

Imbibing Industry requirements in Engineering Curriculum by augmenting it with Industry Standard simulation tool

Anupama R Itagi¹, Sushma V²

^{1,2}Department of Electrical and Electronics BVBCET,
Hubballi, Karnataka, India

¹anupama_itagi@bvb.edu

²sushma_v@bvb.edu

Abstract: At the end of graduation, young engineers will be acquainted with rich theoretical background. However students lack applying this knowledge in real time. Hence darning of curriculum is required to meet at least few employer requirements. This can be done by industry- academia interaction. Considering the input of experts, course instructor should follow a well-focused and designed pedagogical method to train students. In this paper authors discuss about effective Teaching-Learning process for undergraduate students with an aid of Industry Standard simulation tool. It is proposed to use Industry Standard simulation tool PSpice for various mandatory courses like Circuit Analysis (CA), Analog Electronic Circuits (AEC), Digital Electronics Circuits (DEC). This helps in making students meet the preliminary step in the design of Embedded Systems. Typical PSpice simulation examples on each course mentioned above are included and possible outcomes and achievement of Performance indicators are also discussed.

Index Terms—Effective Tecahing-Leraning, Circuit Analysis, Analog Electronic Circuits, Digital Electronics Circuits, Industry Standard PSpice simulator, Attainment of Program Outcomes.

Anupama R Itagi

Department of Electrical and Electronics BVBCET,
Hubballi, Karnataka, India
email – anupama_itagi@bvb.edu

1. Introduction

To succeed in mission, one must have single-minded devotion towards goal - Dr. A P J Abdul Kalam Engineering is a practicing profession, a profession devoted to harness and modify the three fundamental resources that humankind has for the creation of all technology: energy, materials, and information. The overall goal of engineering education is to prepare students to practice engineering [1]. As a step in this direction, todays engineering education provides an opportunity for students to opt their field of interest for practicing in future. To get good success in the field of interest, one should have strong fundamentals. Courses like CA, AEC and DEC are introduced as mandatory courses even in the curriculum of Electrical Science Engineering to strengthen fundamentals at an early stage of engineering education. Knowledge of courses like CA, AEC and DEC is necessary and essential prerequisite for Embedded System design. Embedded System co-design includes design and realization of Analog and Digital Electronic circuits.

Traditionally, these courses were taught in isolation, keeping students in dark about their significance in real time application.

In addition to this, the rapid progressing world, has are many entities that attract young engineers and distract them in acquiring engineering fundamental knowledge entailed to build their career. These are few root causes that hinders student to get industry

required knowledge and skill set. Hence the above referred inspiring quote challenge teachers to drive novices towards accomplishing knowledge and skill set demanded by the employer. This is a wakeup call for all academicians to adopt well focused and designed pedagogical method for effective Teaching-Learning process.

Importance of teaching by simulation is discussed in [2]. At this stage authors would like to suggest use of single Industry Standard simulation tool for effective Teaching- Learning process of electronic circuit courses to make students industry ready. However in higher semester students can be encouraged to use other standard simulation tools to carry out their projects with the knowledge availed in early stage. The advantages of proposed pedagogical practice emphasized are:

- Creates an opportunity to explore use of tool for different courses.
- One can get good proficiency in using the tool.
- User can learn modifying the model as per requirement. Elevates confidence among students to learn any simulation tool.
- Verification of Analog and Digital circuits through simulation, an inevitable and preliminary step in this design process of Embedded System.

Paper is organized as follows. Section II discusses the choice of software for teaching different electronic circuit courses. Section III introduces Industry Standard simulation tool PSpice in brief. Section IV highlights use of PSpice in typical electronic circuit simulation. Section V is on possible attainment of Performance Indicators. Section VI concludes the paper.

