e- Learning and Teaching- Reconfigurable PSPICE for Communication Electronics Courses

Arunkumar K. Walunj
Department of Electronic Science, Poona College, Camp, Pune-411001, India.

Abstract: This paper shreds light on PSPICE software- reconfigurable e-learning and teaching platform for communication electronics courses is proposed. The aim is to realize a platform capable of constructing a wide range of circuit topologies and communication techniques, thus enabling students to gain a better understanding of communication electronics concept of modulation and De-Modulation through practical experiments. A case study of Amplitude Modulation and advanced FFT analysis is discussed in detail. This developed technique is useful for a laboratory, and a user interactive e-learning platform. The reconfigurable communication electronics test-bench can be configured by the students via a Web- based interface to construct a wide variety of AM, PAM, PPM, PWM, FM, FSK, BPSK, QPSK modulations and De-Modulations. The Internet accessible distance laboratory system permits the Teachers and Students to remotely conduct experiments over the Internet with executed on any modern personal computer (PC) without additional hardware. The advantages of the proposed platform include flexibility, user friendly interface, allow for distance waveform measurements, and removal of laboratory time and space constraints.
Keywords: Conceptual Learning, Electronics Courses, e-Learning, PSPICE. Reconfigurable software.

I. Introduction

Laboratories play an important role in teaching and learning technological subjects in colleges. In the era of electronic study, innovative teaching learning methods coupled with appropriate mentoring are required to make the undergraduate students interested in subject. Teaching electronic concept to students by a hands-on approach along with simulation on a topic of interest clearly showed renewed interest among students toward the subject. Additionally, they will gain scientific habits-of-mind (such as the ability to visualize, contemplate, and explain complex concepts and phenomena) that are both encouraged in the recent reform documents and necessary for future careers in science. The computer simulation software could develop the skill set of the electronics student further by introducing various simulations of electronic circuits as the core competencies are achieved by the student[1][2] .

Computer aided learning and laboratory based education can now be performed with computer simulation that can emulate the tools, laboratory environment and experimentation for developing technical education laboratory assignments. Computer simulations are “designed for acquiring skills, problem solving, or obtaining concepts” for technical education [3]. Computer simulation of electronic circuits has become a vital part of learning, discovering and researching the broad understanding of electronics. Throughout the last two decades of the twentieth century, the computer and computer software provides the capabilities for simulation of electronic circuits. Simulated Program with Integrated Circuit Emphasis (SPICE) is an electronic circuit simulation program that was designed by the Electronics Research Laboratory at the University of California at Berkeley in 1973[4]. Computer-based simulations of student practical classes ‘‘virtual laboratories’’ can provide a cheaper and timesaving alternative to traditional practical classes. Indeed, several studies have considered the simulation of a range of skills and practical concepts in the Electronic experiments. These studies highlight many suggested benefits that are common to most types of computer-based learning (CBL) and include:

1. Flexibility of time [students can complete the virtual laboratory at a time convenient to them.
2. Flexibility of location [students can complete the virtual laboratory in a location other than the teaching laboratory [5][7].
3. Control of learning pace (students can take as long as required to understand the concepts with the virtual laboratory) highlight this in the context of students not having to rush to vacate the laboratory; hence, they can work at their own pace [8].

Simulation often goes hand in hand with visualization. The results of changes that a student puts into a model are directly shown on the screen. This generally appeals to students. There are numerous electronic analysis software programs available. One of them is Spice (PC Simulation Program with Integrated Circuit Emphasis). It is a general purpose circuit simulator and one of various versions of SPICE family. It has been extensively used by the Universities and Colleges as a CAD tool because of its simplicity and effectiveness. The role of simulation has been shown to be an effective means in communicating abstract ideas to students. It can be used as a powerful tool in the learning process. Software simulation does not provide finest learning experience by itself, but when combined with hands-on experience, theoretical background of the experiment, however, the simulation software can be a part of a learning environment that will likely produce excellent results. The software allows us to construct a virtual circuit using a schematic, simulate it using various solving techniques, and analyze the results. During our study PSPICE Schematics Version 9.2 was purposely selected being user's friendly and easy to use for the beginners as simulation tool [9] [10]..

