Introduction to SPICE

4/2/2018

BITS Pilani

Pilani|Dubai|Goa|Hyderabad

Contents

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

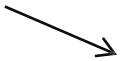
Name	< +ve >	<-ve>	<type>	<value>

Examples:

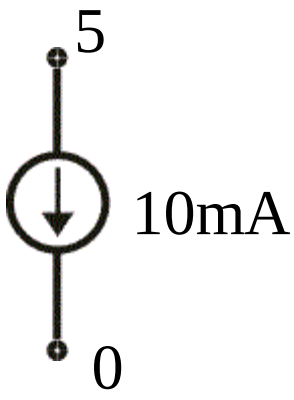
VDD 3 0 5V

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 –

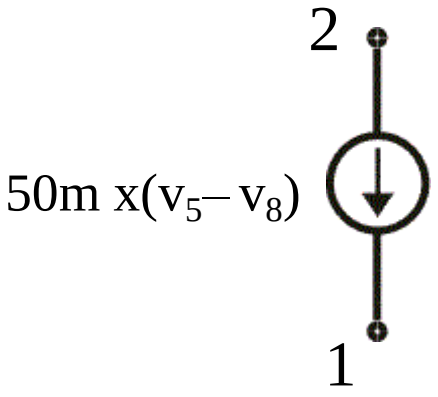
$$v_5 - v_2 = 10\,(v_3 - v_1)$$

E1 5 2 3 1 10 ⟶ Gain

Output node pair input node pair

VCCS –

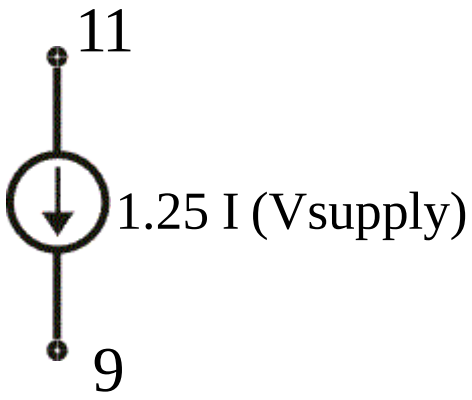
G1 2 1 (5,8) 50m



Dependent Sources

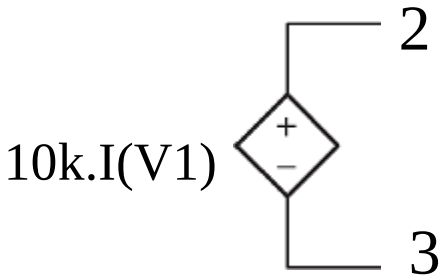
CCCS –

FA2 11 9 Vsupply 1.25



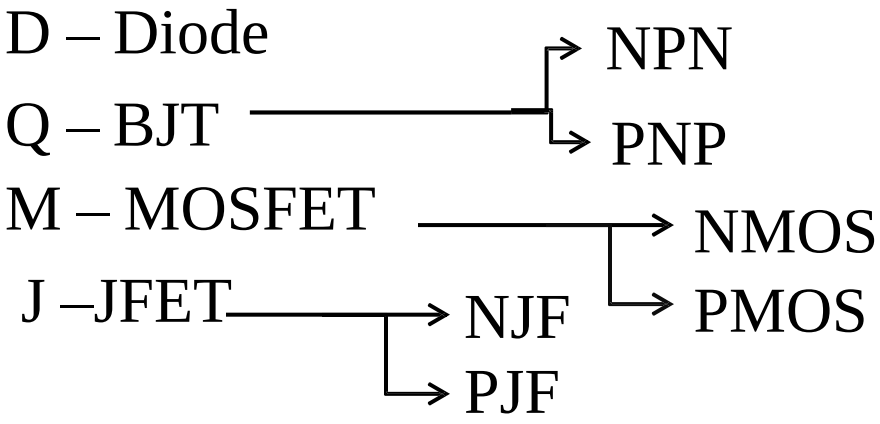
CCVS –

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

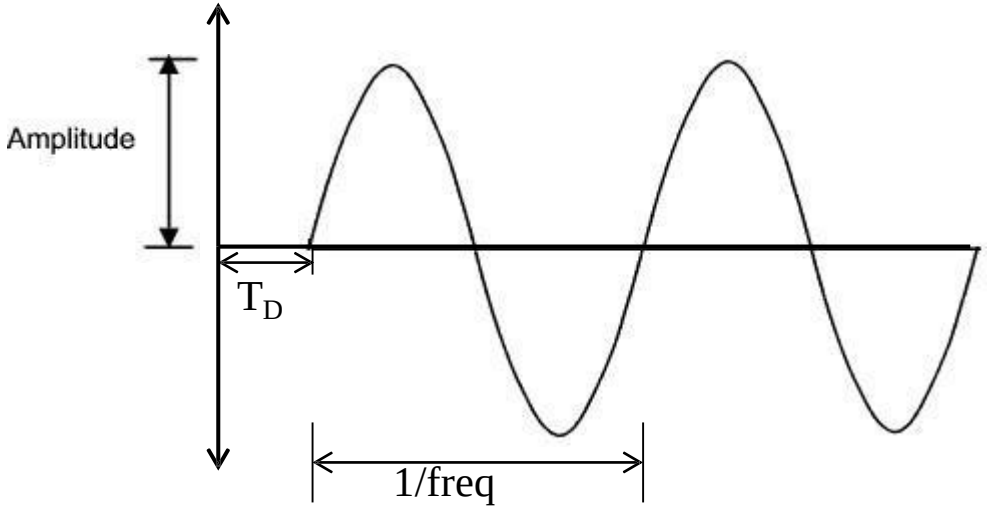
Mname D G S B ModelName L= W=

(L & W in “m”)

