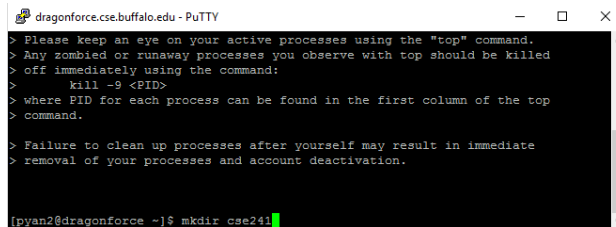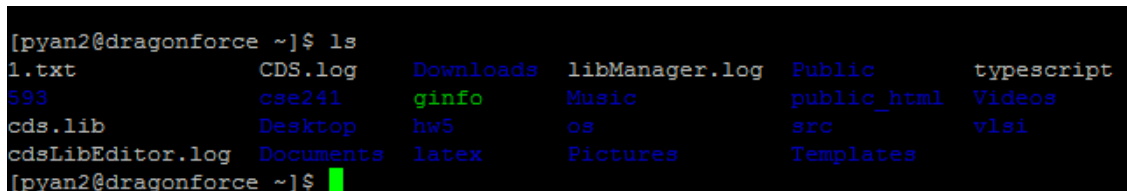# Introduction to Verilog

1. Login your timberlake terminal
2. Create a directory: to create a directory for your Verilog files, type the command 'mkdir <name>' where name is the name of the directory you want to create. An example is below.
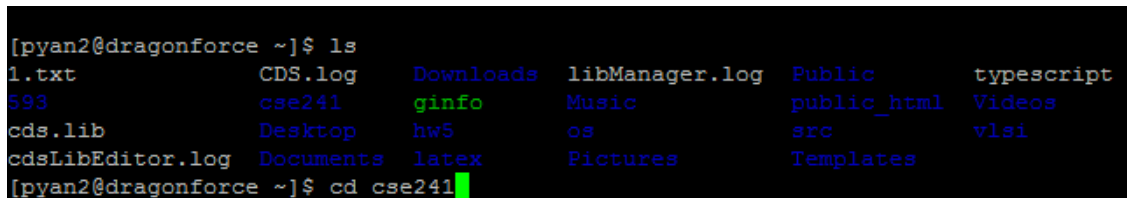

3. View directory: to view the directory you just created, use the command 'ls.' This will show you the new directory and any other directories or files that are in the current directory (in this case your current directory is your home directory). An example is below. Note that you may not have the same files in your home directory.
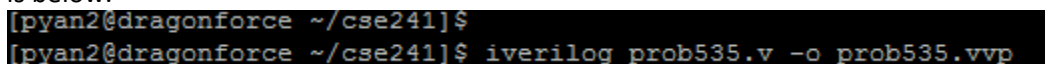

4. Switch directory: to switch from current to a new directory, use the command 'cd <name>' where name is the name of the directory you would like to switch to. An example is below. Notice how the prompt changes to reflect the current directory.

To compile Verilog, you will need a Verilog file. I am using a sample file "prob535.v"(you can find it in "J.D.'s Verilog Guide.pdf") in my cse241 directory on dragonforce. Notice that the file has a .v extension. This denotes it as a Verilog file. The following will compile the Verilog code.

1. First use the command 'iverilog <file_name>.v -o <dump_name>.vvp' where file_name is the name of the file you wish to compile and dump_name is the name of the dump file you are going to create. It is important that the file you are compiling has a .v extension and the dump file you are creating has a .vcd extension. An example is below.

