Quartus Tutorial: 8-bit 2-1 Multiplexer on the MAX7000S Device

Before you begin:

Create a directory in your home workspace called ***csc343***.
Note: You will use this folder to store all your projects throughout the semester.

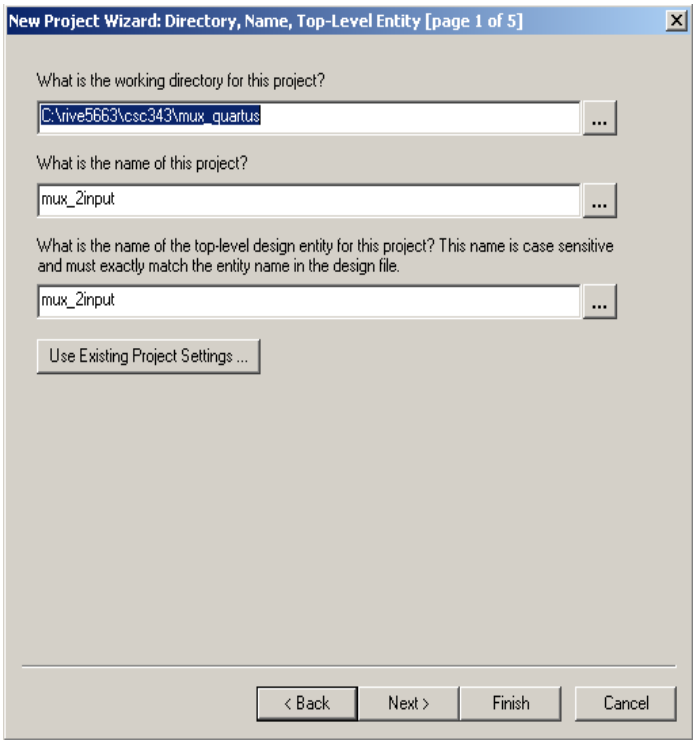
Copy the files ***DEC_7SEG*** and ***mux_2input_pin_assignment*** from the course website into the directory you just created.

1. Open Quartus II: Open a Linux/Unix terminal and type ***quartus***.

2. Create a new Quartus project (***.qpf***):

   2.1. Click ***File -> New Project Wizard***.

   2.2. Click ***Next*** at the bottom of the **New Project Wizard: Introduction** page.
   You will now be on page 1 as shown below.

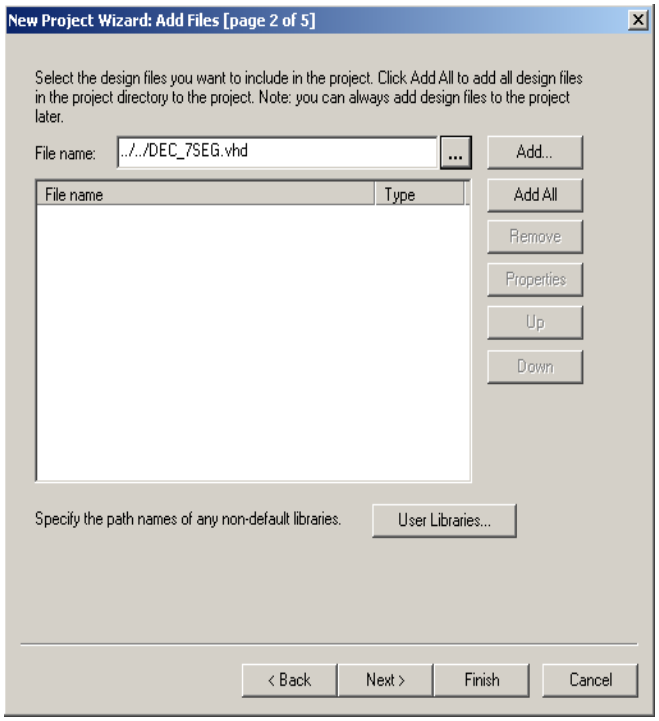


   2.3. Specify the working directory:
   a) Click the **...** button to the right of field **(a)** in the figure above to browse for the ***csc343*** directory you created above.
   The path to this directory will be shown in field **(a)**.

b) Type ***mux_quartus*** after the path that was just displayed in field **(a)**.
Note: This directory does not exist, but Quartus will make it for you.

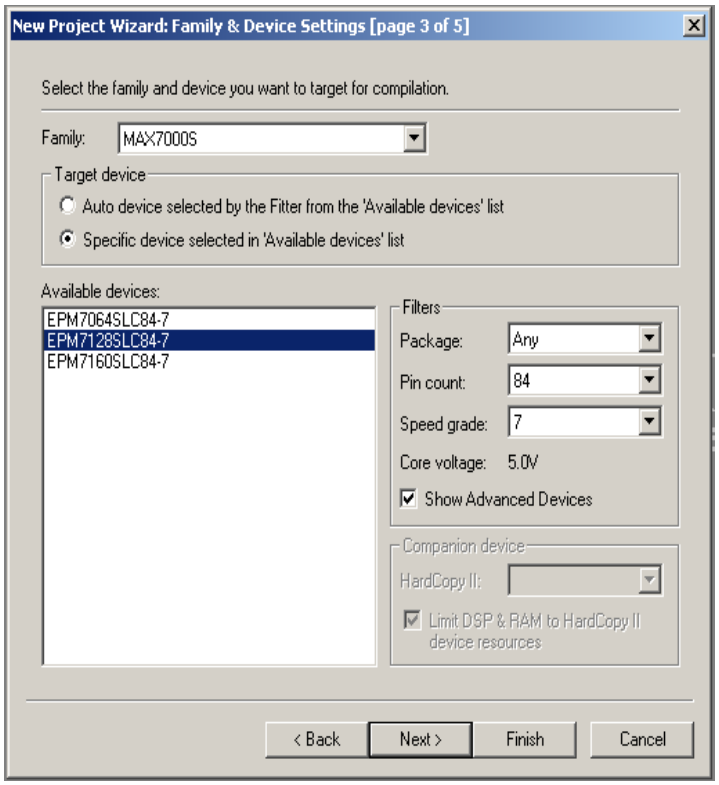
c) Specify the project name: Type ***mux_2input*** in field **(b)**.
Quartus automatically names the top-level design in field **(c)** *the same as the project name*.

