5.2.1 Simulation Using Quartus and ModelSim

This section provides a step-by-step guide for installing Quartus and ModelSim, and simulating the full adder designed in the previous section.
Note: Required setup files have to be downloaded before starting the section.

Setting-up the Environment

Download setup files corresponding to

- Intel Quartus Prime Lite Edition Design Software Version 20.1.1

- ModelSim Intel FPGA Edition Version 20.1.1

- Intel Cyclone IV Device Support file

into a single directory and run the Quartus installer. Proceed until the installer prompts for selecting components to install. Select all the components in the list except ModelSim paid version (see Figure 5.2).
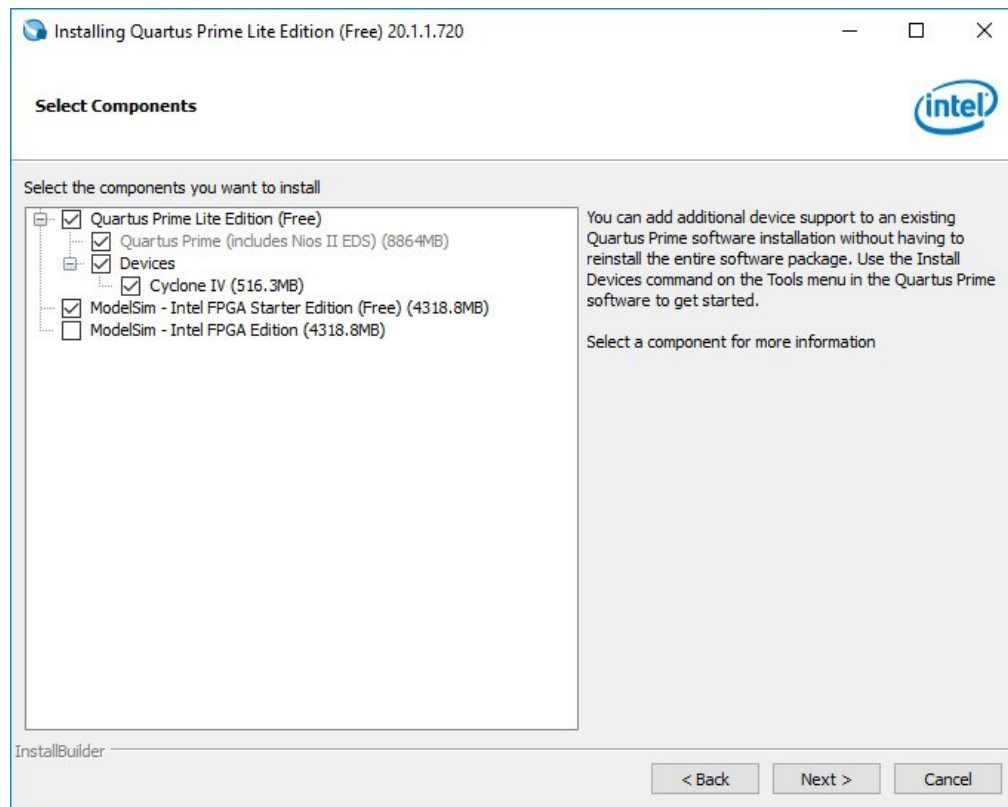

Figure 5.2: Quartus installer: selecting components to install

After setting-up the required software tools, open *Quartus Prime Lite 20.1.1* Design Software. It will bring you to the home screen of Quartus illustrated in Figure 5.3. To verify that the ModelSim has been integrated correctly, complete the steps mentioned below;

1. Click **Tools > Options** from the menu bar. This will open up the **Options** dialog box.

2. Select **EDA Tool Options** listed under **General** category in the left-hand side pane.

3. There should be file paths mentioned in-front of **ModelSim** and **ModelSim-Altera** (see Figure 5.4).

4. From your file manager/explorer, navigate to the mentioned location and check whether **modelsim.exe** file exists. If so, you are good to go. Press **Cancel** button to exit without modifying anything.