2. Now use the vvp to execute your code with the command 'vvp <name>.vvp where name is the name of the dump file you created. An example is shown below.

```
[pyan2@dragonforce ~/cse241]$ vvp prob535.vvp
VCD info: dumpfile prob535.vcd opened for output.
in = 0000 | out = 0011
in = 0001 | out = 0100
in = 0010 | out = 0101
in = 0011 | out = 0110
in = 0100 | out = 0111
in = 0101 | out = 1000
in = 0110 | out = 1001
in = 0111 | out = 1010
in = 1000 | out = 1011
in = 1001 | out = 1100
[pyan2@dragonforce ~/cse241]$
```

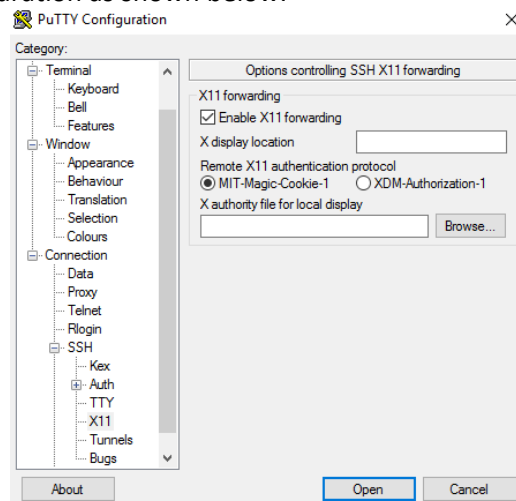3. After vvp command, a .vcd file is generated for the waveform.

```
[pyan2@dragonforce ~/cse241]$ ls
a.vcd  prob535.v  prob535.vcd  prob535.vcd_original  prob535.vvp
[pyan2@dragonforce ~/cse241]$
```

4. In order to use gtkwave to visualize the wave, you need to graphically access the server by help of X-Win32. You can download X-Win32 at https://wiki.cse.buffalo.edu/services/content/x-win32. Please follow the installing and configuration instructions. After starting the X-Win32, a blue Xwin icon will appear in the bottom right corner of the screen. Right click the icon, move to connection and select the timberlake server. You will get pop up window shown below. Enter your UBIT and CSE account password, you will be connected to the timberlake.
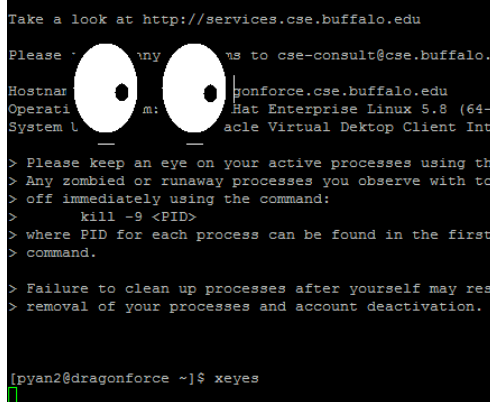Note: If you are doing this off-campus, you do not need to run the UB VPN client.

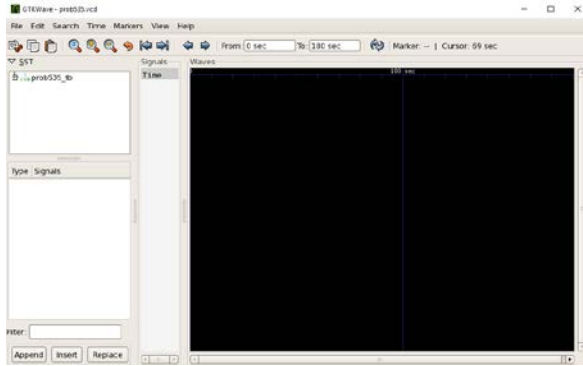5. After you log in you should see the following pop up

6. Then go to the PuTTY. Make sure you have enabled X11 forwarding in your PuTTY. If you have not done that, you can restart your PuTTY, and check the "Enable X11 forwarding" box during the configuration as shown below.

7. To test that you have full graphics capability, type "xeyes". Please select always if it ask your permission. You will see the following eyes pop up at your screen.

8. Close the eyes window. Then visualize the wave we previously produced by typing: gtkwave <name>.vcd, as shown below

9. If you drag your signal into the wave window then you will see the waveform like below