Contents

Appendix A Schematic Entry—Tutorial 1

The MAX+plus 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 MAX+plus 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.

A Student Edition version of the MAX+plus 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 Windows environment, and that the MAX+plus II software has already been installed on your computer. Instructions for the installation of the MAX+plus II software can be found on the CD-ROM. You also must obtain a license file from the Altera website in order for the software to function correctly. Be careful that you obtain the Student Edition license, and not the Baseline Edition license.

The MAX+plus 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.

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 executed in the hardware.

A.1 Getting Started

A.1.1 Preparing a Folder for the Project

Each circuit design in MAX+plus II is called a project. Each project should be placed in its own folder, since the synthesizer creates many associated working files for a project. Using Windows File Manager, create a new folder for your new project.

- For this tutorial, create a folder called `2x8mux` in the root directory of the C drive.

A.1.2 Starting MAX+plus II

After the successful installation of the MAX+plus II software, there should be a link for the program under the Start button. Click on this link to start the program. You should see the MAX+plus II Manager window, as shown in Figure A.1.

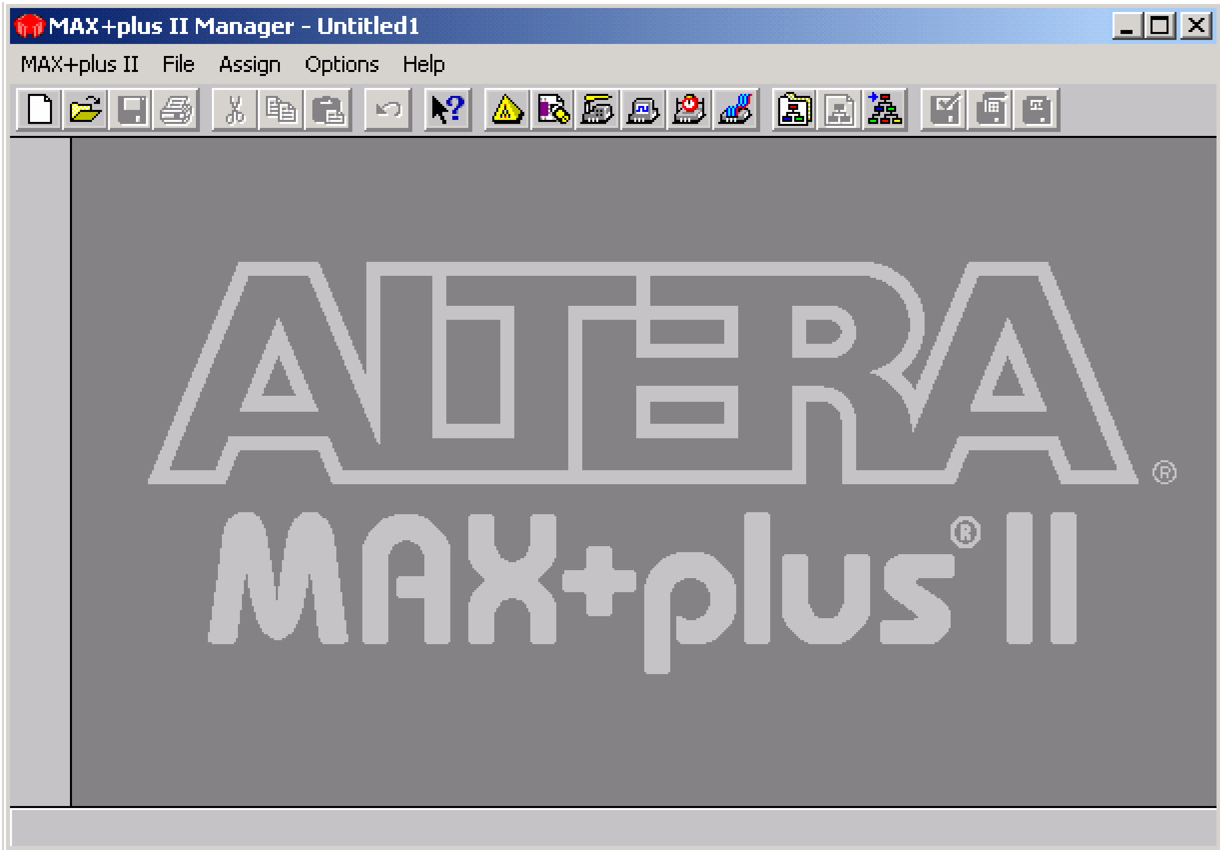


Figure A.1 The MAX+plus II Manager window.

