LABORATORY 1: Sensors, Decisions, and Electronic Math

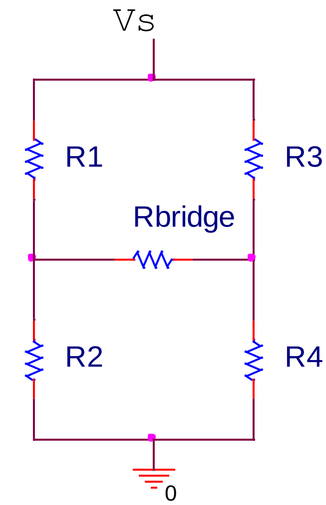
Let's get started with Alpha Labs!!!!
Choose two partners:

1) You can choose **up to two** people for a maximum group of three. Groups of 2 are acceptable. Working alone is discouraged but if necessary, it is possible. Remember, you will likely have to work with this person or persons for the rest of the semester.
2) Only one laboratory report is needed for each group. Make sure you include both group members name on the front of the report. Also remember to add your partner(s) to Gradescope. (Only one person needs to submit it).
3) Please create a WebEx Teams space ***within the Electric Circuits Team*** using your Last names, Alpha/Omega, and Section…for example, "Smith|Jones|Xu|Alpha|Sec 1"
4) You will go through each lab section to learn some basic steps. ***Then you will prove specified concepts listed at the end of each section.*** This simply means that you will demonstrate, with your designated circuit, how a specific concept works through clear and concise comparisons of mathematical analysis, simulation, and experimental measurements

   The template for the Proof of Concepts document that you will submit can be found here: https://sites.ecse.rpi.edu/~ssawyer/CircuitsSum2022_all/Templates/AlphaOmega_ProofofConcepts.docx . Please answer any questions related to those concepts and provide mathematical calculation, simulation, and experimental data to support the proof of concept!
5) You are required to participate in one Proof of Skills Individual check-in by directly posting your video one minute video witin the Gradescope online assignment in the submission space "Proof of Skills Check-In" BEFORE you start this lab. Do so now, if you haven't done it already! In addition you must complete four Alpha Lab INDIVIDUAL check-ins throughout the semester, indicated by Gradescope by posting a short ***one-minute video in your Lab group's WebEx Space about your INDIVDUAL experience*** with the labs. An associated Gradescope online assignment will guide the content of your check-in video. An undergraduate student assistant is assigned to your lab group to continue the conversation, if needed, for mentorship.

PART A – Sensors and Decisions

Material covered: Bridge circuits, Voltage dividers, Op amp configurations



Wheatstone Bridge

Wheatstone Bridge:

A Wheatstone Bridge can be used to measure the value of an unknown resistor. It is a basic type of Ohmmeter. The bridge is shown on the in the above figure. When the bridge is 'balanced', no current flows through the Rbridge resistor. If that is the case, then both the left node and right node for that resistor must have the same voltage. Additionally, since no current is flowing through Rbridge, the left and right paths can be treated as voltage divider circuits with two resistors in series. Circuit analysis then gives us

$$V_{Left} = \frac{R2}{R1+R2}V_S \quad \text{and} \quad V_{Right} = \frac{R4}{R3+R4}V_S$$

Again VLeft = VRight, so we can set these two expressions equal, perform some algebra and obtain a relationship for the resistors when the bridge is balanced (no current through Rbridge) as

$$\frac{R1}{R2} = \frac{R3}{R4}$$

If one of the resistors is unknown, R4 for example, we can then use the bridge to find that value. Holding R1 and R3 fixed, we can vary R2 until we measure zero voltage drop (no current) across Rbridge. Once we have found that value for R2, we apply the above expression and determine R4. Thus, we have an Ohmmeter.

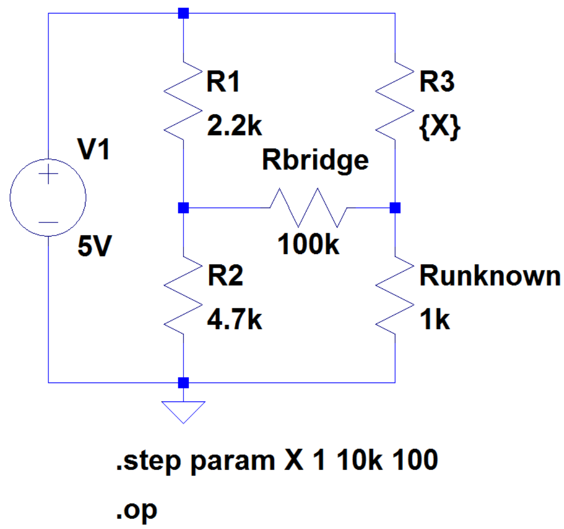
A1: Wheatstone Bridge and Parametric Analysis

