School of Computing
Computer Science Program

CDA 3101

Introduction to Computer Logic

Assignment 1

Student Name			
Assignment Name	**Assignment 1 : Introduction to Multisim and Logic**		
Checklist	**Maximum Available Points**	**Received Points**	**Information**
Quiz	NA		From Blackboard
Section 1 Printout	10		Inputs and outputs clearly marked.
Section 2 Printout	10		Inputs and outputs clearly marked.
Section 3 Printout	10		Inputs and outputs clearly marked.
Section 4 Printout	10		Inputs and outputs clearly marked.
Section 5 Printout	10		Inputs and outputs clearly marked.
Section 6 Logic Circuit (Both circuits)	20		Inputs and outputs clearly marked. Include pin numbers on all gates.
Section 6 Waveform	20		Waveform via Multisim Grapher View. Must compare to truth table..
Other Documents	10		Truth Tables.
Multisim files for Section 6			See Notes for Scoring
Final Grade Grade	**Total =**		

Notes for Scoring:

Note 1: Assignments will not be accepted late.

Note 2: All submitted documents from MultiSim must contain the student's name and UNF n-number printed via the software (insert text). No name and number; no points!!!!

Note 4: You must also submit the Multisim circuit that was used to create the required documents for Section 6. A final grade of "0" will be assigned if the submitted circuit fails to work completely or is not submitted.

INTRODUCTION

This assignment will introduce you to digital circuit simulation via Multisim. Many of you are familiar with Multisim from your previous class work, however, to address all students in the class this tutorial will assume no prior knowledge of Multisim.

Multisim is part of a circuit design/simulation/prototyping product sold by National Instruments. It is one of the most widely used circuit simulation tool in academics. The associated prototyping tool for printed circuit board development called Ultiboard is popular but not as popular as Eagle, which is available for free.

To use Multisim to simulate a circuit requires you to enter the circuit through a process known as "schematic capture". You enter the circuit graphically using icons for components. You connect the components together by drawing lines that simulate the wire connections. Power supplies and function generators are also instantiated graphically as are test equipment devices. All components are represented graphically, but Multisim understands the electronic characteristics of the components too. After the schematic has been captured a simulation can be conducted to analyze how the circuit will perform.

In this assignment you will conduct a series of experiments that will introduce you to using Multisim for digital circuit simulation. You will begin by creating a new project in multisim and you will learn how to place simple components in Multisim and run an interaction simulation. In Section 2 you will learn how to place simple logic gates and simulate their outputs. Section 3 will introduce you to digital constants and digital derivative clocks that can be used for inputs. The next experiment, Section 4, will replace the digital clock inputs with a digital word input tool. The final experiment, Section 5, will introduce the logic analyzer sink, a powerful digital simulation analysis tool.

PRELAB:

1) Read through the entire Assignment.

2) If you are unsure of any concepts, review them using your course resources (textbooks, documents on blackboard, etc.) or other available resources (online, library, etc.) or by visiting the instructor during office hours. Make sure you understand all the experiments that you will be performing during the assignment.

Section 1: Creating a Multisim Project

Section 1 Introduction:

In this experiment you will learn how to create a Multisim project, enter simple components and connect them together for schematic capture, and perform an interactive simulation.

Section 1 Procedures:

1) Open Multisim. The Multisim software environment is shown in Figure 1. Note the different areas and features of the environment.

2) Make sure you have downloaded the blank schematic file "Title Block" from the Course Info section within Canvas. Now in Multisim select Open from the File Menu and browse to the place where you stored the file "Title_Block". Open the file. Save this file as A1S1 (for Assignment 1, Section 1)

3) Double click on the title block to open a pop-up window that will allow you to change the details that will be displayed in the title block. Make sure you enter all of the appropriate data for this assignment and section. Enter the date in a searchable format such as "YRMMDD." The document number should reflect the assignment and section number. Use additional sheets (with the correct sheet number) if you have more than one schematic per section.

4) Place the components shown in Figure 2 in the schematic window, as detailed below.

5) Select Component from the Place Menu. This will bring up the Place Component pop-up window. Note the different areas and features of the pop-up window as shown in Figure 3.

6) In the Place Component pop-up window select the Sources Group and the Power Family and then click on the Vcc component to add a DC voltage source. Click ok. The pop-up window disappears and your cursor in the schematic window is now the icon for the Vcc symbol. Move it to a desired location in the circuit window and click to release it. The pop-up window returns.

7) Next add a digital ground by selecting the DGND symbol in the same pop-up window.

8) Now add four Single-Pole, Double-Throw (SPDT) switches. You will find the SPDT symbol in the Basic Group and Switch Family of the Place Component menu.

9) And finally, add four simple displays using the PROBE-BLUE symbol found in the Indicators Group and Probe Family.

10) Be careful moving the cursor around in the schematic window. By clicking on various parts of a symbol you can change the orientation of the graphic and associated label.

11) Carefully click near a symbol until a dashed box appears around the symbol. This indicates you have selected the object. Now you can move the object and reposition it. Adjust the symbols so they are oriented as shown in Figure 4. Now right-click on each PROBE_BLUE symbol and select properties. On the label tab of the pop-up window change the RefDes to "LED1" (or 2, 3, or 4). This will change the displayed name of the LED (and also rename it for simulation purposes).

12) Now right-click on each switch and select Flip Horizontally, to flip the symbols about their VERTICAL axis.

13) The switches will be used interactively during the simulation. It is possible to toggle the switches during a simulation by pressing a key on the keyboard. Note under each switch symbol the label "Key = Space". This indicates that each switch is toggled (from one switch position to the other) by pressing the space bar during a simulation. Since all four switches are all labeled "Key = Space", all four switches will be toggled at the same time when the space bar is clicked. If you right-click on a switch and select properties, under the value tab the "Key for Toggle" menu selection will allow you to change what key is used. Change the key for each switch as indicated in Figure 4.

6

14) Now it is time to connect the components together. Wires are drawn between component pins to interconnect them. Moving the cursor over a component pin changes the pointer to a crosshair, at which time you may click to initiate a wire from that pin. This causes a wire to appear, connected to the pin and the cursor. Move the cursor to the corresponding pin of the second component (the wire follows the cursor) and click to terminate the wire on that pin. If you do not like the path selected for the wire, you may click at a point on the drawing sheet to fix the wire to that point and then you can move the cursor to continue the wire from that point. You may also initiate or terminate a wire by clicking in the middle of a wire segment, creating a "junction" at that point. This is necessary when a wire is to be fanned out to more than one component input. Connect all of the components as shown in Figure 5.

15) To change the color of a wire you must first right-click a segment of wire. You have two choices for changing the wire color. You can change the color of the wire segment or the color of all of the wire segments that are connected together as one "net". Select Net Color in the right-click menu and then select the appropriate color.

16) Now you are ready to simulate. Select either the green arrow simulate icon at the top, the toggle switch at the top right, or the Run command from the Simulate Menu. See Figure 1 for more information.

17) The simulation should now be running. Toggle the switch keys to watch the probes change from on to off and back. Note that the switch symbol actually changes position when you toggle it.

18) Stop the simulation.

19) Add a DCD_HEX display from the Indicators Group and Display Family. Connect it as shown in Figure 1.

20) Run the simulation. Change the switch positions and note how the LED lights (PROBES) and the display change together.