2. Choice Of Simulation Software

There are growing trends in industry for engineering graduates to possess certain fundamental skills. Educators and trainers of future engineers should take into account all characteristics desired by employers of our graduates [6]. Teaching appropriate computer simulation tool for future engineers has become one of the means to help them meet atleast few demands of employers. Many software development teams have designed tools with

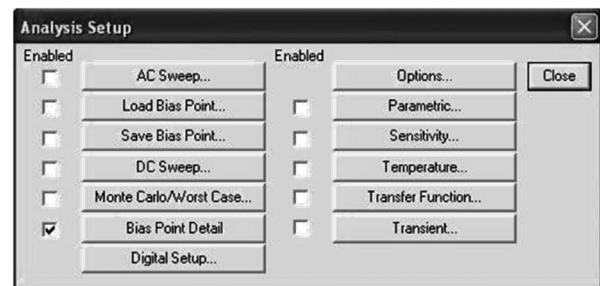


Fig. 1. Analysis Setup in PSpice Schematics

attractive features like schematics, simulation, mathematical modelling, computation, various types of analysis, PCB design and so on. The authors have identified schematics and various types of analysis as fundamental features of computer simulation tool required for CA, AEC and DEC courses. The course instructor has to plan for the effective delivery of the course. Course delivery method chosen should be such that it aid students to do detailed circuit analysis whenever required.

A survey is done to identify a computer simulation tool satisfying the above said features for effective Teaching-Learning process of above mentioned mandatory courses. From Table I, it can be observed that PSpice meets one most important criterion i.e. Industry Standard. Hence to make students industry ready PSpice simulator can be chosen.

3. PSpice Simulator

PSpice (Personal Computer Simulation Program with Integrated Circuit Emphasis) is a circuit analysis tool from OrCAD that allows the user to simulate a circuit and extract vital voltages and currents. The key features of PSpice are:

- It is an industry standard simulator.
- It allows simulation of mixed mode circuits
- Less number of steps is involved in simulating a circuit.
- New Model Editor interface is available.
- Extensive online documentation is also provided.

PSpice is a proprietary software and hence students can access it only during college hours. This is a hindrance for students to carry out the detailed analysis of electronic circuits after college hours. However OrCAD supports student community by providing a Lite/ student version of OrCAD technologies that provides capture and PSpice tools required for course work and labs to get prepared for

academic and career success [3]. The different types of analysis that can be performed using PSpice Evaluation Version is shown in Fig.1.

4. Proposed Methodology

Nowadays it is common practice to teach courses using different simulation tools. However use of different simulation

tool curbs an opportunity of exploring it to the maximum extent. Hence authors propose use of single Industry Standard software for effective Teaching-Learning process of all possible electronic circuit courses in the early stage of academics. The objectives behind proposed methodology are:

- Integration of fundamental mandatory electronic circuit courses in the early stage of engineering education.
- Encouraging students to become self-dependant in analysis of various electronic circuits.

The challenging part here is to identify the training requirements for course instructors to help them get expertise in usage of simulation tool. Authors propose use of Industry Standard simulation tool PSpice for courses like CA, AEC and DEC which are commonly studied during third semester of engineering. PSpice can be used as an aid for class room teaching as well as in laboratories. Collective assessment can be done for these courses by motivating students to apply these concepts in course project. Course instructors can identify various blocks of course project for assessment of their course concepts. Proper communication is necessary and essential among course instructors to take care that students are not heavily loaded with the given task.

5. Illustrations

The different types of analysis reflected in this paper are bias point, AC sweep, DC sweep and transient analysis. In this section authors highlight the relevance of these analyses [4].

- Several of the analyses (small-signal transfer, DC sensitivity, and frequency response) depend upon the bias point calculation. Because so many analyses use the bias point, PSpice calculates this automatically. PSpices bias point calculation computes initial states of analog components.

- Transient analysis is commonly performed on digital circuits. The transient analysis starts using a known solution - the bias point. It then uses the most recent solution as the first guess for each new time point.
- The AC sweep analysis in PSpice is a small signal frequency domain analysis that can be used to observe the frequency response of any circuit at its bias point.
- The DC sweep analysis causes a DC sweep to be performed on the circuit. DC sweep allows you to sweep a source (voltage or current), a global parameter, a model parameter, or the temperature through a range of values.

A. Circuit Analysis

Circuit analysis concepts and network theorems are pillars for analysing any complex electronic circuits. The knowledge of various dependent sources viz Voltage Controlled Voltage Source (VCVS), Voltage Controlled Current Source (VCCS), Current Controlled Voltage Source (CCVS), Current Controlled Current Source (CCCS) can be applied in analysing non-linear semiconductor devices like BJT, MOSFET, IGBT. Fig.2 shows a network with CCVS. The potential at different nodes and current through numerous circuit elements can be determined by bias point analysis.

