About NGSPICE, download and install

- SPICE : Simulation Program with Integrated Circuit Emphasis
- NGSPICE is an open source mixed-signal circuit simulator which can be used to perform different analysis on a circuit
- For SPICE, any circuit is described as an interconnection of various active, passive elements. This interconnection of elements is also called **Net-List**
- Parameters to capture physical behaviour of active devices can be included as **Model File**. For example BSIM1 to BSIM6 are SPICE models for various types of transistors developed by UC Berkley (Berkeley Short-channel IGFET Model)
- DC, transient, AC, pole-zero, noise, PSS - analysis can be performed using NGSPICE
- Result plots can be viewed and saved
- Download NGSPICE from following path and install: *http://ngspice.sourceforge.net/download.html*
- NGSPICE manual can be also downloaded from the same site

General structure of a Net-List

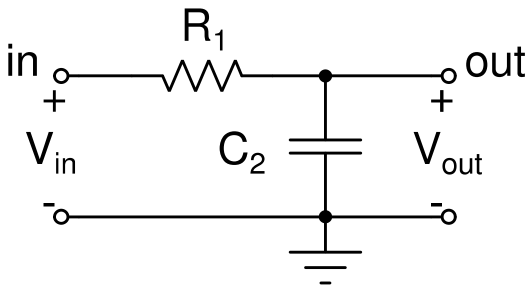
- Circuit description
- Models used to describe circuit elements may be included
- Type of analysis to be done on the circuit
- Control commands to run the simulation and plot/save the results

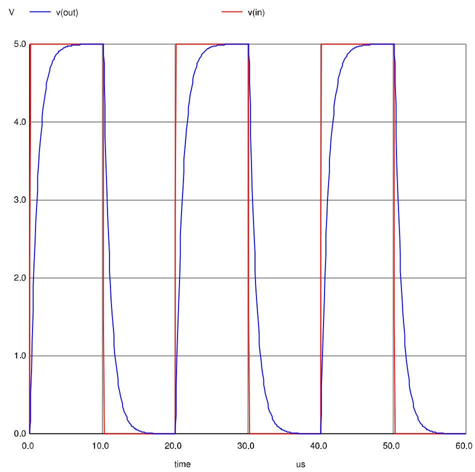
Note: Commands in NGSPICE are case insensitive

Example to illustrate NGSPICE usage

Net list of simple RC circuit

```
Simple RC low pass configuration
* First line is the title. '*' used for comments
* Circuit discription
R1 in out 1k
C2 out 0 1nf

* input pulse
vin in 0 pulse 0 5 0ns 100ns 100ns 10us 20us

* Type of analysis is transient
.tran 10n 60u

.control
run
plot v(in) v(out)

* Saving plots to ps file
hardcopy rc_ckt_tr_out_1.eps v(in) v(out)

.endc
```



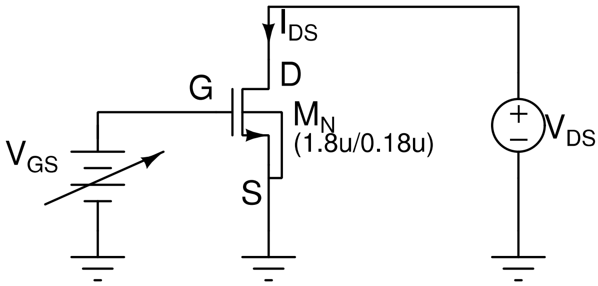


Example to illustrate DC analysis

MOS I_{DS}-V_{GS} curve

Netlist to evaluate MOS I_D-V_{GS}characteristics

```
.include TSMC_180nm.txt
.param SUPPLY=1.8
.param LAMBDA=0.09u
.param width_N={20*LAMBDA}
.global gnd vdd
VGS G gnd 'SUPPLY'
VDS D gnd 1V
M1 D G gnd gnd CMOSN W={width_N} L={2*LAMBDA} +
AS={5*width_N*LAMBDA} PS={10*LAMBDA+2*width_N}
AD={5*width_N*LAMBDA}
PD={10*LAMBDA+2*width_N}
.dc VGS 0 1.8 0.05
.control
run
plot -VDS#branch
set hcopypscolor = 1 *White background
hardcopy mos_id_vg.eps -VDS#branch
.endc
```



