SIMULATION ENVIRONMENT SETUP GUIDANCE AND TOOL PRACTICE

Jiahao Kang, your TA (jiahao_kang@ece.ucsb.edu)

The objective of this session is to give initial exposure to the software environment (HSpice, CScope /AWaves, MAX, and SUE) that will be used in this course throughout the quarter. HSpice (netlist simulator), AWaves (waveform analyzer for Windows), CScope (waveform analyzer for Linux), MAX(layout editor), and SUE(schematic capture tool) 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.

PART 1: SIMULATION ENVIRONMENT SETUP

### STEP 1: BEFORE YOU GO TO THE LAB, APPLY FOR AN ECI ACCOUNT

Make sure your account at engr.ucsb.edu is working. You can apply for a computer account directly on the ECI website (http://www.engr.ucsb.edu/eci/) by clicking the "New Account".

> *If encounter any difficulties with that you can also send an email to help@engineering.ucsb.edu or fill a form in ECI main office in 3110 Engineering 1 (Mon-Thurs 9AM-12 noon && 1PM-4PM). It takes 24hrs for the account to be active. You need to change your password within 7 days of your account being activated.*

For those who do not want to go to the lab, you may do lab homework remotely. Please jump to Appendix 1. However, going to the lab is recommended.

STEP 2: GO TO THE LAB

The lab is at HFH 1140. Lab hour and TA office hour and are the same time (Monday Evening) there.
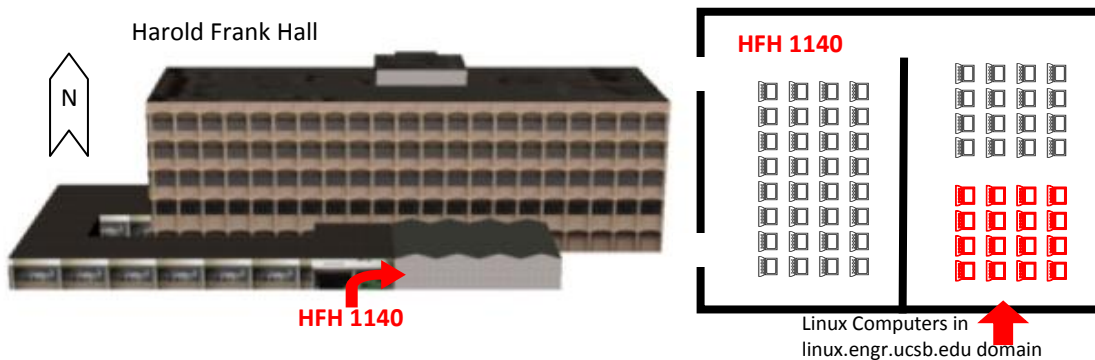
> *The Linux computers in red in the above figure are in the domain of linux.engr.ucsb.edu. Please use one of them.*

STEP 3: SET ENVIRONMENT

3.1.   Choose a computer and login using your ECI account;

3.2.   Option 1: Start an explorer; Download the BASH Shell start file (.bashrc) from our course website under Resources section. Put the file under your home folder (replace it if existing).

> *Option 2: Manually Setup (for advanced Linux users only): See Appendix 2.*

3.3.   Start a Terminal, then type "bash" in the command line to start the shell you set up.

3.4.   Type "hspice", "scope", "max" or "sue" to start software to test the environment.
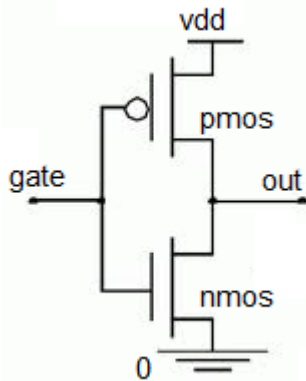
PART 2: TOOL PRACTICE

1. HSPICE PRACTICE

Here gives you an example about using Hspice.

NETLIST AND LIBRARY

Copy the inverter netlist (inv.sp) from ECE 124A course website and .18 micron CMOS process model from http://ptm.asu.edu under latest model (180nm_bulk.txt).

