MAS.836 Sensor Technologies for Interactive Environments
Lab 3: Bias, Active Filters, and Meaningful Design

The purpose of this lab assignment is to familiarize yourself with the implementation of biasing, active filters, and basic design techniques for signals and signal conditioning electronics. The goal is to prepare you to understand how to condition the signals from real sensors. In the next lab, you will synthesize these concepts to design and construct circuits for interfacing with several different sensors.

For now, the signal generator will be used to simulate sensors. This is often a useful design technique, as the output of the signal generator can be easily controlled to simulate different waveforms. This allows you to measure the response of your circuit to different frequencies.

You will construct your circuit from the parts included in your lab kit, in addition to any necessary resistors, capacitors, and diodes from the Responsive Environments stock area.

- Your **lab report** detailing your designs and answering the questions will be due in class, prepared neatly in hard-copy form.
- Your **functioning circuits** should be demoed to a TA for grading by 7 PM on the due date.

1 Your First Op-Amp Circuit

In this section, you will begin building your first simple operational amplifier circuit.

Op amps can only operate on signals that are within the power supply rails. If the op amp's supply terminals are connected to 0 V (ground) and 5 V, it cannot output signals outside this range.

AC signals oscillate around 0 V—that is, they go positive and negative. If our op amp is supplied by 5 V and ground, the negative portion of the input signal will be outside of this range and our signal will be clipped. One option is to power the op amp with a dual power supply providing, for example, both +5 V and -5 V. However, such a power supply adds complexity and expense; often it is desirable to build circuits that operate on a single positive power supply.

Another solution is to bias your op amp circuits. *Bias* refers to the neutral point of a circuit—it is a DC voltage around which the circuit operates. By redefining the resting point of the op amp, AC signals will oscillate around this point instead of zero. This allows active filters and other circuits that operate on AC signals to be constructed without the need for a negative voltage. The entire signal is offset up by the bias voltage to be entirely in the positive range. Usually, the bias point is chosen as half of the power supply voltage. In this case, we will use 2.5V as our bias voltage, giving equal headroom for our most positive signal (5 V) and our most "negative" signal (0 V). Bias will be an important concept in our circuits going forward—more background and suggestions for biasing many different types of op amp circuits can be found in the supplementary reading, *How to Bias an Op Amp* [2].

In this part of the lab, we'll build a way to generate our 2.5 V bias voltage and buffer it so that it can be used in many places throughout future circuits without affecting its voltage (causing it to droop because of too much load).

Familiarize yourself with the datasheet for your op amp, the TLV2374 [4]. Note especially the pinouts on page 3. Many datasheets, like this one, cover a handful of related parts, so make sure you find the pinout that matches the device you are actually using.

Note which pins are the input pins, which are the output, and which are the power supply pins. (Use extra care not to connect the power supply pins backwards, as this will destroy the op amp.)

Now design and build a buffered 2.5 V source. Do your best to keep this compact and neat, as you will use this circuit for the rest of the labs. **Draw a schematic of your circuit in your report, and measure** the output of your op-amp to make sure you have successfully created a 2.5 V bias.

Congratulations on your first working op-amp design!

2 Bandpass Filter Design

The following sections will work through a design example. Imagine that the signal generator is a sensor that we want to use to control an LED. We want the LED to turn on when a signal is present, but only when the signal is within a certain range of frequencies. The design strategies presented here will be useful for future labs and interfacing to sensors in general, so they are worth remembering.

In this part of the lab, we will focus on building the filter that will allow us to select signals within a particular frequency range. Filtering is a common technique for separating signals of interest from noise and other unwanted inputs. For example, imagine that instead of the signal generator we have connected a microphone, picking up all of the sounds outside. If we only want to light up our LED when people are talking nearby, we could design a circuit to try to filter out the low sounds of cars rumbling by or the high sounds of birds in a tree.

Another example could be a sensor that measures motion in a car, in which case we want to notify someone when we go over a hill or around a turn. We're not interested in the little bumps, but just the overall contour of the road—so we need to filter out the high frequencies of the bumps. The sensor may also drift over time, so we need to filter out accumlated offset as well.

These are two examples of times where we may want to use a bandpass filter. A bandpass filter performs the functions of both a high pass and a low pass filter, removing frequencies below a lower cutoff and above a higher cutoff, leaving just what's in between (in the *passband*).

Bandpass filters can be designed using manual techniques, as described in [3]. However, in practice it is often convenient to use an online calculator or design tool to select component values based on your design criteria. Many such tools exist—for this exercise it is suggested that you use the Analog Devices Filter Design Tool [1].

