Schematic Editor Lab

Schematic Editor Lab

Introduction

This lab will enable the user to begin creating the WATCH project. It will also provide the user with experience creating the CNT60 and OUTS macrofunctions. This lab should be completed after the Schematic Editor chapter of the course material has been covered. The CNT60, OUTS, OUTS1, and OUTS2 schematics are located in the Appendix.

Objectives

In this laboratory, it will be shown:

- How to start a project in Foundation
- How to enter primitives in the Foundation Schematic Editor
- How to connect wires and busses to symbols in the Schematic Editor
- How to create macrofunctions in the Schematic Editor
- How to create the CNT60 and OUTS macrofunctions for the WATCH Project

Procedure

Creating the WATCH project in the Foundation Project Manager

1) Open the "**Foundation Project Manager**" with the menu command: ***Start>Programs>Xilinx Foundation Series>Xilinx Foundation Project Manager***.

2) Once Foundation has started, click on the menu command: ***File>New Project***. This action will open the "**New Project**" dialog box, and enable the user to specify a directory for the project to be placed. This window also allows for specification of the chosen family, part, and speed grade.

3) In the "**New Project**" dialog box (see Figure 1), enter the project name "**WATCH**", specify the C:\F1.5labs\ directory, and select the XC4000E family and the XC4003EPC84-3 part. After this information is entered, click on "**OK**".

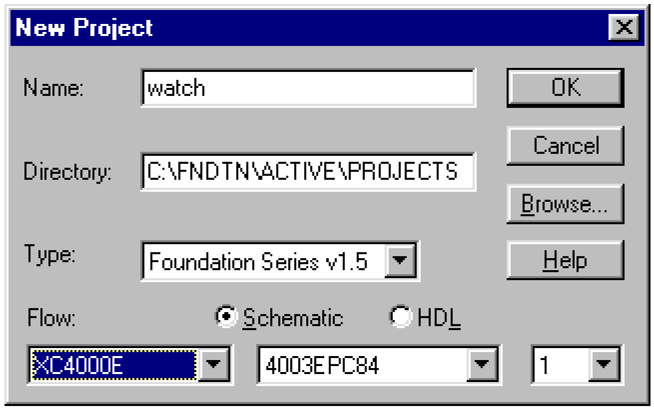


Figure 1. The New Project dialog box.

4) The **WATCH** project has now been created, and the "**Foundation Project Manager**" should look like Figure 2.

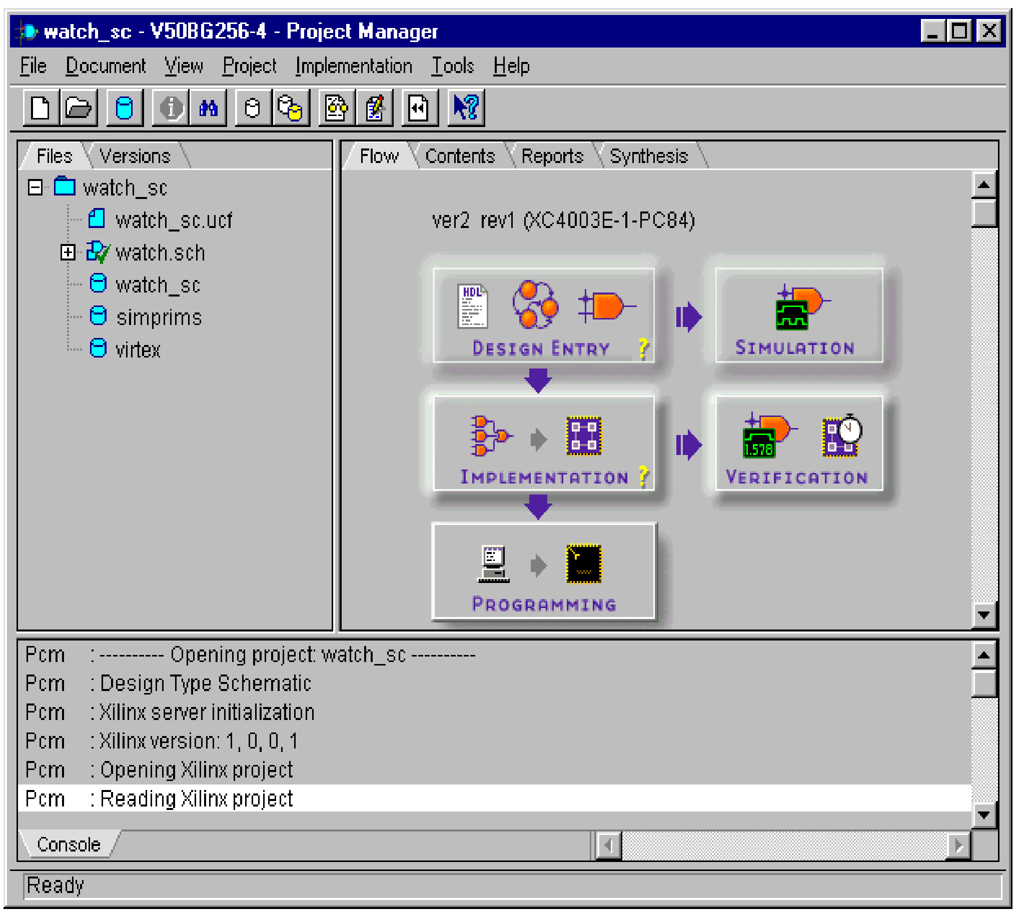
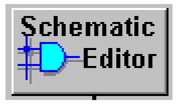
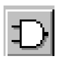