Figure A.2 shows the toolbar for accessing the main development tools. The buttons from left to right are:

- *Hierarchy display*—to show the design files used in the current project
- *Floorplan Editor*—to map the I/O signals from the circuit to the pins on the PLD chip
- *Compiler (synthesizer)*—to synthesize the circuit to its netlist
- *Simulator*—to perform circuit simulation
- *Timing analyzer*—to perform circuit timing analysis
- *Programmer*—to program the circuit to the PLD chip
- *Open existing or new project*—to select a new project
- *Change project name to current filename*—to use the current file as the new project
- *Open top-level design file*—to open the top-level design file for the current project

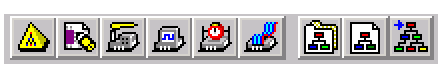


Figure A.2 The MAX+plus II development software toolbar.

In the MAX+plus II software, different commands in the menus are available when different windows are activated. This might cause some confusion at first. If you cannot find a particular command from the menu, make sure that the correct window for that command is the active window.

A.1.3 Starting the Graphic Editor

From the Manager window, select Max+plus II | Graphic Editor. You should see the Graphic Editor window similar to the one shown in Figure A.3. Any circuit diagram can be drawn in this Graphic Editor window.

- Alternatively, you can select File | New from the Manager window menu. Select Graphic Editor file using the extension `.gdf`, and click OK.

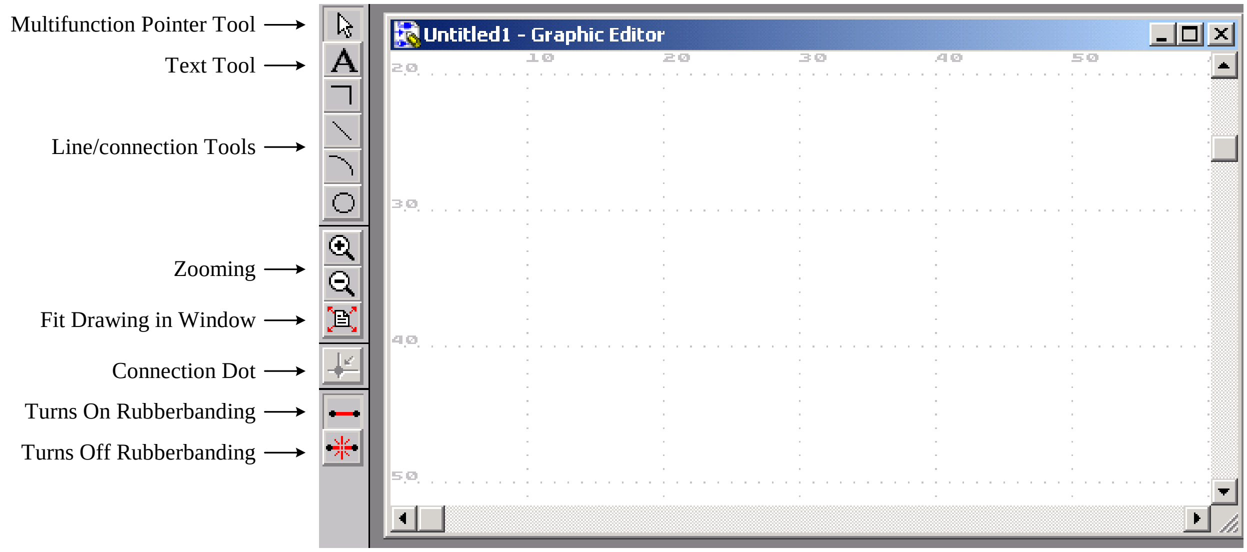


Figure A.3 The Graphic Editor window with the graphics toolbar on the left.

A.2 Using the Graphic Editor

A.2.1 Drawing Tools

In Figure A.3, the tools for drawing a circuit in the Graphic Editor are shown in the toolbar on the left side. There are the standard text writing tool, line drawing tools for making connections between logic symbols, and zooming tools. The main tool that you will use is the multifunction pointer tool. This multifunction pointer allows you to perform many different operations depending on the context in which it is used. Two main operations performed by this multifunction pointer are selecting objects and making connections between logic symbols. The connection dot tool either makes or deletes a connection point between two crossing lines. Finally, the two rubberbanding buttons turn 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.2.2 Inserting Logic Symbols

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

Available symbol libraries are listed in the Symbol Libraries selection 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 primitive logic gates, latches, flip-flops, and input and output connectors that we need are in the primitive library: …`\prim`. Your own circuits that you want to reuse in building larger circuits are in the directory that they are stored in.

