

Lab 4: Multisim and the Oscilloscope

OBJECTIVES

- Learn new Multisim components and instruments.
- Conduct a Multisim transient analysis.
- Gain proficiency in the function generator and oscilloscope.

MATERIALS

- Multisim screen shots sent to TA.

INTRODUCTION

So far we have used Multisim to simulate simple DC circuits. In this lab we will work with new components, instruments, and analyses. We will introduce dependent sources in DC circuits. On the AC side, we will work with the transient analysis, function generator, and oscilloscope.

Dependent Sources

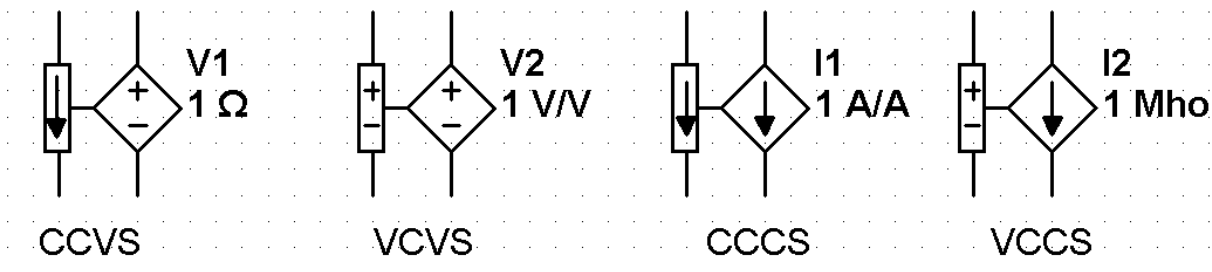


Figure 1 – Dependent sources depicted by Multisim. CCVS = current-controlled voltage source, VCVS = voltage-controlled voltage source, CCCS = current-controlled current source, and VCCS = voltage-controlled current source.

To place a dependent source, go to *Place > Component* and select the *Sources* group. They are found under the headings *CONTROLLED_CURRENT_SOURCES* and *CONTROLLED_VOLTAGE_SOURCES*. As can be seen in Fig. 1, a dependent source has two components. The actual source is on the right hand side. The device on the left hand side acts as a voltmeter or ammeter and measures the quantity which determines the dependent source's output. For the VCVS and CCCS, the quantity 1 V/V and 1 A/A represent the voltage and current gains, respectively. For the CCVS, 1 Ω represents the transresistance, or ratio of output voltage to input current. For the VCCS, 1 Mho is the transconductance, or ratio of output current to input voltage. The unit *Mho* for transconductance is the name for Ω^{-1} . These parameters are the multipliers of the controlling value and must be set to the appropriate multiplier value.

Consider for example, the circuit in Fig. 2. In this case, we have a CCCS which produces a current that is 4 times the value of the current i . In Fig. 3, we have the Multisim implementation of this circuit. Notice that the current i has been redirected through the "meter" of the controlled source. It is important to ensure that the current passes through the "meter" in the same direction as the arrow. The gain has been set to 4. Two handy commands for positioning the CCCS are *alt + X* for flipping the component horizontally and *ctrl + R* for rotating the component.

Lab 4: Multisim and the Oscilloscope

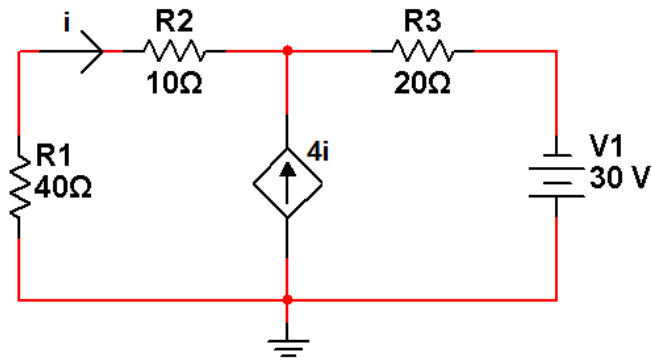


Figure 2 – Example circuit for a CCCS.