We will use the Wheatstone bridge to determine the resistance of an unknown resistor. Pick up any resistor in your kit and consider it "unknown" (If you have an M1K board, you can of course measure the resistor directly so that you can verify your experimental results). In the experiment, a potentiometer is the variable resistor. By adjusting the potentiometer such that the voltage across Rbridge is zero, the value of Runknown can be determined. In the LTSpice simulation, parametric analysis allows varying resistor voltages.

1) You have used parametric analysis in your Proof of Skills. You may use these LTSpice Results to skip to building the circuit in step 5)! If you did not complete this skill, use the instructions below.

LTSpice Parameteric analysis

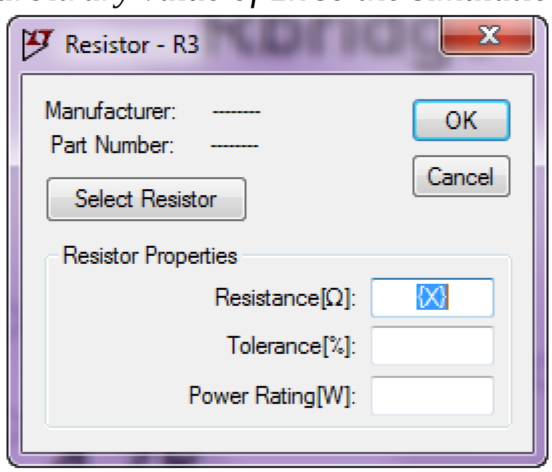
2) Determine the symbolic expression for Runknown when Vbridge is zero (see laboratory introduction).
3) Using values of R1 = 2.2kΩ, R2 = 4.7kΩ, Rbridge = 100kΩ ***(or any comparable values that you have in your kit so you can compare to experiment***), and Runknown = ???. R3 is a 10K potentiometer. *Note: Resistors were renamed by right clicking the given name like R4 and writing "Runknown."*
4) In LTSpice, plot Vbridge vs Rpotentiometer where Rpotentiometer is a parametric value. *In the LTpice simulation, follow the procedure to perform a parametric analysis* (details below). Using the plot and a differential voltage marker, identify the Rpotentiometer value that results in Vbridge = 0. The LTSpice schematic is shown below.



Remote notes:
Remember-You can change these resistor values to any comparable values that you have in your kit. The goal is to compare to experiment…

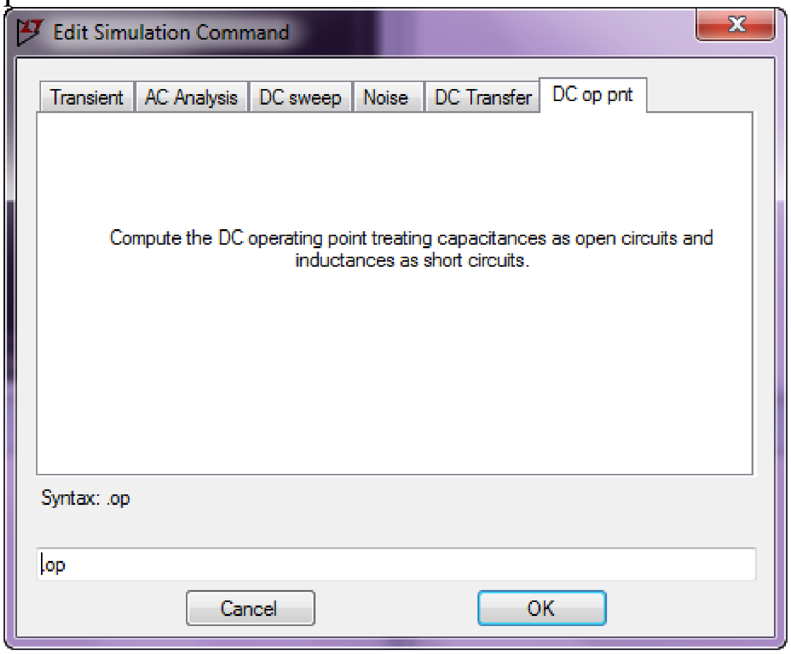
a) *Parametric analysis:* The .step command performs repeated analysis while stepping through specified values of a model parameter, global parameter or independent source.