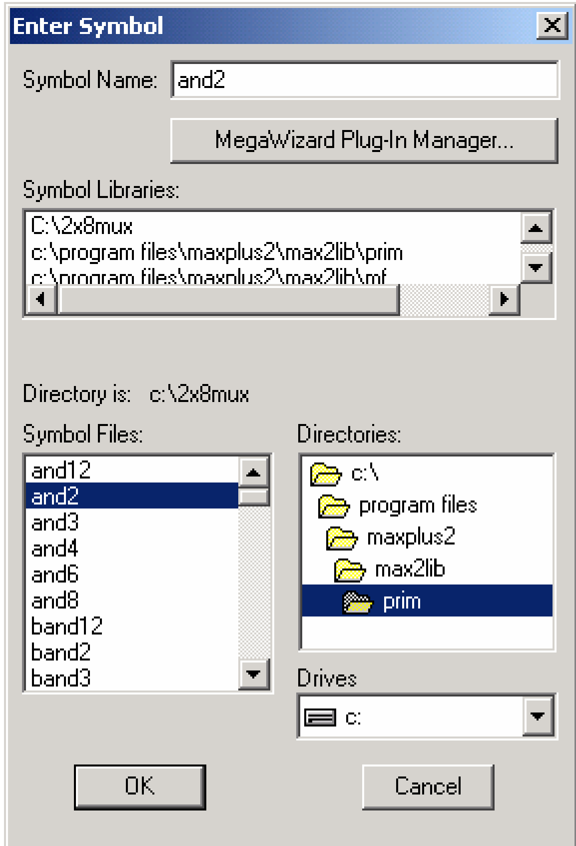


Figure A.4 The Enter Symbol selector window.

2. Double-click on the **prim** library to see a list of logic symbols available in that library. A list of logic symbols is shown in the Symbol Files selection box. The logic symbols are sorted in alphabetical order. To narrow down the list, you can type the first few letters of the symbol name followed by an asterisk in the Symbol Name text box. You need to either press Enter or click on the OK button to update the list. For example, typing `a*`, and pressing Enter will produce a list of all of the symbols whose name starts with the letter "a."

3. 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.

   For this tutorial, insert the following symbols:

   - A 2-input AND gate (and2)
   - A 2-input OR gate (or2)
   - A NOT gate (not)
   - An input signal connector (input)
   - An output signal connector (output)

A unique number is given 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. These numbers may be different from those in the examples.

A.2.3 Selecting, Moving, Copying, and Deleting Logic Symbols

- To select a logic symbol in the Graphic Editor, simply single-click on the symbol using the multifunction pointer tool. You can also select multiple symbols by holding down the Shift key while you select the symbols. An alternative method is to trace a rectangle with the multifunction tool around the objects that you want to select. All objects inside the rectangle will be selected.

- 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 from the drop-down 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 to look like Figure A.5

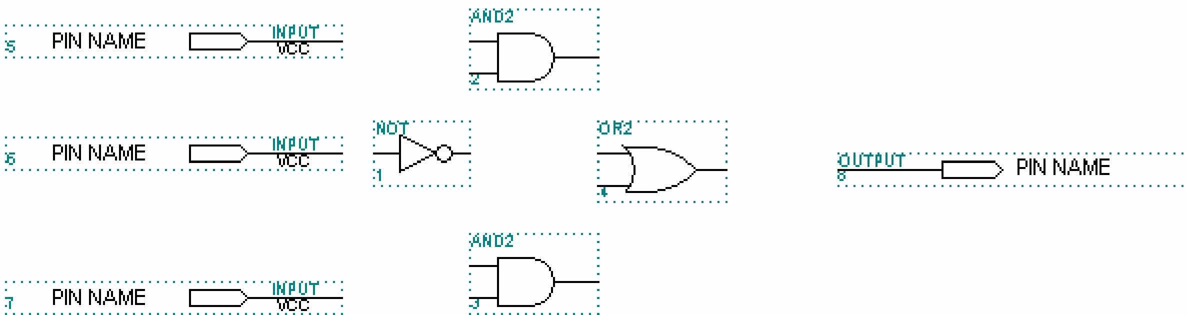


Figure A.5 Symbol placements for the 2-to-1 multiplexer circuit.