21) Change the switch positions to display the letter "F" on the display. Print this circuit to a "pdf" file using procedures within Multisim. <u>Note that print screens (screenshots) will not be accepted.</u> **Submit this document for grading.**

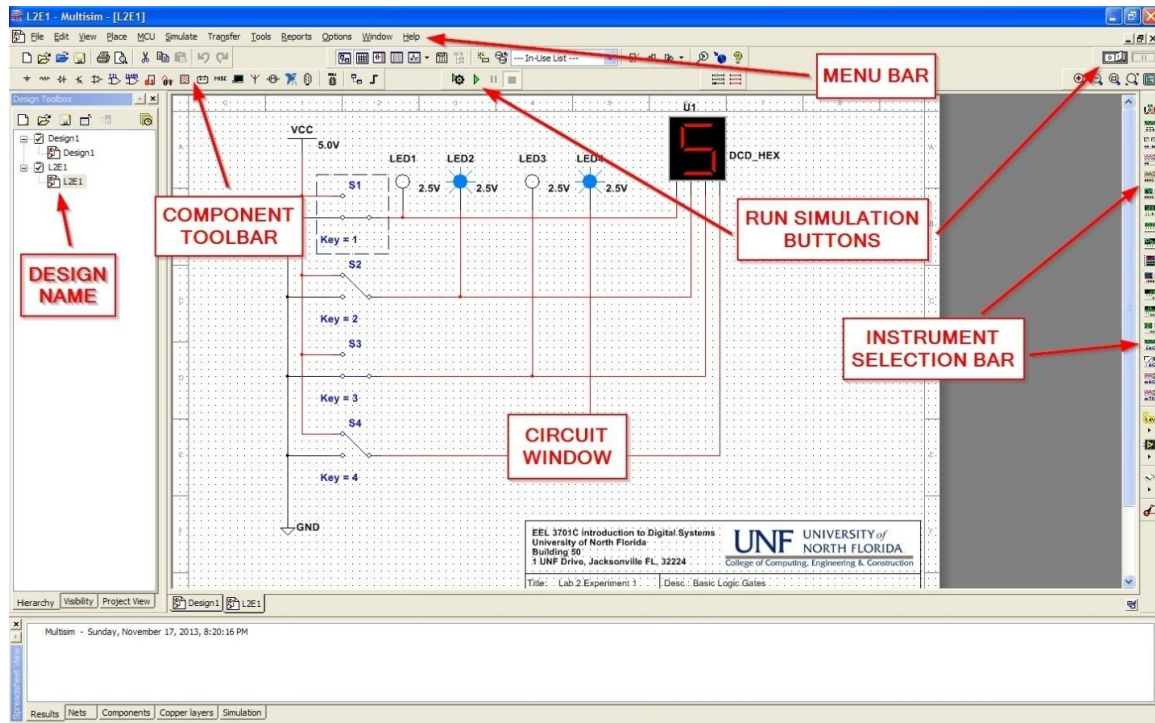22) Save the Multisim file so you can refer to it at a later date.

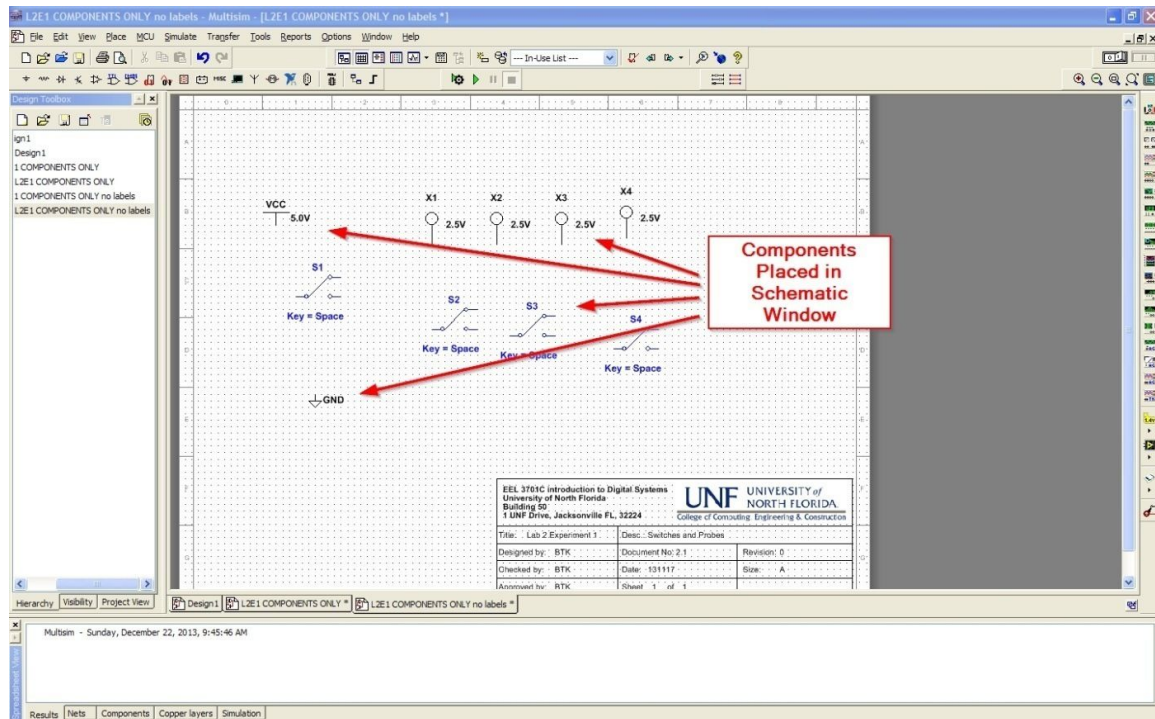Figure 1. Multisim software environment.
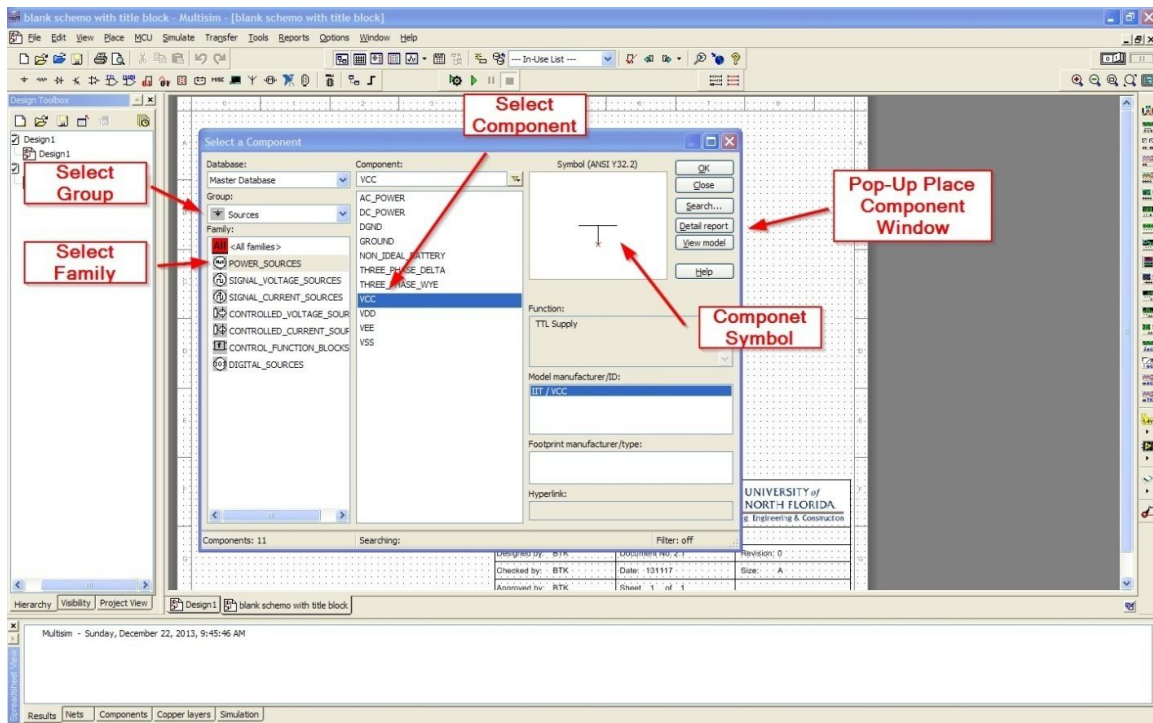

Figure 2. Placing components in Multisim.

