MAS.836 Sensor Technologies for Interactive Environments
Lab 2: Introduction to Circuits, Filters, and Power Supplies

The purpose of this lab is to familiarize yourself with the implementation and measurement of simple circuits. With your kit of parts, you will be required to design, build, and test a number of basic designs. You are welcome to use any design that you choose, as long as it meets the design goals and uses the parts that you are given. Passive components are not included in your kit; they should be obtained as you need them from the Responsive Environments laboratory area. Connections on your breadboard should be made with solid-core hookup wire, which is available in several different colors at the lab benches.

Before beginning the lab, please check that your kit has all of its parts. Please write your name and e-mail address clearly on the bottom of your breadboard.

Lab kit contents:

- breadboard
- 9 Volt power supply
- power jack
- 7805 voltage regulator
- TLV2374 quad operational amplifier
- Metal electrode

For each lab assignment, you will need to complete:

- **Your functioning circuit on your breadboard.** Please try to keep your circuit neat so that it will not fall apart when handled. This should be demoed to the TA before 7PM on the due date of the lab.
- **Your laboratory report.** This should be a document containing the answers to the questions in the lab assignments and details of the designs you create. Include schematics and the reasoning behind your design choices, and plots of performance criteria requested in the assignment. This is due immediately after class on the due date.

1 Filters and Decibels

Now we are going to design a basic RC Low Pass Filter. Remove the voltage dividers from lab 1 from your board, and replace them with a simple Low Pass Filter with a cutoff frequency of 3kHz. You should only need two components.

Remember, try to keep your resistor values in the kΩ range (though a little lower and a few Mohms will still work) and your capacitor values in the nF range (pFs will work fine, up to a couple uF). If you need help, google around for RC low pass filter and RC time constant.

1. **Draw your schematic.**

2. Now sweep your sine wave signal, looking at both the input and the output. Start around 400 Hz and work your way up to 12 kHz. What do you see happening?

   We will practice plotting this on a log-log plot. Many things we perceive (like sound and light) we experience logarithmically. For instance, we hear pitch in octaves (20Hz and 40Hz – one octave – sounds like the same relationship as 10,000Hz and 20,000Hz!) We also experience loudness in decibels.

   Starting at 400Hz, measure the difference in magnitude (the peak to peak level) and Phase (the offset in time) from the input to output, **for each octave** *(an octave is a doubling of frequency)* until 12.8kHz. We're characterizing how the filter changes the input. Use the cursors on the oscilloscope to be more precise in your measurement.

   Plot your findings on the graph (next page). Remember, we need to convert our magnitude difference to dB, and our time offsets to phase (in degrees)!

   We can completely characterize a filter's behavior by comparing its output signal with respect to its input signal. To do so, we need to calculate amplitude and phase responses. To calculate amplitude response using a decibel scale, we use the formula

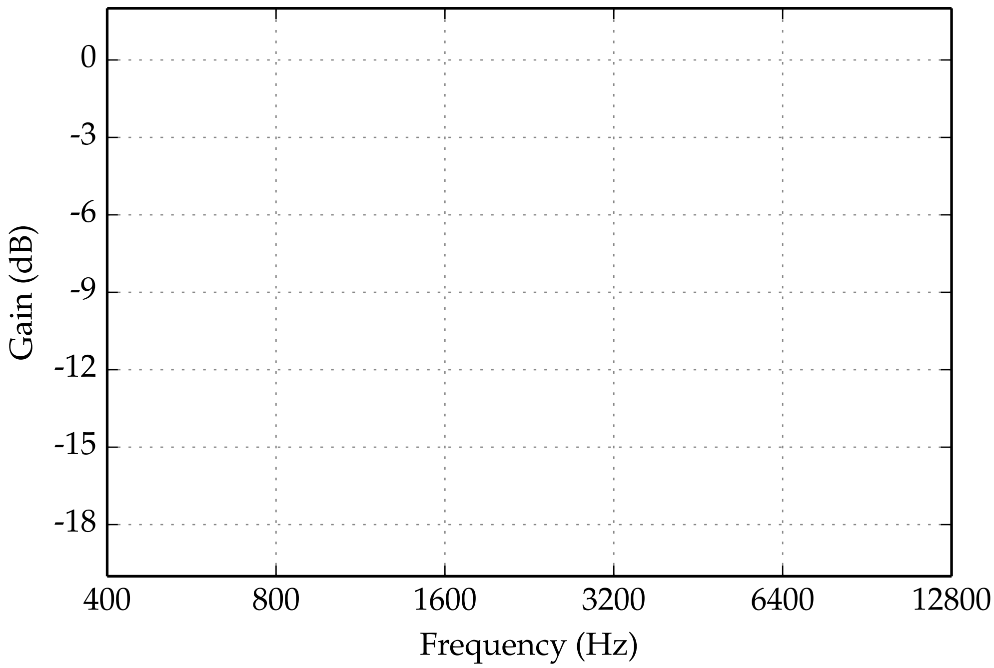
   $$\text{Gain}_{\text{dB}} = 20 \log_{10} \frac{V_{out}}{V_{in}} \tag{1}$$