A.2.4 Making and Naming Connections

- To make a connection between two connection points, use the multifunction pointer tool and drag from one connection point to the second connection point. Notice that, when you position the multifunction pointer tool 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 other connection point.
- If you want to make a connection line that does not start from a connection point, you will need to use either the straight line drawing tool or the vertical and horizontal line drawing tool instead of the multifunction pointer tool.
- 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 multifunction pointer tool to select the junction point (i.e., the point where the two lines cross) and then press the connection tool button, as

shown in Figure A.6. The connection tool button is enabled only after you have selected a junction point.

- To remove a connection point, select (single-click) the point, and then press the connection tool button.

Figure A.6 Making or deleting a connection point with the connection tool button.

- To make a bus connection (for grouping two or more lines together), first draw a regular connection line, then right-click on the line, select Line Style, and select the thicker, solid line style from the pop-up menu.

- A bus must also have a name and a width associated with it. Select the bus line at the point where you want to place the name 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.7.

- 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.7.

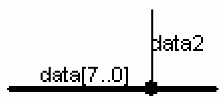


Figure A.7 A single connection line connected to an 8-bit bus with the name **data**.

- 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 double-clicking it, and then 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.

- A bus line connected to an input or output connector must have the same name as the connector.

  Perform the following operations for this tutorial:

  1. Name the three input connectors `d0[7..0]`, `s`, and `d1[7..0]`, as shown in Figure A.8
  2. Name the output connector `y[7..0]`, as shown in Figure A.8
  3. Connect and name the five bus lines `d0[7..0]`, `d1[7..0]`, `and0[7..0]`, `and1[7..0]`, and `y[7..0]`, as shown in Figure A.8
  4. Connect the single lines from the input connector `s` to the inverter and to the two AND gates, as shown in Figure A.8

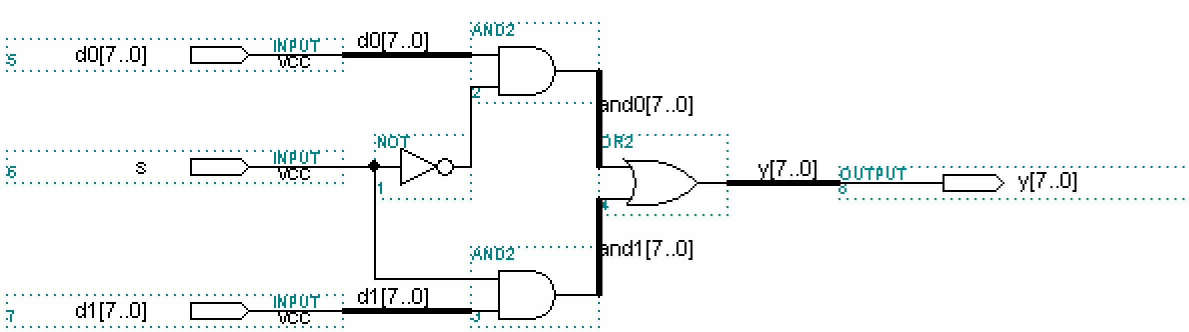


Figure A.8 Connections and names for the 2-to-1 multiplexer circuit.

A.2.5 Selecting, Moving and Deleting Connection Lines

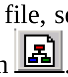
- To select a straight connection line, just single-click on it.
- To select an entire connection line with horizontal and vertical segments, double-click on it.
- To select a portion of a line, trace a rectangle with the multifunction tool around the segment.
- After a line is selected, it can be moved by dragging.
- After a line is selected, it can be deleted by pressing the Delete key.

A.3 Specifying the Top-Level File and Project

A.3.1 Saving the Schematic Drawing

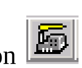
1. From the Graphic Editor menu, select File | Save. Select the `2x8mux` directory that you created on the C drive in Section A.1.1. Type in the filename `2x8mux`. The extension should be `.gdf` (for graphic design file).
2. Click OK.

A.3.2 Specifying the Project

1. To use the schematic drawing file saved in Section A.3.1 as the top-level project file, select File | Project | Set Project to Current File from the Manager window menu, or simply click on the icon .

- You can open any graphic design file (with the extension `.gdf`) using the Manager menu command File | Open, and then select File | Project | Set Project to Current File, or click on the icon  to make that particular file the top-level project file.