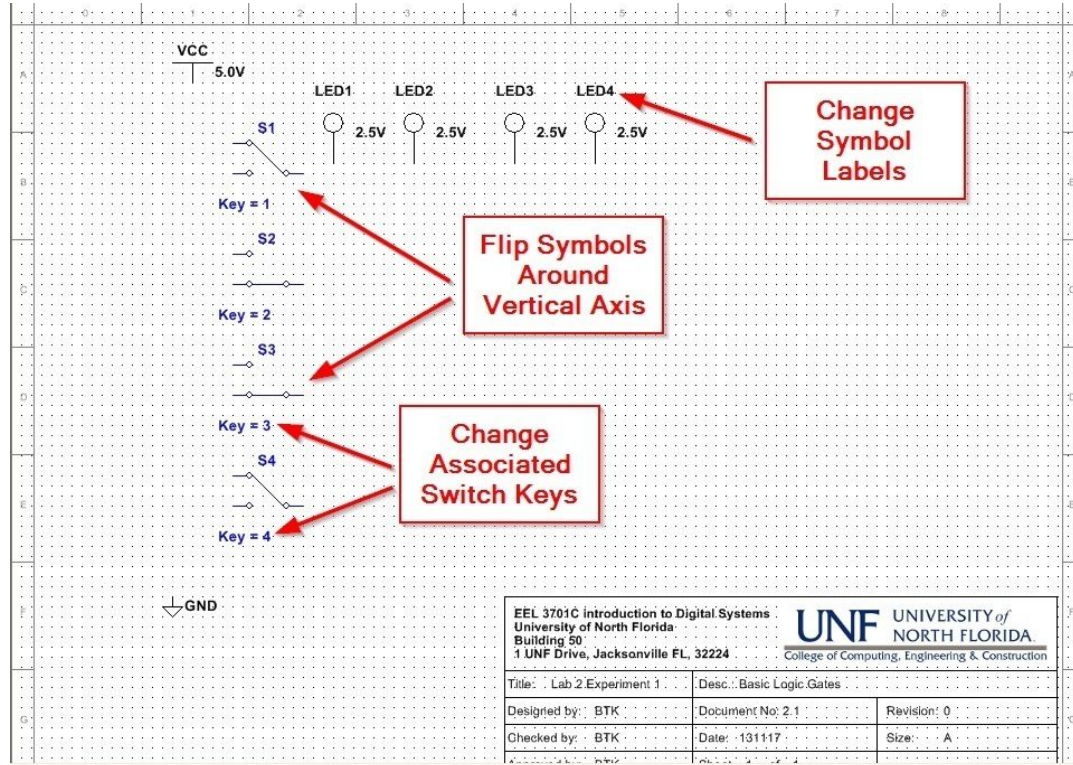Figure 3. Place Component Pop-Up Window

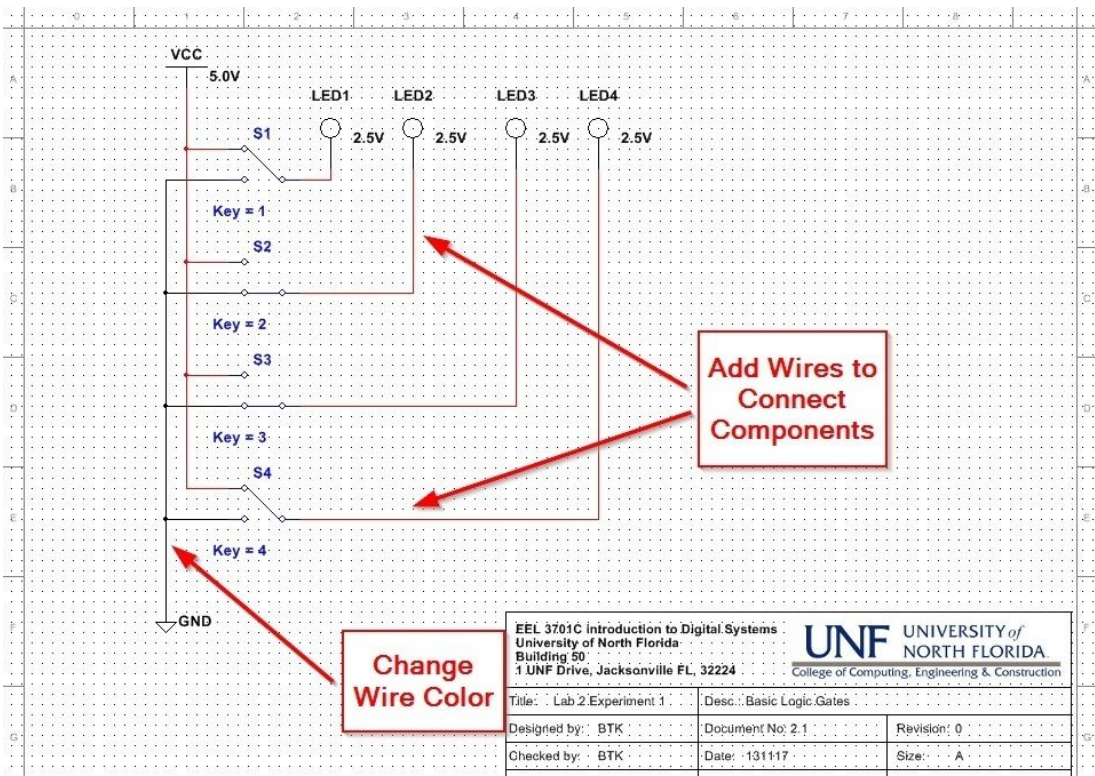Figure 4. Customizing symbol layout and labels.

Figure 5. Connecting components.

Section 2: Working with Logic Gates

Section 2 Introduction:

In this experiment you will learn how to work with digital logic gates in Multisim. We will working extensively with Transistor-Transistor-Logic (TTL) and so we are particularly interested in how Multisim supports TTL devices. In Multisim there are two Component Groups with TTL devices. The first is "Miscellaneous Digital" and the second is "TTL".

The Miscellaneous Digital Group has a "**TIL**" Family (not **TTL**) of digital logic devices that are technology agnostic. That means they have only generic parameters such as delays and voltage sensing levels. These devices are often used for simple simulations intended to test the functionality of a design. Individual gates are provided in this Group and Family. You can place one gate at a time with these components. However, it should be noted that this course will be using the TTL family (74LSxx) for all other labs.

The TTL Group includes components that have realistic parameters and they are used when technology-specific parts are needed. Instead of individual gates, this Group contains different technology Families that have individual integrated circuit chips. The designer will typically select a chip from this Group, and a particular family, and then Multisim will allow the designer to place all or some of the gates or devices from that chip into the schematic window. The symbol labels will indicate that several gates may come from the same chip. Again, we will be using the TTL family (74LSxx series) for all other labs in the course.

Section 2 Procedures:

1) Open Multisim and then open the same blank title block file that you used in Section 1. You will use this file as the start point for all of your Multisim simulations. Change the Title Block components as appropriate. Save the file with the file name L1S2.

2) Using digital devices from the Miscellaneous Digital Group and **TIL** Family, create the circuit shown in Figure 6. Be sure to change the labels, keys, and line colors as indicated in the Figure.

3) Start the simulation and adjust the input switches to watch how the logic states change.

