

Use of Simulation Software for Innovative Teaching & Learning to Undergraduates in Pandemic

Jayshree S. Sonawane ¹, Dr. R. D. Patil ²

¹ *Research Scholar, Shri. J. J. T. University, Jhunjhunu, Rajasthan*

² *I/C Principal, Dr. D. Y. Patil College of Arts, Science & Commerce College, Pune_411018*

Abstract - Simulation is useful to make teaching and learning process interactive for the electronic subject and enhance knowledge of the topic. It is expected that there will be significant change in effectiveness of teaching and learning the electronic practical using simulation software. Since March 2020 students are not coming to institutes, colleges, universities due to CORONA disease situation. So, all practical courses could not conduct so alternative is using simulation. We can conduct some practical's using simulation software. So that students will engage & they will practice at home individually by using trial error method at any time anywhere.

The purpose of this experimental study is to see the effect of simulation software on the attitude towards Electronic subject. When colleges will start already students were aware about circuit, components, connections, instruments, practical set up because of handling of simulation in lockdown period. Now they will perform actually. Virtually they already had done so students will perform laboratory work with lot of confidence. Very less mistakes of students while working laboratory of Electronic science. In this Experimental study, there will be two groups experimental group and control group. Experimental group will performed the experiments using computer simulation software for selected practical in Electronic, while controlled group will studied by traditional method. The target population consists of students offering Electronic subject in undergraduate computer science streams.

From the student side or by taking feedback of students it is observed that in B.sc. computer Science, Electronic subject is difficult to understand. Fundamental of Electronics are the back-bone of computer so due to difficulty they are not able to understand circuit working & so the basic concepts remain unclear. Other objective is to change their mind set that electronic subject is easy to understand and it is important in their IT carrier field. The purpose behind this study is to improve the performance of students in Electronic Science subject. It is must to develop innovative method of teaching Electronic science practical and theory, to understand

the internal mechanism & working of circuits so as to motivate the students of computer science to study the Electronic subject.

Index Terms - Simulation, Electronic Circuits, Software, CORONA, SPICE Software, TINA-PRO Software

I.INTRODUCTION

Advancing technology has opened many doors in education. During the last few years, software tools in various forms have started playing on increasing important role in education. With computer-based tools becoming more affordable, we have the expectation that, time & distance factors will have less impact on the way instruction is delivered to students. In the future instructional software can be used as a complementary tool during laboratory work, the theoretical background of the laboratory work can become available to the students through multimedia software. The simulation software can be used for preparation of the Laboratory exercise & Laboratory experiments can be supported by computer systems, used for collecting data, processing measurements, testing wiring & equipment configurations, simulating behavior of equipment etc. Appropriate use of this educational software allows students to build knowledge by giving them opportunities to explore the equipment to be used before hand in a safe for them, interact with it, Experiment, problem-solve, & collaborate. Interactive multimedia experience cannot replace the real laboratory work but enhance the learning process of many students and also help them to find the relation between the theoretical principles & observed behavior in an easy & spontaneous way. Computer simulation software can also be used to design complex circuit & access supplemental references. These can be effectively used before a

laboratory to increase familiarity with certain laboratory procedures.

II. EXPERIMENTAL

Spice is very popular software for analyzing electronically & electronic circuits. The Microsim Corporation first introduced the PSPICE simulator which can run on personal computers. The student version of PSPICE-9.2, latest version which is freely available on the internet. This software is ideal for classroom use & for practical assignments. One or two lectures on SPICE are required to the students to get some basic knowledge of how to use SPICE. If the SPICE simulator introduced to B.Sc. students then with minimum amount of time & effort teacher can explain practical course to the students. Once the students develops an interest in & an appreciation for the application of the circuit simulator like SPICE. He or She can move on to more advanced materials for the full utilization of SPICE in solving complex circuits & systems. LTSSPICE is linear technology simulation program with Integrated circuits Emphasis. PSPICE strong point is that it helps the user to simulate the design graphically on the computer before building a physical circuit. Hence the designer can make any necessary changes on the prototype without modifying any hardware. As soon as the test design is complete, PSPICE can help you run & check on it before deciding to commit yourself to building a hard model or circuit.