A.4 Synthesis for Functional Simulation

1. From the Manager window menu, select MAX+plus II | Compiler, or click on the icon  to bring up the Compiler window.
2. From the Compiler window menu (that is, with the Compiler window selected as the active window), select Processing | Functional SNF Extractor so that a check mark appears next to it. To actually see whether there is a check mark or not, you need to select the Processing menu again. The Compiler window for functional

extraction is shown in Figure A.9.

3. Click on the Start button to start the synthesis. You will then see the progress of the synthesis.
4. At the end of the synthesis, if there are no syntax errors, you will see a message window saying that the compilation was successful. Click OK to close the message window.
5. If there are errors, go back to Section A.2.4 and double check your circuit with the one in Figure A.8.

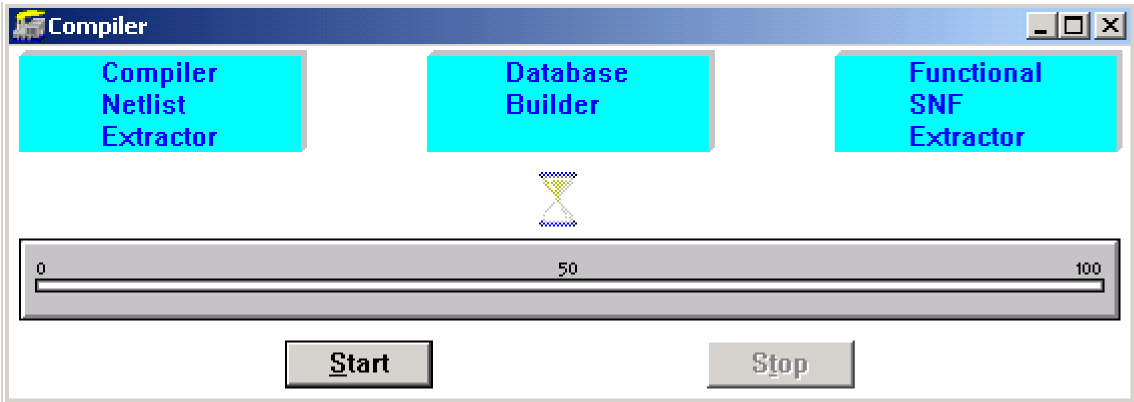


Figure A.9 Compiler window for functional extraction.

A.5 Circuit Simulation

A.5.1 Selecting Input Test Signals

1. Before you can simulate the design, you need to create test vectors for specifying what the input values are. From the Manager window menu, select MAX+plus II | Waveform Editor.
2. From the Waveform Editor window menu, select Node | Enter Nodes from SNF. You can also right-click under the Name section in the Waveform Editor window and select Enter Nodes from SNF from the pop-up menu. You will see something similar to the Enter Nodes from SNF window shown in Figure A.10.

