemented using a SPICE sub-circuit. Details of the Hodgkin-Huxley active membrane equivalent circuit model are outlined in Szlavik et al.[4].

Simulation studies such as these allow students to easily vary key simulation parameters such as the characteristics of the time dependent stimulus current pulse train (i.e. amplitude, pulse width) and see the impact of these changes on the target cell. More advanced simulations that investigate cell membrane capacitive effects are also possible.

Conclusions and Specific Implications Related to Biomedical Engineering Laboratories

The SPICE simulation environment allows for implementation of a virtual simulation laboratory that is applicable to a wide range of physical systems. SPICE is equally versatile at simulating the dynamic characteristics of analog electronic circuits as it is at simulating the behavior of a wide variety of physical and physiological systems. This ability makes it a useful tool in biomedical engineering laboratory education at both the undergraduate and graduate levels.

The availability of various physiological models, such as the Hodgkin-Huxley active membrane, facilitates the use of the simulator in advanced biomedical engineering courses that focus on physiology and modeling. The SPICE simulator, as a tool, is well suited to electrophysiology related simulations since it is possible to simulate hybrid circuits with both biological and synthetic electronic components.

Bibliography

1. N. K. Sinha, *Control Systems*, New York: Holt, Rinehart and Winston, 1988.
2. R. S. Tucker "Large-signal circuit model for simulation of injection-laser modulation dynamics," *IEE Proceedings*, Part I, vol. 128, no. 5, pp. 180-184, 1981.
3. B. Bunow, I. Segev, and J. Fleshman, "Modeling the electrical behavior of anatomically complex neurons using a network analysis program: excitable membrane," *Biol. Cybern.*, vol. 53, pp. 41-56, 1985.
4. R. B. Szlavik, A. K. Bhuiyan, A. Carver, and F. Jenkins, "Neural-electronic inhibition simulated with a neuron model implemented in SPICE," *IEEE Transactions on Neural Systems and Rehabilitation Engineering*, vol. 14, no. 1, March, 2006.
5. J. G. Webster, *Medical Instrumentation, Application and Design*, John Wiley & Sons, 1998.