1 Introduction

In this project, we will build an electrocardiogram (ECG or EKG). An ECG is a noninvasive and painless test that measures the electrical activity of the heartbeat between pairs of electrodes placed at certain points on the skin. The heart is a relatively large piece of tissue, so the flow of electrical current associated with (and immediately preceding) contraction produces detectable voltages (typically a few millivolts) on the surface of the body, that oscillate at a frequency of around 1.3Hz or around 80 beats per minute.

This signal from the electrodes also has a small DC bias or offset, and will also have some high frequency noise from various sources in the lab. To be able to see this signal, we need to build an amplifier circuit that brings the small heartbeat signal up to around 1V (with the same frequency), attenuates the higher frequency noise from various outside sources inside the lab and finally sets the signal within a range that we can view easily through our computers next week (stay tuned!).

By completing this lab, you will:

- Enhance your skills in using the function generator and oscilloscope
- Learn how to use an integrated circuit by reading its datasheet
- Learn how instrumentation amplifiers work and how to get a desired gain
- Be able to analyze general operational amplifier circuits
- Recognize and analyze highpass, lowpass and bandpass amplifier circuits
- Learn some general noise reduction techniques

2 ECG System Diagram

Below is an overall diagram of the circuit we will be building in this lab. We will be working with a simulated heartbeat signal of 1Hz and 0.4mVpp (peak to peak voltage).

Given the limitations of our function generator, we start with a 20mVpp, 1Hz input signal (referenced to 2.5V). This signal then goes through:

- 50:1 voltage divider to go down to 0.4mVpp
- Instrumentation amplifier (at the right gain) to increase in amplitude (and offset)
- Amplifier circuit to remove DC offset, filter unwanted frequencies
3 Prelab

In previous labs, you have used mostly discrete components such as resistors, capacitors, inductors, MOSFETS, and one integrated circuit, the Arduino. Integrated circuits are a number of electronic components (resistors, transistors, and capacitors) wired together on one small piece of silicon that is usually packaged in plastic. In this lab, you will be introduced to two new integrated circuits (ICs).

For this lab, all ICs used will be in a DIP (dual inline package) package, which will be compatible with breadboarding. To enable people to more easily understand how to use ICs, most of them come with a miniuser manual called a datasheet. These are documents specifying a components geometry, internal circuitry, pinout (which pin is connect where) and performance. Such documents often also include information on how to use it for common applications. The data sheets for each of these parts are on the course website.

3.1 Instrumentation Amplifier (INA126P)

This amplifier provides most of the gain to the small heartbeat signal that comes from the electrodes. Because we’re measuring the difference in voltage between the two inputs rather than the absolute voltage, noise that “couples” into both signals is canceled out. If you twist the wires that connect to the inputs of this amplifier, you can create this coupling, and the noise will be the same on both wires and will be canceled out.

P1: Before breadboarding this chip, we need to know what the pins correspond to. Using the datasheet on the class website, label the corresponding pins in the figure below, and on the schematic symbol of the instrumentation amplifier on its right. Note that the V+ and V- pins are not shown in the schematic on the left; these are the power supply voltage pins, which we will still need to connect.

The gain equation for this op amp, with $R_G$ chosen to give a gain of $G$, is:

$$V_{out} = G \cdot (V_+^{in} - V_-^{in}) + V_{ref}$$

For example, if we input a 30mVpp sine wave and a 20mVpp sine wave, set our gain to 100 using $R_G$, and set the Ref pin at 0V, we will get out a $100 \cdot (0.03 - 0.02) = 1$Vpp sine wave, centered at GND (assuming our power supply range, from V- to V+, is large enough to support a ±1V output).

What if our power supply range cannot support this output voltage? Instead of going above $V_+$ or below $V_-$, the output signal will be cut off or ‘clipped ’so that it stays within the power supply voltages. Consider the following example. If we use 0V as our $V_-$ voltage, we will clip off the bottom half of the sine wave. The
transistors inside the op amp cannot drive negative output voltages without a negative voltage source. If we want to keep our power supply voltages at \( V_+ = 5V \) and \( V_- = GND \), we will need to change our reference voltage instead. All of our output voltages are referenced to this new value instead of to \( V_- \). For systems where voltages can be positive or negative, it is more convenient to have this reference voltage be in the middle of the power supply range. If our power supply voltage range goes from 0-5V, we can generate 2.5V as the reference voltage, so we can support sine waves that go above and below the reference without going outside of our power supply voltage range. Our 1Vpp sine wave would thus be centered at 2.5V and go from 2-3V.