Figure 2. The Foundation Project Manager.

Creating the CNT60 macrofunction in the Schematic Editor

1) Start creating the CNT60 macrofunction (refer to Appendix) by opening the "**Schematic Editor**" by clicking on the menu command ***Tools>Schematic Editor*** or by clicking on its icon.



2) Once the program has started, click on the "**Symbols Toolbox**" icon on the vertical toolbar.  This will open the "**SC Symbols**" box (see Figure 3), which contains the available library elements for the chosen device family.

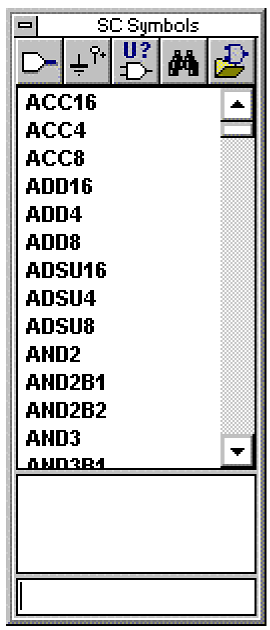
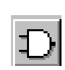
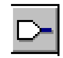


Figure 3. The Symbols Toolbox.

3) To locate the following symbols: **CD4CE**, **CB4CE**, **AND2**, **AND4**, **INV**, and **OR2**, Scroll through the list, or enter the symbol name at the bottom of the "**Symbols Toolbox**". After finding each symbol, move the pointer into the work area and place them by clicking the left mouse button.

4) Duplicate a component in the project by clicking the left mouse button while the pointer is on a placed symbol. Note that the "**Symbols Toolbox**" icon  must still be depressed on the vertical toolbar.

5) To leave any mode, press the "**ESC**" key. This will automatically place the user in **"Select and Drag**" mode.

6) Click on the "**I/O Terminal**" icon on the vertical toolbar.  This will cause the "**I/O Terminal**" dialog box to open (see Figure 4).

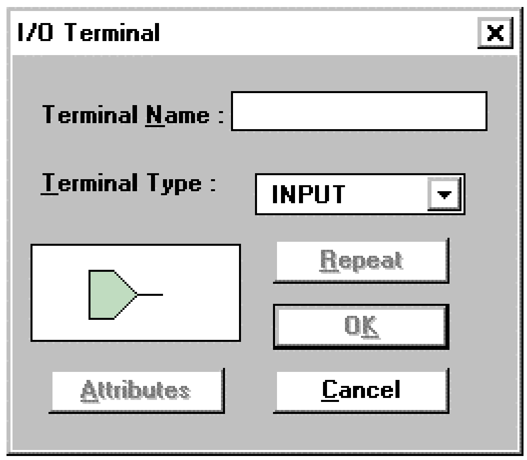


Figure 4. The I/O Terminal dialog box.

7) Enter the **CE** input terminal name, and select a terminal type of "**INPUT**". After clicking on "**OK**", the terminal will be attached to the pointer for placement. Place the terminal as shown in the CNT60 schematic. After placing the input terminal, the mode will automatically be switched to "**Draw Wires**" and the user can connect the terminals to the circuit by clicking once at each end of the wire. In this same manner, add the **CLK** and **CLR** input terminals to the project.