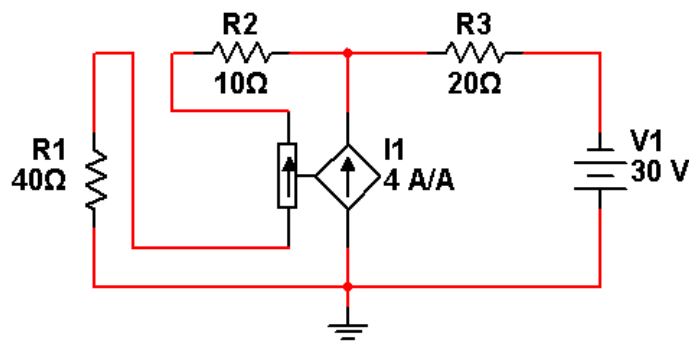


Figure 3 – Multisim implementation of the CCCS.

AC Circuits

Up to this point we have only dealt with DC circuits. In an AC circuit, the source voltages or currents vary with time. In Multisim we can find the AC sources under *Place > Component > Group: Sources > SIGNAL_VOLTAGE_SOURCES*. Consider the circuit in Fig. 4. This circuit has a sinusoidal voltage source *AC_VOLTAGE*, with a 1 V peak amplitude, a frequency of 1 kHz, and a 0 degree phase angle. Other properties of this source can be set by double clicking on it. Suppose we would like to see how this circuit behaves over three periods.

Lab 4: Multisim and the Oscilloscope

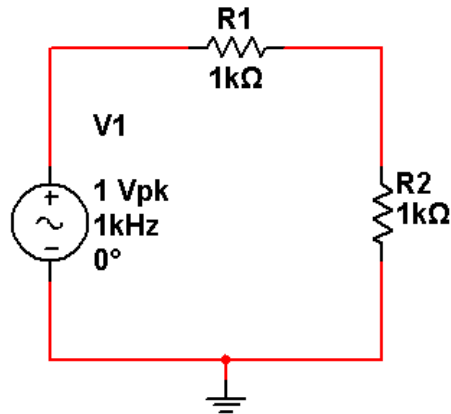


Figure 4 – Example AC circuit.

This can be accomplished with a transient analysis. Select *Simulate* > *Analyses* > *Transient Analysis* to bring up the transient analysis menu. First, we set the analysis duration by setting the start time to 0 s and the stop time to 0.003 s (three periods [since $f = 1 \text{ kHz} = 1000 \text{ Hz}$, then $T = 1/f = 1 \text{ ms} = 0.001 \text{ s}$]). Next, in the *Output* tab, we choose which variables are to be plotted against time. Selecting $V(1)$ and $V(2)$ designates the input voltage and the voltage across resistor 2. Note that we can also plot algebraic combinations of the circuit variables by selecting *Add Expression*. For instance, the voltage across R1 is $V(1) - V(2)$. Finally, after pressing *Simulate*, a plot is output, as can be seen in Fig. 5.