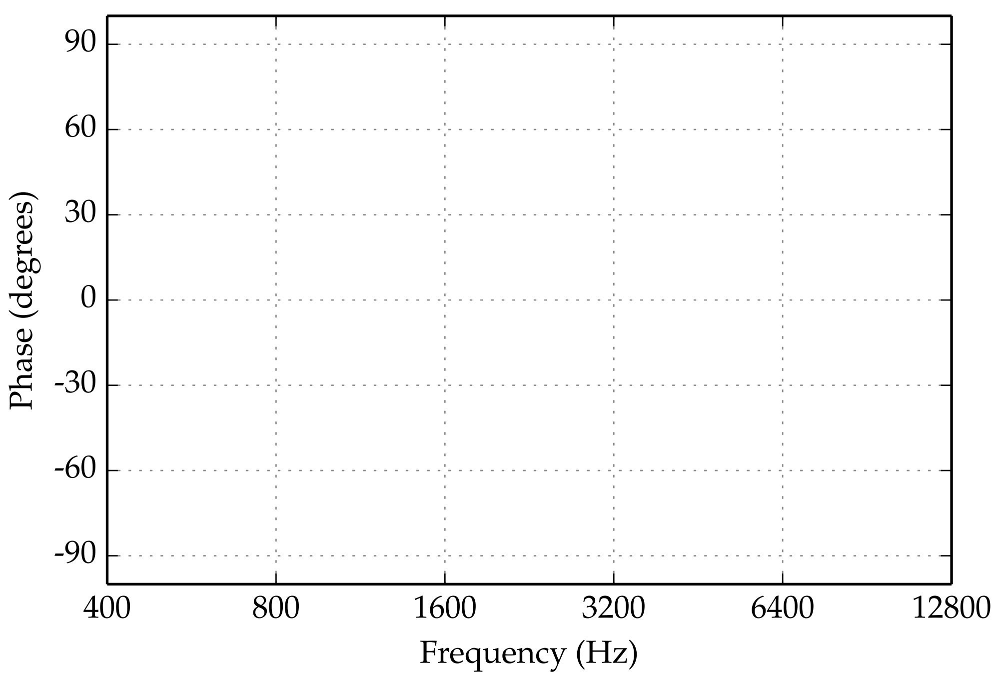
   A decibel scale always communicates a ratio of values. In the above formula, Vout refers to the peak-to-peak voltage of the output signal, and Vin refers to the peak-to-peak voltage of the input signal.

   There are other types of dB units that are changes relative to a fixed, known reference— like dB SPL (how loud sounds are) will always be in reference to 20 uPa, a unit of pressure (this means it would be the denominator in the equation above). If you see 'dB' by itself, it usually means we're look at a system- like a filter- to understand how it changes an output relative to an input.

   Phase refers to an offset of a signal in time and is measured in degrees or radians. One full cycle offset is 360 degrees, one half (out of phase) is 180 degrees, and so on. To calculate this correctly, we need to convert our frequency (in Hz, which is 1/seconds) to the period, lambda (in seconds): lambda = 1/frequency. If our time offset is equal to lamba/2, we are 180 degrees out of phase. If it's lambda/4, we're 90 degrees out of phase. It's up to you to come up with a formula to generalize this rule.

3. **What is the dB value if our filter does nothing to a given frequency**— if it simply lets it pass right through, and Vout = Vin?