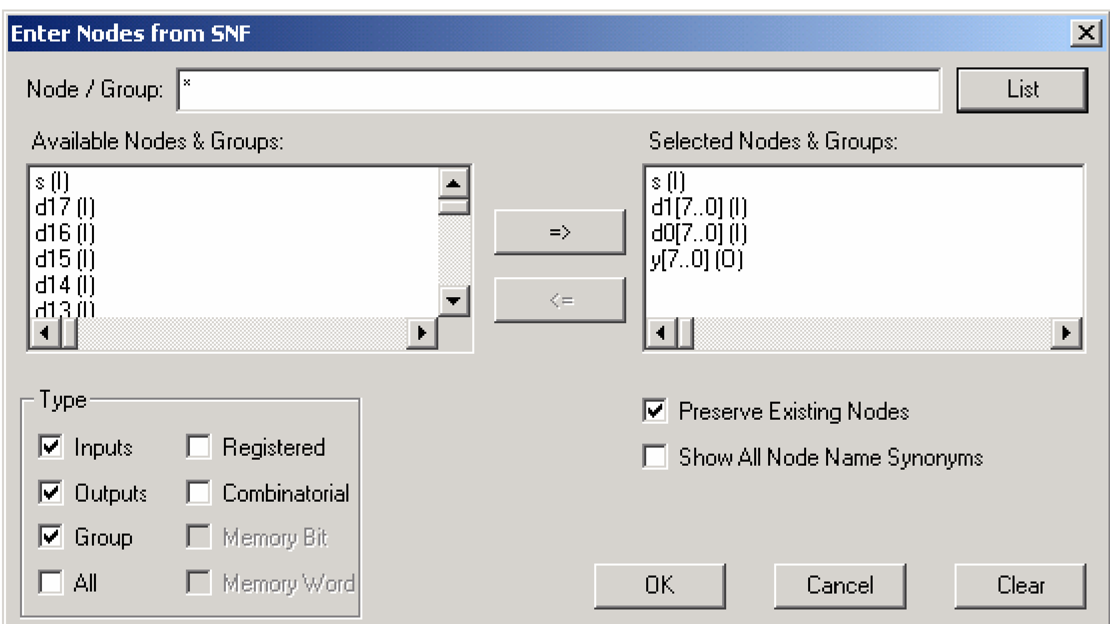


Figure A.10 Window for adding signals for simulation.

3. Click on the List button in the Enter Nodes from SNF window, and a list of available nodes and groups will be displayed in the Available Nodes & Groups box.
4. Select the signals that you want to see in the simulation trace. The signals that we want are: **s (I)**, **d1[7..0] (I)**, **d0[7..0] (I)**, and ***y*[7..0] (O)**. The letters **I** and **O** in parenthesis next to each signal denote whether the signal is an input or output signal, respectively. Note that the signal name such as **y7** is bit seven of the bus named **y**, and **d16** is bit six of the bus named **d1**. Multiple nodes can be selected by holding down the Ctrl or Shift key while clicking on the signal names.
5. After selecting the signals, click on the => button to move the selected signals to the Selected Nodes & Groups box.
6. Repeat Steps 4 and 5 until all of the signals that you want to see in the simulation are moved to the Selected Nodes & Groups box.
7. Click on OK when you are finished. The selected signals will now be inserted in the Waveform Editor window similar to Figure A.11.

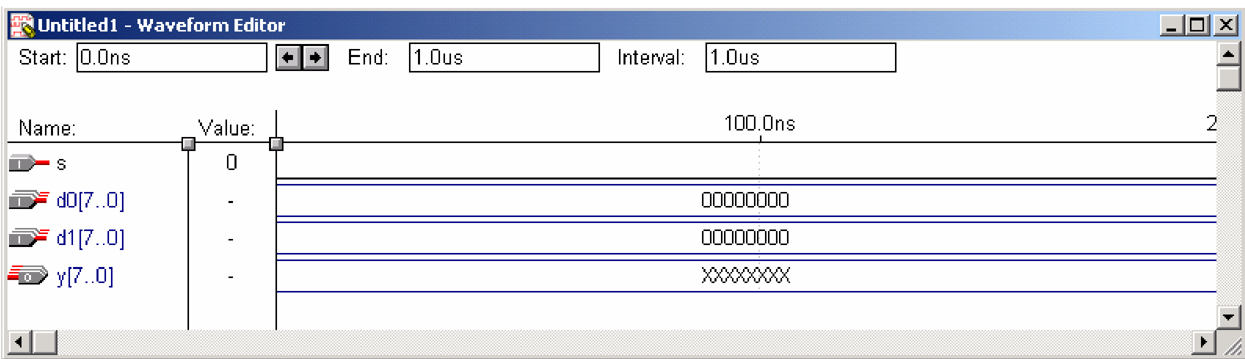


Figure A.11 Waveform Editor window for simulation.

A.5.2 Customizing the Waveform Editor

- You can rearrange the signals in the Waveform Editor by dragging the signal icons like  up or down.
- To delete a signal in the Waveform Editor, just select the signal by clicking on its name and press the Delete key.
- For signals that are composed of a group of bits (such as the data input `d1`), you can separate them into individual bits or change the radix for the displayed value by first selecting that signal and then right-click the mouse. A drop-down menu appears. Select Ungroup to separate the bits. To regroup them, select the bits you want to group and then right-click the mouse. A drop-down menu appears. Select Enter Group. Type in a group name, and select the Decimal radix for the display.

1. We want to simulate for 500 ns. To change the simulation end time, select File | End Time from the Waveform Editor window menu.
2. In the End Time window, type in `500ns`, and click OK.
3. To fit the entire simulation time range inside the window, select View | Fit in Window from the Waveform Editor window menu, or click on the icon  in the toolbar on the left. Your Waveform Editor window should now look like the one in Figure A.12.