2.4. Click ***Next*** at the bottom of page 1 of the **New Project Wizard**.
You will be on page 2 as shown below.



2.5. Click the **...** button to the right of field **(a)** in the figure above to browse for the file ***DEC_7SEG***, then click ***OK***.

2.6. Click ***Add,*** then click ***Next*** at the bottom of page 2 of the **New Project Wizard**.
The You will now be on the **Family & Device Settings** *page as shown below*.



2.7. Set the **Family** name: Select ***Max7000S*** from the drop-down menu in field (a).

2.8. Set the **Speed grade**: Select **8** from the drop-down menu in field (b).

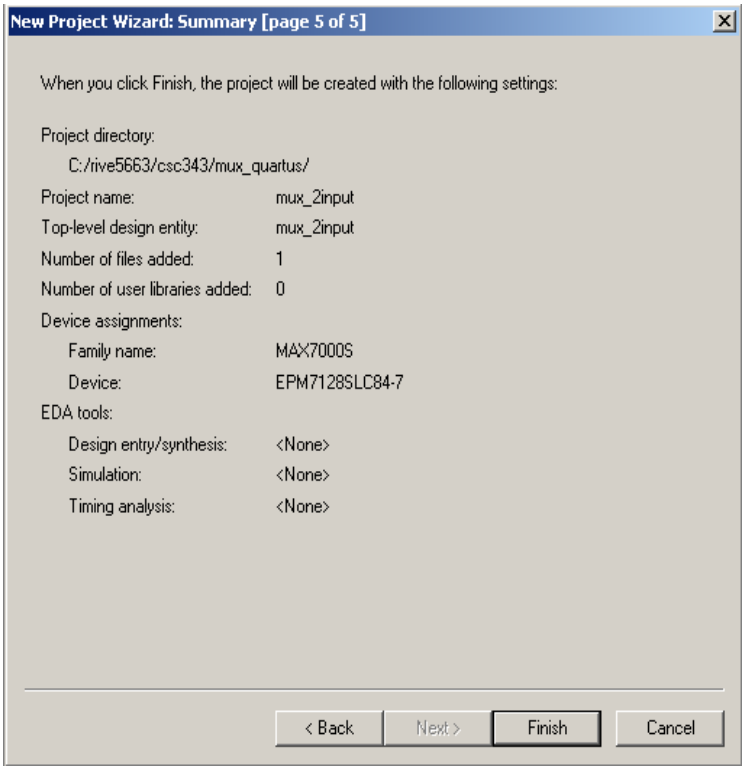
2.9. Set the **Pin count**: Select **84** from the drop-down menu in field (c).

2.10. Select ***EPM7128SLC84-7*** from the available devices in field (d).

Note: These settings are only used when implementing the Altera Max chip on the UP2 Board. The Altera Flex on the UP2 and the Altera Cyclone on the UP3 Board have different settings.

2.11. Click ***Next*** at the bottom of page 3 of the **New Project Wizard**.

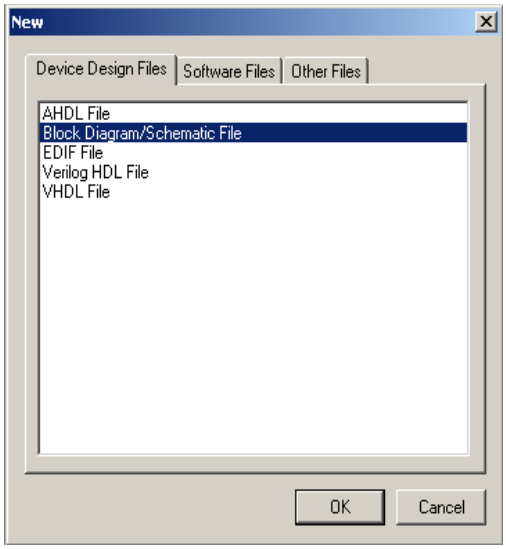
2.12. Click ***Next*** at the bottom of the **EDA Tool Settings** (page 4).
You will now be on Summary page as shown below.



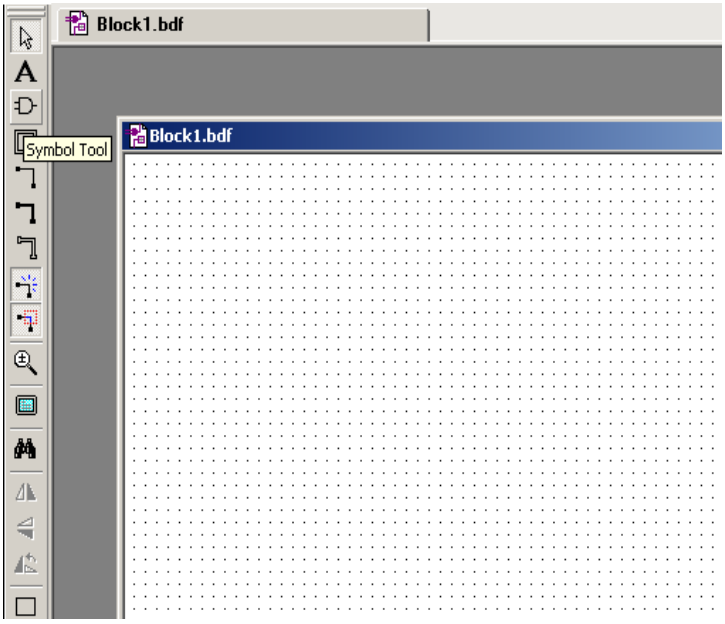
2.13. Click ***Finish*** at the bottom of page 5 of the **New Project Wizard**.