This situation is shown in the figure below. The left pane is the output of an amplifier given a sine wave input, if \( V_+ = 5V \), \( V_- = 0V \) (GND), and \( V_{ref} = 0V \). The output can only be positive, so you get only the positive half of the sine wave. If \( V_{ref} = 2.5V \), we get the middle pane. Now the voltage can be positive and negative relative to this reference and still be within the power supply (between \( V_+ \) and \( V_- \)). In this case the reference is 2.5V above the lower supply, which is the same as saying that the lower power supply voltage is \(-2.5V\) relative to the reference. The right-most pane shows what happens if the amplifier wants to create an output higher than the power supply (or the highest voltage that the amplifier can create, which is generally a little bit less than the positive power supply voltage). In this case the amplifier will “clip” the output waveform, and distort the output signal. We want to keep the output signal within the voltage constraints of the power supply, so we will set \( V_{ref} = 2.5V \).

![Waveform Diagrams](image)

The voltage difference between electrodes connected to your body will contain the signal from your heart, which we want to measure, but there will generally be a DC voltage present too. That means that the signal you want to measure won’t be centered on \( V_{ref} \), but rather it will be centered on \( V_{ref} + \) a voltage, which we call the offset voltage. The output voltage in the right-pane of the figure has a small offset voltage. Now when you measure your heart beat, the offset voltage can be much larger than the signal. It can be up to \( \pm 10mV \), while the signal from your heart will be smaller than 1mV peak-to-peak (which means the sine wave amplitude is smaller than that of 0.5mV sin(2\( \pi ft \))). The next prelab questions look at how to set the gain and the reference of the instrumentation amplifier to ensure it doesn’t “clip” the output waveform.

**P2:** Let’s start off with the “Ref” pin set to GND. This configuration can only amplify positive signals: when the input signal is negative, the output signal is clipped, as shown in the left figure above. Assume the heartbeat signal can be represented by a 1mVpp sine wave, centered at 10mV (which means the max value is 10.5mV, and the min value is 9.5mV). If we connect this signal to \( V_{in}^- \), connect \( V_{in}^- \) to GND, and set \( R_G \) for a gain of 50, sketch the output waveform. Note that the gain applies to the offset voltage as well as the signal amplitude. On your output waveform, label the offset, minimum, and maximum voltages. **Note:** You are lucky that the offset is 10mV, so the signal is not clipped. If it had been -10mV, the entire output waveform would be outside the
Now using the same input voltage and offset, sketch the output of the instrumentation amp, but now connect the “Ref” pin to 2.5V. Can you still see the full waveform?

In truth, signals get cut off at a slightly lower voltage than the power supply. To be safe, we will try to keep our peak output voltages between 1V and 4V. With the same input signal specifications, compute the maximum gain you would like from your IA. That is, what is the largest gain that will stay within our bounds of 1V-4V? Use the same 1mVpk-pk signal with the 10mV offset.
**P5**: The datasheet says that \( V_o = G \cdot (V_{in+} - V_{in-}) + V_{ref} \), where \( G = \left( 5 + \frac{80\,k\Omega}{R_G} \right) \). Using this formula, what resistor value \( R_G \) will give you the gain computed in P4?

### 3.2 Operational Amplifier (LM4250CN)

We use this op-amp to filter the output of the IA (remove the DC offset, and some of the 60Hz power supply noise) and to increase the amplitude of the signal even more. We will use an inverting op-amp configuration, which means that the input signal is connected to the negative input, and the output reference voltage is set by the voltage connected to the positive input of the op-amp. Again to allow this op-amp to produce +/- signals, we will connect this + input of the op-amp to 2.5V.

**P6**: Using the [datasheet](#) on the class website, label the corresponding pins in the figure below, and on the schematic symbol of the operational amplifier on its right. Note that Rset will be connected between the 'Quiescent Current Set' pin and the negative power supply voltage.

Now that we can solve amplifier circuits, let’s analyze part of the circuit we are doing in this lab.

Below is a simpler version of this circuit.
**P7:** What is the overall impedance of the parallel combination of the 100 kΩ resistor and 0.1 µF capacitor in this circuit? This will be a function of frequency.

