Introduction to MAX+PLUS II

1 Overview

This document is intended as an introduction to the major features of Altera's MAX+PLUS II design tools. A tutorial for MAX+PLUS II was developed for EEC 180A and is also available on-line. Altera's MAX+PLUS II consists of a number of applications for designing and analyzing digital logic circuits, and for programming these circuits onto Altera's PLD (Programmable Logic Device) boards. These applications include:

- a **graphic editor** for capturing and editing your schematics,
- a **symbol editor** which allows you to create a single component for your circuit, allowing hierarchical designs,
- a **waveform editor** which allows you to specify and view node values during simulation,
- a **compiler** which compiles your design and creates the appropriate files required for simulation,
- a **simulator** for verifying the functionality of your circuit,
- a **timing analyzer** for determining the path delays and the maximum clock speed for your circuit,
- and a **programmer** for downloading your circuit onto an Altera PLD.

There are a few other applications in MAX+PLUS II, such as the text editor which can be used for entering and editing hardware description language (HDL) design files, and the floorplan editor which can be used to specify the placement of input/output pins and logic blocks on the physical programmable logic device.

2 MAX+PLUS II Basics

2.1 Starting MAX+PLUS II

MAX+PLUS II software has been installed on the CSIF HP machines in the basement of Engineering II. To start up MAX+PLUS II on an HP machine, type the command:

```
maxplus2 &
```

After starting up MAX+PLUS II, the MAX+PLUS II Manager window will show up. Most of the commands in MAX+PLUS II can be accessed through the menus in this window. Short-cut buttons on the tool bar (below the menu bar) can also be used to access certain commands. To see what a button on the tool bar does, move the mouse pointer over that button and read the description provided at the bottom of the Manager window. Some menu commands also have keyboard short-cuts which are listed in the menus.

2.2 Starting Various Applications

Individual applications may be started by 1) using the MAX+PLUS II menu in the Manager window, 2) creating a new file of the appropriate type, or 3) opening a file of the appropriate type. For example, to start the Graphic Editor application, we could use one of the following menu commands:

> **MAX+plus II − > Graphic Editor**

or

> **File − > New**

If we have already created a Graphic Editor file, we can use:

> **File − > Open** (select .gdf file)

The New and Open commands may also be accessed through the two leftmost buttons on the tool bar.

Each time you start up one of the applications (Graphic Editor, Waveform Editor, etc.) another application window will appear within the Manager window. Only one application window will be active at any time. To activate a different window, click the left mouse button anywhere within the new window. The menus and buttons in the Manager window will change depending on which application is currently active, therefore some menu and button commands will only be available for certain applications.

3 Creating a Schematic

This section explains how to create a schematic using the Graphic Editor application. It is a good idea to design your circuit on paper first before attempting to enter the schematic in the MAX+PLUS II Graphic Editor.

3.1 Starting the Graphic Editor

To start the graphics editor with a new file, use one of the following menu commands:

> **MAX+plus II − > Graphic Editor**

or

> **File − > New** (select .gdf file)

Alternatively, you may click on the far-left icon on the tool bar (select .gdf file).

3.2 Placing Components

To place components in your schematic, you may either double-click the left mouse button in the Graphic Editor window, or use the corresponding menu command:

Symbol $->$ **Enter Symbol**

When the Enter Symbol command is executed, a window containing the available libraries and symbols will show up. If you know the name of the symbol you may type the name in the Symbol Name box at the top of the window. Otherwise, you will need to browse through the various libraries until you find the symbol you want. To view the components available within a library, double click the left mouse button on the appropriate directory in the Symbol box. The symbols contained in that directory will show up in the Symbol Files box. Select the desired component by double clicking the left mouse button on its name in the Symbol Files box. The selected symbol will appear in the Graphic Editor window.

3.2.1 MAX+PLUS II Symbol Libraries

In addition to symbols you have created for your own components, MAX+PLUS II also provides libraries containing a number of standard components. These libraries include:

- **prim**: Contains primitives, such as basic gates (and2, or2, not), latches, and flip-flops.
- **mf**: Contains macrofunction components such as multiplexers, decoders, adders, multipliers, and stacked latches and flip-flops.
- **mega_lpm**: Contains Altera's Library of Parameterized Modules (LPM) megafunction components, a number of high-level components which can be configured to meet specific design goals.

3.2.2 Input and Output Pins

In order to compile and simulate your circuit, all inputs to your circuit must be connected to input pins and all outputs from your circuit must be connected to output pins. These components can be found in the **prim** library and are named "input" and "output".

3.3 Connecting Components

After placing various components in your schematic, you will need to connect them with wires or buses. Move the cursor to the end of the first pin until the cursor changes to a "+". Click and hold the left mouse button, drag the cursor to the second pin, and release the mouse button. If your wire requires a number of bends, you may have to first draw a wire from one pin to an intermediate point, and then draw a separate wire from the intermediate point to the other pin.