II. Methodology

Define and verify that how amplitude modulator circuits is expected to work. Use theoretical formulations for AM. Understand how to use PSPICE for basic circuit analysis. Apply Fast Fourier Transform and analyses its frequency spectrum. Verify your hand calculations and observe the said using waveform figures post simulation done. Finally compare calculated with simulated results. A tutorial in PSPICE for analog circuits can be learned easily using internet interaction. Learn the basics of circuit simulation using Cadence Design System's powerful simulation software. This course platform is useful to college students studying electrical and hardware engineering, electronic hobbyists and tinkerers' tool? Graduate students in Science and electrical engineering. Anyone who wants to learn PSPICE for electrical circuit simulation Anyone who wants to understand the basics of electrical circuit simulation. Teaching, Learning and Evaluation is backbone in higher education. The e-learning methodology is self-explanatory. A simplified flowchart model is shown in figure 1.

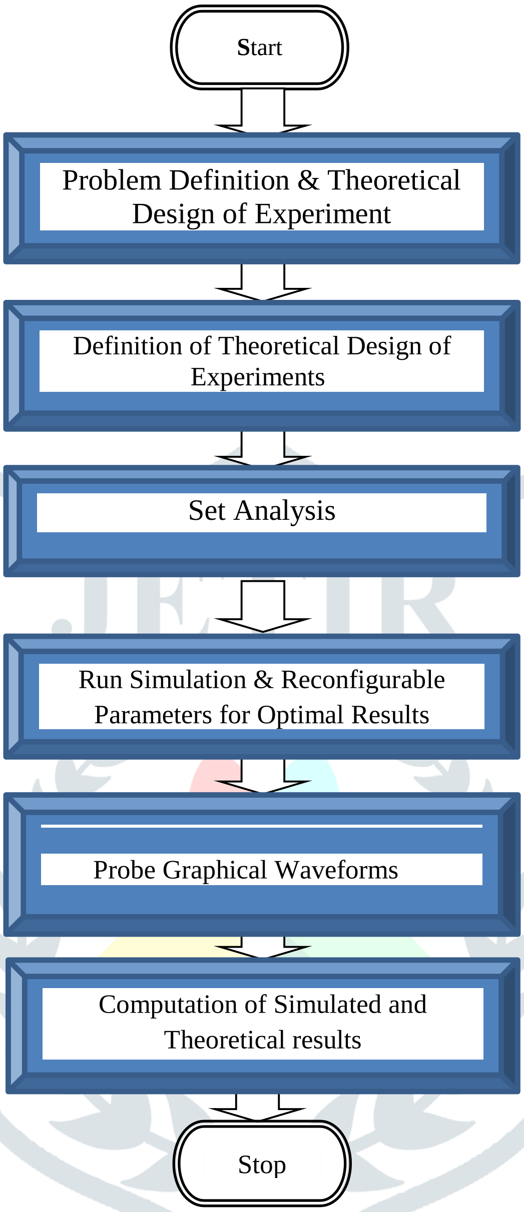

Figure 1: Flow Chart of Process

II. AMPLITUDE MDOULATION (AM) THEORY

The instantaneous amplitude vale of the carrier modulated by a sinusoidal signal is:

$$Vc(t) = [Ec + Em\ Cos2\pi\ \mathrm{fm\ t}]\ \mathrm{Cos2\pi Fc\ t\ Volts}$$

We may consider the part in the square brackets as modifying the amplitude of the carrier. Expand this equation by multiplying out and getting the following result:

$$\mathrm{Vc\ (t) = Ec\ Cos2\pi Fc\ t + Em\ Cos2\pi\ fm\ t\ Cos2\pi Fc\ t\ Volts} \qquad \text{------ (1)}$$

Apply to this equation (1). Using the well-known trignometric expansion formula:

$$\mathrm{(CosA.CosB) = 1/2[Cos(A\text{-}B) + Cos(A\text{+}B)]} \qquad \text{------ (2)}$$

We get: $\mathrm{Vc(t)= Ec\ Cos2\pi fc\ t} + \frac{Em}{2}\ Cos2\pi(fc - fm)\mathrm{t} + \frac{Em}{2}\mathrm{Cos2}\pi\mathrm{(fc\text{-}fm)\ t\ volts.}$

In term of the modulation index m:

$$\mathrm{Vc(t)=Ec\ Cos2}\pi fc\ t + \frac{mEc}{2}\ \mathrm{Cos2}\pi(fc - fm)t + \frac{mEc}{2}\ \mathrm{Cos2\pi(fc+fm)]t} \qquad \text{---- (3)}$$