**P8:** Using the ideal op-amp rules for current and voltage at the op-amp inputs and the impedances in this circuit, write the nodal equation for this circuit. You can substitute $Z_p$ in place of the impedance you solved for above. *Hint: The nodal equation should be for the node at the - input of the op amp.*

**P9:** Now using this equation, find an expression for the transfer function for this circuit as a function of frequency, which should be in the form $\frac{V_o - 2.5}{V_{in} - 2.5} = \frac{A}{1 + \beta \cdot 2\pi f}$. The magnitude of this gain is the absolute value of this transfer function.

**P10:** What is the magnitude of this transfer function, in dB, (i) at very low frequencies? (ii) At very high frequencies? (Remember, the magnitude in dB is $20 \log_{10}(\text{magnitude})$.)
P11: Draw the magnitude bode plot for this circuit, labeling the important cutoff frequencies. Use the figure below for your plot.

![Magnitude Bode Plot](image)

P12: By looking at the magnitude bode plot, can you tell what this circuit does? (What kind of filter is it? E.g. lowpass, highpass, bandpass, bandstop)

Here is another simpler version of this circuit.
P13: Using the same steps as above, find an equation for the transfer function of the circuit, $\frac{V_{out} - 2.5}{V_{in} - 2.5}$.

*Hint: This transfer function will be in the form $\frac{j \omega f \cdot A}{1 + j \omega f \cdot B}$.*

P14: What is the magnitude of this transfer function, in dB, at very low frequencies? At very high frequencies? (Remember, the magnitude in dB is $20 \log_{10}$(magnitude)).
P15: Draw the magnitude bode plot for this circuit, labelling the important cutoff frequencies. Use the figure below for your plot.

P16: By looking at the magnitude bode plot, can you tell what this circuit does?

Now, below is the circuit we are doing in lab.
**P17:** Write the nodal equation at the negative input as function of $Z_s$, the impedance of the $47 \mu F$ capacitor and the $10 \, k\Omega$ resistor in series, and $Z_p$, the impedance of the $0.1 \mu F$ capacitor and the $100 \, k\Omega$ resistor in parallel.

**P18:** Finally, find an equation for the transfer function in the last circuit by rearranging your nodal equation and substituting in $Z_p$ and $Z_s$. You can leave the denominator in factored form (you don’t have to multiply out to find the $(j2\pi f)^2$ term).

**P19:** What is the magnitude, in dB, of this transfer function at very low frequencies? At very high frequencies? What is the magnitude between the two cutoff frequencies? *Hint: In the factored transfer function, your denominator should contain two $(1 + j2\pi f \cdot A)$ terms. Between the two cutoff frequencies, $j2\pi f$ will dominate the first of these terms, while the 1 will still dominate the*
**P20:** Draw the magnitude bode plot for this circuit, labeling the important cutoff frequencies. Use the figure below for your plot.
P21: By looking at the magnitude bode plot, can you tell what this circuit does?

4 Lab procedure

Below is the full circuit schematic. There are two main parts, the IA circuit and the Bandpass filter.

![Circuit Diagram]

4.1 Creating your 2.5V reference node

Before you build your circuit, create 2.5V from your 5V and GND power supply points. Grab two equal resistors (10 kΩ should work fine) and place them on your breadboard. Make sure this point will be easy to access because remember, all your signals need to be referenced to 2.5V!

![2.5V Reference Node Diagram]

4.2 Building the IA circuit

You will need:
- instrumentation amplifier (INA126P)
- 51k, 1k, and Rg resistor

Build the schematic shown below, and use the $R_G$ resistor that you obtained from the prelab. You might not be able to find a resistor with the exact value that you calculated, so it’s fine to use something close but not exact. **Remember to connect the reference terminal of the IA to your 2.5V supply** (this connection is not shown in the schematic). Your circuit will not work if you forget this connection!
Most of the function generators in lab cannot create signals smaller than 20mVpp. Because of this, first create a 50:1 voltage divider before one of the inputs of the IA by attaching a 51 kΩ and 1 kΩ resistor in series. Note: We use a 51 kΩ resistor because 50 kΩ resistors are not available in lab, but we will approximate this as a 50:1 divider because the difference is so small.