8) To make the **LSBSEC** and **MSBSEC** busses, click on the "**Draw Buses**" icon on the vertical toolbar.  Locate the pointer, and click on the left mouse button to start drawing. After the appropriate bend has been made in the bus, double-click the left mouse button to end the bus and open the "**Add Bus Terminal/Label**" dialog box (see Figure 5). Enter the **MSBSEC** and **LSBSEC** bus names, adjust their bus widths, and select "**OUTPUT**" as the I/O Marker type. When each the entry is complete, click on "**OK**". This will automatically append the bus I/O terminal to the bus.

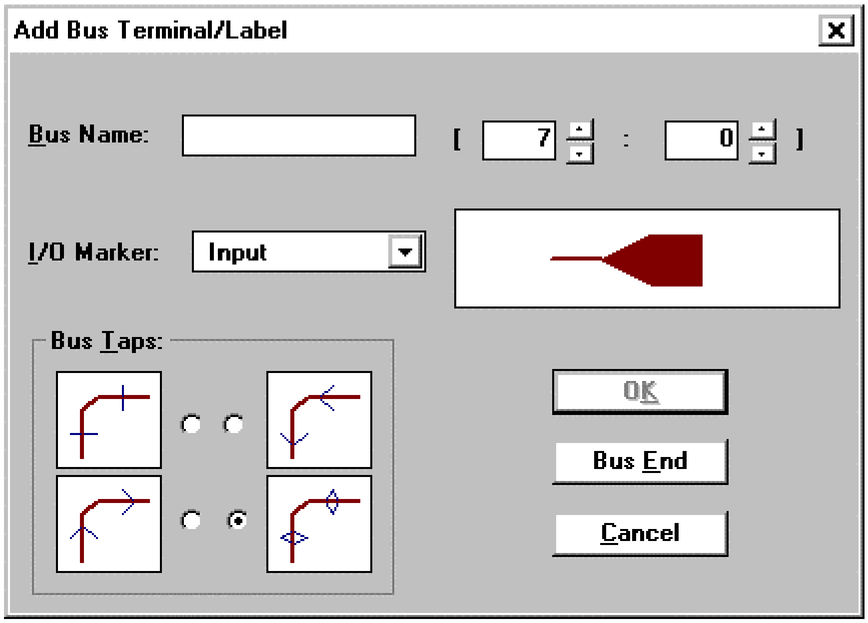
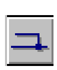


Figure 5. The Add Bus Terminal dialog box.

9) To connect the bus symbols to the counters, click on the "**Draw Bus Taps**" icon on the vertical tool bar,  click on the bus name, and then click on each of the counter outputs starting with the least significant bit. This will automatically place each node name into the schematic. Make sure that each of these has been entered correctly.

10) Often after making changes to a schematic, not all of the wires will appear to connect properly. To fix this, use the command ***Display*>*Redraw***. This command can be used at any time to refresh the screen.

11) After placing the symbols appropriately, click on the "**Draw Wires**" icon on the vertical toolbar.  Connect symbols with wires by clicking on a symbols' node once, dragging the mouse to the destination, and then clicking on the wires' destination once. Wire up the circuit as shown in the CNT60 schematic.

Creating the CNT60 Symbol

1) Once the CNT60 schematic has been created, it's symbol can be created by clicking on the menu command: ***Hierarchy>Create Macro Symbol From Current Sheet*** from within the Schematic Editor. This step opens the "**Create Symbol**" dialog box, which allows entering the name **CNT60** for the macro (see Figure 6). After modifying the symbol name, click on "**OK**". Foundation will create a netlist and symbol for the sheet and add the appropriate files to the projects directory and library. Before the symbol is added to the library, the user is asked to edit the symbol, answer "**NO**".

