Contents

Appendix A Schematic Entry—Tutorial 1

The Quartus II software and the UP2 development board provide all of the necessary tools for implementing and trying out all of the examples, including building the final general-purpose microprocessor, discussed in this book. The Quartus II software offers a completely integrated development tool and easy-to-use graphical-user interface for the design, and synthesis of digital logic circuits. Together with the UP2 development board, these circuits can be implemented on a programmable logic device (PLD) chip. After downloading the circuit netlist to the PLD, you can see the actual operation of these circuits in the hardware.

The Web Edition version 5.1 of the Quartus II software is included on the accompanying CD-ROM and can also be downloaded from the Altera website found at www.altera.com. The optional UP2 development board can be purchased directly from Altera. The full User Guide for using the UP2 board is on the CD-ROM, and can also be downloaded from the Altera website. This tutorial assumes that you are familiar with the Microsoft Windows environment, and that the Quartus II software has already been installed on your computer. Instructions for the installation of the Quartus II software can be found on the CD-ROM. You also must obtain a free license file from the Altera website in order for the software to function correctly.

The Quartus II development software provides for both schematic and text entry of a circuit design. The Schematic Editor is used to enter a schematic drawing of a circuit. Using the Schematic Editor, logic symbols for the circuit can be inserted and connected together using the drawing tools. The Text Editor is used to enter VHDL or Verilog code for describing a circuit. Regardless of whether you used schematic entry or text entry, your circuit can be synthesized, and then downloaded onto the UP2 development board for implementation and execution of the circuit in hardware.

This tutorial provides a step-by-step instruction for the schematic entry, synthesis, and simulation of an 8-bit 2-to-1 multiplexer circuit. Tutorial 3 (Appendix C) will show how a circuit can be downloaded to the PLD on the UP2 development board so that you actually can see this circuit execute in the hardware.

A.1 Getting Started

A.1.1 Starting Quartus II

After the successful installation of the Quartus II software, there should be a link to the program under the Windows' **Start** button. Click on this link to start the program. You should see the main Quartus II window similar to Figure A.1.

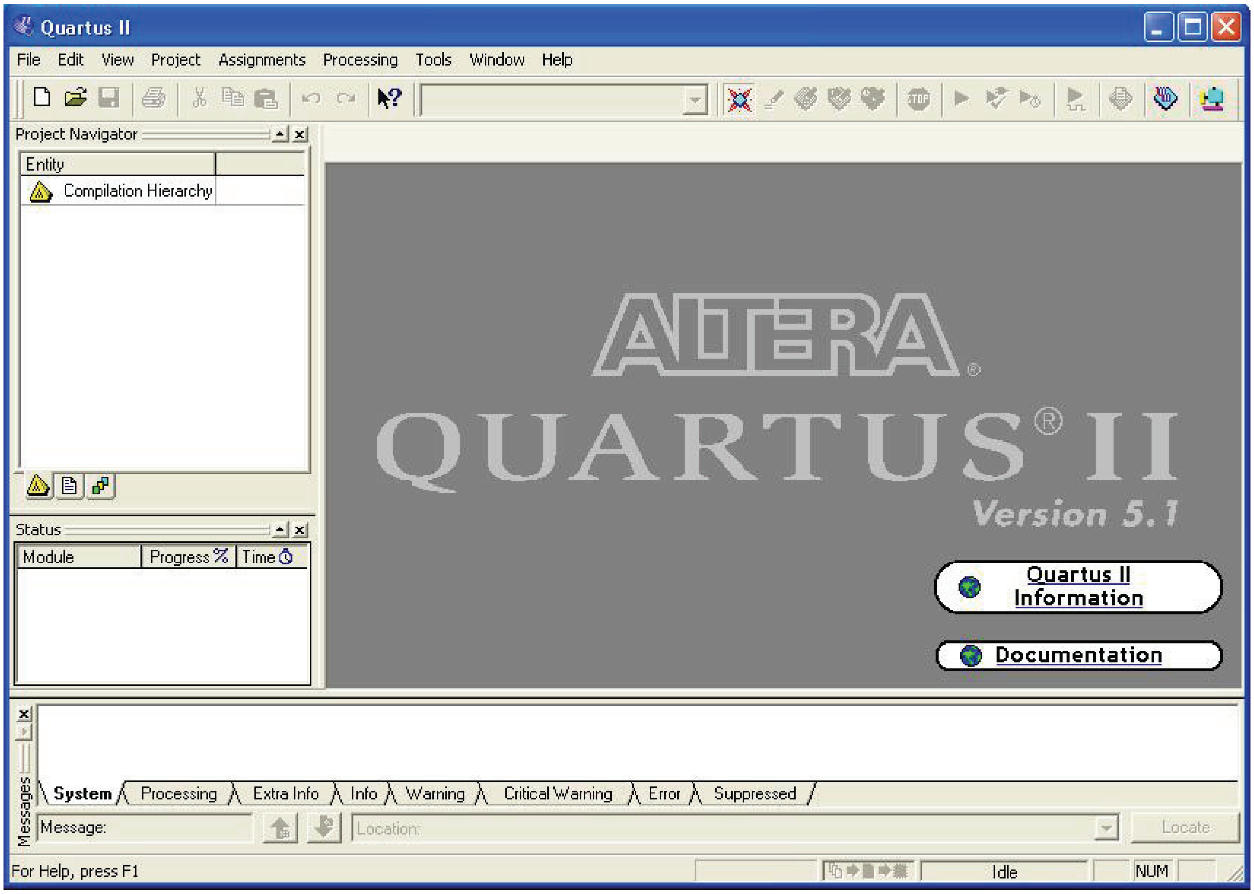


Figure A.1 The Quartus II main window.

A.1.2 Creating a New Project

Each circuit design in Quartus II is called a project. Each project should be placed in its own folder, since the program creates many associated working files for a project. Perform the following steps to create a new project and a new folder for storing the project files.

1. From the Quartus II menu, select **File | New Project Wizard**, and then click **Next** to skip the new project wizard introduction message if it appears. You should see the New Project Wizard: Directory, Name, Top-Level Entity [page 1 of 5] window as shown in Figure A.2.

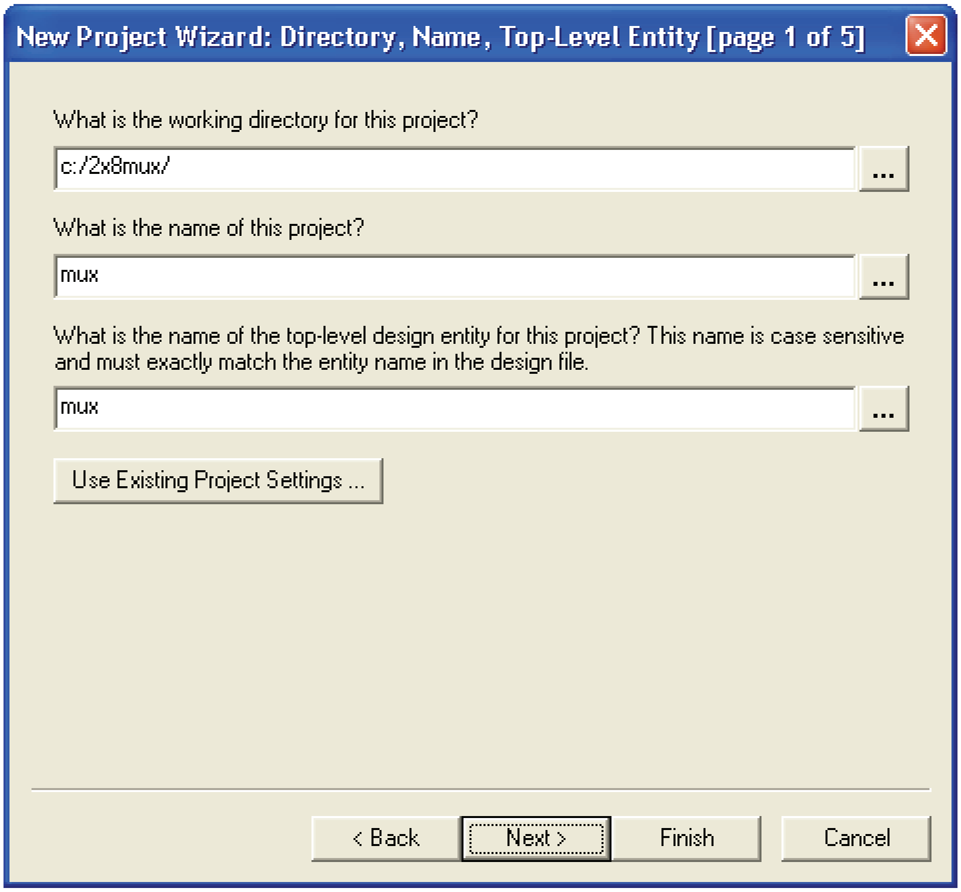


Figure A.2 The New Project Wizard: Directory, Name, Top-Level Entity window with the working directory, the project name, and the top-level entity name filled in.

