Extended Abstract with Poster

Using SPICE Schematics to Deliver the Essence of Microelectronics

Mohammed Arif I. Mahmood
Department of Electrical Engineering

Samir M. Iqbal
Department of Electrical Engineering and Department of Bioengineering
University of Texas at Arlington and University of Texas Southwestern Medical Center at Dallas

Abstract (Extended)

Microelectronics is an integral part of our fast-paced lives. Circuit designs for cutting edge devices change every day to accommodate newer innovations. For a novice designer, physical implementation of even a basic circuit requires certain expertise as well as expensive lab equipments. Fortunately, understanding the fundamental working principles of electrical circuits, starting from designing and implementing those to get some decent outputs can be done outside lab environment. Using ubiquitous computing resources it is possible to simulate simple to fairly complicated circuits. Several software are available for simulations and most of these are fairly intuitive. SPICE, Electronic Workbench, Multisim are to name a few. Softwares to carry out simulations (SPICE, NI Multisim, etc.) are used in industry as well. Most of these tools have student versions available as freeware. PSPICE is commercial software that is extensively used in industry. A student version of PSPICE is available to perform basic electronic circuit design. The freeware version allows limited number of nodes to simulate, but this is good enough to implement fairly complicated circuits. DC/AC circuits with basic understanding of voltage divider, current splitting, and Kirchhoff's laws can be constructed in SPICE environment. The working principles can be understood by investigating the outputs using probes at different nodes and branches showing the parameters of the circuit. The potential of this is not just limited here. Interested students can take their enthusiasm farther and learn advanced topics like basic high-pass, low-pass, band-pass filter design without having to do a formal engineering course. This can lead to designing of amplifiers with off the shelf components like operational amplifiers. The students can get a flavor of the analog world while understanding the basic principles of the devices used on day to day basis.

SPICE stands for *Simulation Program for Integrated Circuits Emphasis*. This simulation tool was developed by Lawrence Nagel and coworkers as a circuit simulator at Electrical Engineering and Computer Science Department of the University of California, Berkeley and was announced on April 12, 1973[1]. From there it flourished and newer device models have been added on regular basis. Many other specialized simulators have been designed subsequently; however all other design tools used for such simulations still roughly follow the format of original SPICE.

Core SPICE program is text command based. A much user friendly PSPICE was initially developed by Microsim Corporation in San Jose, CA, which was later acquired by Cadence Design System, Inc. PSPICE provides a user-friendly graphical interface for circuit design. A student version with certain number of maximum nodes is available for free that can be used by students as learning tool.

SPICE can simulate circuit operations using basic components like resistors, capacitors etc. Included library provides a pool of component models that can be built into circuits. Enthusiast and hobbyists can design and modify circuits, and test output stability before implementing with physical components. At high school level, basic circuit elements are usually introduced whereas physical implementation requires lab facilities as well as expert supervision. Introduction to SPICE can give students an easy and at-home access to the amazing world of electrical engineering without going through the cost, infrastructure and potential electrical hazards. This can bring significant amount of savings in terms of time and money. SPICE can give potential engineering students deep insights and confidence before entering to college environment. SPICE is a well documented software package with numerous, well explained and easily accessible tutorials, both in written as well as in video format [2-4]. The graphical user interface of SPICE Schematics is very intuitive and the procedures resemble closely to the real lab situations. Hence it is easy for any beginner to familiarize with the environment.

The basic steps for a successful simulation are illustrated in Figure 1. Like any design problem, a few error-corrections, adjustments of parameters or even modification of the initial circuit diagram may be required. Although not very difficult, the text-based input method can be a little intimidating for a beginner and hence it is not focused in this paper.

