CSE/ELE 664 – Intro to SoC Design, Fall 2021
Lab1 – HSpice and CosmosScope

Content

1. Introduction

This tutorial introduces you to the Synopsys HSPICE and CosmosScope tools. HSPICE is a program that takes in a netlist, containing a circuit description and analysis options, and outputs the specified analysis it has done on the input circuit. A typical HSPICE netlist file has the .sp extension (e.g. *filename*.sp). HSPICE produces many output files., some are always generated while others are generated if certain options are included in the netlist (e.g. a .mt file is generated if the option .measure is used in the netlist).

CosmosScope is a tool that allows you to graphically display the results of the analysis done in HSPICE. It also allows you to perform different measurements on the loaded waveforms from HSPICE. This tool has some additional functionality that won't be explored in this tutorial.

In this tutorial you will learn the following:

- Invoking HSPICE to perform needed analysis.
- Invoking CosmosScope to display waveforms
- Using CosmosScope to perform different measurements.

2. Setting up HSpice Environment

1. How to Invoke HSpice and Scsope

This section, we will try to invoke HSpice.

In your terminal window, connect to Synopsys/Cadence servers by running the following script:

   qsh

For Linux/OSX user, use command ssh -X ecs-linux.syr.edu -l username(NetID). After connecting the server, you can use it as you are in the qsh window.

For MobaXterm user, set your remote host as **ecs-linux.syr.edu**.

Run the following script to setup the Synoposis environment

   syn

PS: If the terminal says "LD_LIBRARY_PATH: Undefined variable", type in the command **which hspice**, if it shows you a path, then keep going to the next step. If it says "Command not found", specify the server and try it again.

Create a working directory in your root directory. If it already exists, you can skip this step

   mkdir hspice

Change to your working directory

   cd hspice

Copy HSPICE library file "mosfet.lib", input files "mosfet.sp" and "inv.sp" into "hspice " folder. Take a look at these files.

   cp /Linux/sw/request/cadence/local/cse464/mosfet.sp ./
   cp /Linux/sw/request/cadence/local/cse464/mosfet.lib ./
   cp /Linux/sw/request/cadence/local/cse464/inv.sp ./

Run HSpice to simulate the file

   hspice mosfet.sp

A log message will be generated and shown in your qsh terminal. It contains all the info about the netlist, measurements, warning and error messages, model information and more. The message must always be checked to make sure that there is nothing wrong with the netlist or analysis.
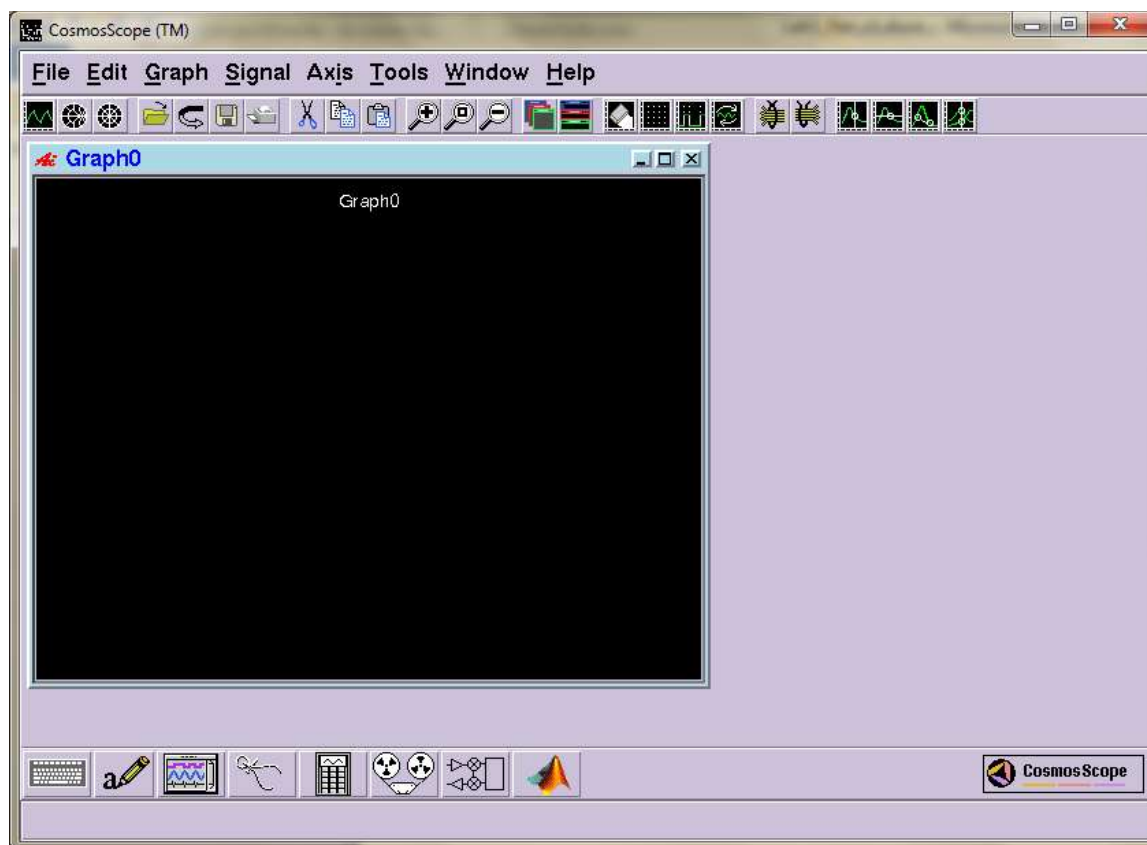
PS: when simulate the given "inv.sp" file, tphl and tplh will aslo be shown in the log message.