Design a bandpass filter, with a gain of 10 dB and a passband (-3dB points) running from 5 kHz to 12 kHz. If you use the filter design tool, you'll see some new terminology to define your filter. The important thing for us (for simplicity), is that your design has a 20 dB/decade[1]roll-off. We can make our roll-off sharper (and our filter more selective), but this will mean we have to use extra op amps and a more complicated design.

In order to achieve a single stage design, we need to define our *stopband* to be -20 dB, at a decade below and a decade above our passband. If our passband spans from 5 kHz to 12 kHz, we should specify our stopband to hit -20 dB over the span roughly from 500 Hz to 120 kHz.

The design from this tool will work with your op-amp. **Build the design on your breadboard, and turn in a schematic of your design.** Use your bias voltage circuit to bias your

[1] A *decade* is an increase or decrease in frequency by a factor of 10, which is convenient on $\log_{10}$ plots. Another commonly used interval is the *octave*, which is a doubling or halving of frequency.

filter.

Now it's time to test it in action! Hook your signal generator up to your new op-amp filter.

Questions

1. Produce a graph showing the absolute value of the amplifier gain as a function of the input frequency. The plot should be on a lin-log graph with the X-axis showing frequency on a logarithmic scale and the Y-axis showing gain in decibels, just like in our last lab. Make sure to base this off of your empirical observations of important points in the frequency response. **Clearly label the passband dB level and the slope of the roll-offs**.

2. Is your filter inverting or non-inverting?

3. How does your bandpass amplifier circuit's performance compare to the idealized version's -3 dB points and rolloff characteristics?

4. What is the amplitude of a 1 V, 7 kHz sine wave through this system? 500 Hz? 40 kHz?

5. What voltage level do you get on the output when you have no AC signal on the input?

6. What is the largest amplitude 7 kHz signal we can tolerate before we clip the output of our op amp? *(In many cases, we will* **know** *or* **measure** *the largest input signal we can expect from a sensor, and pick the gain of our filter (which we just designed) to scale our largest input signal to fit within the full range of our voltage rails.)*

7. What are the benefits of an active filter like this one over the passive filters we designed in the last lab?

3 Envelope Follower

Now we've taken our signal generator—which in this case, is proxying for a sensor we want to condition—and we've filtered it so that only the frequencies we care about remain. We've now hypothetically eliminated the low frequency drift and the high frequency noise from our measurement with our filter, so we're left with a clean, relevant signal.

Now how do we turn that AC signal into a smooth, continuous voltage that represents the amplitude of the signal? Here we use something called an envelope follower or a peak detector, which takes advantage of a new component: the diode. The diode allows current to flow in only one direction through it, effectively connecting (with a small voltage drop) two points, as long as the voltage on the anode side is greater than or equal to the cathode side. If it's not, it's like there is no connection there at all.

Figure 1 shows the schematic of an envelope follower.

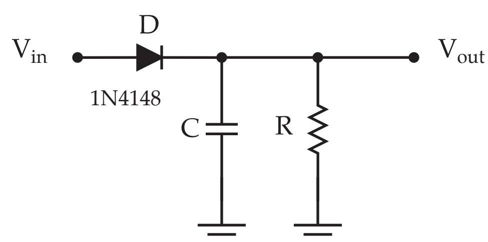

Figure 1: An envelope follower.

Any time we have a circuit with an RC connection like this, it is possible for the capacitor to store charge. When the capacitor and resistor are then isolated from other sources of current, the capacitor will discharge through the resistor. How quickly the RC circuit reacts has to do with its time constant, τ. Google RC time constant and study up!

For our purposes, we have designed our system to isolate a band around 7kHz. That means every $1/7000$ of a second, we expect a new peak. We need to select a time constant that will hold a voltage at least that long (in between peaks), but not *so* long that it won't quickly react and follow the overall shape of the signal, (its envelope).

For this example we can say τ should be 5-10x longer than $1/7000$, which should allow it to hold almost all of its charge over one cycle. Choose values for R and C given this constraint (remember our rules of thumb for resistor and capacitor values?)

Draw your schematic and hand it in with your lab report. Build your circuit on your board, connect it up to the output of your filter, and test your envelope follower.

Questions

1. What happens at V_{out} when the voltage on the anode side of the diode rises quickly?
2. What happens to V_{out} when the voltage on the anode side drops to zero after charging the capacitor?
3. Graph the expected V_{out} in response to the functions below.

