Electronic Instrumentation

ENGR-4300 Fall 2004 Section ____________

Experiment 4
Introduction to Operational Amplifiers

Purpose: Become sufficiently familiar with the operational amplifier (op-amp) to be able to use it with a bridge circuit output. We will need this capability in our first project.

Equipment Required:

- HP 34401A Digital Multimeter
- HP 33120A 15 MHz Function / Arbitrary Waveform Generator
- HP E3631A Power Supply
- Protoboard
- Some Resistors
- 741 op-amp or 1458 dual op-amp

Background

Some of the following material was taken from Introduction to the OP Amp, by John Getty of the University of Denver at http://www.educatorscorner.com/experiments/html/exp26.html The interested reader is also encouraged to look over the op-amp links on the course webpage.

The schematic of Fig. 1 shows a standard ± V_{CC} configuration for op-amps. The schematic symbols for a battery are used in this schematic to remind us that these supplies need to be a constant DC voltage. They are not signal sources.

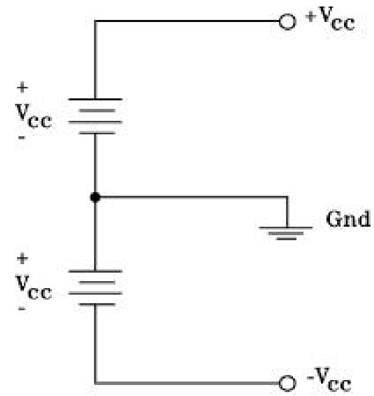


Figure 1

The HP E3631A power supply provides two variable supplies with a common ground and a variable low voltage supply. As shown in Fig. 1, the power supply jack labeled "COM" between the V_{CC} supplies should be connected to circuit ground. Adjust the output so the $+V_{CC}$ and - V_{CC} are equal, but opposite in sign, at 15 V. It is possible to turn the output voltages off without turning off the power supply by pushing the OUTPUT ON/OFF button. Pushing the button again returns the supply to its previous settings. This is the approach to use when testing circuits. However, at this point, we will turn the power supply off. You will have to set it up again later when you connect it to your op-amp circuit.

Study the chip layout of the 741 op-amp in Fig. 2. (You may use a 1458 dual op-amp chip instead of the 741. In that case, the chip layout will be somewhat different. The layouts and other specs for these two chips can be found on the course website and on pages 8 and 9 of the *Radio Shack Mini-Notebook on Op Amp IC Circuits*.) The standard procedure on DIP (dual in-line package) "chips" is to identify pin 1 with a notch in the end of the chip package. The notch always separates pin 1 from the last pin on the chip. In the case of the 741, the notch is between pins 1 and 8. Pin 2 is the inverting input, V_N. Pin 3 is the non-inverting input, V_P and the amplifier output, V_O is at pin 6. These three pins are the three terminals that normally appear in an op-amp circuit schematic diagram. Even though the $\pm V_{CC}$ connections (7 and 4) must be completed for the op-amp to work, they usually are omitted from simple circuit schematics to improve clarity.

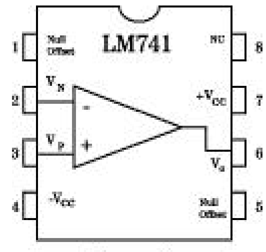


Figure 2

The null offset pins (1 and 5) provide a way to eliminate any "offset" in the output voltage of the amplifier. The offset voltage (usually denoted by Vos) is an artifact of the integrated circuit. The offset voltage is additive with V_O (pin 6 in this case), can be either positive or negative and is normally less than 10 mV. Because the off-set voltage is so small, in most cases we can ignore the contribution V_{OS} makes to V_O and we leave the null offset pins open. Pin 8, labeled "NC", has no connection to the internal circuitry of the 741, and is not used.

Fig. 3 shows a standard op-amp configuration known as an inverting amplifier and Fig.4 shows a non-inverting amplifier.

