Lab 5 Operational Amplifiers

Reference Resource (*optional*):
Practical Electronics for Inventors, Paul Scherz and Simon Monk, 2016, ISBN 978-1-25-958754-2

Lab Overview:
Examine and understand use of Operational Amplifiers to design a simple amplifier. Also understand how to make a resistor ladder voltage reference for operation of a general-purpose amplifier as a single-ended amplifier. Understand the design process of simulation verification, in this case using LTSpice, and then physical validation on a prototype board.

Inventory List:
The following is a list of available inventory for this Lab

Part	Details
Resistors	Assorted from 5 and 10% as needed
Waveform generator	AD 2
Oscilloscope	AD 2
Breadboard	830 Tie points
Op-Amp	Microchip MCP6022 or Analog Devices OP484
Alligator clips	Connectors
Assorted jumper wire	Red, black, white, green, blue, orange
Digital Multimeter	MS8217, DT-830B, DT838, or similar
Software	LTSpice

You will need to refer to **Lab 1 for LTSPICE and Measurement**, **Lab 2 for resistor networks**, and **Lab 3 for voltage dividers**.

Linear Technology has produced a video that shows you how to perform AC analysis in LTSPICE. The link for this video is http://www.linear.com/solutions/4581.

Background Information:

A) **Waveform Generators and Oscilloscopes in your Analog Discovery 2 Kit (Mandatory):**

Check the following two videos:

1- Digilent's explanation video: https://youtu.be/HUAy0J3XqaU

2- ASU's explanation video (also posted in the lab folder): https://youtu.be/rtsABzoV-WI

B) **Waveform Generators and Oscilloscopes in a Practical Lab (Optional):**

A waveform generator (WFG) supplies a variety of time-varying voltages at its output terminals. There are two terminals on the WFG – "Sync" and "Output" as seen in the figure below.

For EEE 202, only the Output Terminal is used – *you will never use the Sync Terminal*.

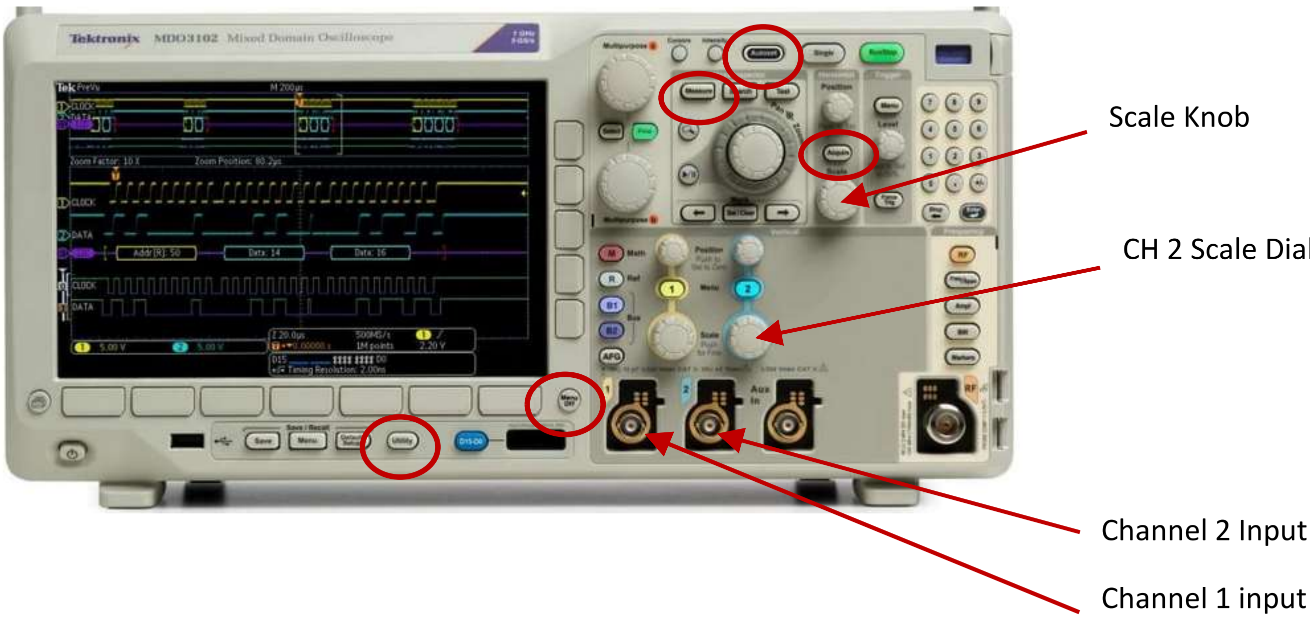
1. Turn on the WFG using the Power button.
    - By default, the WFG will deliver a **100mV peak-to-peak** sine wave with a frequency of **1kHz**

