

# ECE558/658 VLSI Design – Fall 2021

## Lab 0: Setting up tools and accounts

**Objective:** The purpose of this lab is to get you familiar with working on the vlsicad servers, and to set up your access to all of the tools that you will use in the remainder of the course.

**Introduction:** In this course, students taking ECE558 will have accounts on vlsicad1.ecs.umass.edu and students taking ECE658 will have accounts on vlsicad2.ecs.umass.edu. You will need to adjust the instructions according to which server your account is on. If your enrollment status has changed since the accounts were created, then you may not have an account. If you find that you don't have an account, contact the TAs and they will add you. Your username is the string associated with your @umass.edu email.

This document provides a few example Unix and spice commands to help you get started, but it does not provide a full tutorial with everything you may need. This is intentional and not malicious. You are encouraged to seek out additional resources, and to make use of the TA office hours whenever you have questions about the labs. Useful Unix tutorials can be found here:

- <https://people.ischool.berkeley.edu/~kevin/unix-tutorial/toc.html>
- <http://www.ee.surrey.ac.uk/Teaching/Unix/>

### Step 1: log in to server and set up environment

Log into vlsicad2 using SSH with X11 forwarding so that you can work in a terminal and interact with GUI programs from your local machine. Mac users may need to install XQuartz for X11. Here are a couple suggested methods, other programs can also be used:

- On mac or linux, you can log in to vlsicad2 by opening a terminal and using the command  
`ssh -Y username@vlsicad2.ecs.umass.edu`
- Windows users can use the program Mobaxterm: <https://mobaxterm.mobatek.net/download-home-edition.html>  
A demo of Mobaxterm is given here: <https://mobaxterm.mobatek.net/demo.html>

Your account has a temporary password of Vlsi\_2021 that will expire after first login. After logging in, you will be prompted to enter your current (temporary) password once more, and then asked to enter twice a new password. Make sure you record your new password in multiple places so that you don't lose access to your account.

You interact with Unix using a shell to enter commands. Check that your shell is tcsh by using `echo $SHELL`  
vlsicad2:~echo \$SHELL  
/bin/tcsh

The zip file provided on Moodle for this lab contains **.cshrc** and **cadsetup** files. You will need to put these files in your home directory on vlsicad2 to set up your access to the VLSI CAD tools. Note that Windows programs for extracting the files may add Windows-specific linebreak characters that cause problems on Unix. To avoid this, you can copy the zip file over to the server and unzip/unpack it there.

- Example command that will copy lab0.tar.gz from your local computer to vlsicad2:  
`scp lab0.tar.gz dholcomb@vlsicad2.ecs.umass.edu:~/`
- Example Unix command to unzip and unpack the files:  
`tar -xzf lab0.tar.gz`
- File **.cshrc** is a configuration file that will get executed whenever you begin a shell by logging in to vlsicad2
- Note that the provided **.cshrc** file has in it a command to source **cadsetup**. This ensures that the commands in **cadsetup** are executed each time you log in to vlsicad2, which allows you to use the VLSI CAD tools.

Logout from vlsicad2, and then log in again using your new password. Now you should have access to the tools. Check your access to hspice, cscope, dc\_shell, pt\_shell, icc\_shell, virtuoso, and vsim by executing the eight commands shown below. Paste the output in your lab report, similar to what I've shown.

```
vlsicad2:~$pwd
/home/ece658_2021/dholcomb
```

```

vlsicad2:~$which hspice
/usr/synopsys/hspice/hspice/bin/hspice
vlsicad2:~$which cscope
/opt/Synopsys/CosmosScope/F-2011.09/ai_bin/cscope
vlsicad2:~$which dc_shell
/usr/synopsys/E-2010.12-SP5-2/bin/dc_shell
vlsicad2:~$which virtuoso
/opt/cadence/installs/ic/tools/dfII/bin/virtuoso
vlsicad2:~$which pt_shell
/usr/synopsys/primetime_install/F-2011.12/bin/pt_shell
vlsicad2:~$which icc_shell
/usr/synopsys/L-2016.03-SP5-2-new/bin/icc_shell
vlsicad2:~$which vsim
/opt/mentor/modelsim/modeltech/bin/vsim

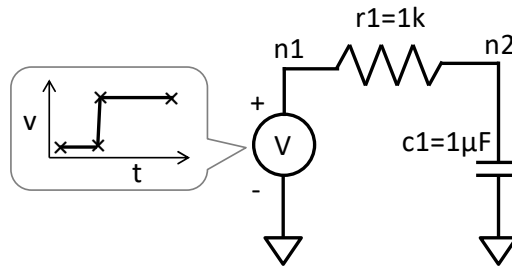
```

## Step 2: Run simulation and analyze results

### Step 2.1: Working with an unmodified test case

The circuit you will simulate, and its spice netlist, are both given below. The circuit has two nodes named n1 and n2, and a piecewise linear voltage source. Note that node 0 refers to ground. The MEASURE statement is a command to measure the delay between when a rising edge of n1 crosses 0.5V and when a rising edge of n2 crosses 0.5V; td is the name we've given the measurement.

