Analog 3 - Amplifiers II

The Operational Amplifier and Instrument Amplifier.

Preparations:

Please make sure you *do* prepare for this lab. This is part 2 of the amplifiers section. Make sure you don't spend an entire second lab period on the transistor amplifier and then not complete this lab up to the instrumentation amplifier. It will be better to abandon the single transistor amp and focus on this one instead. You will need your instrumentation amplifier later, as a pre-amplifier for a thermometer IC, which you will read out using the RPi and an analog to digital converter.

READ:

- Lecture Notes.
- Read Practical Electronics:
  - OpAmps: Chapter 8.3, 8.4, 8.5, then 8.17.
- If you want to read more about the OpAmp amplifier:
  - OpAmp: https://www.electronics-tutorials.ws/opamp/opamp_1.html
  - Inverting OpAmp: https://www.electronics-tutorials.ws/opamp/opamp_2.html
  - Non-Inverting OpAmp: https://www.electronics-tutorials.ws/opamp/opamp_3.html
  - Instrumentation Amplifier: https://www.electronics-tutorials.ws/opamp/opamp_5.html

Goals:

- Learn to make OpAmp amplifiers of various types:
  - Single stage inverting and non-inverting amplifiers.
  - Single stage filtering amplifiers.
  - Instrumentation amplifiers.
- Learn to make measurements on an actual amplifier circuit that works.
- Learn more about noise in a circuit.
- Create Bode plots of amplifier circuits and know what they mean.

Expectations:

- Keep good and accurate notes in your lab book.
- Carefully follow these instructions so you end up with the right measurements.
- For each of the main measurements, export both an image file and a data file. Analyze the data files later.
- Make sure you always record the settings of the oscilloscope.
- Compare measurements with expected values.

Introduction:

In the first part of this lab you learned to build a single transistor amplifier. Though such amplifiers are quite useful, especially for high frequency applications, they also leave lot to be desired: higher input impedance, lower output impedance, higher gain, more stability, to name just a few. All these features could be added using additional transistors, and if these are put in on a single chip, you essentially end up with an OpAmp.

Although Operational-Amplifier (OpAmp) seems to imply that you can make amplifiers with this device, they can form the basis for a very large variety of analog circuits, not only amplifiers and filters. You can create level detectors (Smith-triggers), peak value detectors, oscillators, integrators, differentiators, logarithmic amplifiers, just to name a few.

A Single Stage OpAmp Amplifier:

There are two basic types of single stage OpAmp amplifiers, the inverting amplifier, and the non-inverting amplifier, see the reading materials for details. Each of these has its own use, but here we will examine a non-inverting amplifier.

In this lab we will be using the MCP6002 CMOS OpAmp, see the data sheet on the course website (in Resources). The reason to choose this chip is that is can be powered by a single power supply at 1.8 to 6V. That makes it ideal for circuits that also work with the Raspberry Pi. Note that on circuit diagrams for Opamps, the power connections (Vdd = +5V and Vss = GND) are not shown for simplicity.

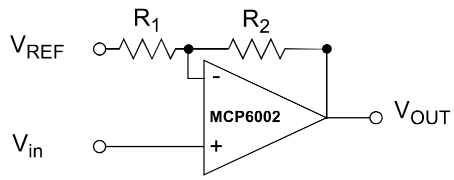


Figure 1, Non-inverting amplifier

Build the non-inverting amplifier shown in Figure 1, with a gain of 11, by choosing R_1=10kΩ and R_2=100kΩ. Start with V_{ref} at ground, and provide V_{dd}=5V power from the AD with V_{ss} to ground.

The amplification of this circuit is given by:

$$V_{out} = V_{in}\left(1+\frac{R_2}{R_1}\right) - \frac{R_2}{R_1}V_{ref}$$

Note that this circuit has a very high input impedance, that of the OpAmp. This is not the case with the inverting amplifiers, where the resistors lower the input impedance.

Tasks:

- Test your circuit using the wave generator to create various input signals and using the scope to look at the output. Initially, set Vref = 0V.
  - What is the problem with this circuit as designed?
  - For what situations would this circuit still be useful?

The issue with the single power supply version of this circuit is that the amplifier cannot amplify the negative part of the input signal. It goes below the lowest voltage available to the OpAmp. You can solve this in several ways:

1. Add an offset to the input so that the input signal is centered at $V_{dd}/2$=2.5 V. Note that we then also need to add this offset to V_{ref}, otherwise the offset is amplified as well and the result is an output that is too large.
2. **Optional:** Create a virtual ground reference with another OpAmp, which you set at $V_{cc}/2$, and set the reference, the input and the output signals relative to this virtual ground. (See lecture).
3. Power the OpAmp with V_{dd} =+2.5 V and V_{ss}=-2.5V, using both supplies of the AD. V_{ref} can now be at actual ground.

