Lab 1: Introduction to the Oscilloscope and the Function Generator

Objectives

In this lab you will learn the basics of operating a function generator and an oscilloscope, two instruments that are used extensively in circuits and electronics. Also, you will learn to build a simple op-amp circuit on a breadboard and analyze this circuit using the oscilloscope. Finally, you will alter your circuit based on a design constraint.

Pre-Lab Instructions (10pts)

Downloads

1. Download the *Tektronix TDS1000 and TDS2000 Series Oscilloscopes Operator Training Kit* from the course web site. (You do <u>not</u> need to print it: It's really long, and there are copies in the lab room.) **Read** the pages listed below, and more if you are interested:

- **1.1–1.9, 1.14–1.25, 1.29**
- **2.1–2.5, 2.13–2.15, 2.20–2.32, 2.37–2.49, 2.67–2.69, 2.88–2.89**

2. Download and print the *Electronics Lab Parts List* from the course web site.

Calculations

1. Compute theoretical values for the quantities requested in Part Five. Work out the solutions on a separate sheet of paper to submit, then write the answers in the spaces provided for the "In-Lab" part of the assignment.

2. Verify your calculations above using PSpice. The schematic for every circuit needs to be printed out. Be sure to include pins for the input and output voltages. In addition, print out the input/output waveforms for parts 5.1. For parts 5.2 and 5.3, be sure to print the output file (with the total power dissipated boxed). (**NOTE: your name must appear in the filename of all circuit and waveform printouts!**)

Simulating an AC Voltage Source in PSpice

1. Load the "Source" library to the parts list and select "VSIN."

2. Double-click on each parameter of the voltage source to set each value. Set "VOFF" to zero, "VAMPL" to whatever the amplitude of the sine wave should be in volts, and "FREQ" to the desired frequency in hertz.

3. After adding the other elements in the circuit, place a voltage marker where the input voltage is defined for the circuit and another marker where the output voltage is defined.

4. Create a new "Time Domain (Transient)" simulation. Set the "Run to Time" to something that would display about 5–10 cycles of the waveform. For example, to display 5 cycles of a 10kHz signal, set the "Run to Time" to

$$5 \text{ cycles} * (1 \text{ sec} / 10{,}000 \text{ cycles}) = 500 \mu s$$

5. Run the simulation. The input and output waveforms should be displayed.

In-Lab Instructions (15pts)

Initial Setup
      1. This lab involves analysis of AC signals. In order to remove any DC bias from these signals when measuring them, the oscilloscope should be set to AC coupling for both CH1 and CH2. Press the CH1 menu button (yellow), and ensure that the coupling is set to "AC." If it is "DC" or "GROUND," push the button next to where the coupling is displayed on the screen, and it should change to "AC." Do the same for CH2.

      2. Inspect the two probes you will be connecting to CH1 and CH2 of the oscilloscope. **Make sure that the PROBE type, displayed when either the CH1 menu button or the CH2 menu button is pressed, matches the type of probe you are using (they will be either 1X or 10X).**

Part One: Probe Check
      1. Connect a probe to CH1 of the oscilloscope. Attach the probe ends to the PROBE COMP terminal on the oscilloscope. Press the round PROBE CHECK button. A 1kHz square wave with an amplitude of 5V should show up on the screen (adjust the VOLTS/DIV, SECS/DIV, and TRIGGER until a steady square wave is displayed on the screen).

Have the TA initial here when finished _____

Part Two: Measuring the Output of the Function Generator
      1. Set the frequency dial on the function generator to 1.2. Push the button marked "1k" on the range setting, then push the button that looks like a sinusoid wave. This will generate a 1.2kHz sinusoidal wave. Connect a BNC-to-alligator clip cable to the output of the function generator. Attach this cable directly to the probe coming from CH1 on the oscilloscope. Set the VOLTS/DIV on the oscilloscope to 1V/div, and adjust the amplitude on the function generator until a sine wave with a peak-to-peak voltage of 5V is seen. If you have an older function generator, you may have to depress the "VOLTS OUT" button to get up to 5 volts. The peak-to-peak amplitude of the waveform is determined by counting the number of divisions the waveform spans vertically from its lowest point to its highest point, and multiplying this number by the VOLTS/DIV shown on the screen.

