LAB SESSION

Mahendra Sakare and Prof. M. Shojaei Baghini

SIMULATOR

- We will use ngspice simulator.
- Ngspice simulator is public domain software.
- It is available easily.
- We understand it by one example.

Ref : IIT Bombay course EE705/707 internal website “http://sharada.ee.iitb.ac.in/~ee705”

CIRCUIT

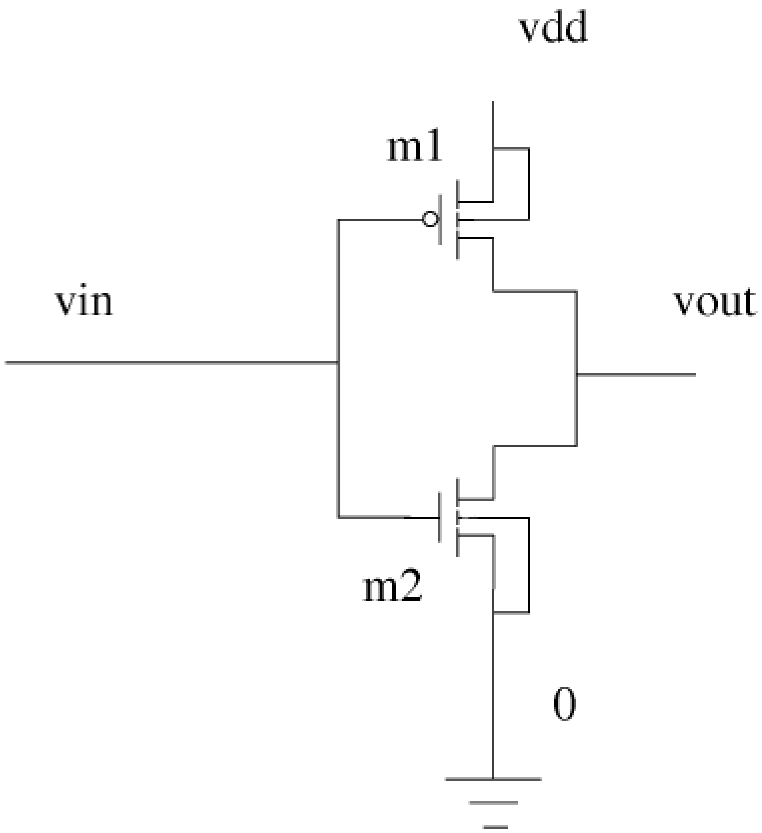


Figure: Inverter

Ref : IIT Bombay course EE705/707 internal website “http://sharada.ee.iitb.ac.in/~ee705”

MODEL FILE DECLARATION

*Create one blank file with extension ".spice"

* script starts here

* The following line includes MOS model file "model.txt"

.include spice model.txt

*spice model file can be HP05, TSMC018 etc.

*Some sites provide model file for public use.

Ref : IIT Bombay course EE705/707 internal website "http://sharada.ee.iitb.ac.in/~ee705"

NETLIST

* Declaring m1 and m2 instances of nmos and pmos, respectively

*Format is

DeviceName drain gate source body DeviceType(pmos or nmos) length width

m1 vout vin vdd vdd CMOSP *L = 0.18u W = 1.71u*

m2 vout vin 0 0 CMOSN *L = 0.18u W = 0.54u*

* *L = x and W = y overwrites default values of L & W*

Ref : IIT Bombay course EE705/707 internal website “http://sharada.ee.iitb.ac.in/~ee705”

COMMAND LINE - SOURCE DECLARATION

*DC source

SourceName Node1 Node2 Sourcetype Value

V1 in 0 dc 1

*For Pulse sourse

V2 in 0 pulse(0 1 1n 10n 10n 100n 200n)

*For AC source

V3 in 0 dc 0 ac 1

Ref : IIT Bombay course EE705/707 internal website “http://sharada.ee.iitb.ac.in/~ee705”

ANALYSIS DECLARATION

* Electrical sources
* Example: input volt source & supply voltage

```
vinput vin 0 dc 0
vsupply vdd 0 dc 1.8
```

* The following line directs ngspice to perform dc analysis

```
.dc vinput 0 1.8 0.1
```

*AC Analysis

```
.ac dec 1 10 100meg
```

*Transient Analysis

```
.tran 1n 100n
```

Ref : IIT Bombay course EE705/707 internal website “http://sharada.ee.iitb.ac.in/~ee705”

- plot the voltage of nodes vout & vin on the output plot

  plot v(vout) v(vin)

* create assign1 1 plot.ps file of output plot

  set hcopydevtype = postscript

  hardcopy assign1 1 plot.ps v(vout) v(vin)

  .endc

  .end

* script file ends here

Ref : IIT Bombay course EE705/707 internal website “http://sharada.ee.iitb.ac.in/~ee705”

NGSPICE EXECUTION COMMANDS

* ngspice execution commands are between ".control" & ".endc" commands.