- You can try option 1 fairly quickly: Use the AD Wavegen 2 output to create a 2.5V constant voltage (set to DC and 2.5V), and use this for your Vref, then change Wavegen 1 to give a sine wave of 100mV amplitude with an offset of 2.5V. Look on your scope to check that the input voltage to your circuit is indeed a sine around 2.5V. The output should now also be around 2.5V but 11x larger.
- **Optional:** Note that method 2 would work well if the wave generator had an independent ground, which could then be connected to the virtual ground of your circuit. Since this is not the case, you can use a 1nF or 10nF decoupling capacitor and a 1MΩ resistor to virtual ground, to provide the required offset for this input. Such a circuit is shown in Figure 2.

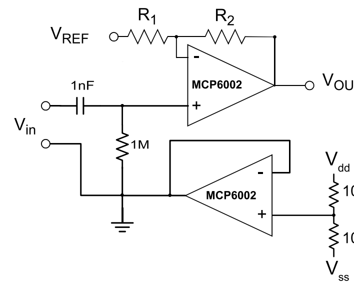
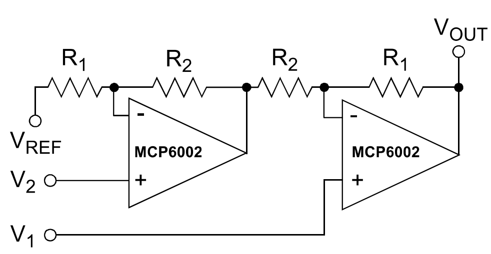


Figure 2, Non-inverting amplifier with virtual ground.

- It should be no surprise that for input signals (like audio) that have both positive and negative parts, the solution #3, with dual power supplies, is one of the more common choices. Change your circuit so that it corresponds to solution #3. Vref is now back to ground, and Vdd is at 2.5V (red wire of power supply) and Vss is at -2.5V (white wire of power supply).
- Look at your output signal now, and make sure it makes sense. Note the wavegen (channel 1) should have an amplitude of 100mV or less.
- For the solution #3 circuit:
  - Create a Bode and phase plot, from 10 Hz up to 10 MHz. (Make sure the input amplitude is 100mV, make sure your scope negative inputs (striped wires) are connected to ground and not -2.5V)
  - What is the approximate new gain-bandwidth (the frequency at which the amplification is 1)?
- **Optional:** We can change the frequency behavior of the amplifier by adding capacitors to our circuit, this way we create an *active filter*. You can do this by adding a passive filter to the input of the circuit, but this will now lower the input impedance. Instead, if you put a capacitor in parallel to R2, you will *reduce* the gain for high frequencies, effectively making a low pass filter. If you put a capacitor in parallel to R1, you would *increase* the gain for high frequencies, making an active high pass filter. You can do better with more complicated configuration, but that is beyond this particular lab.
  - Put a 1nF or 10nF capacitor in parallel to R2, and make a new Bode plot.
  - What is the approximate new gain-bandwidth (the frequency at which the amplification is 1)?
  - Is this behavior the same as you would get with a standard RC filter? How so, how not so?
  - Explain what this capacitor is doing to the amplification of your circuit.

An Instrumentation Amplifier:

An instrumentation amplifier typically has V_+ and V_- inputs so that the difference, $(V_+ - V_-)$, is amplified by some gain compared to a reference voltage V_{ref}. There are many different implementations, some with only one Op-amp stage, some with 3 or more stages, depending on the requirements. Usually, one of the requirements is high input impedance on both inputs, and high "common mode rejection", which means that if $V_+ = V_-$, but both have some common signal, then this common signal does not appear on the output. Figure 3 is a possible circuit for such an amplifier.



$$V_{out} = (V_1 - V_2)\left(1 + \frac{R_1}{R_2}\right) + V_{ref}$$

Figure 3, Instumentation Amplifer

Optional: Your circuit would most likely benefit from some additional capacitors to buffer the power on V_{dd}. Connect a 1μF or larger capacitor right from V_{dd} to the ground (V_{ss}) close to the chip. This circuit will be powered by a single 3.3V supply from the AD.

Build this circuit **on one of the small proto-boards.** That way you can save it for later.

Sensor:

In a later lab you will be connecting a TMP36 temperature sensor to an MCP3208, a 12-bit ADC, which you will read out with the RPi. The TMP36 will be powered at 3.3V. The TMP36 is designed so that the output is 750mV at an ambient temperature of 25°C, and scales at 10 mV/°C. You need to design your amplifier so that the output is 0V at 0°C and 3V at 45°C, thus reducing the normal range of -40°C to 125°C to 0°C to 45°C, and increasing the accuracy of the readout.