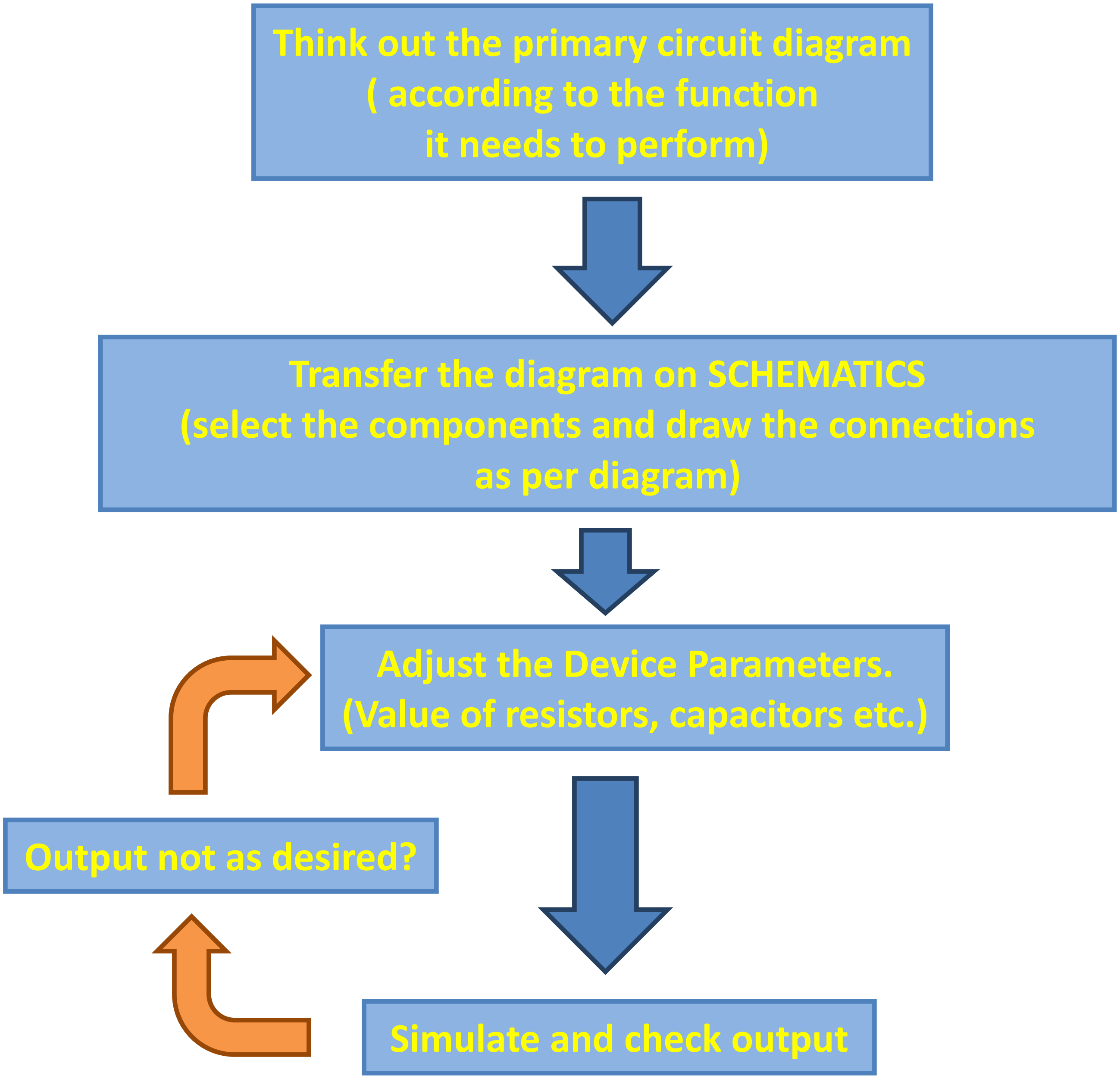


Figure 1: Basic Steps to setup a SPICE simulation.

The graphical user interface (GUI) of SPICE for circuit simulation is called SCHEMATICS. It gives a drawing area for circuit design and implementation. One can choose the components from the given pool and place it on the drawing area by selection. Components are well arranged and categorized in the SPICE library. A large number of component models are available in the default pool, ranging from resistors to commercially available integrated circuits (ICs). These models are usually provided by the component manufacturers. Dependant sources with user customizable parameters are available for advanced users and can be used towards designing modular circuits as well as device models [5, 6]. Connections between the circuit elements can be established using simple mouse click and draw. Circuit parameters can be changed with easy interface and hence provide flexibility of design and modification.

