Laboratory 5: Multisim Simulation of Electronic Circuits

Note! Be sure to keep the inductor you build here for later use in Lab. #6.

1.0 Introduction

Multisim is another type of software used for electronic circuit simulation. Importantly, it differs from PSpice in that run-time adjustments to the circuit (such as switches opening and closing or changing resistance values) can be made while the results are continuously plotted. The program incorporates an easy-to-use graphic interface in which the user can select from a menu of electronic components and instruments to "build" a circuit. Once the circuit is constructed and wired together, the user can run it and see the results on the oscilloscope or multimeter. In this laboratory exercise, you will also construct an actual inductor by winding copper wire in a coil on a torroidal ferrite core. Then, an LR circuit and an RC circuit will be operated and compared to Multisim simulation results.

1. 1 Getting Started

You will need about 12 feet of #22 AWG copper wire and a torroidal ferrite core to wind your inductor. First, strip the insulation off the 2 ends of the wire, exposing about 1/2 inch of bare wire. Neatly wind the wire around the torroid, passing though the center of the core with each turn. Wind enough turns to get an inductance equal to 10±0.5mH as measured on the LCR meter in the lab. You will also need to use the oscilloscope, the function generator, a 1.0μF Mylar capacitor, two 10kΩ resistors, and two 100Ω resistors to build the actual circuits.

2.1 Running Multisim: Series RC Circuit

On the computer, go to START-ENGINEERING-NATIONAL INSTRUMENTS-MULTISIM. A blank schematic diagram screen will appear. It will be used to construct an electrical schematic diagram. Make sure you save the diagram periodically by clicking on the "File" menu and clicking "Save As". Now, you will build a series RC circuit as shown in Figure 2.1.

Click on the resistor symbol in the upper-left menu. Click on the resistor. Click on 10kΩ. Click OK. Click on the schematic window to place the resistor. Place 2 of the 10kΩ resistors on the schematic window. Click on GROUP-SOURCES. Click on POWER SOURCES-DC POWER and place it on the schematic. Double-click on the DC power source and set the voltage equal to1.0V. Click on the GROUND symbol and place it on the schematic. Click on BASIC GROUP and CAPACITOR and 1μF and place it on the schematic. Right-click on the capacitor and rotate it 90 degrees CW. Click on BASIC GROUP and SWITCH and SPST and place a switch on the schematic. Close the SELECT A COMPONENT WINDOW. Notice how the spacebar turns the switch on and off. From the right-hand menu, place an oscilloscope on the schematic.

Place the cursor on the upper terminal of the DC voltage source until a connection dot appears. Press and hold the mouse button down while drawing a wire connection to the switch. Click to end the wire. Connect the rest of the circuit, placing the oscilloscope across the capacitor. Double-click the oscilloscope to open its detailed window. Set the time scale to 10ms per division. Set Channel A to 200mV per division. Click Channel A for DC input. Click EDGE TRIGGER, Channel A, SINGLE, and set the trigger level to 10mV. Move the oscilloscope window to the lower left corner of the screen. Open SIMULATE-ANALYSIS-TRANSIENT and set END TIME = 10SEC. Click OK.

Save your work to a location of your choosing and give the file a name. Click on the schematic window and hold your finger over the spacebar with the switch in the open position. Click the RUN SIMULATION BUTTON in the upper-right corner and immediately click the spacebar 2 times in rapid succession. Notice the exponential relaxation response expected of an RC circuit while the switch closes and opens. Repeat this process a few times until you are comfortable with it. You can save the oscilloscope results to an MS Excel file by clicking the SAVE button on the oscilloscope.

Measure the 10-to-90 rise-time and fall-time, which equal 2.2 times the exponential time constant RC. Do the results agree with your calculations of the RC time constant? Run the simulation while varying the component values and the cycling of the switch. Do the results agree with your understanding of how the circuit works?

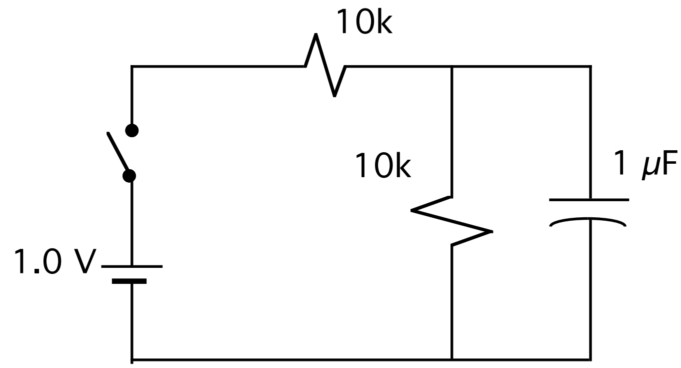

Figure 2.1 Series RC circuit to be used for Multisim simulations.