Needed components:

- One MCP6002 chips, which contains 2 OpAmps.
- Optional: C_1 – 1µF capacitors (not shown).
- Resistors.

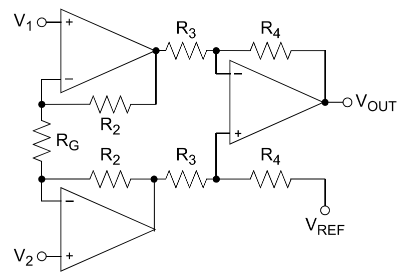
Tasks:

- Calculations. Do this for the circuit in Figure 3, or if you prefer the more complicated circuit in Figure 4:
  - Convert the information about the sensor into an input voltage range. The output voltage range is 0 to 3V.
  - From these values, calculate the gain you need and the offset voltage you need.
  - Choose good values for R_1 and R_2, and justify your choice.
  - **Design a voltage divider** that can provide the needed offset voltage for the V_2 input. Note that there is essentially no load on the divider if you choose R1 relatively large compared to the values of your divider. (Yes, you can "cheat", and use the WaveGen2 at constant output to get your V_2, but this makes your circuit less elegant, tying up another part of the AD.)
  - If you have any doubts or questions about the calculations, come get help.
- **Build the circuit on one of the separate proto-boards (not on the big one), so you can save it for a later lab.**
- Using the AD, power the circuit at 3.3V and test that it has the properties you want.
  - You can use the wave generator to create a slow DC signal (ramp-up) going from 500 mV to 950 mV to simulate the TMP36 over the temperature range and then use the Scope to show the input and output of your circuit.
- Document your findings carefully: input range, output range, gain.
- Measure the linearity of this circuit, i.e. how closely does the output follow the equation.
  - Are there higher order terms? (i.e. does the equation not quite represent the actual circuit?) The best way to do this is to record the DC sweep input and output from the AD, and then in for your progress report do a fit to the data. (Ask for help if you need.)
- Measure the noise of this circuit. How noisy is it?
- Measure the "Common Mode Rejection Ratio" (CMRR), to what extend can the circuit withstand noise that is on *both* inputs.
  - Connect V1 to V2 and use the signal generator to put a common signal on both inputs.
  - What is the amplitude dependence of the CMRR?
  - What is the frequency dependence of the CMRR?
- Measure how sensitive the output is to changes in the power level, i.e. change Vcc between 2.5V and 4.5V for fixed inputs, and see by how much the output voltage changes. Do this for several inputs.

Optional Extensions:

There are several ways in which you could improve this circuit:

- For temperature measurements, we do not need an amplifier that goes to very high frequencies, and 1MHz is clearly not needed. By reducing the band width of the amplifier to something very low (1 Hz would be enough!), see if you can reduce the output noise. Can you do this without making the inputs low impedance or reducing the accuracy of the measurement?
- You can also see if a significantly different choice for the resistor values, while keeping the ratio constant, would make a difference in the noise and output linearity.
- For the ambitious, a different design for a differential amplifier is shown in the circuit in Figure 4 . This circuit uses 3 Op-amp stages, so you need a second chip. It circuit should have a better CMRR due to the pre-stage gain set by R_G, at the cost that the input voltage range is reduced as the ratio: R_2/R_G gets larger. In addition, you can greatly reduce the bandwidth by placing a capacitor in parallel to R_4, which can be desirable to reduce noise.
  - ♦ Test and see if this circuit works better and is more stable than the 2 OpAmp circuit.



$$V_{out} = (V_1 - V_2)\left(1 + \frac{2R_2}{R_G}\right)\left(\frac{R_4}{R_3}\right) + V_{ref}$$

Figure 4, High performance instrumentation amplifier.

For Progress Report:

Required Parts in the report:

- An accurate copy of the schematic that you build in the lab, with a very brief section describing *your* understanding of what it does.
  - Show your calculations.
  - Why did you choose the specific resistor values you used?
- Show the input range and produced output and comment on the gain of the circuit.
- Comment on the noise of this circuit.
  - Do you think it will limit the accuracy of the thermometer?
  - Would the increased accuracy, which is what we set out to do, be obtainable with this amount of noise?
- Extra: Show the linearity of the circuit.
- Comment on the CMRR.
- Comment on the sensitivity to power supply voltage fluctuations. Do you expect this to be a problem?
- Standard sections: what you learned, who helped you, who did you help.