

LAB MANUAL # 1

Course Title: **Digital Electronics**
Course #: TE-301
Taught to: 3rd term Telecom Engineering
Lab Instructor: Engr. Mian Shahzad Iqbal

OBJECTIVE:

To have an introduction of **Spice** design and simulation of electronic circuits.

Design and analysis of some example circuits.

DESCRIPTION:

1. INTRODUCTION:

SPICE, an acronym for *Simulation Program with Integrated-Circuit Emphasis*, is considered as industrial standard for Telecom-aided circuit analysis for microelectronic circuits.

SPICE is a general-purpose circuit simulator capable of performing the following types of analysis:

- **Nonlinear DC analysis:** DC analysis calculates the behavior of the circuit when only a DC voltage or current is applied to the circuit. The result of this analysis is commonly referred to as the DC bias or operating-point characteristic.
- **Nonlinear transient analysis:** Computes the various voltages and currents in the circuit with respect to time.
- **Linear AC analysis:** Linearizes the circuit around the DC operating point and then calculates the output as a function of frequency. Equivalent to calculating the sinusoidal steady-state behavior of the circuit.
- **DC Sweep:** Allows a series of DC operating points to be calculated while sweeping or incrementally changing the value of an independent voltage or current source.
- **Transfer Function Analysis:** Computes the small-signal DC gain from a specified input to a specified output (i.e. voltage gain, transconductance, transresistance or current gain) and the corresponding input and output resistance.
- **Temperature analysis:** Allows a series of analyses to be performed while varying the temperature of the circuit.
- **Noise analysis:** Calculates the noise contribution of each element and the total effect on the output in a mean-square sense.
- **Sensitivity analysis:** Indicates which components affect the circuit performance most critically.

- **Fourier analysis:** Computes the Fourier series coefficients of the circuit's voltages or currents.
- **Mont Carlo analysis:** Performs multiple runs of selected analysis types using a predetermined statistical distribution for the values of various components.
- **Distortion analysis:** Computes several harmonic distortion characteristics using AC small-signal analysis.

SPICE analyses various types of circuits contain one or more of the following components:

- Independent and/or dependent voltage and/or current sources
- Resistors
- Capacitors
- Inductors
- Mutual inductors
- Transmission lines
- Operational amplifiers
- Switches
- Diodes
- Bipolar junction transistors
- Junction field effect transistors (JFET)
- Metal-oxide semiconductor transistors (MOS)
- Metal Schottky field effect transistors (MESFET)
- Digital gates (PSpice for example)

2. SPICE INPUT FILE AND BASICS

A circuit to be simulated must be described in a sequence of lines (cards). Each line is either a statement, which describes a single element, or a control line, which set model parameters, measurement nodes, or analysis types. The first line must be a **title** line, and the last line must be an **.END** statement. A command line must start with **dot** command. Comment line must start with an **asterisk** '*'. Spice input file (deck) format is as follows:

Title statement

circuit description

Power Supplies / Signal Sources

Element Descriptions

Model Statements

Analysis Requests

Output Requests

.END

The circuit is described as elements connected between nodes. Elements must be uniquely labeled, and the nodes must be distinctly numbered with nonnegative integers between 0, for ground, and 9999.

Elements have small or large values can be encoded as follows:

Code	metric prefix	Decimal representation
T	tera	10^{12}
G	giga	10^9
Meg	mega	10^6
K	kilo	10^3
M	milli	10^{-3}
U	micro	10^{-6}
N	nano	10^{-9}
P	pico	10^{-12}
F	femto	10^{-15}

Each element has a unit (dimension) which are as follows:

Spice Suffix	Units
V	volts
A	amps
Hz	hertz
Ohm	ohm ()
H	henry
F	farad

Degree	degree
--------	--------

3. SPICE BASIC ELEMENTS




The basic elements of electronic circuits can be represented by the first letter, the list of these elements and their representations are as follows:

1st letter	Element
B	GaAs Field-Effect Transistor (MESFET)
C	Capacitor
D	Diode
E	Voltage-Controlled Voltage Source (VCVS)
F	Current-Controlled Current Source (CCCS)
G	Voltage-Controlled Current Source (VCCS)
H	Current-Controlled Voltage Source (CCVS)
I	Independent Current Source
J	Junction Field-Effect Transistor (JFET)
K	Coupled Inductors
L	Inductors
M	MOS Field-Effect Transistor (MOSFET)
Q	Bipolar Junction Transistor (BJT)
R	Resistor
V	Independent Voltage Source

4. SPICE ELEMENT LINES

Passive Components

SPICE considers the current always flows from the positive node through the element to the negative node.