4. Now we have built and plotted a first order filter. As we know from the very way we define decibels, a first order filter should have a roll-off of 20db/decade (a decade is a 10x relationship, like 20 to 200 or 1,000 to 10,000) or 6db/octave (an octave is a 2x relationship, like 50 to 100 or 400 to 800). When we measure a filter, the cutoff frequency is the frequency at which the output is 3dB lower than the input, referred to as the -3dB point. **How does your measured cutoff frequency compare to your calculated cutoff frequency?**

5. Frequently with filter design, the magnitude response takes priority, and the phase is treated as a bit of an afterthought. **What happened to the phase in your measurement? Can you think of a circumstance when you would only care about magnitude, and not the phase? Can you think of a circumstance when phase would be really important?**

6. Switch your input from a sine wave to a square wave at 2kHz. **What do you see happen on the output?**

7. Switch the resistor and capacitor in your circuit. Sweep through sine wave frequencies like you did with your previous filter. What happens to the magnitude? What happens to the phase?

   What type of filter did you just create? What happens when you put that same 2kHz square wave through?

2 Power supply construction

It is important to know how to create a constant voltage source, for two reasons. First, the integrated circuits (amplifiers and microcontrollers) that you will commonly use are designed to run at specific voltages. Higher or lower voltages than those specified will result in circuits that don't work, or damage to the components. Second, the power supply voltage must be kept relatively constant (for our purposes, within about $\pm$50 mV) as a reference to which to compare your sensor readings. Your sensor output will necessarily be a function of your power supply voltage, so the tighter you can regulate your power supply, the more accurate your measurements will be.

There are application-specific voltage reference chips that can be used in cases where a greater degree of accuracy is required, but for our cases, the power supply to our amplifiers will be an adequate reference.

Although there are many ways to construct a constant voltage source, one of the simplest is to use a three-terminal voltage regulator. These devices take a higher input voltage and regulate it down to a fixed output voltage that is ideally independent of the input. These devices have a number of important parameters to be aware of; some of the most important include *minimum headroom*, *maximum current draw*, *maximum input voltage*, and *maximum power dissipation*.

The *headroom* is the minimum voltage above the output voltage the regulator needs in order to stay *in regulation*, so that the output voltage is independent of the input. If there is not enough headroom, the regulator drops *out of regulation*, and the output voltage might be lower than specified and might pass through noise and fluctuations from the input. For example, if you have a 5 V regulator with a minimum headroom of 3 V, you must apply at least 8 V to the input of the regulator for it to stay in regulation. The headroom is also sometimes called the *dropout voltage*.

The *maximum current draw* is the limit on the amount of current that the regulator can supply to the connected circuit before it stops working, usually by overheating (possibly damaging the device, though some regulators will detect this and shut down to protect themselves.)

The *maximum input voltage* is the largest voltage which can be applied to the input terminal of the regulator before damage occurs, usually due to junction breakdown within the device, which can result in the regulator exploding.

Finally, the *maximum power dissipation* is the total amount of power that the device can handle before it overheats. The power is defined by equation 2:

$$P = i \cdot V \tag{2}$$

where P is the power, i is the current through the device, and V is the voltage drop across the device. As an example, if we had a 5 V regulator running off of a 10 V supply, through which we are drawing a 1 A current, the total power dissipated by the regulator will be:

$$P_{reg} = (10\,\mathrm{V} - 5\,\mathrm{V})(1\,\mathrm{A}) = 5\,\mathrm{W} \tag{3}$$

It is important to note that the maximum power dissipation and maximum current draw listed in the datasheet will only be attainable under ideal thermal conditions, which would require you to attach a heatsink to the regulator. In most cases, one can conservatively estimate at least 200 mA from a regulator in a TO-220 package without a heatsink.

The regulator that you will be using for the lab assignments is from the ubiquitous 78xx series of linear voltage regulators. The nomenclature for this series shows whether it is a positive or negative regulator, what the current limit is, and the output voltage; see Figure 1 for an explanation. There are other three-terminal regulators with different naming conventions, but this one is the most common.

The reason that these devices are called three-terminal voltage regulators is because they only have three electrical terminals: one for input voltage, one for ground, and one for output voltage. This makes them easy to use: you connect your power input to the input pin, your circuit to the output pin, and the ground pin to the ground of both your input power supply and your circuit.

