LHC Research ModelSim Tutorial

Lev Kurilenko (levkur@uw.edu)
3/28/18

Table of Contents

Introduction

This tutorial covers the more advanced aspects of ModelSim that people in Dr. Hauck's ACME group at the University of Washington should be familiar with. The tutorial assumes a basic understanding of ModelSim and how to go about simulating a design using a .do file.

ModelSim Installation

Due to the nature of some of the projects, an unlicensed and slow version of ModelSim can sometimes be inadequate. Mentor Graphics deliberately slows down the program to motivate customers to purchase a license and increase the speed of their simulations. University of Washington provides a faster version of ModelSim which uses a floating license.

The download link can be found here:

https://downloads.engr.washington.edu/downloads/distribution/Mentor_Graphics/SE%2010.5c/

Download the file named:

modelsim-win64-10.5c-se.exe

Once the download completes, run the file and go through the installation process.

Setting up the License

Setting up the license for the faster ModelSim only needs to be done once. Since the license provided by UW is a floating license, a system environment variable needs to be created containing a value specifying the port number and hostname of the license server.

1. Using the Windows search feature, type in 'environment' in the search input as shown in Figure 1.

2. Select the 'Edit the system environment variables' option.

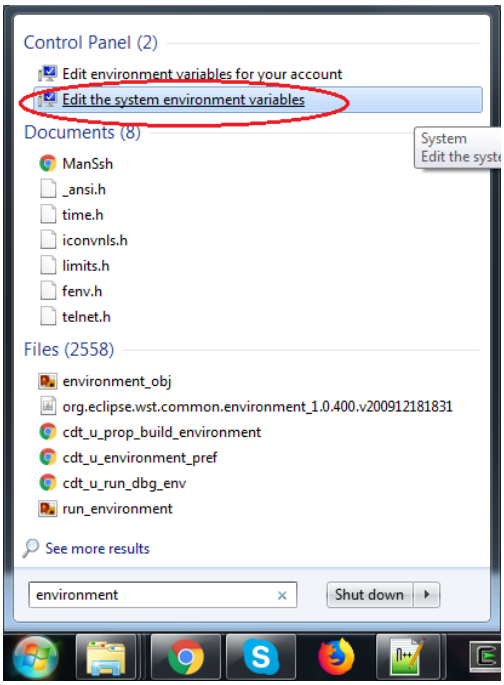

***Figure 1*:** Environment variable search

3. This will take you to a 'System Properties' window. Select the 'Environment Variables' button as shown in Figure 2.

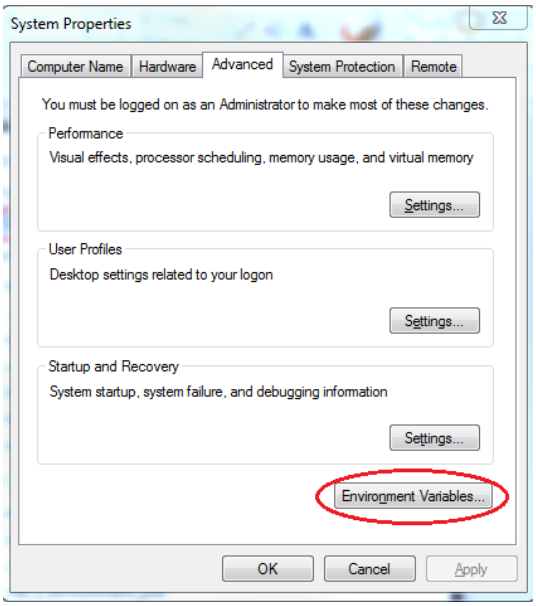

***Figure 2*:** Environment Variables selection in System Properties

4. Next, select the 'New…' button in the 'User variables for PC' as shown in Figure 3.

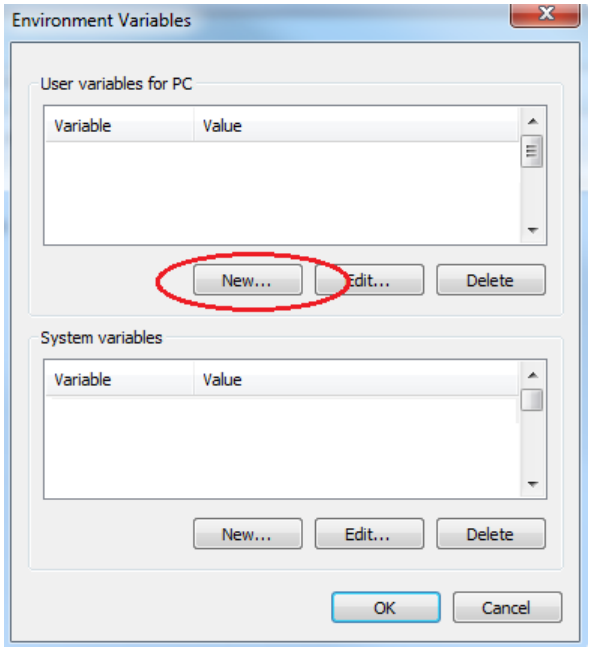