III. CIRCUIT SIMULATION EXPERIMENT

A case Study of Amplitude modulator circuit using LF 411is presented. Create the schematic circuit in PSPICE as shown in the above Figure 2. To generate an amplitude modulation signal, three generators are connected to the summing circuit. Set up the three generators with parameter as shown in figure 2. These were carrier generator. Create two more but set the frequency and amplitude to a value, which will create an AM with 50% modulation

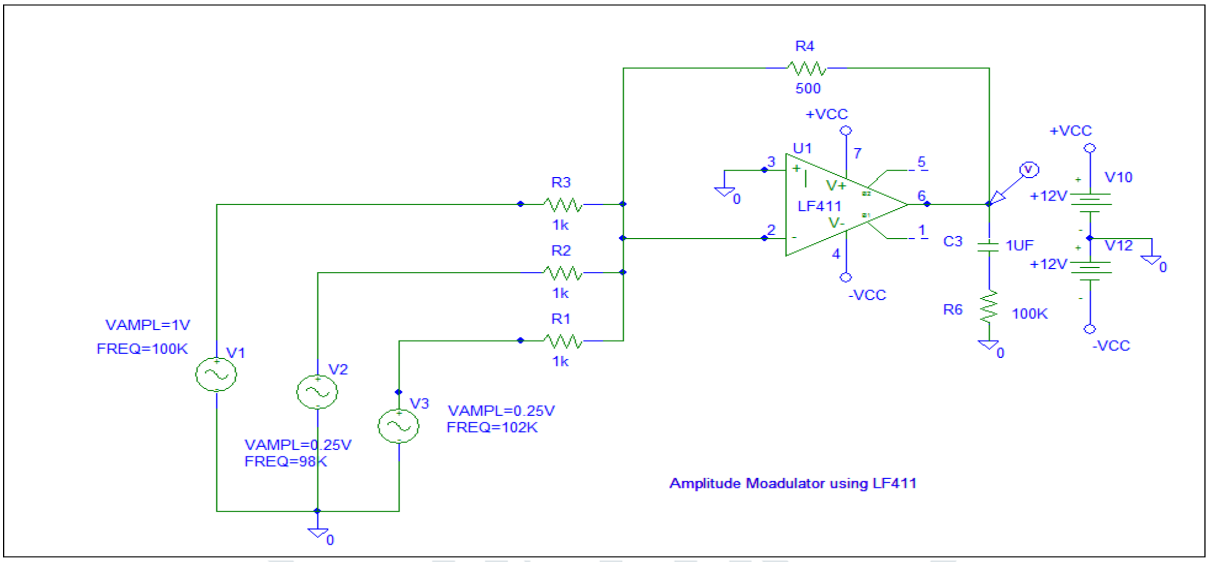

Figure 2: Amplitude Modulation using IC LF411.