Table 1. Comparison Of Different Computer Simulation Tools

Software/Features	PSpice	SEQUEL	eSim
Schematic	✓	✓	✓
Simulation	✓	✓	✓
PCB editing	✓		✓
OS platform	Windows	Windows/ Linux	Windows/ Linux
Open source		✓	✓
Industry Standard	✓		

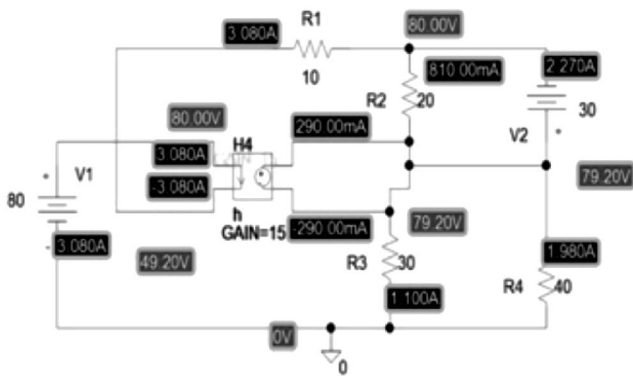


Fig. 2. PSPice schematics of DC network with CCVS

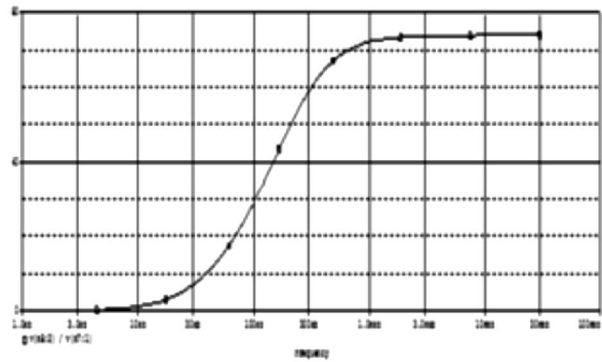


Fig. 4. Frequency response of BJT amplifier

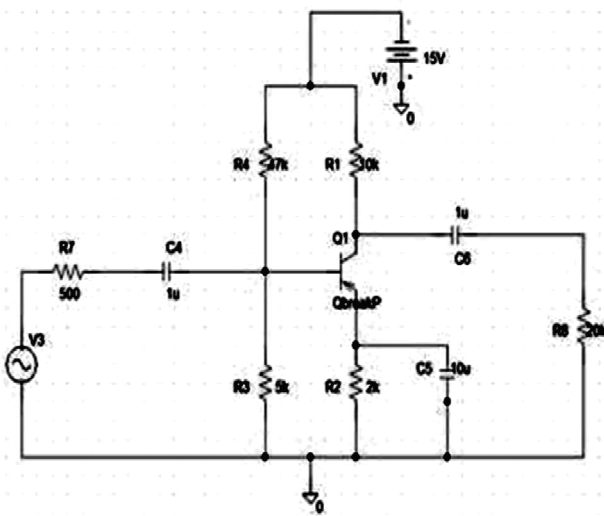


Fig. 3. PSPice schematics of BJT amplifier circuit

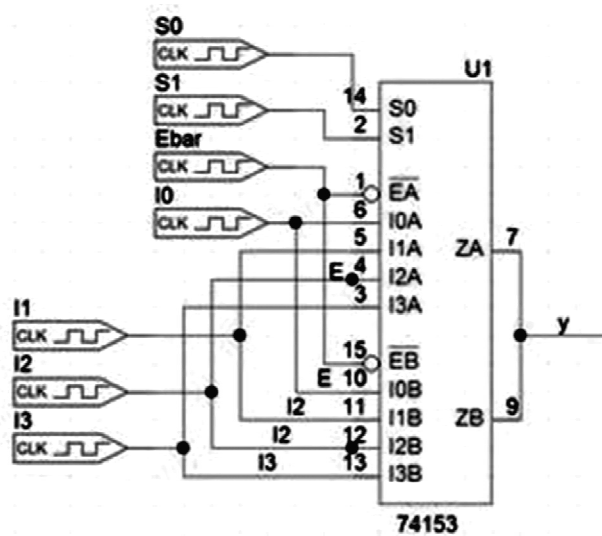


Fig. 5. Logic circuit of 4 to 1 line multiplexer

B. Analog Electronic Circuits

Knowledge developed in analysing network in CA course can be assimilated in analysing typical analog circuits in AEC course. Fig.3 gives PSPice schematics of BJT amplifier. Fig.4 portrays its frequency response after performing AC Sweep analysis in PSPice simulator. Frequency response helps in comprehending information like gain and bandwidth of an amplifier.