1. Define the component parameter by right clicking the resistor R3 and entering "{X}" for the value of resistance (as shown in the diagram below). *Note: Runknown is given the arbitrary value of 1k so the simulation can run.*



2. Add a .step command using a SPICE directive (press "s") which specifies the steps for a parameter
Example: ".step param X 1 10k 1k" steps the parameter S from 1 to 100k in 1k increments.
You may change the increments to a value that will give you more points.

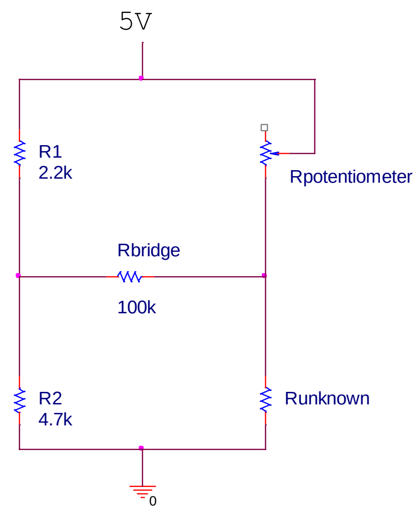
3. Add .op in the SPICE directive. (Click ".op" far right on toolbar and add .op then place anyway on circuit)
4. Run the simulation (click "Running man") go to "DC op pnt tab" and click "ok"



5. Run the simulation again. (click "Running man"). The simulation pop up window should show but without traces with resistor values as the x axis.
6. To specify the differential probes across Rbridge, click the node to the left of Rbridge (a red probe should appear), hold and click the right (a black probe should appear).
7. The trace V(N00n, N00nx) should appear (where n is some number label of node).
8. Now find the variable resistor value when VRbridge = 0V. Use the cursor function by clicking the trace label at the top of the diagram "V(N00n, N00nx)". You can drag the cursor along the curve by clicking and holding where the horizontal and vertical lines meet.
9. Include the screenshot/plot of the balanced bridge point with clear labels in your report.
10. Use the equations in the introduction to calculate the Runknown value from a balanced bridge circuit.

Build the Bridge Circuit and Compare

Build the physical circuit using a 10kΩ Potentiometer, as shown in the circuit below. Note, one leg of the potentiometer is floating. Turn the potentiometer such that the measured Vbridge = 0. Once you find that value, use an Ohmmeter to measure the resistance of the potentiometer (they are on the center table). Be careful not to turn your potentiometer and make sure you disconnect the circuit so you don't measure the other resistors. Compare your result to part 3).
Compare the LTSpice simulated value to the value obtained from your physical circuit. Also compare to a calculation.



A2: Explore another bridge circuit use for practical sensors

5) There is ***ANOTHER practical use of a bridge circuit*** in relationship small changes of resistance from a resistive sensor!
   a) Using an internet search, FIND an example of this practical use for sensors and provide the link (There are more!) https://www.electronics-tutorials.ws/blog/wheatstone-bridge.html
   b) Setup a simulation that demonstrates how a bridge circuit can measure small changes in resistance and convert to voltage
   c) Compare this to using a simple voltage divider. ***Why would one use a bridge circuit instead of a simple voltage divider?***
   d) Show the function of this practical use of a bridge circuit in a built circuit with a potentiometer (in your kit!) or light detecting resistor (ask TA!) or ANY other resistive sensor you can get your hands on! Measure the voltage output as the resistance changes on your potentiometer or sensor.
   e) Compare your experimental results to your simulation and support with calculations. (You may need a way to find resistance of your sensor or potentiometer…data sheet? Impedance measurement?)

Part A: Proof of Concepts List

Prove the concept of a balanced bridge
Prove the concept of using a bridge circuit to measure small variations of resistance

Please include screen shots of your results in your Proof of Concept Report. Make sure they are easy to read with Proof of Skills formatting continued in your Proof of Concept report!