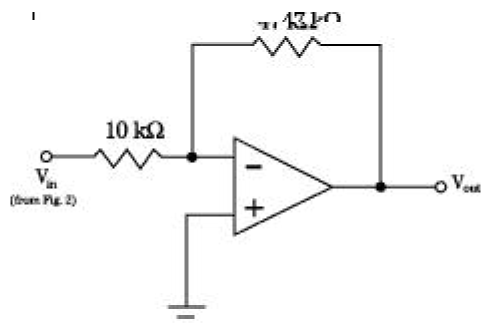


Figure 3

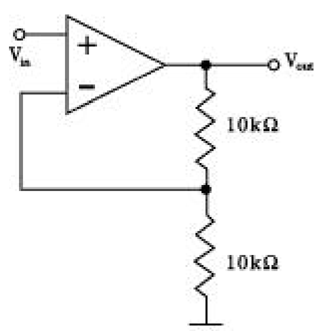


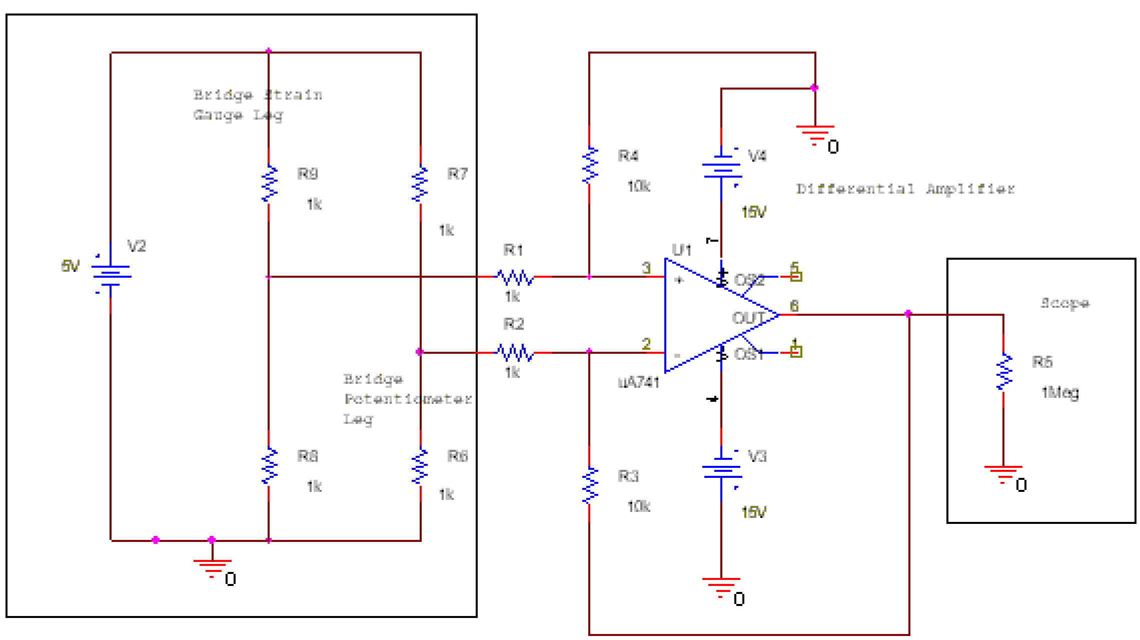
Figure 4

The inverting op-amp is more commonly used than the non-inverting op-amp. That is why the latter amplifier has the somewhat odd name with the double negative in it. If you look at the circuit, you will see that in the inverting op-amp, the chip is connected to ground, while in the non-inverting amplifier it is not. This generally makes the inverting amplifier behave better. When used as a DC amplifier, the non-inverting amp can be a poor choice, since its output voltage will be negative. However, for AC applications, inversion does not matter since sines and cosines are positive half the time and negative half the time anyway.

We will study other op-amp configurations further in a later experiment. However, there is one particular configuration that we need to consider in depth at this time – the differential amplifier. This amplifier is also known as the instrumentation amplifier because it is found to be an essential part of many measurement circuits. You may recall from Experiment 2, that it was difficult to measure the AC voltage across the output of the bridge circuit because both of the output connections had a finite DC voltage. That meant that one could not just connect one of the ‘scope channels across the output, since one of the voltages would be shorted to ground. The differential (instrumentation) amplifier allows us to get by this problem, since neither input is grounded. A very large fraction of measurement circuits use some kind of a bridge configuration or are based on some kind of comparison between two voltages. Thus, the operation of the differential amplifier is very important to understand.