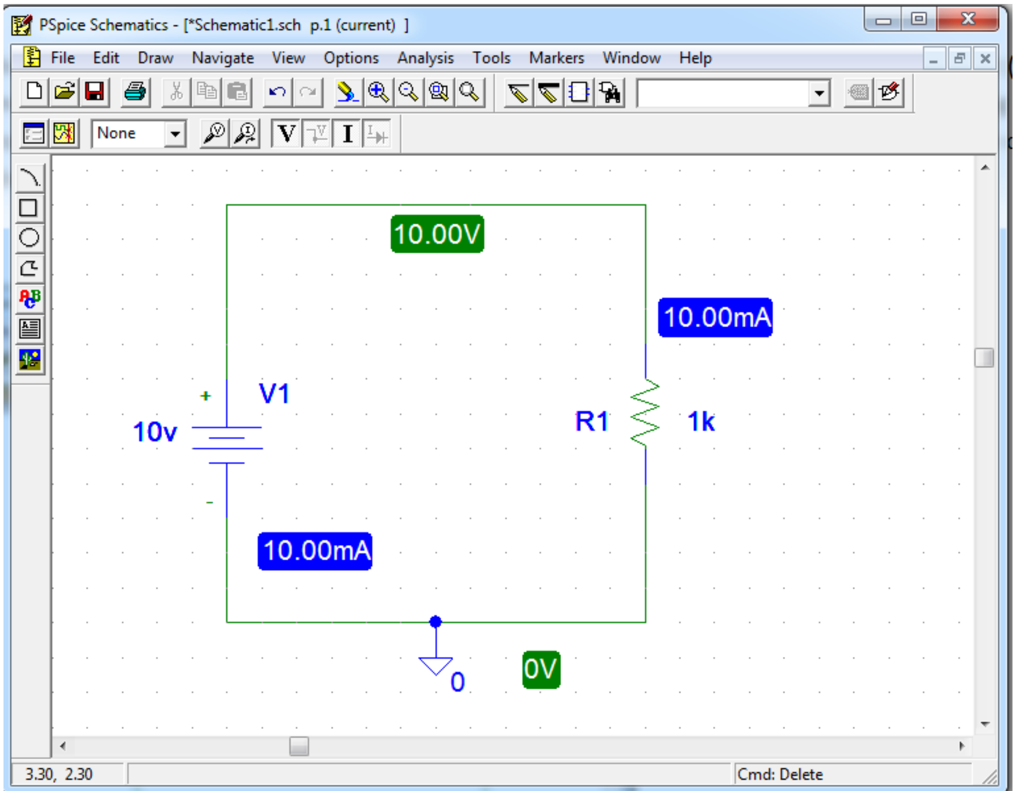
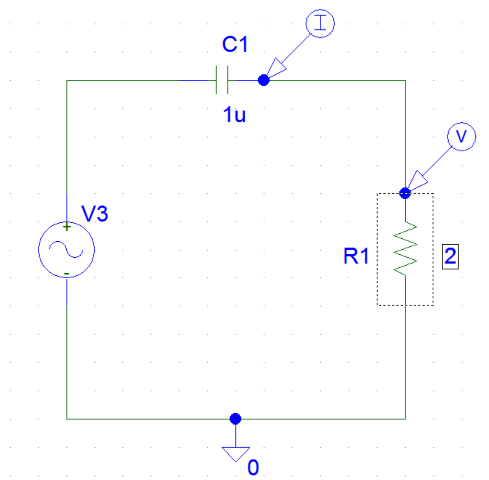
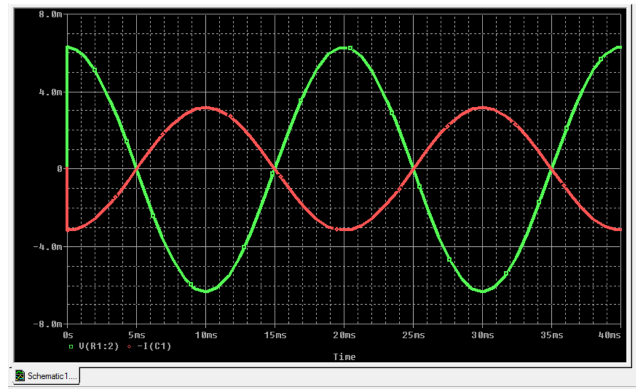


