COMPARATIVE STUDY OF VARIOUS CIRCUIT SIMULATION SOFTWARES

A. V. MANCHARKAR
Head, Dept. of physics, New Arts, Commerce and Science College, Parner 414302, INDIA
Email: mancharkar_av@rediffmail.com

Abstract:

Traditionally, electronic circuit design was verified by building prototypes, subjecting the circuit to various stimuli and then measuring its response using appropriate laboratory equipments. Prototype building is somewhat time consuming, but it produces practical experience from which judge the manufacturability of the design. Computer programs that simulate the performance of an electronic circuit provide a simple, cost-effective means of confirming the intended operation prior to circuit construction and of verifying new ideas that could lead to improved circuit performance. Such computer programs have revolutionized the electronics industry, leading to the development of today's high-density monolithic circuit schemes such as VLSI. The use of computer-aided design tools has proven to be invaluable in the development of new technologies and circuit design. Computer simulations have emerged as a very elegant way to aid device, circuit and system design engineers. Proposed work gives emphasis on comparative aspects of commercially available electronic circuit simulation softwares that would be useful for selecting the best software depending on the area of application.

Keywords: Electronic Circuit Design, Simulation software.

1. Introduction:

The term simulation is used to describe any procedure of establishing a model and driving a solution numerically [1]. The spice oriented electronic simulation software plays the most important role in the today's evaluation process of electronic circuits. Spice is a program that simulates electronic circuit on your personal computer. You can view any voltage or current waveform in your circuit. Spice calculates these voltages and currents versus time or versus frequency. SPICE stands for Simulation Program with Integrated Circuit Emphasis.

Traditionally electronic circuit design was verified by building prototypes subjecting the circuits to various stimuli (such as input signals, temperature changes, resistance changes etc.) and then measuring its response using appropriate laboratory equipment. Prototype building is somewhat time consuming but it produces practical experience from which one can judge the manufacturability of the design. Computer program that simulate the performance of an electronic circuit provide a simple cost-effective means of confirming the intended operation prior to circuit construction and of verifying new ideas that could led to improve circuit performance. Such computer programs have revolutionized the electronics industry, leading to the development of today's high-density monolithic circuit schemes such as VLSI.

Spice the defacto industrial standard for computer aided circuit analysis was developed in the early 1970 at the University of California, Berkeley. Although other programs for computer aided-circuit analysis exists and are used by many different electronic groups spice is the most widespread. Until it was largely limited to mainframe computers on a time-sharing basis but today various versions of spice are available for personal computers (PCs). In general for these programs use algorithms slightly different from spice's for performing elevating the spice input syntax to a programming language [2,3,4,5].

Commercially supported versions of spice can be considered to be divided in to two

1. Mainframe version: - Generally mainframe versions of spice are extended to be used by sophisticated integrated –circuit designers. For this type of simulation software requires large amount of computer power to simulate the complex circuits.

2. PC Version: - Is the version of simulation software that requires the single computer to simulate the electronic circuit. In short is called as PSpice (PC version Simulation Program with Integrated Circuit and Emphasis.) has become a common tool of analog simulation and widely used even for some mixed mode circuit design [6].

2. Simulation Concepts:

It allows the designer to construct entirely new circuit without fabricating the actual circuit hard wires, components and PCB's. The circuit performance and its reliability in any circuit, to minimize failure, can be tested. To meet the required standards of a circuit and hence quality instruments, the circuit analysis is performed. In case of any failure –or- problems observed, one can easily redesign it by modifying the very same circuit with in few minutes using highly sophisticated simulation tools. With acquit number of design and redesign interactions on a computer platform. Where it consumes a very small amount if time and no material, the design can be made robust. The simulated circuit can then be subjected to different analysis that is actual tests. The performance and reliability of circuit and instrument definitely shows results of up most levels. Thus it is faster and low cost and less cumbersome process. [7]

While using the simulation softwares the testing process is as follows

- Draw the circuit on the work bench of given software with the help of relative
- Commands provided in that simulation software.
- Select the type of analysis as per our requirement.
- Simulate the circuit. [With different values of components in circuit]
- Display the results of simulation. [Choose the proper result from the number of results, take the values of component and construct the expected hard wire with the help of real electronic component.]