Now connect the signal generator to this resistor divider. The signal (red lead) should go to the 50 kΩ and the common (black part) to the 1 kΩ. Now input this 0.4mVpp at 1Hz into the differential input of your IA by connecting the output of this divider to the $V_{+}$ pin on your IA and the common to the $V_{-}$ pin. Also connect the common to the 2.5V reference so the sine wave will be centered at 2.5V instead of GND. This will yield the setup shown below:
L1: Measure the output of your instrumentation amplifier (again referenced to 2.5) and compute the gain of your circuit. How is this compared to what you expected? Is your waveform cut off? Take a picture of this output.

4.3 Building the bandpass filter

You will need:
- operational amplifier (LM4250CN)
- 47uF and 0.1uF capacitors
- 100k and 10k resistors
- 1 MΩ bias resistor

Build the op-amp circuit shown in the schematic. Do not connect this to your instrumentation amplifier circuit yet. Make sure to also connect the power supply pins on the op amp to 5V and GND. The 1 MΩ resistor connects to the quiescent current set pin. It’s used to bias the transistors inside the op amp so they will conduct DC as well as AC signals.

As analyzed in the prelab, this circuit is a bandpass filter. Using a test signal of 350mVpp, we will test the amplitude of the output at input frequencies ranging from 0.1Hz to 80Hz. Since we are using a 350mVpp input, we don’t need the attenuator that we built (but leave it on your board since we will need it later). Instead connect the signal output of the signal generator directly to the 47uF capacitor. Remember to leave the common (black lead) connected to your 2.5V reference.

L2: We want to characterize the op amp circuit above. Using a test signal of 350mVpp, find the amplitude of the output at 0.1, 0.2, 0.5, 1, 2, 5, 10, 20, and 50Hz (on a log scale, 1, 2, and 5 are about equally spaced, so this should make it easy for you to plot the data on a log-log scale). Use these data points to draw the bode plot of your filter. What is the approximate upper cutoff frequency of this bandpass filter?
L3: Put in a test signal at 350 mVpp, 1Hz and a 1V DC offset (Note: This offset should be added to the signal generator, as opposed to in the circuit - keep the black lead at 2.5V.). Take a picture of this. What happens to the offset at the output? Are these last two results consistent with the bode plot analysis from the prelab? Hint: What is the frequency of a DC signal? What should your circuit do to signals at this frequency?

4.4 Hooking up both amplifier circuits

Now, test both circuits by inputting a 1Hz, 20mVpp signal into the voltage divider (0.4mVpp after the divider) before the IA. Attach the output of the IA to the bandpass filter input, and measure the output of the op amp.
L4: Take a picture of your final simulated “heartbeat” output and verify that both stages work.

4.5 For Next Week

Next week, we will program the Arduino to take in this 0-5V signal and output it on your computer screens using a program called Processing. By using an isolator board, we can safely hook ourselves up using actual electrodes and be able to view the output on our computer screens. Hooray!

5 Analysis

A1: We can still see some noise at the output of this circuit. Suppose we wanted to try to reduce this noise by changing the filter stages. Could we reduce this noise further? Which components would we need to add or change? Since the heart beat is not a sine wave (it has a spike in it), do you think there might be a downside of doing this?

6 Reflection

Individually, answer the questions below. Two to four sentences for each question is sufficient. Answers that demonstrate little thought or effort will not receive credit!
R1: What was the most valuable thing you learned, and why?

R2: What skills or concepts are you still struggling with? What will you do to learn or practice these concepts?

R3: If this lab took longer than the regular 3-hour allotment: what part of the lab took you the most time? What could you do in the future to improve this?

7 Build Quality rubric

Your build quality grade in this project is based on the quality of your breadboarding and testing setup. You should make use of the oscilloscope probe clip-tips and BNC minigrabbers, and avoid stringing together jumper wires and alligator clips.

Check Plus

- Testing set-up is stable and consistent
- Breadboard layout is clean and organized
- Set-up is very stable with all components fitting comfortably into the breadboard
- Wires are color-coded and easy to trace
- Wire lengths are about right
- All power supply points are cleanly connected

Check

- Testing set-up is mostly stable, although some components or cables could be connected better
- Breadboard layout is organized, but could have been improved with some more careful planning
- Wires are mostly color-coded, and can be traced with minimal difficulty
- Wires lengths are mostly right
- Most power supply points are cleanly connected

Check Minus

- Testing set-up is not very stable, and several components are sub-optimally placed
- Breadboard layout doesn’t follow a consistent pattern
- Lack of color-coding makes wires difficult to trace
- Wires lengths are off and lead to spaghetti wiring, which may cause shorting
- Many power supply points are not very cleanly connected