5. If not, find the location of the **modelsim.exe** file and replace the current paths by the correct path. Press **OK** to save and exit.
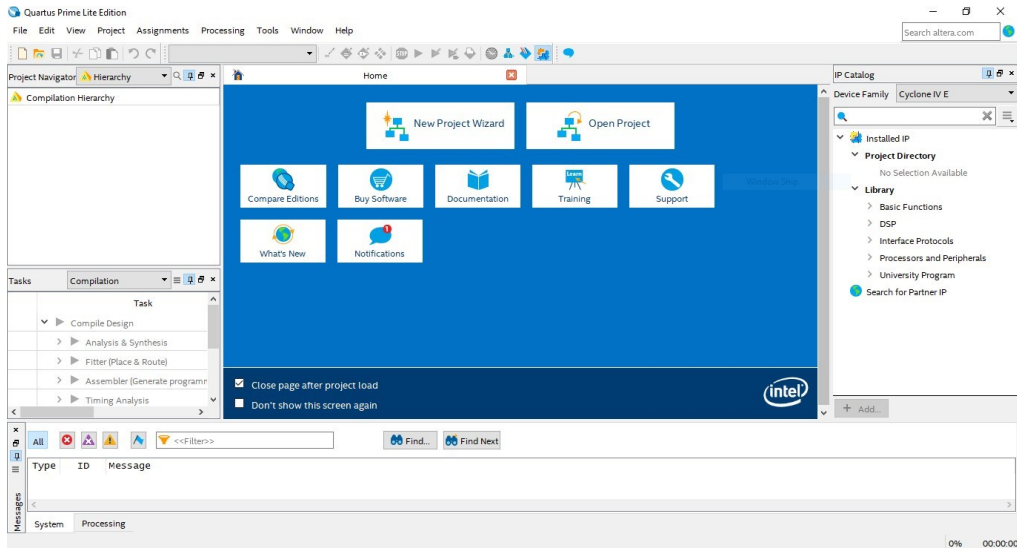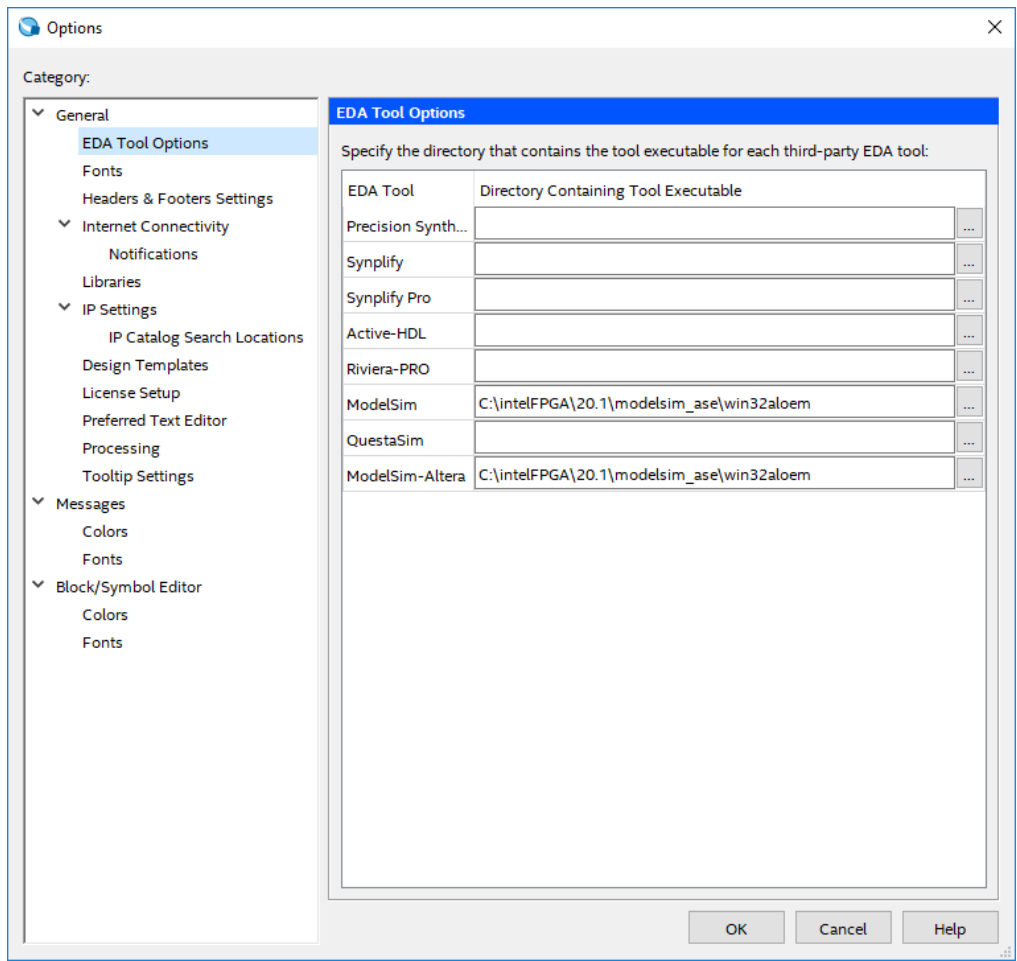
Figure 5.3: Quartus home screen