The schematic of the circuit is shown below:

There are three main sections in the spice netlist:

[1] The models used. A model in hspice is a description of the parameters of the equations used by spice3 to analyze the circuit.	.include '180nm_bulk.txt'
[2] The netlist description. Netlist is a designation for a computer readable representation of the circuit schematic.	vdd vdd 0 5 vin in 0 pulse 0 5 2NS 2NS 2NS 30NS 60NS M1 out gate 0 0 NMOS L=2u W=100u M2 out gate vdd vdd PMOS L=2u W=200u
[3] The analysis to be performed during the simulation. We are requesting a transient analysis and the DC Transfer Characteristic of the circuit.	.options post=2 nomod .op .TRAN 1ns 30ns .DC VIN 0 5 0.1

SIMULATE THE CIRCUIT NETLIST.

In the terminal, type

hspice inv.sp

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

> info: ***** hspice job concluded

2. COSMOSCOPE PRACTICE

DEFINITION OF OUTPUT FILES

inv.sp is the input netlist (your file)

inv.sw0 is the DC sweep data output. (used by cscope/ awaves)

inv.tr0 is the transient data output. (used by cscope/ awaves)

inv.lis is the output listing from HSPICE. (look here for errors or text about the circuit)

inv.st0 is the simulation run information. (not useful)

inv.ic is the information about input to HSPICE (not useful)

VIEW THE RESULT OF THE TRANSIENT ANALYSIS

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

scope

Once CScope loads, click on File/Open. This will open a menu to select which file to display. Your output files should be listed, otherwise, switch to the correct directory. Once you have found your output files, doubleclick on inv.tr0, which should open the Signal Manager to view the transient waveforms.

VIEW THE RESULT OF THE DC ANALYSIS

Open inv.sw0 and view the DC waveforms.

EXAMPLE OF WAVEFORMS
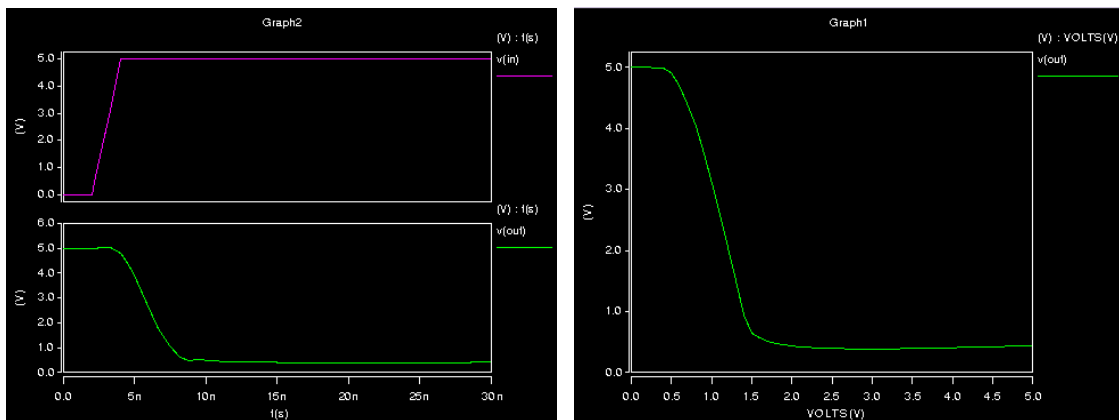
There is a screenshot button on the keyboard.

If you meet any trouble making screenshots, try "File->export image" option in CosmoScope.