2. Type in the directory for storing your project. You can also click on the icon next to it to browse to the directory.
   - For this tutorial, type in `c:\2x8mux` to create a folder called `2x8mux` in the root directory of the C drive.
3. You also need to give the project a name.
   - For this tutorial, type in the project name `mux`.
4. A project may have more than one design file. Whether your project has one or more files, you need to specify which design file is the top-level design entity. The default name given is the same as the project name. However, you can use a different name.
   - For this tutorial, leave the top-level file name as `mux`, and click **Next** to continue to the next window.
5. Since the directory `c:\2x8mux` does not yet exist, Quartus II will inform you of that and asks whether you want to create this new directory. Click **Yes** to create the directory.
6. In the New Project Wizard: Add Files [page 2 of 5] window, you can optionally add the circuit source files associated with your project.
   - For this tutorial, since we are starting a new project, we do not yet have any source files, so click **Next** to continue to the next window.
7. In the New Project Wizard: Family & Device Settings [page 3 of 5] window as shown in Figure A.3, we select

the target PLD device that we will be implementing the circuit on. The UP2 board has two different PLD devices for programming: the FLEX 10K, which is the larger PLD, and the MAX7000S.

- For this tutorial, we will use the FLEX 10K device. In the Family drop-down box, select **FLEX10K**.
- In the Available devices list, select the device **EPF10K70RC240-4**. If this device is not listed, then you need to reinstall the Quartus II program, and make sure that all of the FLEX 10K devices are installed.
- Click **Next** to continue to the next window.

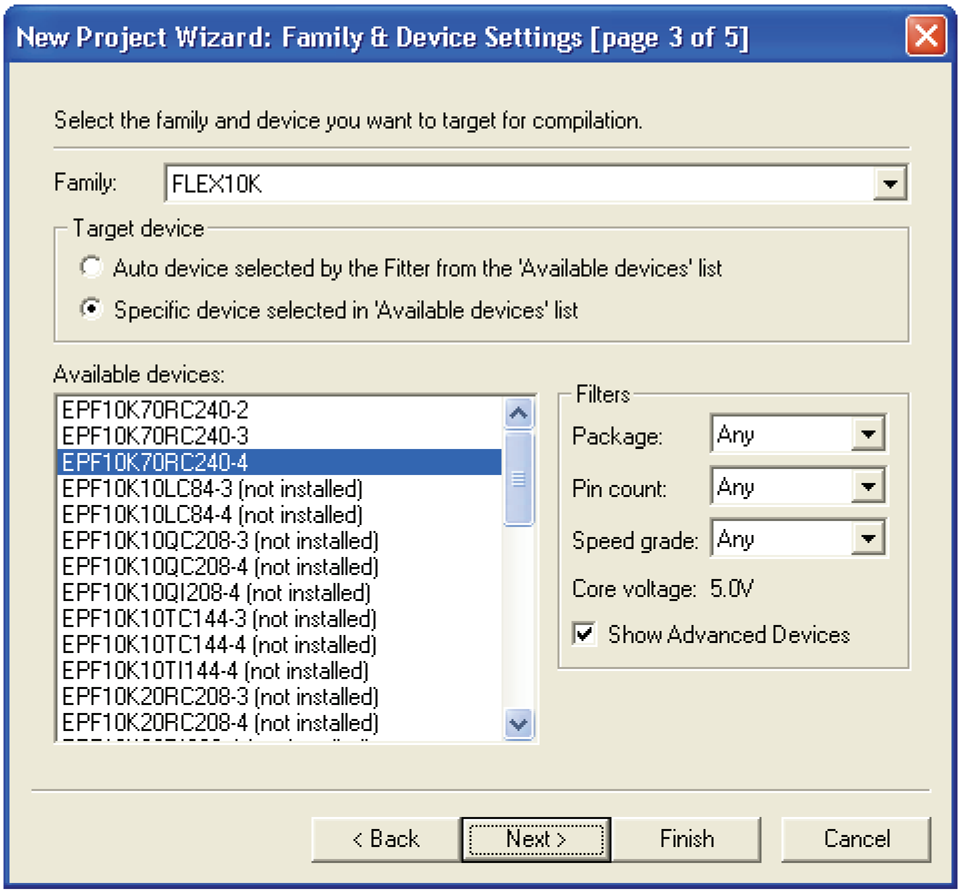


Figure A.3 The New Project Wizard: Family & Device Settings window with the device EPF10K70RC240-4 selected.

8. In the New Project Wizard: EDA Tool Settings [page 4 of 5] window, we do not have any EDA tools to use for this project, so click **Next** to continue to the next window.
9. The final window is a summary of the choices that you have just made. Click **Finish** to create your new project.

A.2 Using the Graphic Editor

After creating a new project, we are now ready to start the Graphic Editor for manually drawing the circuit.

A.2.1 Starting the Graphic Editor

1. From the Quartus II menu, select **File | New**.
2. Under the **Device Design Files** tab, select **Block Diagram/Schematic File**, and then click **OK**. You should see the Graphic Editor window similar to the one shown in Figure A.4. Any circuit diagram can be drawn in this Graphic Editor window.

3. Select **File | Save As** to save the file. Type in `mux` for the filename. The default file extension is `.bdf` (for block design file). Recall that in Step 4 of Section A.1.2, we have specified that the top-level filename is `mux`.

