

## Lab 1 SPICE Simulation

Due 2/14/17 before class

In this lab, we will set up the HSPICE simulation environment, and use it to characterize some devices and a simple diode circuits. The device models can be found in a library `sedra_lib.lib`, which is from our textbook, *Microelectronic Circuits* by Sedra & Smith. You can download this library from Blackboard under Course Materials.

### Set Up and Run HSPICE

Please see the handouts “Using HSPICE” on Blackboard for detailed instructions. If you are new to Linux, read “Basic Linux Commands”. There are also HSPICE manuals for your reference.

### Characterize Diodes

- 1) Find the I-V characteristics ( $I_D$  vs.  $V_D$ ) of a discrete diode D1N4148 using DC sweep.
- 2) Find the saturation current  $I_S$  of the diode using `.MEASURE` command. Assume the effect of its series resistance  $R_S$  can be neglected. Compare to the value specified in the model.
- 3) Find the small-signal diode capacitance  $C_D$  for  $V_D=1V$  using AC analysis.

### Characterize MOSFETs:

- 1) Find the I-V characteristics ( $I_D$  vs.  $V_{DS}$ ) of an NMOS transistor **NMOS0P5** with a size of  $W=20\mu m$ ,  $L=0.6\mu m$ ,  $AS=AD=(W*1\mu m)$ ,  $PS=PD=(W+2\mu m)$ . Note that you need to bias the body of the NMOSFET to ground. Sweep  $V_{GS}$  from 0 to 3V in steps of 0.2 volts, and  $V_{DS}$  from 0 to 3V.
- 2) Find the values of  $V_{DS}$  for which  $I_D$  enters the pinch-off region.
- 3) Find  $f_T$  of the transistor for  $V_{GS}=1V$  and 2V,  $V_{DS}=0$  to 3V. Note: use the same circuit as in the textbook when defining  $f_T$ , i.e., to measure the current gain when the drain is AC short-circuited to ground.
- 4) Repeat the steps above for a PMOS transistor **PMOS0P5** with a size of  $W=20\mu m$ ,  $L=0.6\mu m$ ,  $AS=AD=(W*1\mu m)$ ,  $PS=PD=(W+2\mu m)$ . Note that the source and bulk of PMOSFET needs to be connected to  $V_{DD}$ .

### Voltage Doubler Circuit:

Simulate the following voltage doubler circuit (Fig.1). Plot the transient behavior of the voltage  $V_2$  and  $V_{out}$ . This is the problem from Section 4.6.3 in the textbook Sedra & Smith 6<sup>th</sup> edition. Use the subcircuit in the library `sedra_lib.lib`

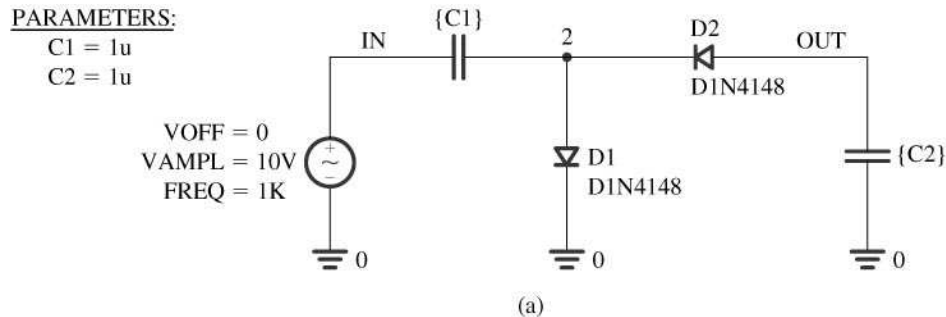


Figure 1. Voltage doubler from Sedra & Smith Exercise 4.6.3.

### Tips:

Here is a tip for HSPICE commands useful for this lab:

DC sweep:

.DC Vin1 0 3 0.2 Vin2 0 3 0.2

This command will change Vin1 from 0 to 3 with 0.2 increments for each value of Vin2 (from 0 to 3V). NOTE that you should not run two .DC commands one after another, which will run two separate simulations.

### Lab Report:

Draw the schematic of all circuits with clear component names and values. Explain your design. Report simulation results except waveforms. Attach printouts of the SPICE netlists and simulation results including waveforms with clear explanations. Discuss any discrepancy from your expectation. **Submit all SPICE netlist files to TA by email.** Note that your SPICE syntax should follow the conventions in the lecture.