It is important to place capacitors at both the input and output of these regulators to keep the

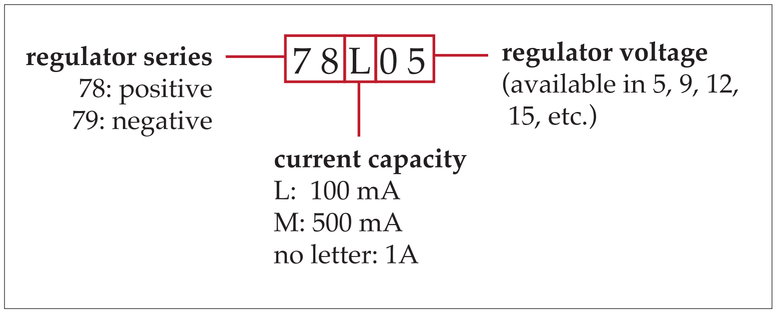


Figure 1: 78xx regulator series nomenclature

power supply voltage from oscillating too much. Your kit includes two 100 μF electrolytic capacitors; one should be placed at the input of the regulator and the other at the output. It also includes two 0.1 μF ceramic capacitors; one for the output of the regulator and the other as close as possible to your op amp.

The electrolytic capacitors have a high capacitance. They will charge up when you apply power to your circuit and will help supply a little extra current if the needs of your circuit fluctuate faster than the regulator can respond. This keeps your power supply nice and stable.

> **Note:** Electrolytic capacitors are polarized, which means that they have one terminal which must be kept positive with respect to the other terminal. If you place an electrolytic capacitor in backwards, it will explode and spew very hot electrolyte on you. Please be careful to note which pin is negative and connect that pin to ground. Also, electrolytic capacitors have a maximum working voltage, which must not be exceeded or the capacitor will explode. This value should be printed on the side of the capacitor. Before placing an electrolytic capacitor into a circuit, make sure that you will not exceed the maximum working voltage.

The 0.1 μF capacitors help filter out high frequency noise from your power supply. The noise can come from electrical interference in the environment and other parts of the circuit that draw varying amounts of current. Recall that capacitors have low impedance for high frequency signals (so the high frequency, unwanted noise is essentially "shunted" away to ground, while the DC power supply voltage is left alone.) Capacitors serving this function are often called *bypass capacitors* (as they are "bypassing" the noise straight to ground.) You'll typically see at least one for every chip in a circuit, placed as close as possible to its power pins for maximum effectiveness.

The datasheet for the 7805 regulator (`http://www.fairchildsemi.com/ds/LM/LM7805.pdf`) details its parameters and operation. Read through it to familiarize yourself with the device. Note which pins are the input, the output, and ground.

Use the regulator to build a regulated 5 V source for your breadboard. Place the regulator near the power connector, and place the two electrolytic capacitors as close as possible to the voltage regulator. Also place one of the ceramic capacitors close to the output of the voltage regulator.

You will be using this power supply for all of the labs, so build it neatly and compactly on your breadboard. You should also hot glue your power jack to the board for robustness.

Questions

1. What is the minimum headroom (dropout voltage) of the 7805 at 500 mA?

2. What is the maximum input voltage of the 7805?

3. What is the input voltage on your circuit board?

4. The maximum power that the regulator can dissipate depends on how quickly the power can be dissipated as heat, as shown in Equation 4, where T_{max} is the maximum allowable temperature, T_A is the ambient air temperature, and $R_{\Theta JA}$ is the thermal resistance between the junction and the air.

$$P_{max} = \frac{T_{max} - T_A}{R_{\Theta JA}} \tag{4}$$

   Assuming an ambient temperature of 20°C, what is the maximum power that your regulator can dissipate without a heatsink?

5. What is the maximum amount of current that your circuit can draw before the regulator overheats?

3 Putting it together

Now you've made two voltage dividers (in lab 1) and an RC filter. Let's pretend we now have a 5V 3Khz AC signal that has high frequency noise. The signal generator will play the role of the signal for this exercise. We want to both (1) attenuate it to 3.3V, and then (2) filter it using our RC filter design.

1. Combine the first voltage divider circuit (kΩ resistors) with your first RC filter.

   Measure the output. **What is the cutoff frequency? How does the measured cutoff frequency compare to the cutoff frequency that you designed your filter for?**

2. Combine the second voltage divider circuit (MΩ resistors) with your first RC filter.

   Measure the output. **What is the cutoff frequency? How does the measured cutoff frequency compare to the cutoff frequency that you designed your filter for? Explain your result.**

3. Look up buffer amplifiers. What do they do? Why and when do we need them?

   Draw the schematic of a circuit that uses a buffer to properly connect the attenuation circuit and filter circuit you designed.

4. If we assume we're going to connect the output of our filter to a High Impedance Load (like a digital input of a microcontroller), **do we need to buffer the output of the filter?**

5. If we assume we're going to connect the output of our filter to a Low Impedance Load (like a 4Ω speaker, to play music), **do we need to buffer the output of the filter?**