2. Press the "Freq" softkey to see the frequency
    - To change the frequency, press the "Freq" softkey and enter the magnitude of the desired frequency using the numeric keypad and select the desired units

3. Press the "Ampl" softkey to see the peak-to-peak voltage
    - To set the peak-to-peak voltage, press the "Ampl" softkey and enter the desired amplitude using the numeric keypad and select the desired units

4. The rotating knob can also be used to adjust the amplitude and frequency values, rather than the numeric keypad

Oscilloscopes are used for measuring output waveforms and unlike a multimeter, they provide a visual image of the waveform. The accuracy of oscilloscopes can vary greatly from model to model. Some scopes have higher resolution, some are very sensitive to input noise, and some have built in statistical measurements that can jump around when in use. In industry, you would have to select the appropriate oscilloscope to meet your specific needs. The scopes used in the EEE 202 lab (or similar) are shown below.



For EEE 202, only Channels 1 and 2 will be used – *you will never use the AUX Channel (Trigger)*.

The specifications for each Channel can be found on the oscilloscope noted above the inputs. The resolution of an oscilloscope is the ability to tell two points apart. Looking at the oscilloscope, you can find the minimum voltage and time that can be displayed.

Connect the WFG output to the input of Channel 1 on the scope. Use the adapters (one on the WFG and one on the scope) – the red terminal is positive and the black terminal is negative. The black terminal ALWAYS connects to the ground side of the circuit. Push the "Autoset" button to reset the scope and push the "Menu off" button to get rid of the menus. Each division is one FULL box. The horizontal axis is time and the display will tell you how much time is represented by 1 division of the screen (bottom of the display, just right of center). The vertical axis is voltage and the volts per division is displayed bottom left corner for each channel.

To manipulate the display of the signal on the screen, use the **Scale Knob** for the horizontal axis and the **Scale Dial** associated with the desired Channel. Try adjusting both scales and observe what happens – as you change the scales, does the information on the oscilloscope still match the output of the waveform generator?

The waveform generator is computer controlled – the chip inside makes an assumption about the devices connected to it. The WFG assumes the oscilloscope has an input impedance of 50 Ohms and creates a model with that assumption, when in reality, it has a much larger input impedance. This erroneous model used by the WFG causes a discrepancy between the WFG display and the oscilloscope result.

To match the oscilloscope with the waveform generator, do the following on the oscilloscope:

1. Press the “Utility” button
2. Navigate the menu to set the output termination
3. Press the “Output Setup” menu key
4. Press the “Load” menu key to choose “High Z”
5. Press the “Done” menu key (*notice “High Z Load” in the upper right corner of the display*)

For all EEE 202 labs, we will use the High Z Load setting.

A few notes for taking measurements on the Oscilloscope:

1. Always use high resolution mode: “Aquire” → Mode → Hi-Res
2. Use the scope to display various statistics (peak-to-peak voltage, frequency, period, etc.) using the “Measure” button

**** End of Optional Part *****

Data Sheet Part 0

1) Note the maximum voltage of the Waveform Generator.
2) Adjust the WFG to output a 1V (peak to peak) sine wave with a frequency of 2 kHz.
3) Record the Oscilloscope input impedances of Channel 1 and Channel 2.
4) Record the maximum input voltages that can be applied to Channel 1 and Channel 2.
5) Record the voltage resolution of the oscilloscope.

Lab Instructions:

For the Final Design Project, Sparky Solar has asked you to measure the voltage and current of their solar panel systems. To effectively measure current, without impacting the load circuit that the Sparky Solar is providing power to, an op-amp will be used. Op-amps are very effective tools for measurement as they can boost voltages or currents to higher magnitudes (to make them more readable), they can reduce high voltages or currents to lower magnitudes (to protect measurement equipment), and they can change the phase of the signal, all without impacting the load that is being supplied.