Template:
https://sites.ecse.rpi.edu/~ssawyer/CircuitsSum2022_all/Templates/AlphaOmega_ProofofConcepts.docx

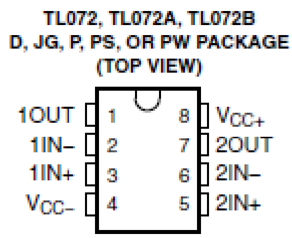
Part B: Comparators and Voltage Dividers

General Operational Amplifier Basics

Overall notes:
TL072CP chip (dual op-amp):
The data sheet for the chip can be found online from any number of sites. One is provided below (it is long and contains several chips)

http://www.ti.com/lit/ds/symlink/tl071.pdf

A copy of the pin connections is shown below

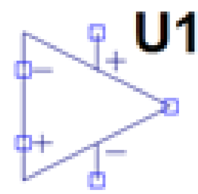


There are two op-amps on the chip, indicated by the ‘1’ and the ‘2’ pin labels. For example, 1IN+ is the V+ and 1IN- is the V- of the first op-amp, with 1OUT being the Vout. Power connections are +Vcc at pin 8 and –Vcc at pin 4.

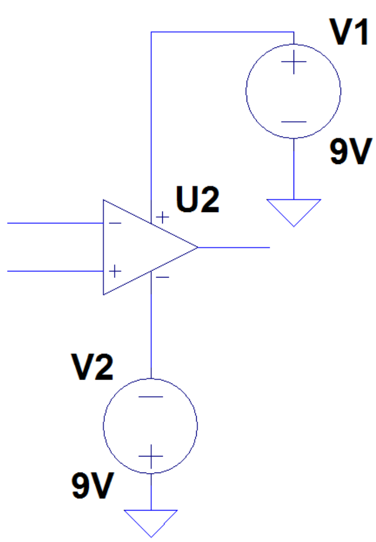
In LTSpice, you can use the "UniversalOpamp2" component or "opamp" component. The "opamp" component does not have power levels and is assumed ideal. It is useful for simplified drawings, but your simulations will not be the same as the experiments. As such, please use the "UniversalOpamp2" component, with LTSpice details shown below.

A summary of the connections for LTSpice "UniversalOpamp2" component:

- input, - (left), inverting input
- input, + (left), non-inverting input
- -(bottom), V-: Negative power, 9 V
- +(top), V+, Positive power, 9V
- Right node: Vout, output voltage

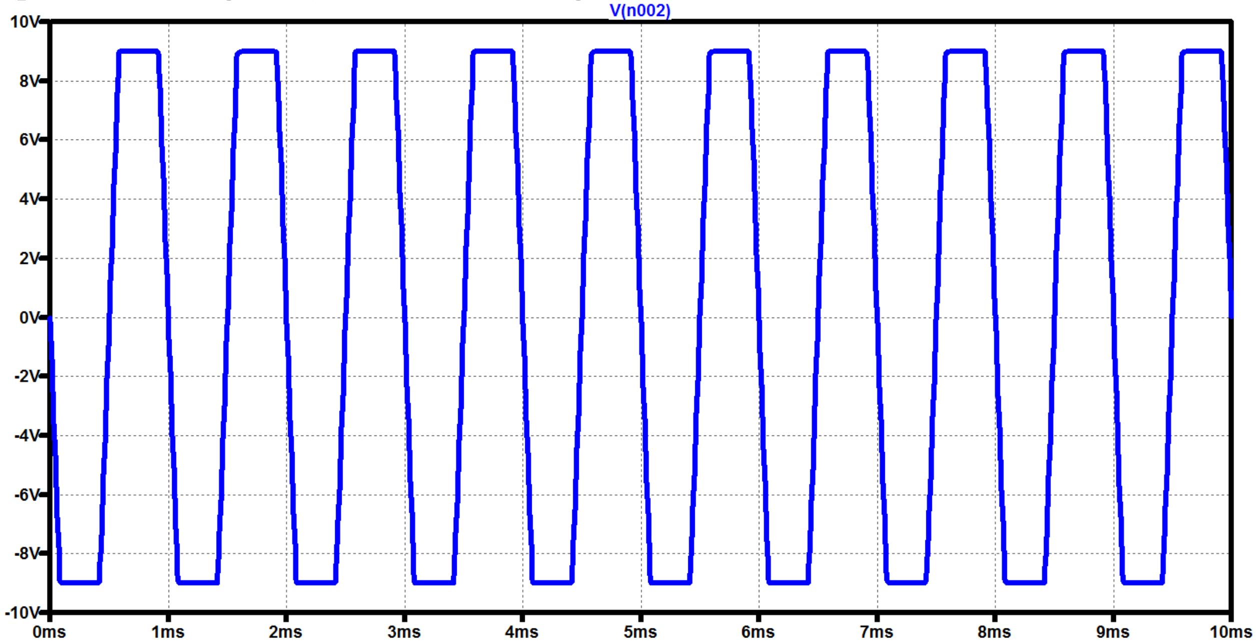


The DC power sources will be the 9 Volt batteries that you have in your kit. Note the orientation of the batteries when you connect the leads.

