C. Nguyen

INFORMATION ON ANALOG CIRCUIT SIMULATION USING H-SPICE

A. Introduction

SPICE is a general-purpose circuit simulation program for nonlinear dc, nonlinear transient, and linear ac anayses. Circuits may contain resistors, capacitors, inductors, mutual inductors, independent voltage and current sources, four types of dependent sources, transmission lines, switches, and the five most common semiconductor devces: diodes, BJTs, JFETs, MESFETs, and MOSFETs.

SPICE has built-in models for the semiconductor devices, and the user need specify only the pertinent model parameter values. The model for the BJT is based on the integral charge model of Gummel and Poon; however, if the Gummel- Poon parameters are not specified, the model reduces to the simpler Ebers-Moll model. In either case, charge storage effects, ohmic resistances, and a current-dependent output conductance may be included. The diode model can be used for either junction diodes or Schottky barrier diodes. The JFET model is based on the FET model of Shichman and Hodges. Several MOSFET models are implemented, including: MOS1, which models the basic square-law I-V characteristic, and is useful for matching hand analysis (i.e., for checking your analytical answers); MOS2[1], which is a more accurate analytical model that includes second order effects; MOS3 [1], which is a semi-empirical model that better models actual devices; MOS4 [2,3], which is the BSIM (Berkeley Short-channel IGFET Model) that models more modern short channel devices; and several additional models capable of modeling the latest generation of submicron transistors. MOS2, MOS3, and MOS4 include second-order effects such as channel length modulation, subthreshold conduction, scattering limited velocity saturation, small-size effects, and charge-controlled capacitances.

There are 4 types of analysis you will use most often in this class:

B.1. DC analysis

The dc analysis portion of SPICE determines the dc operating point of the circuit with inductors shorted and capacitors opened. 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 non-linear devices. 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.

<u>General Form:</u>

.DC SRCNAM VSTART VSTOP VINCR (SRC2 START2 STOP2 INCR2)

<u>Examples:</u>

.DC VIN 0.25 5.0 0.25

.DC VDS 0 10 .5 VGS 0 5 1

.DC VCE 0 10 .25 IB 0 10U 1U

This line defines the dc transfer curve source and sweep limits. SRCNAM is the name of an independent voltage or current source. VSTART, VSTOP, and VINCR are the startng, final, and incrementing values respectively. The first example will cause the value of the voltage source VIN to be swept from 0.25 Volts to 5.0 Volts in increments of 0.25 Volts. A second source SRC2 may optionally be specified with associated sweep parameters. In this case, the first source will be swept over its range for each value of the second source. This option can be useful for obtaining semiconductor device output characteristics.

B.2. Operating Point Analysis

General Form:

.OP

The inclusion of this line in an input file will force SPICE to determine the dc operating point of the circuit with inductors shorted and capacitors opened. Note: 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. SPICE performs a dc operating point analysis if no other analyses are requested.

B.3. 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 (voltage gain, transimpedance, etc). 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.

General form:

.AC DEC ND FSTART FSTOP

.AC OCT NO FSTART FSTOP

.AC LIN NP FSTART FSTOP

Examples:

.AC DEC 10 1 10K

.AC DEC 10 1K 100MEG

.AC LIN 100 1 100HZ

DEC stands for decade variation, and ND is the number of points per decade. OCT stands for octave variation, and NO is the number of points per octave. LIN stands for linear variation, and NP is the number of points. FSTART is the starting frequency, and FSTOP is the final frequency. If this line is included in the circuit file, SPICE will perform an ac analysis of the circuit over the specified frequency range. Note that in order for this analysis to be meaningful, at least one independent source must have been specified with an ac value.

B.4. 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.