Sparky Systems wants to build an amplifier design prior to the final design project to verify their assumptions. Using the lab instructions below, design an inverting amplifier, so that the Sparky representatives are confident in their design decisions (*and so you can get some experience before the Final Design Project*).

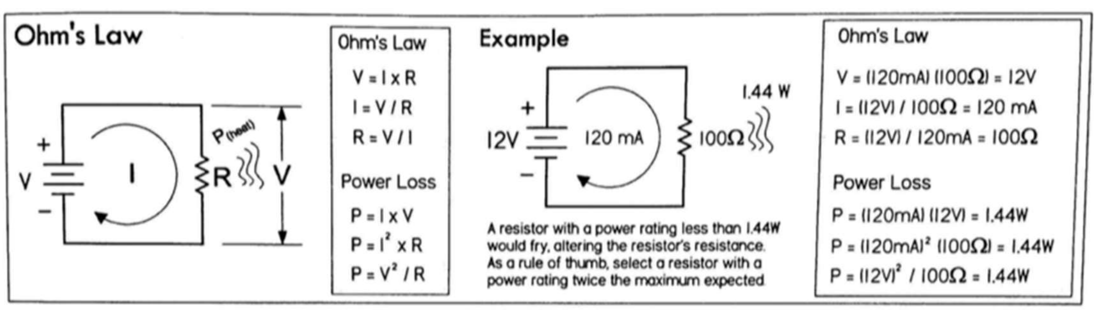
Sparky Systems has given you some specifications for your amplifier design:

1) The input AC signal has voltages ranging from 2 to 3 Volts
2) The output voltage desired from the amplifier is greater than 0 and less than 4 Volts.
3) The op-amp circuit output should be 2x the amplitude of the input.
4) The op-amp circuit output should be 180 degrees out of phase with the input (in other words, v_o is "inverted" compared to v_i).
5) You will have a single DC supply of 4 Volts to run your amplifier.

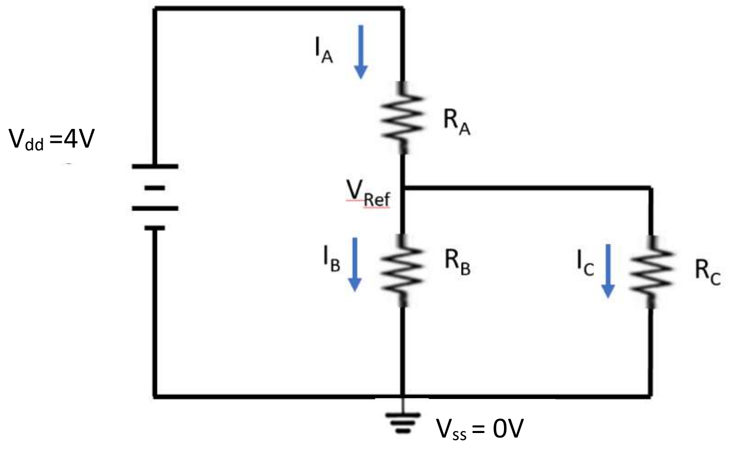
Step 1: Build a voltage reference

Virtual ground voltage reference

To operate an op-amp, you will need a high-side DC supply (V_{dd}), a low-side DC supply (V_{ss}), and a DC voltage reference (V_{Ref}). In this design, we only want to use one 4V power supply for the op-amp. We can set V_{dd} =4V and set V_{ss} to ground (0V). This means, we must create a "**virtual ground**" for V_{Ref}. If V_{dd} was 4V and V_{ss} was -4V, we could make our reference point in the middle at 0V, but that would require two distinct power supplies (a 4V supply and a -4V supply). Instead, we'll design a virtual voltage reference so that we can use a single 4V power supply. Remember this information from Lab 2 concerning Ohm's Law and power and power dissipated by a resistor. Feel free to refer to Lab 2, and the lecture materials for a review of these concepts.