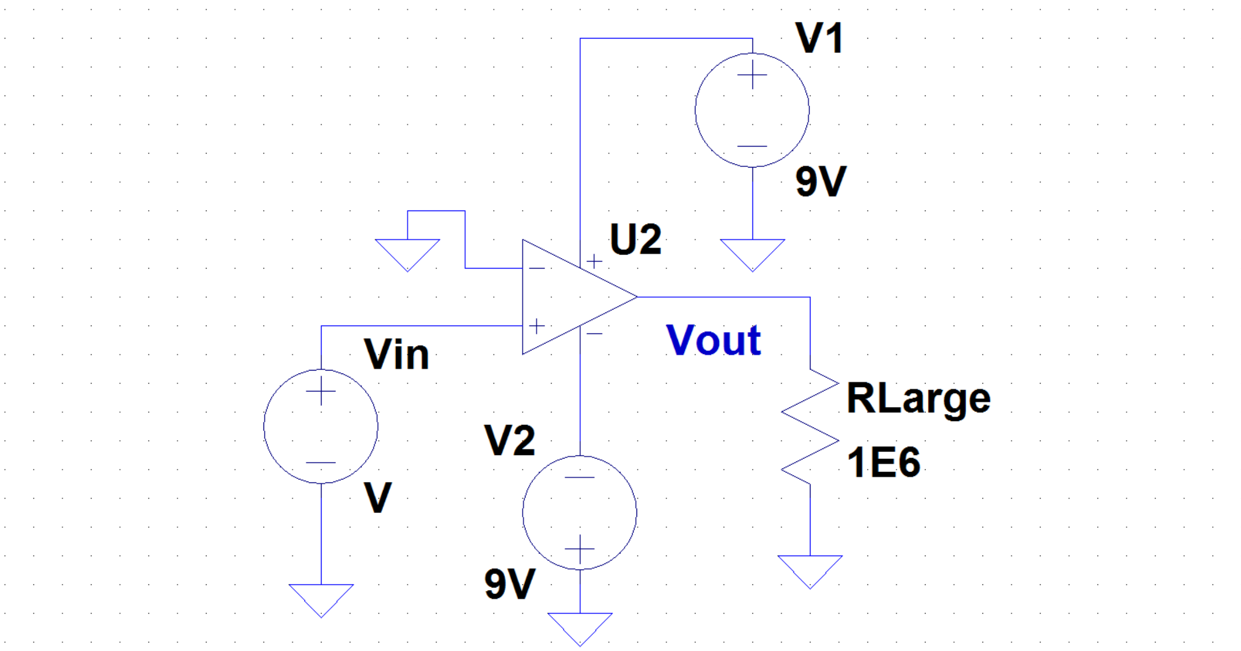


Again, for LTSpice simulations, the circuits on the following pages indicate how to power a uA741 op-amp. The input and output connections depend on the circuit.

An example of an op-amp reaching saturation is shown below. The input is a sinusoidal. If the op-amp was ideal, the output would also be a sinusoid with a scaled amplitude. However, saturation occurs and the output voltage cannot exceed (positive or negative) the source voltages.



B1: Comparator Circuits



Build the comparator circuit shown above. V+ and V- will be your inputs and Vout will be the output. In Analog Discovery experiments, use the TL072CP chip (or equivalent. (You only need one amplifier for this part). In the LTSpice simulations use the "UniversalOpamp2" component.

1) We will use W1 and W2 for out amplifier inputs. The Voltmeter channels inputs will act as the RLarge.
   a. Connect W1 (yellow wire) to the V+ op-amp input and ground (orange striped wire) to the V- op-amp input.
   b. Ground the V- op-amp input.
   c. To compare input voltage to output voltage, use the Voltmeter to measure the output voltage (refer to Lab 1).
   d. Using the Discovery board, set the W1 output voltage to DC mode and check the output voltage of the op-amp for the following input voltages (This chart and others below should end up in your Proof of Concepts report…)

Vin [V]	Vout [V]
2	
1	
0	
-1	
-2	

   e. Comment on your results and expectations when Vin = 0 V.
   f. What happens when you switch ground to the V+ op amp input and Vin to the V- op amp input? Provide a similar chart. Explain the difference.

<u>Remote notes:</u> Remember-You can change the input voltages in your chart for any of the op amp circuits below. Explicitly write these changes in your proof of concept document with reasons why you made them.

