Introduction to SPICE

BITS Pilani

Pilani|Dubai|Goa|Hyderabad

07/02/2016

Contents

1. What is SPICE?

2. Why do we need SPICE?

3. A Simple Example using SPICE.

4. Analysis options is SPICE.

    Operating point

    DC Analysis

    AC Analysis

    Transient Analysis

5. References

What is SPICE?

SPICE is an Acronym of

Simulation

Program with

Integrated

Circuit

Emphasis

Introduction

- Developed in 1970 by UCLA, Berkeley.
- Ronald Rohrer and Nagel developed CANCER program.

  CANCER – Computer Analysis of Non-Linear Circuits.
- SPICE1 was derived from CANCER program.
- SPICE uses nodal analysis to solve the circuits.

Introduction

- HSPICE
- PSPICE – Microsim (first PC version)
- LT-SPICE – Linear Technology (free version)
- NG-SPICE (free and trusted)

Introduction

- SPICE is an interpreter based language.
- The input file is called "**netlist**"
- It is also referred as "**DECK**"
- Each line in the SPICE netlist is called "**CARD**"
- The input file comes with an extension **".cir" / (.sp .ckt)**
- Such file can be written using any text editor.
- Key to make any SPICE simulation is the **"model library"**
- Model libraries are very costly.
- Certain free libraries are also available.

  Ex: Stanford PMT model files

Structure of SPICE File

- Each file should have **.cir** extension

  TITLE

  ELEMENT DESCRIPTIONS

  .MODEL STATEMENTS

  ANALYSIS COMMANDS

  OUTPUT COMMANDS

  .END

- Single Line Comments are indicated by "*****"
- Following line comments can be indicated using "**;**"
- Two or more lines can be combined using "**+**"

Circuit Description

Elements and Representation –

R – Resistor Default Unit " Ohm"
C – Capacitor Default Unit "Farad"
L – Inductor Default Unit " Henry"

- Both upper case and lower case letters are equivalent
- Element is identified by first character only.

Circuit Description

Symbol	Prefix	Value
T	Tera	10^{12}
G	Giga	10^{9}
MEG	Mega	10^{6}
K	Kilo	10^{3}
F	Femto	10^{-15}
N	Nano	10^{-9}
P	Pico	10^{-12}
U	Micro	10^{-6}
M	Milli	10^{-3}

Table.1.List of pre-defined suffixes in SPICE

Circuit Description Example

- Nodes is associated with a number.
- Ground is to be always denoted by “node-0”

Examples -

R1	5	0	20K
C1	Vin	Gnd	25pF
C_{load}	Vout	Gnd	0.30 UF
L	1	2	0.5 UH

- The data that follows valid suffix is ignored by the SPICE Simulation

Rules for Naming in SPICE

- Can start with a number/alphabet.
- But once the first character is number then the following part should have only numbers. Ex. 2A is invalid node name.
- Model name can’t start with a number.
- All the letters that follow a number (Value) are neglected unless they are a part of standard pre-fix table.

Independent Sources

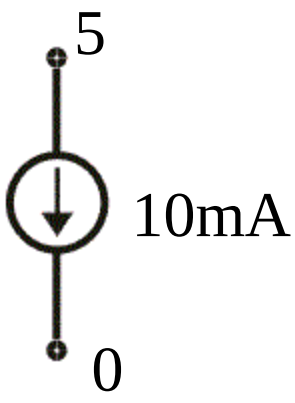
- V – Independent Voltage Source
- I – Independent Current Source

Name	< +ve >	<-ve>	<type>	<value>

Examples:

```
VDD      3    0     5V
IBIAS    10   0     DC   100A
ISS      5    0     DC   10mA
```

Current flows from node-5 to node-0



Dependent Sources

- VCVS (E)– Voltage Controlled Voltage Source
- VCCS (G)– Voltage Controlled Current Source
- CCCS (F)– Current Controlled Current Source
- CCVS(H) – Current Controlled Voltage Source