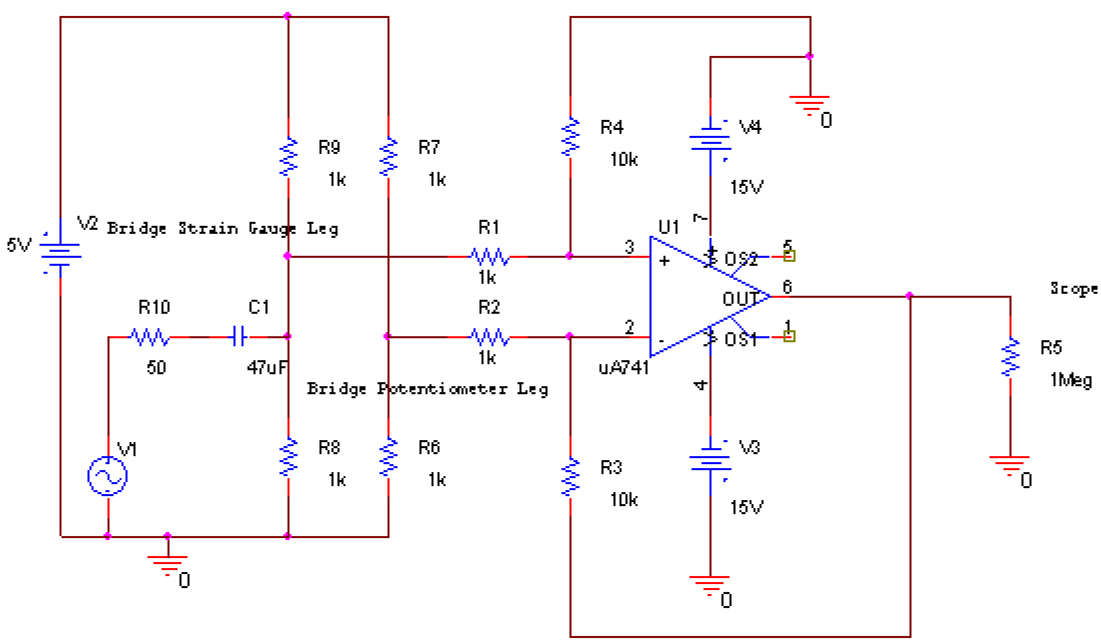
Part A – PSpice Simulation of Amplified Bridge Circuit

To see how this more complex amplifier configuration can amplify the difference between two voltages, we will do a PSpice simulation that includes a bridge circuit as the input. The configuration we will consider is shown below. Again, recall the operation of the strain gauge bridge observed previously. There were two legs to the bridge – one consisting of a fixed resistor (R9) and the strain gauge (R8) and the other ideally consisting of a 2k potentiometer (which has been divided into the two resistors R7 and R6). The final component in the strain gauge bridge is the 5V DC power supply (V2). When the pot is adjusted correctly, the voltage at the node between R7 and R6 will equal the voltage at the node between R9 and R8 (both should be about 2.5 volts). These five components represent the bridge. Resistor R5 represents the oscilloscope input. All the other components (R1, R2, R3, R4, V3, V4 and U1) represent the differential amplifier.



Unfortunately, it is not possible to simulate the operation of the strain gauge bridge directly using PSpice, since there is no simple way to make the resistance of the strain gauge itself oscillate with time. We can, however, add some components to the bridge to produce the kind of voltages we actually observed. The voltage observed at the node between R9 and R8 had both a DC level of about 2.5 volts and a small AC signal that oscillated at a frequency f of about 20 Hz. To make our simulation work, we add an additional sinusoidal voltage source in such a manner that we will be able to test our simulation experimentally. The new circuit is shown below. The source V1 and the resistor R10 represent a function generator (recall that our function generator has an internal resistance of 50 ohms). We have also incorporated a capacitor in the circuit so that the DC voltage at the node between R9 and R8 is not seen by the function generator. Recall that a capacitor is an open circuit at DC (frequency = 0), so that only AC signals can pass through it.