Another option for connecting components is to use the Rubberbanding function. When Rubberbanding is on, two components may be connected by moving one component until its pin touches the pin of the second component. When the components are moved apart, a net is formed between the pins. Rubberbanding may be turned on using the menu option:

Options – > Rubberbanding

You may also connect various wires in your circuit by giving them the same label. For example, if two wires are both labeled "CLK", then these two wires will be considered as connected by the program.

3.4 Labeling Nodes and Buses

All nodes (i.e., wires) that you wish to view in the waveform editor must have a label. To label a wire, first select the wire by clicking the left mouse button on it so that it is highlighted, and then type in the label name. If the label is moved away from the wire, it may cease to be a label. To see if a label is valid, select the wire. If the label is still valid, it will have a highlighted box around it.

Buses are basically a lot of wires bunched together. The format for a bus label is BUSNAME[$i..j$], where i and j are integers. Some examples are DATA[0..7] or ADDRESS[31..0]. To label a bus, select the bus by clicking on it with the left mouse button, and type in the desired label.

A wire connected to the bus must have a label which matches the label of the bus. For example, a wire connected to the bus DATA[0..7] can be labelled DATA0, DATA1, DATA2, etc., but cannot be labelled D2.

3.5 Labeling Input and Output Pins

Input and optput pins must be labeled in order for the circuit to compile. Input and output pins can be labelled by double clicking the left mouse button on PIN_NAME and typing the desired name, or selecting the pin, clicking the right mouse button, selecting Edit Pin Name, and then typing the desired name. If you press return after editing a pin name, the next pin name below it is automatically selected for editing.

3.6 Editing Your Circuit

There are a number of commands for editing your circuit. Some of these are listed below.

3.6.1 Selecting Components

Individual components such as symbols or wires may be selected by clicking on them with the left mouse button. Multiple components may be selected by holding down the Shift key while clicking on the different components with the left mouse button. Another way to select multiple components is to click the left mouse button on an empty area in the Graphic Editor window, and holding the button down, drag the mouse such that the resulting box surrounds the desired components.

3.6.2 Move

To move a symbol, select it by clicking on it with the left mouse button, and keeping the left mouse button down, drag the symbol to its desired location.

3.6.3 Undo

The Undo command undoes the last thing you did. Only the most recent command is remembered, so you will not be able to undo something that you did two or more commands ago.

Edit $->$ Undo

3.6.4 Cut

The Cut command removes the selected component from the schematic and saves it to the clipboard. The component can be replaced by using the Paste command.

Edit $->$ Cut

3.6.5 Copy

The Copy command copies the selected component to the clipboard. Copies of the selected component can be added to the schematic using the Paste command. The Copy command is very useful and can save you a lot of time.

Edit $->$ Copy

3.6.6 Paste

The Paste command adds the last component saved to the clipboard to your schematic.

Edit $->$ Paste

3.6.7 Delete

To delete a symbol, first select it by clicking on it with the left mouse button, then hit the Del key, or use the menu command:

Edit $->$ Delete

3.6.8 Flipping

A component may be flipped either horizontally or vertically using one of the following commands:

Edit − > Flip Horizontal

Edit − > Flip Vertical

3.6.9 Rotation

You can rotate any component by either 90, 180, or 270 degrees by selecting it and using the menu command:

Edit − > Rotate − > 90, 180, 270

3.7 Creating Symbols

When you draw a circuit in the Graphic Editor, you are creating a *schematic*. The components you added using the Enter Symbol command are *symbols* for circuits which have already been defined. The idea of a symbol representing a circuit is an important one, as it allows for modular and hierarchical designs. In MAX+PLUS II, you can create symbols for your own circuits which you can then use in higher levels of your design. To create a symbol for your schematic, use the menu command:

File − > Create Default Symbol

This command creates a file ending with ".sym". To view the symbol that you have created, use the menu command:

File − > Edit Symbol

A Symbol Editor window containing the symbol will show up. The symbol will be a block with inputs and outputs corresponding to the input and output pins in your schematic.

Once you have created a symbol for you circuit, you may place the symbol in another schematic. When you use the Enter Symbol command, go to the directory containing your symbol, and double click on its name in the Symbol Files box. Do not attempt to place a symbol into its own schematic.

4 Simulating Your Design

After you have created a schematic and a symbol for each component in your design and a top level schematic which ties all of your components together, you will want to simulate your design to verify its functionality. In order to simulate your design, you first need to compile your design and then you need to specify the nodes that you want to view in the Waveform Editor.

4.1 Setting the Project

A *project* consists of all of the files in your design. The project name should be set to the name of the top level schematic of your design. To set the name of the project, use the menu command:

File $->$ Project $->$ Name

You should then enter the name of your top level schematic in the Project Name box, or double click on the name of your top level schematic in the Files box.

If your top level schematic is currently open in a Graphic Editor window, you can also set the project name by making this window active and using the menu command:

File $->$ Project $->$ Set Project to Current File

To view the hierarchy of your design, you can open the Hierarchy Display using the command:

MAX+plus II $->$ Hierarchy Display

The Hierarchy Display shows your design as a hierarchy tree with your top level schematic at the root of the tree and lower-level design files as branches of the tree. You may open any of the schematics in your design by double clicking the left mouse button on the appropriate node in the tree.