You will need to make a resistor divider to create the voltage reference (V_{Ref}), which you will use as a "virtual ground" for the op-amp. Remember these are the considerations in making a resistor ladder:



Normally you build a resistor divider without R_C, but it has been included to **emphasize the assumptions of the resistor divider**, which is that the current $I_A = I_B$, that is $I_C \sim 0$, or at least can be considered <u>insignificant</u> compared to I_B. I_C is the current that this voltage reference V_{Ref} supplies to the op-amp pin it is connected to. We will see the impact of this in our op-amp design configurations.

$$\frac{\dfrac{R_B R_C}{R_B + R_C}}{\dfrac{R_B R_C}{R_B + R_C} + R_A} = \frac{V_{Ref}}{V_S}$$

Data Sheet Part 1 (You can use the 34401A or Portable Multimeters for this part of the lab)

1) Assuming $I_C \sim 0A$ for V_{dd} = 4 V what should be the relationship between R_A and R_B to set V_{Ref} = 2 V? On your breadboard, build the circuit **without R_c** and using large values for R_A and R_B (around 10 kΩ). With a multimeter, measure the value of V_{Ref} and record the value along with your values of R_A and R_B in the data sheet below. (*<u>Note</u>: the multimeter draws current out of the circuit during measurement, but it is assumed to be an insignificant amount so that it doesn't impact the measurement significantly*.)

2) Assume R_C = 1 MΩ and R_B = 100 kΩ in your circuit and calculate R_A so that V_{Ref} = 2 V. On your breadboard build the circuit with these values. If you cannot find a resistor with the value R_A in your inventory, use a combination of series and parallel making sure your V_{Ref} is close to 2 V. Measure I_A, I_B, and I_C in this case and record that along with your value of R_A in the Datasheet below (*refer to Lab 1 if you don't remember how to measure current – hint…break the circuit!*).

3) Assume R_C = 10 kΩ and R_B = 1 kΩ in your circuit and find R_A so that V_{Ref} = 2 V. On your breadboard build the circuit with these values, and make sure V_{Ref} is close to 2 V. Measure I_A, I_B, and I_C in this case and record that along with your value of R_A in the Datasheet below.

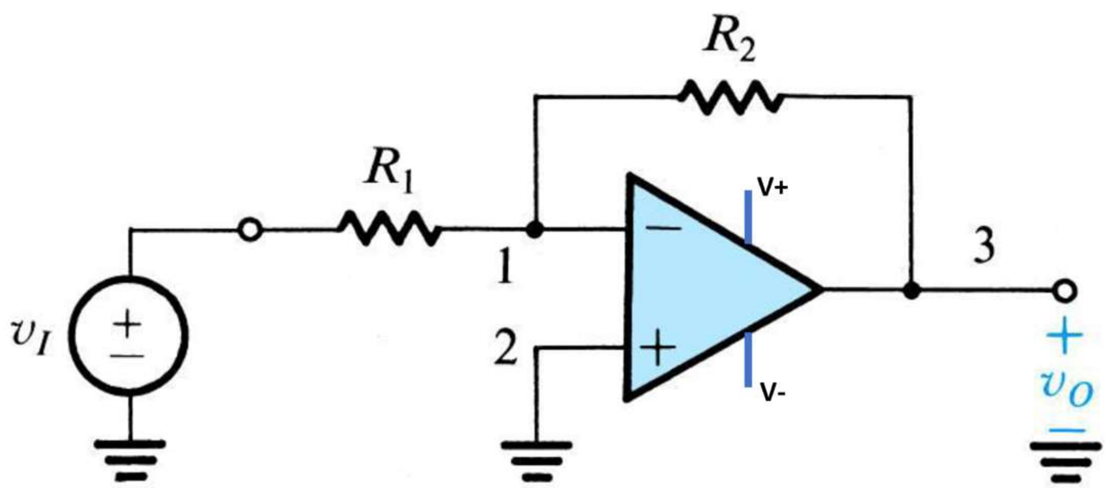
4) Assume $R_A = R_B$ and we want V_{Ref} to be at 2 ± 0.05V (that is somewhere between 1.95V and 2.05V) where R_C is 1 MΩ. What are R_A and R_B? On your breadboard build the circuit with the values you have chosen, then measure V_{Ref} with a multimeter. Get as close as you can to V_{Ref} and record R_A, R_B, and your measured V_{Ref} in the datasheet below.

5) Assume $R_A = R_B$ and we want V_{Ref} to be at 2 ± 0.05V where R_C is 10 kΩ. What are R_A and R_B? On your breadboard build the circuit with the values you have chosen, then measure V_{Ref} with a multimeter. Get as close as you can to V_{Ref}, and record R_A, R_B, and your measured V_{Ref} in the datasheet below.