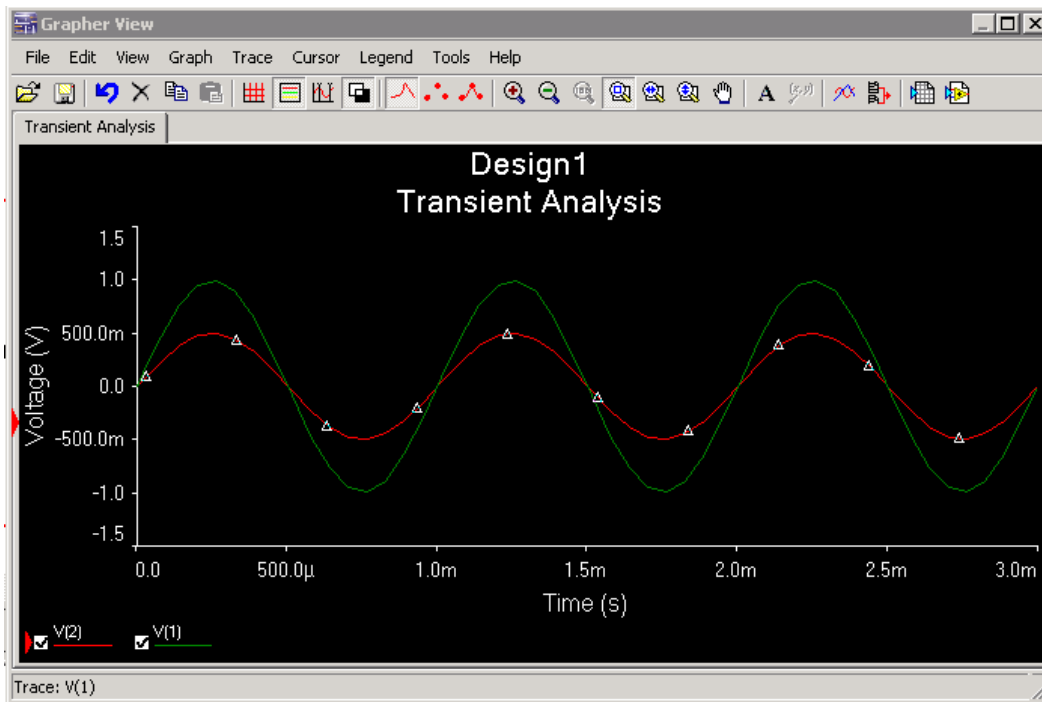


Figure 5 – Transient analysis result. Input voltage in green, output in red.

Lab 4: Multisim and the Oscilloscope

In a lab setting, our AC voltage source is called a *function generator* and we view AC voltage waveforms with an *oscilloscope* (often abbreviated as oscope). Multisim provides these tools (*Simulate > Instruments*), so we can repeat the above procedure as we would with real lab instruments. In Fig. 6, the circuit is now supplied with a sinusoidal waveform from the function generator. The oscilloscope has two channels, and each channel is dedicated to one of our desired voltages. The leads of the oscilloscope are placed in parallel with the desired voltage, just as we would with a voltmeter.

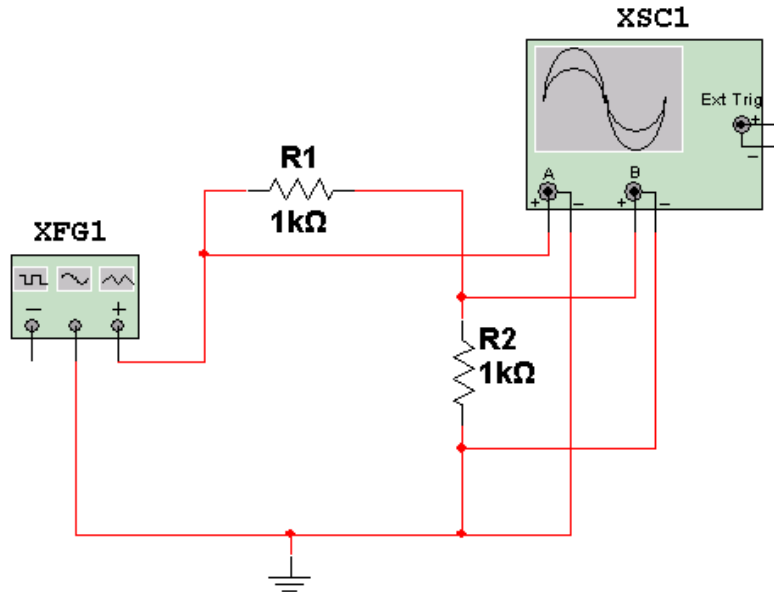


Figure 6 – Circuit of Fig. 4 as we would construct in the lab.

Double clicking the function generator, we can set its parameters to that of the AC voltage source from our previous example. The menu can be seen in Fig. 7. The sinusoidal waveform has been selected, the frequency set to 1 kHz, and the peak amplitude set to 1 V.

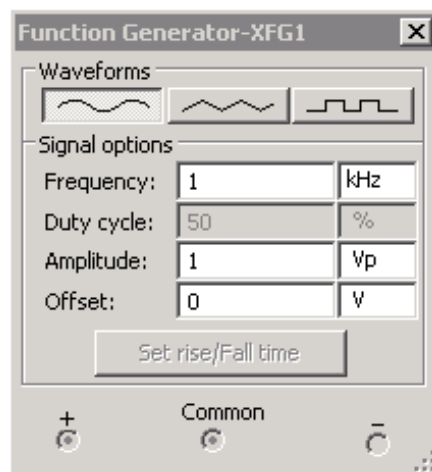


Figure 7 – Function generator menu.

