A review on SPICE Simulation Educational Tool to Enhance Student Centered Learning

Pratibha Bhanudas Sakhare
Department of Electronics,
Kamla Nehru Mahavidyalaya Nagpur -440024
pratibhasakhare@gmail.com

Abstract:-

Carrying out real-world software projects &simulation techniques in their academic studies helps students to understand what they will face in industry, and to experience first-hand the challenges involved when working collaboratively. Microelectronic circuits are integral components to electrical, electronics and computer undergraduate curriculums of science and engineering. Most of the instructional strategies used to help enhance student's educational tool knowledge and understanding. SPICE is an electrical and electronics circuit simulation tool that has been widely adopted for industrial applications and education. The effectiveness of SPICE simulation tools for incorporating simulation into lecture, in class active learning, as well as assignments, and industrial projects. The aim of this study is to analyze simulation tools for student centered learning using simulation program with integrated circuits emphasis (SPICE) reliability simulation method which shifts the focus of circuit analysis, functionality and characteristics. This educational tool can provide student teacher interactions, practical work in lab, assignments in classrooms and parallel effective communications.

Keywords:-

Student-centered learning, SPICE, active-learning, simulation educational tool, student teacher interactions.

I. INTRODUCTION

Teaching and learning process executed by an instructor or lecturer previously now replaced by student-centered learning (SCL) [1].Simulation-based educational products are excellent ''illustrative tools'', used exceedingly in student centered learning methodologies [2]. Simulation tool is help to students compatible with the technology jobs in order to solve real-world problems of the current era [3].In the fields of electronics, electrical and computer simulation is particularly important in the context of electronic design and in the design of systems that contain microelectronics, since fabricating such circuits is expensive and physical prototypes require a large investment of resources. Thus, simulation is tightly coupled with the circuit design process as a general standard of practice in industry [4].In particular; simulation is very valuable to introductory microelectronics courses, which are core to most electrical engineering and electronic science curriculums, as simulation provides students with deeper insights. With introductory microelectronics courses, students must transition from analyzing electrical circuits containing simple linear elements such as resistors, capacitors, and inductors to analyzing, for the first time, circuits that contain complex, non-linear components such as diodes, transistors, or logic gates. The behaviors of these non-linear components depend on a large number of parameters and have the ability to exhibit different behaviors based on a wide range of operating conditions [5, 6, 7].Current flow through these devices to the voltages across the terminals does not have linear relationship these electronic systems cannot be analyzed using straight forward linear algebra, and differential equations for circuit analysis, as in prerequisite courses [4].Voltage transfer characteristic curves, current voltage (I-V) curves analysis done by student. A novel approach to teaching microelectronics that entirely revolves around active learning based simulation in the classroom. The objective of instructional approach is not only to enhance students' ability to solve and analyze microelectronic circuits, but also to provide them with the intuition and deep insights they will need in order to be effective real-world designers. The simulation environments most often used (i.e., SPICE simulators) are mature and widely used in both academia and industry [8, 9, 10].The norm in education is to incorporate SPICE circuit simulations as supplementary homework problems or as part of laboratory assignments. After theory is taught in lecture, students are given assignments afterwards to do on their own. To enhance students' educational experience created an active classroom in which technology was used in conjunction with traditional instruction [11]. The benefits to learning of active learning, or having students "do work" in class beyond listening and note-taking, have been well published within STEM education [12].

II. LITERATURE REVIEW

The American Psychological Association identified learner-centered psychological principles. The domains of the learner-centered principles the meta cognitive and cognitive, affective, personal and social, developmental and individual differences factors emphasize both the learner and learning [13].Analysis of NAAC Accreditation System using ABCD framework [14].