For an **ideal op-amp**, we assume that the input resistance is **infinitely large** (*i.e. R_C is very big*). For actual op-amps in the lab, there will be a small current going into the input pins (*R_C will not equal infinity*) – think about how that relates to what you saw in parts 4 and 5 above.

Operational Amplifier (Op-Amp)
In the lecture, you learned about the calculations and applications of basic ideal op-amp circuit configurations with resistive feedback. Op-amp circuits are one of the most useful circuits across industries because they are fairly simple to design, low cost to prototype, and can accomplish a number of functions including amplification, buffering, mathematical operations, measurement, and more. All of the calculations and analysis done on op-amps in this course have assumed ideal op-amps. Let's start to simulate and build some physical prototypes to better understand how real op-amps work.

Data Sheet Part 2
The figure below shows an inverting configuration op-amp. Note that it requires a V+ and V- supply. For the MCP6022 and OP484 a DC supply voltage of $\pm$ 4V will work. This type of supply and operation with a positive and negative supply is referred to as dual-ended. This means input v_i and output v_o can have voltages that are positive or negative with values approaching (but never up to) the supply voltages you are using.

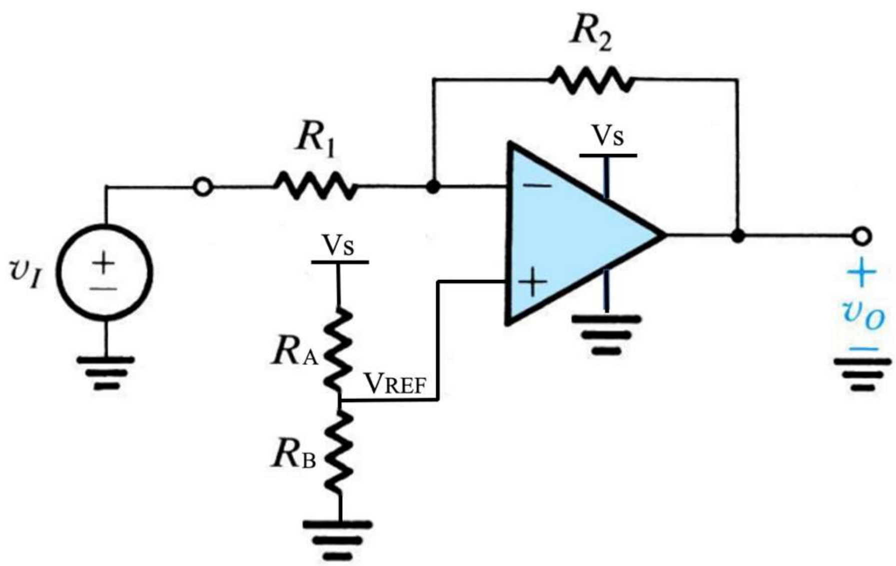


The design you will build will use a single battery supply, which does not have positive and negative voltages available. This is shown in the figure below. In this case v_i and v_o can have voltages between 0 and close to the supply voltage, Vs. This is referred to as a single-ended design.

In LTspice build the circuit shown below using a DC power supply Vs = 4V using the op-amp, resistors R1, R2, RA, and RB. In LTSpice use **Op07** (aka OP-7) for the Op-Amp model to simulate the physical Op-Amp.

Given what we know about the input impedance of the op-amp, which scenario from Part 1 should we use for R_A and R_B and how does the value of R_1 impact this?

This is an inverting configuration, single-ended amplifier circuit:



The inverting configuration equation is equal to $v_O = \frac{-R_2}{R_1} v_I$ (you derived this in the lecture), but we need to modify this equation slightly for our single ended design: $\left(v_O - V_{REF}\right) = \frac{-R_2}{R_1}\left(v_I - V_{REF}\right)$.

To achieve the amplification required, what ratio of R2/R1 do we need? If you wish you can assume a value of R1 = 10kohm.

What value will you use for R1 and R2?