3. Create a new block diagram file(***.bdf***).

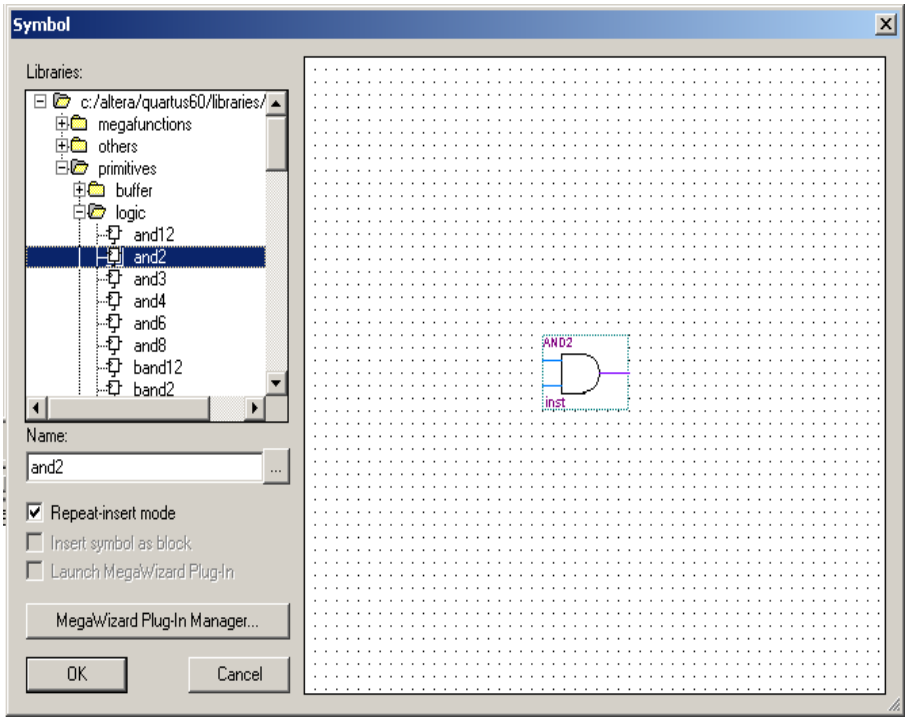
3.1. Click ***File->New***.
A new window appears like the one below.



3.2. Select the ***Block Diagram/Schematic File*** option and click ***OK***.
A new window appears like the one below.



3.3. Select the ***Symbol Tool*** from the toolbar on the left of the block diagram.
A new window appears like the one below.

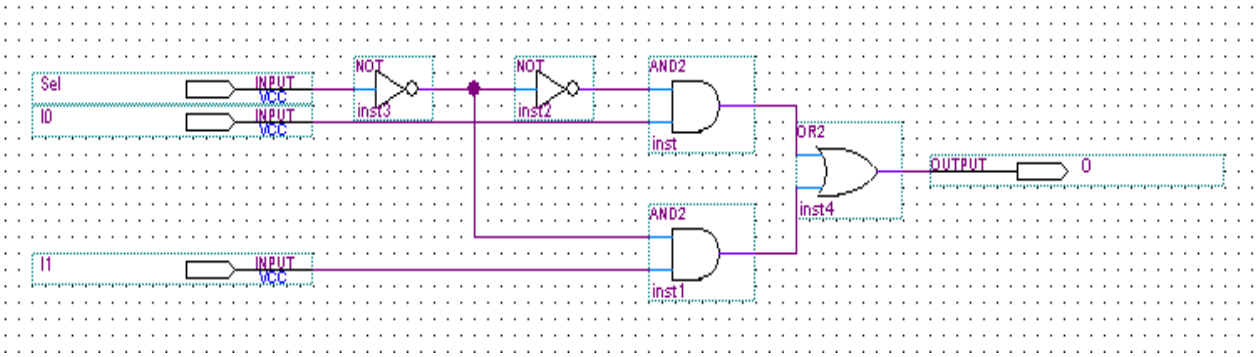


3.4. Click on ***primitives -> logic -> and2*** to get the and symbol and click ***OK***.
The cursor now acts as a stamp in the block diagram window. It will stamp an ***and*** *gate every time you click inside the window.*

3.5. Place two and gates inside the block diagram window referencing the Multiplexer schematic below.

3.6. Right-click inside the block diagram window or click on the ***Selection and Smart Drawing Tool*** at the top of the toolbar to finish using the ***and*** gate.

Note: You can delete mistakes by right-clicking on an object and selecting **Delete**.



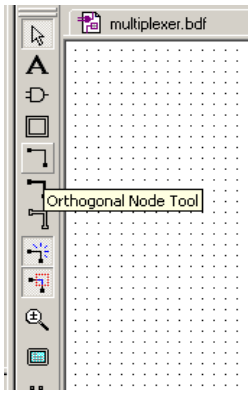
3.7. Place one ***or2*** symbol and one ***not*** symbol in the block diagram window referencing the schematic above.

Note: These symbols can be found in the same place as ***and2.***.

3.8. Place three ***input*** pins and one ***output*** pin in the block diagram window according to the schematic above.

Note: These symbols can be found in ***primitives -> pin*** in the Symbol window.

3.9. Select the **Orthogonal Node Tool** from the toolbar on the left as shown below.



3.10. Connect each object as shown in the schematic above:

a) Click as close as possible to one object.
b) Drag the connection toward a second object until a small box appears.

Note: A unconnected wire shows an **X** along the wire when the wire is not highlighted.

3.11. Name each pin:

a) Double click on the upper left-most ***input*** pin.
b) Type ***Sel*** in the **Pin name(s)** field and click ***OK***.
c) Name the second ***input*** pin ***I0*** and the third ***input*** pin ***I1***.
d) Name the ***output*** pin ***O***.

