EXPERIMENT 7

Simulation of Logic Circuits

In-Lab Procedure and Report (30 points)

After meeting in the lab with your instructor (Broun 266) for a preliminary lecture and quiz, you will be given further instructions for accessing the necessary software and performing the experiment in a "self-paced" mode. You will move into Broun 308 and/or you will be informed of times when your instructor or assistants will be available to help you in lab. **Your report is due at the end of the lab period** unless your instructor tells you otherwise.

This experiment will be performed on a computer using PSPICE. The PC lab in Broun 308 is available for your use except when reserved for other classes. PSPICE is installed on each of the machines in this lab. Alternatively, you may order/download an evaluation version or a student copy from http://www.orcad.com or get it locally from http://www.eng.auburn.edu/~troppel/91pspstu.exe and install it on your own PC (*make sure to select the 'schematics' options when installing this on your PC*).

The laboratory tutorial is designed to be self-explanatory, and regardless of the availability of your instructor, you should complete this experiment on your own time in Broun 308. ***Report computer and network problems in 308 to Mr. Les Simonton in Broun 321: email s*imonton@eng.auburn.edu.* Please also cc to your Lab instructor and *kirkih@eng.auburn.edu***

Objective: In this lab you will be simulating digital logic circuits in PSpice. At the end of the lab, your schematic should perform correctly when:

- the CLR input is 0, the output should reset to 0
- the CLR input is set back to 1, the output should count up

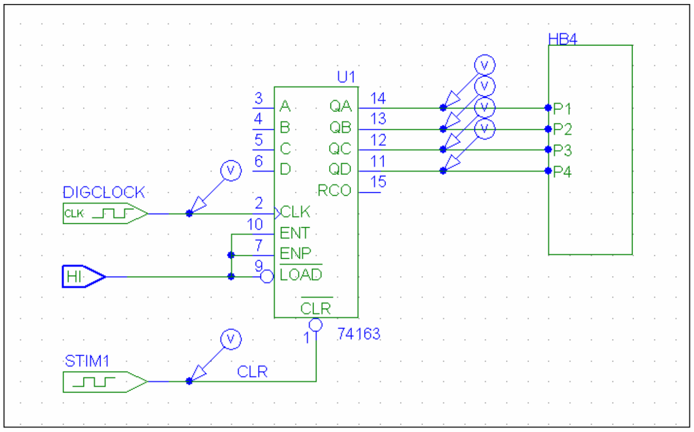
(1) Complete the Digital Circuit Simulation Tutorial (Optional).

If you have not worked the tutorial before coming to the lab, you must go through the Tutorial. The circuit simulation tutorial is part of your lab manual and is designed to introduce you to the basic functionality of PSPICE. Feel free to go online for other resources on PSPICE. A good place to start would be http://www.ewh.ieee.org/soc/es/Nov1999/02/BEGIN.HTM. Work through the tutorial and build a simple circuit to learn how to use the software and then move on to this week's circuit.

(2) Draw and Simulate the 4-bit binary counter circuit shown in Figure 1.

Draw and simulate the test circuit shown in Figure 1 (*due to difference in software versions, your IC diagram might look slightly different, the number of pins and the basic connections however* *should be the same*), which is based on the 74163 binary counter. The 74163 is very similar to the 74169 we used in Expt. 6, except it does not have the option to count down. Also, the P and T enable inputs are not complemented on the 74163, so they must be connected HIGH for the chip to work.

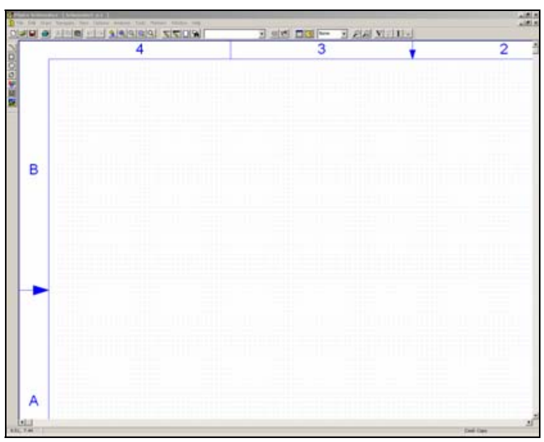
Figure 1: Circuit Diagram after Placing Components and Drawing Wire



(3) Open Schematics

Open PSpice Schematics from the Start menu. Click All Programs -> ECE Applications -> Orcad Family Release 9.2 -> Schematics (this path may change depending on the semester). A white background with grids should appear as shown in Fig. 2.

