

# Instructions for logging into ECE Linux

Login to any linux system in the ece linux lab (Klauss 1448)

OR

Connect to ece linux server using vpn and fast x using the following instructions.

- For connecting remotely follow the instructions in <https://anyc.vpn.gatech.edu/> . Note that the portal is different for a normal gatech account and a 2-factor account. For the Group name field, use either gatech-2fa-Duo or gatech depending on whether you have a 2-factor account or a normal account. Download the “Cisco AnyConnect Secure Mobility Client” .Enter your credentials . Use the appropriate group(gatech or gatech-2fa-Duo). If you chose ‘gatech-2fa-Duo’ for the second password, use “push” or phone” depending on your preferred method of authentication.
- Now that you are connected remotely, we need a terminal to access Linux. FastX Client can be used. <https://software.oit.gatech.edu/request.php>? Install the Fast X Client. In the FastX Terminal, Click on the “+” on the top right corner and select ‘SSH’ to add a new connection. Fill the following details  
Name => Any user defined name  
Host => “ecelinsrvy.ece.gatech.edu  
User => gatech user name (e.g : gburdell3).  
Click on the connection you created and enter your password. To create a new window, click on the ‘+’ on the top right corner. Select “GNOME”

## Instructions for using modelsim

(The text in green font are comments, and the text in red font are the commands to be executed)

%To enable tcsh,

➤ tcsh

Download the tar-ball file from t-square and move to the home directory (or wherever you want your work to be

%Assuming you want to work in your home directory, copy the tar into your home directory

➤ cp ~/Downloads/lab\_1.tar.gz ~

% Assuming you want to work in your home directory, move into your home directory

➤ `cd ~`

% Untar the lab\_1 assignment

➤ `tar -xzf lab_1.tar.gz`

% Move into the project directory

➤ `cd lab_1`

% List the folders

➤ `ls`

Note that you have four folders => tb,src,run,scripts

- i. 'tb' refers to the folder which contains the testbench (in this case tb\_barrel\_shifter.vhd)
- ii. 'src' refers to the folder which contains the RTL (source files) (in this case barrel\_shifter.vhd)
- iii. 'run' is the folder where you should run simulations
- iv. scripts contains the simulation scripts, in this lab we do not have any

This hierarchy structure will be constant for all the labs. For this lab, you are only expected to make changes in the testbench to add the 3 test-cases.

(You can of course use any editor, but a personal preference is gvim ☺).

