INTERNATIONAL ATOMIC ENERGY AGENCY
UNITED NATIONS EDUCATIONAL, SCIENTIFIC AND CULTURAL ORGANIZATION
INTERNATIONAL CENTRE FOR THEORETICAL PHYSICS
I.C.T.P., P.O. BOX 586, 34100 TRIESTE, ITALY, CABLE CENTRATOM TRIESTE

The United Nations University

SMR/748 - 5

ICTP-INFN-UNU-MICROPROCESSOR LABORATORY
THIRD COURSE ON BASIC VLSI DESIGN TECHNIQUES
2 November - 16 December 1994

SPICE

Andres CICUTTIN
Microprocessor Laboratory
ICTP
P.O. Box 586
34100 Trieste
Italy

These are preliminary lecture notes, intended only for distribution to participants.

BRIEF INTRODUCTION TO SPICE

SPICE is a family of circuit simulators originally developed at the University of California at Berkeley. As members of this family there are some commercial versions like HSPICE, PSpice, etc. Now SPICE is practically a standard tool for analog circuit simulation. In this course we will use Spice-3e1.

SPICE is a general-purpose circuit simulation program, which accepts a description of a circuit and provides several forms of accurate and detailed simulation, including small signal ac, dc, and time-domain transient solutions. Circuit may contain passive devices (resistors, capacitors, inductors, mutual inductors, etc.), Semiconductor devices (diodes, BJT's, JFET's, MESFET's and MOSFETs), different kinds of voltage and current sources, lossless and lossy transmission lines, switches and uniform distributed RC lines.

The circuit to be analysed is described to SPICE by a set of **element lines**, which must define:

- Circuit topology
- Element values

And a set of **control lines** that defines:

- Model parameters
- Run controls

Circuit Topology

It is basically a description of the network, with components and their connections. To describe it, each element in the circuit must be specified by an element line that contain:

- The element name (where the first letter specifies the element type)
- The different current and voltage sources (time constant or generating any stimulus to the circuit)
- The circuit nodes to which each element is connected
- The values of the parameters required by the mathematical model that defines completely the (modelled) electrical behaviour

Element Values

These are the numbers that characterise quantitatively the element (for example: the capacity in a capacitor, the AREA in a MESFET or a JFET, etc.). Usually are the variables handled by the designers.

Model Parameters

These are the numbers needed by the model to characterise qualitatively the type of element. In the semiconductor devices, are fixed by the technological process of fabrication (for example the threshold voltage in a MESFET, layers sheet resistances, etc.).

Run Controls

These are to specify the type of analyses within the circuit description file (ac, dc, noise, etc.), and to select the options to control the numerical procedure (iteration limits, error tolerances, numerical integration method, etc.).

Precision and Limitations

With respect to precision and limitation SPICE is not different from any other simulation tools in the sense that it has two principal sources of error:

1) **The mathematical models**: since they are the responsible to reflect the principal aspect of what will be simulated, the simulation cannot offer to us valid results beyond the validity limit of the model (for example the parasitic capacitance of an inductor in the high frequency domain require a more complex model),(experimental vs. simulated curves).

2) **The numeric methods**: to solve the equations that relate the variables of interest, or to find the initial conditions for that equations, limits the accuracy in the results. This point is partially controlled through the special options (number of iteration, tolerances, etc.), normally is not critical.

Examples and Applications

In order to illustrate the capabilities of SPICE we start directly with some examples, after that we will use it to analyse and characterise an elemental logic gate (not) in CMOS and GaAs technology showing some problems connected to **Very Large Scale of Integration**.

First hour

Introduction to SPICE

- A simple ohmic circuit
- Low-Pass and High Pass passive Filter
- Low-Pass and High-Pass Active Filter
- Band-Pass filter
- Bode Diagrams

Second hour

DC Characteristics of MOS and GaAs transistors

- NMOS, PMOS and D-MESFET: Transconductance, output conductance, linear and saturation region.
- Two inverters: CMOS and BFL (dc transfer curve)

Third hour

Comparison of Two Logical Families: CMOS (Si) and BFL (GaAs)

- Logic swing and noise margin
- Power consumption
- Rise and fall time, delay propagation time
- Speed-power product (and the power/area problem)
- Fan-out
- Ring Oscillators

Bibliography

1. B. Jhonson, T. Quarles, A.R. Newton, D.O. Pederson, A.Sangiovanni-Vicentelli
SPICE Version 3e User's Manual. April 1, 1991.
Department of Electrical Engineer and Computing Sciences. University of California, Berkeley. CA 94720.

2. Chattergy, R.
Spicey circuits : elements of computer-aided circuit analysis / Rahul Chattergy. Boca Raton, Fla. : CRC Press, c1992.
Series title: CRC Press computer engineering series.

3. Connelly, J. Alvin.
Macromodeling with SPICE / J. Alvin Connelly, Pyung Choi. Englewood Cliffs, N.J. : Prentice Hall, c1992.

4. Keown, John (John L.)
PSpice and circuit analysis / John Keown. New York : Merrill ; Toronto : Collier Macmillan Canada ; New York : Maxwell Macmillan International Pub. Group, c1991.
Series title: Merrill's international series in engineering technology.

5. Nagel, Laurence W.
SPICE2 : a computer program to simulate semiconductor circuits / by Laurence W. Nagel. [Berkeley] : Electronics Research Laboratory, College of Engineering, University of California, Berkeley, 1975.
Series title: University of California, Berkeley. Electronics Research Laboratory Memorandum ; no. ERL-M520.

6. Nilsson, James William.
Introduction to PSpice. Supplement to Electric circuits, 4th edition / James W. Nilsson, Susan A. Riedel. Reading, Mass. : Addison-Wesley Pub. Co., c1993.

7. Rashid, M. H.
SPICE for circuits and electronics using PSpice / Muhammad H. Rashid. Englewood Cliffs, N.J. : Prentice Hall, c1990.

8. Rashid, M. H.
SPICE for power electronics and electric power / Muhammad H. Rashid. Englewood Cliffs, N.J. : Prentice Hall, c1993.

9. Semiconductor device modeling with SPICE / Paolo Antognetti, editor, Giuseppe Massobrio, coeditor. New York : McGraw-Hill, c1988.

10. Thorpe, Thomas W.
Computerized circuit analysis with SPICE : a complete guide to SPICE, with applications / Thomas W. Thorpe. New York : Wiley, c1992.

11. Tuinenga, Paul W.
SPICE : a guide to circuit simulation and analysis using PSpice / Paul W. Tuinenga. Englewood Cliffs, N.J. : Prentice Hall, c1988.

12. Tuinenga, Paul W.
SPICE : a guide to circuit simulation and analysis using PSpice / Paul W. Tuinenga. 2nd ed. Englewood Cliffs, N.J. : Prentice Hall, c1992.

13. Vladimirescu, Andrei.
The SPICE book / Andrei Vladimirescu. New York : J. Wiley, c1994.