Element	Spice Description
	Rname n+ n- value <>
	Lname n+ n- value [IC=initial_voltage_condition]
	Cname n+ n- value [IC=initial_current_condition]

The first field of each statement describes its type and provides a unique name for each element. This is followed by two fields that describe how each element is connected to the rest of network through node numbers. Each element is assigned a positive and a negative terminal. The fourth field is used to specify the value of the passive element. Resistance is specified in Ohms, inductance in Henries and capacitance in Farads. For the capacitor or inductor, an initial (t=0) voltage or current condition can be specified in its fifth field.

```
R1 1 2 10ohm
C1 2 0 2.65mF
Rload 3 0 100ohm
```

Independent Sources

Voltage and current sources that are independent of any circuit variables are defined by the following statements:

Type of Signal	Type of analysis	Element statement
DC Voltage	All types	Vname n+ n- DC value
DC Current	All types	Iname n+ n- DC value
AC Voltage	Frequency sweep	Vname n+ n- AC mag phase
AC Current	Frequency sweep	Iname n+ n- AC mag phase
SIN wave Voltage	Transient	Vname n+ n- SIN (Vo Va freq td damping)

SIN wave Current	Transient	Iname n+ n- SIN (Vo Va freq td damping)
Pulse wave Voltage	Transient	Vname n+ n- PULSE (V1 V2 td tr tf PW T)
Pulse wave Current	Transient	Iname n+ n- PULSE (V1 V2 td tr tf PW T)
Piecewise linear Voltage	Transient	Vname n+ n- PWL(t1,v1 t2,v2...tn,vn)
Piecewise linear Current	Transient	Iname n+ n- PWL(t1,v1 t2,v2...tn,vn)

DC:

Generic statement:

Vname n+ n- DC value

Example:

V1 1 0 DC 10V

This is a DC voltage source with a value of 10V named “1” and its connected with its positive terminal at node 1 and its negative terminal at node 0 (ground). The name is arbitrary; you can call it anything you want.

AC:

Generic statement:

Vname n+ n- AC (mag phase)

Example:

Vnew 5 0 AC (5V 2)

This statement describes a frequency-swept AC source called “new” that is connected to nodes 5 and 0, has a magnitude (*mag*) of 5V, and contains phase shift of 2 degrees. The *phase* can be left blank and HSpice will assume a zero degree phase shift.

SIN:

Generic statement:

Vname n+ n- SIN (Vo Va freq td damping)

Example:

Vinput 2 1 SIN (0V 5V 10e3 5e-3 0)

Vo is the initial voltage.

Va is the voltage amplitude.

freq is the frequency.

td is the time delay in seconds.

damping is the damping factor.

This element statement describes a sinusoidal signal named “input” at it is connected between nodes 2 and 1. The signal has an initial voltage of 0V and a magnitude of 5V. It has a frequency of 10kHz and a time delay of 5milliseconds. The signal has no damping factor. If the statement does not contain values for *td* or *damping*, then HSpice assumes a value of zero for both.

PULSE:

Generic statement:

Vname n+ n- PULSE (V1 V2 td tr tf PW T)

Example:

V50 15 0 PULSE (2 5 0 2e-3 4e-5 5 10)

V_1 is the lower voltage value.
 V_2 is the upper voltage value.
 t_d is the time delay in seconds.
 t_r is the time in seconds it takes for the pulse to rise.
 t_f is the time in seconds it takes for the pulse to fall.
PW is the pulse width of the peak value (or the time the pulse remains at the upper voltage value in seconds).
 T is the time of one pulse period in seconds.

PWL:

Generic:

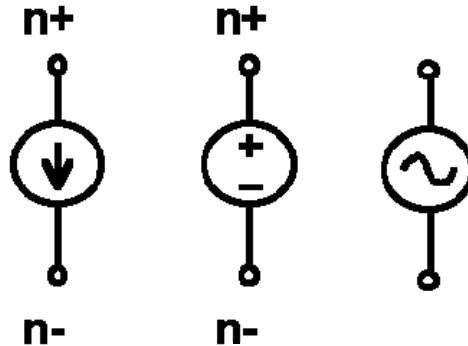
$Vname\ n+\ n-\ PWL(t_1,v_1\ t_2,v_2\dots t_n,v_n)$

Example:

V100 25 10 PWL(0,0 3e-3,5V 6e-3,5V 10e-3,-3V)

To use a piecewise linear source, state a voltage and the corresponding time you wish the source to reach that value. In the example above at time 0, the source will have a value of 0V. The source will then perform a linear increase for the next 3ms till it reaches 5V. The next point given has the same voltage value, so the source will have the value of 5V for the next 3ms. The source will then follow a linear degradation till it reaches -3V four milliseconds later.

