
ECE 436 Pre-Lab

XILINX ISE TUTORIAL

Description:

For this laboratory, you will become familiar with the Xilinx ISE design environment by following the ISE Quick Start Tutorial.

Software Access:

We will be using the Xilinx ISE WebPACK and ModelSim XE II software packages, which are free to download. This software is also installed on the ITC computers in the Stacks computer cluster.

To install the software to your own computer please see the TAs and get a copy of the CD, you can also try to download but that will take a *LOOOONG* time:

- Go to: www.xilinx.com/products/design_resources/design_tool/index.htm
- Click on *ISE WebPACK* under *Logic Design Products*.
- Click on *Order & Register*.
- Click on *Create an Account*.
- After you have registered, go to: <http://www.xilinx.com/webpack/index.htm>
- Click on *Xilinx WebInstall* to download it to your C:\ directory.
- Run *WebInstall*, following all of the instructions. Choose to have it install everything when it gives you the option.
- After you do the *WebInstall*, you still have to download the *Complete MXE Simulator*, which is at that same link. When you install it, select "MXE II Starter – Limited Version of MXE II (Free)" and "Full VHDL" during the installation process.
- After you are done installing *ModelSim*, you will need to submit a license request. From the *start* menu in Windows, find the newly installed *ModelSim* in the *Programs* directory and select *License Request Instructions* and follow the instructions.

To access the WebPACK from Stacks:

- Open: Start→S.E.A.S→Xilinx ISE 5→Project Navigator

Once the Project Navigator opens, access the Quick Start Tutorial at:

<http://toolbox.xilinx.com/docsan/xilinx6/books/docs/qst/qst.pdf>

The document is quite large, so you probably do not want to print it out.

Process:

Follow the steps of the tutorial, starting on page 11 and ending on 34. You do NOT need to do the steps in the Appendix. While you are already familiar with *ModelSim* and may eventually develop your own approaches to simulation (including writing simulation scripts), you should go through the simulation steps as defined in the tutorial. But this is also a good opportunity to experiment with different ways to use *ModelSim*.

A couple of deviations from the tutorial:

-When creating your project, save it in the C:\LocalData directory when using the ITC machines. When you are done working, make sure to save your *project* and *work* directories to your Home directory for future access and editing.

-When selecting the "Project Device Options", select the following settings:

Device Family: Spartan2

Device: xc2s100
Package: tq144
Speed Grade: -5
Design Flow: XST VHDL

-When you are asked to use the counter design from the ISE Language Templates, instead use the VHDL file posted on the toolkit website.

Deliverables:

Please submit electronically a lab report with the following items:

1. An **annotated** printout of the "Top-Level Schematic".
2. An **annotated** printout of the "Timing Simulation Waveform" from the ModelSim tools that tests your top-level design.

(Note: By annotate, I mean explain what is on the printout.)