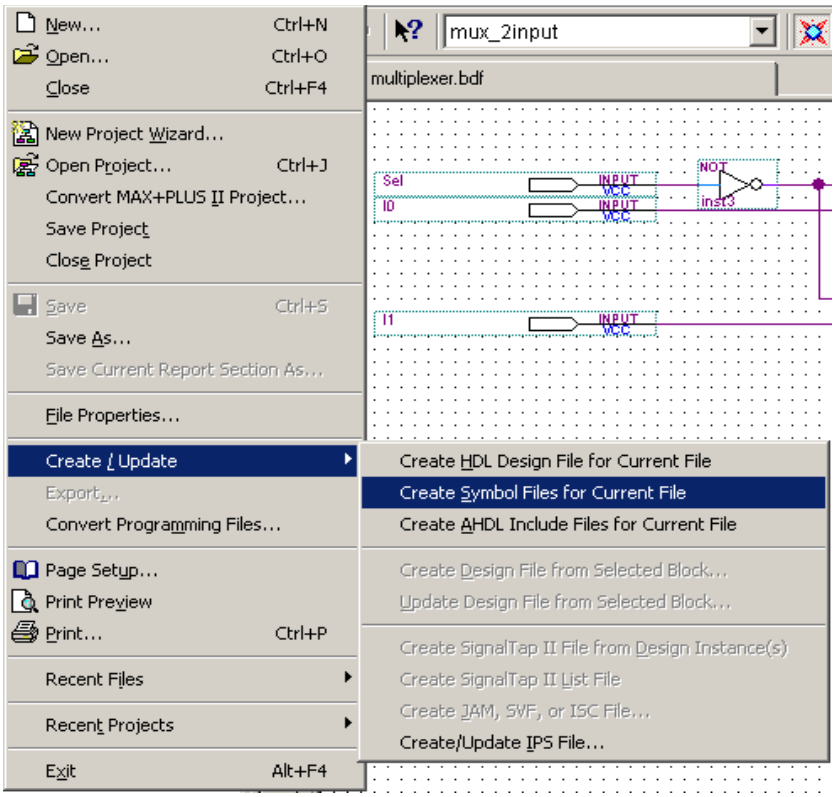
3.12. Save the file as ***multiplexer.bdf***.

4. Create a symbol to represent the above file:

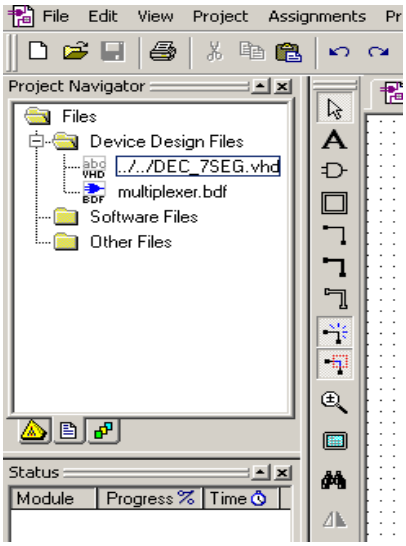
    4.1. Click inside the ***multiplexer.bdf*** to make sure this is the active window.

    4.2. Click ***File -> Create/Update -> Create Symbol Files for Current File*** as in the figure below.

    Note: Make sure the ***multiplexer.bdf*** file is highlighted before creating the symbol.



5. Click on the ***Files*** tab in the **Project Navigator** window shown below.



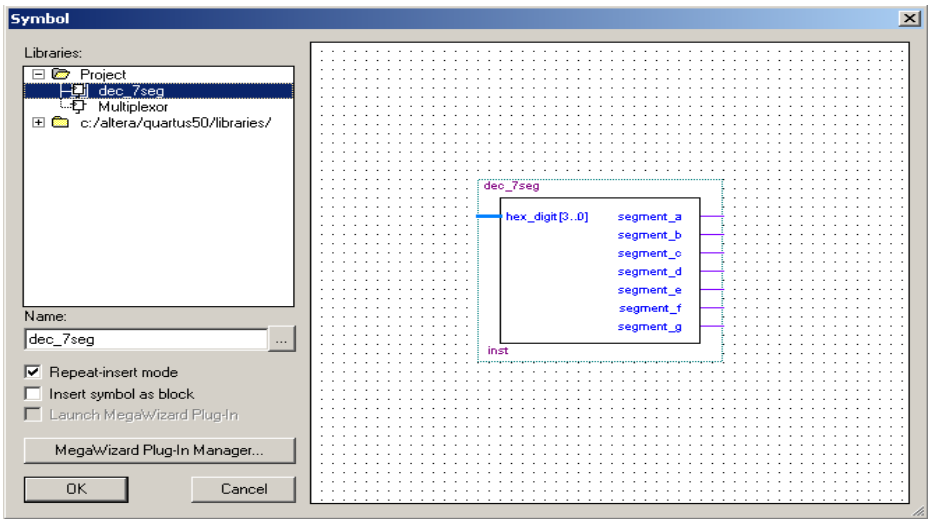
6. Double-click on ***DEC_7SEG.vhd*** to open the file.