Hence PSPICE allows you to check the operability of the circuit model in real life simulations to validate its practicality. Since all the tests, design & modification are made over a terminal, the designer can save a lot of money that would have otherwise has been spent on the building circuits & modifying them [6], [7], [8].

Here the Researcher is using PSPICE 9.2version & LTSPICE software and TINA-PRO for Experimental course at F. Y. B. Sc. In above discussion we get importance of simulation software. Along with simulation software, Animation effects are given to the circuits. The Animation will probably be remembered better to help understand & recall it later. In this sound & motion is truly interactive learning experience & it will an excellent self study material. Specific series of practical, designed for practical applications offer a project or practical based approach that gives students a sense of real world. Capability of

software application extensive screen captures provide easy to follow visual examples for each major textual step while visual summaries reinforce the concepts building upon students knowledge, watching the simulation while reading about a concept help students to better understand.

In LTSPICE many facilities are given, at a time we can see many graphical out put at different mode. At particular node we can get signal value along with separate screen, circuit and its output can see on a ones screen so teacher can explain step-by-step circuit action to the students.

Another electronic simulation software is TINA-PRO in which all Electronic instruments present in the software like a function generator , CRO, digital multi meter, which is gives the student real lab experience . So the use of TINA PRO simulation software as nothing but working in virtual lab.

Researcher selected only following .B.sc. Computer, science Electronic Practical- Using LTSPICE- Rectification using filter and without filter. Using TINA-PRO software- using zener diode regulator circuit LTSPICE, PSPICE, and TINA-PRO this Electronic simulation software freely available on internet so down load it use it for practical purpose. We are getting following types of screen.

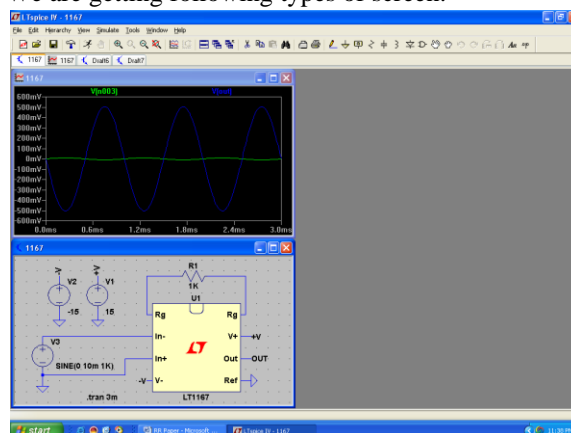


Fig. 1 Screen of LTSpice

III. SPICE SOFTWARE

LTSpice IV is a fourth generation switching regulator design program from Linear Technology. The program consists of a high-performance SPICE simulator extended with a mixed mode simulation capability that includes new intrinsic SPICE devices for macro-modeling Switch Mode Power Supply

(SMPS) controllers and regulators. The program includes an integrated hierarchical schematic capture program that allows users to edit example SMPS circuits or design new circuits. An integrated waveform viewer displays the simulated waveforms and allows further analysis of the simulation data. There is a built-in database for most of Linear Technology's power ICs and many passive components. The device database, schematic editing, simulation control and waveform analysis are integrated into one program.

Due to the mixed mode simulation capability and many other enhancements over previous SPICE programs, the simulation speed is greatly improved while simulation accuracy is retained. Detailed cycle-by-cycle SMPS simulations can be performed and analyzed in minutes. A user can get a detailed simulation of power systems with a few mouse clicks without knowing anything about the device, SPICE or the schematic capture program. Pre-drafted demo circuits can be used as a starting point to build the custom circuit to fit different power supply requirements. After the new schematic is created, the system can be simulated and reports generated.

The program's integrated hierarchical schematic capture and SPICE simulator are available for general use. The improved performance of the SPICE simulation engine is a benefit for simulating general analog circuits and should be of interest to all electronic engineers. There are no arbitrary limits on component count or content. With an installed base of over 2,000,000 licenses so far, LTspice has arisen as the de facto standard SPICE simulator. We hope you enjoy the program and find it useful.