<http://www.openvim.com/> is a good place for reference

You are advised to open the source file and testbench and peruse the code

% Move into the run folder

➤ `cd run`

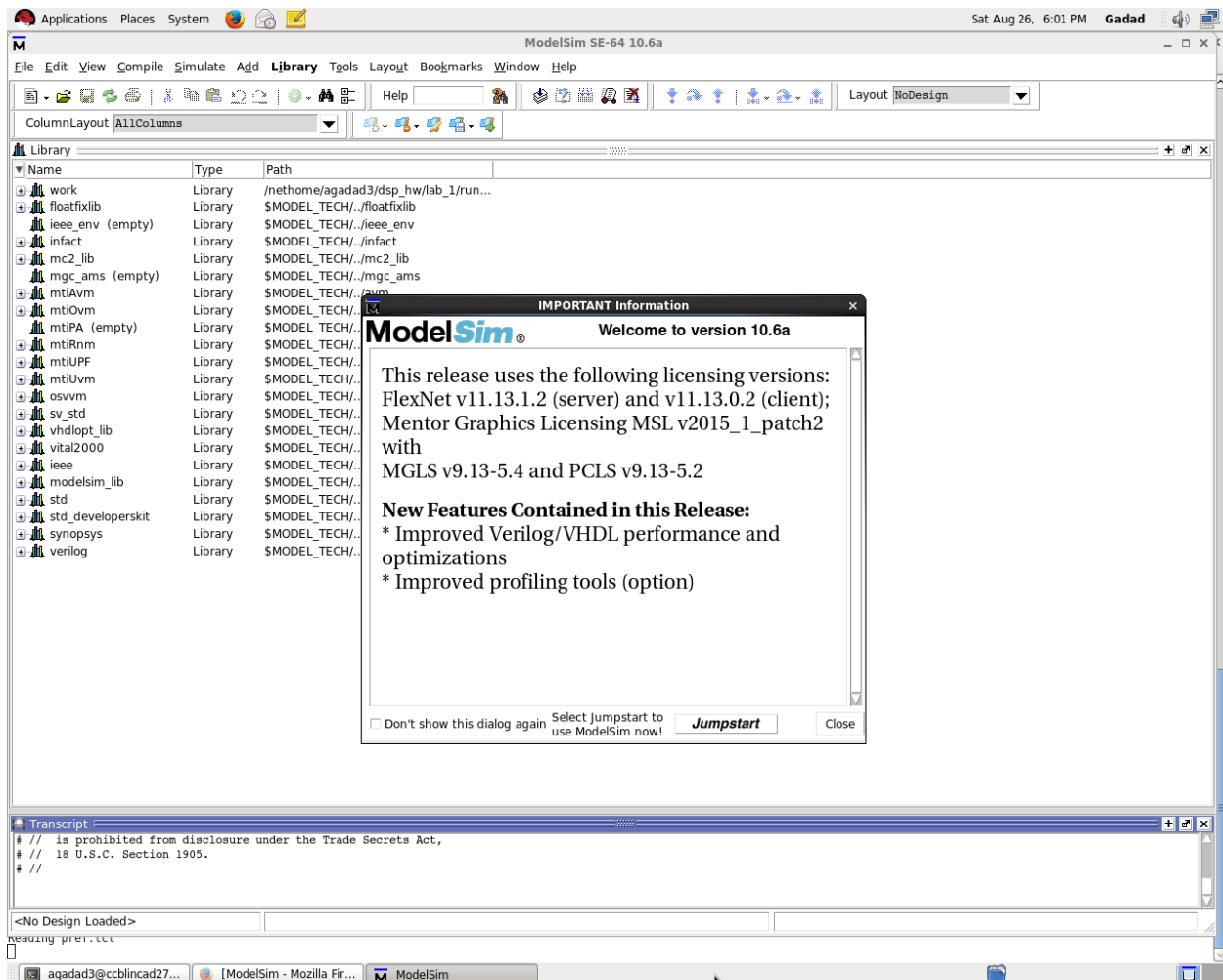
% Source the script

➤ `source /tools/mentor/modelsim/ms106a/cshrc.modelsim`

% run modelsim

➤ `vsim`