Lab 4: Multisim and the Oscilloscope

Opening the oscilloscope, and running the circuit, the two waveforms are plotted in real time. The simulation is paused and a screen shot is shown in Fig. 8.

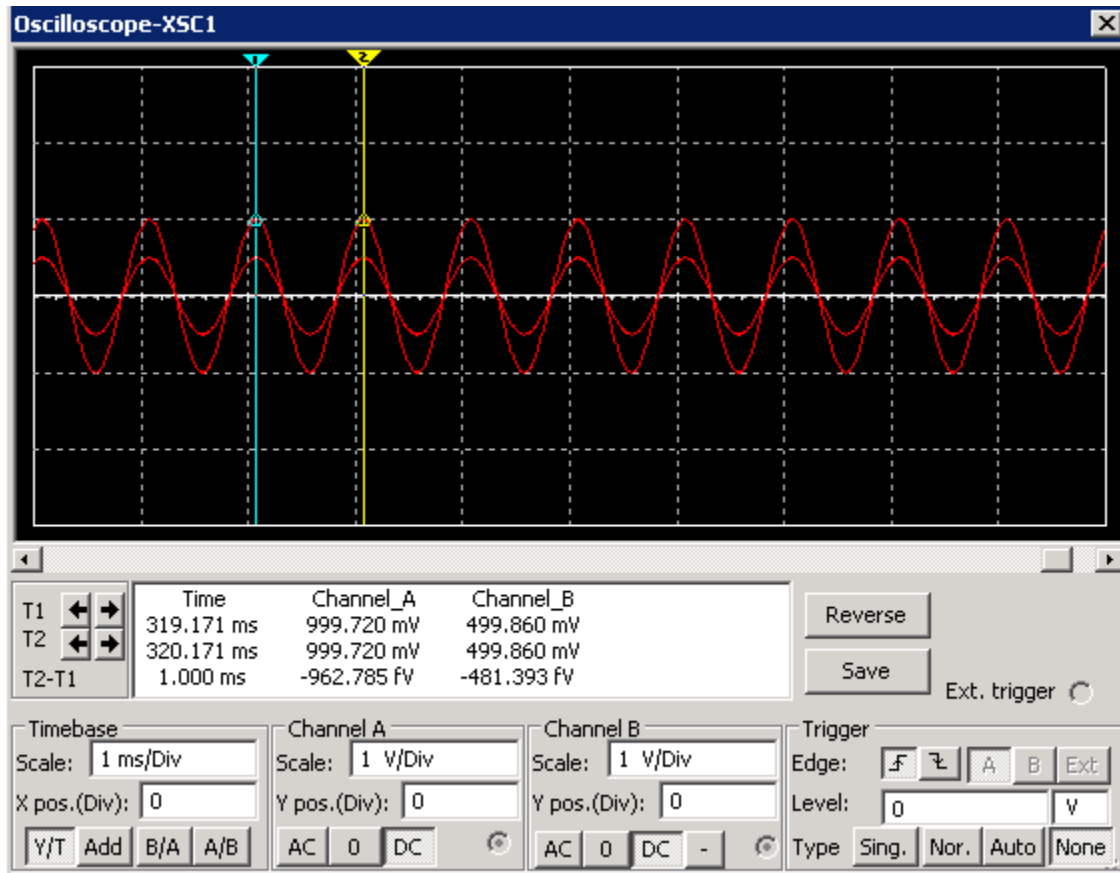


Figure 8 – Multisim oscilloscope.

Let us examine Fig. 8. The oscilloscope plots voltage in the y-axis and time in the x-axis. The timebase is 1 ms/Div, meaning that the length of a grid block is 1 ms. This can be adjusted to get a good view of the signal. Channel A contains the input voltage. The scale has been adjusted to 1 V/Div, meaning that the height of a grid block represents 1 V for the channel A signal. As expected, the source voltage peaks at 1 V, just as we instructed it. Channel B contains the voltage across R2, and has the same scale as channel A.