4) Adjust the switches so all of the LEDs are turned on. Print this circuit to a "pdf" file using procedures within Multisim. <u>Note that print screens (screenshots) will not be accepted.</u> **Submit this document for grading.**
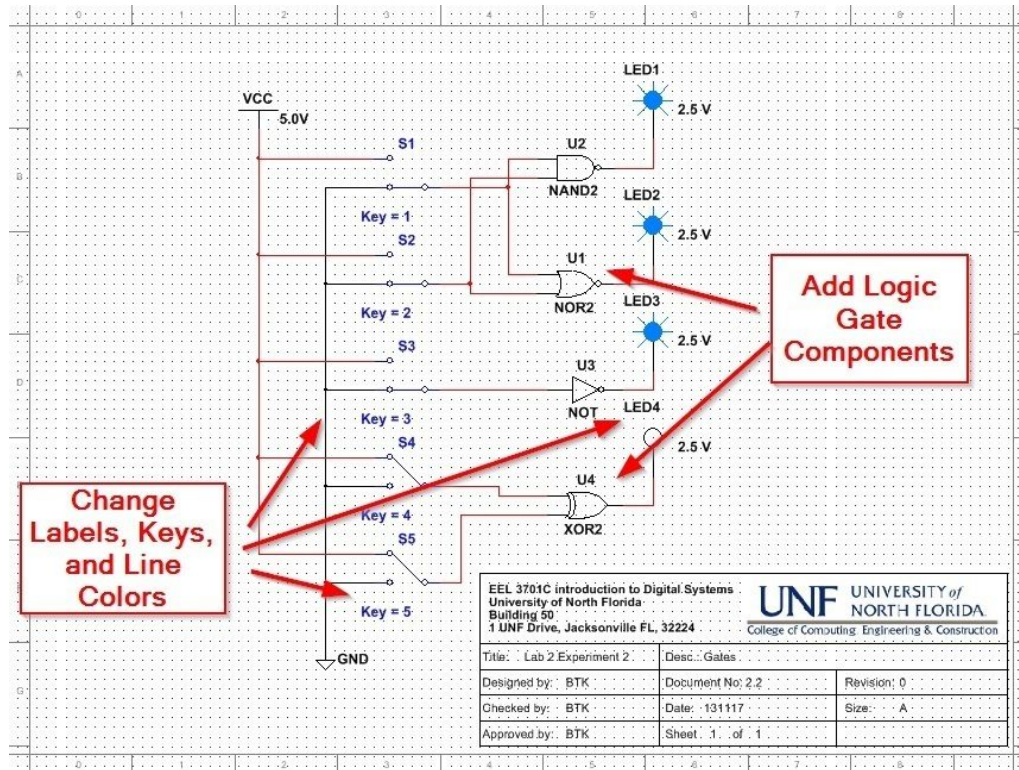

Figure 6. L1S2 Circuit.

Section 3: Using Digital Sources Instead of Switches

Section 3 Introduction:

In the first two experiments the input signals to the circuits were created from switches (SPDT) and power sources (VCC and DGND). By changing the Key associated with each switch it was possible to toggle the inputs between the two power sources. Multisim includes digital sources that are much easier to use than the SPDT switch. Specifically, three digital sources are the DIGITAL_CONSTANT, INTERACTIVE_DIGITAL_CONSTANT, and the DIGITAL_CLOCK. They are all found in the Sources Group and DIGITAL_SOURCES Family of the Place Component menu in Multisim.

Section 3 Procedures:

1) Open Multisim and then open your L1S2 simulation file. Now save the file as L1S3. Make the appropriate changes to the title block as you did before.

2) Delete the SPDT switches and power sources (Vcc and DGND).

3) For each switch insert a DIGITAL_CONSTANT source and connect it to the gates as the switch was connected. Your schematic should look like Figure 7. Set the values of each source by right-clicking the properties for each DIGITAL_CONSTANT symbol and changing the input on the value tab in the properties pop-up window.

4) Run the simulation. Note that you can NOT change the values of the DIGITAL_CONSTANT sources once the simulation has started.

5) Now replace the two DIGITAL_CONSTANT sources for the NAND gate at the top of the schematic with INTERACTIVE_DIGITAL_CONSTANT sources. Change the toggle keys for the two new symbols to A and B as shown in Figure 8.

6) Run the simulation. Note that you CAN change values of the INTERACTIVE_DIGITAL_CONSTANT sources after the simulation has started.

7) Now replace the DIGITAL CONSTANT sources for the XOR gate with DIGITAL_CLOCK sources. Replace the XOR gate with a 2-input OR gate (OR2) from the

Miscellaneous Digital Group and TIL Family. Connect the DIGITAL_CLOCK sources to the OR gate and then connect the OR gate to the PROBE_BLUE symbol.

8) Change the clock rates of the DIGITAL_CLOCK sources using their properties pop-up window. Use the values shown in Figure 9.

9) Run the simulation. Study the flashing rate of the OR gate and be prepared to discuss it in the Review Questions for this Assignment. Print this circuit to a "pdf" file using procedures within Multisim. <u>Note that print screens (screenshots) will not be accepted.</u> **Submit this document for grading.**

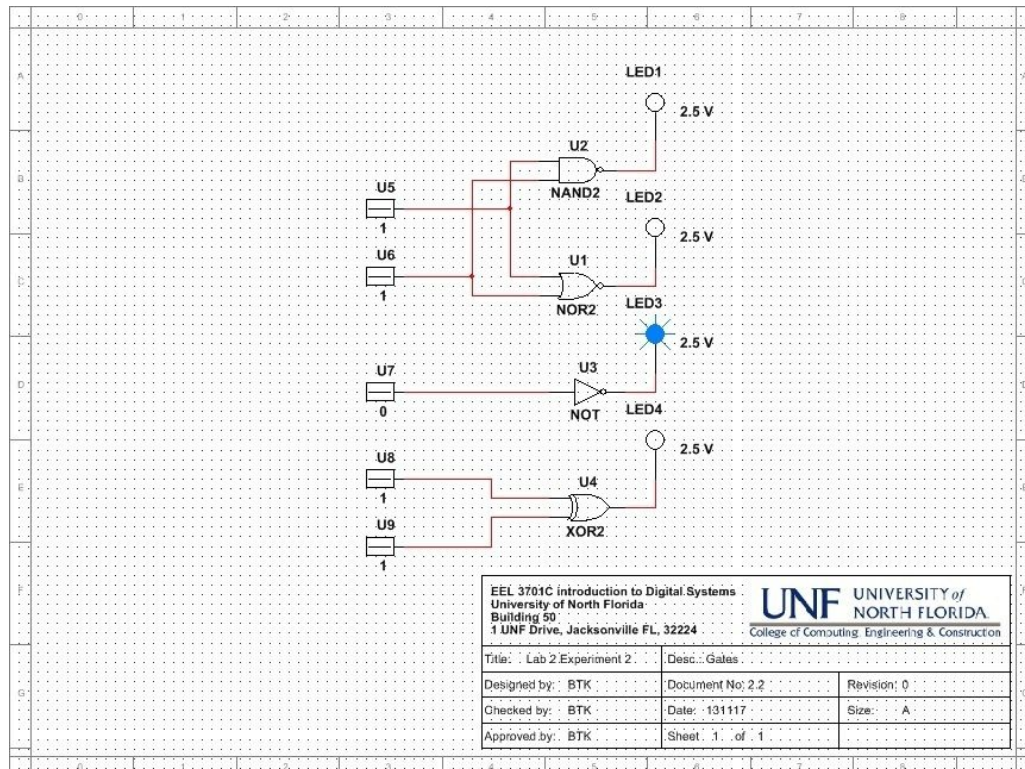10) Save the Multisim file so you can refer to it at a later date.
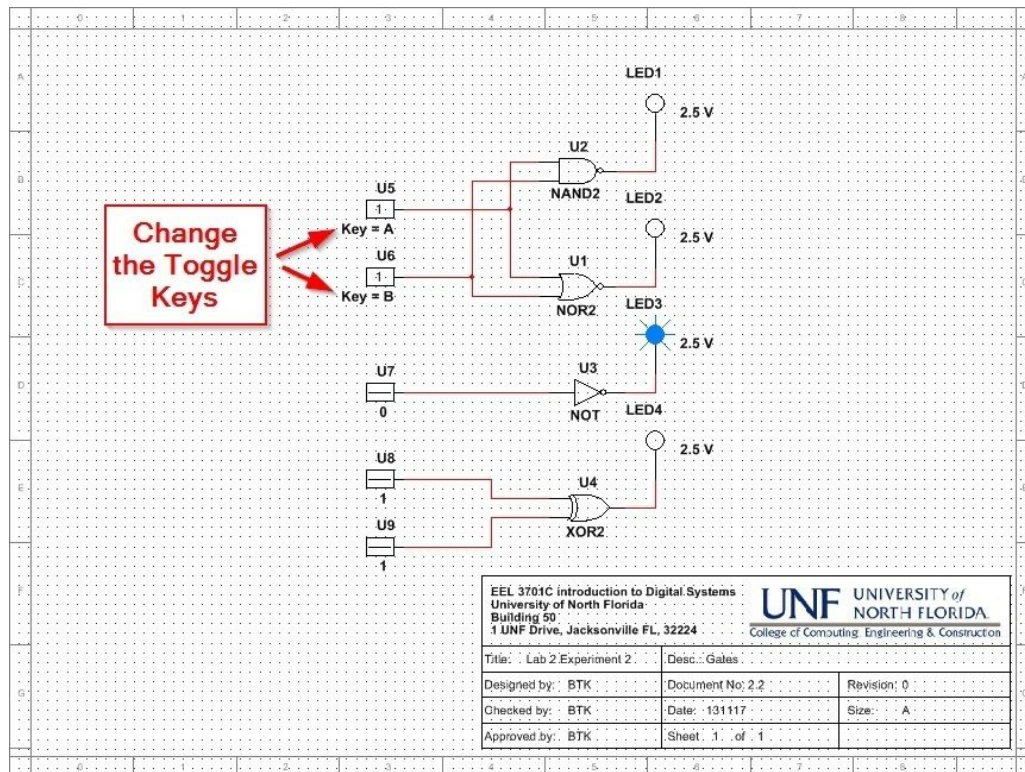