General form:

.TRAN TSTEP TSTOP <TSTART <TMAX>> <UIC>

Examples:

.TRAN 1NS 100NS

.TRAN 1NS 1000NS 500NS

.TRAN 10NS 1US UIC

TSTEP is the printing or plotting increment for line printer output. For use with the post-processor, TSTEP is the suggested computing increment. TSTOP is the final time and TSTART is the initial time. If TSTART is omitted, it is assumed to be zero. The transient analysis always begins at time zero. In the interval <zero, TSTART>, the circuit is analyzed (to reach a steady state), but no outputs are stored. In the interval <TSTART, TSTOP>, the circuit is analyzed and outputs are stored. TMAX is the maximum step size that SPICE will use (by default the program chooses either TSTEP or (TSTOP-TSTART)/50.0, whichever is smaller. TMAX is useful when one wishes to guarantee a computing interval which is smaller than the printer increment, TSTEP.

UIC (use initial conditions) is an optional keyword which indicates that the user does not want SPICE to solve for the quiescent operating point before beginning the transient analysis. If this keyword is specified, SPICE uses the values specified using IC=... on the various elements as the initial transient condition and proceeds with the analysis. If an .IC line has been given, then the node voltages on the .IC line are used to compute the intitial conditions for the devices. Look at the description on the IC line for its interpretation when UIC is not specified.

C.SPICE input file structure - Circuit description

C.1. General structure and conventions

The circuit to be analyzed is described to SPICE by a set of element lines, which define the circuit topology and element values, and a set of control lines, which define the model parameters and the run controls. The first line in the input file must be the title, and the last line must be ".END". The order of the remaining lines is arbitrary (except, of course, that continuation lines must immediately follow the line being continued).

Each element in the circuit is specified by an element line that contains the element name, the circuit nodes to which the element is connected, and the values of the parameters that determine the electrical characteristics of the element. The first letter of the element name specifies the element type. The format for the SPICE element types is given in what follows. The strings XXXXXXX, YYYYYYY, and ZZZZZZZ denote arbitrary alphanumeric strings. For example, a resistor name must begin with the letter R and can contain one or more characters. Hence, R, R1, RSE, ROUT, and R3AC2ZY are valid resistor names. Details of each type of device are supplied in a following section.

Fields on a line are separated by one or more blanks, a comma, an equal ('=') sign, or a left or right parenthesis; extra spaces are ignored. A line may be continued by entering a '+' (plus) in column 1 of the following line; SPICE continues reading beginning with column 2.

A name field must begin with a letter (A through Z) and cannot contain any delimiters.

A number field may be an integer field (12, -44), a floating point field (3.14159), either an integer or floating point number followed by an integer exponent (1e-14, 2.65e3), or either an integer or a floating point number followed by one of the following scale factors:

T = 1e12 G = 1e9 Meg = 1e6 K = 1e3 mil = 25.4e-6

m=1e-3 u=1e-6 n=1e -9 p=1e-12 f=1e-15

Letters immediately following a number that are not scale factors are ignored, and letters immediately following a scale factor are ignored. Hence, 10, 10V, 10Volts, and 10Hz all represent the same number, and M, MA, MSec, and MMhos all represent the same scale factor. Note that 1000, 1000.0, 1000Hz, 1e3, 1.0e3, 1KHz, and 1K all represent the same number.

Nodes names may be arbitrary character strings. The datum (ground) node must be named '0'. Note the difference in SPICE3 where the nodes are treated as character strings and not evaluated as numbers, thus '0' and '00' are distinct nodes in SPICE3 but not in SPICE2. The circuit cannot contain a loop of voltage sources and/or inductors and cannot contain a cut-set of current sources and/or capacitors. Each node in the circuit must have a dc path to ground. Every node must have at least two connections except for transmission line nodes (to permit unterminated transmission lines) and MOSFET substrate nodes (which have two internal connections anyway).