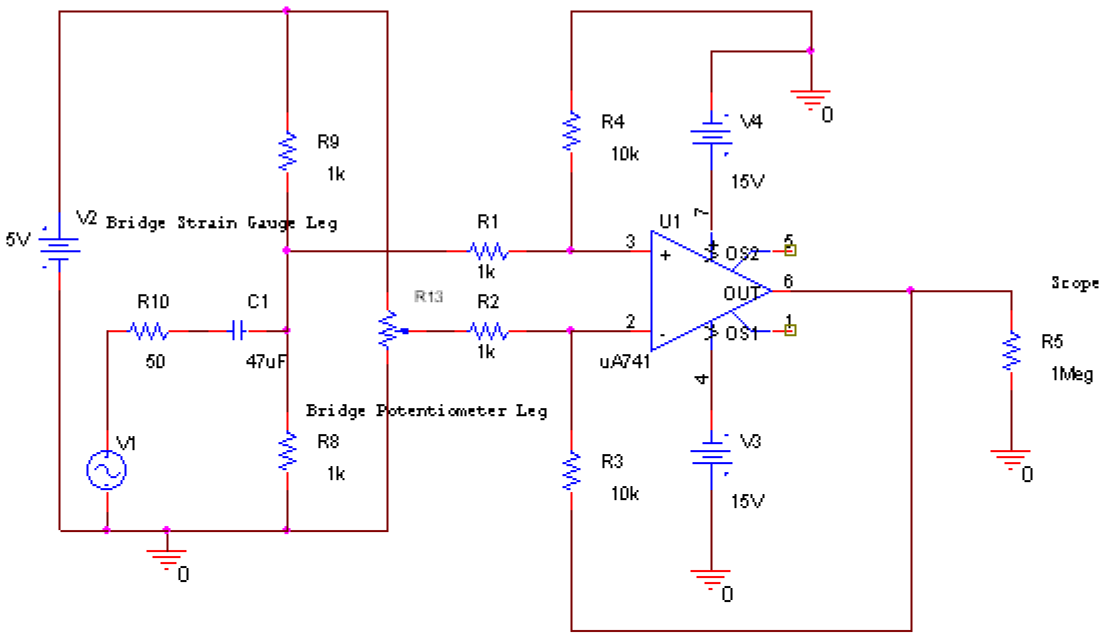
In summary, the two sources V1 and V2, the capacitor C1 and the five resistors R6 – R10 are used to produce a signal something like that of the output of the bridge circuit we build when we use the cantilever beam strain gauge. V2 is a constant value of 5 volts while V1 has a sinusoidal amplitude of 100 mV at 20 Hertz These values are chosen somewhat arbitrarily, but are typical of actual strain gauge behavior. The resistance values are chosen to either equal the resistances of the bridge (1k) or the output impedance of the function generator (50). The capacitor is one of the larger ones found in your parts kit and is used to AC couple the function generator output so that it adds to the DC voltage between the two 1k resistors of the bridge circuit. There is no function generator in the usual bridge circuit, but we use it here to give a voltage like the one that results when the strain gauge resistance varies sinusoidally. You will likely have to change some of these numbers to use this circuit to model the actual output of your bridge, since you will have to use the actual values for the components in your circuit. This schematic is set up to obtain a particularly simple gain. **Do the transient simulation, displaying about 4 cycles of the output and determine what this gain is. Print out one probe plot per group and discuss why you think the output is correct.** How close is it to the theoretical gain?



Circuit A

In this circuit, you will notice that R1=R2 and R3=R4. It is necessary that these resistors be the same or the circuit will not work properly. To see that this is indeed the case, **change the values of the resistors R2 to 1.5 k and R1 to 1k. Print out this plot and describe what happened to the output voltage. That is, how did this voltage change and why.**

Now set R2 back to 1k and replace the two 1K resistors in the potentiometer leg with a pot, as shown below.