7. Create a symbol for this file as well using the process described above.

8. Create a new block diagram file:

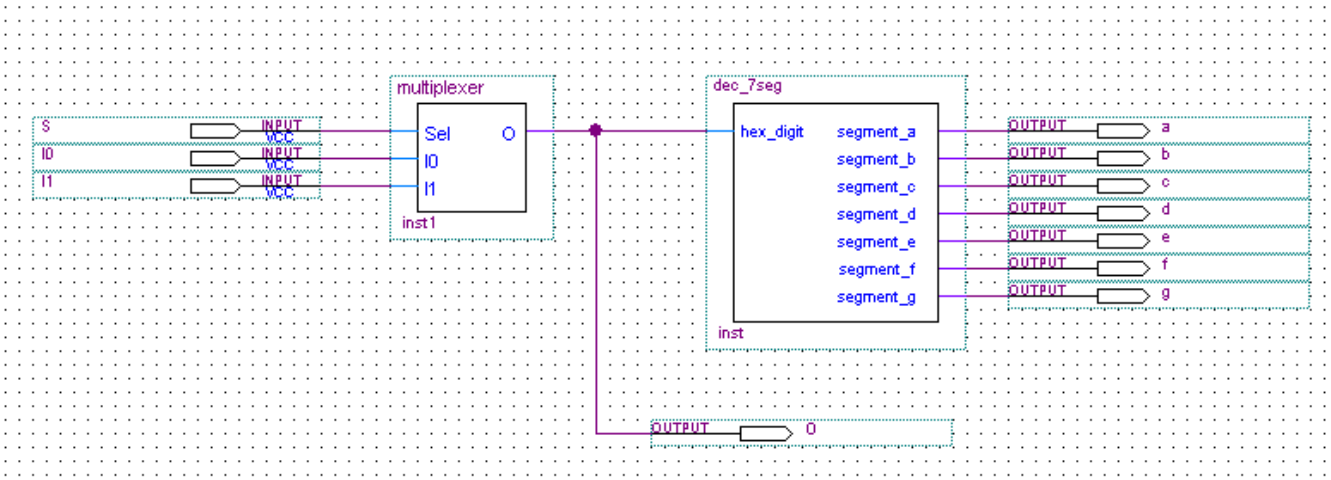
8.1. Repeat steps 5.1, 5.2, and 5.3.

A new **Project** *folder will appear above the altera libraries folder as shown in the figure below.*



8.2. Click on the **Project** folder, select ***dec_7seg*** and click ***OK***.

8.3. Place one ***dec_7seg*** symbol in the block diagram referencing the schematic below.



8.4. Place one ***Multiplexer*** symbol in the block diagram.

8.5. Place three ***input*** pins and seven ***output*** pins as shown in the schematic above.

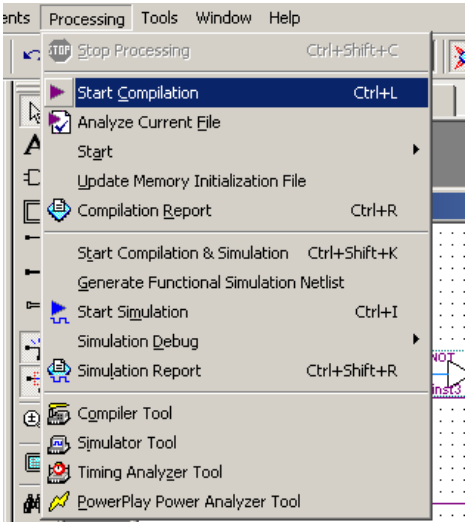
8.6. Connect all objects as shown above using the **Orthogonal Node Tool**.

8.7. Name all pins in the following order from top to bottom, left to right:
Input: **S, I0, I1**
Output: **O, a, b, c, d, e, f, g**

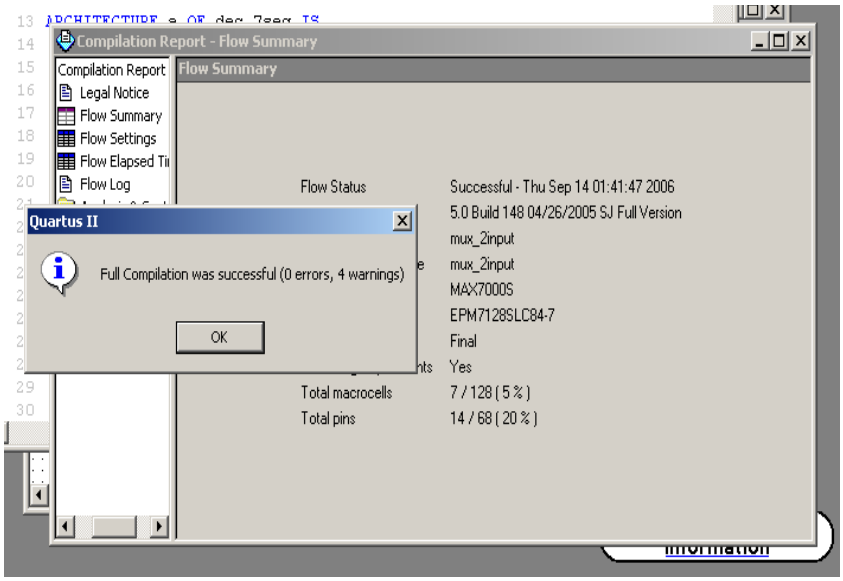
8.8. Save the file as ***mux_2input.bdf***.

9. Compile your design:

9.1. Click ***Processing -> Start Compilation*** as shown below.
Quartus will show the status of the compilation process in the **Status** *window on the left side of the screen and will show a confirmation of completion.*

