Appendix B
Instruction to Download the Schematic to the Board

---

1. Open a new project - one project for each of the files you will be working on.

2. Perform a functional simulation that checks the logical operation of your circuit. Check the waveform for all the input combinations and make sure that the output is correct for all conditions. If the output is correct, print your schematic and functional analysis output files. You have to select "Current Page" option before you send any file to the printer.

3. Open you schematic file. Delete the input and output terminals. Instead, add **IPAD**s (inputs) and **OPAD**s (outputs) in their places.

4. Now, you will have to map the input and output pins on the board to these pads. For this, select the components (IPAD or OPAD) and the click on the right mouse button. You will see a popup menu. Select symbol properties. You come to a square dialog box. In the space for name, type in capital letters, **LOC**, which stands for location. In the space for description, type the corresponding pin number, say p46 to an IPAD. Then, click add. You will see that the pin number you specified is added to the corresponding IPAD or OPAD. Do this for all inputs and outputs using the following tables which specify the input and output pin numbers. For example, $X_i \rightarrow$p48, $Y_i \rightarrow$p47, $C_i \rightarrow$p46, $C_{i+1} \rightarrow$p19, and $S_i \rightarrow$p23.

XC95108 pin	Xsport
p46	B0
p47	B1
p48	B2
p50	B3
p51	B4
p52	B5
p81	B6
p80	B7

XC95108 pin	LED seg
p21	S0
p23	S1
p19	S2
p17	S3
p18	S4
p14	S5
p15	S6
p24	S7

5. Save the file. Now, go back to the opening screen. You are ready for implementation.

6. Click on the **Implementation Design** in the dialog box. Select device name **95108PC84**. Then, click run. If an error message appears on the screen, then you need to find out the cause of the error. Click on verification and find out the cause of your error. Rectify it.

7. Now, you are ready for downloading the schematic on to the board. Click on **Programming**. You will come to a screen **JTAG Programmer**. This window lets us produce a stream of bits that is suitable for programming a set of XC9500 chips. Select the **Output → Create SVF File…** menu item. Then an **SVF Options** pop-up window will appear. Select Through Test-Logic-Reset, then click on OK.

8. Do not save the file in the rev (revision) directory that appears on the popup box but instead to save the **.svf** file we have to go into the **C:\XSTOOLS\BIN** directory and save the file there as our

executable format will be built there. Therefore go to that directory and save the file as **yourfilename.svf**.

9. Now turn on the power for the XS prototype board. Make sure that the power supply is in the **+6V** range and adjust the voltage to about 6V.

10. Select **Operation → Get Device ID** and then click on OK.

11. In the JTAG popup window that you are in go to **Operations → Program** and select "Erase Before Prog" option. In this step erase the program on the board.

12. As the board has a ROM you probably will need to erase the previous program stored on the board. Go to **Start → Run** and type **C:\XSTOOLS\BIN\Xstest.bat xs95-108**. This will clear the old program from the board.

13. You are now ready to download your program. Go to **Start → Run** and type **C:\XSTOOLS\BIN\Xsload.exe yourfilename.svf.**

14. To test the program, we test it for all possible combinations of the input. For example, if we have 3 inputs the test vector shall range from 000 to 111. To test for each condition go to **Start → Run** and type **C:\XSTOOLS\BIN\Xsport.exe yourtestvector** where you have to manually change your test vector from 000 to 111 for a 3 input circuit and so on. The corresponding output will be displayed on the LED on the board. For 4 inputs case, you have to change the test vector from 0000 to 1111.

15. When you are ready for the experiment, let the TA check your results.

16. When you finish your lab experiment, delete all the .svf files under the BIN directory.