Or take a photo of your screen using your cell phone…

3. MAX PRACTICE

Draw a standard inverter cell (as shown in Fig. 1.40 in the textbook) with a unit of NMOS and double-sized for PMOS by MAX. **Your print out should show that no DRC error exists on graph by including the "DRC clean" string at the left bottom corner of MAX screen.** Grid and labels for Vdd, Gnd, In and Out are needed. Refer to MAX manual on course webpage for help.
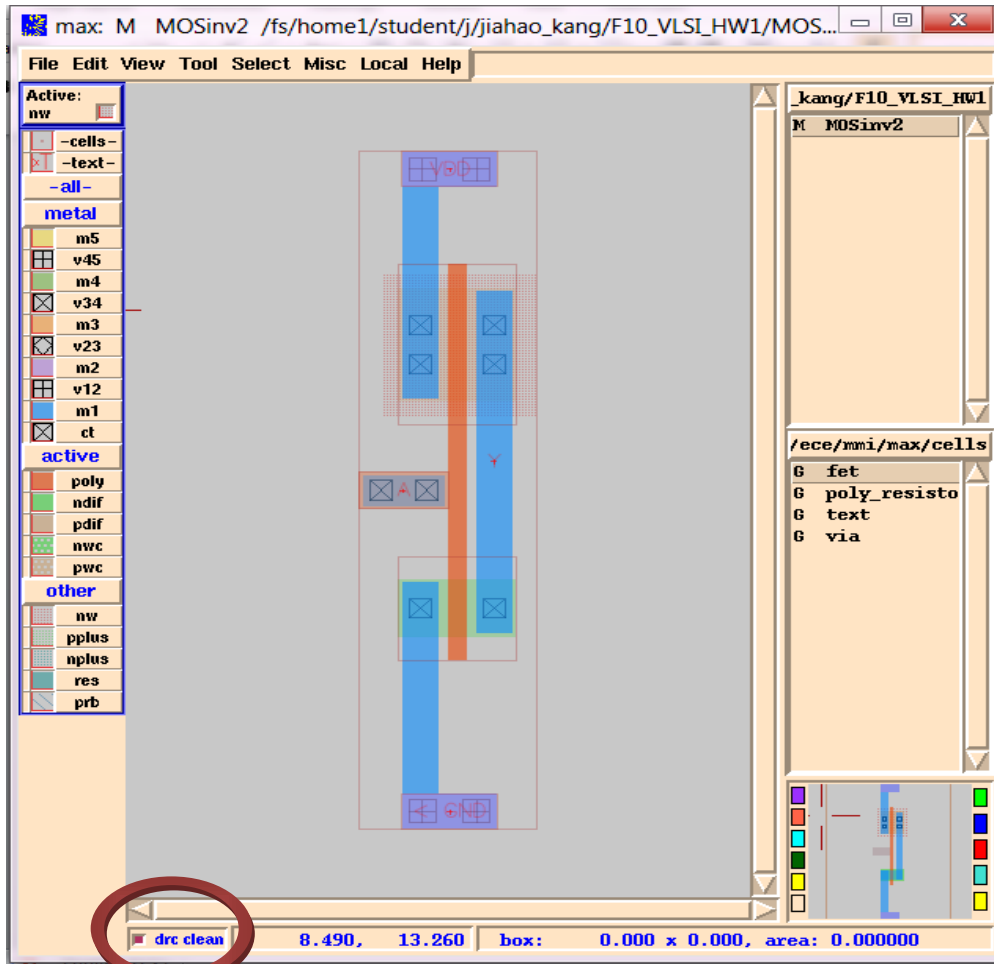
Please make sure:

- Your print out should show that no DRC error exists on graph by including the "DRC clean".

- You should have an N-Well under your PFET.

- Your layout should have texts "Vdd", "Gnd", "A" and "Y" at input/output nodes. To do so, select the metal electrode, and then press "t" to add a text.