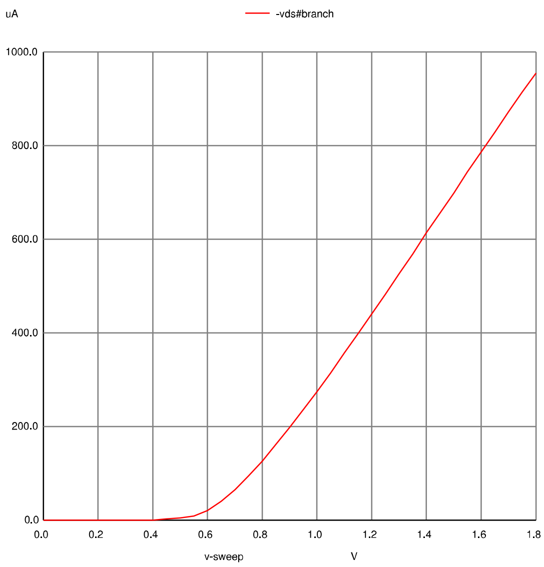
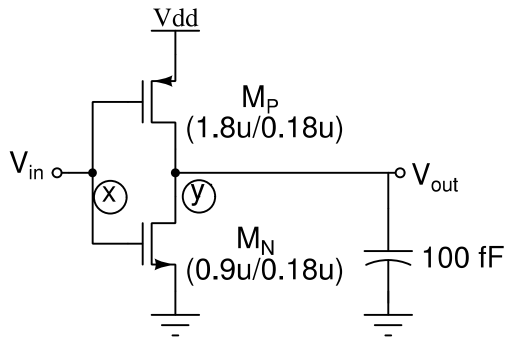


Figure: I_{DS} Vs V_{GS}

Example to illustrate Transient analysis

CMOS Inverter transient analysis

```
CMOS inverter transient response
.include TSMC_180nm.txt
.param SUPPLY=1.8
.param LAMBDA=0.09u
.param width_P=20*LAMBDA
.param width_N=10*LAMBDA
.global gnd vdd

Vdd vdd gnd 'SUPPLY'
vin x gnd pulse 0 1.8 0ns 1ns 1ns 10ns 20ns

M1 y x gnd gnd CMOSN W={width_N} L={2*LAMBDA} +
AS={5*width_N*LAMBDA} PS={10*LAMBDA+2*width_N}
AD={5*width_N*LAMBDA} PD={10*LAMBDA+2*width_N}

M2 y x vdd vdd CMOSP W={width_P} L={2*LAMBDA} +
AS={5*width_P*LAMBDA} PS={10*LAMBDA+2*width_P}
AD={5*width_P*LAMBDA} PD={10*LAMBDA+2*width_P}

Cout y gnd 100f

.tran 0.1n 200n .control

run
plot v(y) v(x)
set hcopypscolor = 1
hardcopy inv_transient_resp.eps v(x) v(y)
```



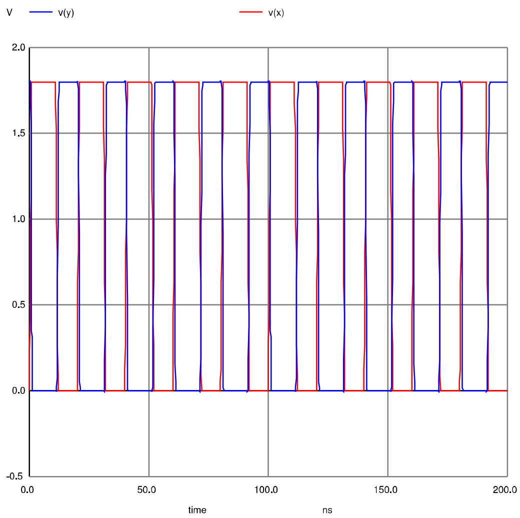
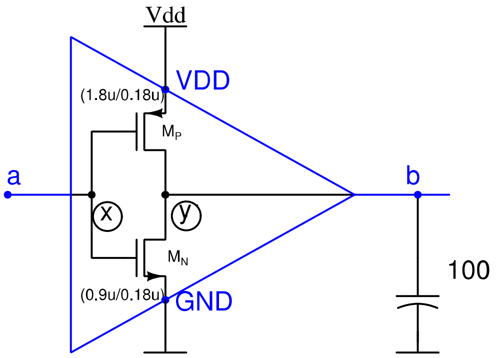