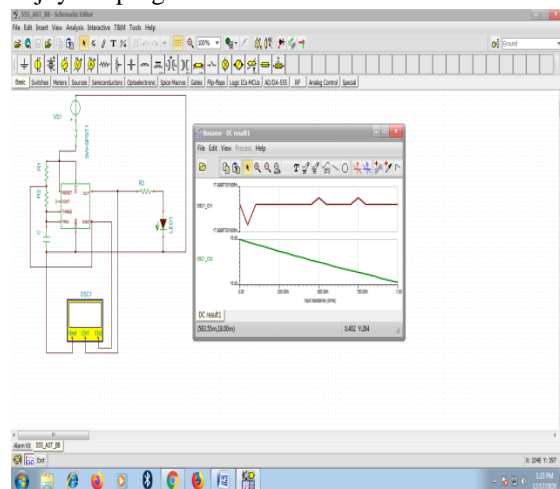


Fig. 2 Screen of TINA-PRO

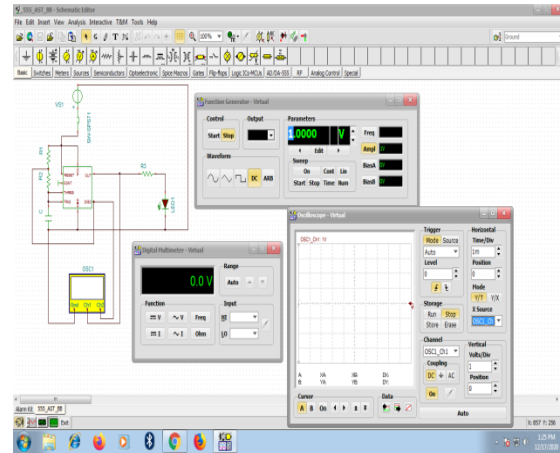


Fig. 3 Analysis screen of TINA-PRO

This shows typical screen shot of TINA-PRO, which shows IC555 circuit along with multi-meter, Function generator, Oscilloscope. In this all components visible on the screen with symbols, instruments so it becomes user friendly to use the software. We can get 2D/3D views of electronic components. It has different tools, Instruments, Analysis like AC, DC, and Transient for linear and nonlinear circuits. It has different functions -AC time function, AC Transient function, auto, repeat, built in function, bus/wire properties, nodal, analysis, clear, lose, control, compile, etc TINAS oscilloscope displays an electrical waveform as function of time on its screen. TINAS virtual oscilloscope has more input channels than a conventional oscilloscope. Some more signals can be displayed simultaneously.

IV. CONCLUSION

Individual feedback on students understanding is also possible through simulation software. Remedial teaching to each & every student is not possible conventionally by the teacher but with the help of simulation software it can be made possible. Individual self paced learning can play a role for remedial teaching through simulation.

Following benefits are expected through such experiments:

- Encourage deeper thinking by the student,
- Reinforce & link concepts
- Promote students-students & lecturer students interactions.
- Provide instantaneous feedback to lecturer & students on the student's progress.
- Students are comfortable with its use.

ACKNOWLEDGMENT

The researcher expresses sincere thanks to J. J. T. University. Researcher would like to thank Principal of Dr. D. Y. Patil College of Arts, Commerce and Science Collage, Pimpri, Pune 18.

REFERENCES

- [1] Ausubel, David P. (1968). Educational Psychology, a Cognitive View. New York: Holt, Rinehart and Winston Inc
- [2] Best John W. (1978). Research in Education (3rd Edition), Prentice-Hall of India private Ltd. New Delhi.
- [3] Brasell, H. (1987). "The effect of real-time laboratory graphing on learning graphic representations of distance and velocity", Journal of Research in Science Teaching, 24 (4),385-395.
- [4] Busch, T. (1996). "Gender, group composition, cooperation, and self-efficacy in computer studies". Journal of Educational Computing Research, 15(2), 125-135.154
- [5] Y. B. Gandole (2005). "Changing the nature of Undergraduate electronics science practical work", International Journal of Instructional Technology and Distance learning, Vol. 2. No.4,
- [6] Busch, T. (1996). "Gender, group composition, cooperation, and self-efficacy in computer studies". Journal of Educational Computing Research, 15(2), 125-135.154
- [7] Y. B. Gandole (2005). "Changing the nature of Undergraduate electronics science practical work", International Journal of Instructional Technology and Distance learning, Vol. 2. No.4, 9-22.
- [8] www.pspice.com
- [9] www.linear.com
- [10] www.designsoftware.com.
- [11] www.tina.com