

UNIVERSITY OF CALIFORNIA
College of Engineering
Department of Electrical Engineering and Computer Sciences
Last modified on August 20, 2012 by Elad Alon

Elad Alon

HW #1: Circuit Simulation
Due Thursday, Aug. 30th, 5pm, box in 240 Cory

EECS 141

1. Objective

The objective of this session is to give initial exposure to the software environment that will be used in this course throughout the semester.

HSPICE (a circuit simulator) and WaveView Analyzer (a waveform viewer) will be used to execute some of the procedures that are necessary in many lab and homework assignments, in addition to your project, during the course.

2. Tasks

- a. Copy the work files from the EE141 master account (which is ~ee141/) to your home directory. You may want to do this as follows:

```
> mkdir LAB1  
> cd LAB1  
> cp ~ee141/LAB1/CMOSInv.sp .
```

- b. Examine the SPICE deck for the CMOS inverter by typing in the following:

```
> cat CMOSInv.sp
```

The file (CMOSInv.sp) contains the description of a CMOS inverter and the analyses to be performed by SPICE. The schematic of the circuit is shown below in Figure 1.1:

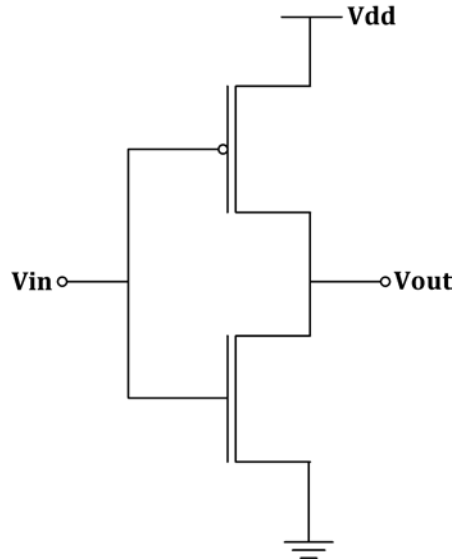


Figure 1.1

CMOSinv.sp:

Simple CMOS Inverter

```
.lib '/home/ff/ee141/MODELS/gpdk090_mos.sp' TT_S1V
```

```
**Begin Circuit Netlist**
```

```
**Power Supplies**
```

```
VDD vdd 0 1.2
```

```
VIN in 0 PULSE 0 1.2 200ps 100ps 100ps 2ns 4ns
```

```
**Transistors**
```

```
M0 out in vdd vdd gpdk090_pmos1v L=100e-9 W=120e-9 AD=69.6e-15 AS=69.6e-15 PD=1.16e-6 PS=1.16e-6 M=1
```

```
M1 out in 0 0 gpdk090_nmos1v L=100e-9 W=120e-9 AD=69.6e-15 AS=69.6e-15 PD=1.16e-6 PS=1.16e-6 M=1
```

```
**Control Information**
```

```
.options post=2 nomod
```

```
.op
```

```
**Transient Analysis**
```

```
.tran .01ns 3ns
```

```
**DC Analysis**
```

```
.dc VIN 0 1.2 0.001
```

```
.END
```

There are three main sections in the spice deck:

- The netlist description. A netlist is a computer readable representation of the circuit schematic.
- The models used. A model in SPICE contains the parameters of the equations used by SPICE to analyze certain elements (such as transistors) in the circuit.
- The analysis to be performed during the simulation. We are requesting a transient analysis and a DC analysis of the circuit.

c. Simulate the circuit netlist.

```
> hspice CMOSinv.sp >! CMOSinv.out
```

Upon proper completion of the simulation, you should see the following:

```
> info:          ***** hspice job concluded
```

d. Definitions of the output files:

- CMOSinv.sp is the input netlist (your file)
- CMOSinv.sw0 is the DC sweep data output (used by awaves)
- CMOSinv.tr0 is the transient data output (used by awaves)
- CMOSinv.out is the output listing from HSPICE (look here for errors or text about the circuit)
- CMOSinv.st0 is the simulation run information (usually not useful)
- CMOSinv.ic0 is the information about the initial condition computed by HSPICE (usually don't need to look at this)

e. View the results of the transient analysis.

Load WaveView Analyzer and examine the results of the transient analysis by entering the following:

```
> awaves &
```

or

```
> wv &
```

Once WaveView loads, click on **File/Import Waveform File** or use shortcut Ctrl+O. This will open a menu to select which output file to display. Your output *CMOSinv.sw0* as well as *CMOSinv.tr0* should be listed; if not, switch to the correct directory using the tab or the arrows. Once you have found your output files, double-click on *CMOSinv.tr0*, which should import the transient data into **Output View** window.

To view the transient waveforms, click on the “+” sign in front of the file name “D0:CMOSinv.tr0” and then double click on “*toplevel*”. Now you should see the waveform data listed below. To see the waveform, you can just double click on the waveform name and it will appear in a new panel on the right. In our case, we want to plot the waveforms for **in** and **out** in the same panel. To do this, select **v(in)** by single

clicking on it, then hold “Ctrl” and double click on **v(out)**. You should see both waveforms appear on the same panel.

Print the waveforms for **in** and **out**.

- f. View the results of the DC analysis.

Open a new panel for the DC waveform by clicking on **Panels/New/X-Y**. Click on **File/Import Waveform File** again to import *CMOSinv.sw0*. Then click on **v(out)** to print the output waveform in a new panel.

Print the waveform for **out**.

- g. Transform the CMOS inverter into an NMOS inverter with a passive load.

Replace the PMOS transistor with a resistor as shown in Figure 1.2

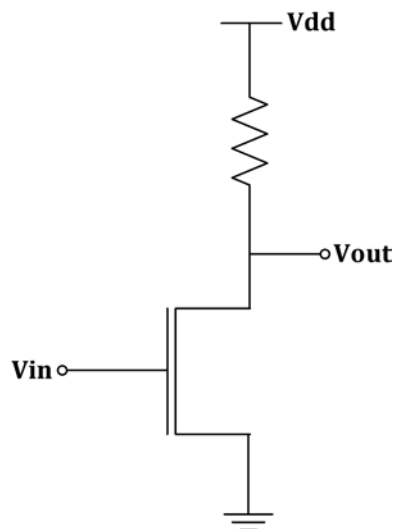


Figure 1.2

Edit the CMOSinv.sp file by replacing the PMOS transistor with a 20k resistor in your netlist.

Delete: M0 out in vdd vdd gpdk090_pmos1v L=100e-9 W=120e-9 AD=69.6e-15
AS=69.6e-15 PD=1.16e-6 PS=1.16e-6 M=1

Replace with: R1 vdd out 20k

- h. Simulation and analysis of the NMOS inverter.

Repeat steps c-f with this modified file.

3. Report

For your report, all that is required is the following:

- A printout of the modified SPICE input file
- Plot of the transient response of the CMOS and NMOS inverters
- Plot of the DC transfer characteristic of the CMOS and NMOS inverters