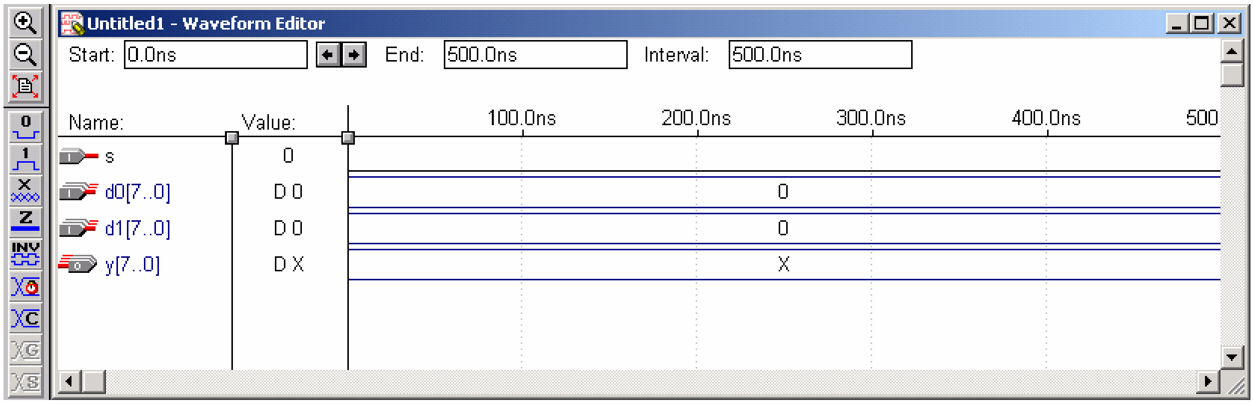
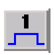


Figure A.12 Waveform Editor window after changing the value radix and fitting the entire time range inside the window. Notice also the toolbar buttons on the left.

A.5.3 Assigning Values to the Input Signals

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

1. Using the multifunction pointer tool, drag from time `200ns` to `400ns` for the `s` signal only, as shown in Figure A.13.
2. Click on the icon  in the toolbar on the left to set the signal in the selected range to a logical 1 value.
3. Drag from time `0ns` to `100ns` for the `d0` signal only, as shown in Figure A.13.
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.
5. Repeat Steps 3 and 4 for the remaining input values for `d0` and `d1`, as shown in Figure A.13.

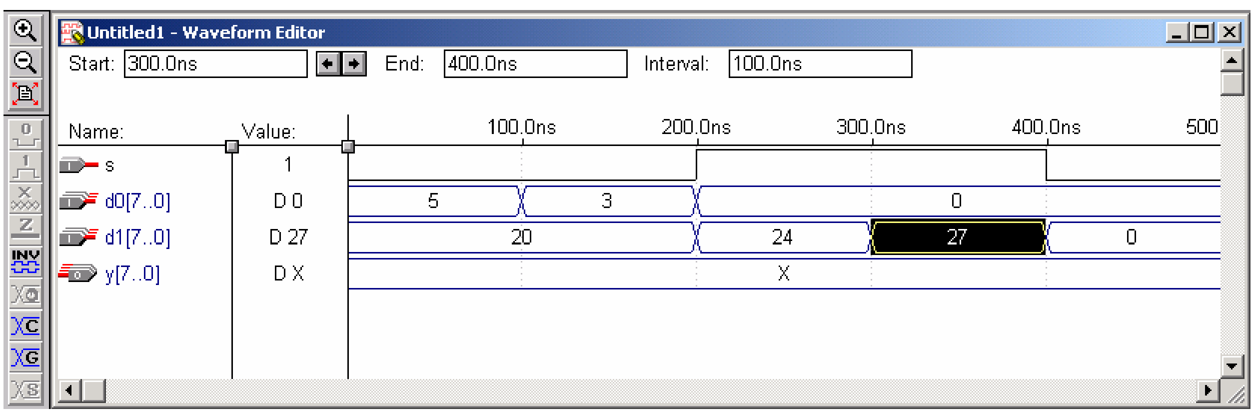


Figure A.13 Changing the input signal values.

A.5.4 Saving the Waveform File

1. Save the Waveform Editor window by selecting File | Save. The Save As window appears. Notice that the default file name is the same as the top-level entity name, and the extension is `.scf`. For this example, the name is `2x8mux.scf`.
2. Click on OK to save the file.

A.5.5 Starting the Simulator

1. We are now ready to simulate the design. From the Manager window menu, select MAX+plus II | Simulator, or click on the icon to bring up the Simulator window.

- You can also save the waveform file and start the simulator in one step by selecting from the Manager window menu File | Project | Save & Simulate.