Using LTspice, simulate your design. Use an input sine wave with a peak-to-peak amplitude of 1V and an offset of 2V with a frequency of 1kHz. Your input sinusoidal waveform should be a sinusoid that goes from 1.5 to 2.5 volts and has a period of 1ms (this means that the waveform repeats after 1 millisecond or 1/1000th of a second)

In the run control set the simulation type to transient simulation. Simulate for 0.01 sec with samples ever 0.1ms. Note the supply voltage is 4 V (DC). This sets the label VCC to 4 VDC. The resistor values are hidden by gray boxes, so you will have to fill in those boxes and then simulate.

When the simulation runs place a probe at the output of the Op-Amp and the positive terminal input source V1.

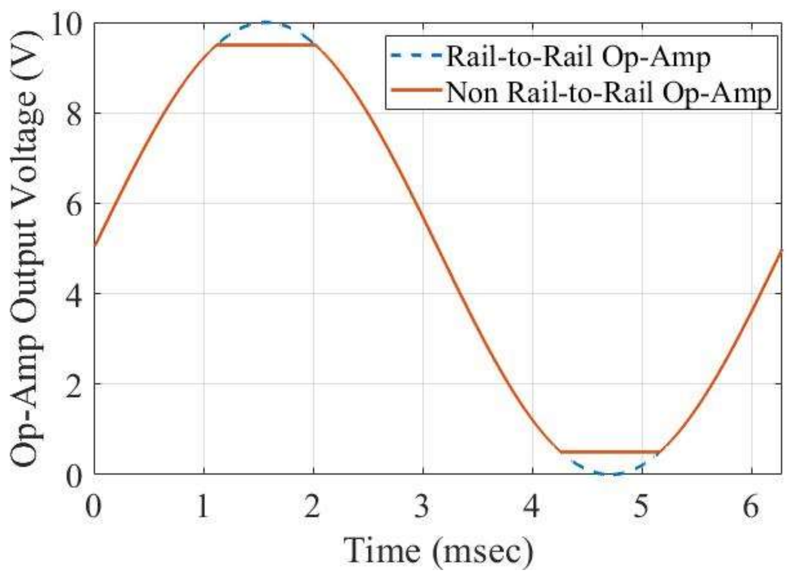
Data Sheet Part 2

- Record your calculations and values for R1, R2, RA, RB in the datasheet.
- Capture an image of your LTSpice circuit and include it with the datasheet.
- Capture the results of your simulation and include it with the datasheet.

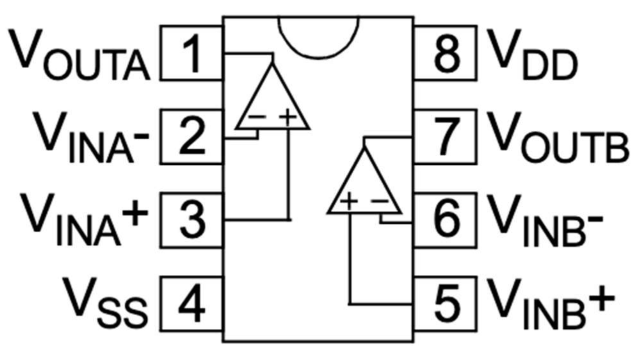
Data Sheet Part 3: Physical Validation

You will be using a rail-to-rail op-amp, the Microchip MCP6022, which will operate correctly with a supply voltage of about ±2V to ±6V or the Analog Devices OP484, which will operate correctly with a supply voltage of about ±3V to ±18V. A "rail-to-rail" op-amp means that the op-amp is designed to work very closely to the high and low supplies. An op-amp that is not designed to be rail-to-rail, will start to lose functionality as the inputs and outputs get too close the supplies (V_{dd} and V_{ss}). For this lab you will use an 8-pin or 14-pin through hole dual in line package (DIP). This is commonly used for prototyping with a breadboard. Surface mount devices have a smaller, more compact package and are more common on modern circuit boards. Most modern op-amps are available in either package.

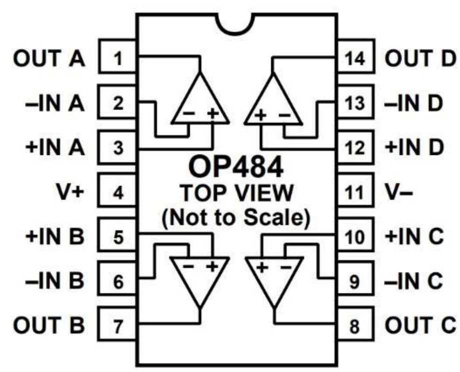
An Op-Amp is called a "rail-to-rail" op-amp if it can output voltage values that can reach the two "rails" of its power supply (i.e. the 0 and the +4V for your design). The following figure shows the output of a rail-to-rail op-amp and that of a non-rail-to-rail one with a supply voltage of 10V. The non-ideality in the latter results in a 0.5V "clipping", and thus distortion, to the signal.



