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

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 \( \Omega \) 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 \( \Omega^{-1} \). 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 kHz = 1000 Hz, then T = 1/f = 1 ms = 0.001 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.

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.

![Function Generator Menu](image)

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.

![Multisim oscilloscope](image)

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.

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.](image)

2. Build the circuit in Fig. 10 in Multisim, using the \textit{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.](image)

3. Start the NI myDAQ Oscilloscope and Function Generator by selecting ‘Scope’ and ‘FGEN’, respectively, from the NI ELVISmx Instrument Launcher.
   a. \textbf{Generate} a 4Vpk 2kHz Sine Wave by connecting the Function Generator (AO 0, AGND) to the Oscilloscope (A1 0+ and A1 0-) on your myDAQ device and adjusting the necessary settings on
Lab 4: Multisim and the Oscilloscope

the Function generator interface shown in Fig. 11. (Note: A1 1+ and A1 1- can also be used for the Oscilloscope.)

b. **Display** 5 periods of the signal above by adjusting the ‘Timebase’ (Time/Div) on the Oscilloscope interface and clicking ‘Stop’. Explain why the resulting Time/Div gives 5 periods. Take a screenshot of the measurement.

c. Repeat part a) for a 4Vp-p 2 kHz Square wave and this time display 4 periods. Email all screenshots to the TA before your lab.

d. Build the circuit of Fig. 10. Show this circuit it to your TA at the beginning of the lab. (Note: You will also have to complete this exercise during lab on the ELVIS machine.)

i) A1 0+ and A1 0- or A1 1+ and A1 1- are the myDAQ oscilloscope inputs. To measure the input, connect a wire from A1 0+ to AO 0- and a wire from A1 0- to ground. In the same way, make the connections so that Channel 1 measures the output.

ii) Make sure the display for both Channels 0 and 1 is enabled on the Oscilloscope. 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 ‘Stop’ to freeze the scope so that we can work with it.

iii) 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. Sketch the two signals.

iv) Set cursor C1 to channel A and cursor C2 to channel B. Use the cursors to measure the delay \( \Delta T \) between the two signals. Calculate the phase angle \( \Phi \), between the two signals:

\[
\Phi = \Delta T \times \text{frequency} \times 360^\circ
\]

**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 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 **FGEN** 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.

1. Build the circuit from Fig. 10 on your ELVIS board, just as you built it on your myDAQ breadboard in the pre-lab.
Lab 4: Multisim and the Oscilloscope

2. Open the oscilloscope from the ELVIS menu. This is a two channel oscilloscope, just like the Multisim scope. Put the input voltage on Channel 0 and the output voltage on Channel 1. Connect CH0 to BNC1 and CH1 to BNC2; these are on left side of the ELVIS board. BNC1+ and BNC1- (or BNC2+ and BNC2-) are the ELVIS oscilloscope inputs. To measure the input, connect a wire from BNC1+ to FGEN and a wire from BNC1- to ground. In the same way, make the connections so that Channel 1 measures the output.

3. Make sure the display for both Channels 0 and 1 is enabled on the Oscilloscope. 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 ‘Stop’ 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. Sketch the two signals.

5. Set cursor C1 to channel A and cursor C2 to channel B. Use the cursors to measure the delay $\Delta T$ between the two signals. Calculate the phase angle $\Phi$, between the two signals:

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