Dependent Sources

VCVS –

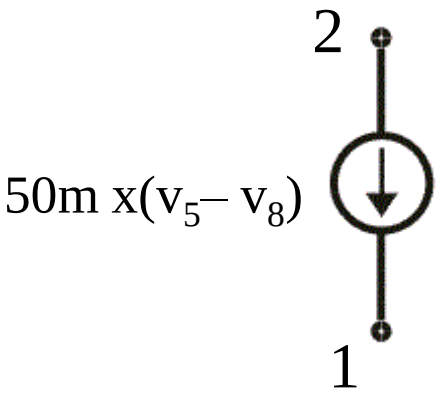
$$v_5 - v_2 = 10\,(v_3 - v_1)$$

E1 5 2 3 1 10 ⟶ Gain

Output node pair input node pair

VCCS –

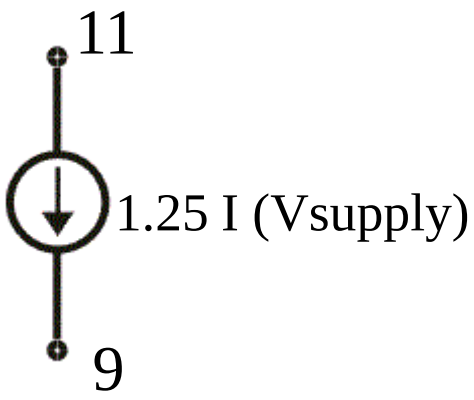
G1 2 1 (5,8) 50m



Dependent Sources

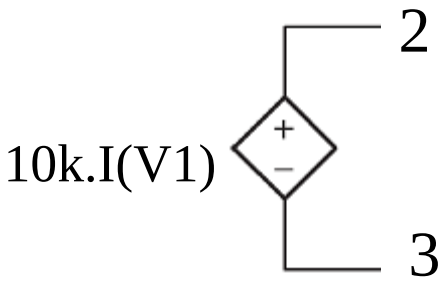
CCCS –

FA2 11 9 Vsupply 1.25



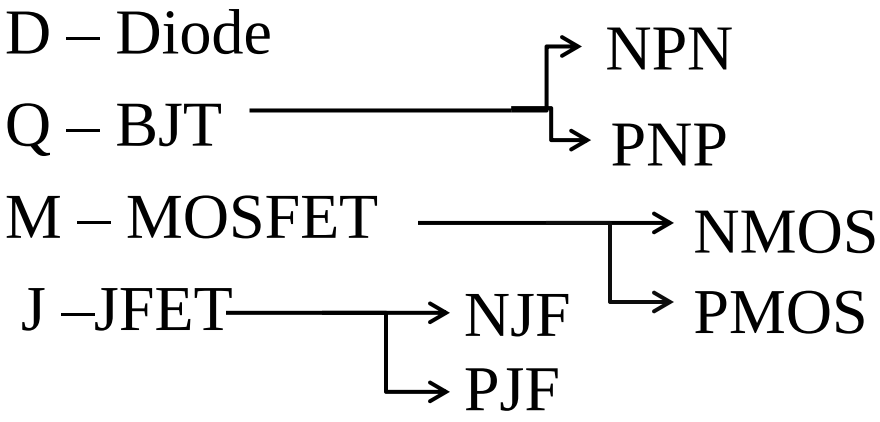
CCVS –

Hsup 2 3 V1 10K



- To sense the current sometimes we need to use a “0v” voltage-source (DC) in series.

Semiconductor Devices



X –SUBCIRCUITS

- All semiconductor devices can have their own model definitions and parameters.
- All the parameters need not be specified since SPICE can call default values for thr unspecified values.

```
.Model   ModName     Type    <parameters>
```

Diode Model Declaration

- **Is-** Saturation Current (10fA)
- **N-** Emission Coefficient (1)
- **Rs-** Series Resistance (0 ohm)
- **Cjo-** Junction Capacitance (0 F)
- **TT-** Transit Time (0 sec)
- **BV-** Reverse Bias Breakdown Voltage (infinite)
- **IBV-** Reverse Bias Breakdown Current (1 e^{-10})
- **VJ-** Built in Junction Potential (0.6V/0.7V)

Diode Model Declaration

Example :

```
.MODEL   DIN4007   D    Is = 7.0276N     +
N = 1.88    Rs = 0.0345      CJO = 10p        +
TT = 0.10   BV=1K    IBV = 50N     VJ = 0.7
```

- D1 12 15 DIN4007

Bipolar Transistors (BJT)

- **BF** – Forward Active Current Gain (100)
- **Is** – Saturation Current (.1 fA)
- **Vaf** – Early Voltage (infinite)

```
Qname  C   B   E   BJT_modelName
```