Simulation-based educational products in academics are becoming wide spread and ample literature is available on this area. These products are categorized as scenario, simulation, and game-based e-learning [15].Circuit simulation is inextricably linked to the microelectronic circuit design process. Unlike circuits that are constructed using discrete electronic components that can be easily prototyped by soldering devices to printed circuit boards, microelectronics have nanoscale features that require expensive equipment for fabrication, and testing. These are typically not readily available even to the most well-known manufacturers of consumer electronics, let alone students [16].Extensive use of simulation prior to manufacturing from long years in industry. The most widely used circuit simulation programs are derivatives of SPICE (Simulation program with integrated circuit emphasis) [17, 4].Presented a multimedia module for climate-simulation-experiments application based on the energy balance model. The proposed method was free from numerical or algebraic computations. To motivate the students into learning, the fundamental principles of the subject are taught in an active learning environment [9]. SPICE was developed in the 1970's by Pederson and Nagel, having started as a class project in the 1960's [2, 18, and 19].The class project led to research, which was quickly adopted by industry [2]. The adoption accelerated over the decades [8] and is prevalent today, being used in many commercial products (e.g. Cadence's PSPICE, Synopsis' HSPICE, and Analog Devices'LTSPICE [20, 21, and 22]). SimSE simulation environment allows students to apply their conceptual knowledge in projects [23].

III. METHODS

Classroom instructional method

The learning objectives for students:

1) To Acquire fundamental knowledge on the purpose and functioning of microelectronic components, specifically transistors, and diodes; 2) To obtain knowledge and skills in the design of application circuits using these components (e.g., amplifiers, MOSFET, rectifiers, power supplies, and logic circuits); 3) To apply the fundamental techniques and procedures for analysis of circuits containing these microelectronic components; and 4)To effectively use modern tools (e.g. SPICE) for electronic design. A SPICE-based simulation tool was selected for the course, given its benefits in teaching material that involves complex, non-linear electrical systems [24]. Also used in digital circuits, analog circuits, or power systems, SPICE is currently utilized in electrical and computer engineering education [25]. SPICE was incorporated into the classroom in a holistic manner, with daily use by the instructor as the basis for lectures, including inductive lectures, student assignments and in-class activities often involving pair simulation, which we adapted from the pair-programming approach [26]. SPICE simulation tools, instruction coupled with frequent use of other active learning techniques. The most frequently used active learning techniques were think-pair-share (TPS) [27], predict-observe explain (POE) [28], and pair simulation (PAIR), with the majority of activities involving simulation in some way. With think-pair-share, students worked on problems individually, and then in groups to improve upon their answers, with eventual sharing with everyone [25].Students can solve examples from traditional "textbook" approaches (e.g., Sedra's textbook [29]). Amplifier analysis equivalent circuit models are used to analyze the AC and DC responses separately then determine the amplifier voltage gain [27].Circuit simulators use advanced models for microelectronic components for example MOSFET can operate in (cut off, saturation, active) region [30] SPICE simulator can calculate accurate solution and current voltage curve characteristics [20].

IV. ASSESSMENT METHOD

Simulation-centric tool can helpful in combination of indirect and direct assessment methods for assess and compare learning and understanding of electronic& electrical circuits as compare to previous approach. Students can beneficial to class work and project through active learning [31].

V. CONCLUSION

By incorporating simulation into effective instruction can executed active learning.Based on review of the literature instructional methods SPICE simulation as the basis of inductive lectures, in-class active learning, and follow up assignments. Students can perform the best, enhance practical knowledge. SPICE simulators have robust computational abilities and provide advanced models for microelectronic components. Simulation-based teaching methodology used to solve open-ended design problems. Simulation-centric instructional approach provides a framework to implementation in STEM courses. Simulation-centric instructional approach can provides a framework for other faculty by comprehensive study and assessment. Recently simulation-based power estimation framework used for FPGAs, called FPGA-SPICE tool extends the VTR architecture description language to include transistor-level modeling parameters of FPGA components [32].