C. Digital Electronic Circuits

Digital Electronic Circuits lay a great foundation for the study of Microprocessor/ Microcontroller. Transient analysis is commonly performed for digital circuits.

1) 4 to 1 line multiplexer: Multiplexing means

transmitting a large number of information units over a smaller number of channels or lines. A digital multiplexer is a combinational circuit that selects binary information from one of many input lines and directs it to a single output line [5]. Fig.5 shows logic diagram of 4:1 multiplexer and its corresponding timing diagram is given in Fig.6

With the help of this simulation result, students were able to appreciate the following important concepts [2]:

- When enable (E) pin is active high, multiplexer is activated.
- The control inputs select one of the data inputs and connect it to the output one at a time.

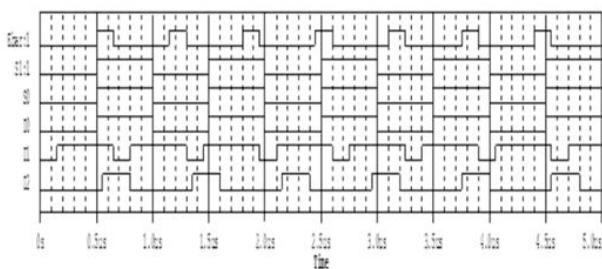


Fig. 6. Timing diagram of 4 to 1 line multiplexer

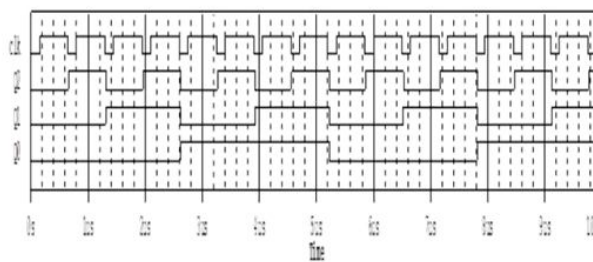


Fig. 8. Timing diagram of a 3 bit synchronous counter

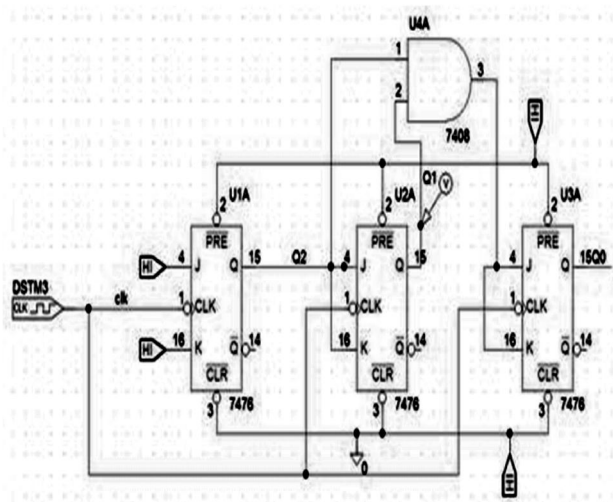


Fig. 7. Logic circuit of a 3 bit synchronous counter

2) 3 bit synchronous counter:

A sequential circuit that goes through a prescribed sequence of states upon the application of input pulses is called a counter. Counters are found in almost all equipment containing digital logic. They are used for counting the number of occurrences of an event and are useful for generating timing sequences to control operations in a digital system [2]. Logic diagram of 3 bit synchronous counter and its timing diagram are shown in Fig.7 and Fig.8 respectively. With an aid of simulation students were able to comprehend the following:

- Recognise the role of clock in sequential circuits.
- The count starts repeating from 7th negative edge of the clock.