Figure: Transient response of CMOS Inverter

Example to illustrate .SUBCKT usage

Describing CMOS inverter using subckt

```
CMOS inverter transient response
.include TSMC_180nm.txt
.param SUPPLY=1.8
.param LAMBDA=0.09u
.global gnd vdd
Vdd vdd gnd 'SUPPLY'
vin a gnd pulse 0 1.8 0ns 1ns 1ns 10ns 20ns
.subckt inv y x vdd gnd width_P=20*LAMBDA
width_N=10*LAMBDA
.param width_P=20*LAMBDA
.param width_N=10*LAMBDA
M1 y x gnd gnd CMOSN W={width_N} L={2*LAMBDA} +
AS={5*width_N*LAMBDA} PS={10*LAMBDA+2*width_N}
AD={5*width_N*LAMBDA} PD={10*LAMBDA+2*width_N}
M2 y x vdd vdd CMOSP W={width_P} L={2*LAMBDA} +
AS={5*width_P*LAMBDA} PS={10*LAMBDA+2*width_P}
AD={5*width_P*LAMBDA} PD={10*LAMBDA+2*width_P}
.ends inv
x1 b a vdd gnd inv width_P=20*LAMBDA width_N=10*LAMBDA
Cout b gnd 100f
.tran 0.1n 200n .control
run
plot v(b) v(a)
set hcopypscolor = 1
hardcopy inv_transient_resp_subckt.eps v(b) v(a)
```



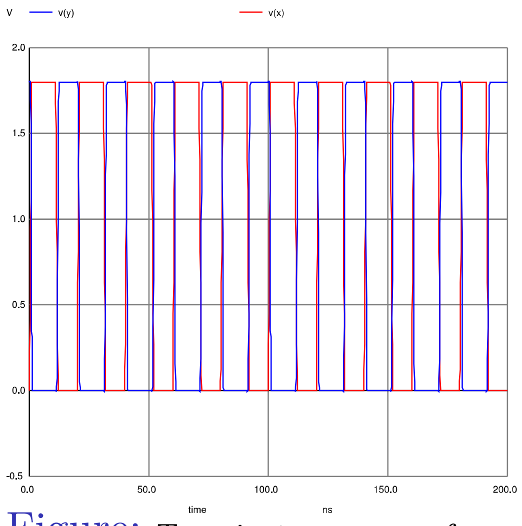


Figure: Transient response of CMOS Inverter

Quick keys I

- To launch NGSPICE
  ngspice : It will take you to ngspice shell
  ngspice file_name.<cir or sp> : It will execute the specified net-list
- Type quit or exit to end the ngspice shell
- To execute a net-list in ngspice shell
  ngspice —> source filename
- To edit in ngspice shell
  ngspice —> edit file_name ; :wq! to save and quit editing
- To run an analysis specified in net-list in ngspice shell
  ngspice —> run
- To plot in ngspice shell
  use ‘plot’ command

Quick keys II

- To plot voltages
  ngspice —> plot v(node_name)
  Ex. plot v(out) v(in)
- To plot branch currents
  plot voltage_source_name#btranch
  Ex. plot VDS#branch
  – Note that direction of current is entering into the positive terminal of a voltage source
  – If required a dummy 0 V DC source can be inserted in the net-list to plot branch current
- To save plots as ps
  hardcopy file_name.eps variables_to_plot
  Ex. hardcopy inv_transient_resp.eps v(x) v(y)
- To change background colour of saved plot
  set hcopypscolor = 1 *White background

Quick keys III

- To change background colour plot window
  set color0=white ** color0 is used to set the background of the plot (manual sec:17.7))
  set color1=black ** color1 is used to set the grid color of the plot (manual sec:17.7))
- Specifying pulse
  vin in+ in- pulse V_{Low} V_{High} delay rise-time fall-time on-period time-period
  vin in 0 pulse 0 5 0ns 100ns 100ns 10us 20us
- Specifying sinusoidal signal
  SIN(VO VA FREQ TD THETA PHASE)
  Ex. vin a 0 sin(.849 0.25 50Meg 0 0)
- DC analysis
  .dc $voltage\_to\_be\_swept$ $V_{initial}$ V_{final} $step\_size$
  Ex: .dc vgs 0v 1.8v 0.1

Quick keys IV

- Transient analysis
  .tran *step_size stop_time* < *start_time* >
  Ex: .tran 10n 60u
- AC analysis
  .ac lin number_of_points start stop
  .ac lin 100 1 100Hz
- Measure statement example (sec 15.4.5):
  .measure tran tpdf
  + TRIG v(1) VAL='SUPPLY/2' RISE=1
  + TARG v(2) VAL='SUPPLY/2' Fall=1
  measures the time difference between v(1) reaching 'SUPPLY/2' V for the first time on its first rising slope (TRIG) versus v(2) reaching 'SUPPLY/2' V for the first time on its first falling slope (TARG), i.e. it measures the fall time delay between v(1) and v(2).
- Refer NGSPICE manual for more details