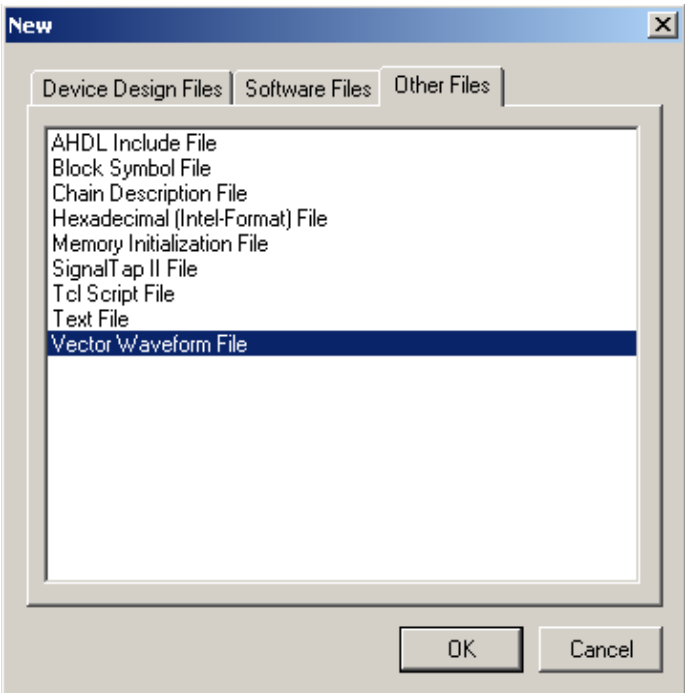


9.2. Click ***OK*** in the confirmation window and exit out of the Compilation Report window that is shown below.

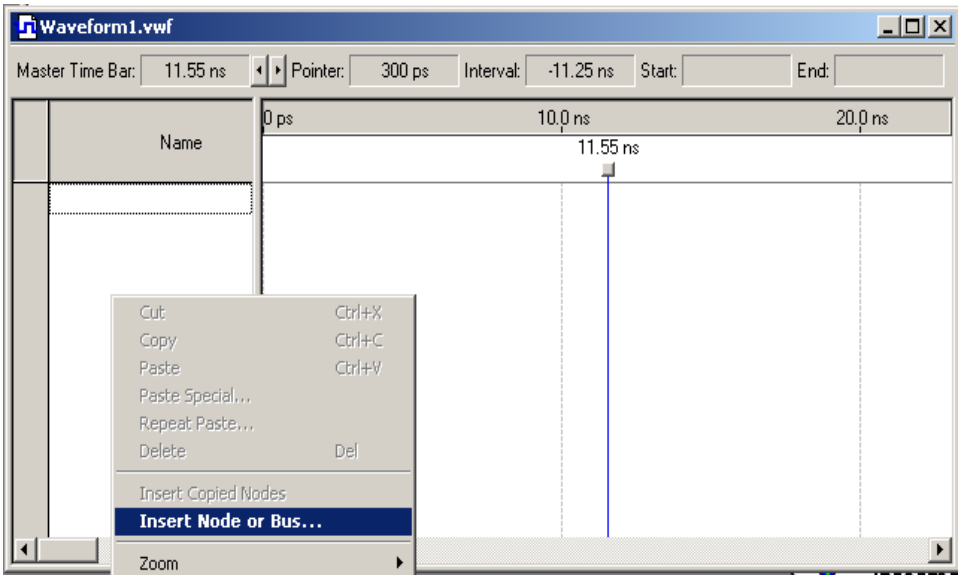


10. Create a vector waveform file (**.vwf**):

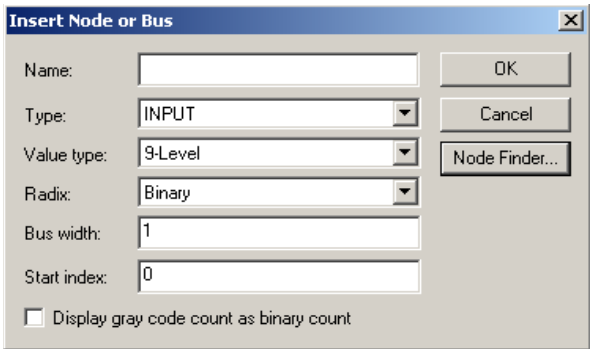
10.1. Click ***File -> New***, and click on the **Other Files** tab as shown below.



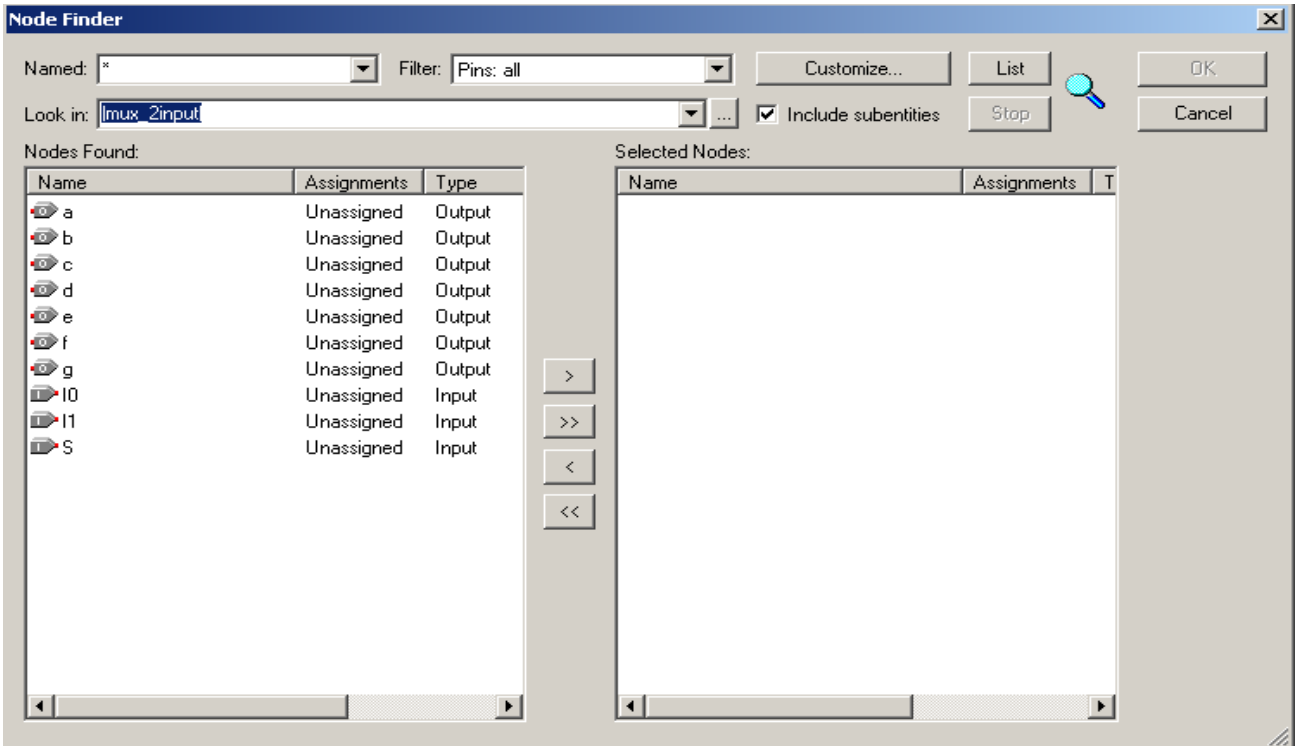
10.2. Select **Vector Waveform File**.
A new window will appear as shown below.



