UNIVERSITY OF CALIFORNIA, SANTA BARBARA

Department of Electrical and Computer Engineering

ECE 122A VLSI Principles

LAB 2 – CMOS Circuit Simulation with HSpice

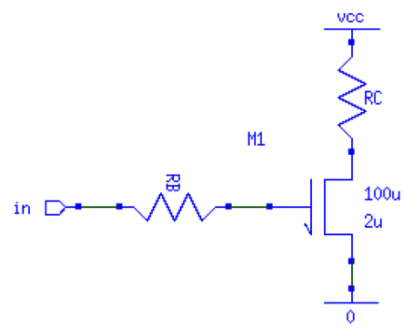
Due Date: ***Friday, 10/15/2021, 5:00 pm***

Part 1: HSpice Syntax

In this part, you will learn to read and write basic netlist file for HSpice simulation.

Here is a typical HSpice netlist (the schematic is shown on right):

```
.title 'Resistor-load inverter'
.include '180nm_bulk.txt'
VCC vcc 0 5
VIN in 0 PULSE 0 5 2NS 2NS 2NS 30NS 60NS
RB in gate 10k
M1 out gate 0 0 NMOS L=2u W=100u
RC vcc out 1k
.options post=2 nomod
.op
.TRAN 1ns 30ns
.DC VIN 0 5 0.1
.END
```



The following is the explanation of each line:

.title 'Resistor-load inverter'	The title **Syntax: .TITLE 'string of up to 72 characters'**
.include '180nm_bulk.txt'	include a file (a library) **Syntax: .INCLUDE 'filepath filename'**
VCC vcc 0 5	Voltage source **Syntax: Vxxx node+ node- voltage**
VIN in 0 PULSE 0 5 2NS 2NS 2NS 30NS 60NS	Pulse source function **Syntax:** **PULSE v1 v2 delay rise_ramp fall_ramp pulse_width period**
RB in gate 10k RC vcc out 1k	Resistor **Syntax: Rxxx node+ node- resistance**
M1 out gate 0 0 NMOS L=2u W=100u	MOSFET **Syntax: Mxxx drain gate source base TYPE Length Width**
.options post=2 nomod	Output binary waveforms
.op	Calculate DC operating point of the circuit
.TRAN 1ns 30ns	Transient Analysis **Syntax: .TRAN step_length duration**
.DC VIN 0 5 0.1	Perform DC Sweep **Syntax: .DC voltage_source start end**

	step_length
.END	The Star-Hspice input netlist file must have an .END statement as the last statement.

For detailed syntax, see HSpice Manual posted on the course website. Note that the manual has 1800+ pages. Use find command to look for the syntax you need.

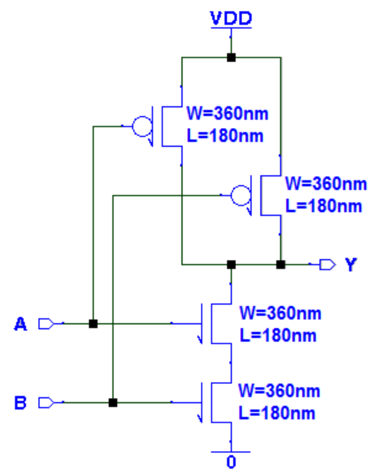
Now write a netlist for the following circuit on yourself. Given Vdd=2.5V, input A=1 and B changes from 0 to 1. Fill the blanks below:

```
.title 'NAND2'
.include '180nm_bulk.txt'

* Transistors:
M1 _____ _____ node1 0 NMOS L=180n W=____
M2 node1 B 0 0 NMOS L=____ W=____
M3 Y A VDD VDD ____ ______ ______
M4 _____ _____ _____ _____ ____ ______ ______

* Sources
VDD VDD 0 ____
VA A ___ ____
VB ___ ___ PULSE 0 ___ 5ns 0.1ns 0.1ns 10ns 20ns

.options post=2 nomod
.TRAN 0.1ns 20ns
.END
```



Save your netlist to file name 'NAND2.sp'. **Attach your printed NETLIST in your report.**

PART 2: Simulation with HSPICE

Add the following statement before '.END', which measures the delay from B to Y.

.MEAS TRAN delay TRIG V(B) VAL='2.5/2' RISE=1 TARG V(Y) VAL='2.5/2' FALL=1

Syntax:
.MEAS TYPE name TRIG V(node) VAL='value' RISE/FALL=number (determine which rise/fall edge)
+ TARG V(node) VAL='value' RISE/FALL=number

Save the change to 'NAND2.sp'.

Put the library file '180_bulk.txt' (from Lab 1) in the same folder with 'NAND2.sp'. Run HSpice to simulate your netlist. Launch CScope , load the file 'NAND2.tr0', and plot the waveforms of V(B) and V(Y).

Attach your waveforms in the report.

In the output info in the file 'NAND2.lis', you can find these lines, which tells you the delay from B to Y.

```
****** transient analysis tnom=  25.000 temp=  25.000 *****
 delay= [redacted]   targ= [redacted]   trig= [redacted]
```

Make a note of the delay.