4.2 Checking and Compiling Your Design

Once you have completed entering your design and have set the name of the project, you may check your design for errors using the command:

File $->$ Project $->$ Save & Check

If your design has an error, highlight the error message using the left mouse button and press the Locate button to view the location of the error in your design file.

If your design has no errors or warnings, you may then proceed to compile your design. There are two different types of simulation that can be performed on the design: functional and timing. Each type of simulation requires a separate compilation. Functional simulation allows you to check for logical errors in your circuit. Timing analysis is used to determine the path delays in your circuit and helps to determine the maximum clock frequency at which sequential circuits can operate.

For a functional simulation, select the option:

Processing $->$ Functional SNF Extractor

If you want the nodes in your simulation to maintain the same names as the labels you used for the wires, you should also select the option:

Processing − > Preserve All Node Name Synonyms

To compile your design for the functional simulation, use the command:

File − > Project − > Save & Compile

You may also compile your design by pressing the Start button in the Compiler window. If your design compiles correctly, a Simulator Netlist File (.snf) containing all of the nodes in your circuit will be created.

To compile your design for a timing analysis, select the option:

Processing − > Timing SNF Extractor

And compile by pressing the Start button in the Compiler window or by using the command:

File − > Project − > Save & Compile

4.3 Waveform Editor

After compiling your design for a functional simulation, you will need to use the waveform editor to specify the inputs to your circuit as well as the nodes you wish to view in the simulation. Start the Waveform Editor using the command:

MAX+plus II − > Waveform Editor

4.3.1 Selecting Nodes

To add nodes to the Waveform Editor window, begin by using the command:

Node − > Enter Nodes from SNF

Another window titled "Enter Nodes from SNF" will appear. Each of the nodes in your design will be classified under one of several categories. These categories are listed in the Type box. To view all nodes of a given type, select the type by clicking the appropriate box, and then click on the List button in the upper right corner of the window. A list of nodes of the given type will appear in the Available Nodes box. Whenever you change the selection buttons, you must press the List button again to see the updated list of nodes. To observe a given node in the Waveform Editor window, highlight that node in the Available Nodes box by clicking on it. Multiple nodes may be selected by holding down the Shift key when selecting nodes, or by clicking and dragging across a number of nodes in the Available Nodes box. Once the nodes have been selected, press the => button to move the nodes to the Selected Nodes & Groups box. Nodes may be removed from the Selected Nodes & Groups box by highlighting them and clicking on either the Clear button or the <=button. When you are done adding nodes, click on the OK button.

Individual nodes may also be entered by double clicking the left mouse button in the Name column in the Waveform Editor window or by using the following command:

Node − > Insert Node

After adding nodes, you should see the selected nodes in the Waveform Editor window.

4.3.2 Grouping Nodes

A number of nodes may be combined into a single group that is displayed as a single waveform in the Waveform Editor window. Grouping is especially useful for bus signals. To group nodes, first highlight the nodes by clicking on them in the Name column of the Waveform Editor window. Multiple nodes may be selected by holding down the Shift key or by clicking and dragging across a group of nodes. Once the nodes are selected, use the command:

Node − > Enter Group

You will then need to enter a name for the group and choose the radix (binary, decimal, octal, or hexadecimal) for viewing the waveform. The naming convention for a group is similar to that of a bus, i.e., GROUPNAME[$i..j$] where i and j are integers.

4.3.3 Setting Inputs

You now need to set the inputs for your simulation. An input node can be set high or low for the entire duration of the simulation by selecting the node in the Waveform Editor window and using one of the following commands:

Edit − > Overwrite − > High
Edit − > Overwrite − > Low

You may set an input high or low for a selected interval of a waveform by clicking and dragging the cursor across the desired range on the waveform in the Waveform Editor window, and using one of the Overwrite commands.

To set clock signals, select the clock node and use the command:

Edit − > Overwrite − > Clock

A group's value may be set by using either the Overwrite Count Value or Overwrite Group Value commands:

Edit − > Overwrite − > Count Value
Edit − > Overwrite − > Group Value

Overwrite Count Value increments the value of the group by n after every $m \cdot 100.0$ns interval, where n and m are integers specified in the Overwrite Count Value window. The starting value for the group should also be specified in this window.

Overwrite Group Value sets the value of the group to the desired value over any selected interval.

The command:

Edit − > Overwrite − > Invert

changes the value of the selected node over the selected interval from high to low or from low to high. If used on a group, the command inverts the value for each of the nodes in the group.

4.4 Running the Simulation

To run the simulation, use the command:

File − > Project − > Save & Simulate

If there are no errors, the output waveforms in the Waveform Editor will be updated to reflect the simulation results.

5 Timing Analysis

To run timing analysis, the design should be compiled with the Timing SNF Extractor option (see Section 4.2). The Timing Analyzer can be started with the command:

MAX+plus II − > Timing Analyzer

When the Timing Analyzer window shows up, select the option:

Analysis − > Registered Performance

In the Timing Analyzer window, press the Start button. After the analysis is complete, you may select one of the clock signals in the Clock box to see the maximum frequency and minimum period for that clock.