To learn more about hspice, you can view the hspice documentation on vlsicad2 with this command, which you may need again later when modifying the spice netlist: `firefox /usr/synopsys/hspice/hspice/docs_help/index.html &`



```

* Simple RC circuit
.OP
.OPTION POST
R1 n1 n2 1K                * Resistance
C1 n2 0 1uF                * Capacitance
V1 n1 0 PWL (0s 0V 2ms 0V 2.001ms 1V 10ms 1V) * PieceWise Linear
.TRAN 1us 10ms             * Transient analysis until 10ms
* Measure delay from v(n1) to v(n2)
.MEASURE TRAN td TRIG=v(n1) VAL=0.5 RISE=1 TARG=v(n2) VAL=0.5 RISE = 1
.END

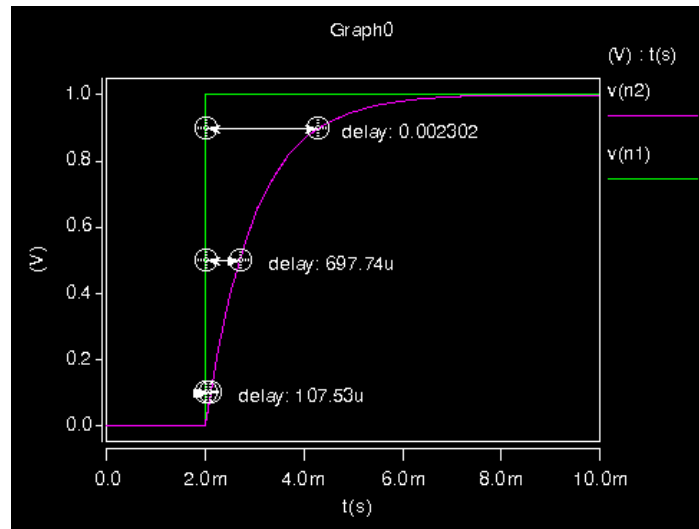
```

**Step 2.1.1 Run hspice:** The steps for simulating the spice file using hspice are as follows:

- Create a text file on vlsicad2 named lab0.spice and write in it the spice netlist text above.
- Simulate lab0.spice with hspice tool and capture its output to a log file: `hspice lab0.spice > lab0.log`
- Read lab0.log to verify that the hspice simulation run was successful. If not, debug and retry.
- If simulation succeeded, lab0.mt0 now contains the results of the measurement statement in lab0.spice
- If simulation succeeded, lab0.tr0 contains transient waveforms that can be viewed in a waveform viewer such as cscope.

**Step 2.1.2 Use cscope:** Open cscope using shell command `cscope` or `cscope&` (try both, note the difference). Then perform the following steps in the cscope gui:

- open plotfile lab0.tr0
- Plot signal n1 and signal n2 from the plotfile, then drag n2 from its plot onto the same axes as n1
- In cscope, use the measurement tool to make a time domain measurement. Specifically, measure the delay between when n1 crosses 0.5V (i.e. 50%) and when n2 crosses 0.5V (i.e. 50%). This is the same measurement as the measure statement from spice. Check that cscope agrees reasonably well with the td value reported by hspice. Add similar measurements using 0.1V to 0.1V as the crossing point, and 0.9V to 0.9V. If you've done this correctly, your plot should look like the one shown below. If you've reached this point, you can now try making some modifications in the next step.



## Step 2.2: Modify Netlist and Repeat Simulation and Analysis

Modify the spice file and repeat the analysis. The three modifications you will need to make in the spice file are as follows. You may need to refer to hspice documentation to learn how to make the desired changes:

- Modify the piecewise linear voltage source so that it reaches 1.0V at time 2.XYZms where XYZ are the final 3 digits of your student ID number. The voltage must still start rising at 2.0ms as in the original netlist.
- Modify the capacitance between node n2 and ground to be 1.XYZ uF, where XYZ are the same as above.
- Modify the measurement statement in a way that causes hspice to measure the delay between n1 and n2 using the 10%-10%, 50%-50%, and 90%-90% crossings. This will give you three different delay numbers.

As before, use cscope to plot the waveforms generated by hspice. In cscope, perform the following:

- Measure with cscope the three delay values. Make sure the measurement results on the plot are readable. The annotations can be moved by dragging. Feel free to explore cscope and modify the plot appearance to your preference.
- Change the plot name from "Graph0" to "<username>\_<student\_id>" (e.g. "dholcomb\_0123456").
- Export the plot to a file and copy it to your local machine. You can, for example, use scp for this purpose as shown:  
`scp dholcomb@vlsicad2.ecs.umass.edu:~/lab0_demo.png .`

## Step 3: Submitting report

Reports must be submitted on moodle by the deadline. The lab report should be a typed document with name and student id number. The text should briefly and concisely provide the following:

- (1) The pasted results from running the eight shell commands in step 1 (pwd, which hspice, etc). This lets us verify that everything is working in your account.
- (2) The results reported by hspice for the three delay measurements in step 2.2. Describe how you modified the MEASURE statement in the spice file to obtain the results. Pasting the MEASURE statement into the lab report will suffice for this.
- (3) The annotated plot with proper title showing the same 3 measurements from step 2.2 according to cscope. Explain briefly (no more than a couple sentences) whether you think the hspice and cscope results are consistent with each other.