2) In LTSpice, build the comparator circuit using a UniveralOpamp2 op-amp. You will need to add a load resistor at the output node since LTSpice does not allow nodes to float (be unconnected). A 1E6Ω load is fine (use exponential notation since M in LTspice is 1E-3). Compare the output voltages between LTSpice and Analog Discovery. You should see some differences, what causes these differences?

Vin [V]	Vout [V]
2	
1	
0	
-1	
-2	

3) **Remove the ground connection at V- and use a voltage divider with one of your 9V sources to create a reference voltage to compare to**. Clearly label the reference voltage you will compare to on your schematic and in the table below.
   1.)
      a. Repeat the output voltage measurements again

Vin [V]	Vout [V]
Reference voltage – 2n	
Reference voltage - n	
Reference voltage	
Reference voltage + n	
reference voltage + 2n	

Where n is a voltage interval that you specify such as 1V or 0.5V

4) Again, compare your Analog Discovery experiment to the LTSpice simulation.

Vin [V]	Vout [V]
Reference voltage – 2n	
Reference voltage - n	
Reference voltage	
Reference voltage + n	
reference voltage + 2n	

Where n is a voltage interval that you specify such as 1V or 0.5V

Part B: Proof of Concepts List

Prove that a comparator with a reference of 0V saturates to positive or negative voltage values

Demonstrate that you can create a reference voltage to compare to saturate to negative or positive voltage values

Note: The word prove means demonstrate using simulation, mathematical calculation, and experimental results to show that a concept is valid. It is not a mathematical definition of a proof. If you find limitations to that concept, include it!

Please include screen shots of your results in your Proof of Concept Report. Make sure they are easy to read and simulation results are labeled.

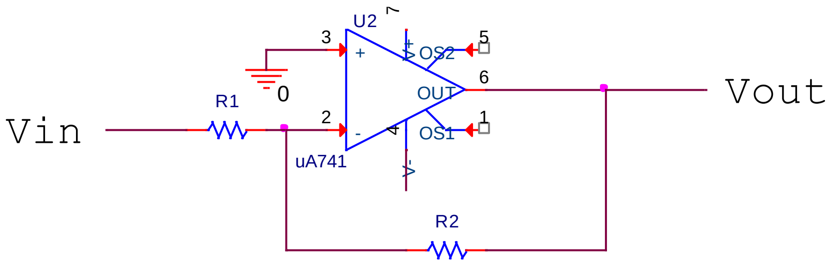
Template:
https://sites.ecse.rpi.edu/~ssawyer/CircuitsSum2022_all/Templates/AlphaOmega_ProofofConcepts.docx

Part C: Mathematical Operational Amplifiers

C1: Multiplication and Division: Inverting Op-amp, Non-inverting op-amp

Build and test the following circuits. *(Again, this schematic was created in PSpice. Recreate in LTSpice. Remember the power connections. They have been removed to simplify the drawing but they must still be included in the circuit).*

1) Inverting Op-amp with a gain of -3.5 (Multiplication and inversion)



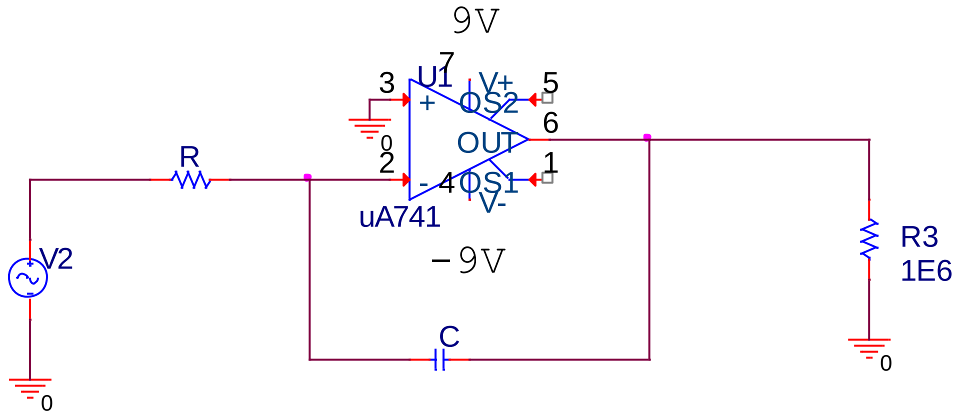
   a. When considering the saturation voltage, what is the maximum Vin such that Vout = -3.5 Vin? Choose appropriate resistors.