Figure 7. Using the DIGITAL_CONSTANT source.

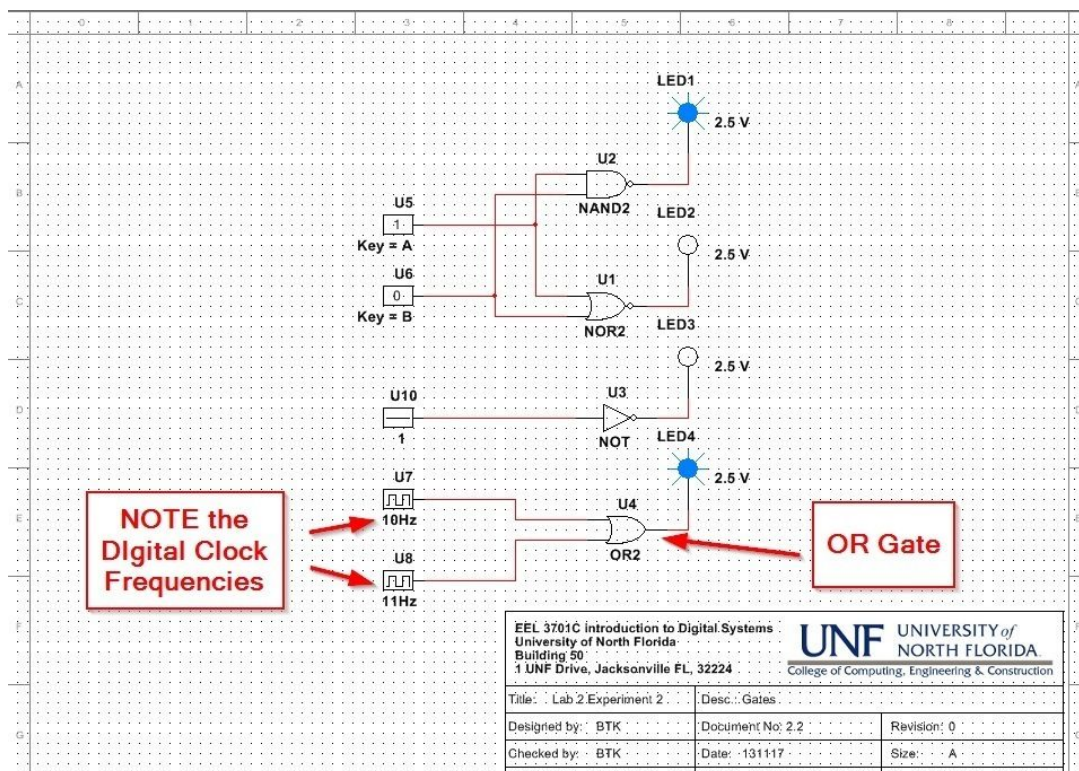Figure 8. Using the INTERACTIVE_DIGITAL_CONSTANT source.

Figure 9. Using DIGITAL_CLOCK sources.

Section 4: Using Digital Clocks and Digital Words

Section 4 Introduction:

In this assignment we will continue investigating digital sources using first the DIGITAL_CLOCK source from Section 3. In particular the use of clocks to create counters will be explored. After using the DIGITAL_CLOCK source, the digital Word Generator source that is in the Multisim instruments menu will be explored.

Section 4 Procedures:

1) Open Multisim and then open the same blank title block file that you used in Section 1. Change the Title Block components as appropriate. Save the file with the file name L2S4.

2) Create the schematic shown in Figure 10 using DIGITAL_CLOCK sources and PROBE_BLUE symbols. Change the clock frequencies as shown in the Figure so that they are 1Hz, 2Hz, and 4Hz respectively.

3) Run the simulation.

4) Stop the simulation. Then remove the digital clocks. Replace them with a digital Word Generator source. This tool is available from the Simulate menu by selecting Instruments and then Word Generator. You can also use the short-cut on the instrument selection bar. Multisim will let you drag and place the Word Generator source in the schematic window as you did with the components. Check the orientation of the Word Generator and flip the symbol as necessary so the pin numbers start with 0,1,2… running down the right side, top to bottom.

5) Once you have placed the Word Generator source you need to wire it. Connect the first three pins on the right side of the Word Generator symbol (pins 0, 1, and 2) to the three PROBE_BLUE symbols as shown in Figure 11.

6) To configure the Word Generator source, double-click its symbol to access its parameters in a control window. The control window is shown in Figure 12. This tool is used to create specific test vectors for digital circuits. The test vectors can be complex or they can be a simple counting or shifting sequence of 1's and 0's. The panel on the right of the window

contains the bit sequences that will be output on the 32 output pins of the Word Generator source. When the Word Generator's clock increments, another bit sequence is placed on the output pins. The sequences are placed on the output pins one after the other, from top to bottom in the panel on the right of the control window. The bit sequences can be displayed in various number system formats, including hexadecimal, decimal, and binary. These are selected in the center of the control window. The Word Generator clock frequency is set at the bottom right. Set it now to 10 Hz. Bit sequences can be custom patterns that are set bit by bit in the panel on the right or they can be generated automatically. To automatically generate a three bit counting clock pattern select the "Set…" button. This will bring up the Settings window shown in Figure 13. Select the "Up Counter" preset pattern. Next change the buffer to 8 to limit the length of the sequence to $2^3 = 8$. Hit ok to return to the control window. Now you can see the patterns that will be sent out in the panel on the right.