If .sp file is correct, some new files will be generated. Type in *ls* to check them.
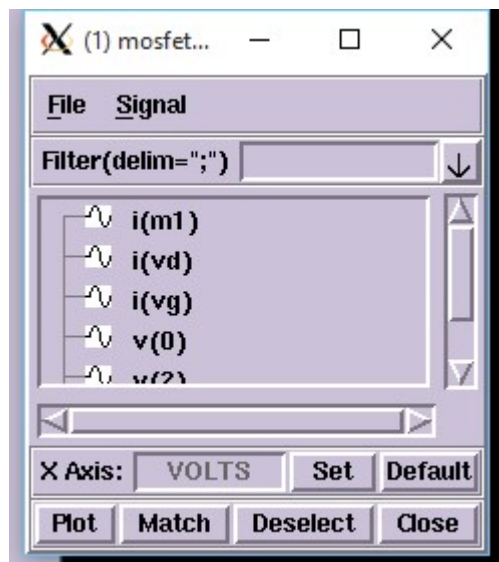
Invoke cscope waveform viewer

   cscope &

Note that *&* is used to keep the current shell active while using the tool.

2. Load a Waveform File and Display Plots

Select the menu item *File > Open > Plot files*, then select the plot file "mosfet.sw0"

In the signal window, select *I(vd)* and *I(m1)* (choose multiple targets by pressing "Ctrl" button while click) and click plot.
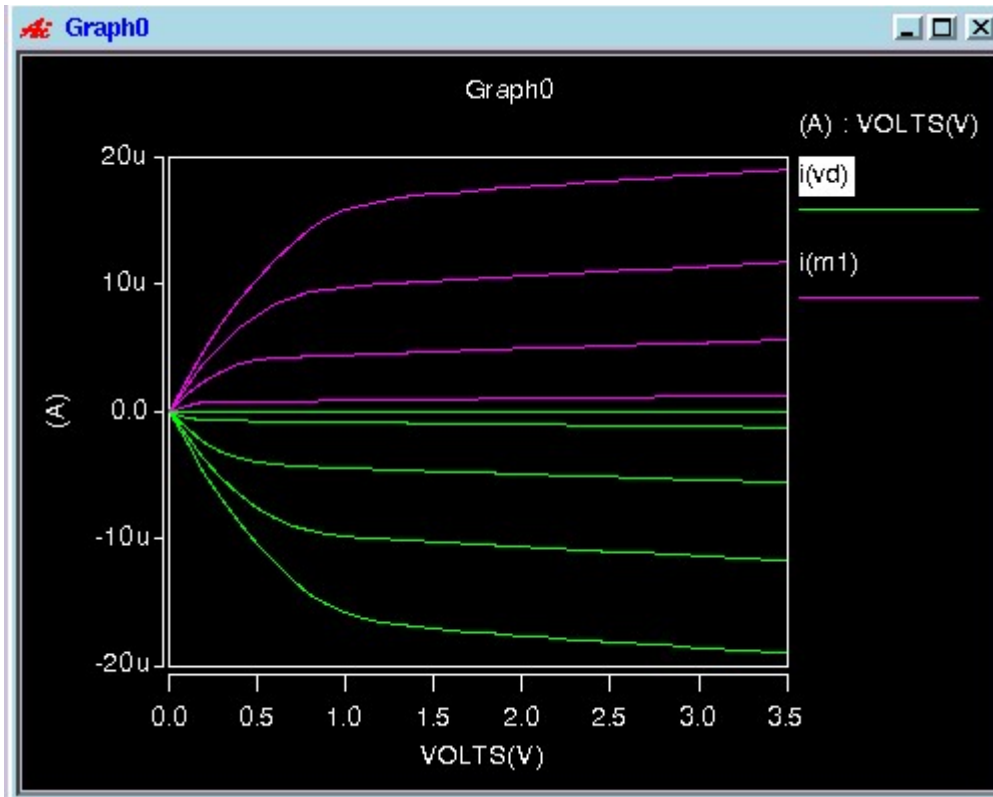
3. Merging Multiple Waveforms

Assume we want to merge the waveforms into a single plot. We want to move the plot of i(m1) to the plot of i(vd).