Vin [V]	Vout [V]
5	
3	
1	
0	
-1	
-3	
-5	

   b. Build the circuit in LTSpice and verify that simulation is constant with experiment (within the limits of the respective saturation voltages).
   c. Show this gain with a sine wave instead of a DC signal. Compare the amplitudes.

2) Non-Inverting Op-amp with a gain of 0.5 (Division by 2)
   a. Build, simulate, and test a circuit you create.
   b. Come up with a simple chart of inputs and record output.
   c. Find the circuit drawing of a non-inverting op amp and find any resistor values that achieve a gain of 0.5.
   d. Show this gain with a sine wave instead of a DC signal. Compare the amplitudes.

C2: Integrators and Differentiators



The above circuit is called an integrator. The relationship between the input and the output is $V_{out} = -\frac{1}{RC}\int V_{in}dt$.

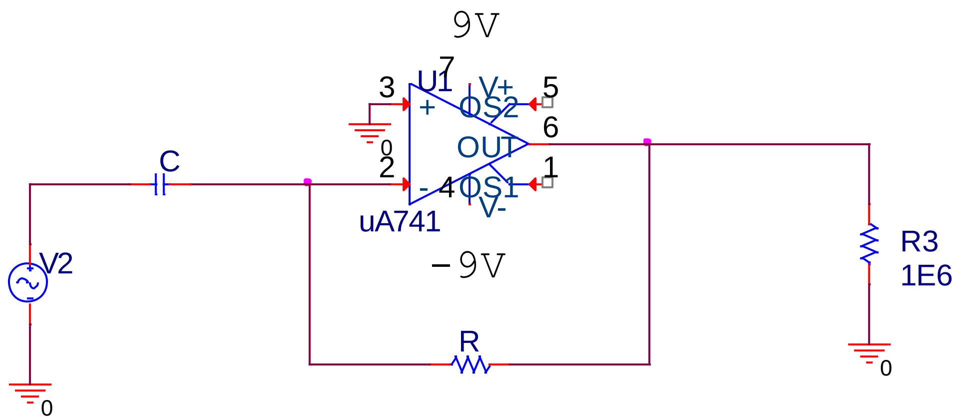
a) Build the Integrator circuit using R = 2kΩ and C = 0.2E-6F in LTSpice. You may need to add components together to get the values above.

(Note: The diagram above was made in PSpice but you should be now know how to do it in LTSpice.)

b) On the Discovery Board, set your signal to a 2.0 Vp-p *sine* wave with a DC offset of 0.0 Volts (no DC component), $V_{in}(t) = V_o \sin(2\pi f t)$. Set the frequency to 1kHz initially. (Discovery Board only)
   a. For the above settings, mathematically determine $V_{out}(t)$. (yes, you have to do Calculus)
   b. Connect Oscilloscope channel 1 input to the source and Oscilloscope channel 2 input to the output of the op-amp. Does the output voltage agree with expectations?
   c. Increase and decrease the frequency. Does the waveform change as expected?
   d. Change the coupling to DC and set the DC offset to 0.1 V. Explain what you see at the output. If you don't have this option on your board, skip it.
   e. Repeat part a-c. with a square wave input. Make sure you set the offset to 0 V. Again, does the output waveform behave as expected?
   f. Repeat part a-c with a triangle wave input. Again, does the output waveform behave as expected.
   (LT*Spice: The triangle wave simulation is optional and extra credit.)*

Clearly label extra credit in the heading of your Concept.
BE SURE TO DISCUSS all questions especially c. and d. in your proof of concepts…

Differentiators



The above circuit is called an differentiator. The relationship between the input and the output is $V_{out} = -RC\frac{dV_{in}}{dt}$ (we will derive this in class). ***(Note: The diagram above was made in PSpice but you should be now know how to do it in LTSpice.)***

Build the Differentiator circuit using R = 2kΩ and C = 0.2E-6F. (Discovery Board only)

a) On the Discovery Board, set your signal to a 2.0 Vp-p *sine* wave with a DC offset of 0.0 Volts (no DC component), $V_{in}(t) = V_o \sin(2\pi f t)$. Set the frequency to 1kHz initially.
   a. For the above settings, mathematically determine $V_{out}(t)$.
   b. Does the output signal agree with expectations?
   c. Increase and decrease the frequency. Does the waveform change as expected?
   d. Set the DC offset of your input signal to 0.1 V. Do you notice any change in the output?
   e. Change the waveform to a triangle wave. Is the output consistent with expectations?
   f. Change the waveform to a square wave. Is the output consistent with expectations? *(we will discuss what you see in a fair bit of detail when we get to second order transient responses)*

PART C: Proof of Concepts list
Prove that multiplication of a signal is possible. (either DC or sinusoidal).
Prove that a signal can be inverted.
Prove that division of a signal is possible (either DC or sinusoidal).
You may combine a couple of the above in one circuit.