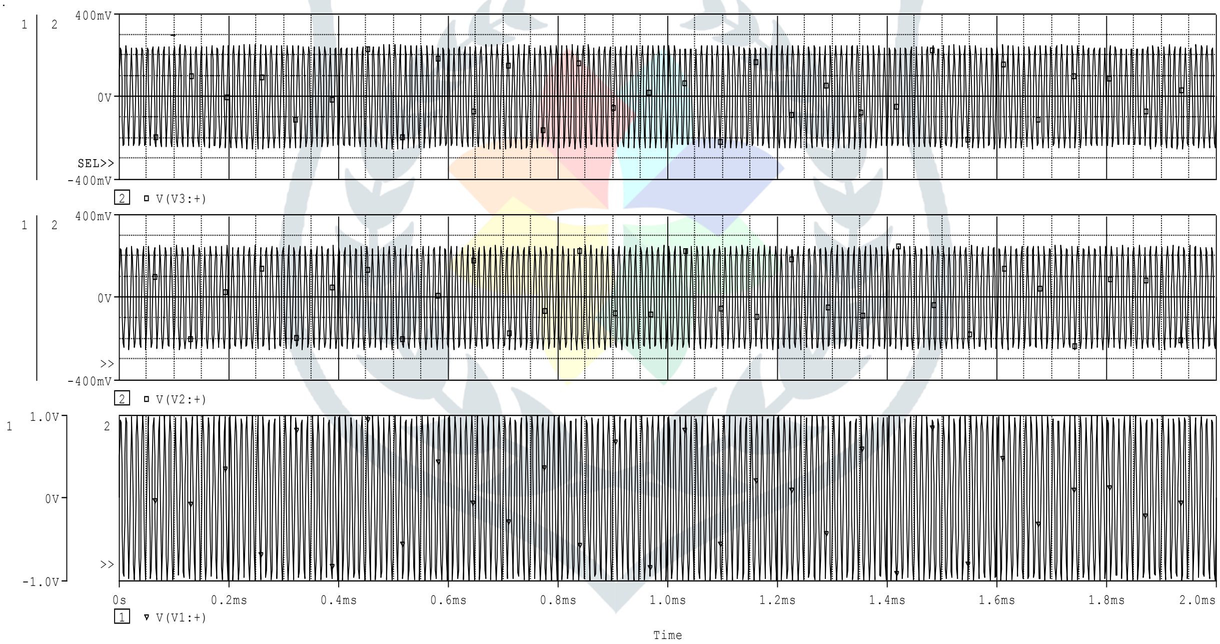

Figure 3: Carrier Frequency, USF and LSF Signals

III. TEST/ PROBE PROCEDURE

Set the Analysis / Transient parameters: Run to time = 10 ms, Maximum Step size =0.1µ. Press F11 to simulate. Note the gain of the op-amp is 0.5, so the magnitude of the un-modulated carrier is 0.5v. Use the two cursors to measure the maximum and minimum value of the AM output displayed in fig.3. The modulated carrier outline, or envelope, has an upper and lower envelope with the same shape as the modulating signal. The envelope of an AM signal modulated by a complex signal is difficult to interpret on an oscilloscope, so, in general, a spectrum analyzer is more useful in that it allows us to examine the frequency content of these complex signals. However, for a signal modulating frequency and time domains are both useful [11].