Right click on the legend name of the waveform i(m1).
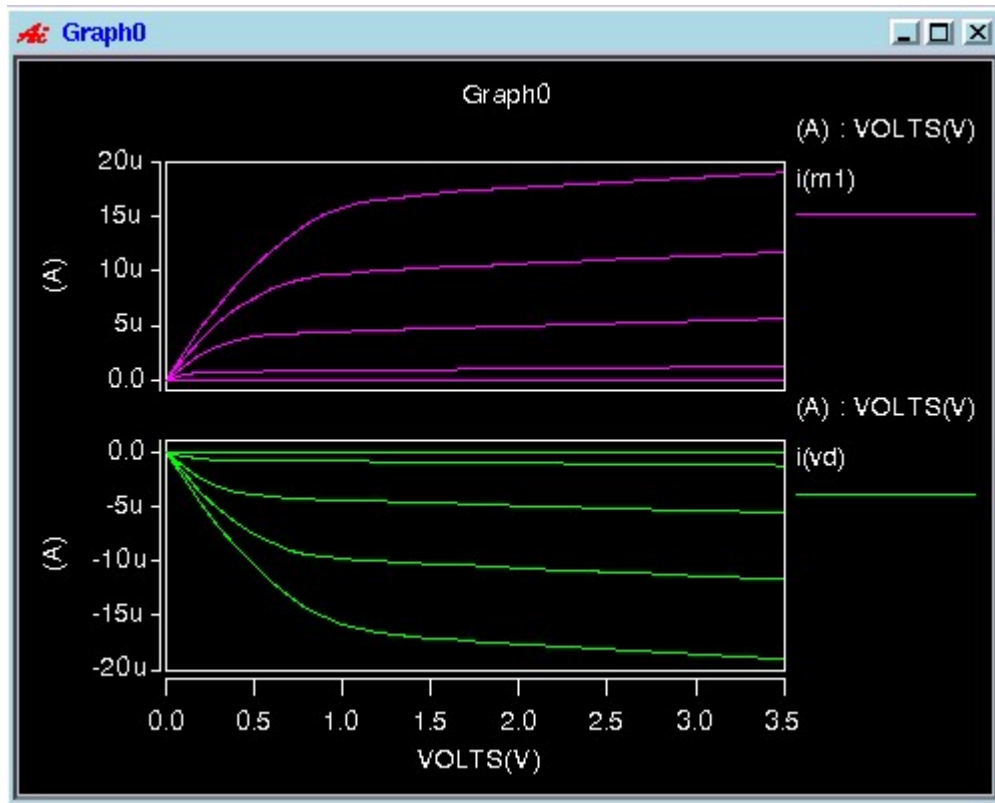
On the displayed menu click **Move to Stack Region > Analog0**

The top most waveform displayed is the most recent one. So the bottom one has the index 0, and this index increases as you go up. The graph would look as follows after the 2 signals are merged together.

4. Unmerging Multiple Waveforms

If you want to unmerge 2 or more waveforms and have them displayed in different plots, you need to do the following:

- Right click on the legend name of the waveform you want to display separately.
- On the displayed menu click **Move to Stack Region > New Analog**

5. Deleting a Waveform

To delete a waveform from the graph, right click on legend name of the waveform then click on **delete.**

6. Performing Measurements

We will learn how to measure the value of current in a given voltage.

- Click on the measurements icon on the bottom of the main window.

On the "Measurement window":

- In "Measurement" click **At X > General > Vertical Marker**
- Select **Apply to all signals in current graph**
- In "Location" type in the X value 3.0
- In "Apply measurement to" select **Entire waveform**
- Click **Apply.**

On the graph you will see a vertical line displayed with an x value of 3.0V and small circle on it. You can drag the circle up and down to choose which line to measure.

7.   Measure from point to point

Now we go back to "qsh terminal" and run hspice to simulate "inv.sp". After simulation is done, open "inv.tr0" in comos scope (cscope). Plot v(1) and v(2) in one graph.