7) Note the upper left area of the control window contains a control panel that includes several control mode buttons. We are using the default Cycle mode. In this mode after the 8 patterns are cycled through the output pins, the sequence repeats using the same 8 digital words. If the Burst mode selected Multisim will iterate through all of the patterns in the sequence one time and then stop. Move the control window so that you can see the 3 PROBE_BLUE symbols. Use the step button to iterate the highlighted word until the highlighted digital word is at the top of the list on the right of the control window (In MS13 there is a Reset button that can do this too). Now hit the Burst mode button. Note how Multisim iterates through all of the digital words, changing the blue LEDs of the PROBE_BLUE symbols as it iterates, and then stops. Now click the Step mode and observe the final control mode. In this mode it is possible to manually step through all of the digital word patterns. Hit the Step mode button several times to observe this. Also note the bit sequence currently being output is present in the icon bit sequence at the bottom of the control window.

8) Print this circuit to a "pdf" file using procedures within Multisim. <u>Note that print screens (screenshots) will not be accepted.</u> **Submit this document for grading.**

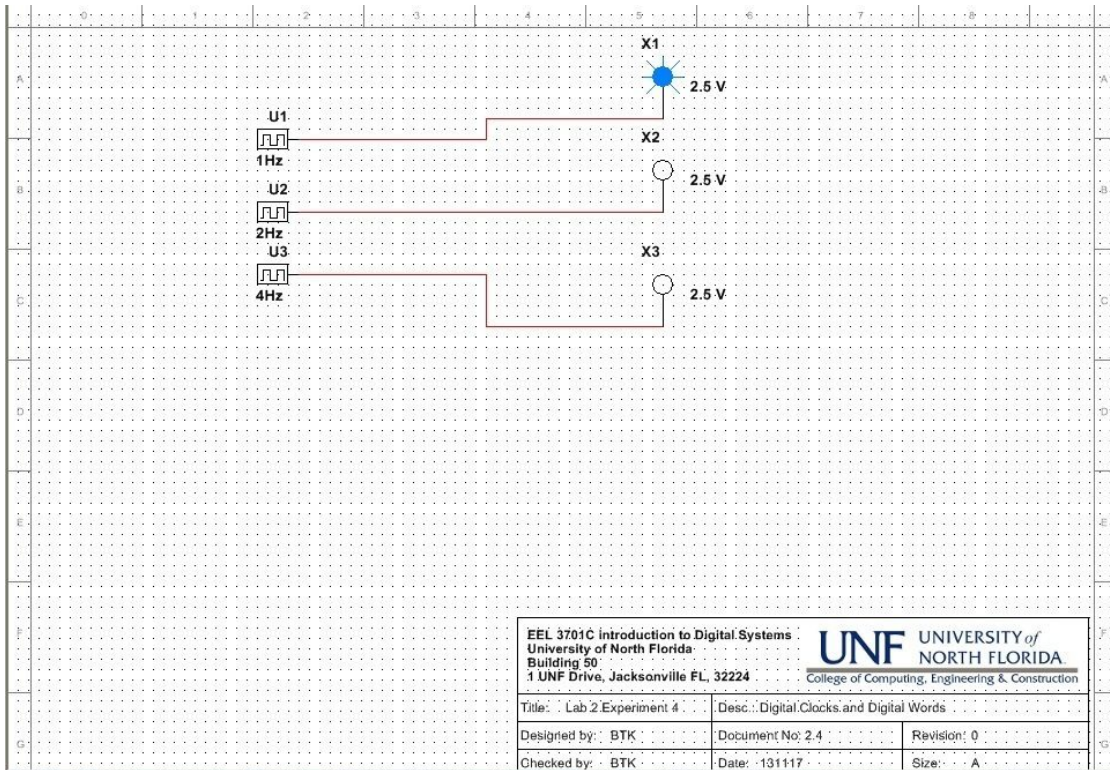9) Save the Multisim file so you can refer to it at a later date.

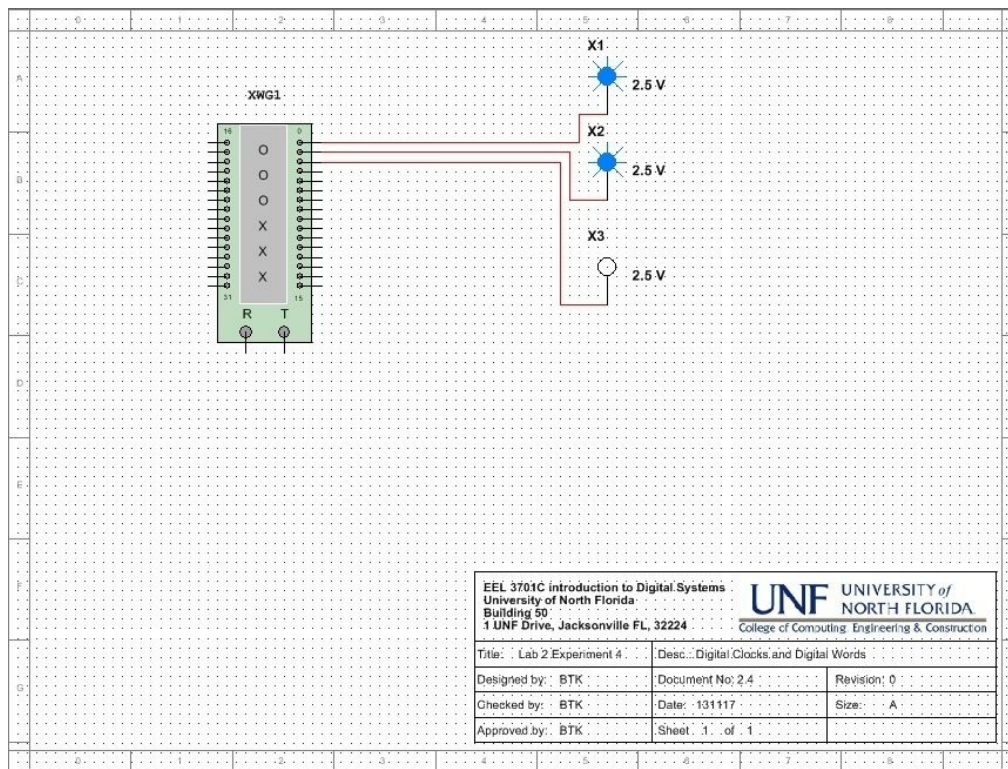Figure 10.Using DIGITAL_CLOCK sources as counters.

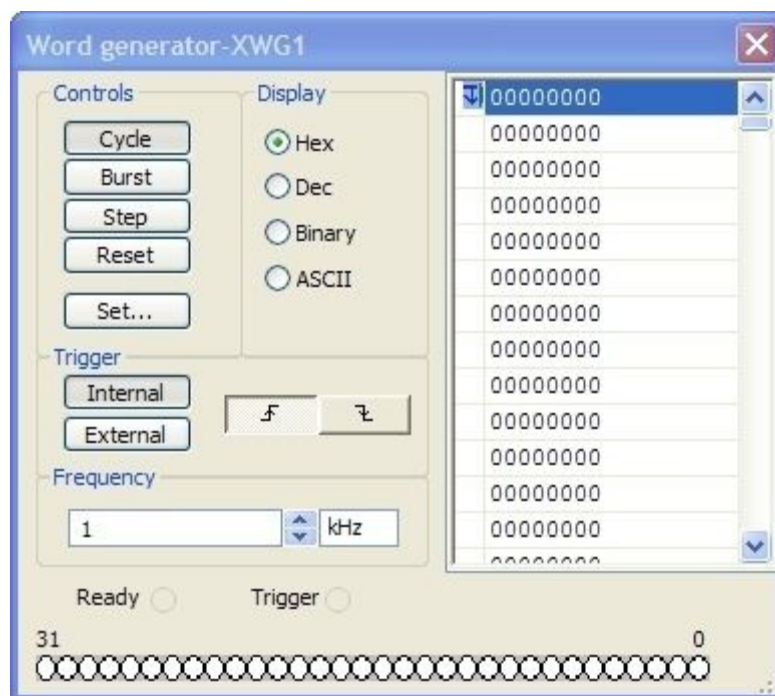Figure 11. Using the Word Generator source.