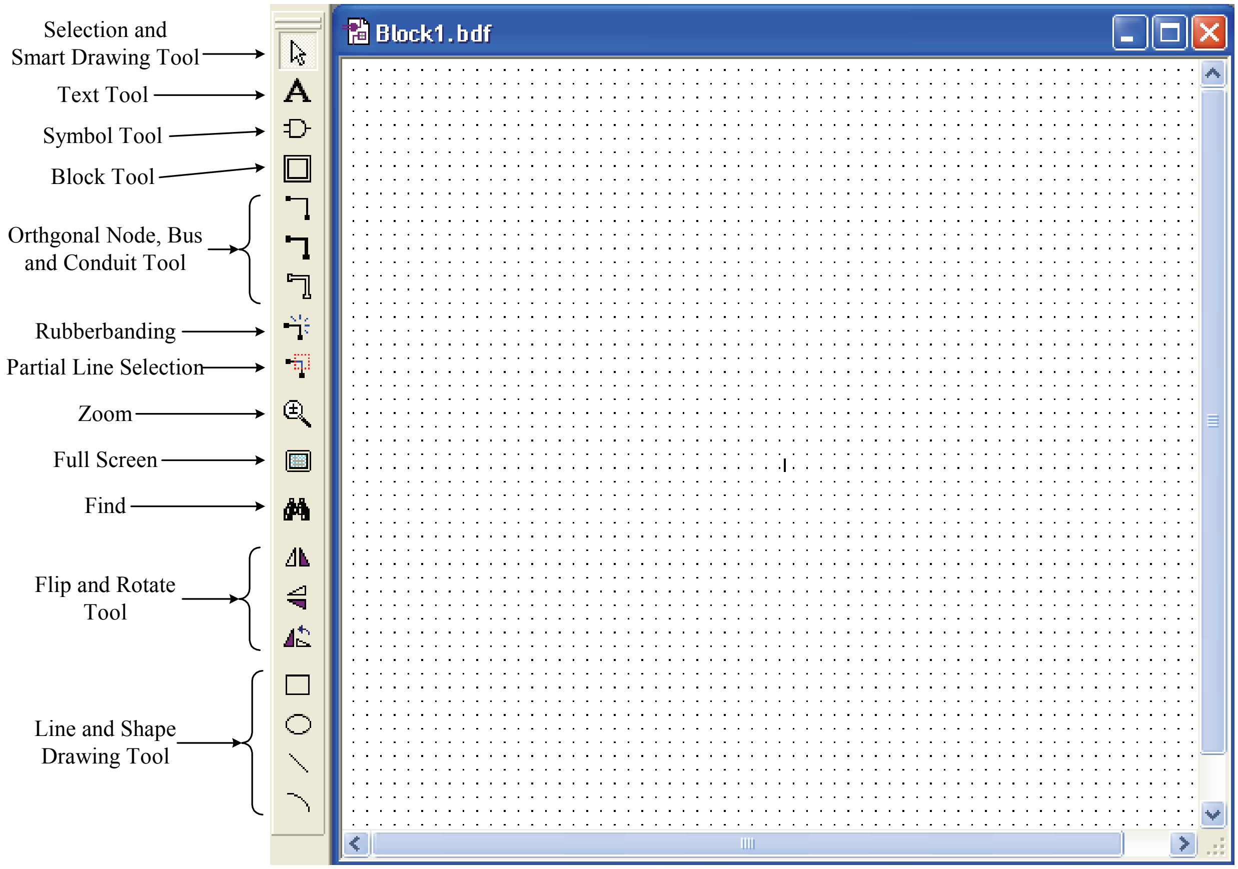


Figure A.4 The Graphic Editor window with the graphics toolbar showing on the left.

A.2.2 Drawing Tools

In Figure A.4, the tools for drawing circuits in the Graphic Editor are shown in the toolbar on the left side. There are the standard tools such as text writing, zoom, full screen, find, flip and rotate, and line and shape drawing. The main tool that you will use is the Selection and Smart Drawing tool. This smart drawing tool allows you to perform many different operations depending on the context in which it is used. Two main operations performed by this tool are selecting objects and making connections between logic symbols. The Symbol tool allows you to select logic symbols from the library or from your own design files. The three Orthogonal tools allow you to draw connection lines that are not connected to another object. The Rubberbanding button turns on or off the rubberbanding function. When rubberbanding is turned on, connection lines are adjusted automatically when symbols are moved from one location to another. When rubberbanding is turned off, moving a symbol will not affect the lines connected to it.

A.2.3 Inserting Logic Symbols

4. To insert a logic symbol, first select the Selection and Smart Drawing tool, and then double-click the pointer on an empty spot in the Graphic Editor window. You should see the Symbol window as shown in Figure A.5.

   - Alternatively, you can click on the Symbol tool to bring up the Symbol window.

Available symbol libraries are listed in the Libraries box. These libraries include the standard primitive gates, standard combinational and sequential components, and your own logic symbols located in the current project directory.

All of the basic logic gates, latches, flip-flops, and input and output connectors that we need are located in the `primitives` folder. If this folder is not listed, then click on the plus (+) sign to expand the libraries folder. Within the `primitives` folder are several subfolders. The basic gates are in the `logic` subfolder; the latches and flip-flops are in the `storage` subfolder; and the input and output connectors are in the `pin` subfolder. Your own circuits that you want to reuse in building larger circuits (if any) are found in the directory where they are stored in, and listed in the `Project` folder.

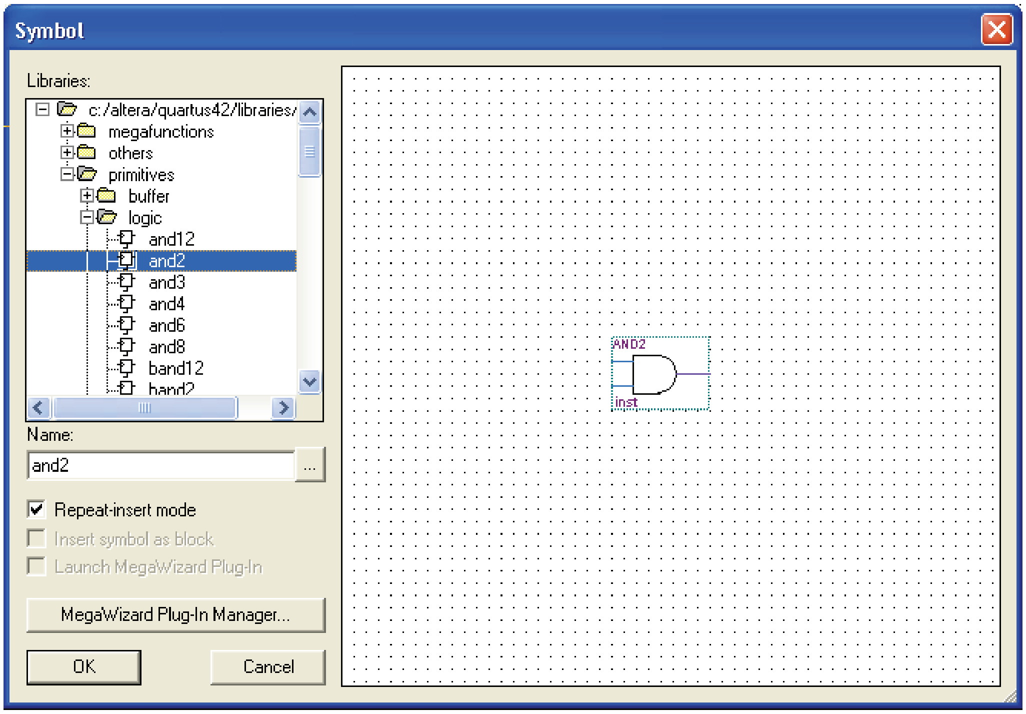


Figure A.5 The Symbol selector window.

5. Expand the `logic` subfolder by clicking on the plus sign next to it to see a list of logic gate symbols available in that library. The logic symbols are sorted in alphabetical order.
6. Double-click on the logic symbol name that you want in order to insert that symbol into the Graphic Editor. If you just select the symbol with a single-click, then you will also have to click on the **OK** button after making the selection.
7. If the Repeat-insert mode box is checked, then you can insert several instances of the same symbol until you press the **Esc** key.

   For this tutorial, insert the following symbols into the Graphic Editor:

   - A 2-input AND gate (`and2`) found in the `logic` subfolder.
   - A 2-input OR gate (`or2`) found in the `logic` subfolder.
   - A NOT gate (`not`) found in the `logic` subfolder
   - An input signal connector (`input`) found in the `pin` subfolder.
   - An output signal connector (`output`) found in the `pin` subfolder.

A unique number is assigned to each instance of a symbol and is written at the lower-left corner of the symbol. This number is used only as a reference number in the output netlist and report files. The numbers that you see may be different from those in the examples.

A.2.4 Selecting, Moving, Copying, and Deleting Logic Symbols

- To select a logic symbol in the Graphic Editor, simply click on the symbol using the Selection and Smart Drawing tool. You can also select multiple symbols by holding down the **Ctrl** key while you select the symbols. An alternative method is to trace a rectangle around the objects that you want to select. All objects inside the rectangle will be selected.
- To de-select a symbol, simply click on an empty spot in the Graphic Editor.
- To move a symbol, simply drag the symbol.
- To copy a symbol, first select it and then perform the Copy and Paste operations. An alternative method is to hold down the **Ctrl** key while you drag the symbol.
- To delete a symbol, first select it and then press the **Delete** key.
- To rotate a symbol, right-click on the symbol, select **Rotate by Degrees** from the pop-up menu, and select the angle to rotate the symbol.

