

NGSpice Tutorial

Akshay Adlakha

Teaching Assistant

CMOS Analog VLSI Design (EE-618)

I.I.T. Bombay

1 August, 2015

- A Circuit Simulator is used to replicate the behaviour of an electronic circuit using a mathematical model in a software environment.
- SPICE stands for Simulation Program with Integrated Circuit Reference.
- It was developed as a part of research project at University of California, Berkley in 1960s.
- NGSPICE is an open source mixed signal circuit simulator. It is freely available for windows and linux.

Installing NGSpice

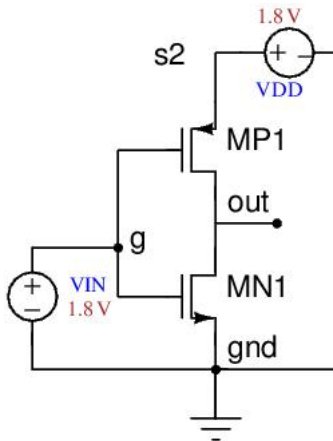
- For LINUX user type following command in terminal-
sudo apt-get install ngspice
- For Windows user-
 - i. Download the binaries from
[http : //sourceforge.net/projects/ngspice/files/latest/download?source = files](http://sourceforge.net/projects/ngspice/files/latest/download?source=files).
 - ii. Extract ngspice-25_130104.zip
 - iii. Goto ../spice/bin/
 - iv. Run ngspice.exe

Key features

- NGSPICE can analyse circuits containing-
 - i. Passive Components - Resistors, Inductors, Capacitors, etc.
 - ii. Dependent or Independent Voltage/Current Sources.
 - iii. Transistors, Diodes, MOSFETs, OpAmps and many more.
- It is also capable of analysing circuits with new technologies or devices given their model files are included.
- It can perform following types of analysis-
 - i. DC Analysis
 - ii. AC small signal Analysis
 - iii. Transient Analysis
 - iv. Noise Analysis and more

NGSPICE Netlist

- A circuit is represented in the form of a netlist. A netlist is a text file that describes the circuit being simulated.
- For example consider the following CMOS inverter circuit.

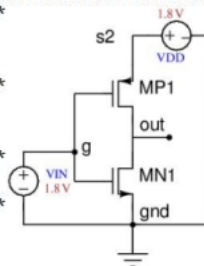


- The netlist associated with it is as follows-

```

1 * CMOS Inverter
2 *****
3 * Include Model File
4 *****
5 .include tsmc_spice_180nm.txt
6 *****
7 *Transistor Description
8 *Note: Name of transistors begin with an "M"
9 *nmos description
10 *name    drain    gate    source    body    d_type    length    width
11 MN1      out      g      gnd      gnd      cmosn    L=0.18u    W=1.8u
12
13 *pmos description
14 *name    drain    gate    source    body    d_type    length    width
15 MP1      out      g      s2       s2       cmosp     L=0.18u    W=3.6u
16 *****
17 *Power Supply
18 VDD s2 dc 1.8v
19 *****
20 *Input Voltage
21 VIN g  gnd dc 1.8v
22 *****
23 *Analysis Statements
24 *****
25 .dc VIN| 0 1.8 0.01
26 .end

```



NGSPICE Netlist: Summary

- Any text editor can be used to write an NGSPICE netlist (notepad for windows and gedit for linux). Netlist is usually saved with an extension '.txt', '.cir' or '.spice'.
- First line in the netlist is **Title**. It can be used to describe the aim of circuit being simulated.
- All **comments** start with an asterix(*).
- All circuit nodes are named with alphanumeric characters only. Circuit components are commonly identified by the first letter of their name like r for resistor, c for capacitor, m for MOSFET, q for BJT, v for voltage source, etc.
- A **ground** or reference node must be there. It is usually represented as gnd or 0.
- The netlist consists of 5 specific parts-
 - i. Title
 - ii. Model File
 - iii. Circuit Elements
 - iv. Analysis Statements
 - v. End Statement

Some Basic SPICE Elements

Table 1: Value Representation

Code	Metric
T	Tera
G	Giga
Meg	Mega
K	Kilo
M	Milli
U	Micro
N	Nano
P	Pico
F	Femto

Table 2: Common SPICE Elements

Spice Keyword	Element
C	Capacitor
L	Inductor
R	Resistor
M	MOSFET
Q	BJT
D	Diode
J	JFET
V	Independent Voltage Source
I	Independent Current Source

Table 3: Unit Representation

Spice Suffix	Unit
V	Volts
A	Ampere
Hz	Hertz
F	Farad
H	Henry
Ohm	Ohm

Common Voltage Sources

- Voltage Sources of various types dc, ac, sinusoidal, pulse and piece wise linear are important for simulations.