***Figure 3*:** Select 'New…' button in 'User variables for PC'

5. Enter ' LM_LICENSE_FILE' for the variable name and ' 27007@mentorls.s.uw.edu' for the variable value as shown in Figure 4.

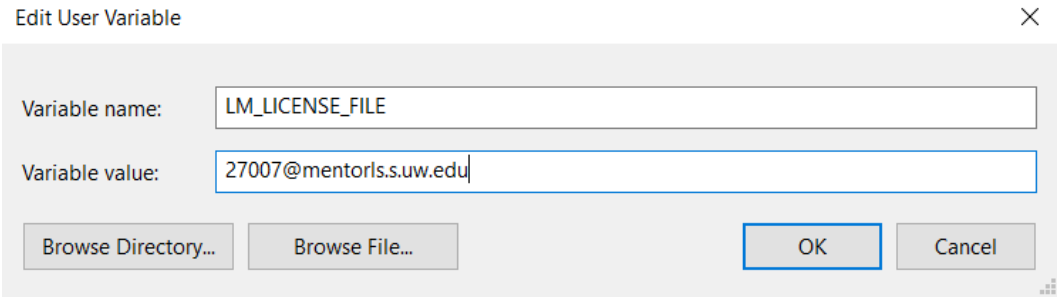

***Figure 4*:** Enter the variable name and variable value

6. Hit OK, for the 'New User Variable' and the 'Environment Variables' windows. Hit Apply for the 'System Properties' window.

7. Finally, reboot the system and open ModelSim. ModelSim should now be licensed and ready to go!

Compiling Vivado Simulation Libraries

Oftentimes, IP cores provided by the vendor are used in a digital design. The IP cores contain HDL code, and sometimes hardened IP blocks, that target specific functionality. When an IP core is included in a design, it is important to still have the ability to simulate the design. To accommodate for this, vendors usually provide a functional simulation model of the IP core. This allows the user to understand the functionality of the IP core and incorporate it into the simulation flow.

The functional simulation model uses libraries created by the vendor specifically for simulation. Examples of Xilinx libraries include 'unisim', 'secureip', and 'unimacro'. These libraries need to be included as a ModelSim library to simulate any design containing an IP core that references these libraries. Described below are the steps for compiling Vivado simulation libraries and including them into your ModelSim simulation.

1. Open the 2017.4 Vivado IDE.

2. In the top, select the 'Tools' tab and select 'Compile Simulation Libraries…' as shown in Figure 5.

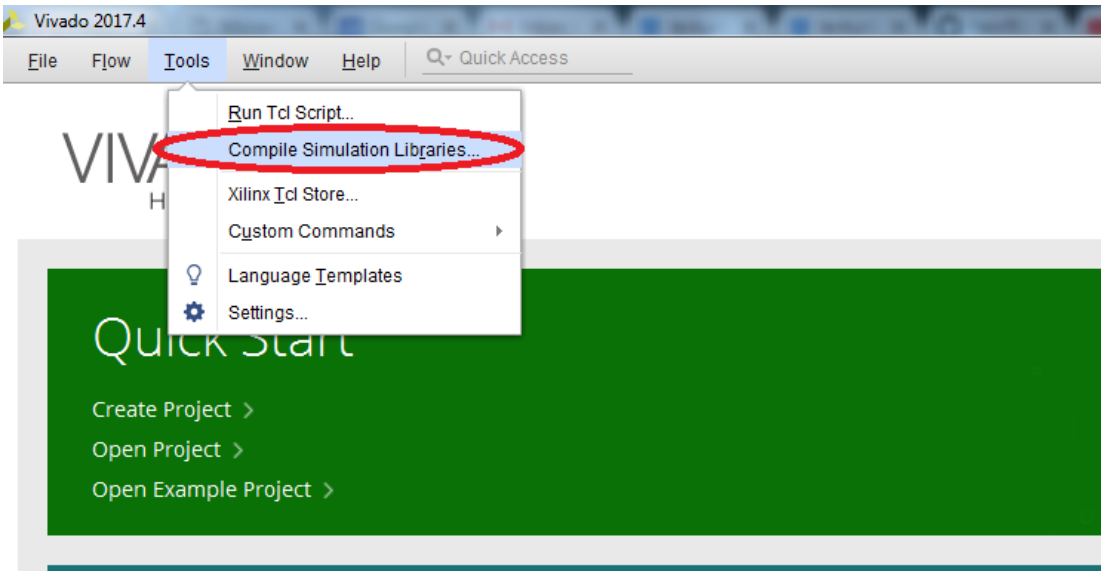

***Figure 5*:** Select compile simulation libraries

3. A 'Compile Simulation Libraries' window will open as shown in Figure 6. Make sure all the options shown are selected and proper paths specified, then hit Compile.