2. The Simulator window as shown in Figure A.14 is displayed. Make sure that the Simulation Input filename is `2x8mux.scf`. This is the same name as your top-level entity.

3. Click on the Start button and watch the progress of the simulation.

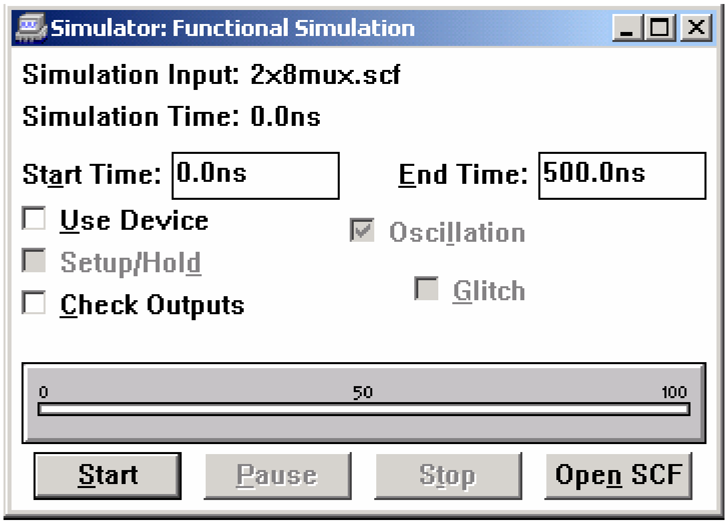


Figure A.14 Simulator window for the multiplexer design.

4. At the end of the simulation, if there are no errors, you will see a message window saying that the simulation was successful. Click OK to close the message window.

5. Click on the Open SCF button in the Simulator window to bring up the Waveform Editor with the resulting simulation waveforms. The simulation result is shown in Figure A.15. The signal `y` is the multiplexer output.

   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.

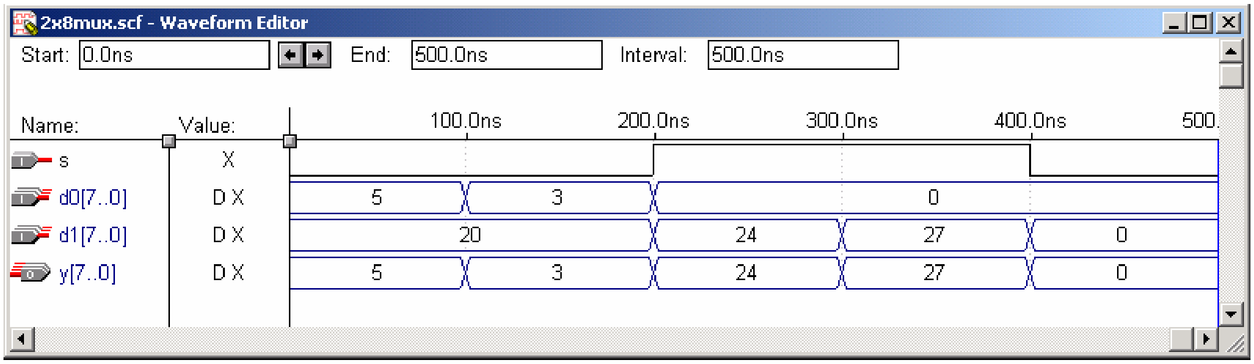


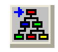
Figure A.15 Resulting waveform after the simulation.

6. You can change the input signal values of `s`, `d0`, and `d1` to something different and run the simulation again.

A.6 Creating and Using the Logic Symbol

If you want to use this circuit as part of another circuit, you need to create a logic symbol for this circuit.

1. To create a logic symbol for the current active circuit diagram that is in the Graphic Editor window, from the Graphic Editor menu, select File | Create Default Symbol. The name of this logic symbol is the same as the name of the current active circuit diagram in the Graphic Editor, but with the extension `.sym`.

   - If your Graphic Editor window is closed, you can click on the icon  to open the top project circuit diagram again.

2. You can view and edit the logic symbol by selecting File | Edit Symbol. The placements of the input and output signals can be moved to different locations by dragging them. The size of the symbol can also be changed by dragging the edges of the symbol rectangle.
3. To use this circuit in another project, you need to copy this `.sym` file and the corresponding `.gdf` circuit file to the other project's directory. This new symbol name will now show up in the Enter Symbol window like the one shown in Figure A.4.