The polarity conventions are shown below:



Dependent Sources

Generic statement:

$Ename\ n+\ n-\ p+\ p-\ A$

Example:

Eone 2 3 1 0 50

There are four nodes in the dependent source command. The first two nodes of the voltage controlled voltage source (VCVS), n_+ and n_- , represent the node numbers for the positive and negative ports of the output. The second set of node numbers represents the positive and negative nodes of source's reference voltage. The gain of the VCVS is indicated by the value of A . In the example, the controlling, or reference, voltage of the VCVS is between nodes 1

and 0, while it has a gain of 50. The output of the VCVS is connected between nodes 2 and 3. The value of the output is dependent on the controlling voltage by a factor of the gain.

5. ANALYSIS REQUESTS:

a. **.OP Statement:** This causes Spice to solve to solve for the DC operating point of the given circuit. This includes finding all the DC node voltages and currents and power dissipation of all voltage sources.

b. **.DC Statement:** This statement allows you to increment (sweep) an independent source over a certain range with a specified step. The format is as follows:

.DC SRCname START STOP STEP

in which SRC name is the name of the source you want to vary; START and STOP are the starting and ending value, respectively; and STEP is the size of the increment.

Example: *.DC V1 0 20 2*

You can also vary the temperature of the circuit this way by simply replacing the name of the source in the field labeled SRCname by TEMP. You can nest the DC sweep command which is often used to plot transistor characteristics, such as the Drain current i_{ds} versus the Drain-source voltage V_{ds} for different gate voltages V_{gs} . This can be done as follows:

.DC SRCname1 START STOP STEP SRCname2 START STOP STEP

Example: *.DC Vds 0 5 0.5 Vgs 0 5 1*

c. **.AC Statement:**

This statement is used to specify the frequency (AC) analysis. The format is as follows:

.AC LIN NP FSTART FSTOP

.AC DEC ND FSTART FSTOP

.AC OCT NO FSTART FSTOP

in which

LIN stands for a linear frequency variation,

DEC and OCT for a decade and octave variation respectively.

NP stands for the number of points and

ND and NO for the number of frequency points per decade and octave.

FSTART and FSTOP are the start and stopping frequencies in Herz.

d. **TRAN Statement:** This statement specifies the time interval over which the transient analysis takes place, and the time increments. The format is as follows:


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

TSTEP is the printing increment.

TSTOP is the final time

TSTART is the starting time (if omitted, TSTART is assumed to be zero)

TMAX is the maximum step size.

UIC stands for Use Initial Condition

e. **IC Statement:**

This statement provides an alternative way to specify initial conditions of nodes (and thus over capacitors).

```
.IC Vnode1 = value Vnode2 = value etc.
```

6. OUTPUT REQUESTS

These statements will instruct Spice what output to generate. If you do not specify an output statement, Spice will always calculate the DC operating points. The two types of outputs are the prints and plots. A print is a table of data points and a plot is a graphical representation. The format is as follows:

```
.PRINT TYPE OV1 OV2 OV3 ...
```

```
.PLOT TYPE OV1 OV2 OV3 ...[lower_plot_limit,  
upper_plot_limit]
```

in which TYPE specifies the type of analysis to be printed or plotted and can be:

- DC
- TRAN
- AC

The output variables are OV1, OV2 and can be voltage or currents in voltage sources. Node voltages and device currents can be specified as magnitude (M), phase (P), real (R) or imaginary (I) parts by adding the suffix to V or I as follows:

- M: Magnitude
- DB: Magnitude in dB (deciBells)
- P: Phase
- R: Real part
- I: Imaginary part

Examples:

```
.PLOT DC V(1,2) V(3) I(Vmeas)
```

```
.PRINT TRAN V(3,1) I(Vmeas)
```

```
.PLOT AC VM(3,0) VDB(4,2) VM(2,1) VP(3,1) IR(V2)
```

EXAMPLE:

Transient Response of a Linear Network

**Circuit Description **

* Input Signal Source

Vi 1 0 SIN (0V 10V 60Hz)

*Linear network

R1 1 2 1ohm

C1 2 0 2.65mF

Rload 3 0 10ohm

E1 3 0 2 0 10

** Analysis request **

.TRAN 1ms 50ms 0ms 1ms

** Output Requests **

.PRINT TRAN V(3) V(1)

.PLOT TRAN v(3) V(1) (-100, +100)

* indicate end of Spice deck

.end