Prove that the integrator circuit integrates at least two different signals (i.e. sine wave and square wave) as expected.
Prove/discuss DC offset as it relates to integration of your signals

Prove that the differentiator circuit integrates at least two different signals (i.e. sine wave and square wave) as *expected*. Note: Speculate why the square wave and/or triangle output looks the way it does which is likely NOT what you expected!…(mathematical analysis will help but a little physics is needed too….)

Part D: Design Your own Application by Integrating the Parts Above
D1: Voltage Divider as a Component (not just a calculation)

1) Demonstrate how you can connect a sensor (or potentiometer) to a bridge circuit (Part A) to a comparator circuit (Part B) to make decisions about your environment. Simulate and build this sensor/decision circuit. Write some context about how you would use this in real life!

2) Discuss applications for either an integrator or differentiator in a sensor circuit. (Simulate only)

Part D: Proof of Concepts List

Demonstrate a sensor circuit that changes to a binary output (a decision) depending on resistance induced by the environment (can use potentiometer). Simulate and build this circuit.

Discuss applications for either an integrator or differentiator in a sensor circuit. (You can write or create schematic in LTSpice. Simulate if possible)

EXTRA CREDIT: Write in your metacognition journal (instructions and template in the link below, feel free to continue to edit a Google doc throughout the course to add entries).

https://ecse.rpi.edu/~ssawyer/CircuitsFall2019_all/Labs/Circuits_OmegaLabDocs/04_Deliverables/05_Circuits_Metacognition%20and%20Reflections.docx

SUMMARY of Concepts

Concept List that must be accounted for in your Proof of Concepts

PART A:

1. **Prove the concept of a balanced bridge**
2. **Prove the concept of using a bridge circuit to measure small variations of resistance**

PART B:

1. **Prove that a comparator with a reference of 0V saturates to positive or negative voltage values**
2. **Demonstrate the you can create a reference voltage to compare to saturate to negative or positive voltage values**

PART C:

1. Prove that multiplication of a signal is possible. (either DC or sinusoidal).
2. Prove that a signal can be inverted.
3. Prove that division of a signal is possible (either DC or sinusoidal).
    a. You may combine a couple of the above in one circuit.

4. Prove that the integrator circuit integrates at least two different signals (i.e. sine wave and square wave) as expected.
5. Prove/discuss DC offset as it relates to integration of your signals
6. Prove that the differentiator circuit integrates at least two different signals (i.e. sine wave and square wave) as expected. Note: Speculate why the square wave and/or triangle output looks the way it does which is likely NOT what you expected!…(mathematical analysis will help but a little physics is needed too….)

PART D:

1. **Demonstrate a sensor circuit that changes to a binary output (a decision) depending on resistance induced by the environment (can use potentiometer). Simulate and build this circuit.**

2. **Discuss applications for either an integrator or differentiator in a sensor circuit. (You can write or create schematic in LTSpice. Simulate if possible)**

Standards Based Assessment:

You will be graded on the following Standards. Please ensure to achieve each standard. If you do not, you can resubmit to the missing standard to the end of the semester. CLEARLY mark the changes you make in you Proof of Concept submission by either Tracking Changes in Word or highlighting changes by writing comments in a different color and/or changing the color of the updated work.

Lab 01 Standards

1. **I can use a bridge circuit to measure small variations of resistance**
2. **I can make a reference voltage with a comparator to help make decisions**
3. **I can multiply or divide a signal with a gain that I specify by changing component values in an op amp circuit.**
4. **I can integrate or differentiate a signal by changing component values and positions in an op amp circuits.**
5. **I can design a potential solution to a sensor problem by integrating two or circuits learned in this lab.**
6. **I can identify non-idealities or unexpected results and attempt to explain why they may exist.**
7. **I can answer for myself "Is this right?" by comparing mathematical calculations to simulation and experimental results.**
8. **I can show plots and diagrams that are easy to read, scaled correctly and clearly labeled.**
9. **I can use consistent variable labels and component values in mathematical calculation, simulation and experimental results for easy comparison.**
10. **I can accurately answer conceptual questions found throughout the lab.**