- **VT0** – Threshold Voltage when VSB = 0
- **U0** – Mobility ($cm^2/V\text{-}s$)
- **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 (damping factor)

Piece-Wise Linear (PWL)

Vname N1 N2 PWL (T1 V1 T2 V2 …)

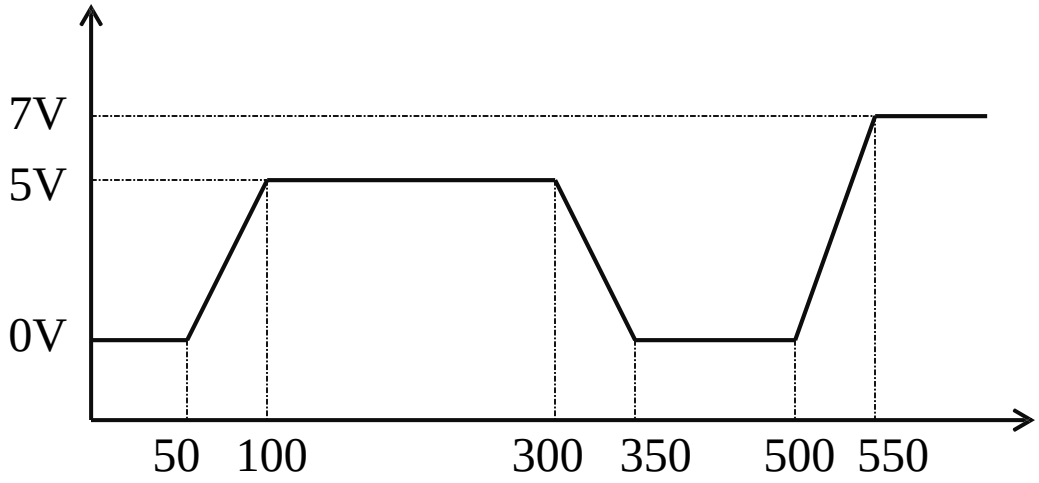
- Ti –Vi = Voltage Value at a given time Ti

V1 1 2 PWL +

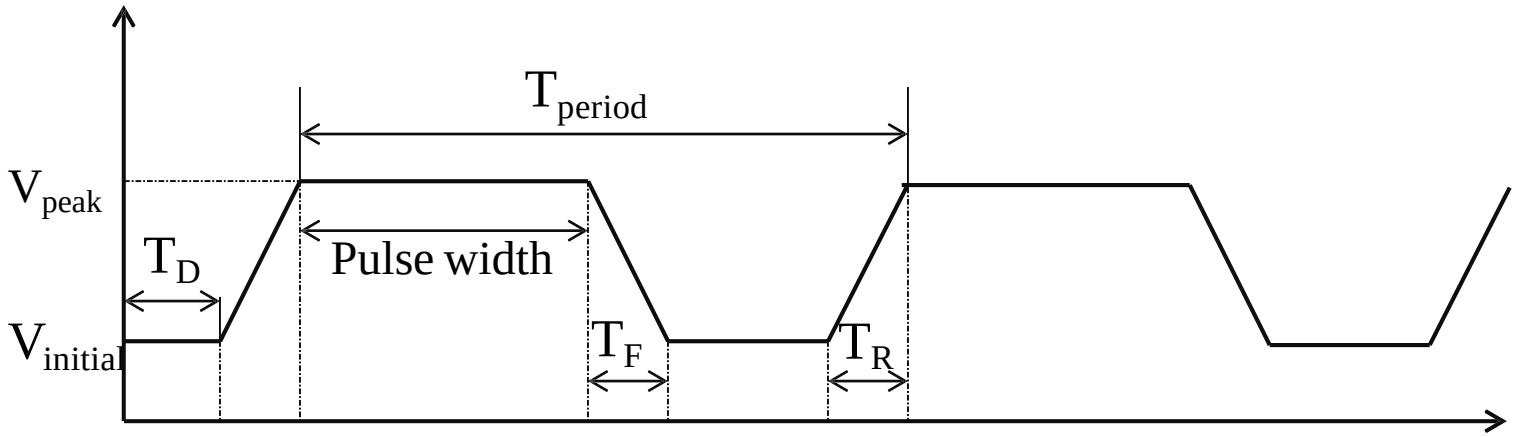
0 0 50 0 +

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 TransientAnalysis

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

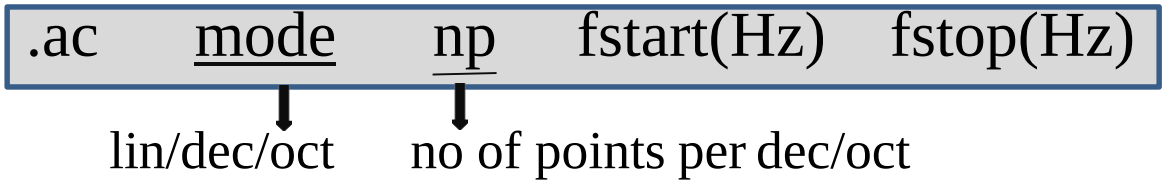
.tran	tstep	tstop	tstart (default zero)

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

Additional Resources

1. https://vision.lakeheadu.ca/eng4136/spice/index.html
2. http://www.ecircuitcenter.com/SPICEsummary.htm
3. *List of Teaching Assistants* - https://docs.google.com/spreadsheets/d/1mZVkgLm2miW1pYADgRn2tXi6tSlGk4C-xHr2PJEK3Ss/edit?usp=sharing