The following window should pop up



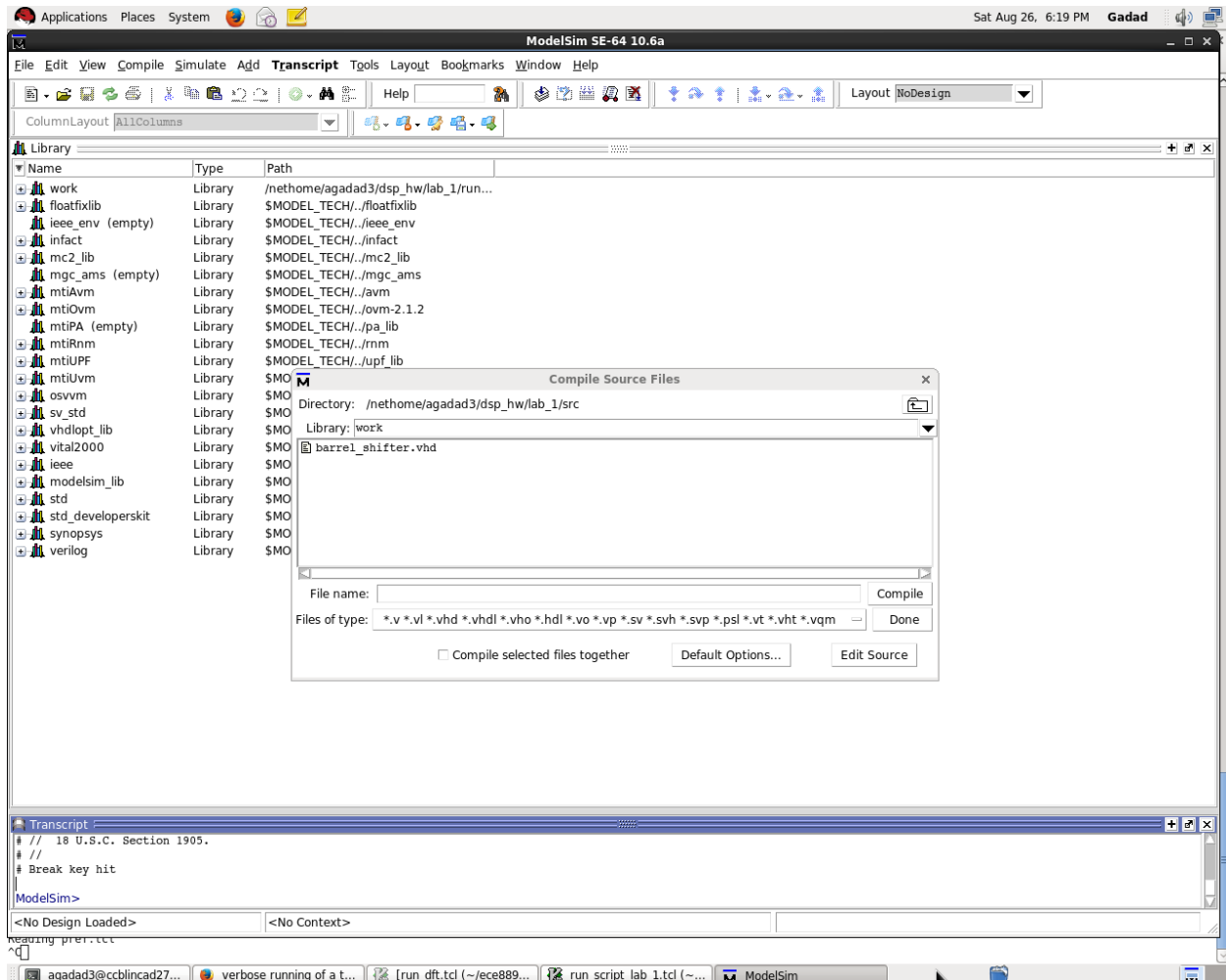
\*Click on close

%Create a work library

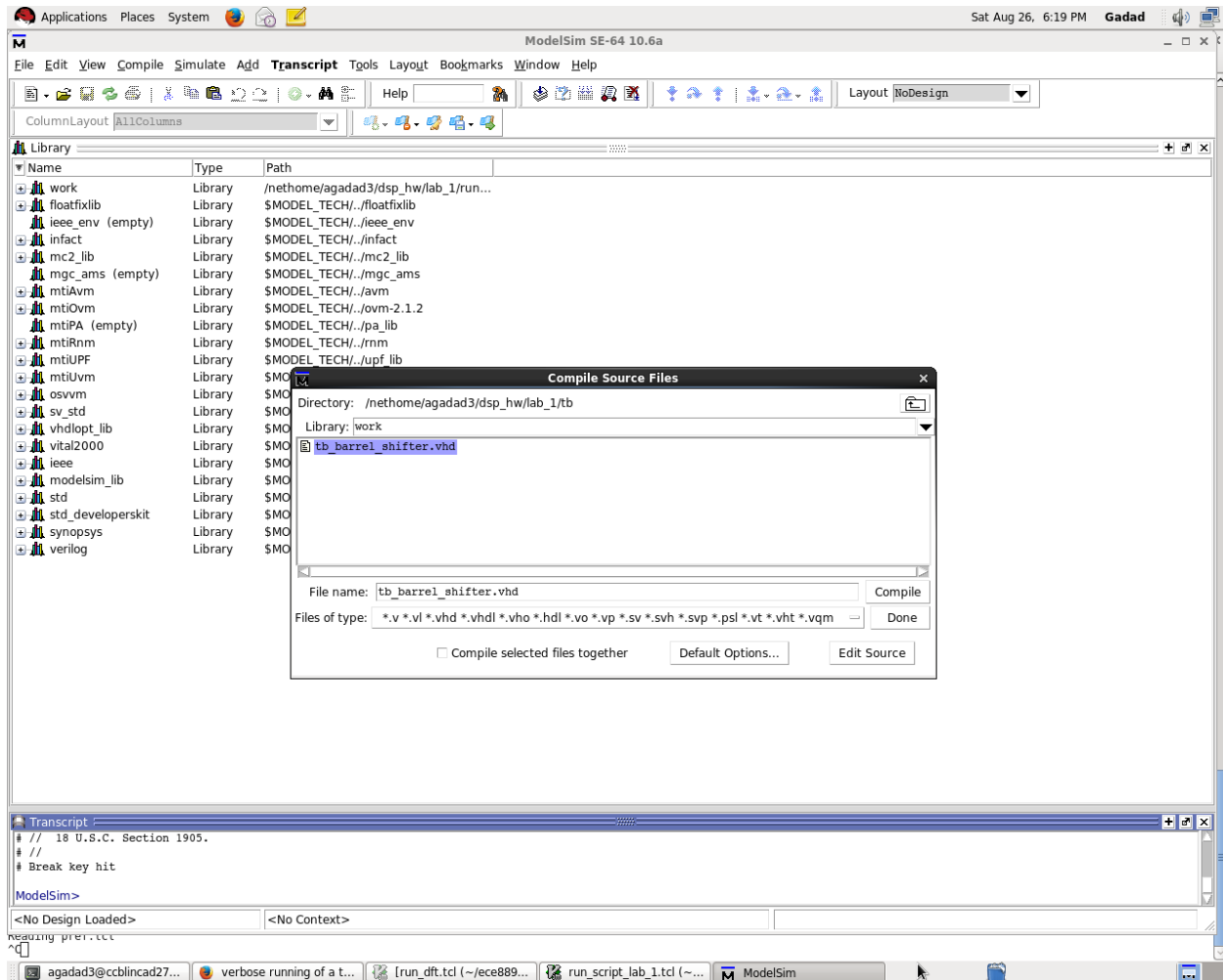
➤ vlib work

To compile the source files, go to Compile on the toolbar; click on “Compile” and Select “Compile”

Locate the barrel\_shifter in the src folder and click compile.