Circuit B

The default resistance of the pot is 1K and the default value of the set parameter is 0.5. This means that if the pot were redrawn as a voltage divider, the top and bottom resistors would both be 500 ohms. Run the simulation again. In theory, since the pot divides the source voltage in half just like the two 1k resistors, the output should be the same. However, you will notice that the bridge is no longer perfectly balanced and therefore, the output has a small DC offset. This is caused by the influence of the additional components in the circuit on the smaller resistance values of the voltage divider in the pot. (You have already seen how circuits are influenced to varying degrees by the measuring equipment. Different parts of a circuit can also influence each other and cause some variation from theoretical expectations.) You can still balance the bridge by changing the set parameter in the pot to compensate for this influence. Try tweaking the value of the set parameter in the pot spreadsheet until you find a value that balances the bridge. (The two inputs will be centered around the same voltage and the output signal will be centered around 0 volts). **Print out the plot which proves the bridge has been balanced and write the value of set on it.** What would be the values of the upper and lower resistances if the pot were redrawn as a voltage divider?

Please note that our primary purpose here is to learn a little about an op-amp configuration that we can use with our cantilever beam bridge circuit output. We will return to op-amps to look at the simpler inverting and non-inverting amplifiers more thoroughly. However, you might want to play around a little with your PSpice simulation at this point and configure both of these circuits. If you do, you will have to be sure that your test input voltages are not too large. The output of an op-amp cannot exceed $\pm V_{CC}$. (At $\pm V_{CC}$ we say it is saturated.) Thus, if you have an overall gain of 100, the largest input must be somewhat less than $\pm V_{CC}/100$. This is one of the other properties of op-amps we must take into account when we use them. There are several others which we will introduce when necessary.

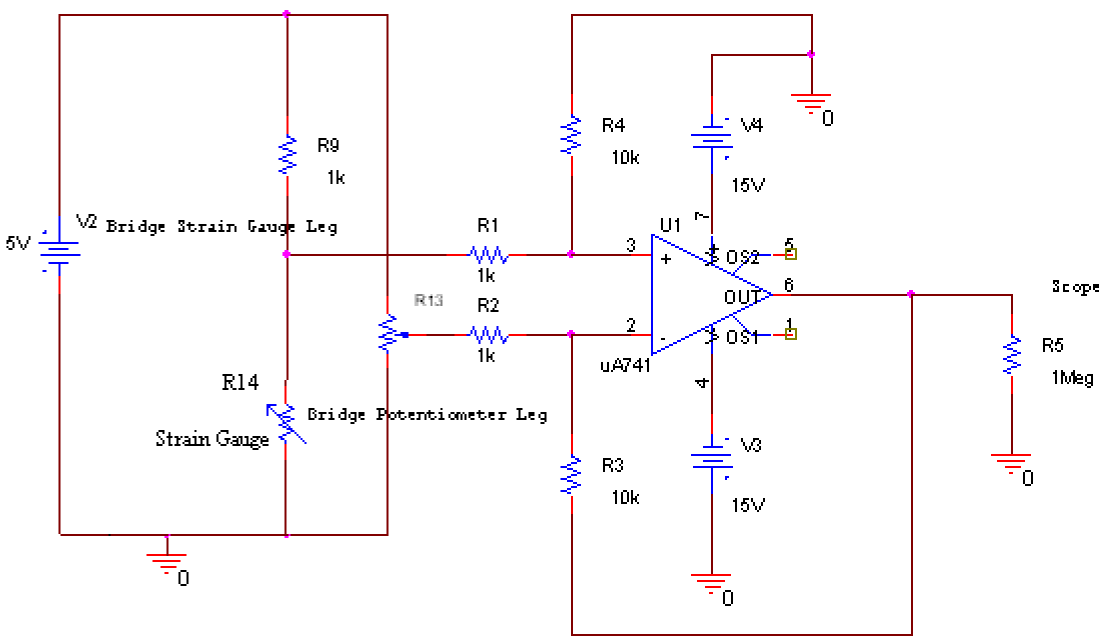
Part B – Build an Amplified BridgeCircuit

Now you must build the circuit you have modeled. You can either build circuit A as shown.or you can build the circuit with an actual strain gauge (instead of the function generator model).