C.2. TITLE LINE, COMMENT LINES AND .END LINE

c.2.a: Title lines

<u>Examples:</u>

POWER AMPLIFIER CIRCUIT

TEST OF CAM CELL

The title line must be the first in the input file. Its contents are printed verbatim as the heading for each section of output.

c.2.b: END Line

.END

The "End" line must always be the last in the input file. Note that the period is an integral part of the name.

c.2.c: Comments

<u>General Form</u>

*<any comments>

Example

* RF=1K   Gain should be 100

* Check open-loop gain and phase margin

The asterisk in the first column indicates that this line is a comment line. Comment lines may be placed anywhere in the circuit description. Note that SPICE3 also considers any line with leading white space to be a comment.

C.3. DEVICE MODELS

General Form

.MODEL MNAME TYPE(PNAME1=PVAL1 PNAME2=PVAL2 ...)

Example

.MODEL MOD1 NPN (BF=50 IS=1E-13 VBF=50)

Most simple circuit elements typically require only a few parameter values. However, some devices (semiconductor devices in particular) that are included in SPICE require many parameter values. Often, many devices in a circuit are defined by the same set of device model parameters. For these reasons, a set of device model parameters is defined on a separate .MODEL line and assigned a unique model name. The device element lines in SPICE then refer to the model name.

For these more complex device types, each device element line contains the device name, the nodes to which the device is connected, and the device model name. In addi- tion, other optional parameters may be specified for some devices: geometric factors and an initial condition

MNAME in the above is the model name, and type is one of the following fifteen types:

R   Semiconductor resistor model

C   Semiconductor capacitor model

SW   Voltage controlled switch

CSW   Current controlled switch

URC   Uniform distributed RC model

LTRA   Lossy transmission line model

D   Diode model

NPN NPN BJT model

PNP PNP BJT model

NJF N-channel JFET model

PJF P-channel JFET model

NMOS N-channel MOSFET model

PMOS P-channel MOSFET model

NMF N-channel MESFET model

PMF P-channel MESFET model

Parameter values are defined by appending the parameter name followed by an equal sign and the parameter value. Model parameters that are not given a value are assigned the default values given below for each model type. Models, model parameters, and default values are listed in the next section along with the description of device element lines.

C.4.: SUBCIRCUITS

A subcircuit that consists of SPICE elements can be defined and referenced in a fashion similar to device. The subcircuit is defined in the input file by a grouping of element lines; the program then automatically inserts the group of elements wherever the subcircuit is referenced. There is no limit on the size or complexity of subcircuits, and subcircuits may contain other subcircuits.

c.4.a: .SUBCKT Line

<u>General Form:</u>

.SUBCKT subnam N1 <N2 N3 ...>

<u>Example:</u>

.SUBCKT OPAMP 1 2 3 4

A circuit definition is begun with a .SUBCKT line. SUBNAM is the subcircuit name, and N1, N2, ... are the external nodes, which cannot be zero. The group of element lines which immediately follow the .SUBCKT line define the subcircuit. The last line in a subcircuit definition is the .ENDS line (see below). Control lines may not appear within a subcircuit definition; however, subcircuit definitions may contain anything else, including other subcircuit definitions, device models, and subcircuit calls (see below). Note that any device models or subcircuit definitions included as part of a subcircuit definition are strictly local (i.e., such models and definitions are not known outside the subcircuit definition). Also, any element nodes not included on the .SUBCKT line are strictly local, with the exception of 0 (ground) which is always global.

c.4.b: .ENDS Line

Example:

.ENDS OPAMP

The "Ends" line must be the last one for any subcircuit definition. The subcircuit name, if included, indicates which subcircuit definition is being terminated; if omitted, all sub-circuits being defined are terminated. The name is needed only when nested subcircuit-definitions are being made.

c.4.c: Subcircuit Calls

General Form:

XYYYYYYY N1 <N2 N3 ...> SUBNAM

Example:

X1 2 4 17 3 1 MULTI

Subcircuits are used in SPICE by specifying pseudo-elements beginning with the letter X, followed by the circuit nodes to be used in expanding the subcircuit.

