

# ECE 332 Lab 2s

## Multisim® for RC Circuit

How do you test a new circuit design? Well, you could just build it and hope it works, which tends to be expensive because Murphy's Law often wins. You could build it with laboratory parts and test it in the lab, which might be more efficient. You could "build" the circuit by describing it to a computer and then simulate it, which sounds a lot cheaper than the other methods. Finally, you could just write equations that describe the circuit and do a mathematical analysis, which is really cheap.

In this exercise we are going to take the simulation approach and in the next, build the circuit and test it with instruments.

### Goals

There are two goals for this lab:

- Observe by simulation the time-domain response of a simple RC circuit
- Learn to use Multisim® so we can use it to simulate other circuits in this course.

### Introduction

The circuit for this lab is the RC circuit of Figure 1. We will study this by applying a known voltage waveform and observing the output. Our interest is in the *time-domain* response of this circuit, which is the response as a function of time. You will later learn about the frequency response of a circuit. We are going to do this using a software simulator called *Multisim*, a program that can simulate both our simple circuit and much more complex circuits, both analog (like ours) and digital.

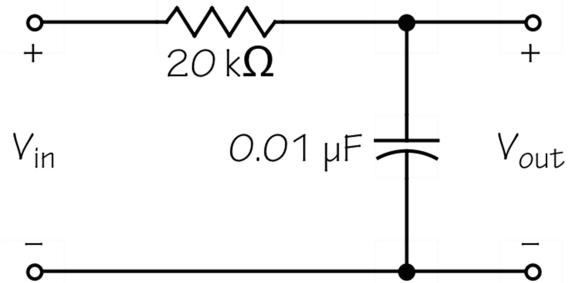


Figure 1: RC Circuit

The Multisim software is part of a circuit design package that comes in the National Instruments system that also supports the *myDAQ*® interface that turns your laptop into a suite of electrical instruments. We'll use *myDAQ* in the next lab exercise. You have already experienced both Multisim and *myDAQ* in ECE 210.

This lab will be a refresher for Multisim and also a refresher for first order circuit response.

### Procedure

There are four steps for carrying out this lab:

- Create the circuit description graphically.
- Simulate the circuit's response to a step function.
- Simulate again under improved input conditions
- Determine the circuit's time constant.

#### A. Create the circuit for simulation

The first job when simulating via Multisim is to draw a graphical model of the circuit by placing circuit elements in your drawing, adding an input source and an instrument for measuring output, and connecting them all together.

1. Launch Multisim via All Programs→National Instruments→Circuit Design Suite 11.0→Multisim 11.0.
2. Start a file to save your work where you can find it again: File→Save as \_\_\_\_\_
3. Select Place→Component...
4. In the new window, in **Group:**, select Basic—take a moment to read the resultant list because you'll need some of these items in later work
5. In the list of basic parts, select **RESISTOR** and double-click the name, which will load your cursor with a resistor to place in your circuit drawing.
10. Likewise, select and place VCC.
11. Dismiss the Components window.
12. Double-click on the resistor symbol to bring up the **Value** table; change the **Resistance** to 20k and click **OK**.
13. Double-click on the capacitor symbol and change its value to 0.01u where *u* is used for  $\mu$  for *mico*.
14. Double-click on **VCC** and change its value to 1 V.
15. Wires are added by placing the cursor on a terminal of a circuit element, where the cursor will change to cross hairs, clicking, and then clicking on the destination terminal. Your circuit should end up looking like Figure 3.

