Setting Up For the First Lab – V 2.2

You can (1) work from a computer in room 209 or (2) work from your own laptop by connecting remotely to a KTH server.

In case you choose option (2), open a Unix terminal and type:

ssh –Y YourKTHUsername@malavita.it.kth.se

If you have a Windows computer and want to work from home, it is recommended that you install Linux near Windows. The Ubuntu linux distribution can very easily be installed along with Windows without requiring you to partition your filesystem, as long as you have some free Gigabytes. You can then delete Ubuntu at any time through the Windows control panel. In alternative you can use Xming and Putty to connect remotely to the KTH server, but be aware that this solution is sometimes buggy.

From this time on, everything you type in the terminal happens remotely on the server. The remainder of the instructions are identical for both scenarios (1) and (2), i.e. both if you work in the lab and if you work at home. If you work in the lab, open a terminal using the graphical interface and type all commands there.

First we have to make questasim work. Give the command: *vsim*

- if questasim opens, you are fine.
- if nothing happens or modelsim opens then follow the next instructions:
  gedit ~/.bashrc
  An editor will pop out, search for all lines that contain the string
  "modelsim" and comment them out with a hash or delete them. Save the .bashrc file, then close the editor, log out, log in, try again giving the command *vsim*. This time you should get the message: "command not found".
- if you get the message "command not found" or similar, then type on the console the two following commands:
  export LM_LICENSE_FILE=1727@lic02.ug.kth.se
  module add contrib-ict-nveg
  module add questasim105
  Now try giving the command *vsim*, questasim 10.5 should open. Note: every time you open a new terminal, you should retype the two *export* commands.

Close vsim, then from the terminal give the following commands:

mkdir Gullfaxi

cd Gullfaxi

firefox

The commands create a directory Gullfaxi in your home directory, move to the new directory, then

open Firefox.

Go to the course webpage, locate the files needed for the first lab:

- *GullfaxiRELx.v* (choose your favourite release, higher numbers have less bugs)
- *GullfaxiTOP.v*
- *GullfaxiTB-SC.v*

Download them in the Gullfaxi directory, which is a subdirectory of your home directory. Close firefox.

Give the command:

vsim

to run questasim.

Select the menu entry *file->new project* and define the directory in the *Project Location*, as well as the project name.

A new window should pop up, where you can click *Add Existing File.* Click *Browse* and select the files that you want to add to your project (GullfaxiRELx.v, GullfaxiTOP, GullfaxiTB-SC.v). Close the pop up window. Select all the files in project window of questasim and use right click to edit the properties. In the *Verilog & SystemVerilog* tab, select Use *SystemVerilog* and click *OK*. Now you can compile your files, click the compile all button to do so.

Your design now should be compiled and ready to be simulated.

To edit a file, you can do that inside questasim by double clicking on the file you want to edit. Alternative, you can edit the file using a text editor externally. After you edit a file you need to recompile for the changes to take effect.

Now you can find the compiled designed called *Gullfaxi_tb_top module* under *work* in the *library* tab in the left side of questasim. Right-click on the name and select *simulate*. Questasim should automatically start the wave tab on the right, where you can add the signals you want to view. If you cannot see the *Wave* window tab, go to *View->Wave.*

On the left side of questasim you can find the *sim* tab, where you can navigate through the hierarchy of your system. You can select and add signals to the wave window from there. Right-click on any object or scope and select "*Add to -> Wave*", or simply drag&drop them on the wave window. Select the menu entry "*simulate -> run"*, or press the run or run-all button to run the simulation. Console output will appear on the transcript window on the bottom.