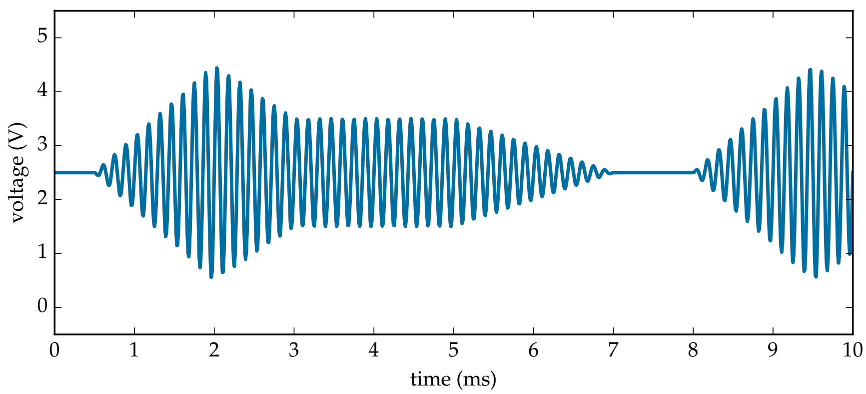

Figure 2: ADSR Waveform.

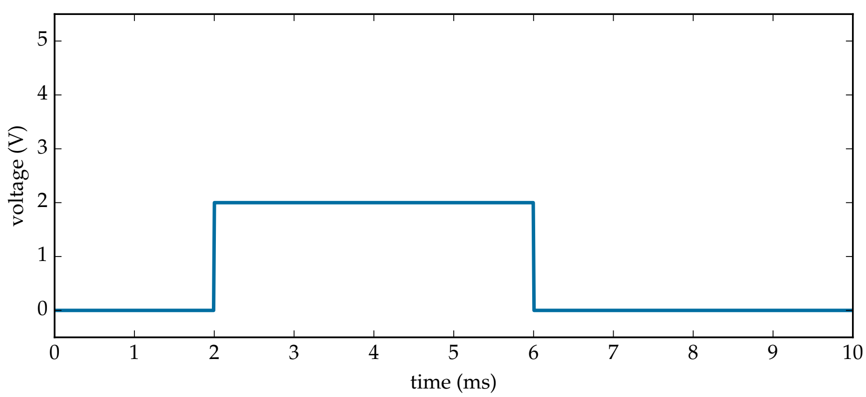

Figure 3: Step Waveform.

4. What is the voltage coming out of the peak detector when you have 0V at the input of the filter? When you have your maximum 7 kHz sine wave from the last problem?
5. Do you need to bias the peak detector section of your circuit? Why or why not?

4 Finally! Make that LED Shine!

Now we've taken our signal, filtered it, and figured out how to turn it into a nice, smooth voltage that will go up and down as the size of our frequency of interest gets larger and smaller. It's time to make it light up an LED—make it brighter when the signal gets stronger!

The brightness of an LED is approximately proportional to the current that passes through it. If we place a resistor in series with our LED, we can generally assume that the voltage drop across the LED is small enough to ignore it. This makes it easy for us to approximate the current we drive through the LED by just using $v = iR$ for the series resistor.

A reasonable maximum current to run through an LED is 20 mA (though your op amp may have trouble sourcing this much current.) The LEDs given in the kit can produce a significant amount of light at less than 5mA. Differences in brightness will also be most apparent for small currents through the LED.

Pick a resistor value, and design a circuit to hook up your LED to your peak detector. Assume you want your LED to shine brightly (20 mA) when it sees 5 V.

You should be able to change the brightness of the LED by changing the amplitude of your 7 kHz sine wave. You should be able to see that higher frequencies and lower frequencies outside of the passband of your filter don't cause the LED to change its brightness.

Demonstrate this circuit (with LED and also using the oscilloscope or multimeter) to one of the TAs. Also include your final schematic in your lab report.

Questions

1. Did you need to buffer your LED circuit from the peak detector?
2. What is the maximum voltage the TLV2374 can reach if it is at room temperature, sourcing 20 mA, and running from a single 5 V supply?
3. What is the voltage drop across the LED in your kit when it is on? (You should measure this.)

References

[1] Analog Devices. *Filter Design Tool*. 2016. URL: http://www.analog.com/designtools/en/filterwizard.

[2] Mark Feldmeier. *How to Bias an Op Amp*. 2007. URL: http://resenv.media.mit.edu/classarchive/MAS836/bias.pdf.

[3] Paul Horowitz, Winfield Hill, and Thomas C Hayes. *The Art of Electronics*. Cambridge University Press, 1989.

[4] Texas Instruments. *TLV2374 Datasheet*. 2001. URL: http://www.ti.com/lit/gpn/tlv2374.