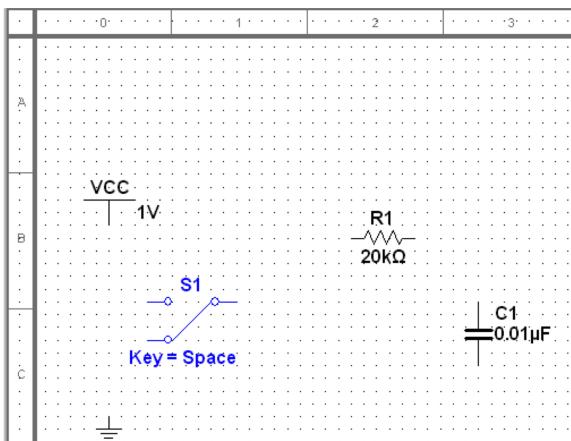


Figure 2: Initial Element Placement

6. (See Figure 2.) Place this resistor in your circuit by positioning the loaded cursor where you want the resistor and clicking (we'll set its value later).
7. Similarly, load the cursor with a capacitor, but use **Control-R** to rotate it before placing it as shown in Figure 2.
8. Select **Switch** and then **SPDT** (single-pole double-throw) and add it to your design, using **Control-R** twice to flip it around.
9. In the Components widow, change **Group:** to **Sources**, then select **POWER SOURCES**, and double-click on **Ground** so you can place it in your circuit.

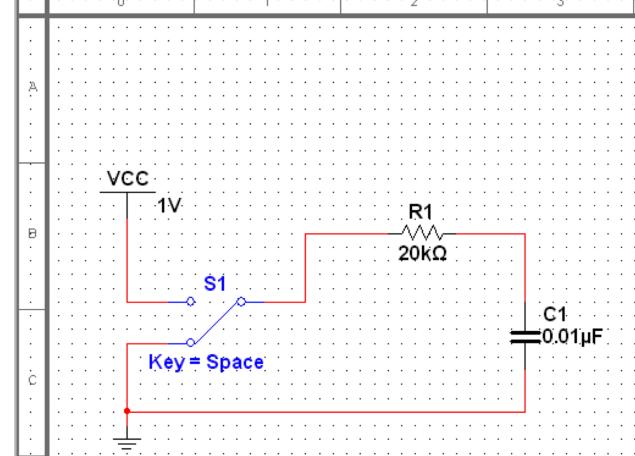


Figure 3: Wiring

Now the circuit is described to Multisim and we are ready to simulate its operation.

## B. First Simulation

We need to be able to observe the results of our simulation, so we'll add an oscilloscope to see the response as a function of time. Then we'll flip the switch back and forth to simulate a step function and see what happens.

1. **Simulate→Instruments→Oscilloscope**  
will load your cursor with a scope. Position it a little above the resistor to leave room for connections, as shown in Figure 4.

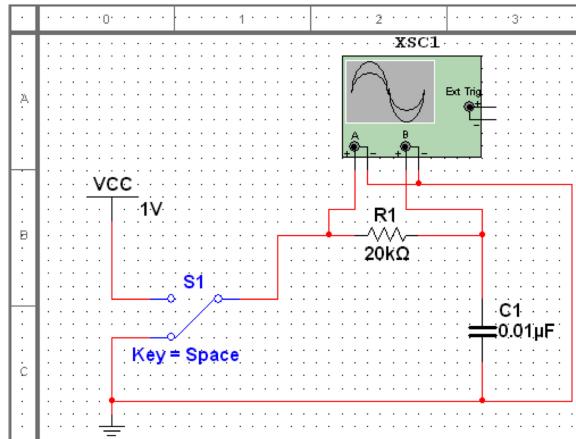


Figure 4: Scope Connections

2. Connect the scope to the circuit.
  - a. **A+** to the left end of the resistor (which is  $V_{in}$ ).
  - b. **B+** to the top of the capacitor (which is  $V_{out}$ ).
  - c. Both **A-** and **B-** to ground (bottom of circuit).
3. Play with the switch for a moment by clicking your cursor somewhere in the drawing space, then pressing the space bar to flip the switch back and forth.
4. Double-click on the scope to bring up the real scope display (see Figure 5).
5. Set the scope's parameters:
  - a. **Timebase** =  $200\mu s/Div$
  - b. Both **Channel A** and **Channel B** =  $500 mV/Div$
  - c. **Trigger Level** = 0.5 V.
  - d. Select **Normal**