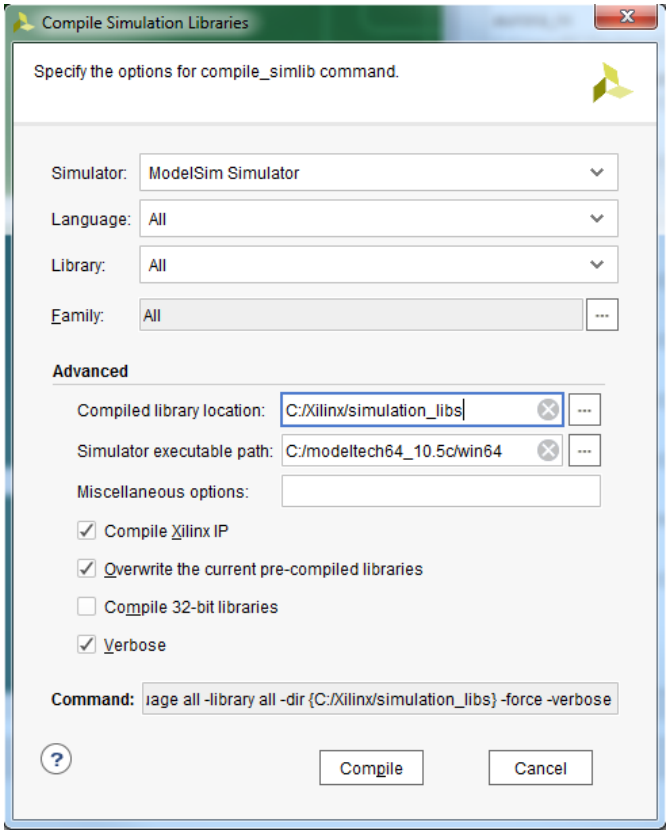

***Figure 6*:** Compile simulation libraries window

4. Compiling the libraries may take several minutes.

5. Once the libraries are compiled, any library used by an IP core must be included as a ModelSim library.

6. Right-click in the ModelSim Library pane, hover the mouse over ‘New’, and select ‘Library…’ as shown in Figure 7.

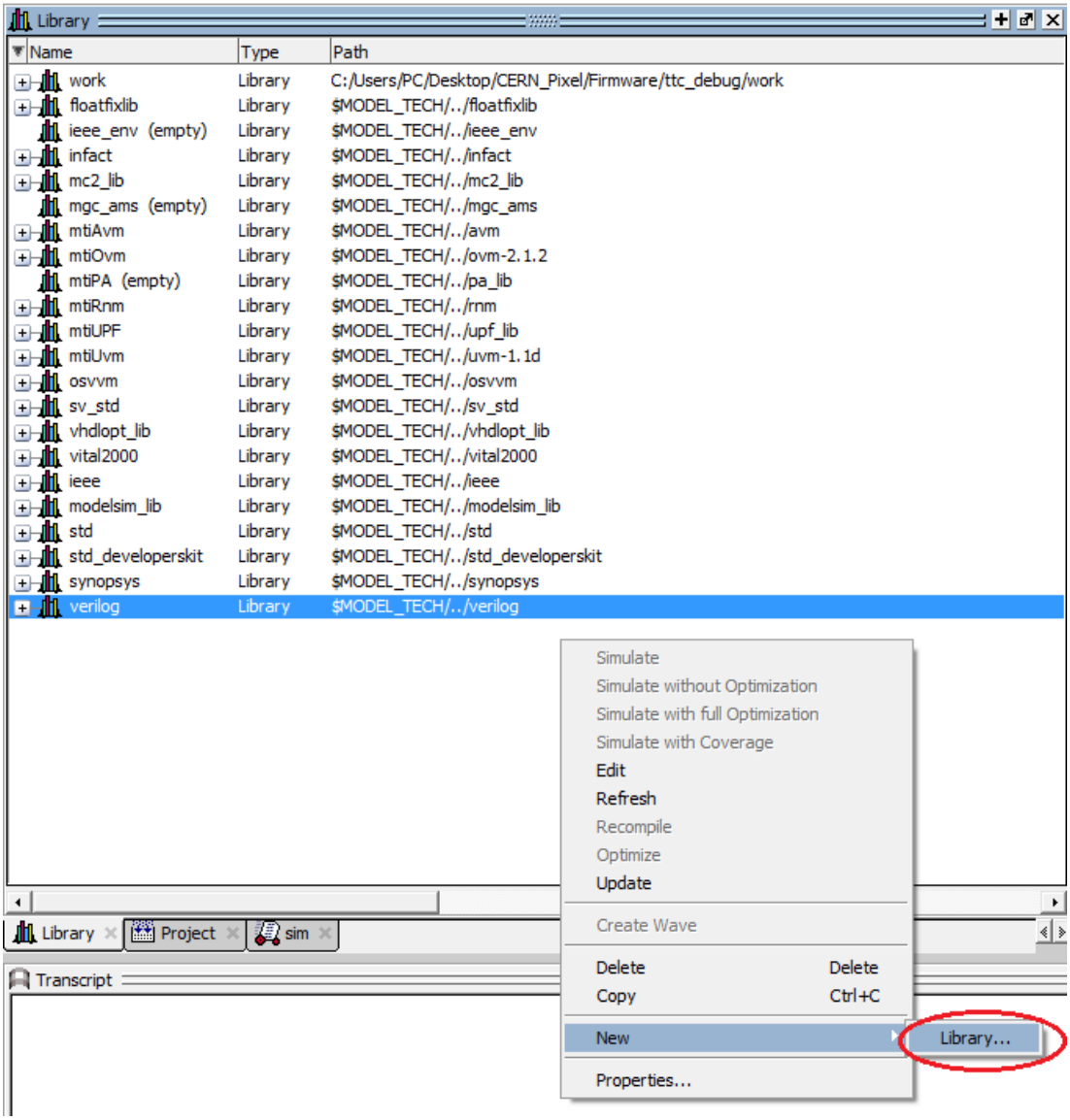

Figure 7: Add Library to ModelSim

