Notes for Installing Software Tools for SE2DA4 Labs

Robert Li

All the software tools for SE2DA4 labs have been installed on the department lab computers and are ready to use. On our department lab computers, the installed Quartus Prime and Questa-Intel FPGA are the Standard Edition of Version 22.1.1. The other simulator, ModelSim-Intel FPGA version 20.1, is also available on the lab computers. These notes are for students who want to install those software tools on their own computers.

These notes are for Window 10 operating system, which is used on our lab computers.

1. Install Quartus Prime Version 22.1.1

There are three editions of Quartus Prime. They are: Pro Edition, Standard Edition and Lite Edition. The Lite Edition is free and no license is required, but it has less features and supports limited devices, and its performance is not as good as the other two editions. Our department has license for the Standard Edition, and this is the edition that is installed on our lab computers. You can either download and install Quartus Prime **Standard** Edition or the Quartus Prime **Lite** Edition on your computer. Both the Standard Edition the Lite Edition support the DE1-SoC boards that will be used for our lab projects, and both of them work fine with our lab projects. If download and install the Standard Edition, you need to set up license to use it.

1.1 Download Quartus Prime

The Quartus Prime can be downloaded from http://www.intel.com/. Under its PRODUCTS category, find section "FPGAs & Programmable Devices". Then click "Intel Quartus Prime Design Software". From that page you can download software of different Editions and different versions for Windows or for Linux.

You have three ways to download the Quartus Prime and its necessary supporting files for our lab projects:

1) Complete Download. This will download the Quartus Prime, Questa-Intel simulator, and all the device support files
2) Multiple Download : For this choice, you need to download: 1. Quartus (including Questa) , 2. At least the Device Package 3, which is for Cyclone devices. The DE1-SoC boards which we will use for our lab projects use Cyclone V.
3) For Individual File Download: You need to download: 1. Quartus, 2. Questa, 3. At least the Device Package for Cyclone V.

1.2 Install Quartus, Questa and Device Package for Cyclone V

The following notes is for Complete Download.

The downloaded files are in a .tar file. You need to un-tar or unpack it before running the setup.bat file to install the software.

Un-tar can be done by "7zip" utility for Windows or by using the command line tool "tar -xvf" followed by the .tar file name.

Once the file is un-tared, find the setup.bat file, and run it to start to install.

At the stage to select the components to be installed:

1) You need to check the device support at least for Cyclone V, because our labs only use DE1-SoC boards which use Cyclone V.
2) By Default, the "Questa-Intel FPGA Starter Edition" is automatically checked and included. Questa-Intel FPGA Starter Edition has limited functions and performance. We recommend you to check the checkbox for "Questa-Intel FPGA Edition" to install this standard edition as well, because this edition has more potentials.
   Both editions of Questa-Intel FPGA need license to be setup, but the "Questa-Intel FPGA Starter Edition" it is a zero cost license.