      2. Using the "Measure" button on the oscilloscope, record the frequency and the peak-to-peak amplitude shown on the screen. If you do not have frequency and peak-to-peak on the screen press one of the five rectangular buttons just to the right of the screen and select the channel you want to measure and the type of measurement you want. You can have up to five different measurements on the screen at a time. Once you have selected, press the "Measure" button.

Frequency _____   Amplitude (peak-to-peak) _____

How does the frequency setting read from the dial on the function generator compare to the measured value read from the oscilloscope?

What happens when the range setting is changed to "100k"? What happens when it is changed to "10"? What needs to be adjusted to see these waveforms correctly?

3. Change the range back to "1k" on the function generator so you again have frequency of 1.2kHz. Determine the peak-to-peak amplitude as described in section 2.1 above and record the result below. Counting the number of divisions that comprise one cycle of the waveform and multiplying this number by the SECS/DIV shown on the screen, one can obtain the period of the waveform. Taking the inverse of the period will yield the frequency. Determine the frequency and record the result below.

Frequency _____   Amplitude (peak-to-peak) _____

4. Percent error is defined as:

% Error = expected value – measured value * 100
                    expected value

Calculate the percent error between the frequency and amplitude measurements made in the last two parts (where the expected value is the value calculated above, and the measured value what is read from the scope).

Frequency % Error _____   Amplitude % Error _____

What are some possible sources of this error?

5. Set the VOLTS/DIV to 50mV, and next generate a 100mV peak-to-peak sine wave with a frequency of 1kHz. If the amplitude knob on the function generator changes the waveform too much to accurately generate 100mV peak-to-peak, try depressing the "VOLTS OUT" button on the function generator. This will change the output voltage range from 0-20V to 0-2V. Now change the scope setting to the VOLTS/DIV shown below, and record the peak-to-peak amplitude (using the "Measure" button).

50mV/DIV _____

100mV/DIV _____

200mV/DIV _____

500mV/DIV _____

1V/DIV _____

Why does the reading change? Does the frequency reading change as well?

Part Three: Analyzing the Difference between a Sine Wave and a Square Wave
      1. From the function generator produce a sinusoidal waveform with a peak-to-peak voltage of 5V and a frequency of 5kHz (change the VOLTS OUT back to 0-20V). On the function generator, press the button that generates a square wave. Record the peak-to-peak voltage as well as the frequency of the square wave.

Frequency _____   Amplitude (peak-to-peak) _____

Part Four: Root-Mean Square (RMS)
      1. The RMS value of an AC voltage (or current) is the effective value of the sinusoidal signal. This means that if a DC voltage (or current) with that effective value was put across (or through) a resistor, the power dissipated would be the same as the power dissipated if the AC voltage (or current) source were used. **The RMS value of a sine wave is the amplitude of the sine wave (which is half of the peak-to-peak voltage) divided by the square root of two.** Calculate the RMS value of the sine wave generated in part 3.1. A square wave has an RMS value equal to its peak voltage (Note: not peak-to-peak). Calculate the RMS value for the square wave measured in part 3.1.

Calculated RMS (sine wave) _____

Calculated RMS (square wave) _____

      2. Using the "Measure" button on the oscilloscope, record the displayed RMS value for the sine wave and the square wave. If you do not have RMS measurements on the screen, follow the instructions in part 2.2 to select RMS measurements.

Measured RMS (sine wave) _____

Measured RMS (square wave) _____

      3. Calculate the percent error between the previous two results.

RMS % Error (sine wave) _____

RMS % Error (square wave) _____

Part Five: Analysis of an Op-Amp Circuit
      1. Consider the op-amp circuit in Figure 1. Compute the ratio Vout/Vin and record your values for Vin, Vout, and Vout/Vin in the spaces provided at the bottom of the page. The ratio Vout/Vin is called the *gain*. Next, simulate this circuit in PSpice with a 2V (peak-to-peak) 1kHz source for Vin and compute the gain. For this simulation, the ideal OPAMP will suffice. (You will need to place a large resistor, say 100k or 1M, connecting Vout to ground.) Record your values below.
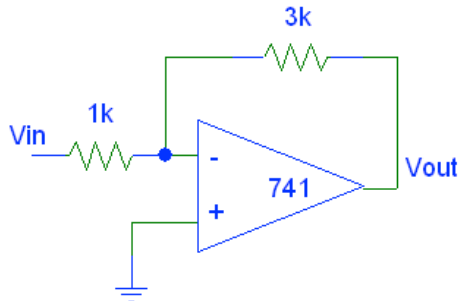

