Introduction to MAX+PLUS II and Verilog

Created by Richard Freeman
Step 2-6 Modified by Souvik Ray

Objectives

This lab will introduce you to the CAD package we will use the rest of the semester. Over the next several labs, you will be introduced to, and expected to use different tools within Altera MAX+PLUS II. In this lab, you will be introduced to three things:

1. Coding the truth table into Verilog HDL syntax.
2. Using the Waveform editor

In future labs, you may want to refer back to this one.

Step 1: Setup

On the desktop of the computer you should find an icon labeled **MaxII+plus**. This is the software we will be using for the rest of the semester. Double-click the icon and start the program. If you cannot find the icon, click on **Start** menu, **Program** submenu, **Altera** submenu, and select **Max+Plus II 10.2**.

MAX+PLUS II stands for **M**ultiple **A**rray Matri**X** **P**rogrammable **L**ogic **U**ser **S**ystem. It provides three kinds of basic design entries: **Graphic Editor**, **Text Editor**, and **Waveform Editor**. You can see these from the **MAX+PLUS II** dropdown menu.

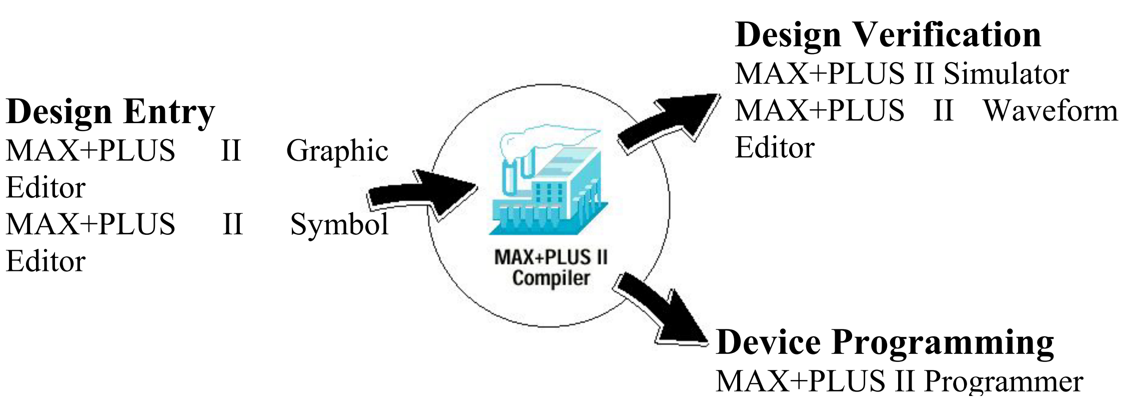

Figure 1: A General Overview of MAX+PLUS II Design Environment
(Figure modified from Figure 2-1 of *ALTERA Max+Plus II Getting Started*)

Step 2. Creating a project in Altera Max

Go to File → Project → Name and set a name for your project. For uniformity and convenience, you can set your project name to the name of the lab, e.g lab1, lab2 etc.

Note: For uniformity and convenience, you should work on your U: drive and have a directory/ file structure as U:/cpre305/LAB1 and save your files in this directory.

Verilog

Step 3 -- To write Verilog code, do a File → New and then select the file type as a text editor file. Save the file with .v extension. Write the verilog code in this file and save it.

Step 4 – Compiling

Before compiling the code, you must set the project to the current file using File → Project → Set Project to Current File. To compile the code, go to File → Project → Save and Compile. If you get errors, you can double click on the errors in the error window and the cursor will be set to the error in the code (not always true). Correct the errors and recompile it.

Waveform Editor

Step 5 -- Now it is time to test the behavior of the circuit. For this, we need to use the Waveform Editor to create a Simulator Channel File (**.SCF*), which we will be using to input the test data into the design. This file will be the input for the Simulator to run and the result will be recorded in the same file.

Select **MAX+plus II | Waveform Editor** to create a simulator channel file. An empty waveform editor file will be opened for you. First, the length of the simulation must be changed. Select **File | End time** to bring up the end End Time dialog.

Next, select **View | Fit in Window** (or press **Ctrl-W**) to view the whole simulation duration in the current window. In this window, you should see a **Ref** field, **Time** field and **Interval** field. Click once under the **Name** column and you should notice that the Ref field and Time field changed to **Start** field and **End** field respectively. Start field indicates the starting time of the simulation, End field indicates the end time of the simulation, and the Interval field indicates the duration between each clock tick.

Right click on anywhere under the **Name** column, and a menu will pop up. Select **Enter Nodes from SNF…** from the menu, and a dialog box will appear. From the **Type** column, un-check the **Group** Type, and click on the **List** button. Upon clicking the List button, the **Available Nodes & Groups** column will show a list of input and output nodes available. Click on the => button to add the whole list into the **Selected Nodes & Groups** column, which will be added to the *SCF* file when you click **OK**. This process allows you to select nodes you want to monitor and view the corresponding waveforms. Finally, select **File | Save As** and save the waveform file.

Simulation

Step 6 -- Select **MAX+PLUS II | Simulator** to run the Simulator. When the Simulator dialog box appears, check and make sure that the Simulation Input file is the one that you saved in the last step. If this file is not shown as the simulation input, check if you have set the project to the current file. Then select **File | Inputs/Outputs**, and choose the correct input file for your simulation. To run the simulation, simply click on the **Start** button. Click on the **Open SCF** button in order to see the simulation result. If you cannot see the complete waveform within the current window select **View | Fit it Window,** or press **Ctrl-W**. Observe the results carefully, and convince yourself and your lab instructor that it is correct.

.