1.3 Setup License

By default, after Quartus Prime is successfully installed it will be automatically launched. The first time when Quartus Prime is launched, users will be reminded to set up license for it:
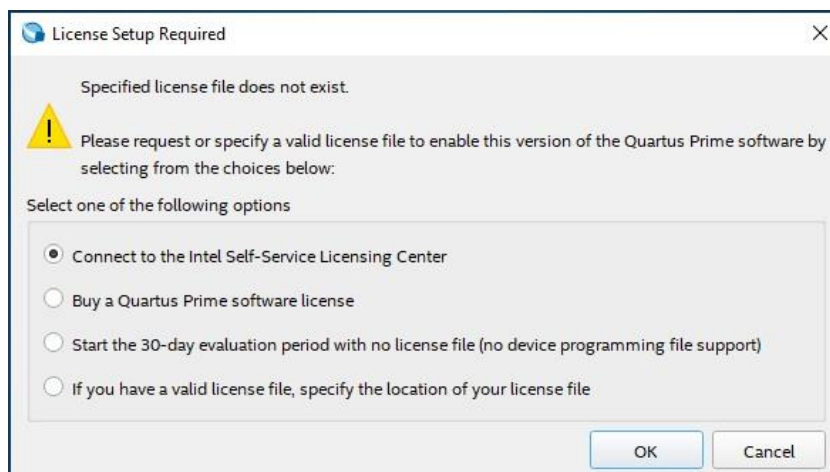
To use our department license, please select choice to specify the location of our department's license file. Then in the License Setup window, input "**27000@alteralm.cas.mcmaster.ca**" as the license file, as shown in the following Figure.
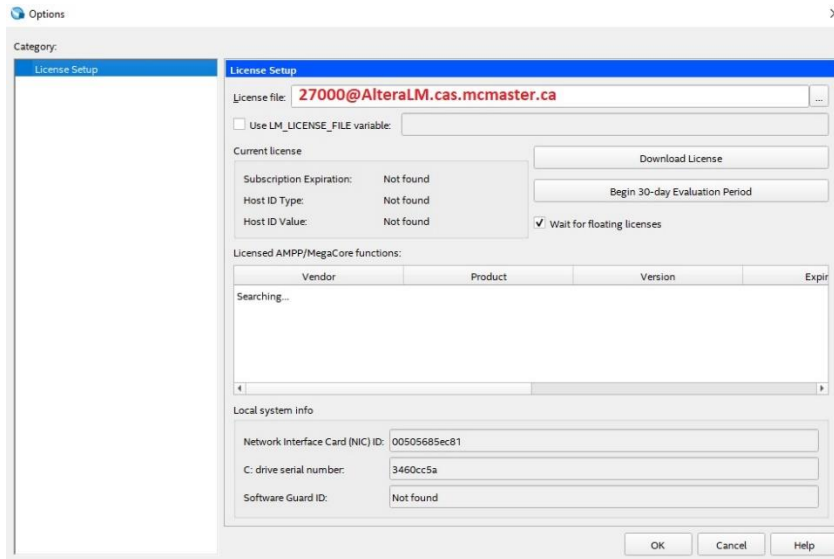
In case you need to set up license for Quartus Prime later, you can access the License Setup window by going to **Tools | License Setup …** in Quartus Prime.

Our department license covers all the components of Quartus Prime that are required for our lab projects, including NIOS II EDS, Questa-Intel, and ModelSim-Intel.

For both the standard edition and the starter edition of Questa-Intel, and for ModelSim-Intel standard edition, users need to create a new system environment variable "LM_LICENSE_FILE" and set its value to our department's license file, which is at: "27000@AlteraLM.cas.mcmaster.ca".

The steps to set up the environment variable are as follows:

1) On Windows task bar, right click the Windows icon and select "System".
2) In the opened window, under the "Related Setting" section, click "Advanced system settings".
3) In the opened "System Properties" window, click the tab "Advanced", then click the button "Environment Variables".
4) Under the section "System Variables", click button "New…" to create a new environment variable.
5) In the "New System Variable" window, input "LM_LICENSE_FILE" as the variable name, and "27000@AlteraLM.cas.mcmaster.ca" as the variable value.

If you use Quartus Prime from off-campus, you need to use VPN to be able to connect to our department license server.

2   Install ModelSim-Intel 20.1

Since Version 21.1, the simulator that comes with the Quartus Prime is Questa-Intel FPGA. To use Questa-Intel to simulate a design, users need to write a testbench, which is basically a (Verilog) module generating and providing stimuli for the design unit that is under test, and then the output of the design unit is verified.

Writing testbenches requires uses to have good grasp of Verilog language. However, for lab 1 project, students may have not had enough time to pick the necessary Verilog knowledge yet. So on the department lab computers, another simulator, ModelSim-Intel, is also installed.

With ModelSim-Intel, users do not have to write testbenches. Instead, users can create waveform input files by using the provided GUI tool to simulate the design unit. ModelSim is easy to learn and easy to use. It is a good entry level simulator and is good for teaching purpose. Although Questa-Intel offers higher performance and advanced debugging capabilities, for our lab projects, ModelSim is well competent and time saving. Actually, students can use ModelSim for all of our lab projects.

The latest version of available ModelSim-Intel is 20.1. To download this version of ModelSim-Intel, go to the Quartus download page, chose Standard Edition and select version of 20.1. Under the Individual Files downloading option, download the ModelSim setup file.