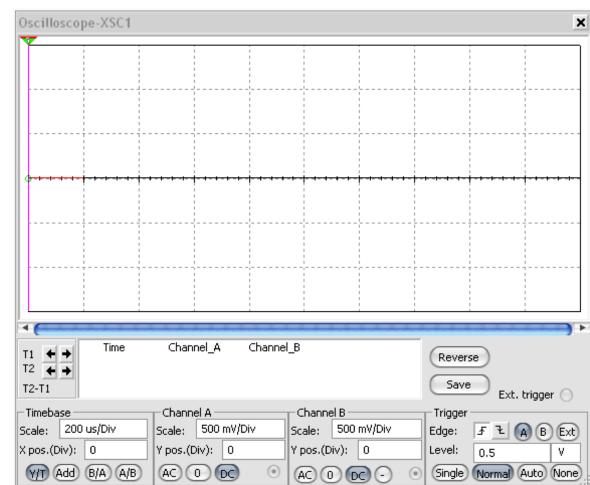


Figure 5: Scope setup

6. **Simulate→Run** will set Multisim on the task of running the circuit.
7. Press the space bar to flip the switch back and forth to drive the circuit with steps and see what happens on the scope.

Umm, you don't see a good scope picture of what's happening? Can you determine what the output waveform looks like? Can you measure the time constant? No?

This is not a good way to make observations. The step isn't wrong, but it isn't timed so the waves display in a usable way on the scope. Our simulation will produce better, more observable results if we drive the circuit with a repeating step function. We'll get this by using a square wave as the input.

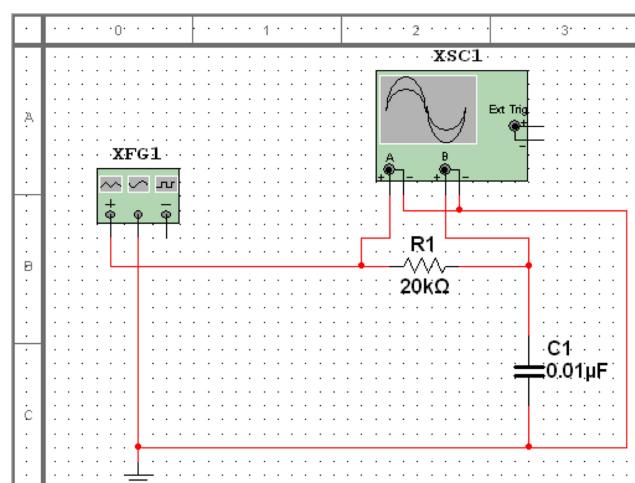


Figure 6: Square Wave Source

### C. Step-function simulation

Changing the input to a repeating step function, which is a square wave, will give us a repetitive scope display where we can measure the time constant of this RC circuit.

1. **Simulate→Stop** (or use the red button above your circuit).
2. Delete the switch and VCC by clicking on each and then pressing the Delete key.
3. **Simulation→Instruments→Function Generator** and place it in your circuit (see Figure 6).
4. Connect the function generator as shown in Figure 6: the + terminal goes to the resistor and the common (middle) terminal goes to ground.
5. Double-click on the function generator and set its values as shown in Figure 7:
  - a. Click the square-wave button
  - b. **Frequency** = 500 Hz
  - c. **Duty cycle** = 50
  - d. **Amplitude** = 0.5 Vp (peak)
  - e. **Offset** = 0.5 V
6. **Simulate→Run** or the green triangle above your circuit. Your scope should look like Figure 8.

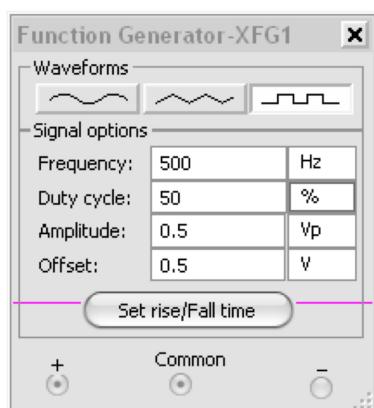


Figure 7: Function Generator Setup

Now you can see the step and the resulting exponential. If you want to play for a moment, stop the simulation. Then double-click on the wire that connects the right end of the resistor and the scope's B channel. Change the wire's color to green, restart the simulation, and see what happens on the scope.