  Perform the following operations for this tutorial:

  1. Make a copy of the 2-input AND gate
  2. Make two more copies of the input signal connector
  3. Position the symbols similar to Figure A.6

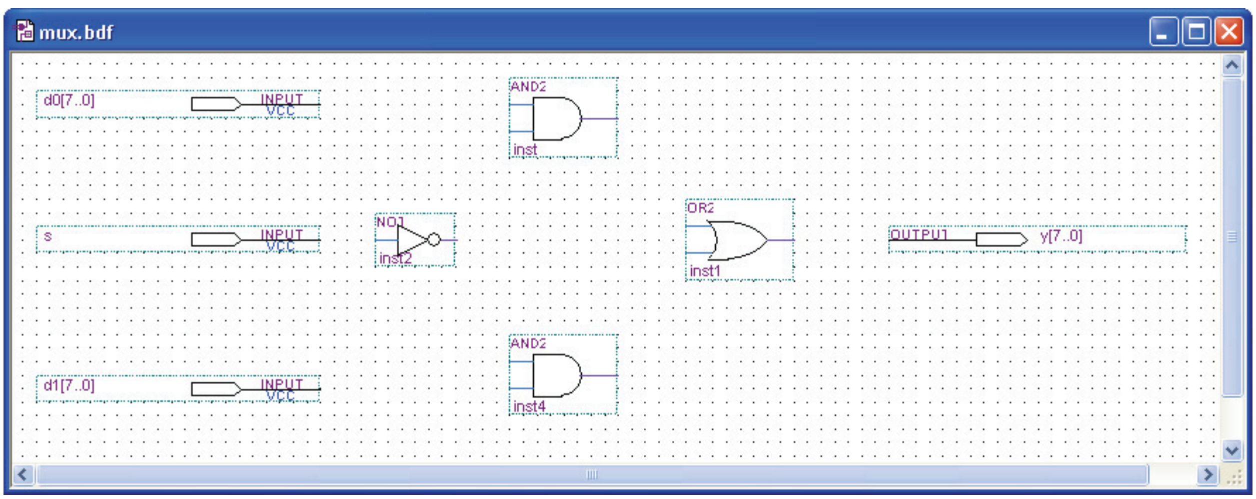


Figure A.6 Symbol placements for the 2-to-1 multiplexer circuit.

A.2.5 Making and Naming Connections

- To make a connection between two connection points, use the Selection and Smart Drawing tool and drag from one connection point to the second connection point. Notice that, when you position the pointer to a connection point, the arrow pointer changes to a crosshair.
- To change the direction of a connection line while dragging the line, simply release and press the mouse button again, and then continue to drag the connection line.
- You can also make a connection between two connection points by moving a symbol so that its connection point touches the connection point of the second symbol. With rubberbanding turned on, you can now move one

symbol away from the second symbol, and a connection line is automatically drawn between them.

- If you want to make a connection line that does not start from a connection point, you will need to use the Orthogonal Node tool instead of the Selection and Smart Drawing tool.
- Do not use the Line tool to make connections; this tool is only for drawing lines and not actually making a connection.
- Once a connection is made to a symbol, you can move the symbol to another location, and the connection line is adjusted automatically if the rubberbanding function is turned on. However, if the rubberbanding function is turned off, the connection will be broken if the symbol is moved.
- To make a connection between two lines that cross each other, you need to use the Orthogonal Node tool and double-click the junction point (i.e., the point where the two lines cross). To remove a connection point, double-click on it again with the Orthogonal Node tool.

Figure A.7 Making or deleting a connection point with the Orthogonal Node tool.

- To select a line segment, simply single click on it. To select the entire line (with several line segments connected in different directions), you need to double-click on it.
- Use the Orthogonal Bus tool to draw a bus connection.
- To change a single node line to a bus line, right-click on the line and select **Bus Line** from the pop-up menu. Select **Node Line** from the pop-up menu to change it back to a node line.
- A bus must also have a name and a width associated with it. Right-click on the bus line at the point where you want to place the name. Select Properties, and then type in the name and the width for the bus. For example, `data[7..0]` is an 8-bit bus with the name `data`, as shown in Figure A.8.
- To change the name, just double-click on the name and edit it.
- To connect one line to a bus, connect a single line to the bus, and then give it the same name as the bus with the line index appended to it. For example, `data2`, is bit two of the `data` bus, as shown in Figure A.8.

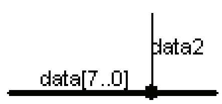


Figure A.8 A single connection line connected to an 8-bit bus with the name **`data`**.

- To check whether a name is attached correctly to a line, select the line, and the name that is attached to the line will also be selected.
- To name an input or output connector, select its name label by single-clicking it twice, and then double-clicking it once. You can now type in the new name. Pressing the **Enter** key will move the text entry cursor to the name label for the symbol below the current symbol. Alternatively, you can just double-click on the input or output connector, and the Properties window for that pin will open up which allows you to enter the pin name, among other things.
- A bus line connected to an input or output connector must have the same bus width as the connector.

For this tutorial, perform the following operations to look like Figure A.9:

1. Name the three input connectors `d0[7..0]`, `s`, and `d1[7..0]`
2. Name the output connector `y[7..0]`
3. Connect and name the five bus lines `d0[7..0]`, `d1[7..0]`, `and0[7..0]`, `and1[7..0]`, and `y[7..0]`

4. Connect the single lines from the input connector `s` to the inverter and to the two AND gates
5. Select **File | Save** to save the design file.

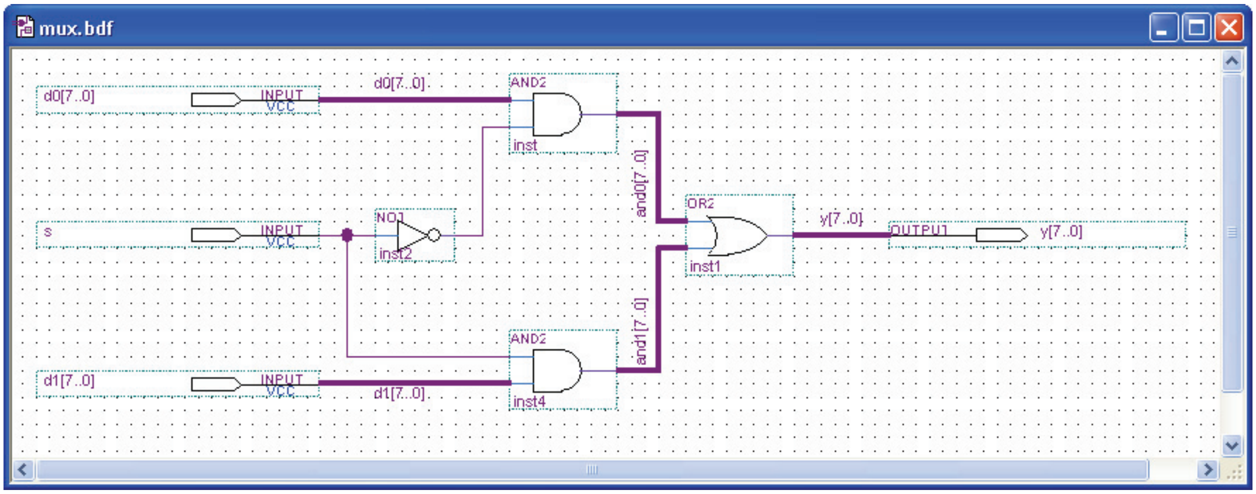


Figure A.9 Connections and names for the 2-to-1 multiplexer circuit.

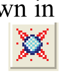
A.2.6 Selecting, Moving and Deleting Connection Lines

- To select a straight connection line segment, just single-click on it.
- To select an entire connection line with horizontal and vertical segments, just double-click on it.
- To select a portion of a line segment, first turn on the Use Partial Line Selection button, and then drag a rectangle around the line segment. Only the portion of the line segment that is inside the rectangle will be selected.
- After a line is selected, it can be moved by dragging.
- After a line is selected, it can be deleted by pressing the **Delete** key.

A.3 Managing Files in a Project

A project may have one or more design files associated with it.

A.3.1 Design Files in a Project

- To see the files that are currently associated with a project, click on the **Files** tab  in the Project Navigator window as shown in Figure A.10. If the Project Navigator window is not displayed, then click on the Project Navigator button  to display it. Figure A.10 shows that this project has only one file named `mux.bdf`.

