International Journal of Scientific Research in Science, Engineering and Technology
Print ISSN: 2395-1990 | Online ISSN : 2394-4099 (www.ijsrset.com)

Why Simulate?

Dr. Sanjay K. Tupe[1], V.P. Shirsat[1]
[1]Department of Physics, Kalikadevi Arts, Commerce and Science College, Shirur (K.), Dist.- Beed. 413249, Maharashtra, India

Article Info
Volume 9, Issue 5
Page Number: 153-156

Publication Issue :
July-August-2021

Article History
Accepted : 02July2021
Published:25 July, 2021

ABSTRACT

This paper addresses why simulation is necessary and various types of circuit analysis. Traditionally electronic circuit design was verified by building prototypes, subjecting the circuits to various stimuli and measuring its response using appropriate laboratory equipment's. It is time consuming, instead of this if we use various SPICE software's, we can perform the number of analysis of the same circuit virtually. Virtual results are very close to the actually build circuits. It gives new ideas that could lead to improve the circuit performance.

Keywords: - SPICE, Circuit Analysis, Virtual Components

I. INTRODUCTION

SPICE is a great tool to learn a lot in a short time. Also, busy lives and limited budgets can make experimenting with real parts and expensive equipment nearly impossible. What may take you an hour to wire up in the lab to get a minor concept could be covered in a few minutes with SPICE. For example, how does an amplifier's gain vary with bandwidth? Before the circuit parts were even collected, you can get hands on experience with the gain-bandwidth tradeoff. While text and equations tell you the story, a simulation can clarify the concept and drive it home.

It's true getting a circuit to work as you envisioned can be fun and satisfying. Trying one more RC combination can be addicting as you optimize a circuit. Simulation gives you an open-ended sense of play, a set of circuit blocks ready to be combined in some interesting or useful way. There's a challenge in creating a SPICE model for an electrical or non-electrical component in your system. It's easy to get lost in a circuit adventure. What better way to learn the art and develop a passion for circuit design?

Measuring some circuit voltages and currents can appear like a mission impossible. Here are some difficulties simulation can avoid. Some measuring equipment may load your circuit producing misleading results. Other measurements may require special test equipment you don't have or can't afford. Still others may be dangerous (high voltage or current measurements) or may inadvertently destroy the real circuit. [1,2,3,4,5]

II. TYPES OF ANALYSIS

DC Analysis: -The dc analysis portion of SPICE determines the dc operating point of the circuit with inductors shorted and capacitors opened. The dc analysis options are specified on the .DC, .TF, and .OP control lines. A dc analysis is automatically performed prior to a transient analysis to determine the transient initial conditions, and prior to an ac small-signal analysis to determine the linearized, small-signal models for nonlinear devices. If requested, the dc small-signal value of a transfer function, input resistance, and output resistance is also computed as a part of the dc solution. The dc analysis can also be used to generate dc transfer curves: a specified independent voltage or current source is stepped over a user-specified range and the dc output variables are stored for each sequential source value.

AC Small-Signal Analysis: -The ac small-signal portion of SPICE computes the ac output variables as a function of frequency. The program first computes the dc operating point of the circuit and determines linearized, small-signal models for all of the nonlinear devices in the circuit. The resultant linear circuit is then analyzed over a user-specified range of frequencies. The desired output of an ac small- signal analysis is usually a transfer function. If the circuit has only one ac input, it is convenient to set that input to unity and zero phase, so that output variables have the same value as the transfer function of the output variable with respect to the input.

Transient Analysis: -The transient analysis portion of SPICE computes the transient output variables as a function of time over a user-specified time interval. The initial conditions are automatically determined by a dc analysis. All sources which are not time dependent (for example, power supplies) are set to their dc value. The transient time interval is specified on a .TRAN control line.

Pole-Zero Analysis: -The pole-zero analysis portion of SPICE computes the poles and/or zeros in the small-signal ac transfer function. The program first computes the dc operating point and then determines the linearized, small-signal models for all the nonlinear devices in the circuit. This circuit is then used to find the poles and zeros of the transfer function.

Two types of transfer functions are allowed: one of the form (output voltage)/(input voltage) and the other of the form (output voltage)/(input current). These two types of transfer functions cover all the cases and one can find the poles/zeros of functions like input/output impedance and voltage gain. The input and output ports are specified as two pairs of nodes. The pole-zero analysis works with resistors, capacitors, inductors, linear-controlled sources, independentsources, BJTs, MOSFETs, JFETs and diodes. Transmission lines are not supported. The method used in the analysis is a sub-optimal numerical search.