The pin-outs for the op-amps ICs (integrated circuits) can be found below:



MCP6022 Pin-Out



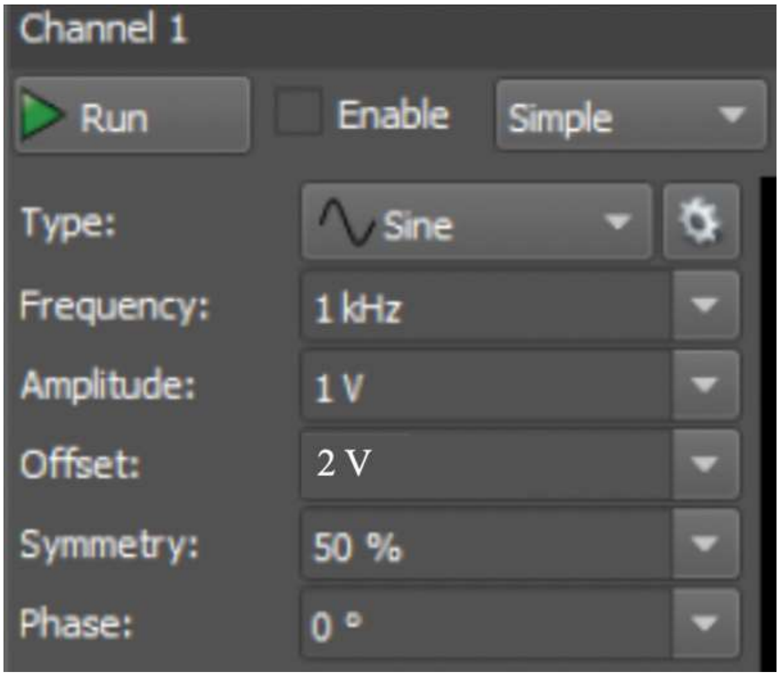
OP484 Pin-Out

As seen above, each IC contains more than one op-amp on the chip, so you could build multiple circuits with multiple op-amps using a single chip. You can determine the location of the pins on each physical chip using the semi-circle cut-out at the top (which is also on the physical chip). These pin-outs act as a map for you to build the circuit. *Note: The high-side and low-side DC supplies of the op-amps are shared on a single IC. For the MCP6022, VDD is the high-side supply, VSS is the low-side supply. For the OP484, V+ is the high-side supply, V- is the low-side supply.*

Use your prototype board (breadboard), op-amp and resistors and build the single-ended inverting op-amp circuit you simulated in LTspice.

Capture an image of your prototype board and include the image with the Datasheet.

Use the DC power supply for the 4V supply. Use the waveform generator to create the input signal with 2 V DC offset and 1V amplitude at 1kHz.



View this on an oscilloscope, connecting Channel 1 to the output of the waveform generator and Channel 2 to the output of the Op-Amp. Use Channel 1 as a trigger (or reference) and get a stable figure showing the input and output relationship plotted against time. This result should be similar to what you expected in your design and to your simulation results. If it is not, remember that the objective of physical validation is to circle back to your design calculations, simulation or breadboard work and correct mistakes or misunderstandings. It is very typical to find that the physical results don't match the ideal results exactly.

Data Sheet Part 3

- Capture an image of your physical prototype board and include the image with the datasheet
- Capture the oscilloscope screen image entirely and include it with the datasheet.
- Is your Op-Amp a "rail-to-rail" Op-Amp? If not, what are the upper and lower clipping voltages?
- Answer the discussion questions about the lab

Fill in your data sheet and print the required LTspice schematics, images, and waveforms. Labs can be done as a team, but each member must complete and submit their own data sheet.

Lab CLOS:

- Understand how to make a single-ended inverting op-amp design
- Use LTSpice to do a simulation verification of your design

- Use a waveform generator and oscilloscope
- Use a breadboard to do a physical validation of an op-amp design
- Understand how to design with inverting configurations of an op-amp