D.Circuit Elements and Models

D.1.: Resistors:

General Form:

RXXXXXXX N1 N2 VALUE

Example:

R1 1 2 100

RC1 12 17 1K

N1 and N2 are the two element nodes. VALUE is the resistance (in ohms) and may be positive or negative but not zero.

D.2.: Capacitors

General Form:

CXXXXXXX N+ N- VALUE <IC=INCOND>

Example:

CBYP 13 0 1UF

COSC 17 23 10U IC=3V

N+ and N- are the positive and negative element nodes, respectively. VALUE is the capacitance in Farads.The (optional) initial condition is the initial (time-zero) value of capacitor voltage (in Volts). Note that the initial conditions (if any) apply 'only' if the UIC option is specified on the .TRAN control line.

D.3.: Independent Sources:

General Form:

VXXXXXXX N+ N- <<DC> DC/TRAN VALUE> <AC <ACMAG <ACPHASE>>>

\+ <DISTOF1 <F1MAG <F1PHASE>>> <DISTOF2 <F2MAG <F2PHASE>>>

IYYYYYYY N+ N- <<DC> DC/TRAN VALUE> <AC <ACMAG <ACPHASE>>>

\+ <DISTOF1 <F1MAG <F1PHASE>>> <DISTOF2 <F2MAG <F2PHASE>>>

Example:

VCC 10 0 DC 6

VIN 13 2 0.001 AC 1 SIN(0 1 1MEG)

ISRC 23 21 AC 0.333 45.0 SFFM(0 1 10K 5 1K)

VMEAS 12 9

VCARRIER 1 0 DISTOF1 0.1 -90.0

VMODULATOR 2 0 DISTOF2 0.01

IIN1 1 5 AC 1 DISTOF1 DISTOF2 0.001

N+ and N- are the positive and negative nodes, respectively. Note that voltage sources need not be grounded. Positive current is assumed to flow from the positive node, through the source, to the negative node. A current source of positive value forces current to flow out of the N+ node, through the source, and into the N- node. Voltage sources, in addition to being used for circuit excitation, are the 'ammeters' for SPICE, that is, zero valued voltage sources may be inserted into the circuit for the purpose of measuring current. They of course have no effect on circuit operation since they represent short-circuits.

DC/TRAN is the dc and transient analysis value of the source. If the source value is zero both for dc and transient analyses, this value may be omitted. If the source value is time-invariant (e.g., a power supply), then the value may optionally be preceded by the letters DC.

ACMAG is the ac magnitude and ACPHASE is the ac phase.The source is set to this value in the ac analysis. If ACMAG is omitted following the keyword AC, a value of unity is assumed. If ACPHASE is omitted, a value of zero is assumed. If the source is not an ac small-signal input, thekeyword AC and the ac values are omitted.

DISTOF1 and DISTOF2 are the keywords that specify that the independent source has distortion inputs at the frequencies F1 and F2 respectively (see the description of the .DISTO control line). The keywords may be followed by an optional magnitude and phase. The default values of the magnitude and phase are 1.0 and 0.0 respectively.

Any independent source can be assigned a time-dependent value for transient analysis. If a source is assigned a time-dependent value, the time-zero value is used for dc analysis. There are five independent source functions: pulse, exponential, sinusoidal, piece-wise linear, and single-frequency FM. If parameters other than source values are omitted or set to zero, the default values shown are assumed. (TSTEP is the printing increment and TSTOP is the inal time.

D.4.: Junction Diodes

General Form:

XXXXXXX N+ N- MNAME <AREA> <OFF> <IC=VD> <TEMP=T>

Examples:

DBRIDGE 2 10 DIODE1

DCLMP 3 7 DMOD 3.0 IC=0.2

N+ and N- are the positive and negative nodes, respectively. MNAME is the model name, AREA is the area factor, and OFF indicates an (optional) starting condition on the device for dc analysis. If the area factor is omitted, a value of 1.0 is assumed. The (optional) initial condition specification using IC=VD is intended for use with the UIC option on the .TRAN control line, when a transient analysis is desired starting from other than the quiescent operating point. The (optional) TEMP value is the temperature at which this device is to operate, and overrides the temperature specification on the .OPTION control line.