10.3. Right-click under the **Name** field in the left section of the waveform window and select **Insert Node or Bus**.
A new window will appear as shown below.



10.4. Select **Node Finder** on the right side of the window.
A new window will appear as shown below.

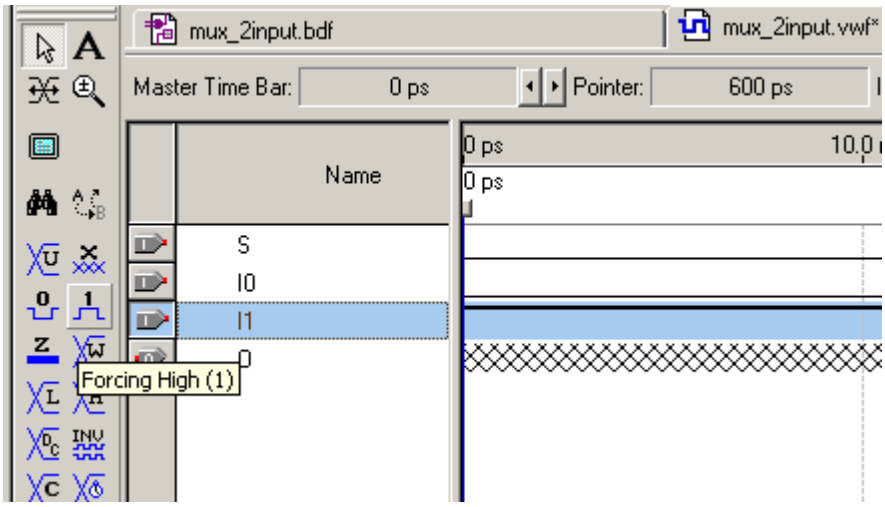


10.5. Click on the **List** button the right side of the window.
All input and output pins in the design will appear in the left window as show above.

10.6. Select **S** then click on the **>** button in the middle of the window.

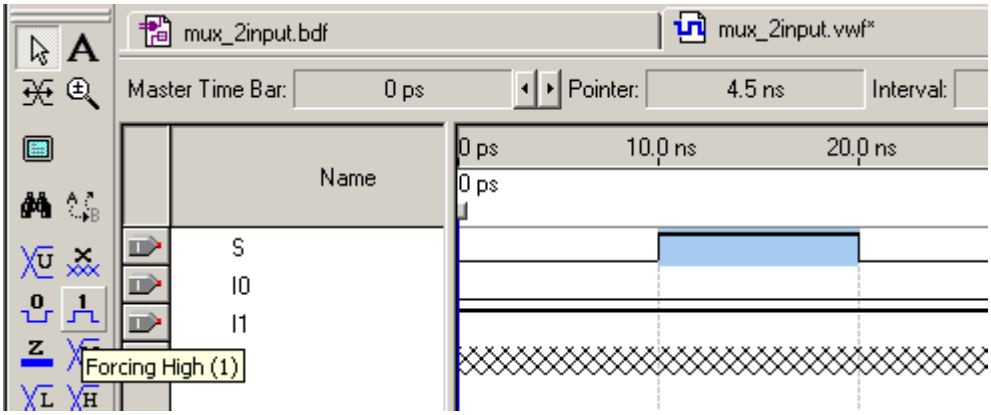
10.7. Repeat step 12.6 for **I0, I1,** and **O** in that order.

10.8. Click ***OK*** to exit the **Node Finder** window and ***OK*** again to insert these items into the vector waveform as shown below.



10.9. Click on **I1** and select the **Forcing High** button from the toolbar on the left.
*This sets **I1** to always input* 1 *as seen above*.

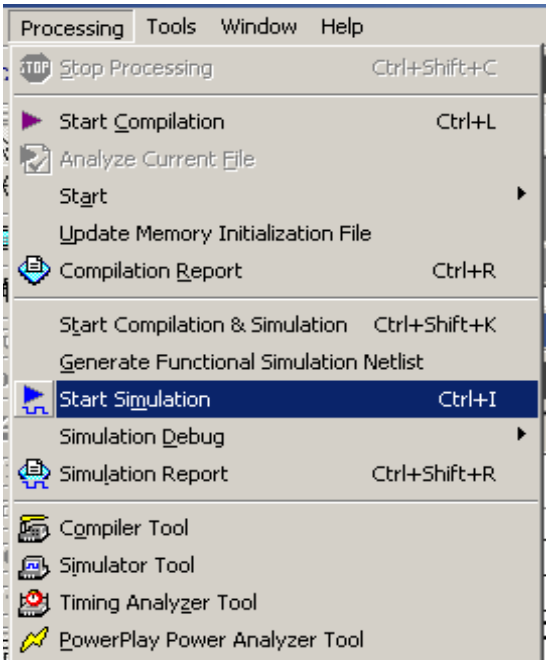
10.10. Click and drag over the interval from 10.0ns to 20.0ns in the **S** signal and select the **Forcing High** button.
This produces a step in the signal shown below.



10.11. Save the file as ***mux_2input.vwf***.

11. Simulate your design:

11.1. Click ***Processing -> Start Simulation*** as shown below.



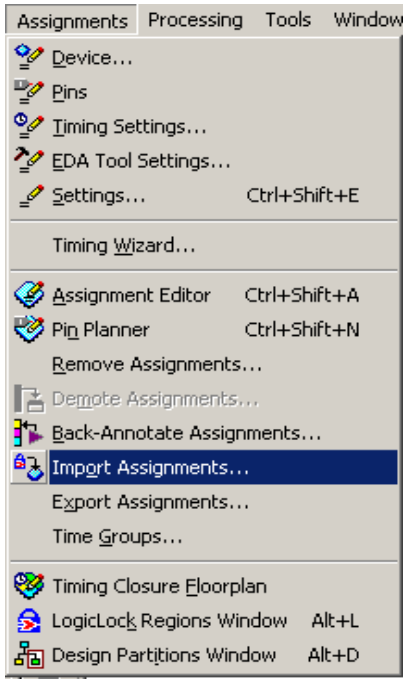
11.2. Click ***OK*** once the confirmation window appears, but DO NOT exit out of the **Simulation Report** window.