Figure 5.4: Setting ModelSim path

Creating a New Project

To start the **New Project Wizard**, either click on the "New Project Wizard" button on the home screen or goto **File > New Project Wizard**. This will open up the window illustrated in Figure 5.5.
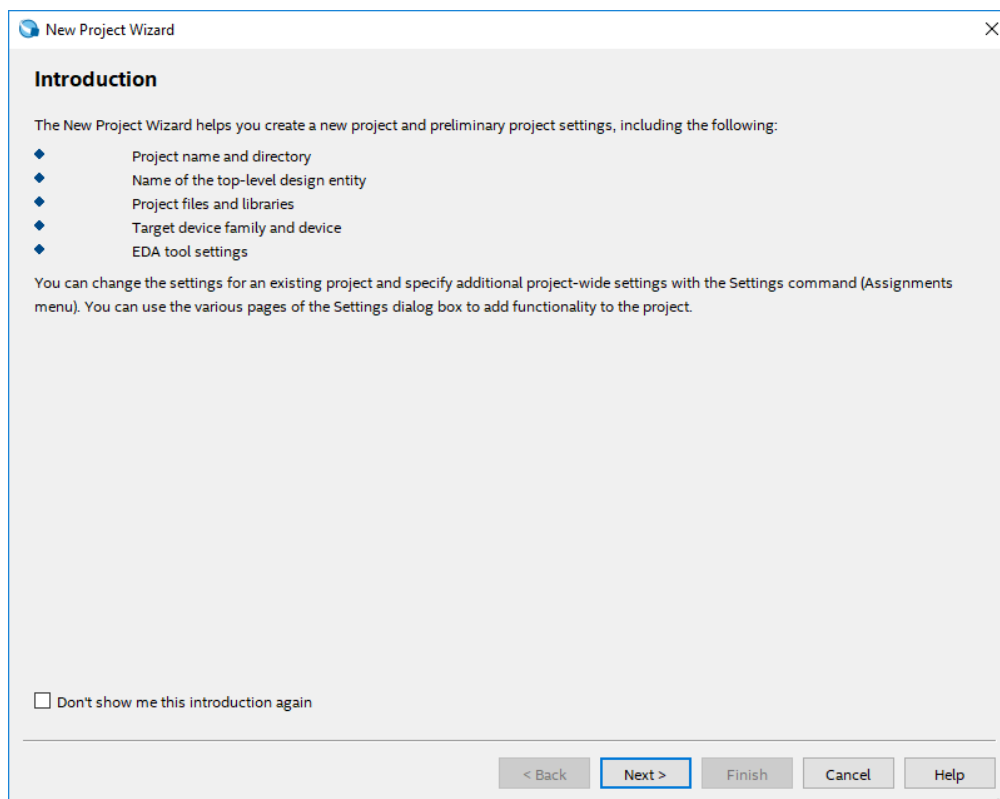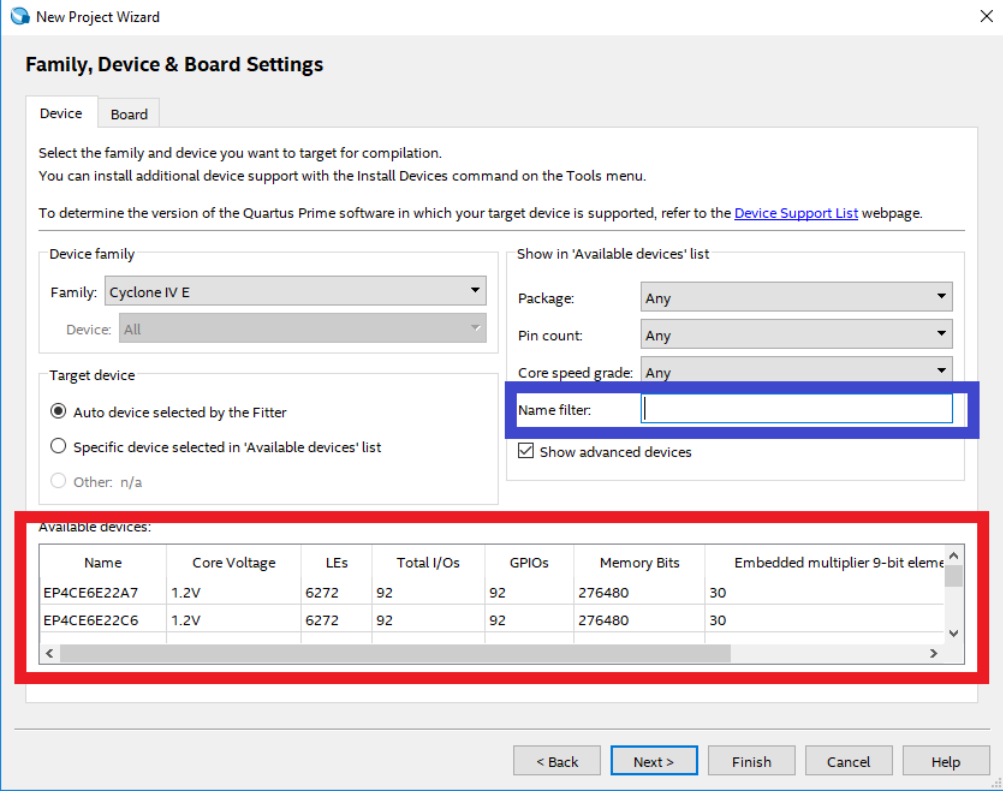

Figure 5.5: New Project Wizard

Follow the instructions listed below to create a new project.

1. Click **Next** to navigate to the project details form.

2. Specify a working directory and a suitable name for the project and click **Next**.

3. Specify the project type. Since we are creating this project from scratch, select **Empty project**. Click Next.

4. If we need to use any already-existing files, we can specify them here. Since there are no such files, click **Next** to continue.

5. Carefully select the device family and model. In order to use *DE0-Nano* development board (which will be used in a later experiment), select the following;

   - Family: Cyclone IV E
   - Device: EP4CE22F17C6N

   See Figure 5.6. Click _____ to continue.

6. If we need to use any additional third part software tools along with Quartus, we can specify them here. Since we are not going to use any, click **Next** to move on.

7. Finally, a summary of all the attributes of the new project will be shown. If everything is correct, click **Finish**. This will create the new project and will bring you to the screen shown in Figure 5.7.

Figure 5.6: Family, device and board selection. Red square: select device type here. Blue square: type a partof the device name to filter the list