

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 III software packages, which are free to download. Somewhat older versions of this software are also installed on PCs in the ECE labs in Thornton E-225 and the same version in C-123.

To install the software to your own computer please ask the TAs for the CDROMs (preferred) with the tools, alternatively you can download but be prepared to wait:

- Go to: http://www.xilinx.com/products/design_resources/design_tool/index.htm
- Click on *ISE WebPACK™*
- Click on *Register*.
- Click on *Create an Account*.
- After you create an account, you will receive an email confirmation from Xilinx with a link to the WebPACK tools. You will have to go to: <http://www.xilinx.com/webpack/>
- This time log in with your user id and password.
- 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 *ModelSim Xilinx Edition-III*, which is at that same link. During the installation process, select "MXE III Starter – Limited Version of MXE III (Free)" and "Full VHDL."
- 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/xilinx8/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 7 and ending on page 28 with the "Download Design to ... Board" section (which you cannot do without a board). 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 the tool.

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 (these correspond to the device you will use for the project):

Device Family: Spartan2
Device: xc2s100
Package: tq144
Speed Grade: -5
Design Flow: XST VHDL

Because of this change in the part, the pin allocation also needs to be changed compared to the tutorial. Please use this pin allocation instead:

clock : P18
count_out_0 : P4
count_out_1 : P5
count_out_2 : P6
count_out-3 : P7
direction : P10

Deliverables:

Please submit electronically a lab report with the following items:

1. An **annotated** printout of the "Top-Level Schematic".
(To prepare the Schematic, refer to the directions given in the tutorial for the older version of the tool. It is available at:
<http://toolbox.xilinx.com/docsan/xilinx6/books/docs/qst/qst.pdf>)

2. **Annotated** printout of simulation of the design in the tutorial.

(Note: By annotate, I mean explain what is on the printout, you can use the ink capabilities of a Tablet PC, or you can explain with text separate from the figure.)