Figure 2: PSpice Opening View



(4) Get the parts from the library

Click Draw -> Get New Part… or press the toolbar button with the binoculars and the AND gate. In the Part Name box, enter the following part names, click Place, move into the drawing area, and place the part. Continue this process until you have all of the components on the drawing area.

- HI
- DigClock
- STIM1
- 74163

(5) Arrange the parts

Place the parts on the drawing area as shown in Fig. 1.

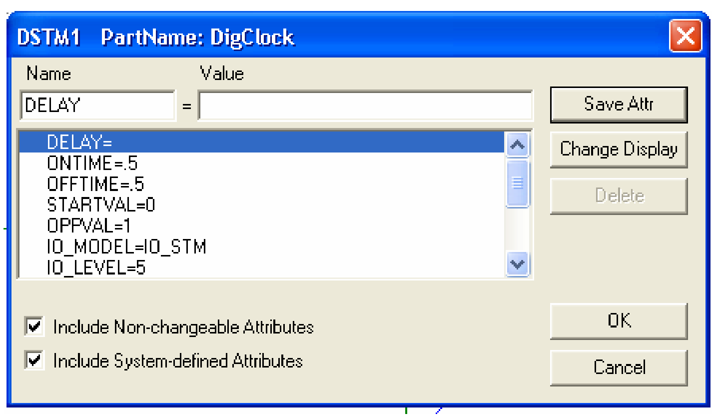
- You can Ctrl+C to copy the components and Ctrl+V to place components if you need more than one component.
- Click Draw -> Wire to connect your components with wire. Wires at nodes should have a blue dot to show that there is a connection. Wires without the blue dot at the intersection are not connected.
- Click Draw -> Text… or the ABC button on the toolbar to place text such as your name or a title. By selecting your text and pressing Edit -> Text Properties, you can change properties such as the size of your font.

You should have a diagram similar to Fig. 1.

(6) Setting Digital Clock:

For the clock input, use the digital clock part. Pick a clock period and record this in your lab report. Set the parameters for this clock by double clicking on the part schematic. An example to set the clock is shown in Figure 3. Set the duty cycle based on the selected clock period. We will set up the window for the transient analysis later on.

Figure 3: Digital Clock settings



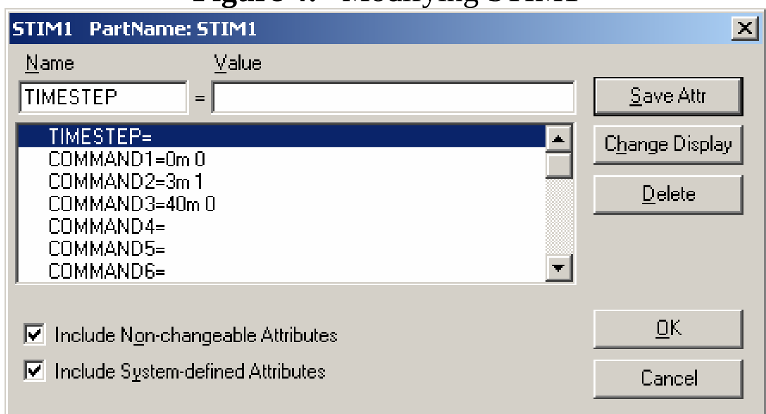
(7) Place the voltage markers

Click Markers -> Mark Voltage/Levels and place the markers as shown in Fig. 1. The tips of the arrows must touch the appropriate wires. These markers will show up in the waveform you will generate later.

(8) Re-setting the counter

Set up STIM1 to reset the counter. The counter should count up after it has been reset. To properly set up the counter, the CLR pin will need to be set and reset. Since the CLR pin is active low (as shown by the circle at the pin and the bar above the pin name), CLR is high when STIM1 is low. Double click on STIM1, which will allow you to change the state of CLR at specified time intervals. You can specify at which times you want the CLR line to change using the COMMAND selection. The first number to enter is the time at which the change will occur, while the second is the state (1 or 0). One choice for these values can be seen in Fig. 4.