  .control

* Running the script

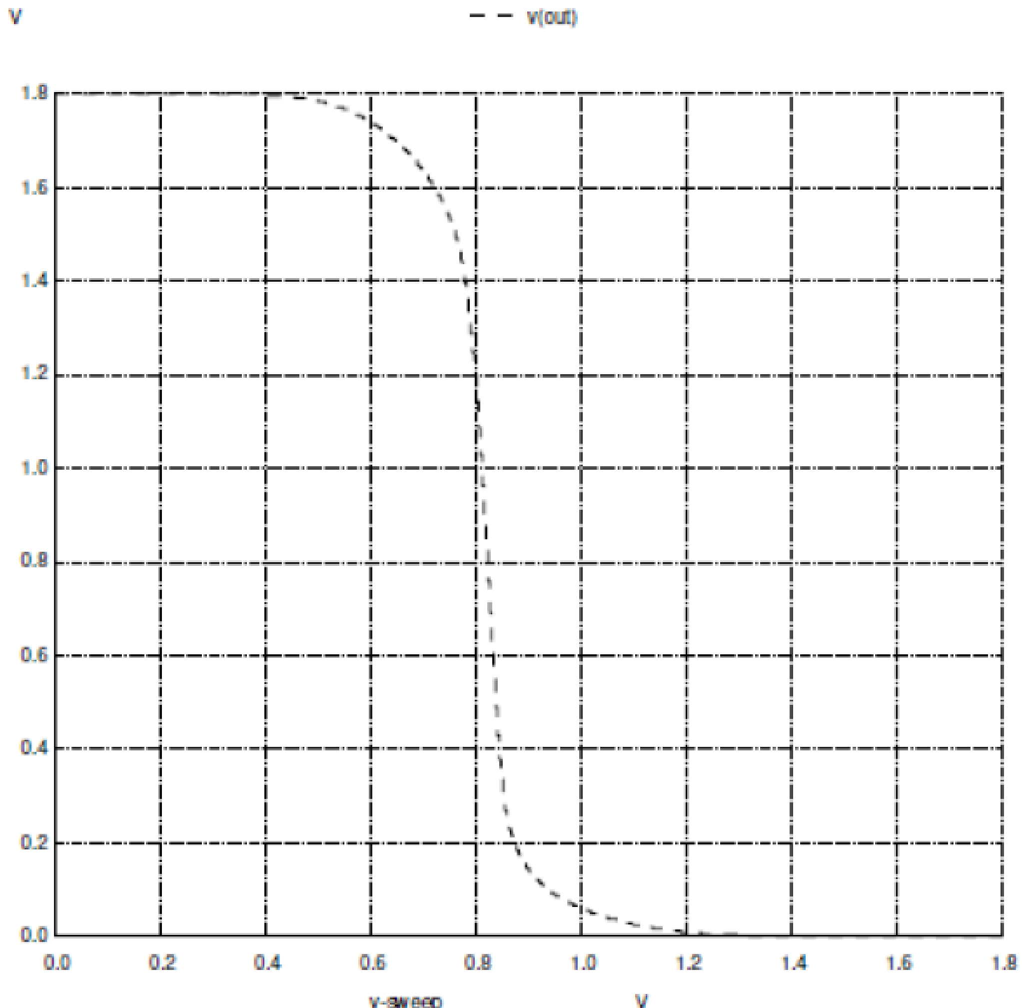
  run

* "print v(N)" prints the voltage at node "N"

  print v(vout)

Ref : IIT Bombay course EE705/707 internal website "http://sharada.ee.iitb.ac.in/~ee705"

OUTPUT PLOT



Ref : IIT Bombay course EE705/707 internal website “http://sharada.ee.iitb.ac.in/~ee705”

EXECUTION

- Save the script as ”filename.spice” or ”filename.cir”
- ”.spice” or ”.cir” extention is not compulsory

but it is a good practice

- The following command in linux will execute the script

  > ngspice filename.spice

Ref : IIT Bombay course EE705/707 internal website “http://sharada.ee.iitb.ac.in/~ee705”

REFERENCES

- IIT Bombay course EE705/707 internal website http://sharada.ee.iitb.ac.in/~ee705”

- http://newton.ex.ac.uk/teaching/CDHW/Electronics2/userguide/

Thank You