7. In the ‘Create a New Library’ window select the ‘a map to an existing library’ option. Enter the library name of the library you wish to create. Press ‘Browse’ in the ‘Library Maps to:’ section as shown in Figure 8.

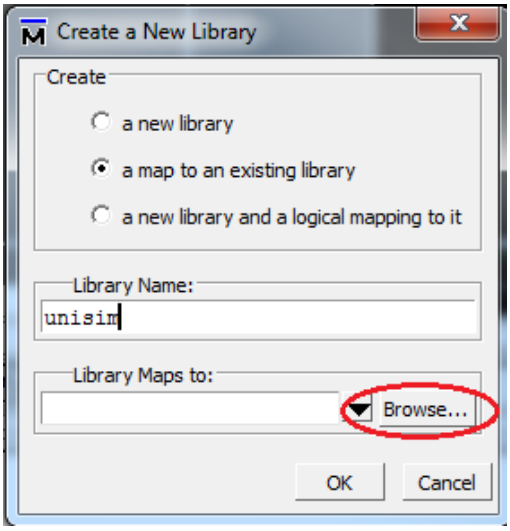

Figure 8: Create a New Library

8. Navigate to the simulation_libs folder where the Xilinx simulation libraries were compiled. In there, find the library corresponding to the library name specified in step 7. See figure 9.