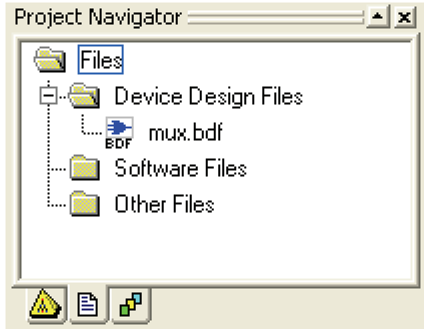


Figure A.10 Files associated with a project as shown in the Project Navigator window.

A.3.2 Opening a Design File

- To open a design file, simply double-click on the file that is listed in the Project Navigator window. Depending on the type of file, the associated editor will be used. The Graphic Editor is used to edit a Block Diagram/Schematic File, and a text editor is used to edit a VHDL or Verilog text file.

A.3.3 Creating a New Design File

- To create a new schematic drawing design file, select **File | New** from the Quartus II menu. In the **Device Design Files** tab, select **Block Diagram/Schematic File**. The newly created design file is not automatically added into the current project. If you want to include this new design file in the project, you have to add it in by following the instructions in Section A.3.4.

A.3.4 Adding Design Files to a Project

- To add another design file to the current project, select **Project | Add/Remove Files in Project** from the Quartus II menu. Alternatively, you can right-click on the folder icon labeled **Device Design Files** in the Project Navigator window, and then select **Add/Remove Files in Project** from the pop-up menu.
- This will bring up the Files category under the Settings window.
- From the Files window, you can choose additional files to be added into the project by either manually typing in the file name or browsing to the directory and then selecting it.
- Click on the **Add** button to add individual files, or click on the **Add All** button to add all of the files in the selected directory.

A.3.5 Deleting Design Files from a Project

- To delete a design file from a project, simply select it in the Project Navigator window, and then press the **Delete** key. Alternatively, you can right-click on the file that you want to delete, and then select **Remove File from Project** from the pop-up menu.

A.3.6 Setting the Top-Level Entity Design File

- In Section A.1.2 where it discusses how to create a new project, you also had to specify the name of the top-level design file. If you want to change the top-level entity to another design file, you can do so by right-clicking on the file that you want to be the top-level entity in the Project Navigator window. From the pop-up menu, select **Set as Top-Level Entity**.

A.3.7 Saving the Project

1. Select **File | Save Project** to save the project and all of its associated files.

A.4 Analysis and Synthesis

After drawing your circuit with the Graphic Editor, the next step is to analyze and synthesize it. During this step, Quartus II collects all of the necessary information about your circuit, and produces a netlist for it.

1. From the Quartus II menu, select **Processing | Start | Start Analysis & Synthesis** to synthesize the circuit. Alternatively, you can click on the icon .
2. If there are no errors in your circuit, you should see the message "Quartus II Analysis & Synthesis was successful" in the Message window at the bottom.
3. If there are errors then they will be reported in the Message window and highlighted in red. You can double-click on the error message to see where the error is in the circuit. Go back and double check your circuit with the one shown in Figure A.9 to correct all of the errors.

A.5 Circuit Simulation

Circuit simulation allows you to observe the behavior of the circuit before actually implementing the circuit in hardware. The Quartus II program can perform either a timing simulation or a functional simulation of a circuit. A functional simulation provides only information on the logic values of each signal simulated, whereas a timing simulation includes also the signal propagation delay information. In this tutorial, we will only perform a functional simulation.

A.5.1 Creating a New Vector Waveform File

To simulate a circuit design, you need to first create test vectors for specifying what the input values for your design are. Test vectors are specified in a vector waveform file having the file extension `.vwf`. In the vector waveform file, you will also specify what output signals you want to observe in the simulation.

1. From the Quartus II menu, select **File | New**.
2. Select the **Other Files** tab in the New window.
3. Select **Vector Waveform File** and click **OK**. A new Vector Waveform window similar to Figure A.11 opens up.
4. To change the time scale grid size increments to 5 ns, select **Edit | Grid Size**. Type in `5` for the time period, and select `ns` for the unit.
5. To overwrite the simulation input file with the simulation result, select **Tools | Options** to bring up the options window. Expand the **General** category if it is not already expanded by clicking on the plus sign icon. Select **Processing**, and put in a check mark in the check box for **Overwrite simulation input file with simulation results**. Click **OK** to accept the changes and close the window.
6. Select **File | Save** to save the Vector Waveform file. Use the default filename `mux.vwf`, which is the same as the top-level entity name but with the file extension `.vwf` (for vector waveform file). Make sure that the check box for **Add file to current project** is checked. The file will now be listed in the Project Navigator window under the folder **Other Files**.

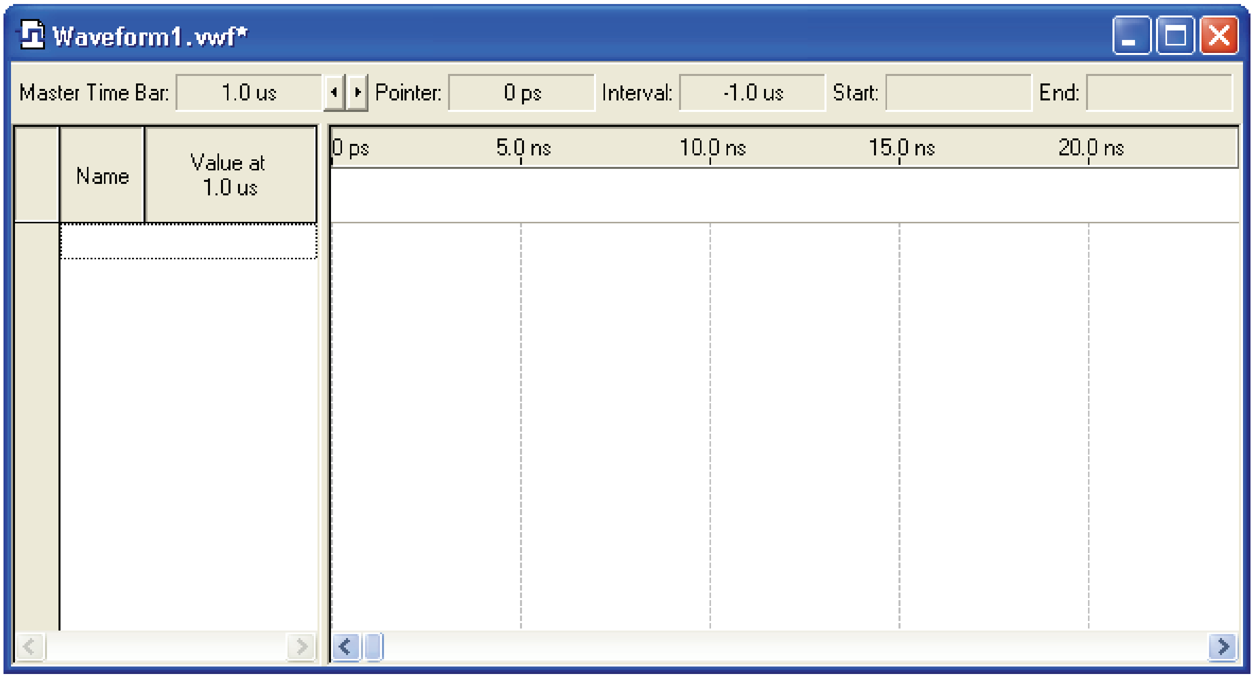


Figure A.11 A new Vector Waveform window for specifying the test vectors for simulation.

A.5.2 Selecting Input Test Signals

From the Vector Waveform window as shown in Figure A.11, you will select the test vectors to be simulated. Test vectors are the input and output signals from your circuit design.