*Figure 2: Drawing space of the SCHEMATICS. Well laid out buttons provide easy access to numerous features from selection of components to one click simulation. Complex functions are also provided for advanced users. In the above figure, a 10 Volt DC source is shown connected to a 1 K**Ω** resistor. In SPICE, a reference voltage needs to be defined and hence placed here at the bottom. After simulation, the output voltage at different nodes and the current flowing in the branch (in this case 10 mA) are displayed with proper units. The outputs are similar to the result one would see using physical Voltmeter or Ammeter. One can easily observe from the output quantities how voltage and current follows Ohm's law.*

Simple DC circuits can be constructed using constant voltage sources and resistors (Figure 2). After simulation one can observe Ohm's law relationship between voltage and current in the circuit elements. Ohm’s law is the foundation of electrical engineering. The series and parallel combination principles of the electrical components can be investigated. One can also observe the voltage divider law or current divider law by simple modifications of the circuits. This gives students an intuitive idea of how electric current and potential works in the circuit. Slightly advanced users can extend these ideas using dependant sources.



(a)



(b)

Figure 3: (a) A simple AC circuit connected with a capacitor and a resistor. The voltage and current "probes" are connected to the circuit. The output voltage and current are measured at these points. (b) Shows the voltage and current in the output window. The Green and Red graphs represent voltage and current, respectively. The phase difference of a typical AC circuit is noticeable in the output.This probe window closely mimics the expensive oscilloscopes commonly used in the laboratories. The output parameters can also be programmed to find, for example, voltage difference between two specific nodes etc.

AC sources are ubiquitous in our everyday life and sinusoidal AC sources are provided within the SPICE component library to manipulate as desired. Voltage amplitude, phase and frequencies of these sources can be adjusted and optimized accordingly. Root mean square (RMS) output, as well as time varying outputs can be investigated by putting "probes" at different points in the circuit (Figure 3). Probes facilitate output display in graphical format. Features like AC sweep, where amplitudes and frequencies of the sources can be varied smoothly over a range, make it easy to check device performance and optimize the design. For example, using this feature on can investigate the working principle of the very important class of circuits like "Filters". One can investigate the output with varying parameters and pick the most suitable one for the design. These features are in fact much easier to implement on SPICE environment than in physical labs, facilitating faster design and optimization of time. This alleviates need for expensive oscilloscopes, frequency generators, power supplies, etc.

A vast range of analog electronic circuit components are found in the SPICE library including diodes, bipolar junction transistors (BJTs), or metal oxide field effect transistors (MOSFETs). For curious students it is easy to build simple electronic circuits like rectifiers using diodes for AC-DC conversion and check the output as desired. Without delving into the device theory and based on the working principles one can design many such useful and everyday devices. Experiment like this with physical devices involve electrical hazard which is avoided in SPICE environment without compromising the design experience. Accomplishments like these can certainly encourage students towards pursuing an Engineering degree while making them confident towards working in a physical lab.

Many commercially available and highly customizable ICs are included in the SPICE library. For advanced users, this will provide a way towards designing many interesting circuit implementations (Figure 4). Hobbyists have been building such circuits for long and SPICE platforms can immensely increase the flexibility of their design and experiments. For example, SPICE includes several commercial versions of operational amplifiers (Op-Amp). These are fairly simple devices but can be manipulated to design a range of circuits with very useful functions like amplifiers, adders, integrators and active filters, etc. It is also possible to make modules using basic electronic devices or ICs, giving students a flavor of how to design modern ICs and a peek in the Large Scale Integration (LSI) world.