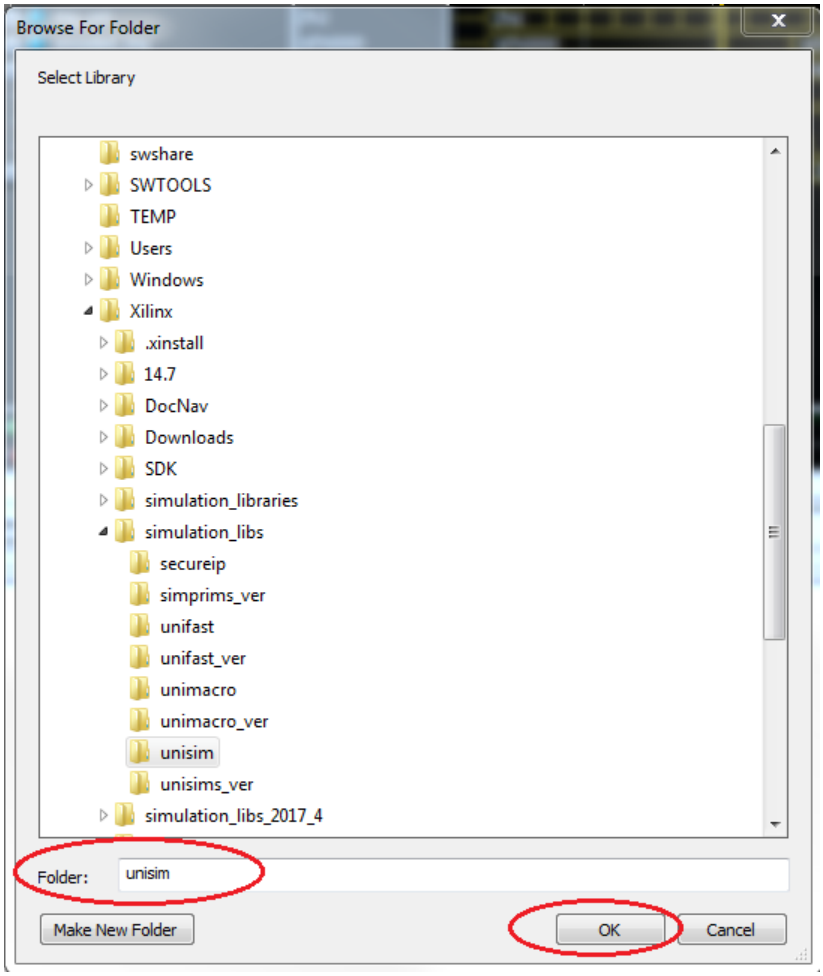

Figure 9: Select Library folder corresponding to Library Name

9. The library is now included in the library pane as shown in Figure 10.

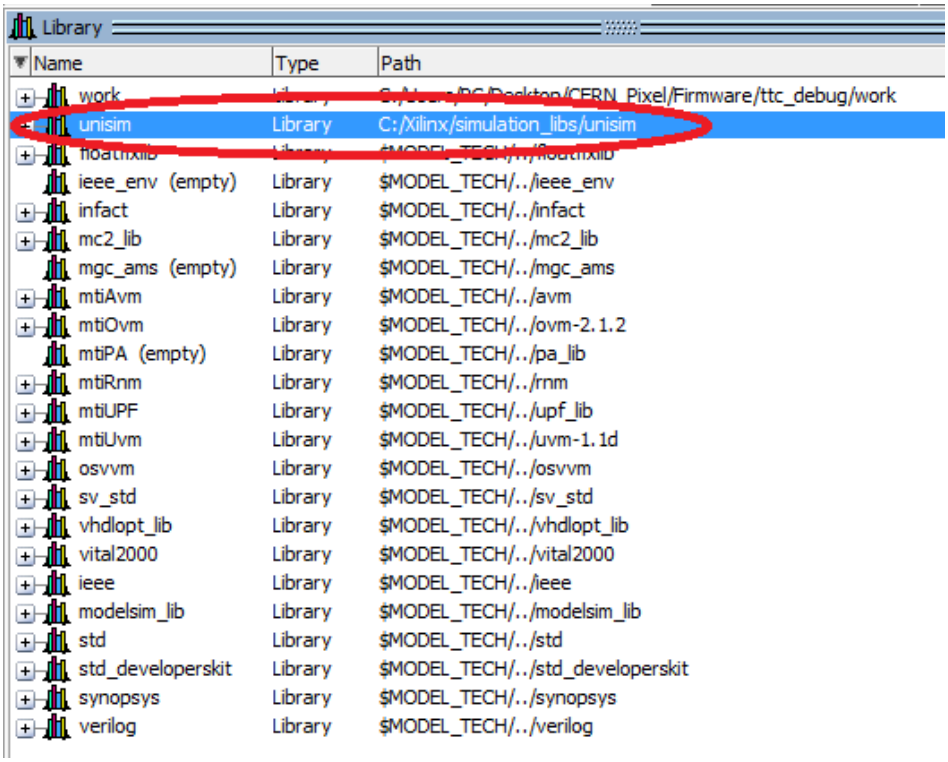

***Figure 10*:** Xilinx simulation library shows up in ModelSim Library pane

After these steps are completed, you are ready to simulate designs containing functional simulations of Xilinx IP blocks.

Simulation

Simulations can be created in the ModelSim Transcript window using tcl commands. The simulation creation process can be automated by creating a .do file containing tcl commands and executing the .do file by running:

do filename.do

in the Transcript window. A detailed tutorial can be found in Dr. Hauck's EE271 tutorial on pages 6-14:

https://class.ee.washington.edu/271/hauck2/labs/Quartus%20Tutorial.pdf