1. From the Quartus II menu, select **Edit | Insert Node or Bus**. Alternatively, you can double-click on the white area below the heading label **Name** in the Vector Waveform window.
2. In the Insert Node or Bus window, click on the **Node Finder** button to bring up the Node Finder window.
3. Make sure that the **Filter** selected is **Pins: all**. Click on the **List** button to list all of the available pins from your circuit design in the Nodes Found (left side) pane as shown in Figure A.12
4. Select the signal(s) that you want to see in the simulation from the Nodes Found pane, and then click on the **>** button to move the selected signal(s) to the Selected Nodes pane. You can hold down the **Ctrl** key to select multiple signals. Clicking on the **>>** button will move all of the signals listed in the Nodes Found pane to the Selected Nodes pane.
   - For this tutorial, select the three input signals `d0`, `d1` and `s`, and the one output signal `y`. Note that for the three bus signals (`d0`, `d1`, and `y`), you need to select the entire bus (such as `d0`) and not the individual bits of the bus (such as `d0[1]`).
   - The symbols  and  next to the signal names denote whether the signal is an input or output signal respectively.
5. Repeat Step 4 until you have selected all of the signals that you want to see in the simulation.

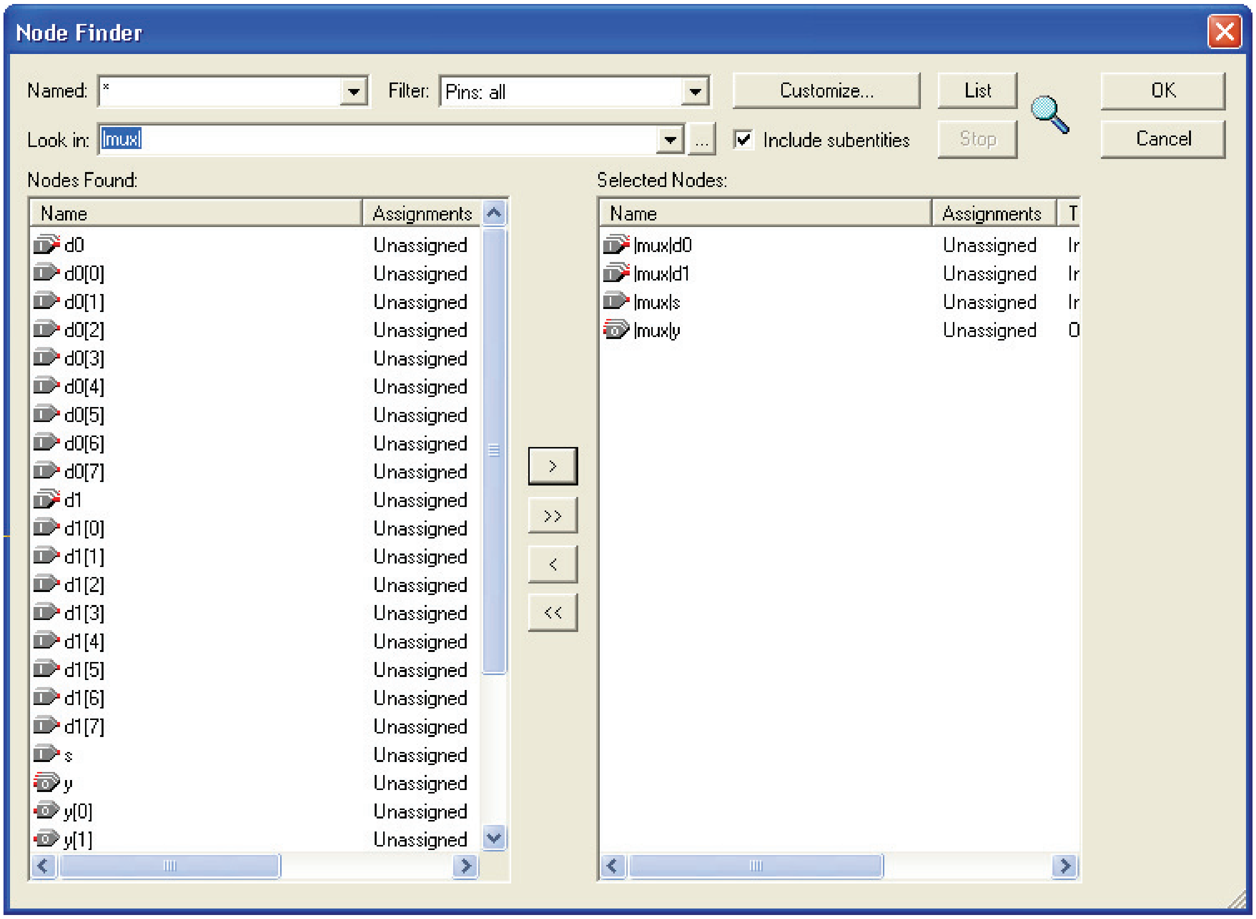


Figure A.12 The Node Finder window for adding input and output signals for simulation.

6. Click on **OK** when you are finished. The four selected signals will now be transferred to the Insert Node or Bus window having the generic name "Multiple Items."
7. Click on the **OK** button in the Insert Node or Bus window to insert the selected signals to the Vector Waveform window as shown in Figure A.13.

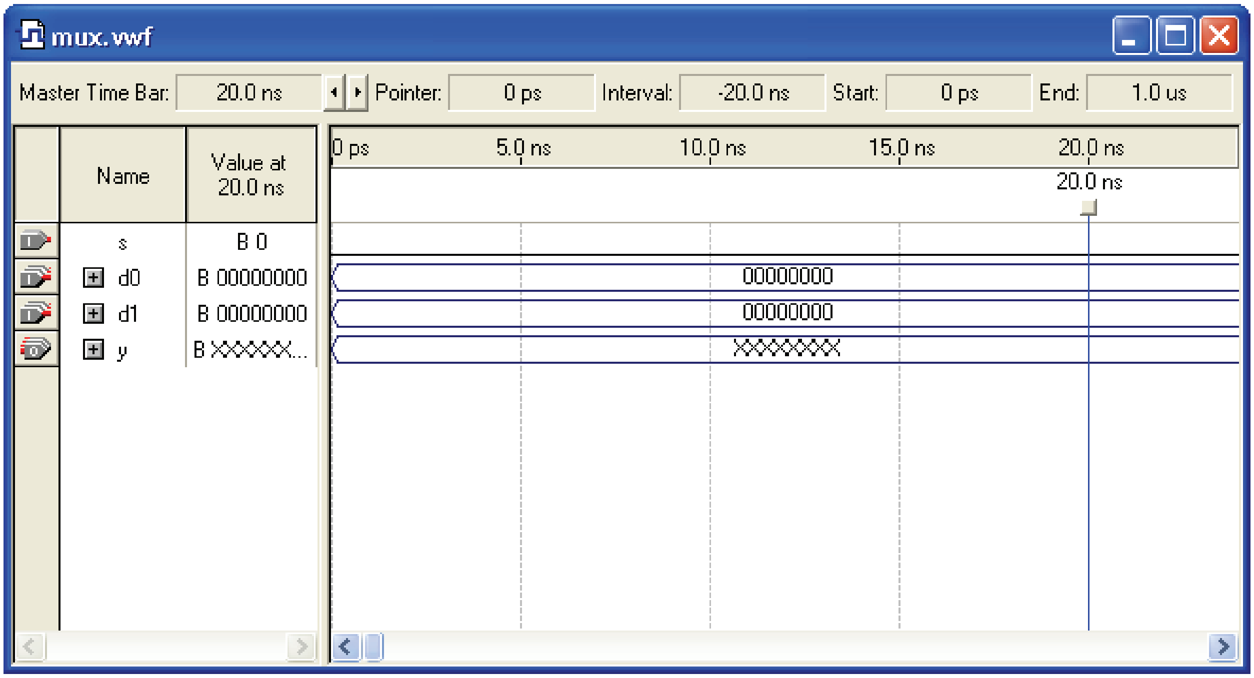


Figure A.13 The Vector Waveform window with the four signals for simulation.

A.5.3 Customizing the Vector Waveform Window

- You can rearrange the signals in the Vector Waveform window by dragging the signal icons such as up or down. First select the signal by clicking on the icon that you want to move. Then drag it to the new location.
  1. For this tutorial, move the signal `s` from where it is to be the first signal listed.
- To delete a signal, just select the signal by clicking on its name and then pressing the **Delete** key.
- For bus signals, you can see the individual bits by clicking on the plus icon to expand it.
- To change the radix of the displayed value, right-click on the signal name and select Properties. In the Node Properties window, change the Radix to what you want.
  1. For this tutorial, change the radix for the three bus signals, `d0`, `d1` and `y`, to be Unsigned Decimal.