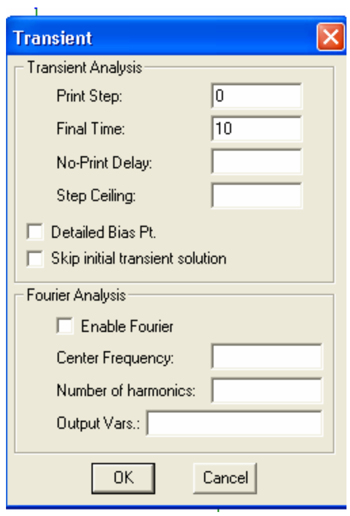
Figure 4: Modifying STIM1



(9) Set up the simulation

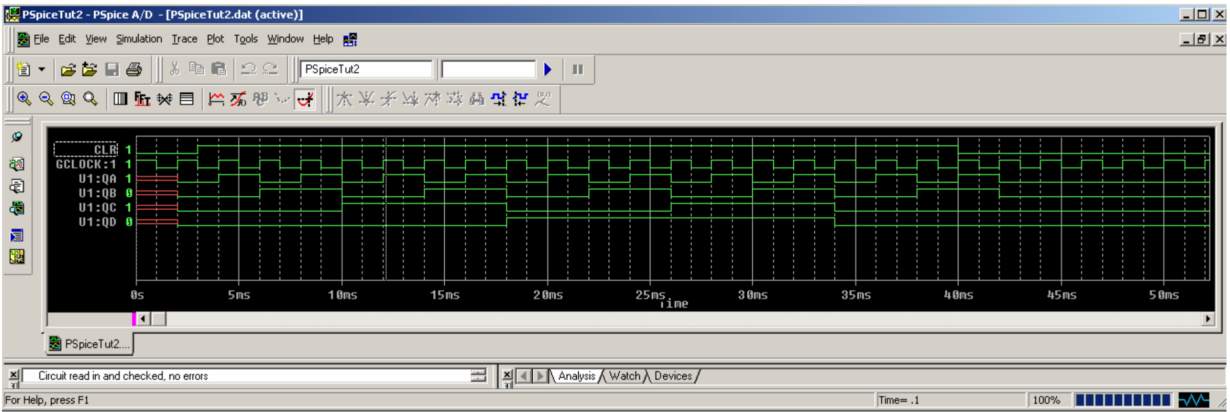
Click on Analysis -> Setup… and click the checkbox next to Transient…. Click on the button for the Transient Analysis. You should see a window like that in Figure 5. We will be viewing the circuit length of time appropriate for your selection of pulse period, which should be long enough to get an idea of the circuit's function.

Figure 5: Transient analysis window



Now simulate your circuit by clicking on the Simulate button. A plot should appear in a new window that shows you the output for your circuit for each voltage probe on your circuit schematic. A window like the one shown in Figure 5 should appear.

Figure 6: Simulation output



(10) View Simulation

Click on Trace -> Cursor -> Display to show the cursors. Click on the waveform to view the output along the left side next to QA, QB, QC, and QD. You can also click on the clock signal on the left side and then click Trace -> Cursor -> Next Transition or Previous Transition to move through the clock transitions. These commands can also be found on the toolbar. Verify that your waveforms conform to the requirements for the lab.

(11) 20Hz Clk with 75% duty cycle

Modify your schematic to create a clock with a frequency of 20Hz and a 75% duty cycle. Show your lab instructor your simulation results. You should display at least one full count cycle in your simulation.

(12) 30.3kHz Clk with 50% duty cycle

Modify your schematic to create a clock with a frequency of 30.3kHz and a 50% duty cycle. For this simulation, the counter should:

- Count to 7
- Reset to 0
- Count to 10
- Reset to 0
- Count to 3
- Reset to 0
- Stop counting

Show your lab instructor your simulation results. Your simulation output should display all the points above in an adequate timeframe.

(13) Email your circuit to your instructor - this is your lab report.

When your circuit is working properly, put your name, class and the date on the diagram. Then save your circuit on your H: drive and/or on a flash drive.

Then email your circuit schematics file (.SCH) and the simulation outputs to your GTA as an attachment. The circuit output can be mailed in PSPICE format or as a separate report in Microsoft Word format. The subject line in your email should read "ELEC2010-Expt7-*AUID*." Replace *AUID* with your Auburn University User ID. Your instructor should be able to 'Simulate' your .SCH file successfully. This counts as your lab report submission.

Follow your lab instructor's directions and deadlines.