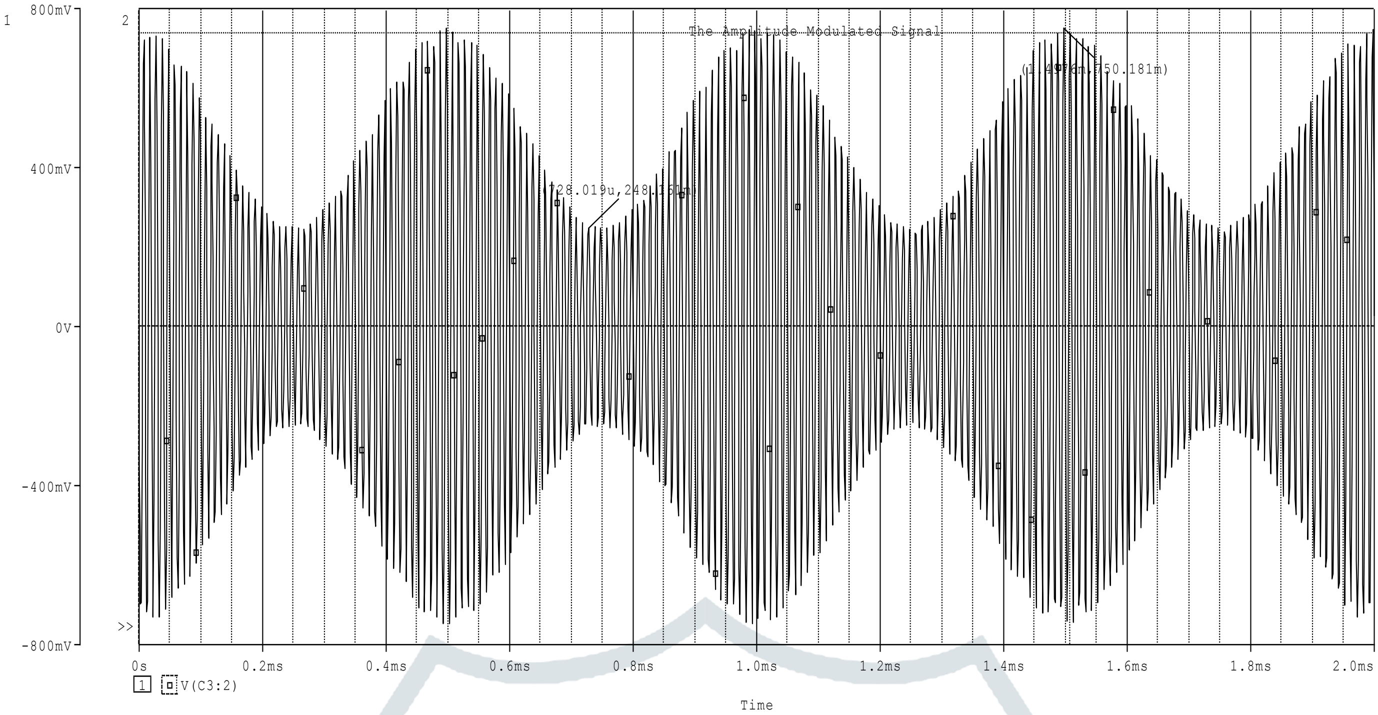

Figure 4: The AM Signal

The outline of the signal is called the envelope shown in figure 4, and the upper or lower portion of the envelope has the same shape as the modulating signal.

IV. FREQUENCY SPECTRUM ANALYSIS USING FAST FOURIER TRANSFORM (FFT)

Use the FFT icon to display the signal in the frequency domain. The resolution in this domain is determined by the parameters set in the transient set-up. The greater the time axis the better the frequency resolution. We can see the spectral components in figure, for the 100 KHz carrier modulated by a 2 KHz modulating signal. The components are located at the locations given by the above expression i.e. spectral components at 98 KHz, 100 KHz and 102 KHz. Another point to notice from the expression is that the side-bands are not equal in magnitude to the carrier component.

The bandwidth for this signal is obtained by subtracting the lowest frequency component from the highest frequency component. The Probe FFT icon, when pressed produces a spectrum of the AM signal showing the carrier and side frequencies as in figure 4. Increased spectral resolution is achieved by increasing the Run to time and / or reducing Maximum Step size in the transient setup menu.

The bandwidth is measured as the difference between the upper and lower sidebands: BW = (fc+fm) - (fc-fm) = 2fm Hz. Thus, the bandwidth for a 2-KHz modulation frequency is 2fm = 4 KHZ.