1. Build circuit A on your protoboard. Use the function generator for V1 and R10 and one of the DC supplies for V2. For this circuit, observe the ratio of the op-amp output (channel 1) to the function generator output (channel 2). Find the gain. Is it more or less than the gain found with Pspice. Why? KEEP THIS CIRCUIT FOR PROJECT 1.

- OR --

*2. Build the circuit using the strain gauge and a 1k pot as shown below in Circuit C***.** For this circuit, you will observe the ratio of the op-amp output (channel 1) to the difference between the two legs of the strain gauge bridge (channel 2). In order to balance the bridge, you must turn the pot until the output trace on the 'scope has no DC offset (is centered around zero). Once the bridge is balanced, generate an amplified version of the decaying sinusoidal output. (The STOP button on the 'scope really helps here.) Find the gain. Is it more or less than the gain found with Pspice? Why? KEEP THIS CIRCUIT FOR PROJECT 1.



Circuit C

Be sure that you have your circuit checked over before you do your experimental test. Observe the function generator output (or strain gauge output) AND the output of the amplifier on the scope. **Use the Agilent software to print out a copy of the scope traces,** but only after you show the scope traces to a TA or instructor. **Clearly mark which trace is which and determine the amplitude of both signals.** When you are finished testing your circuit, *do not dismantle it*, since you will need a version of this circuit for the first project.

Report and Conclusions

The following should be included in your written report. Everything should be clearly labeled and easy to find. Partial credit will be deducted for poor labeling or unclear presentation.

Part A

Include the following plots:
1) Transient simulation from PSpice with R1=R2=1K. The plot should show 4 cycles of the input and output to the differential amplifier. (1 pt) Indicate the amplitudes of the input and output clearly on the plot. (1 pt) Write on the plot what the gain is and how you found it. (1 pt)
2) Transient simulation from PSpice with R1=1K and R2=1.5K. The plot should show 4 cycles of the input and output to the differential amplifier. (1 pt)
3) Transient simulation from PSpice with R1=R2=1K and a 1K pot. The plot should show 4 cycles of the input and output to the differential amplifier and have the value of set which balanced the bridge. (1 pt)

Answer the following questions:
1) What is the theoretical gain of the amplifier you modeled in PSpice (all resistors balanced)? (1 pt)
2) What overall gain did you find for the basic differential amplifier in plot 1)? Why does the output look correct? (Consider both the amplitude and phase shift.) (2 pt)
3) Describe the most significant difference between the output of the circuit with the balanced input resistors and the output when they were unbalanced. (1 pt) Explain why you think this has happened using the derivation of the equations for the differential amplifier given in class. (1 pt)
4) Redraw the 1k pot in the balanced bridge as a voltage divider. Provide values for the upper and lower resistances. (1pt)

Part B
Include the following plots:
1) Benchmark software copy of the scope traces for channel 1 and channel 2 of the circuit you built. (1 pt) Clearly mark which trace corresponds to the input and output to the differential amplifier. (1 pt) Write the amplitude of both signals on the plot. (1 pt) Show the gain and how you calculated it. (1 pt)

Answer the following questions:
1) What overall gain did you find in the circuit that you built? (1 pt)
2) How does the actual gain compare to the theoretical gain of the amplifier you built? (1 pt) ... to the gain in the Pspice simulation? (1 pt) Name two things which can account for the discrepancy. (2 pt)
3) Find the maximum current the 741 op-amp can supply. This can be found on the data sheet for the device. (1 pt)
4) Give an example of a system (electrical, mechanical, chemical or some combination) with negative feedback and an example of a system with positive feedback. (2 pt)

Summarize Key Points (1 pt)

Mistakes and Problems (0.5 pt)

Member responsibilities (0.5 pt)

Total: 25 points for write up

Attendance: 3 classes (5 points) 2 classes (3 points) 1 class (0 points) out of 5 possible points
No attendance at all = No grade for experiment.