- **DC Source-**

Example : `Vin 2 0 dc 1.8v`

`Vin 2 0 dc=1.8v`

- **AC Source-**

Example : `Vin 2 0 dc 1.8v ac 1v`

`Vin 2 0 dc=1.8v ac=1v`

- **Sinusoidal-**

General Form : `SIN (VO VA FREQ TD THETA)`

Example : `Vin 2 0 SIN(1 0.5 1MEG 0 0)`

- **Pulse-**

General Form : `PULSE (V1 V2 TD TR TF PW PER)`

Example : `Vin 2 0 PULSE(0 1 2NS 2NS 2NS 50NS 100NS)`

- **Piece Wise Linear (PWL)-**

General Form : `PWL(T1 V1 <T2 V2 T3 V3 T4 V4 . . . >) < r = value >
< td = value >`

Example : `Vin 2 0 PWL(0 1 10NS 2 20NS 3 30NS 4 50NS 5) r=0 td=5NS`

Types of Analysis

- DC ANALYSIS- Determines the dc operating point of the circuit. It can be performed using .dc command. The .dc line defines dc transfer curve source and sweep limits.

General Form : . dc srcname vstart vstop vincr

Example : .dc VIN 0 1.8 0.1

.dc VGS 0 1.8 0.1 VDS 0 1.8 0.3

.dc VCE 0 10 0.2 IB 0 10u 1u

.dc Rload 1k 2k 100

.dc TEMP -15 75 5

Types of Analysis

- TRANSIENT ANALYSIS- It does the time domain analysis,i.e., the voltages and currents are plotted with respect to time. A transient analysis always starts at $T=0$. From $T=0$ to T_{start} circuit is analysed but no outputs are stored. In the interval T_{start} to T_{stop} outputs are stored. By default T_{start} is 0.

General Form : `.tran Tstep Tstop < Tstart >`

Example : `.tran 1ns 100ns`

`.tran 1ns 1000ns 500ns`

Types of Analysis

- AC ANALYSIS- It does the frequency domain analysis,i.e., the voltages and currents are plotted with respect to frequency. An ac voltage or current source is required to perform an ac analysis. It is useful in finding the small signal characteristics like gain that vary with frequency.

General Form : `.ac dec np fstart fstop`

`.ac oct np fstart fstop`

`.ac lin np fstart fstop`

Example : `.ac dec 10 1 100MEG`

How to Run NGSPICE?

- For Windows User-
 - i. Goto ../spice/bin/
 - ii. Open Notepad, create and save netlist as <.cir> file.
 - iii. Run ngspice.exe
 - iv. In command line interface type "source <filename>.cir."
 - v. Type "run".
 - vi. Now use "plot" command to see the results - plot v(<nodename>) i(<voltage source>).
- For Linux User-
 - i. Open "gedit" and create and save netlist as <.cir> file.
 - ii. Open terminal and go to the directory where you have saved the <.cir> file.
 - iii. Type "ngspice" in terminal and press enter.
 - iv. In the command line interface type "source <filename>.cir"
 - v. Type "run".
 - vi. Now use "plot" command to see the results - plot v(<nodename>) i(<voltage source>).
- To save plot on disc use command- hardcopy <filename.ps> v(<nodename>) i(<voltage source>).

References

- [http : //ngspice.sourceforge.net/docs/ngspice25 — manual.pdf](http://ngspice.sourceforge.net/docs/ngspice25-manual.pdf)
- [http : //iec.iitd.ernet.in/academics/videolectures.html](http://iec.iitd.ernet.in/academics/videolectures.html)
- [http : //www.brunel.ac.uk/ eestmba/usergS.html](http://www.brunel.ac.uk/eestmba/usergS.html)
- [http : //www.ngspice.com/examples.php](http://www.ngspice.com/examples.php)

Thank You