### D. Measure the time constant

Recall from class the time constant is the time it takes for the exponential to fall from its starting value to  $1/e = 0.368$  of its starting value. First, though, we need to readjust the scope for a better picture.

1. Stop the simulation if it's running
2. Change some of the scope settings:
  - a. At the bottom right, click **Single**
  - b. Change the **Timebase** to  $100 \mu\text{s}/\text{Div}$
  - c. Click on the falling-edge (downward-going) symbol.
3. Start the simulation
4. The exponential starts at 1 volt, so we want the time when it reaches  $0.368$  volts. We'll use the T1 cursor on the scope
5. Near the left edge of the scope below the display, click T1's right arrow to move that cursor into the exponential
6. Stop the simulation
7. Move the cursor until you get the Channel B voltage as close to  $0.368$  as you can (see Figure 9)
8. Note the time shown for the T1 cursor. This is the time to reach  $1/e = 0.368$  of the starting value, so this time is the time constant,  $\tau$ , of this RC circuit

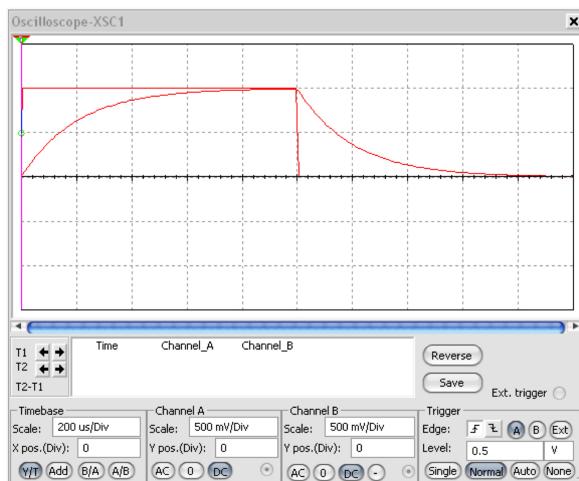


Figure 8: Output from Square Wave

## The End

Did we meet our goals? We observed the exponential response of a simple RC circuit and measured its time constant. We simulated a circuit using Multisim and learned something about creating and simulating circuits. Yes, we met the goals.

Multisim is pretty easy to use for creating a

circuit and then simulating its operation under different stimuli. You will probably find it harder to locate various parts in the menus than to create the circuit and run the simulation.

There are some tutorials and help files in the Circuit Design Suite 11.0 package (in Documentation) that you might find useful. We'll be using Multisim a number of times in this course.

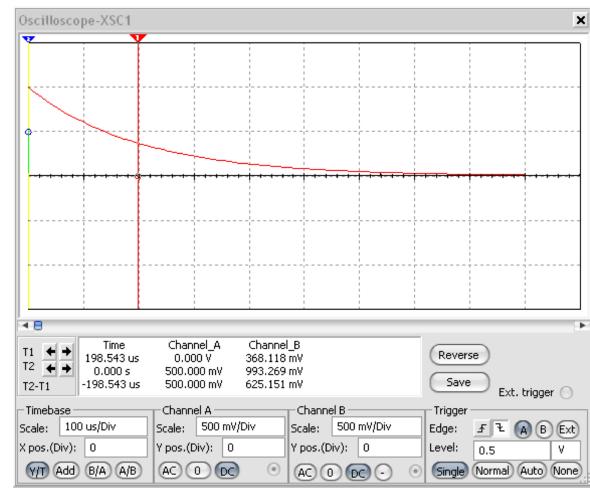


Figure 9: Time Constant Measurement