The cursors are the blue and yellow vertical lines. They trace along the waveform, and the value of the signal at the point of the cursor is given in the big white box. Cursor 1 is at time 319.171 ms, where the Channel A signal is 999.720 mV. The second row lists the data for cursor 2. The third row contains data pertaining to the differences between the two cursor points. Notice that T2-T1 is 1.000 ms. Because the two cursors are at adjacent peaks, the time difference between them is the period. By right-clicking, the cursors can be set to trace along either waveform.

Lab 4: Multisim and the Oscilloscope

PRE-LAB AND QUESTIONS

1. Build the circuit in Fig. 9 in Multisim and measure the node voltage V1 relative to ground. Take a screen shot of your circuit.

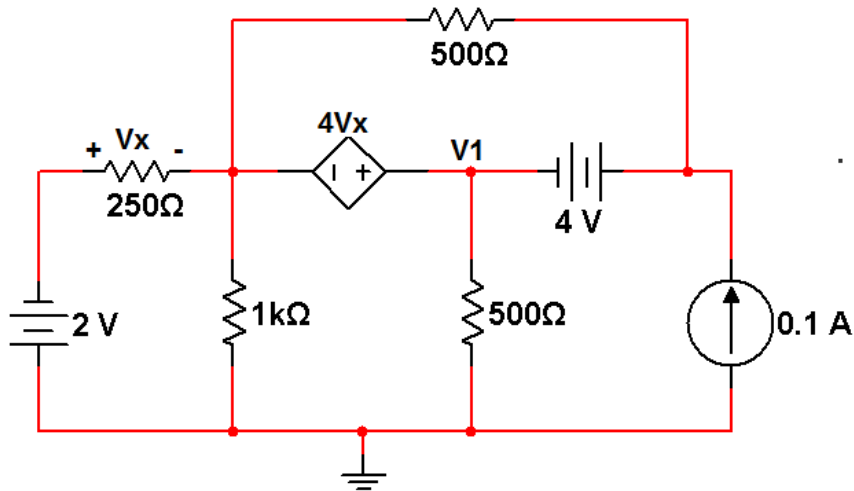


Figure 9 – Circuit for pre-lab question 1.

2. Build the circuit in Fig. 10 in Multisim, using the *AC_VOLTAGE* component.
 - a. Determine the period of the input signal.
 - b. Run a transient analysis for a duration equal to 3 periods of the input signal. In the plot include the input voltage and the voltage across the capacitor. Provide a screen shot.
 - c. Now attach the oscilloscope channel A to the input and channel B to the capacitor voltage. Run the circuit, scale the oscilloscope appropriately, and provide a screen shot.

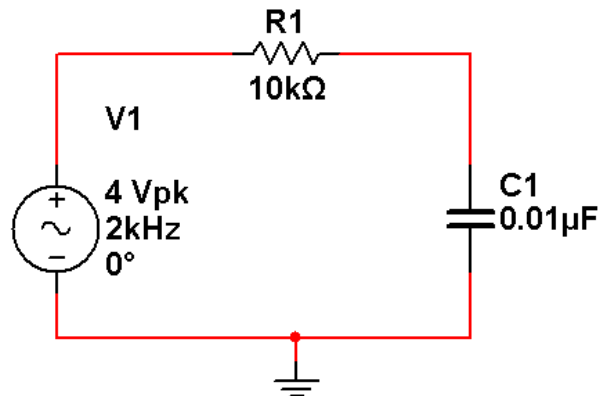


Figure 10 – Circuit for pre-lab question 2 and lab procedure.

LAB PROCEDURE AND QUESTIONS

From the ELVIS menu, open the function generator. The variable power supply only supplies DC voltages, so we use the function generator to generate time-varying periodic voltage signals. The function generator interface is seen in Fig. 11. We can set the frequency, select a waveform, set the

Lab 4: Multisim and the Oscilloscope

amplitude, and add a DC offset (the offset is equivalent to adding a constant to a sinusoid, which shifts the wave up or down).

The output of the function generator is designated *FUNC_OUT* on the left hand side of the ELVIS board, and offers 4 slots for wires. This voltage is internally grounded, just as with the DC supplies.

