Lab 2 QuickSim II

In this lab you will use the logic simulator QuickSim II to verify the operation of the circuits you created in Lab 1. The following lab provides a step by step procedure to use the simulator QuickSim II.

In this lab you will:

Part 1 Using QuickSim II

1. Invoke QuickSim II
2. Set up a schematic view window
3. Create the List, Trace, and Monitor windows.
4. Setup the Transcript window.
5. Issue stimulus to the simulation.
6. Run a simulation
7 Exit QuickSim II.

PART 1

Once you have logged on you should create a terminal window and move into the ~/ee552/Lab1 directory. You should then invoke Design Manager. (QuickSim II can also be invoked from the shell as “quicksim”).

1. Invoking QuickSim II from Design Manager

a. Click the Select mouse button on the Tools window.

b. Open the QuickSim II Session window, double click the Select mouse button on the QuickSim II icon. Alternatively you could select the QuickSim II icon by clicking once and using the pull down menu

(Menu) > Object > Open

c. In the dialog box the path should be

~/ee552/Lab1/full_adder

use the Navigator to select the required path.

d. Expand the QuickSim II window to fill the entire screen area.

2. Create a schematic view window.

a. In the Setup palette click on the Open Sheet icon. The schematic view of the full_adder should appear. Use the scroll bars to center the circuit in the view area.

3. Selecting Nets

a. Click the Select mouse button inside the Schematic view window.
b. Select all the nets in the schematic view window by clicking on each **portin** and **portout** icon. (The order you select the nets is the order they will appear in the Trace window.)

4. Open the Trace window

a. Click the Trace icon in the palette window.
b. The Trace window appears. The default position for this window is the bottom of the QuickSim II session window. The selected signals are placed in the Trace window.

5. Open the list window.

a. Click the Select mouse button on the List icon in the palette window.
b. The List window appears in the default location to the right of the Schematic view window.

6. Unselecting objects

a. Use the function key F2 to unselect all the selected objects.

7. Open the Monitor window.

a. Open the Monitor window by using the menu bar and selecting

(Menu) > Add > Monitors

b. When the dialog box appears, click on "Named Signals". The entry field for "Signal Name" appears.
c. Enter the following names, one name per field, (use the tab key not the Return key):
SUM
COUT
d. Click OK in the dialog box.
e. The Monitor window appears in its default position at the top of the session window. Signal states are presented for each of the monitored signal names.
f. Unselect everything using the F2 function key.

8. Open the Session Transcript Window

    a. From the palette window select

        (palette) > WINDOWS > Show Transcript

    b. Examine the contents of the Transcript window. Notice that functions have been transcribed for the menu items you have entered.
    c. Make the Transcript Window an icon by clicking on the Minimize button on the Transcript window.

9. Force the CIN signal to the "0" state.

    a. Unselect all signals using the F2 function key.
    b. Click in the Trace window to make it the active window
    c. Select the Trace signal CIN by clicking in the /CIN label.
    d. Use the RMB in the Trace window to bring up the pop-up menu.
    e. Set the logic 0 level by selecting

        (Trace pop-up) > Force > 0 > S

10. Force the AIN signal to the "0" state.

    a. Unselect all signals using the F2 function key.
    b. Click in the Trace window to make it the active window
    c. Select the Trace signal A1 by clicking in the /A1 label.
    d. Use the RMB in the Trace window to bring up the pop-up menu.
    e. Set the logic 0 level by selecting

        (Trace pop-up) > Force > 0 > S

11. Force the BIN signal to the "0" state.

    a. Unselect all signals using the F2 function key.
    b. Click in the Trace window to make it the active window
    c. Select the Trace signal CIN by clicking in the /CIN label.
    d. Use the RMB in the Trace window to bring up the pop-up menu.
    e. Set the logic 0 level by selecting

        (Trace pop-up menu) > Force > 0 > S

12. Run the simulation for 100 ns.

    a. Use the RMB to bring up the Trace pop-up menu.
    b. Select the

        (Trace pop-up menu) > Run > Simulation > For time...

c. In the prompt bar that appears at the bottom of the work area type in 100, then click OK.

13. Changing CIN signal.

a. Unselect all objects using the F2 function key.
b. Select the CIN signal
c. Use the pop-up menu to select

(Trace pop-up menu) > Force > Multiple values

d. The force multiple values dialog box appears.
e. Enter the following and then OK the dialog box.
Signal Name CIN
Value 1 Time 0
Value 0 Time 100
Value 1 Time 200
f. Now run for the simulator for 300 ns, using the Trace pop-up menu.

14. Change the AIN signal.

a. Unselect all objects using the F2 function key.
b. Select the A1 signal
c. Use the pop-up menu to select

(Trace pop-up menu) > Force > Multiple values

d. The force multiple values dialog box appears.
e. Enter the following and then OK the dialog box.
Signal Name AIN
Value 1 Time 0
Value 0 Time 100
Value 1 Time 200
f Now run for the simulator for 300 ns, using the Trace pop-up menu.

15. Change the BIN signal.

a. Unselect all objects using the F2 function key.
b. Select the BIN signal
c. Use the pop-up menu to select

(Trace pop-up menu) > Force > Multiple values

d. The force multiple values dialog box appears.
e. Enter the following and then OK the dialog box.
Signal Name BIN
Value 1 Time 0
Value 0 Time 100

Value 1 Time 200

f. Now run for the simulator for 300 ns, using the Trace pop-up menu.

16. Reset the simulation time to 0

` a. Use the RESET button on the Setup palette window to reset the state (simulation time).

17. Connecting clock inputs

a. Unselect all objects using the F2 function key.
b. Select the CIN signal
c. Use the pop-up menu to select

(Trace pop-up menu) > Force > Clock...

d. In the dialog box set the period to 100, the duty cycle to 50%, and one transition per period. Click the OK button
e. Repeat for the AIN signal but set the period to 200.
f. Repeat for the BIN signal but set the period to 400.
g. Run the simulation for 800 ns.

18. Look for glitches in the output, and determine the maximum delays. Does this correspond to the delays you had put into the inverter and muxs? You may use the

(Menu) > View > Area

to expand a given segment of the trace window. You can use function key F5 to select a cursor or cursors to get exact timing values.

19. Changing delay models

a. Use the QuickSim menu bar and select

(Menu) > Setup > Kernel > Analysis...

b. When the dialog box appears select
Delay

20. Rerun the simulation in section 17 and answer the questions in 18.

PART 2

Carry out timing simulation for the add4 circuit. Determine the input transitions that will lead to the worst case delays in S0 and COUT. How large are these delays? Hand in the simulation waveforms.

PART 3

Design a 2-bit ripple carry adder with the existing full_adder and D-flipflops ("dff" in gen_lib)) added to all inputs (`Carry_in, A0, A1, B0, B1`) and all outputs (`Sum0, Sum1, Carry_out`). Set the D-flipflop rise and fall times to 1ns. Connect the preset and clear signals to the appropriate logic level. Create a new **portin** for `Clock`. Simulate the circuit and determine the maximum clock frequency of correct operation.

PART 4

Take a copy of the 2-bit ripple carry adder from Part 3 and further pipeline it to achieve a higher clock frequency. Note: correct data should arrive at all of the outputs at the same time. What is the maximum clock frequency for correct operation? What is the latency (time from a given input to the output of the result of that given input)?

Hand in schematics and simulation output, including those that support your findings and answer all questions.