On the "Measurement window":

- In "Measurement" click **At X > General > Delta X**
- In "Number of Signals" choose 2 if the two points locate in different waveforms
- In "Signal" choose one of the waveforms and in "Ref. signal" choose another
- In "Location" type in the Y1 and Y2 value. In our case of inv, they should be both 1.5
- In "Apply measurement to" select **Defined X and Y Range**
- In "X range", type in "200n" and "500n", for example.
- Click **Apply.**

Then you will get the tplh time of the inverter.
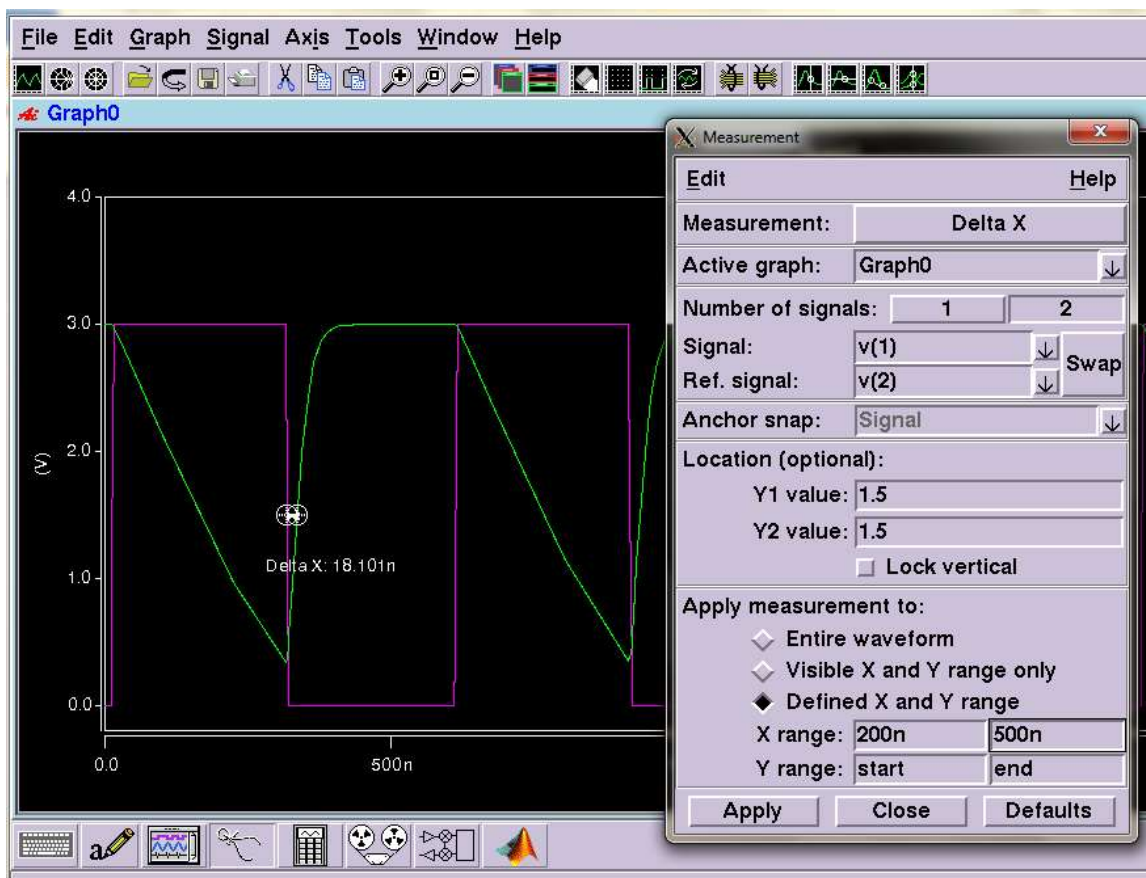
8.   Deleting Measurements
- Delete a single measurement: right click anywhere on the measurement and click **Delete.**

- Delete all measurements: right click anywhere on the graph and select **Delete All Measurements.**

3. Task

1. Find out the equivalent resistance of the transistor in (mosfet.sp) for V_{DD}=3.0V.
   - (Hint: measure the current of the transistor when the V_{DS}=V_{GS}=3V, also measure the current when V_{GS}=3V, V_{DS}=1.5V. Calculate the average resistance over the range V_{DS}=1.5~3V.)
   - Note that since we use the 0.5u technology, the threshold voltage is more than 0.9V, which is different from the data given in our textbook.
2. Measure tphl and tplh of the inverter (inv.sp).
3. Increase the load capacitance in "inv.sp" from 0.2pF to 2.0 pF in step of 0.2pF and measure the delays. Plot the relation between capacitance and delay. (In report, you should include measurement for all capacitance values. You can use EXCEL, MATLAB or other software to plot the capacitance and delay relationship.) Hint: You should observe and think whether your plots are correct.
4. Design a CMOS circuit to implement the NAND gate. Set the gate size of PMOS and NMOS to be L=0.5U and W=1.0U, the load capacitance to be 0.2 pF. Measure the low to high delay and the high to low delay when both inputs switches from 00 to 11 and 11 to 00. How many possible ways to make the output switch from low to high? Do they all have the same propagation delay?

The lab report should be submitted through Blackboard. An electronic version is highly recommended. If you write on the papers by hand, please scan them into electronic version and make sure that it is clear enough. You will have **unlimited** attempts to hand in the report and only the newest one will be graded.