MOSFETS

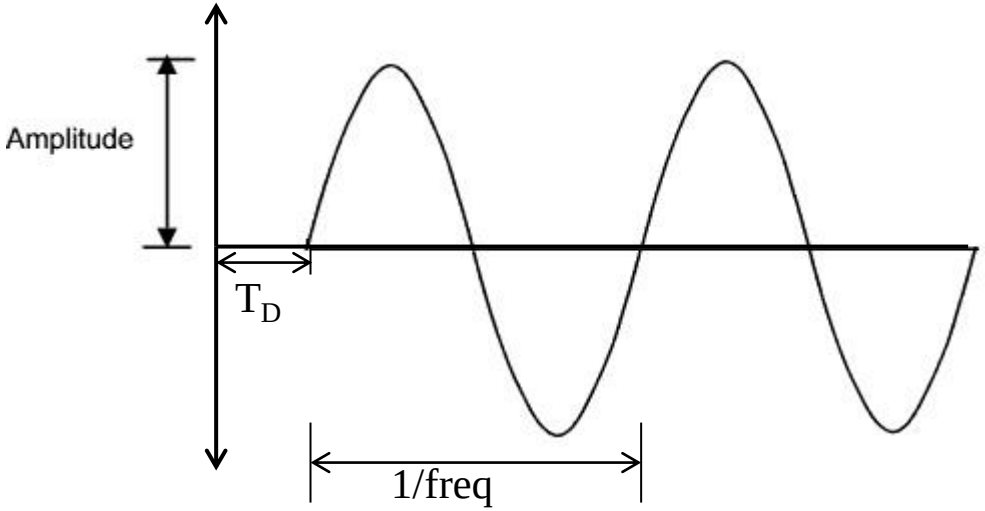
Mname	D	G	S	B	ModelName	L=	W=

(L & W in “m”)

- **VT0** – Threshold Voltage when VSB = 0
- **U0** – Mobility (cm^2/V-s)
- **KP** – Transconductance Parameter (A/V^2) (2 e^{-5})
- **GAMMA**– Body Effect Parameter ($V^{0.5}$)
- **LAMBDA**– Channel Length Modulation (V^{-1})
- **TOX** – Oxide Thickness (m)
- **Level no.** is the most crucial data for MOSFET modelling.

Voltage Sources (AC)

Sinusoidal Source:-



$V_{sinewave}$ N1 N2 Sin(V0 VA Freq TD Θ Φ)

Ex:- V_{car} N1 N2 Sin(0 50m 1k)

- Defines 1kHz sinewave of V_{peak} = 50mV
- By default TD=0, Θ=0, Φ=0
- Θ is for exponential damping sinewave

Piece-Wise Linear (PWL)

Vname N1 N2 PWL (T1 V1 T2 V2 …)

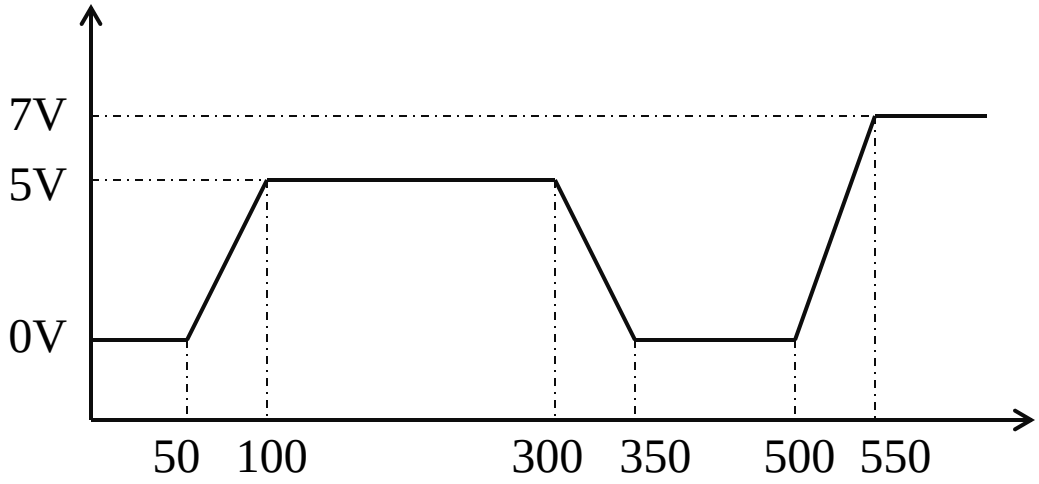
- Ti –Vi = Voltage Value at a given time Ti