Small-Signal Distortion Analysis: -The distortion analysis portion of SPICE computes steady-state harmonic and intermodulation products for small input signal magnitudes. If signals of a single frequency are specified as the input to the circuit, the complex values of the second and third harmonics are determined at every point in the circuit. If there are signals of two frequencies input to the circuit, the analysis finds out the complex values of the circuit variables at the sum and difference of the input frequencies, and at the difference of the smaller frequency from the second harmonic of the larger frequency.

Distortion analysis is supported for the following nonlinear devices: diodes (DIO), BJT, JFET, MOSFETs (levels 1, 2, 3, 4/BSIM1, 5/BSIM2, and 6) and MESFETS. All linear devices are automatically supported by distortion analysis. If there are switches present in the circuit, the analysis continues to be accurate provided the switches do not change state under the small excitations used for distortion calculations.

Sensitivity Analysis: -Spice will calculate either the DC operating-point sensitivity or the AC small-signal sensitivity of an output variable with respect to all circuit variables, including model parameters. Spice calculates the difference in an output variable by perturbing each parameter of each device independently. Since the method is a numerical approximation, the results may demonstrate second order affects in highly sensitive parameters, or may fail to show very low but non-zero sensitivity. Further, since each variable is perturb by a small fraction of its value, zero-valued parameters are not analyzed.

Noise Analysis: -The noise analysis portion of SPICE does analysis device-generated noise for the given circuit. When provided with an input source and an output port, the analysis calculates the noise contributions of each device (and each noise generator within the device) to the output port voltage. It also calculates the input noise to the circuit, equivalent to the output noise referred to the specified input source. This is done for every frequency point in a specified range. The calculated value of the noise corresponds to the spectral density of the circuit variable viewed as a stationary Gaussian stochastic process.

After calculating the spectral densities, noise analysis integrates these values over the specified frequency range to arrive at the total noise voltage/current (over this frequency range). This calculated value corresponds to the variance of the circuit variable viewed as a stationary Gaussian process. [6, 7, 8, 9, 10, 11]

III. CONCLUSION

The sky is the limit with electronic devices and topologies.You can start with some high-level functional blocks. As the design takes shape, fill in the details with components until Presto. Your creative synthesis has given birth to a circuit ready for actual prototype and further verification.

Note: -Now a day due to Covid -19 Pandemic condition education field is totally affected in comparison with other fields. So simulation is necessary for to give the realistic experience to the students. For this teachers should be techno friendly and sufficient technical infrastructure should be available.

IV. REFERENCES

[1]. Mancharkar A. V. Ph.D. Thesis, Development, comparison and simulation studies of universal sensor interface, December 2005.

[2]. Attia, John Okyere, PSPICE and MATLAB for Electronics: An Integrated Approach, Boca Raton, FL: CRC Press, 2002.

[3]. Roy W. Goody, OrCAD PSpice for windows Volume I: DC and AC Circuits, 3d ed. Upper Saddle River, New Jersey: Prentice Hall, 2000.

[4]. Roy W. Goody, OrCAD PSpice for windows Volume II: Devices, Circuit and Operational Amplifiers, 3d ed. Upper Saddle River, New Jersey: Prentice Hall, 2000.

[5]. Mark E. Herniter, Schematic Capture with Cadence PSpice, Upper Saddle River, New Jersey, Prentice Hall, 2001.

[6]. Muhammad H. Rashid, Introduction to PSpice Using OrCAD for Circuits and Electronics, third edition, Prentice –Hall of India Private Limited, New Delhi-110 001, 2006.

[7]. Lavagno, Martin, Scheffer, Electronic Design Automation For Integrated Circuits Handbook, ISBN 0-8493-3096-3, 2006.

[8]. Dirk Jansen., Kluwer, The Electronic Design Automation Handbook, Academic Publishers, ISBN 1-4020-7502-2, 2003.

[9]. M. H. Rashid, "SPICE For Circuit And Electronics Using PSPICE", Prentice Hall of India, second Edition, 2004.

[10]. R. M. Kielokwski, “Inside SIPCE overcoming the obstacles of circuit simulation”, McGraw Hill, Inc, 1994.

[11]. A. K. Walung, A. V. Mancharkar and A. D. Shaligram, “PSPICE Simulation performance and Relability testing of sensor signal conditioning circuits”, Journal of Instruments Society of India, vol. 30(2)65, 2000.