Note: The **Simulation Report** window shows the output of your design.

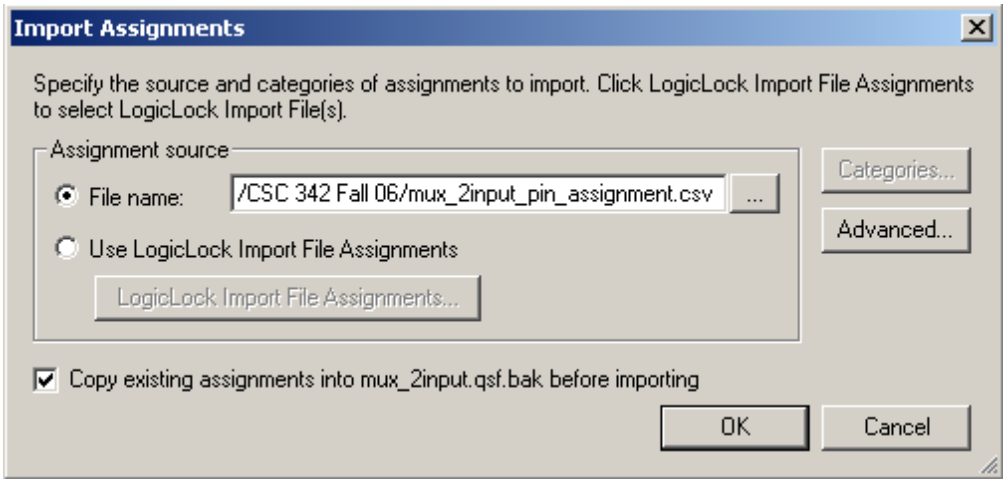
11.3. Analyze the output in the **Simulation Report**. Notice that when **S** is 0, the output is whatever **I0** is, and when **S** is 1, the output is whatever **I1** is.

12. Download your design onto the chip:

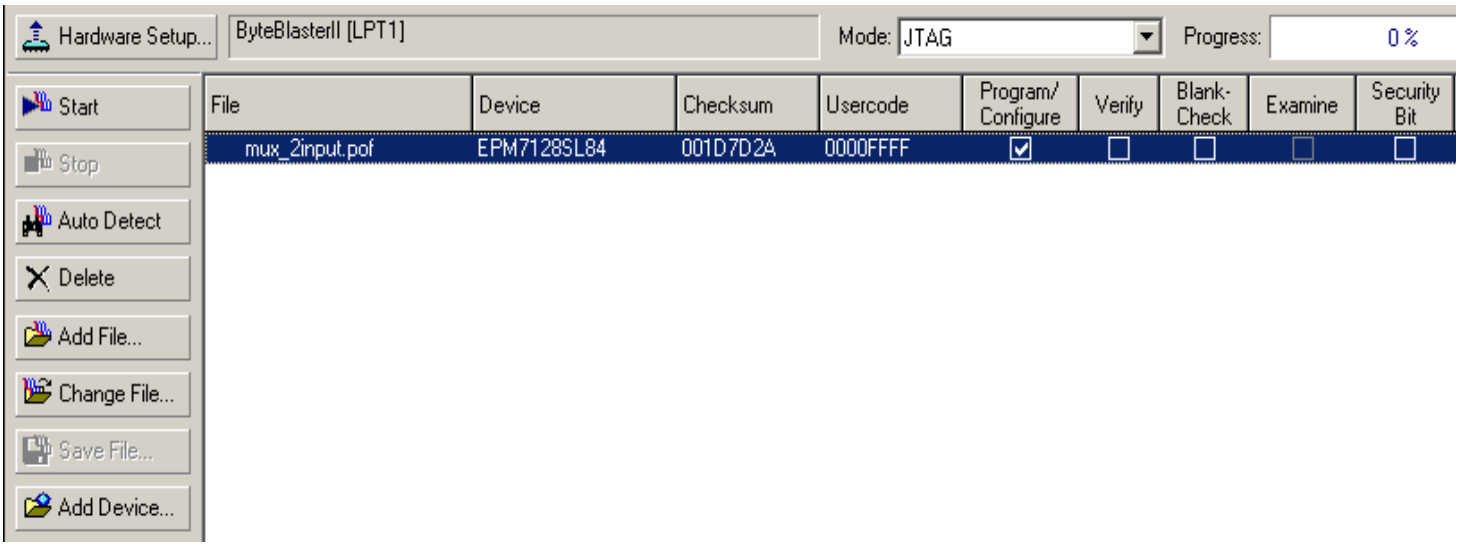
12.1. Click ***Assignments -> Import Assignments***.



12.2. Click on the **...** button in the **Import Assignments** window to browse for the ***mux_2input_pin_assignment*** file you copied at the beginning of the lab and click ***OK***.



12.3. Save your design and recompile to include the pin assignments.
Click ***Tools -> Programmer***.
A new window appears as show below.



12.4. Click **Hardware Setup** and make sure that the ByteBlaster(MV) hardware is listed.

If it is not listed,

a) Click **Add Hardware** and select ByteBlaster(MV) for the Hardware Type
b) Select Parallel Port LPT1 and click ***OK***.

12.5. Click on the **Program/Configure** checkbox of your ***mux2_input.pof*** file in the

Programmer window and click **Start**.
If your file is not listed in the window, click **Add** File, select your file and click ***OK***.

13. Compare your results on the board with that of the waveform. Try as many combinations as you can and see how many outputs you can get to display correctly.