Figure 12. Word generator control pop-up window (MS13, not MS11 or MS12).
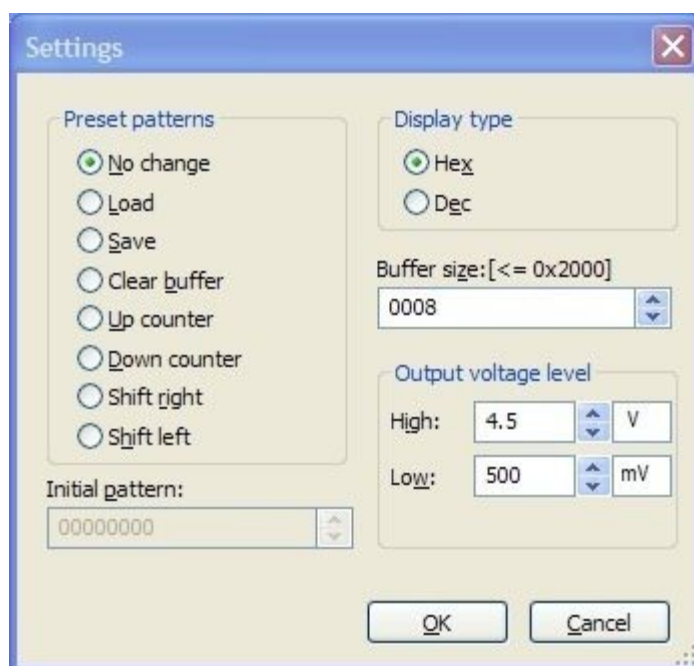

Figure 13. Word generator settings window.

Section 5: Using the Logic Analyzer

Section 5 Introduction:

In this assignment we will use the circuit from the end of Section 4 to investigate a powerful instrument in Multisim called the Logic Analyzer. Since this instrument is where we send data from the simulated circuit, in the simulation industry it is considered a data "sink". This term is used in contrast to the term source which we used in the previous sections. Technically, the way we have used them in this assignment, the PROBE_BLUE symbols are data sinks as well, however they can serve many purposes so the specialized name "sink" is not usually applied to them.

The Logic Analyzer sink can display one or more digital signals over time. It uses a display (similar to an oscilloscope) and plots the logic level on the vertical axis and time on the horizontal axis. The Multisim logic analyzer can plot up to 16 digital signals at a time. It works by sampling the digital signals at specific points in time and displaying the sampled value. This means the Logic Analyzer sink must be clocked. The Multisim Logic Analyzer sink can use an internal clock to sample the digital signals at a user-defined rate or it can be clocked externally from the circuit being simulated. If an internal clock is used to sample the digital signals, the clock rate should always be set greater than or equal to twice the clock speed of the fastest digital signal you want to measure.

Section 5 Procedures:

1) Open Multisim and then open your L1S4 simulation file. Now save the file as L1S5. Make the appropriate changes to the title block as you did before.

2) Open the Word Generator source and set the clock to 10 Hz, internal.

3) Place a Logic Analyzer sink in the schematic window and wire it as shown in Figure 14. This tool is available from the Simulate menu by selecting Instruments and then Logic Analzyer. You can also use the short-cut on the instrument selection bar. Multisim will let you drag and place the Logic Analyzer sink in the schematic window as you did with the components.

4) Double-click the Logic Analyzer sink to view its display and control window. There are several control panels at the bottom of the window. In the Clock panel, click the Set button to access the Clock parameters. Your Multisim screen should now look something like Figure 15. Set the clock rate to 20 Hz, internal. Set the pre-trigger samples to 1 and the post-trigger samples to 10000. This will ensure you collect enough data to view the entire sequence.

5) Press the Reset button in the Logic Analyzer display and control window to clear out its memory and to reset the simulation clock before the next simulation run. Run the Multisim simulation using the standard Multisim simulation start button (the green arrow). After the PROBE_BLUE LEDs indicate the 8 patterns have been cycled through one time, hit the stop button in the Logic Analyzer display and control window.

6) Note the Word Generator source clock speed and the Logic Analyzer sink speed in your assignment report. Study the waveforms on the Logic Analyzer so that you can answer any associated questions. HINT: You many need to analyze the time between transitions and then use this interval to back-up in time from the first transition in order to determine when the 0 digital word started. Print this circuit to a "pdf" file using procedures within Multisim. Note that print screens (screenshots) will not be accepted. **Submit this document for grading.**
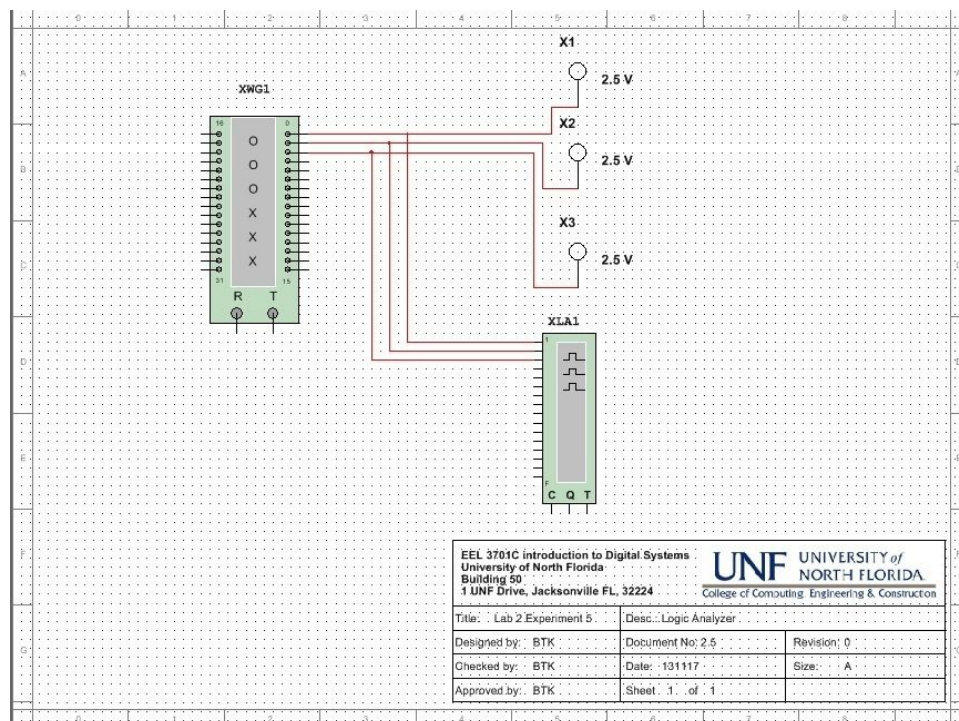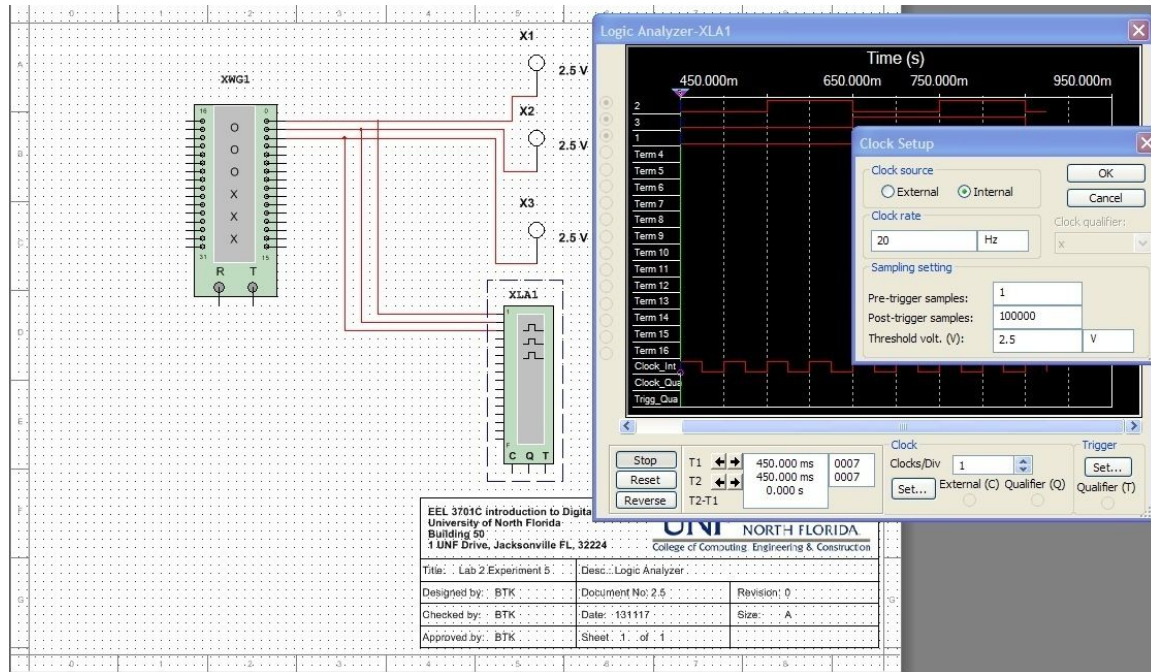