V1 1 2 PWL +

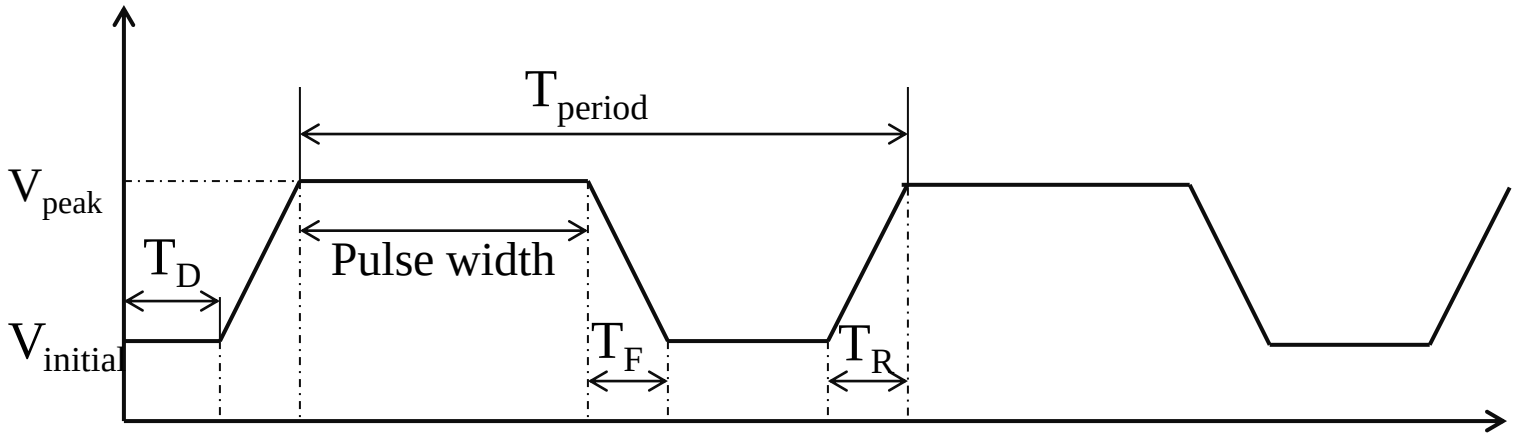
0 0 50 0 +

100 5 300 5V +

350 0 500 0V 550 7V



Pulse Waveform (clock waveform)



Vpulse N1 N2 Pulse (V1 V2 TD TR TF PW PER)

Ex:- Square Waveform 33% Duty Cycle 10kHz

Vsquare N1 N2 Pulse (0 5 10p 10p 10p 33u 0.1m)

- Specify Rise Time and Fall Time in ps if they are of no specific importance.

Sub-Circuits (letter X is reserved for them)

.SUBCKT subckt_name nodes_externally

Element Statements

.ENDS subckt_name

- Node 0 still remains ground
- Node name can’t be node 0
- Internal node names are always local
- Opamps are always modelled as subcircuits

Control Statements

DC Operating Point - .op

- Voltages at the nodes
- Current in each voltage source
- Operating point for each element
- Automatically done before AC or Transient Analysis

Control Statements

DC Analysis - .dc

- Better understood as dc sweep analysis
- Can be done on any (one or more) independent sources

```
.DC   source_name   Start   Stop   Step
```

Ex : .DC Vapplied 0 1.2 .01

- Nesting of dc sweep is allowed and done often

Ex : .DC Vds 0 3.3 0.1 Vgs 0 1.8 0.6

Control Statements

Transient Analysis - .tran

- Time varying signals used to plot Vs time
- Internally operating points are always calculated

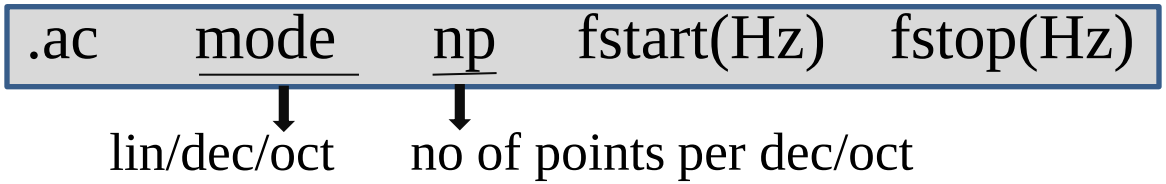
.tran	tstep	tstop	tstart (default zero)

Note. Used in general to observe the time response of the circuits.

Control Statements

AC analysis - .ac

- Used to perform frequency analysis
- Filters , Amplifiers and Resonant Circuits



Ex:- .ac dec 10 1 10k

- Plots frequency response from 1 Hz to 10Khz with each decade having 10 sample points.
- In general decade sweep is used for frequency response.

Control Statements

Transfer function - .tf

- Plots the transfer function (small signal)

```
.tf   output_variable  source
```

- Can be used to calculate the thevenin equivalent
- Output also contains : a) Resistance with respective input source
  b) Resistance with respective output source

Ex:- .tf vout vin

Control Statements

Sensitivity Analysis - .sens

- Plots the sensitivity of the variable with respect to every circuit parameter (small signal values)

```
.sens    variable
```

- Can be used to calculate power supply dependencies, resistance tolerance etc

A Presentation By –
Mr. K. Babu Ravi Teja, M.Tech. (VLSI Design)
Lecturer, Dept. of EEE, BITS Pilani (Pilani)
email – baburaviteja.k@pilani.bits-pilani.ac.in