When install Modelsim, you can select either the Standard Edition, or the Starter Edition. Both of them are competent for our lab projects. The starter edition of MoldelSim-Intel is a free version. Our department has license for the Standard Edition, which is supposed to offer more features and has better performance.

Students are recommended to read two reference materials for starting to use ModelSim-Intel for our lab projects. These two reference materials are available in the reference folder in lab 1. They are:

1) [Notes for Using ModelSim.pdf](#)
   Some notes for using ModelSim-Intel 20.1 with Quartus Prime 22.1.
2) [Quartus_II_Simuatioin.pdf](#)
   Tutorial Materials from Intel for Version 17.1. The latest tutorial materials available from Intel for using ModelSim.

Students who prefer to use Questa-Intel or have interest to learn how to use Questa-Intel to simulate can use the following documents as the references to start:

1) [Questa Quick Start.pdf](#)
   A very brief documentation form Intel website.
2) [Writing a Testbench.pdf](#)
   How to start to write a testbench in Verilog.

3 Install Eclipse IDE for NIOS II EDS

In lab 5, students will be required to use NIOS II EDS to develop a controller circuit to access the SDRAM chip on the DE1-SoC board. In the project, the Eclipse IDE is needed to create NIOS C/C++ projects. Starting from version 19.1, users are required to manually install the Eclipse IDE.

For Windows system, please follow the following steps to install Eclipse IDE for NIOS II:

1. Download the Eclipse C/C++ IDE for Mars.2:
   1) For Windows:
      https://www.eclipse.org/downloads/download.php?file=/technology/epp/downloads/release/mars/2/eclipse-cpp-mars-2-win32-x86_64.zip

Or download it from :
https://archive.eclipse.org/technology/epp/downloads/release/mars/2/eclipse-cpp-mars-2-win32-x86_64.zip

2) For Linux:
https://www.eclipse.org/downloads/download.php?file=/technology/epp/downloads/release/mars/2/eclipse-cpp-mars-2-linux-gtk-x86_64.tar.gz

2. Extract the downloaded file into directory: <*Quartus Prime installation directory*>/nios2eds/bin. This will create a folder: <*Quartus Prime installation directory*>/nios2eds/bin/eclipse. By default the <*Quartus Prime installation directory*> would be: C:/intelFPGA/22.1std

3. Rename the folder <Quartus Prime installation directory>/nios2eds/bin/eclipse to <*Quartus Prime installation directory*>/nios2eds/bin/eclipse_nios2.

4. Locate the file <*Quartus Prime installation directory* >/nios2eds/bin/eclipse_nios2_plugins.zip, and extract this file to folder: <*Quartus Prime installation directory*>/nios2eds/bin. The extraction overrides files in < *Quartus Prime installation directory*>/nios2eds/bin/eclipse_nios2. (For Linux, the file is <*Quartus Prime installation directory*> /nios2eds/bin >/nios2eds/bin/eclipse_nios2_plugins.tar.gz)

After the above steps, clicking **Tools | Nios II Software Build Tools for Eclipse** will get Eclipse IDE launched.

4   Install WSL and Necessary Tools for Eclipse IDE

Starting from version 19.1, Window Subsystem for Linux(WSL) and two utility tools, **dos2unix** and **make**, are needed for NIOS II EDS and Eclipse IDE to work. You can follow the following steps to install WSL and the two utilities:

1. Open PowerShell or Command Prompt in Administrator mode (by right clicking it and selecting "Run as administrator").
2. Run: wsl --install
   Most of time, the command "wsl --install" will work and the WSL will be installed. At the end of installation, the system will prompt you to create an account for a new user for the WSL and set password for the user. But sometimes because of your system, the above command may not work:
   1) In case the command "wsl --install" does not work but give you WSL help text, please use command:
      "wsl --install -d Ubuntu" instead of "wsl --install".
   2) If at the end of installation, the system gives you error: "WslRegisterDistribution failed with error: 0x80370102", go back to step 1 and run command:
      wsl --set-default-version 1
      Then run command:
      wsl --install -d Ubuntu
3. Once the WSL is installed successfully, run: sudo apt update
4. Run: sudo apt upgrade
5. Then Run "sudo apt install dos2unix" to install dos2unix.

6. Run "sudo apt install make" to install make for Eclipse IDE.