A.5.4 Assigning Values to the Input Signals

The next thing is to assign values to all of the input signals.

1. Using the Selection pointer tool, drag from time `10 ns` to `20 ns` for the `s` signal only. While dragging the mouse, the selected signal range will be boxed in, and when you release the mouse the selected area will be highlighted in blue.
2. Click on the icon in the toolbar on the left to set the signal in the selected range to a logical 1 value as shown in Figure A.14.
3. Next, drag from time `0ns` to `5ns` for the `d0` signal only.
4. Click on the icon in the toolbar on the left. Type in the value `5` and click **OK** to set the value for the `d0` bus signal to decimal `5` in the selected time interval as shown in Figure A.14.
5. Repeat Steps 3 and 4 for the remaining input values for d0 and d1 according to Figure A.14.

6. Select **File | Save** to save the vector waveform file. The filename should be `mux.vwf`.

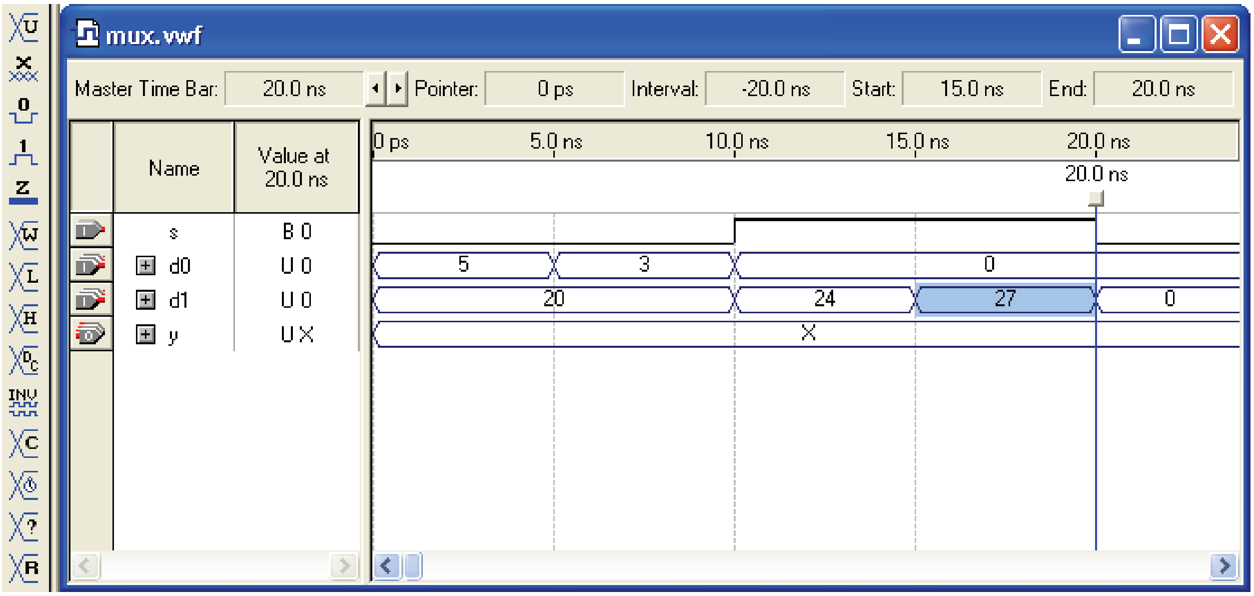


Figure A.14 Changing the input signal values.

A.5.5 Generate Functional Simulation Netlist

In order for the Quartus II program to perform a functional simulation of the circuit, it must first generate a functional simulation netlist of the circuit.

1. From the Quartus II menu, select **Tools | Simulator Tool** to bring up the Simulator Tool window as shown in Figure A.15.
2. In the Simulator Tool window, make sure that the Simulation Input filename is `mux.vwf`.
3. From the **Simulation mode** drop-down menu, select **Functional**.
4. Click on the **Generate Functional Simulation Netlist** button.
5. If there are no errors in your circuit, you should see the message “Quartus II Analysis & Synthesis was successful” in the Message window at the bottom.
6. If there are errors then they will be reported in the Message window and highlighted in red. You can double-click on the error message to see where the error is in the circuit. Go back and double check your circuit with the one shown in Figure A.9 to correct all of the errors.

A.5.6 Starting the Simulator

We are now ready to simulate the design.

1. In the Simulator Tool window as shown in Figure A.15, put a check mark in the box that says **Overwrite simulation input file with simulation results**.
2. Click on the **Start** button  and watch the progress of the simulation.
3. At the completion of the simulation, you should see in the Message window at the bottom, a line that says the simulation was successful.

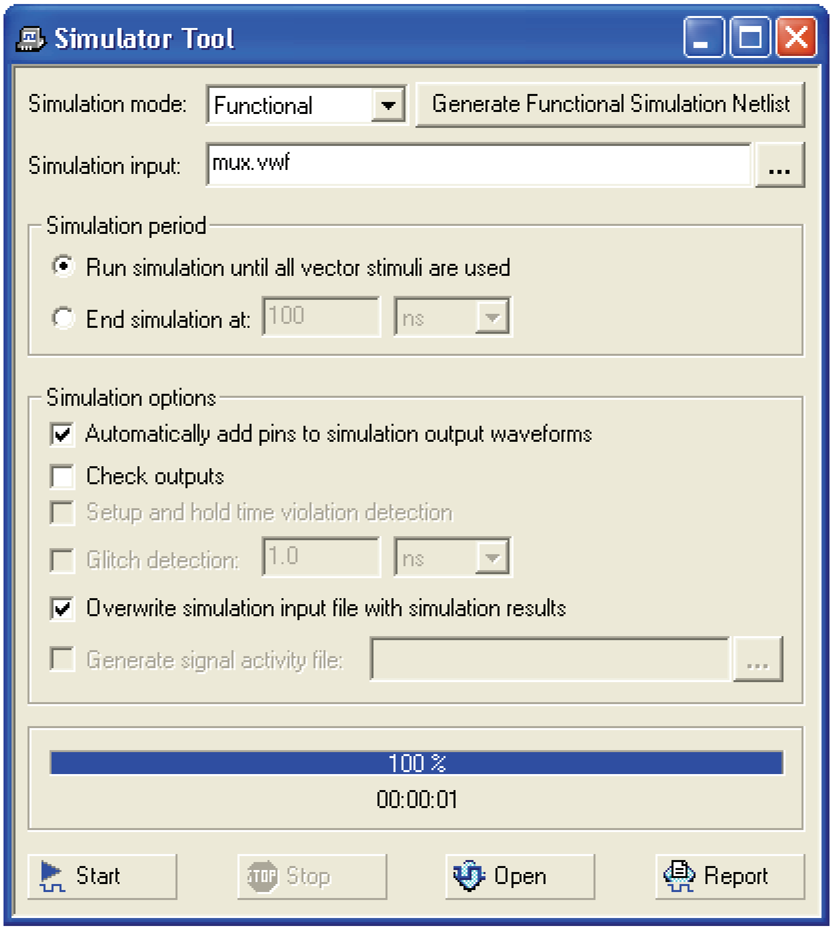


Figure A.15 Simulator Tool window.

A.5.7 Viewing the Simulation Results

1. To see the result of the simulation, you can either select the Vector Waveform window `mux.vwf`, or click on the **Open** button.
2. If a message window appears asking whether you want to reload the vector waveform file to update the changes made by the simulator. Click **Yes**.
3. The resulting waveform from the simulation is shown in Figure A.16. Notice that when `s` is a 0, the `y` output follows the `d0` input, and when `s` is a 1, the `y` output follows the `d1` input.
4. You can change the input signal values of s, d0, and d1 to something different. After making any changes to the simulation input file, you need to simulate the design again by clicking on the Start button . However, it is not necessary to re-generate the functional simulation netlist.