The design center software package has three major interactive programs: Schematic, Spice and Probe. Schematic are a powerful program that let's you build circuits by drawing a window on a screen. Spice analysis the circuit created by schematics and generates voltage and current solutions. Probe is a graphic post processor that allows you to display plots of parameters such as voltage, current, impedance power etc. [8].

3. Study of Softwares:

3.1 Multisim:

Multisim is a complete system design tool that offers a large component data base schematic entry, full Analog / Digital spice simulation, VHDL / Verilog HDL design entry FPGA / CPLD Synthesis, RF Capabilities, Post processing features and seamless transfer to PCB lay out packages, such as multiboard, also from electronic workbench. It offers single easy to use graphical interface for your design needs.

Multisim provides all the advanced functionality you need to take design from specification to production and because the program tightly integrates schematic capture, simulation, PCB layout and programmable logic, you can design with confidence knowing that you are free from the integration issues. Often found when exchanging the data between the applications from different vendors.

The following chart gives us the information about various versions, types of analysis & no. of components in multisim.

Module	Personal Version	Professional Version	Power Profess-ional Version
Basic Schematic Capture	,,	,,	,,
Interactive Simulation	,,	,,	,,
Symbol Editor	,,	,,	,,
SPICE Analog / Digital Simulation	,,	,,	,,
Editable Footprint Field	,,	,,	,,
Electro-mechanical Components	,,	,,	,,
DC Operating Point Analysis	,,	,,	,,
AC Analysis	,,	,,	,,
Transient Analysis	,,	,,	,,
Fourier Analysis	,,	,,	,,
Noise Analysis	,,	,,	,,
Distortion Analysis	,,	,,	,,
DC Sweep Analysis	,,	,,	,,
AC and DC Sensitivity Analysis	,,	,,	,,
Virtual Instruments	,,	9	11
Component Database and Editor	standard with 6000 parts	standard with 12000 parts	standard with 16000 parts
Model Expansion Package	Optional	Optional	,,
SPICE Import		,,	,,
Distortion Analysis Instruments		,,	,,

Virtual Wiring		,,	,,
Menu-driven Simulation from		,,	,,
Netlist (without schematic)		,,	,,
Multiple Circuit Windows		,,	,,
Parameter Sweep Analysis		,,	,,
Temperature Sweep Analysis		,,	,,
Pole Zero Analysis		,,	,,
Transfer Function Analysis		,,	,,
Worst Case Analysis		,,	,,
Monte Carlo Analysis		,,	,,
Trace Width Analysis		,,	,,
Component Search Engine		standard	advanced
Bill of Material		standard	advanced
VHDL		optional	design / debug and simulation
Project / Team Design Module		optional	,,
RF Module		optional	,,
Analog & Digital Model Maker		optional	,,
Code Modeling			,,
Postprocessor			,,
Batched Analysis			,,
Nested Sweep Analysis			,,
User Defined Analysis			,,
PSpice Import		,,	,,
Network version		,,	,,
			[9]

3.2 Circuit Maker:

With just a minimum of electronics theory, you can successfully use Circuit Maker to design and simulate circuits. For beginners, Circuit Maker is perfect for learning and experimenting with electronics and circuit design. For advanced users, Circuit Maker's powerful analyses provide a sophisticated environment for testing and trying all the "what if" scenarios for your design. Best of all, you can accomplish more in less time than traditional prototyping methods. Circuit maker is the most powerful, easy to use schematics capture and simulation tools in its class it provides the features of high-end design software at a function of a cost. Using it advanced schematic capabilities, you can design electronic circuits and output net lists for tracks maker and other PCB design tools and auto routers you can also performs fast and accurate simulations of digital analog or mixed analog / digital circuits using circuit makers Berkeley SPICE 3P5 / XSpice based simulator. [10]

By using the circuit maker simulation software we can performs the following analysis.

- DC Analysis.
- AC Analysis.
- Transient / Fourier
- Noise Analysis.
- Transfer function
- Parameter sweep
- Temperature sweep
- Monto Carlo

Analysis data is saved in raw file.

1. Node voltage and supply current.
2. Node voltage, supply and device current.
3. Node voltage, supply device current and power.
4. Node voltage, supply device current subscript VARS.

This software provides us the facility for to design the circuit by two ways by using the total hot keys and by using the software window. With the help of this window we can take the required value component and keep it in to the designed circuit or for to design the circuit [11].

3.3 Tina:

Tina is one of the complete electronic software that provides us the following analysis.