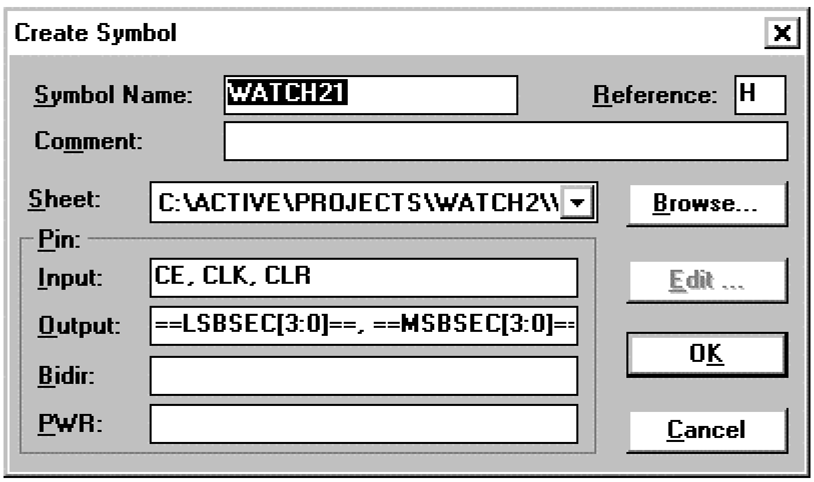


Figure 6. The Create Symbol dialog box.

2) After the symbol has been created, a new sheet can be added to the project by clicking on the menu command: ***File>New Sheet*** from within the Schematic Editor.

3) The top-level schematic can be created by placing the **CNT60** symbol on a blank sheet, and wiring it up with the other necessary symbols contained in the **WATCH** project.

Creating the OUTS macrofunction in the Schematic Editor

1) Add a new sheet to the **WATCH** project by clicking on the menu command: ***File>New Sheet*** from within the Schematic Editor. The schematic editor should now have a blank work area open.

2) Start creating the **OUTS** macrofunction (refer to the Appendix), by clicking on the "**Symbols Toolbox**" icon on the vertical toolbar.

3) Locate the **OBUF** and **OPAD** symbols by scrolling through the list of symbols in the "**Symbols Toolbox**".

4) Copy each symbol by clicking the left mouse button while the pointer is on a placed symbol. After all the symbols have been entered, connect each **OBUF** to an **OPAD**. Name each node by double-clicking on the net and entering the name shown on the schematic. Naming the node connected to an input or output pad effectively names the pad. By naming the pad, the Alliance M1 software will report any pin assignments made in the Pad Report generated by M1.

5) Add the **INPUTS** bus to the project by clicking on the "**Draw Buses**" icon on the vertical toolbar. After drawing the bus, name the bus, provide the bus width, and add the necessary input terminal.

6) Finally, connect the bus to the **OBUF** symbols entered in the sheet by clicking on the "**Draw Bus Taps**" icon, clicking on the bus name, and then clicking on each of the **OBUF** symbols starting with the least significant bit.

7) After the schematic has been completed, create the **OUTS** symbol by clicking on the menu command: **Hierarchy>Create Macro Symbol From Current Sheet**.

8) Enter a new sheet into the project by using the command ***File>New Sheet***. This sheet will be used to create the **OUTS1** schematic

9) Open the **OUTS** schematic by using the Schematic Editor command ***File>Open*** to select the correct sheet.

10) To save time and energy, copy the entire **OUTS** schematic by dragging the cursor around the entire schematic. After the entire macrofunction has been selected, it will be colored red. Copy the selected schematic using the menu command: ***Edit>Copy***.

11) In the **OUTS1** sheet, paste the schematic by using the command: ***Edit>Paste***. Modify the sheet to resemble the **OUTS1** schematic in the appendix, and create the **OUTS1** symbol with the command: **Hierarchy>Create Macro Symbol From Current Sheet**.

12) Create the **OUTS2** macro in the same way.

13) Now that the **CNT60**, **OUTS**, **OUTS1**, and **OUTS2** macrofunctions have been created, they will be entered into the top-level schematic and wired together in another lab.

14) Exit the "**Schematic Editor**" by clicking on ***File>Exit***.

Conclusion

In this laboratory, it was shown:

- "**Select and Drag**" mode is used to move symbols in a schematic
- The "**Hierarchy Push/Pop**" button allows access to lower and higher level macros
- The "**Symbols Toolbox**" allows the user to insert macros library elements into a schematic
- I/O Terminals represent input and output signals from a macro
- Bus Taps are used to separate signals from a bus
- The command ***Hierarchy>Create Macro Symbol From Current Sheet*** is used to make a symbol that represents a schematic sheet

Questions

1) What command is used to refresh a schematic sheet?

2) What mode allows the user to enter macros into a schematic?

3) What must be used to represent input and output signals in a macro?

4) Why is naming nets attached to input or output pads important?