- You may use 3D view to check your layout. It is located at "View->View in 3D", or just press "Alt+Z". To exit 3D mode, right-click or press "Esc".
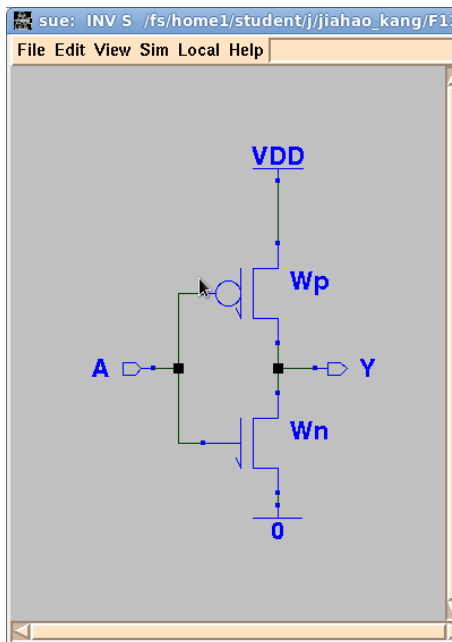
4. SUE PRACTICE

Draw a standard inverter cell (as shown below) with a unit of NMOS and double-sized for PMOS by SUE.

The nmos and pmos models are in the library on the right side of SUE window under:
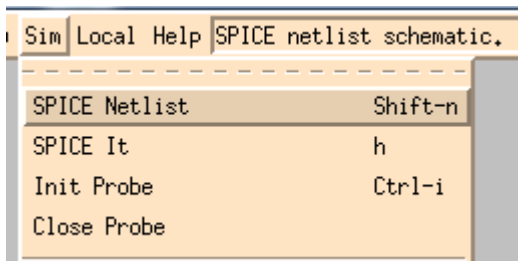
/ece/mmi/sue/schematics/mspice

VDD, ground, pins are under:

/ece/mmi/sue/schematics/devices

Now generate a hspice netlist using:

You will see the info:

Wrote spice netlist to /fs/home1/student/j/jiahao_kang/VLSI_F12_Lab1/inv2.sp

APPENDIX I (FOR ADVANCED USERS ONLY) REMOTE

If you want to do lab homework remotely, for example, at home, you may try the following steps:

FOR WINDOWS COMPUTERS

In order to work remotely, you will need tools in three categories:

> SSH Client *(to run simulation tools on the server) Examples:* **PuTTy**, **OpenSSH**, **SecureCRT**, *etc.*

> X Server *(to start graphic UI) Examples:* **Xming**

> SFTP Client *(to upload/download your files) Examples:* **FileZilla**, **SecureFX**, *etc*

I recommend using **Bitvise Tunnelier** *(combining* SSH Client *and* SFTP Client*) with* **Xming***.*

Step 1:   Download **Bitvise Tunnelier** (free software) at http://www.bitvise.com/tunnelier and install;
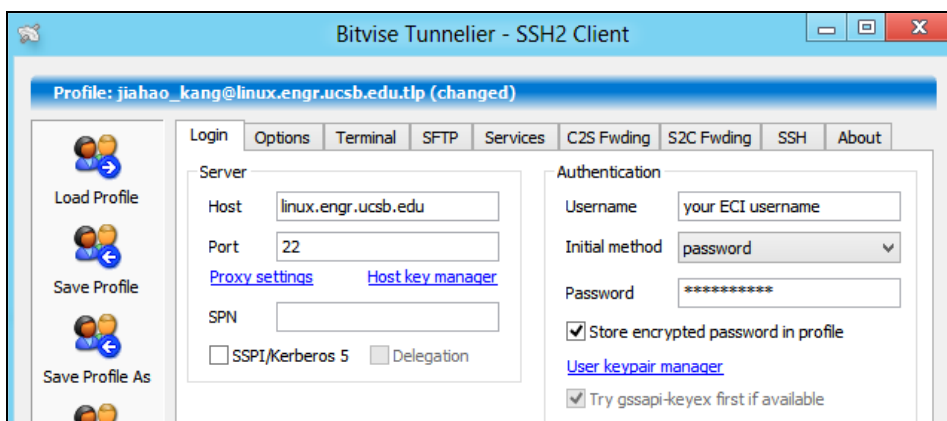
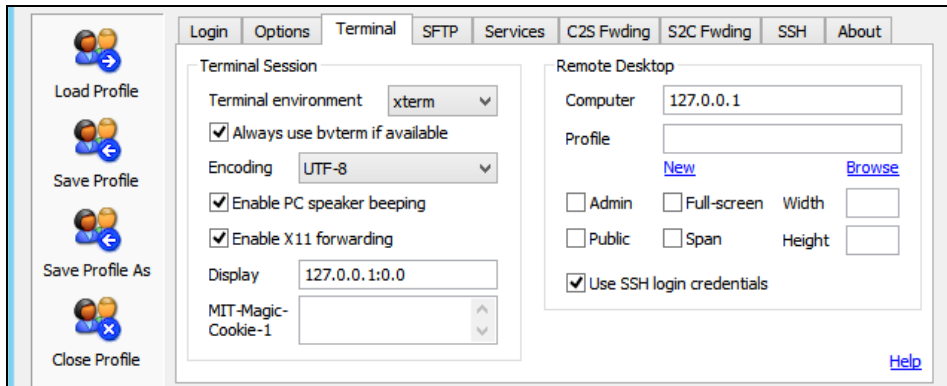Step 2:   Download **Xming**(free software) at http://www.straightrunning.com/XmingNotes/ and install;