Repeat the same for the testbench



You now have the source and testbench compiled

To simulate ,descend into the work hierarchy =>right click on the tb\_barrel\_shifter(this is the top module and the design barrel\_shifter is instantiated in this) and click on Simulate

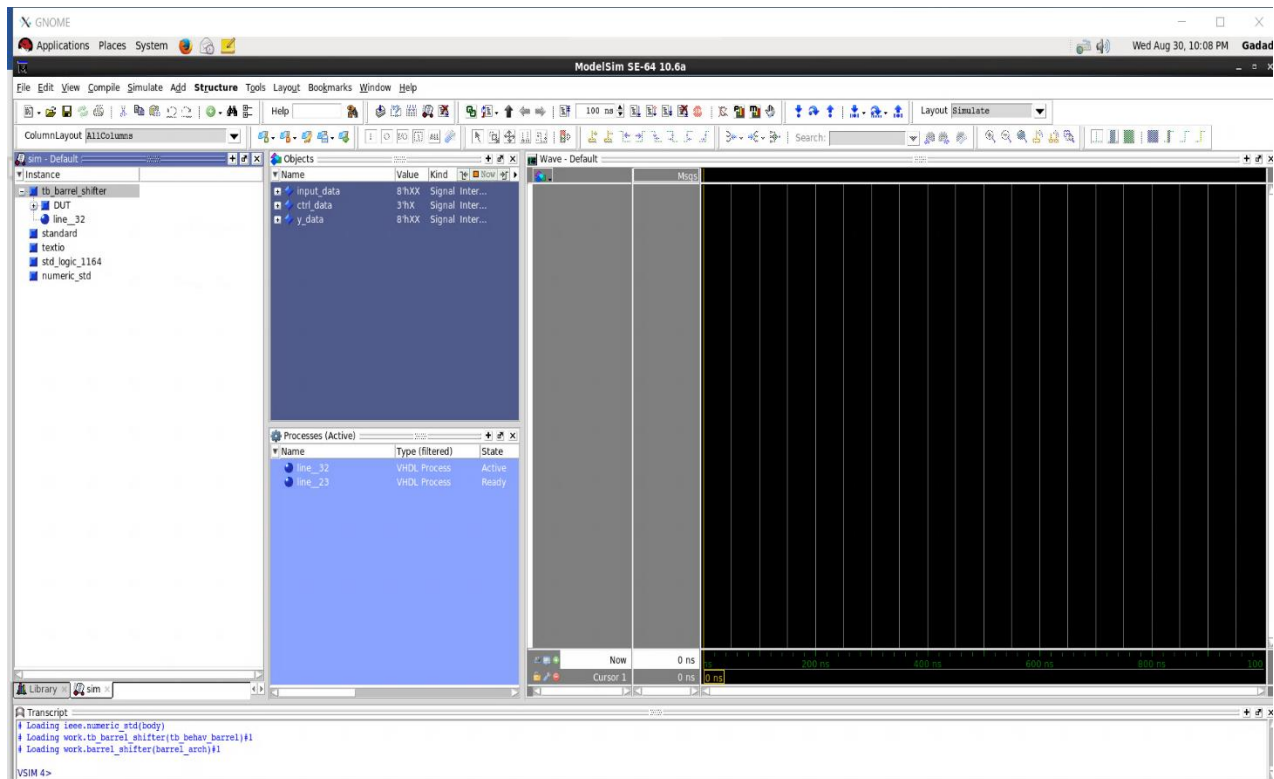
%Type the following command into modelsim to log all the wave values,

➤ log -r \*

%Type the following command into modelsim to run the simulation

➤ run -a

Go to the wave window; Note that 3 signals have appeared corresponding to tb\_barrel\_shifter ( which is the top module) . These correspond to the signals in the test bench.



Right click on 'input\_data' and select Add Wave. Add all the signals

Now click on DUT. This is the barrel\_shifter design. Again, 3 signals have appeared. These are the signals in the design. Add all the signals

Study the waveform.

In this way, you can descend into any design and use signals for debugging.

For more information on modelsim, the reference manual is given in the following link

[https://www.microsemi.com/document-portal/doc\\_view/131619-modelsim-user](https://www.microsemi.com/document-portal/doc_view/131619-modelsim-user)