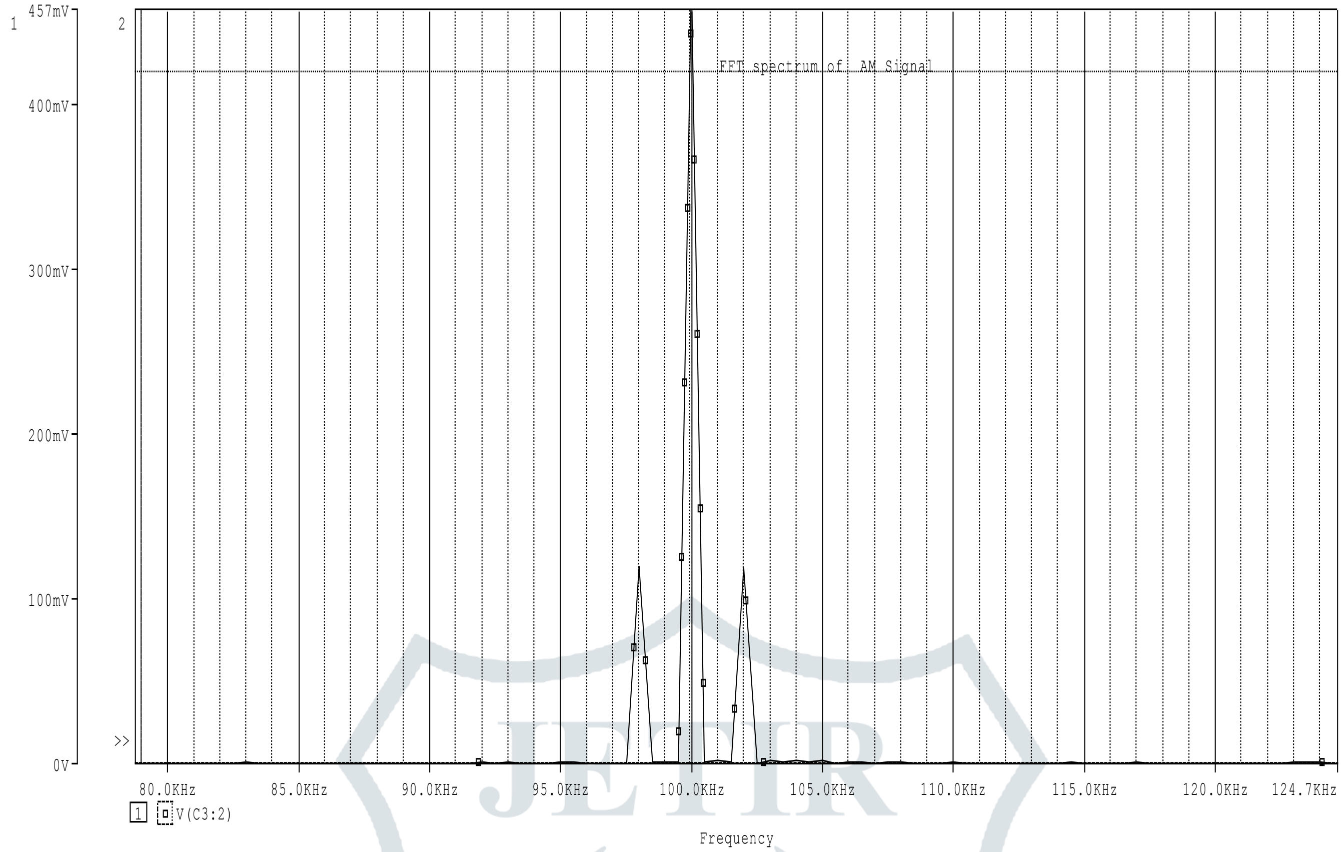

Figure 4: FFT Modulation Spectrum

From simulated waveforms and FFT frequency spectrum analysis the following results were calculated:

1. Modulating signal voltage = 1V
2. Modulating signal frequency = 2KHz
3. Carrier signal voltage = 1V
4. Carrier signal frequency = 100 KHz
5. Maximum Amplitude of modulated signal Emax = 746.572mV
6. Minimum Amplitude of modulated signal Emin = 487.224 mV
7. Modulation Index (MI) = Emax+ Emin / Emax –Emin x 100

=1233.796/ 259.348= 0.4757x100 = 47.57 %

8. Centre frequency = 100 KHz with LSB= 98 KHz and USB= 102KHz
9. AM Signal Bandwidth = USB-LSB = (102-98) KHz = 4 KHz
10. AM Signal Spectrum Voltage = (746.572+ 696.412) mV= 1442.984mV.

V. CONCLUSION

PSPICE-based experiments in teaching Communication electronics conducted successfully. AM circuit used as the sample student's task. The presented technique designed to represent all activities in the real laboratory experiment to understand theory as well. Thus, it is observed that simulated and calculated results are almost meeting to 50%. The implementation of the study shows that the students are satisfied with this approach. Most of them have a good performance in this teaching approach. Therefore, teachers are invited to build their knowledge in the simulation software to solve the Students learning problems by cooperating with each other's.

REFERENCES

[1] Tina Hudson, Matthew Goldman, Shannon Sexton "Using Behavioral Analysis to improve Student Confidence with Analog Circuits" IEEE Transaction in Educ., Vol.51, pp. 364-369, August 2008.
[2] L. D. Feisel, A. Rosa "The role of Laboratory in Undergraduate Engineering Education" J. of Eng. Edu., vol. 94, pp. 57-71, Jan. 2005
[3] Shaw, E. &Okey, J. (1985). Effects of microcomputer simulations on achievement and attitudes of middle school students. Paper presented at the annual meeting of the National Research in Science Teaching, French Lick, IN., 1-6.
[4] Nicola J. Gibbons, Chris Evans, Annette Payne, Kavita Shah, and Darren K. Griffin. Computer Simulations Improve

University Instructional Laboratories Cell, Biology Education Vol. 3, page 263–269.
[5] Race, P. (1994). The Open Learning Handbook, 2nd ed. London: Kogan Page.
[6] Gilbert, L., Wang, C., &Sim, Y.-W. (2005). An e-learning system engineering methodology. ICALT 2005, Fifth IEEE International Conference on Advanced Learning Technologies, 150 - 154.
[7] Tuinenga, P. (1988). SPICE, a guide to circuit Simulation and analysis using PSpice. Englewood Cliffs, NJ: Prentice-Hall
[8] Heerman D.W., Fuhrmann T.T. Teaching physics in the virtual university: the mechanics toolkit. Computer. Phys. Communication 2000, 127:11–15.
[9] PSpice User's manual, OrCAD Corp. (Cadence Design Systems, Inc.)
[10] OrCAD Capture User's Guide, OrCAD Corp., Cadence Design Systems, Inc.)
[11] PSPICE for Analog Communications Engineering by Pual Tobin, Dublin Institute of Technology, Ireland