Introduction to digital logic gates and their implementation is important as first glimpse into the working of computer systems. These can be introduced without much background of the analog devices (Figure 6). SPICE environment has a pool of logic gates including AND, NAND, OR, NOR, X-OR etc. One can design simple circuit like a basic binary adder to fairly complex circuits like Arithmetic logic Unit (ALU) and even 4-bit computers using the student version of SPICE. Such playground of digital circuit design can open new horizons for young students in computer engineering.

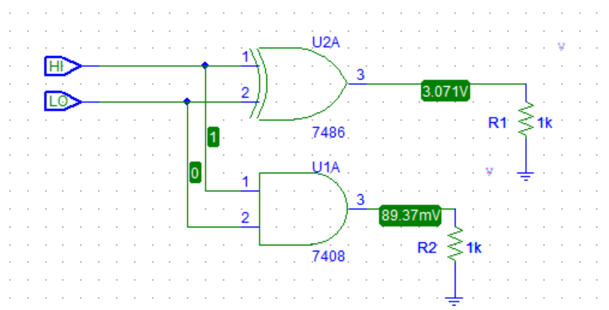


Figure 4: Binary adders take two inputs in binary form and display the output after binary addition. A binary adder circuit employing an XOR and an AND gate is shown above. With the digital inputs provided, the output is shown as analog voltage. This adder is the building block for modern central processing units. Combining this with the clock/timer circuit can give an idea of how modern computers perform binary calculations.

SPICE can provide a very effective introduction to electrical engineering and electronics for young students. They can develop simple skills related to circuits with a head start in their engineering endeavors. The student version is sufficient enough for users ranging from beginners to enthusiasts. With the vast arrays of tutorials, instructions and discussion forums available online, and with the processing power of a modest computer, one can implement decent circuits to learn the underlying principles. Minimal supervision is necessary to setup these circuits in SPICE. This in home simulation tool can be very useful for directing a new generation towards engineering.

Reference

1. Nagel, L.W. and D.O. Pederson,1973, SPICE: Simulation program with integrated circuit emphasis. Electronics Research Laboratory, College of Engineering, University of California.
2. Vladimirescu, A.,1994, The SPICE book. John Wiley & Sons, Inc.
3. Sandler, S.M. and C.E. Hymowitz,2006, SPICE circuit handbook. McGraw-Hill.
4. Tuinenga, P.W.,1995, SPICE: a guide to circuit simulation and analysis using PSpice. Prentice Hall PTR.
5. Foty, D.P.,1997, MOSFET modeling with SPICE: principles and practice. Prentice-Hall, Inc.
6. Massabrio, G. and P. Antognetti,1998, Semiconductor device modeling with SPICE. McGraw-Hill, Inc.

MOHAMMED ARIF I. MAHMOOD
M. A. I. Mahmood did his BS in Electrical and Electronics Engineering from Bangladesh University of Engineering and Technology, Dhaka, Bangladesh and joined University of Texas at Arlington, Arlington, TX, USA in direct Ph.D. program. His research is focused on modeling of nano-scale interactions and development of novel bio-electronic devices. He is affiliated with the Nano-Bio Lab, Department of Electrical Engineering, Nanotechnology Research and Education Center, University of Texas at Arlington.

SAMIR M. IQBAL
Dr. Iqbal received his Bachelor's degree in Electrical Engineering from NED University of Engineering and Technology, Karachi, Pakistan, in 1997, and earned his doctorate in Electrical and Computer Engineering from Purdue University, West Lafayette, IN, a decade later. He is a faculty member of the Department of Electrical Engineering, University of Texas Arlington, Arlington, Texas, USA. He is affiliated with the Nano-Bio Lab, Department of Electrical Engineering, Nanotechnology Research and Education Center and the Department of Bioengineering, Joint Graduate Committee of Bioengineering Program, University of Texas at Arlington and University of Texas Southwestern Medical Center at Dallas. Correspondence regarding this paper should be addressed: Samir M. Iqbal, Ph.D., SMIQBAL@uta.edu