Step 3:   Run **XLaunch** in your **Start Menu** and select the display option you like:
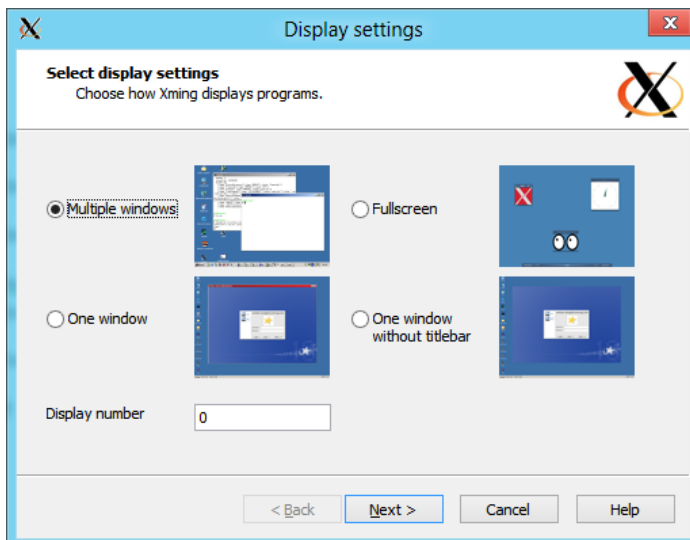
Press **Next>** for several times and then **Finish**. Next time, run **Xming** directly.

Step 4:   Run **Tunnelier** in your **Start Menu** and enter the information below:
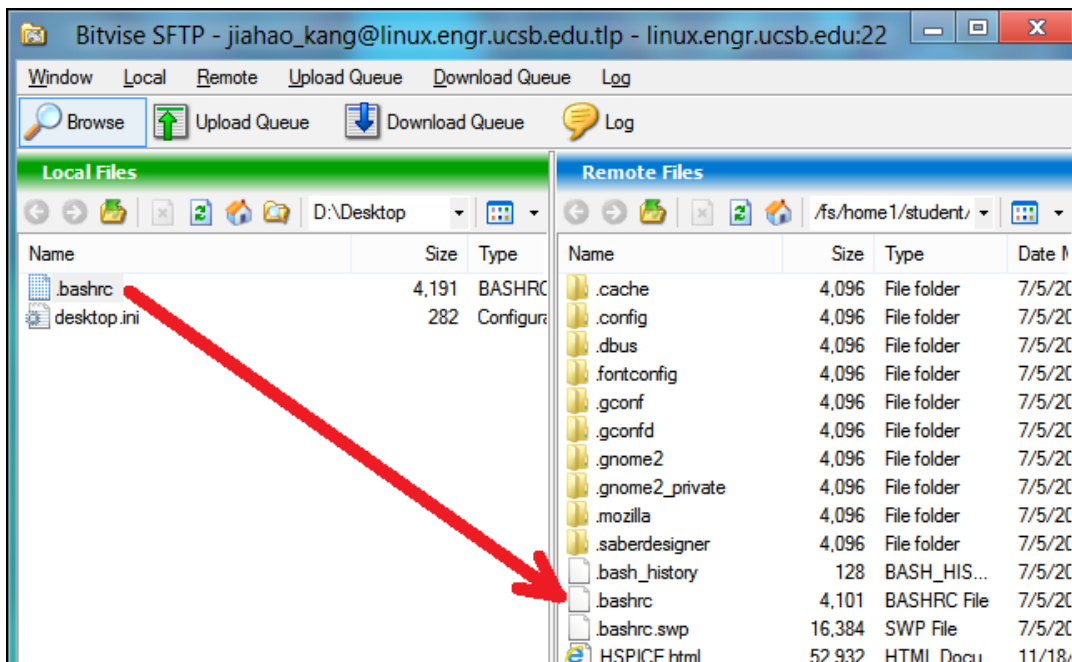
Press icon **Save Profile** for future use.