D.5.: Bipolar Junction Transistors (BJTs)

General Form:

QXXXXXXX NC NB NE <NS> MNAME <AREA> <OFF> <IC=VBE, VCE> <TEMP=T>

Examples

Q23 10 24 13 QMOD IC=0.6, 5.0

Q50A 11 26 4 20 MOD1

NC, NB, and NE are the collector, base, and emitter nodes, respectively. NS is the (optional) substrate node. If unspecified, ground is used. MNAME is the model name, AREA is the area factor, and OFF indicates an (optional) initial condition on the device for the dc analysis. If the area factor is omitted, a value of 1.0 is assumed. The (optional) initial condition specification using IC=VBE, VCE is intended for use with the UIC option on the .TRAN control line, when a transient analysis is desired starting from other than the quiescent operating point. See the .IC control line description for a better way to set transient initial conditions. The (optional) TEMP value is the temperature at which this device is to operate, and overrides the temperature specification on the .OPTION control line.

D.6.: MOSFETs

General Form:

MXXXXXXX ND NG NS NB MNAME <L=VAL> <W=VAL> <AD=VAL> <AS=VAL>

+ <PD=VAL> <PS=VAL> <NRD=VAL> <NRS=VAL> <OFF>

+ <IC=VDS, VGS, VBS> <TEMP=T>

Examples:

M1 24 2 0 20 TYPE1

M31 2 17 6 10 MODM L=5U W=2U

M1 2 9 3 0 MOD1 L=10U W=5U AD=100P AS=100P PD=40U PS=40U

ND, NG, NS, and NB are the drain, gate, source, and bulk (substrate) nodes, respectively. MNAME is the model name. L and W are the channel length and width, in meters. AD and AS are the areas of the drain and source diffusions, in meters . Note that the suffix U specifies microns (1e-6 m) and P sq-microns (1e-12 m). If any of L, W, AD, or AS are not specified, default values are used. The use of defaults implifies input file preparation, as well as the editing required if device geometries are to be changed. PD and PS are the perimeters of the drain and source junctions, in meters. NRD and NRS designate the equivalent number of squares of the drain and source diffusions; these values multiply the sheet resistance RSH specified on the .MODEL control line for an accurate representation of the parasitic series drain and source resistance of each transistor. PD and PS default to 0.0 while NRD and NRS to 1.0. OFF indicates an (optional) initial condition on the device for dc analysis. The (optional) initial condition specification using IC=VDS, VGS, VBS is intended for use with the UIC option on the .TRAN control line, when a transient analysis is desired starting from other than the quiescent operating point. The TEMP value is the temperature at which this device is to operate, and overrides the temperature specification on the .OPTION control line. The temperature specification is ONLY valid for level 1, 2, 3, and 6 MOSFETs, not for level 4 or 5 (BSIM) devices.

E. Others often used:

.options (simulation variables); .print, .plot, .ic and etc.

How to invoke HSPICE and View Output Waveforms

1. Invoke HSPICE

After you finish editing the H-spice file, for example, opamp.sp, HSPICE can be invoked by typing:

% **hspice opamp.sp > opamp.lis**

opamp.lis is the outputfile where you can find all kinds of usefule information you may want to check.

2. Viewing the waveforms

There is a graphic tool called **mwaves** you can use to view the hspice waveforms. In the unix shell, type:

% **mwaves**

A window will pop up. Then go to:

Design-->Open

In the "Open Deign" window, choose the project you want to view. Click it.

Then in the "Results Browser" window, choose the results you want to view.