21

Figure 14. Using the Logic Analyzer sink.

Figure 15. Logic Analyzer sink display, control and clock windows.

Section 6: Testing Logic Gates

Section 6 Introduction:

You will now focus on logic gates and their operation. As you may have learned in previous courses, logic operations can be implemented using electronics. We will be using TTL (Transistor, Transistor, Logic) gates in this course to build logic circuits. This exercise will help you become familiar with this technology.

Equipment needed:

Locate and use the these gates within Multisim for the following procedures:

74LS00, 74LS02, 74LS04, 74LS08, 74LS32, 74LS86

Section 6 Procedures:

1) You will be using one logic gate from each device above. Note that each integrated circuit may have multiple logic gates within; however, we will be using only one gate from each part.

2) Using only two switches, connect each to both inputs of each logic gate in parallel with the exception of the inverter. That gate will only have one input connected. The switches can be either a SPDT or an "INTERACTIVE_DIGITAL_CONSTANT" from the "Sources" library within Multisim. Label all inputs and outputs.

3) Verify the operation of all the gates using the switches and indicator probes. Print this circuit to a "pdf" file using procedures within Multisim. Note that print screens (screenshots) will not be accepted. **Submit this document for grading.**

4) Create truth tables showing the "expected" and "measured" data. **Submit a pdf version of this document for grading as well.**

5) Create another circuit using a dual input AND gate with one input going to a logic switch and the other going to a clock. Connect the output to an indicator probe and adjust the clock for 10MHz. Also add an Oscilloscope to your diagram. Run the simulation and set the switch to allow the clock signal to be reflected on the output. Double-click on the Oscilloscope icon and set the Timebase Scale to 20ns/div. Set the Trigger to Normal with a negative going edge (see figure 16). Observe the circuit's built-in delay using the output-timing diagram (waveforms) as described in class.
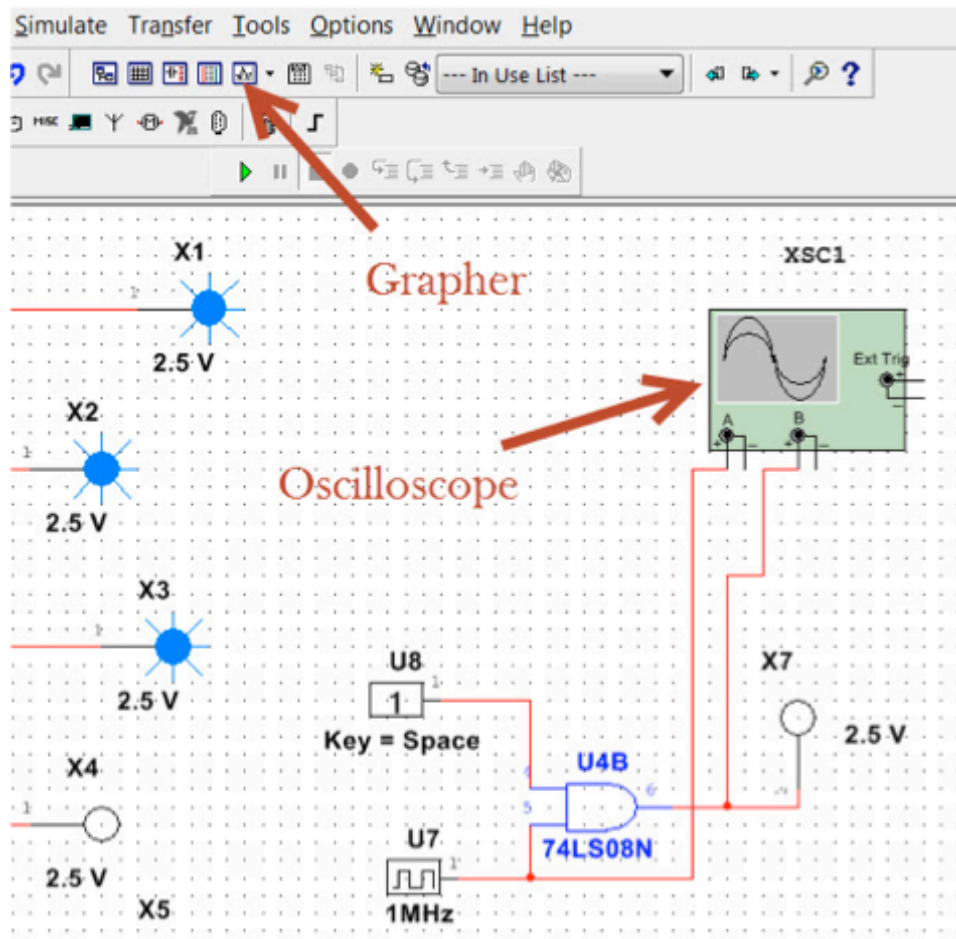
Figure 16

6) In future assignments, these diagrams may be used for building your circuits using the hardware trainers. Therefore, print chip pin numbers for all gates.. Note: See instructions on Canvas for printing pin numbers. Print this circuit to a "pdf" file using procedures within Multisim. Note that print screens (screenshots) will not be accepted. **Submit this document for grading.**

6) Printout your waveform diagrams using techniques discussed in class. Submit these documents for grading. Note: use the Grapher to print the waveforms to pdf. **Submit this document for grading.**

Grading:

<u>You should have eight "pdf" printouts and one Multisim circuit to submit (see below)</u>.
Do not zip / archive / compress them together. Just upload to Canvas.

Submit the following documents:
Section 1
Section 2
Section 3
Section 4
Section 5
Section 6 Multiple Gate diagram and Dual Input AND gate diagram
     (both circuits can be placed on the same sheet)
Section 6 Waveform
Section 6 Truth Tables
Section 6 Multisim Circuits (both circuits can be placed on the same sheet)

ACKNOWLEDGEMENTS: