Introduction

At multiple stages of the CAD flow, it is important to simulate your design for functionality, speed, and power to make sure it is continuously meeting your expectations. This lab will help you become familiar with simulation of your digital circuits using HSPICE. Your tasks will be separated into three sections. The first section involves setting up the necessary folders and files in your Unix environment, and understanding how to set up simulations in 0.25µm. In the second section we will begin with some HSPICE inverter simulations and familiarize ourselves with some simple HSPICE commands, as well as the tool used to view output waveforms. Section three will have you determine an approximation of the $V_{th}$ of the MOSFET by looking at its I-V curve. By the end of this lab you should have a greater understanding of the following objectives: how to feed a signal into a digital circuit and verify proper functionality by viewing output waveforms; and how to determine the delay, static and dynamic power, and energy.

Setup

Use the following steps to setup your lab.

1. From your Unix home directory, create a folder called ECE471 and within that, a sub-folder titled Lab1.
2. Download the `inverter.spi` file from the lab website and place it in your Lab1 folder. This is an HSPICE netlist for an inverter, with a supply voltage `vdd`, a ground node labeled `vdd!`, an input labeled `in`, and an output labeled `out`.
3. Download the `sim_deck` file from the lab website, this is the template we will use the make any changes to our simulation parameters. Before advancing, take a moment to read through the file and understand what each command is doing. (There should be descriptions for all of them) If you have questions don’t hesitate to ask.
4. Make the necessary changes to the `sim_deck` to make it work for the 0.25µm technology. This involves changing the voltage supply value, un-commenting two lines, and placing comments before one other. (Commented lines are those that start with a `*` symbol) By un-commenting the lines for the 0.25µm library, you are telling HSPICE to run the simulation using transistors from that technology.

Simulating the inverter

The following steps will guide through the process of simulation.

1. Before running the simulation make sure you understand which command controls the input pulse to the inverter, and what the `.measure` statements do.
2. To run an HSPICE simulation, on the Unix command line type:

```
hsypressim_deck > sim_deck_out
```

This runs a simulation using your `sim_deck`, and puts the output in a file called `sim_deck_out`. It will create this file if it does not already exist, and overwrite it if it does. Note: If there is an error and your HSPICE simulation is aborted, look in this output file to see the description of the error.
3. Once the simulation is finished, take a look at the files that exist in your Lab1 folder. The `sim_deck.mt0` file contains all of the values calculated using the `.measure` statements, in this case it will be the $t_{PL\to H}$ and $t_{PH\to L}$ values. You should also see a `.st0` file. This will be necessary for the next step.

4. On the Unix command line, type `Scope`. Once Cosmos Scope is opened, go to `File → Open → Plotfiles`. Select the filetype to be 'HSPICE(*.tr*,*.ac*,*.sw*,*.ft*)'. And open the `sim_deck.tr0` file.

5. In the `sim_deck.tr0` window, select your $v_{in}$ and $v_{out}$ signals. The waveforms will show up in the graph window, and if your output is inverting your input, then your simulation was successful. See figure 1.

![Figure 1: Screen shot of example Cosmos Scope window.](image)

6. Now we are going to determine the static power, dynamic power, and energy/computation of the inverter. This will be done by inserting `.measure` statements into the `sim_deck`. Take a look at the description of the `.measure` statement in the HSPICE command reference guide found on the lab website.

   - Finding the static power is a matter of determining the current flowing while no transistors are switching, and multiplying that by `vdd`. This can be found by measuring the average current over a small sample of no switching activity.
   - To determine the dynamic power, find the average of the current during the period that the transistors are switching. For the inverter, we will define this period from when the output voltage is at 10% of `vdd` to when the output voltage is at 90% of `vdd` (this is when current is being sourced from `vdd`).
   - Finally, energy can be computed by taking the integral of the current over a period of time and multiplying it by `vdd`. Energy/computation is found by doing this during the switching period.

7. Finally, change the rise and fall times of the input voltage source to 500p and rerun the simulation. How does this change the measured values?
Determination of $V_{th}$ from Simulated I-V Curve

Use these steps to determine $V_{th}$.

1. First you will need to create a new netlist for this simulation. In this, you need to instantiate just one nmos transistor. Connect the source and base to ground, the gate to a voltage source node $V_G$, and the drain to a source node $V_D$.

2. In your sim_deck you will need to create a constant voltage source $V_D$. Make this value small (~50mV).

3. Instead of doing a transient analysis (using the .tran statement like we did before) we will be doing a voltage sweep using the .dc parameter. Look at the HSPICE reference manual for more information on this command. For this simulation we will want to sweep the $V_G$ voltage source from 0V to 2.5V.

4. Before running the simulation, be sure that you have included your new netlist, and excluded the inverter netlist. Also, you will need to comment out the .tran statement.

5. Run the simulation, then open up Cosmos Scope and view the graph of the current $I_D$. This is your I-V curve. $V_{th}$ can be found by finding the slope once $I_D$ begins to linearly increase, and extrapolate it down to the $V_{GS}$ value where it crosses zero.

What to turn in

For the lab write-up, please give a brief overview of what changes were necessary in each section to successfully run the simulations. Include any important results (delay/power/energy values, waveforms, extrapolated $V_{th}$ value, etc.) as well as your final sim_deck and new HSPICE netlist. Also, if you ran into any difficulties, briefly explain them and what you had to do to fix them. Turn in one lab per group, in PDF form, to TEACH by 11:59pm on Friday, January 20th 2012.