EE 231L

Design with Altera Graphics Design Files

Here is a brief tutorial for designing a digital circuit using an Altera Graphics Design File. For a more detailed tutorial, look in the *MAX+PLUS II Getting Started Manual*, which is available on the Altera website.

1. Find the MAX+plus II program. It should be a button in the lower-left corner of the screen next to the Start menu. Open the Max+plus II program.

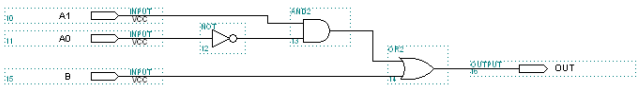
2. From the **File** menu choose **Project** and choose **Name**. (Or click on the  button on the menu bar.) Change to the subdirectory for this lab, or make a subdirectory for this lab if you haven't already done so.

3. Enter a name such as Boolean. This name will be given to the many files created by the program, and should also be the name of your SUBDESIGN.

4. From the **File** menu choose **New**, select a .gdf file. Click OK. Now choose **File**, **Save As**, and save with the same name as the project name.

5. Double-click at an appropriate location of the canvas. An **Enter Symbol** dialog box should pop up. Choose an appropriate symbol from the symbol libraries. The *prim* library has basic symbols, such and AND and NOR gates. The *mf* library has more complex symbols, many of which correspond to 7400 series logic. The *mega_lpm* library has even more complex symbols, which allow you to edit some parameters, such as the number of inputs or outputs you want. (If you know the name of a symbol, such as `and2` or `input`, you can in the bf Symbol Name box.)

6. After placing the needed logic symbols on the canvas, enter symbols for inputs and outputs (from the *prim* library). Double-click on the `PIN_NAME` label and enter the appropriate name. Make connections by clicking on a pin stub and dragging a wire to another pin stub. A design may look something like this:



7. Choose **File**, **Project**, **Save and Check** (or click on the  button on the menu bar). After it saves with no errors, proceed as discussed in the *Getting Started with Max+plus II* tutorial.