Now keep input B=1 and A changes from 0 to 1. You can do this by modify the netlist file. Make a note of the delay again this time. Similarly, you could measure the delays for the following cases, and fill in the table:

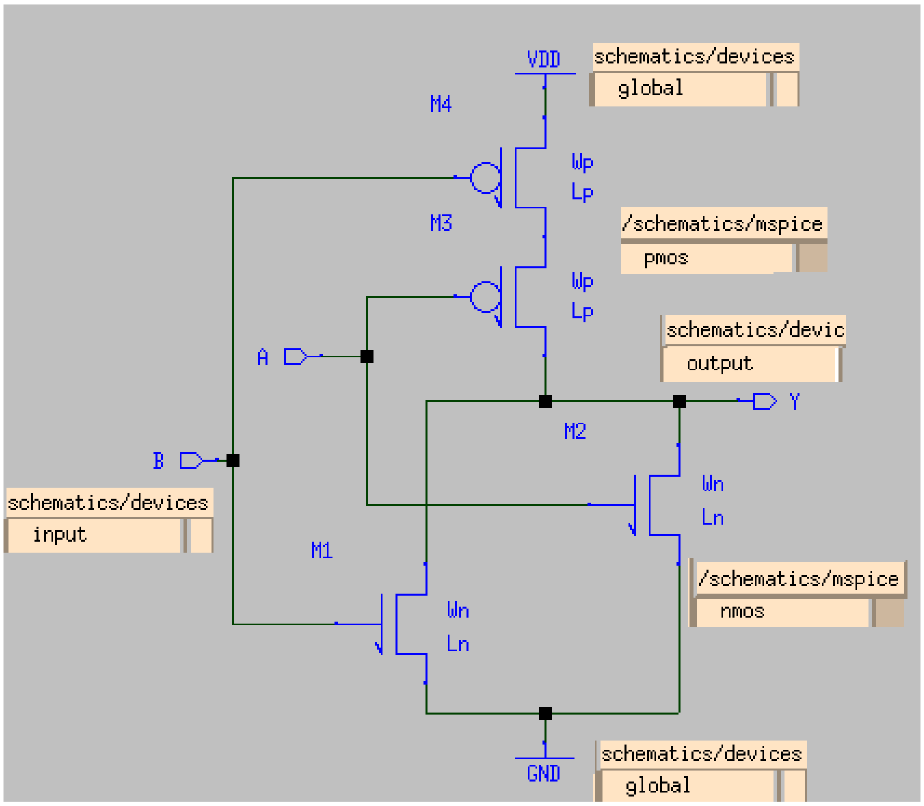
CASES	INPUT A	INPUT B	OUTPUT Y	DELAY
CASE 1	1	0→1	1→0	
CASE 2	0→1	1	1→0	
CASE 3	0→1	0→1	1→0	
CASE 4	0	1→0	0→1	
CASE 5	1→0	0	0→1	
CASE 6	1→0	1→0	0→1	

Explain briefly why these delays are different, and attach your table in the report.

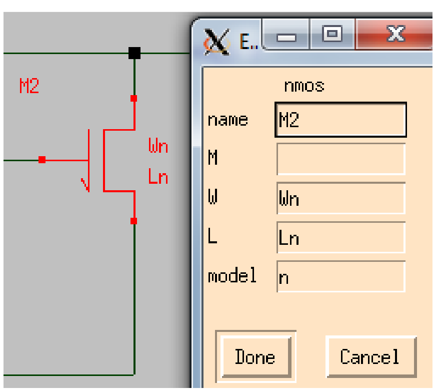
PART 3: Netlist Generation from SUE

Now we practice how to generate HSpice netlist from SUE.

Build NOR2 gate circuit as follows:



The models you will use are marked on the figure. You should give every model a name by double clicking on it. Note that you can modify the sizes of MOSFETs by double clicking on the symbol:



Save your Schematic to NOR2.sue. Click on Sim\SPICE Netlist. You will see this message 'Wrote spice netlist to /…/NOR2.sp'.

Modify your generated netlist as follows:

Orignial	Change to
.OPTIONS post NOMOD post_version=9601	.OPTIONS post=2 NOMOD
.include '${MMI_TOOLS}/sue/ schematics/mspice/mmi25.mod'	.include '180nm_bulk.txt'
.PARAM vddp=2.25	.PARAM vddp=2.5
.TEMP 105	(Remove)
.TRAN 5p 10n	.TRAN 0.1ns 20ns
* .SUBCKT NOR2 A B Y	.SUBCKT NOR2 A B Y
(In all the MOSFET Statements)	Change ‘n’ and ‘p’ to ‘NMOS’ and ‘PMOS’
* .ENDS $ NOR2	.ENDS $ NOR2
(add before .END)	.param Ln=0.18u .param Lp=0.18u .param Wn=0.18 .param Wp=0.72 X1 A B Y NOR2 VA A gnd 0 VB B gnd PULSE vddp 0 5ns 0.1ns 0.1ns 10ns 20ns .MEAS TRAN tpLH TRIG V(B) VAL='vddp/2' FALL=1 + TARG V(Y) VAL='vddp/2' RISE=1 .MEAS TRAN tpHL TRIG V(B) VAL='vddp/2' RISE=1 + TARG V(Y) VAL='vddp/2' FALL=1

Save your changes, and attach your printed netlist in your report.

PART 4: Netlist Simulation

Simulate your NOR2.sp file with HSpice. Plot the waveforms of V(A) and V(Y) in CScope.

Attach the waveforms in your report.

You will also find delays 'tpHL' and 'tpLH' in 'NOR2.lis' file.

Now change the widths of PMOSFETs 'Wp' from '0.72' to '0.36' and '0.9'; Simulate for these two cases and fill this form:

CASES	Wp / Wn	tpLH	tpHL	Average delay tp=(tpLH+tpHL)/2	Rise/fall imbalance |tpLH - tpHL|/ tp
CASE 1	0.36 / 0.18				
CASE 2	0.72 / 0.18				
CASE 3	0.90 / 0.18				

Attach your form to your report.