Figure 1: Op-Amp Circuit with a Fixed Gain

      Construct the same circuit using a breadboard. The pin layout of the LM741 op-amp is shown in Figure 2. To wire the circuit as shown in the symbolic diagram of Figure 1, follow the wiring diagram shown in Figure 3. The 15V voltage source will come from the variable voltage source on the breadboard. The 1V (amplitude) sine wave will come from the function generator (the frequency can be set to 1kHz).
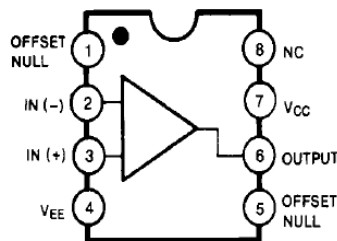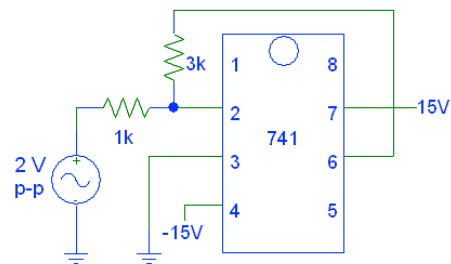

Figure 2:
Pin Layout of the LM741 Op-Amp

Figure 3:
Wiring Diagram for the Op-Amp Circuit

      Build the circuit, but **before connecting the power, have a TA check the wiring of your circuit**.

      Once the circuit is wired as shown in Figure 3, connect a probe from CH1 on the oscilloscope to the input voltage (coming from the function generator). Connect a second probe from CH2 on the oscilloscope to the output voltage (pin 6 of the op-amp). Display both readings on the oscilloscope. Record the peak-to-peak voltages below and compute the gain.

Quantity [units]	Hand Calculation	PSpice Simulation	Circuit Measurement
Vin [Vpp]			
Vout [Vpp]			
gain (Vout/Vin)			

1. (continued)

Looking at the positive and negative peaks of both signals, why is this op-amp considered to be "inverting"? What is the phase difference between Vin and Vout?

2. Replace the 2V (pp) AC source with a 1V DC source. We do this to check the maximum power dissipated by the circuit. Run the simulation and this time check the output file (under the <u>V</u>iew menu). Near the bottom you should see the Total Power Dissipation. Record your value here.

Total Power Dissipation _____

3. Now, minimize power consumption while keeping the same gain across the amplifier, and also keeping Vin and Vout the same. Change only the two resistors, and be sure to only use resistors available in the lab for your simulation. (This is where the Electronics Lab Parts List is useful.) Design for a specification of 0.2mW. (In other words, make sure the power dissipated is less than this value.) Indicate the values you have chosen below.

R1 (which used to be 1k) _____

R2 (which used to be 3k) _____

4. Simulate the circuit in PSpice with the values chosen (and a 1V DC source), and verify that the gain has not changed and also that the design constraint has been met. Record the Total Power Dissipation from the output file here, and then fill out the appropriate spaces in part 6 on the next page.

Total Power Dissipation _____

5. Before constructing the circuit, let's make sure you have the right resistors. Use the multimeter to measure each value of resistance and then compute the percent difference as compared to its labeled value. Comment on whether or not these resistors are within the labeled tolerances.