6. Possible Attainment Of Program Outcomes

Outcome-based education is an approach to education in which decisions about the curriculum are driven by the exit learning outcomes that the students

should display at the end of the course. Outcome-based education can be summed up as results-oriented thinking and is the opposite of input-based education where the emphasis is on the educational process and where and whatever may be the result is accepted [7]. Table II comprehends the list of possible Program Outcomes that can be attained for the proposed methodology. Comparison of attainment in Program Outcomes in conventional method and proposed method is presented in Table III, where degree of compliance are L: Low M: Medium

7. Conclusion

In this paper authors have put forth the advantages of using a Industry Standard computer simulation tool for various mandatory electronic courses of engineering. Typical examples quoted in section reveal use of PSpice for CA, AEC, and DEC courses. Numerous other analysis that can be performed on electronic circuits are parametric, temperature, worst case and transfer function analysis. It is evident that proposed method-ology helps in enhancing fundamentals, thus laying strong foundation for primary embedded system design. Integration of these fundamentals can be done using common assignment for above mentioned courses. This technique can be further extended for theme based/capstone project work with an additional demand of new knowledge. Various program outcomes that can be achieved are also discussed.

Acknowledgment

The authors would like to express their gratitude to Dr. Ashok Shetter, Vice-Chancellor, KLE Technological University and Dr. P. G. Tewari, Principal, B. V. Bhoomaraddi College of Engineering and Technology, Hubballi for their continuous encouragement. Authors would also like to thank Dr A. B. Raju, H O D Department of Electrical and Electronics Engineering for his support.

References

Vol.51,NO.1, February 2008

- [1] Lyle D. Feisel and Albert J. Rosa “The Role of the Laboratory in Un-dergraduate Engineering Education,”Journal of Engineering Education January 2005
- [2] Anupama R Itagi and Varsha Tatti “Effective Teaching of Digital Electronics for Undergraduate Students Using a Free Circuit Simula-tion Software-SEQUEL.International conference on transformations in Engineering Education 2014.
- [3] Milo D. Koretsky, Danielle Amatore, Connelly Barnes, and Sho Kimura “Enhancement of Student Learning in Experimental Design Using a Virtual Laboratory ”IEEE Transactions on Education ,
- [4] Users Guide, OrCAD PSpice
- [5] M. Morris Mano,“”Digital Logic and Computer Design”,Prentice Hall ofIndia Private limited, New Delhi, 2000.
- [6] A. Ayob, S.A. Osman, M.Z. Omar, N. Jamaluddin, N.T. Kofli, S. Johar “Industrial Training as Gateway to Engineering Career: Experience Sharing,6th International Forum on Engineering Education (IFEE 2012)
- [7] Margery H. Davis “Outcome-Based Education ”JVME 30(3)copyright 2003 AAVMC

Table 2
Description Of Rprogram Outcomes Attained Using Pedagogical Method

Sl.No	Program Outcomes	Competency	Indicators
1	PO1: Engineering knowledge: Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialisation for the solution of complex engineering problems.	1.4 Demonstrate competence in Electrical engineering knowledge	1.4.1 Apply discipline specific laws and pri nciplcs to solve an engineering problem.
2	PO2: Problem analysis: Identify, formulate, research literature, and analyse comp lex engineering problems reaching substantiated con clusions using first principles of mathematics, na tural sciences, and engineering sciences.	2.3 Demonstrate an ability to formulate a solution plan and methodology for an engineering problem	2.3.2 Identify existing processes/solution methods for solving the problem, including justified approximations and assumptions.
3	PO3: Design/Development of Solutions: Design solutions for comp lex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for public health and safety, and cultural, societal, and environmental considerations	3.4 Demonstrate an abilit y to advance an engineering design to defined end state.	3.4.1 Refine a conceptual design into a detailed design within the existing constraints (of the resources)
4	PO5: Modern tool usage: Create, select, and apply appropriate te chniques, resources, and modern engineering and IT tools including prediction and modelling to complex engineering activities wit h an understanding of the limitations.	5.2 D emonstrate an ability to select and a pply discipline specific tools, techniques and resources	5.2.2 Demonstrate pro ficiency in using computing, mathematical, circuit simulation, and document presentation/preparation software. (MATLAB/Scilab, PSPICE, SABER, PROTEUS software tools, AutoDesk, project management tools, Latex and others).

Table 3
Comparison Of Program Outcomes Attained In Conventional And Proposed Method

Sl.No	PO	Conventional method	Proposed method
1	PO1	H	H
2	PO2	L	M
3	PO3	L	M
4	PO5		M