2.2 Running Multisim: Series LR Circuit

On the computer, go to START-ENGINEERING-NATIONAL INSTRUMENTS-MULTISIM. A blank schematic diagram screen will appear. It will be used to construct an electrical schematic diagram. Make sure you save the diagram periodically by clicking on the "File" menu and clicking "Save As". Now, you will build a series LR circuit as shown in Figure 2.2.

Click on the resistor symbol in the upper-left menu. Click on the resistor. Click on 1Ω. Click OK. Click on the schematic window to place two 1 Ω resistors. Click on GROUP-SOURCES. Click on POWER SOURCES-DC POWER and place it on the schematic. Double-click on the DC power source and set the voltage equal to1.0V. Click on the GROUND symbol and place it on the schematic. Click on BASIC GROUP and INDUCTOR and 10mH and place it on the schematic. Right-click on the resistor and the inductor and rotate them 90 degrees CW. Click on BASIC GROUP and SWITCH and SPST and place a switch on the schematic. Close the SELECT A COMPONENT WINDOW. Notice how the spacebar turns the switch on and off. From the right-hand menu, place an oscilloscope on the schematic.

Place the cursor on the upper terminal of the DC voltage source until a connection dot appears. Press and hold the mouse button down while drawing a wire connection to the switch. Click to end the wire. Connect the rest of the circuit, placing the oscilloscope across the inductor. Double-click the oscilloscope to open its detailed window. Set the time scale to 10ms per division. Set Channel A to 500mV per division. Click Channel A for DC input. Click EDGE TRIGGER, Channel A, SINGLE, and set the trigger level to 10mV. Move the oscilloscope window to the lower left corner of the screen. Open SIMULATE-ANALYSIS-TRANSIENT and set END TIME = 10SEC. Click OK.

Save your work to a location of your choosing and give the file a name. Click on the schematic window and hold your finger over the spacebar with the switch in the open position. Click the RUN SIMULATION BUTTON in the upper-right corner and immediately click the spacebar 2 times in rapid succession. Notice the exponential relaxation response expected of an LR circuit while the switch closes and opens. Repeat this process a few times until you are comfortable with it. You can save the oscilloscope results to an MS Excel file by clicking the SAVE button on the oscilloscope.

Measure the 10-to-90 rise-time and fall-time, which equal 2.2 times the exponential time constant L/R. Do the results agree with your calculations of the L/R time constant? Run the simulation while varying the component values and the cycling of the switch. Do the results agree with your understanding of how the circuit works?

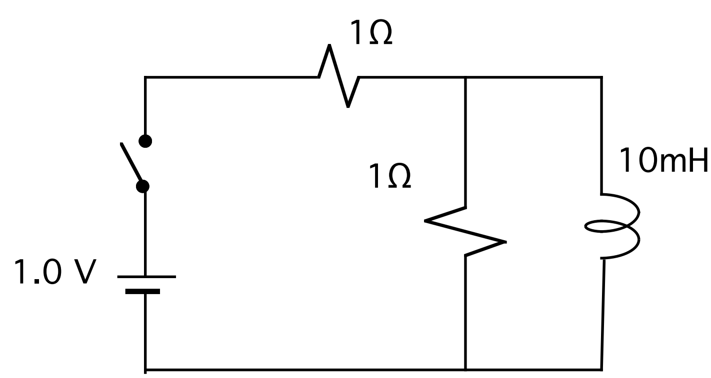

Figure 2.2 Series LR circuit to be used for Multisim simulations. The actual circuit will use two 100Ω resistors.

3.1 Experimental Series RC Circuit

Construct the RC circuit shown in Figure 2.1 using a Mylar capacitor. However, instead of the switch and voltage source, use the function generator to apply a 1.0V amplitude square wave to the RC circuit. Connect the oscilloscope across the capacitor. Run the circuit using conditions similar to those used in the Multisim simulations. The experimental results should resemble the simulated results.

3.2 Experimental Series LR Circuit

Construct the LR circuit shown in Figure 2.2 using the inductor you wound. ***However*, instead of the 1Ω resistors shown in Figure 2.2, use 100 Ω resistors!** Instead of the switch and voltage source, use the function generator to apply a 1V amplitude square wave to the LR circuit. Connect the oscilloscope across the inductor. Run the circuit using conditions similar to those used in the Multisim simulations. The experimental results should resemble the simulated results, allowing for the change in resistor values between the actual circuit and the simulated circuit.

4.1 Report

Write a report giving:

A. The rise-time and fall-time of the square-wave signal from the RC and LR circuits. Present hardcopies of the actual waveforms you observed on the oscilloscope as a result of using the RC and LR circuits.

B. Carry out an analysis of the RC and LR circuit step responses and show how well or how poorly the results agree with the measurements of Part A, including the voltage amplitudes and the rise and fall-times. How well do your analytically derived waveforms agree with the actual measured waveforms? Give a quantitative discussion. Remember, the function generator acts like an ideal voltage source in series with 50 Ω.