R1, tolerance _____

R1, measured _____

R1, % error _____

R2, tolerance _____

R2, measured _____

R2, % error _____

6. Construct the circuit with the values chosen, now using the 2V (pp) AC source, and verify that the gain is the desired value. Measure Vin and Vout, and compute the gain.
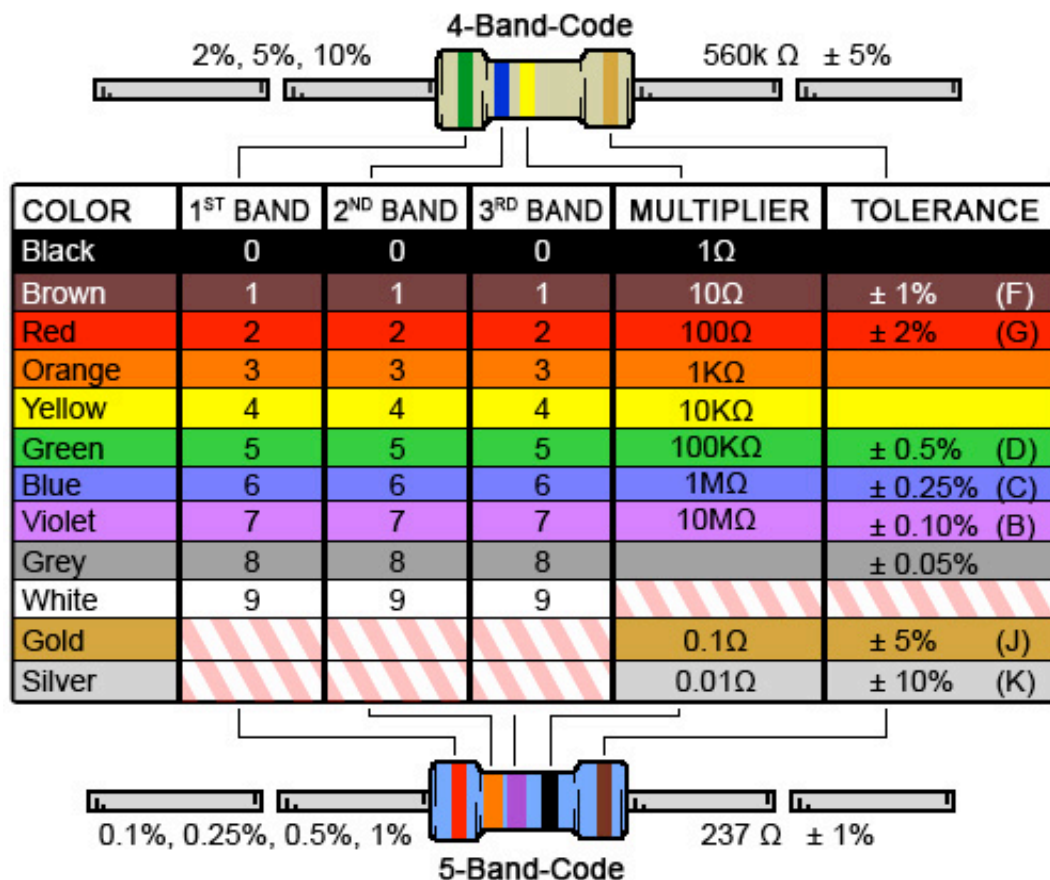
Quantity [units]	Hand Calculation	PSpice Simulation	Circuit Measurement
Vin [Vpp]			
Vout [Vpp]			
gain (Vout/Vin)			

When you have completed the lab, sign and print your names below and have the TA initial next to each name.

TA

_____   _____

_____   _____

Appendix: Resistor Color Code Chart

Here is a chart to show you how to convert the colored bands on a resistor to a value of resistance and a tolerance.

(source: http://www.digikey.com/~/media/Images/Marketing/Resources/Calculators/resistor-color-chart.jpg?la=en-US)

Skills Assessment (5pts)

Everybody must demonstrate that he/she can perform certain lab skills, as described below.

Fill in your names in the spaces provided. A TA will initial the appropriate space when a skill has been successfully demonstrated.

		Name #1	Name #2
		_____	_____
1.	Measure resistance with a multimeter.	_____	_____
2.	Adjust the secs/div and the volts/div on an oscilloscope to get a given signal into view with reasonable clarity.	_____	_____
3.	Demonstrate how to measure continuity with a multimeter, and explain why a continuity test is useful.	_____	_____
4.	Know which cables are intended for which instruments.	_____	_____
5.	Demonstrate that you can interpret the color code on a resistor.	_____	_____