- DC analysis:
  * Calculate nodal voltage
  * Table of DC results.
  * DC transfer characteristics.
  * Temperature analysis.
- AC analysis:
  * Calculate nodal voltage
  * Table of AC results.
  * AC transfer characteristics.
  * Phaser diagram.
  * Time function.
  * Network analysis.
- Transient analysis:
- Fourier analysis:
  * Fourier series.
  * Fourier spectrum.
- Symbolic analysis:
  * DC result.
  * Semi symbolic DC results.
  * AC results.
  * Semi symbolic AC results.
  * Poles and zeros.
  * Semi symbolic transient.
  * AC transfer.
  * Semi symbolic AC transfer.
- Noise analysis:

- Optimization: * DC optimization analysis.
  * DC optimization transfer analysis.
  * Temperature analysis.
  * Optimization analysis.
  * AC optimization transfer analysis.
- Trace mode analysis.
- Fault finding analysis: Exam manager is one of the great features of this simulation software. [12]

Tina version -7 provides us the Analog, Digital, Mixed, VHDL, MCU, Interactive, symbolic, Spice, Tolerance, S-parameter, Network Analysis etc. Tina library includes over 20,000 components that can be modified by users. In Tina we found Analog, Digital, mixed - signals and RF Devices.

Student's version of Tina system requirements – Windows 98 / Windows NT / 2000 / XP or later (Vista) IBM PC / AT Compatible Pentium 120 or better computer. 16 MB RAM, 50 MB Hard Disk Spice Tina has provide the facility to check your results in Tina's training mode, which can help get ready of your fear examples. [13].

Three different simulation software's and Spice's observed at different sites e.g. various versions of Spice such as PSpice, LTspice, TOPSPICE etc. [14]

4. Conclusion:

There are several simulation software's are present now a days. Each software has its powerful simulation engine which helps us to save the time of simulation each has its own identity about the design of components, design facility, number of components, types of analysis and different spice orientation. In above mentioned simulation softwares Circuit Maker is easy for to handle the new comers. Circuit Maker provides Berkeley SPICE 3P5 / XSpice based simulator. Multisim is one of the software that provides us the different versions as per our requirement with different no. of components generally its professional version is used in industrial sector. TINA is one of the simulation software that contains highest number of components. It helps to the teachers for to improve the basic concepts in electronics in laboratory due to its feature of fault finding analysis & exam manager. In short the concepts of spice, simulation & simulation software's should become familiar to for the teachers & students. This improves the interest in study of electronics,

5. References:

[1] Geoffrey Gordon, System simulation, 2nd Ed. Feb. 2003.

[2] M.H. Rashid, Micro Electronic Circuit Analysis and Design. PWS Company. University of Florida, 1999.

[3] M. H. Rashid, SPICE For Circuits And Electronics Using PSPICE Prentice Hall of India, Second Edition, 2004.

[4] P. W. Tunenga. SPICE: A Guide to circuit simulation and analysis using PSPICE, Prentice Hall. Englewood Cliffs, N. J. USA, 1998.

[5] A. V. Mancharkar. Development, Comparison And Simulation Studies of Universal Sensor Interface. Dec. 2005.

[6] Martino' Hara, Modeling board Level DC – DC converts in SPICE (Electronic Product Design), July 1998.

[7] Walunj A. K., Mancharkar A. V. and Shaligram A. D., PSpice Simulation for Performance and Reliability Testing of Sensor Signal Conditioning Circuits, J. Instrument. Soc. India, 30 (20), pp 65 – 74, 2001

[8] Mancharkar A. V. and Behere S. H., PSpice Simulation for Performance Testing of Signal Conditioning Circuits for Resistive Temperature Sensors, Proceedings (section of physical sciences) of 91st The Indian Science Congress, Chandigarh, pp 41 – 42., 2004.

[9] Multisim [TM] & Electronic Workbench [TM] Version 6.22 Demo.

[10] Microcode Engineering Support http://www. Microcode. com.

[11] Circuit Maker Students Version 6.2c 1999 Portel Technology Inc.

[12] TINA™ for Windows. The Complete Lab Version 6.01.01.DT.

[13] TINA Version – 7. [www.tina.com], [www.designsoftware.com]

[14] eCircuit Centre 2003–2005 [http://ecircuitcenter.com/demos.htm].