Press **Login** and then **Accept and Save** the Host Key Verification.

Step 5: Option 1

Download .bashrc file from course website;

Press icon **Open New SFTP Window** in tunnelier window;

Drag .bashrc to your remote home folder (replace it if it exists)

Step 5: Option 2

Set the starter file by yourself. Refer to Appendix 2

Step 6: Press **Open New Terminal Console** in tunnelier window; Type "**bash**" to start shell;

Step 7: Type "hspice", "scope", "max" or "sue" to test simulation tools.

FOR LINUX COMPUTERS

Step 1: Your computer should support **X11** display; To start X Server, type "xinit" (or "sudo xinit") in terminal;

Step 2: then use SSH command to login remotely:

ssh linux.engr.ucsb.edu –l yourusername

Step 3: Option 1

Download .bashrc file from course website ;

Replace the .bashrc to your remote home folder (replace it if it exists)

Step 3: Option 2

Set the starter file by yourself. Refer to Appendix 2

Step 4: Open a new **terminal**; Type "**bash**" to start shell;

Step 5: Type "hspice", "scope", "max" or "sue" to test simulation tools.

APPENDIX 2 (FOR ADVANCED USERS ONLY) SET ENVIRONMENT VARIABLES

If you would like to set the starter file by yourself, determine which shell you are using: Bourne Again Shell (BASH) (recommended) or C shell (CSH). Then open the starter file (.bashrc or .cshrc) with any text editor (vi, vim, gedit), and add these contents to the end of the file: (green lines are commentaries)

Bourne Again Shell (BASH) .bashrc
Synopsys HSPICE & CSCOPE
export PATH=$PATH:/ece/synopsys/hspice/E-2010.12/hspice/bin
export PATH=$PATH:/ece/synopsys/CosmosScope/D-2009.12/ai_bin
export LM_LICENSE_FILE=$LM_LICENSE_FILE:1717@license.engr.ucsb.edu
export LM_LICENSE_FILE=$LM_LICENSE_FILE:1781@license.engr.ucsb.edu
Micro Magic (MAX & SUE)
export MMI_TOOLS=/ece/mmi
export PATH=$PATH:${MMI_TOOLS}/bin.i486-linux
export MMI_LICENSE_SERVER=license.ece.ucsb.edu

C Shell (CSH) .cshrc
Synopsys HSPICE & CSCOPE
set path=(/ece/synopsys/hspice/E-2010.12/hspice/bin $path)
set path=(/ece/synopsys/CosmosScope/D-2009.12/ai_bin $path)
setenv LM_LICENSE_FILE 1717@license.engr.ucsb.edu:1781@license.engr.ucsb.edu
Micro Magic (MAX & SUE)
setenv MMI_TOOLS /ece/mmi
set path=(/ece/mmi/bin.i486-linux $path)
setenv MMI_LICENSE_SERVER license.ece.ucsb.edu