The maximum amplitude for the function generator is 2.5 V. Use the large amplitude waveform generator (available at the top of the function generator menu), to produce a larger signal. The large amplitude waveform generator is output from the slots marked *DAC0*.

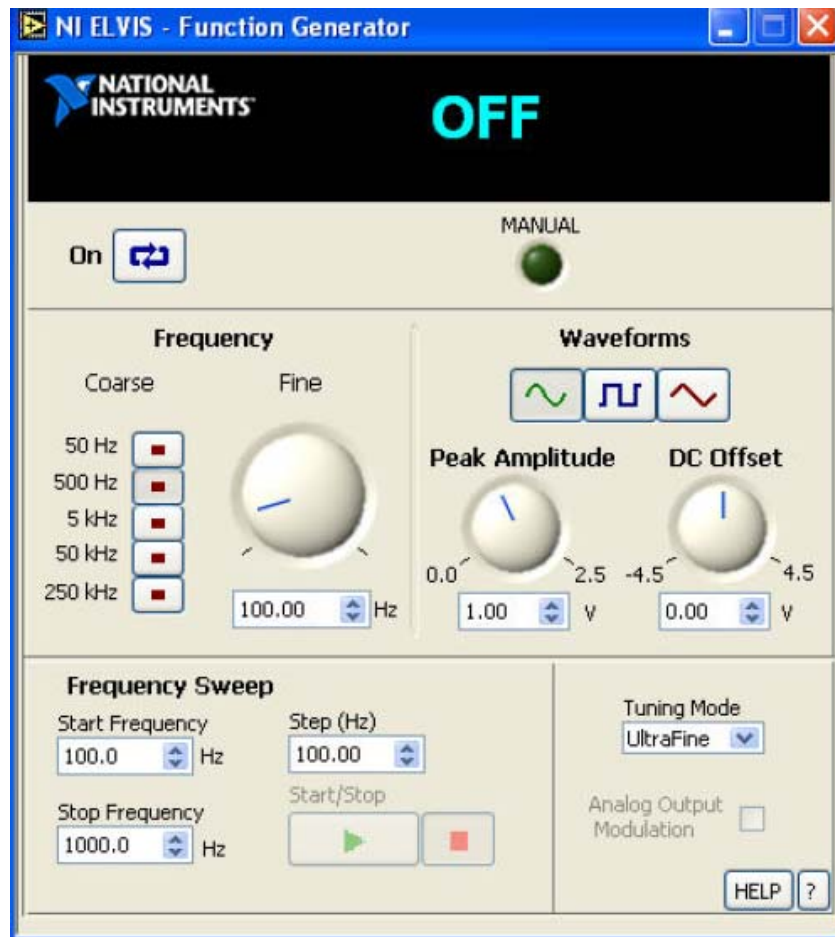


Figure 11 – Function generator interface.

1. Build the circuit from Fig. 10 on your ELVIS board. The $0.01 \mu\text{F}$ capacitor is labeled with the code 103 on it. Use the large amplitude waveform generator because the function generator cannot supply a 4 V amplitude. (Remember output is from *DAC0*.)
2. Open the oscilloscope from the ELVIS menu. This is a two channel oscilloscope, just like the Multisim scope. Let's put the input voltage on channel A and the output on channel B. On the top left of the ELVIS board are the oscilloscope inputs. To measure the input, connect a wire from

Lab 4: Multisim and the Oscilloscope

channel A+ to *DAC0* and a wire from channel A- to ground. In the same way, make the connections so that channel B measures the output.

3. Make sure the display for both channels A and B is toggled on. Adjust the two channels to have the same scale. Adjust the timebase so that we can see 2 or 3 periods of the signals. Select *Single* to freeze the scope so that we can work with it.
4. Below the screen the scope displays the root-mean-square voltage, the frequency, and the peak-to-peak voltage of the signal on each channel. It also displays the voltage value of each cursor, and the time difference between them.
5. Sketch the two signals.
6. Set cursor C1 to channel A and cursor C2 to channel B. Use the cursors to measure the delay ΔT between the two signals. Calculate the phase angle Φ , between the two signals:

$$\Phi = \Delta T * frequency * 360^\circ$$