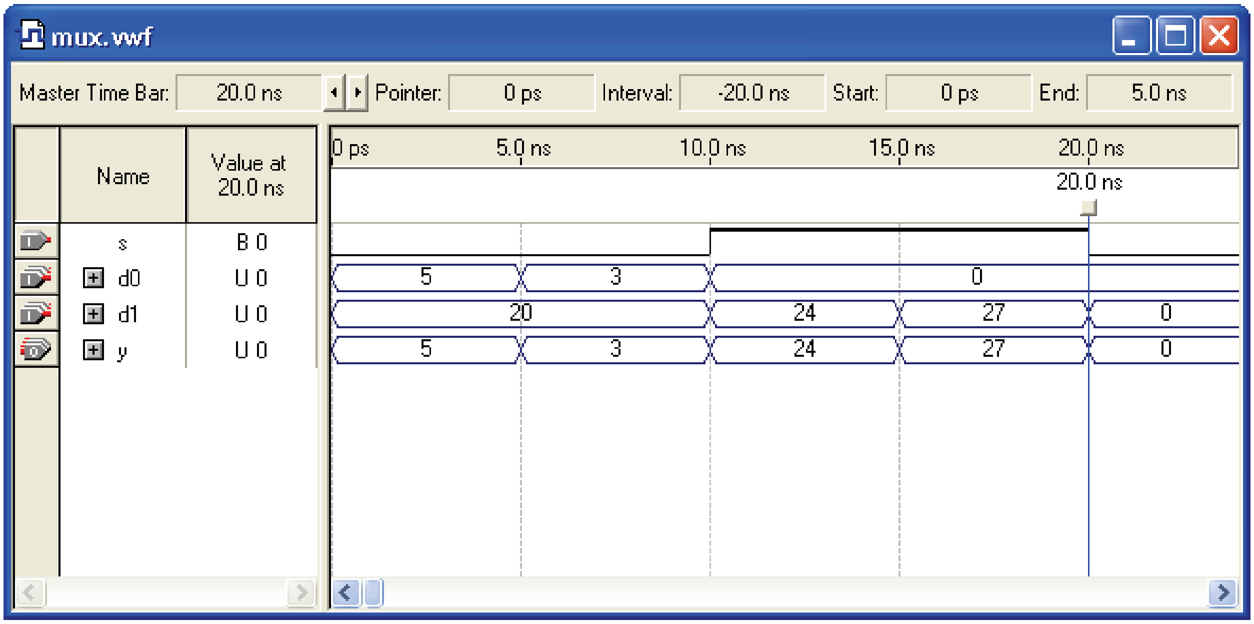


Figure A.16 Resulting waveform after the simulation.

A.6 Creating and Using a Logic Symbol

If you want to use this circuit as part of another circuit in a schematic drawing, you can create a logic symbol for this circuit. However, this is not necessary if you use this circuit in another Verilog or VHDL design file.

1. To create a logic symbol, first select the Graphic Editor window `mux.bdf` as the active window.
2. Select **File | Create/Update | Create Symbol Files for Current File**. The name of this symbol file will be the same as the name of the current active circuit diagram (`mux`) in the Graphic Editor, but with the file extension `.bsf` (for block symbol file).
3. You can view and edit the logic symbol by first opening the file. Select **File | Open** and type in the filename `mux.bsf`. Click on the **Open** button. A window similar to Figure A.17 will open showing the logic symbol.

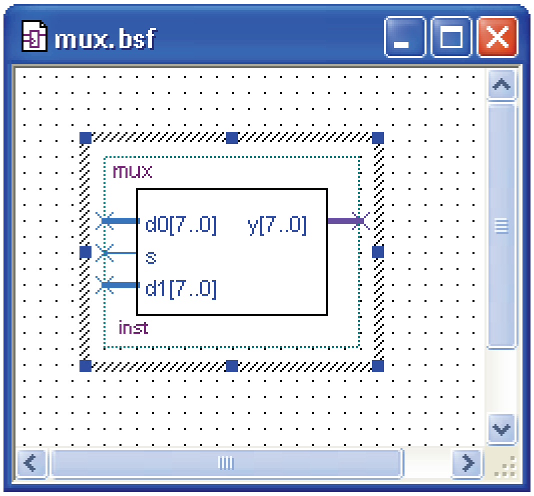


Figure A.17 Logic symbol of the mux circuit.

4. The placements of the input and output signals can be moved to different locations by dragging the signal

connection line around the symbol box. The signal label will also be moved. You can then drag the label to another location if you wish.

5. The size of the symbol can also be changed by dragging the edges of the symbol box.
6. This new symbol name can now be used in the Graphic Editor. It will show up in the Symbol window under the `Project` folder as shown in Figure A.18. You can follow the same steps as discussed in Section A.2.3 to insert this logic symbol into another schematic circuit design.
7. To use the mux circuit that is represented by this logic symbol in another project, you need to first copy the `mux.bsf` symbol file and the corresponding `mux.bdf` circuit design file to the other project's directory.

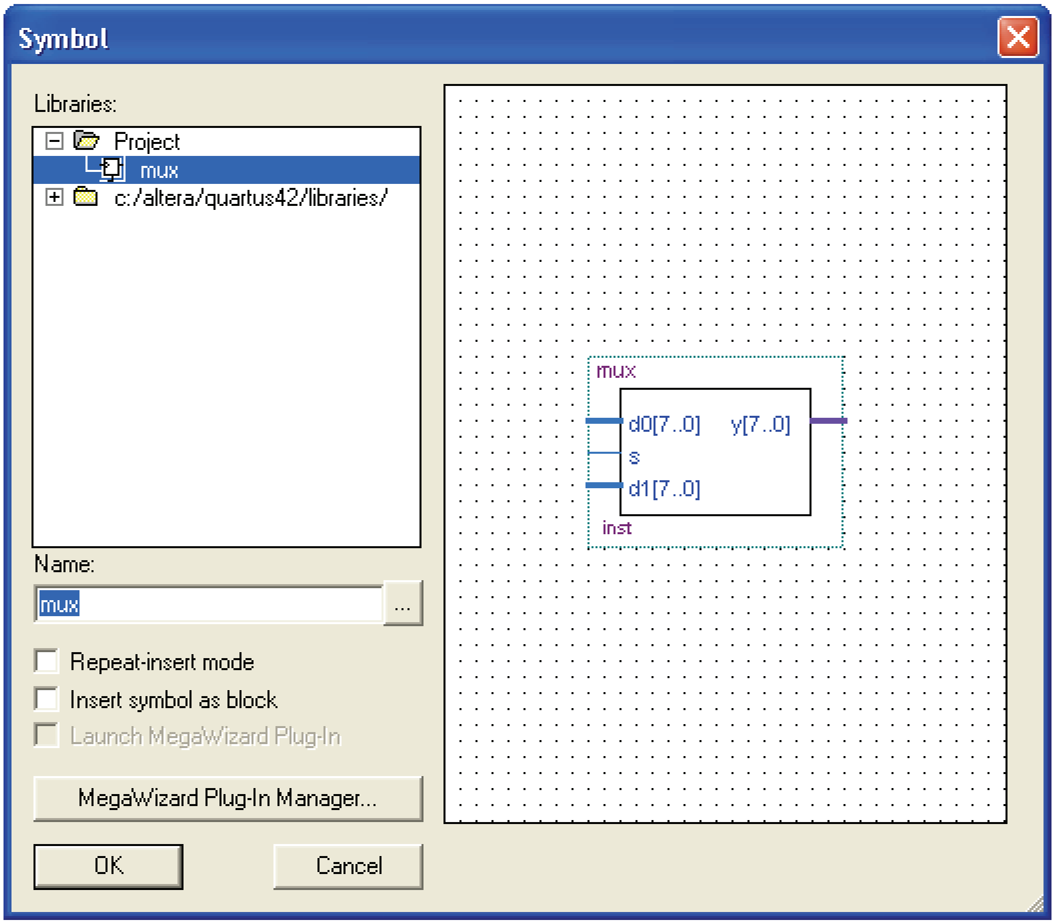


Figure A.18 Selecting the mux logic symbol to be inserted into another circuit design.