VI. REFERENCES

[1]. Hairullia, M. J., &Noraidah, S. (2013) Student centered learning in statistics: analysis of systematic review, https://doi.org/10.1016/j.sbspro.2013.10.406
[2]. A. Bogdanowicz, "Spice circuit simulator named ieee milestone,"The Institute, IEEE News Source, IEEE, 20117.
[3]. Z. H. Khan and M. I. Abid, Role of laboratory setup in project based learning of freshmen electrical engineering in Pakistan, Int. J.Electr. Eng. Educ. 54(2017), 150–163.
[4]. Atiq Siddiqui *, Mehmood Khan, Sohail Akhtar Department of Systems Engineering, King Fahd University of Petroleum and Minerals, 31261 Dhahran, Saudi Arabia, 2007
[5]. P. Horowitz & W. Hill, The art of electronics, Cambridge University Press, New York, NY, 2015.
[6]. R. Thomas, A. Rosa and G. Toussaint, The analysis and design of linear circuits, Wiley. 2011.
[7]. J. Williams, "The art and science of analog circuit design, "Elsevier Science& Technology Books, San Diego, CA, 1998, p.416 p.
[8]. A.Vladimirescu, Spice-the third decade, Proceedings on Bipolar Circuits and Technology Meeting. 1990, 96–101
[9]. L. W. Nagel and D. O. Pederson Spice: Simulation program with integrated circuit emphasis, Electronics Research Laboratory, College of Engineering, University of California, 1973.
[10]. B. Balamuralithara and P. Woods, Virtual laboratories in engineering education: The simulation lab and remote lab, Comput.Appl. Eng. Educ. 17 (2009), 108–118.
[11]. R. Carr, S. Palmer, and P. Hagel, Active learning: The importance of developing a comprehensive measure, Act. Learn. High.Edu. 16(2015), 173–186.
[12]. T. De Jong, M. C. Linn, and Z. C.Zacharia, Physical and virtual laboratories in science and engineering education, Science. 340(2013), 305–308.
[13]. Yun-Joan and Charles Reigeluth, Journal of Digital Learning in Teacher Education | Volume 28 Number 2
[14]. Aithal P. S., Shailashree V. T., & Suresh Kumar P. M., Analysis of NAAC Accreditation System using ABCD framework, International Journal of Management, IT and Engineering (IJMIE), Vol. 6, Issue 1, pp.30 - 44, January 2016
[15]. John Randall, higher education quaterly09551-5224 volume56, no2, April 2002, pp.188-203
[16]. C.Duhigg and K.Bradsher, how the US lost out on iphone work, the New York Times, January 21, (2012).
[17]. A. Bogdanowicz, "Spice circuit simulator named ieee milestone,"
The Institute, IEEE News Source, IEEE, 2011.
[18]. L.Nagel and R Rohrer, Computer analysis of nonlinear circuits,excluding radiation (cancer), IEEE J.Solid-St. Circ. 6 (1971), 166–182.
[19]. D. Lakens, Calculating and reporting effect sizes to facilitate cumulative science: A practical primer for t-tests and ANOVAs, Front. Psychol. 4 (2013), 1–12
[20]. "Hspice," Synopsys, 2017
[21]. "Ltspice," Analog Devices, 2017.
[22]. "Orcad pspice," Cadence Design Systems, 2017.
[23]. E. Oh Navarro, A. van der Hock, SimSE: An interactive simulation game for software engineering education, Proceedings of Computers and Advanced Technology in Education (CATE), (2004 August), Kauai, Hawaii
[24]. A. Abramovitz, Teaching behavioral modeling and simulation techniques for power electronics courses, IEEE Trans. Educ. 54 (2011), 523–530.
[25]. A. Beg, A web-based method for building and simulating standardcell circuits—a classroom application, Comput. Appl. Eng. Educ.23 (2015), 304–313.
[26]. N. Salleh, E. Mendes, and J. Grundy, Empirical studies of pair programming for cs/se teaching in higher education: A systematic literature review, IEEE Trans. Software Eng. 37 (2011),509–525
[27]. R. M. Felder and R. Brent, Teaching and learning stem: A practical guide, John Wiley & Sons, 2016
[28]. C. W. Liew and D. F. Treagust, A predict-observe-explain teaching sequence for learning about students' understanding of heat and expansion of liquids, Australian Science Teachers' Journal. 41(1995), 68–71.
[29]. A. S. Sedra and K. C. Smith, Microelectronic circuits, Oxford University Press, New York; Oxford, 2015
[30]. B.J.Sheu, D. L. Scharfetter: Berkeley short channel igfet model for MOS transistors, IEEE J. Solid-St. Circ. 22(1987), 500–566.
[31]. K. A. Neuendorf, The content analysis guidebook, Sage Publications, Thousand Oaks, CA, 2002.
[32]. Xifan Tang, Pierre-Emmanuel Gaillardon and Giovanni De Micheli,FPGA-SPICE: A Simulation-based Power Estimation Framework for FPGA: 33rd IEEE International Conference on Computer Design (ICCD), 2015