ECE-205 Lab 10
Lowpass, Highpass, and Bandpass Filters

Overview

In this lab you will build a simple lowpass filter, a simple highpass filter, an all pass (gain) amplifier, and then put them together to construct a simple bandpass filter. The filters we are building are actually very simple, and we are being very inefficient with our use of op amps. However, we are again going to build the final circuit in stages and construct the magnitude portion of the Bode plot for each part and then put them together.

PART 1 : System Overview

Our bandpass filter has three parts, shown in the different shaded regions in Figure 1. The equivalent block diagram is shown below the circuit, so you can match elements of the block diagram with the circuit subsystems. The first subsystem is the lowpass filer, the second subsystem is the highpass filter, and the last subsystem is an all pass filter which just adjusts the gain of the system. As with most of the circuits in this class, this design has not been optimized and is very inefficient, but should be fairly easy to build. All of the resistors are either 1 kΩ resistors, or are variable resistors. The capacitors are 1 μf capacitor. We will be utilizing TL072 chips, which include two op amps on each chip. *We will build and debug this circuit in stages, so do not start building yet!*

<u>***It is a good idea to disconnect the power from your circuit while you are building it.***</u>
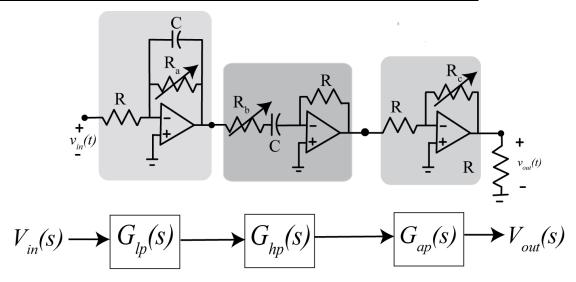

Figure 1. Bandpass filter consisting of a lowpass filter, a highpass filter, and an all pass (gain) filter.

The first thing we want to do is set the ground and the two power rails, just as we did for the previous labs. First connect one of the grounds (a down arrow) to the blue "rail" and then use another wire to connect the top and bottom blue rails together. It is better to do this somewhere away from the board where the USB and power connectors are. Next connect a wire from VP+ to the top red rail (this will be our V_{CC}) and connect a wire from VP- to the bottom red rail (this will be our $-V_{CC}$). Both V_{CC} and $-V_{CC}$ are just to provide the necessary voltages to our trop amp, *they are not the input signals.*

Next start Waveforms (the Digilent software), select the voltage icon, then set Positive Supply VP+ to 9 volts and Negative Supply VP- to -9 volts. Set the maximum currents for the power supplies to 100 and -100 mA. Do not turn the power on until you have constructed the circuit each part of the circuit, then turn the power on.

PART 2: Lowpass Filter

1) Build the lowpass, shown in Figure 2. We have included a load resistor for measuring the output. The input to this circuit should be the signal generator, and the output should be the voltage across the load resistor. Set the input to a 10 Hz sine wave with an amplitude of 1 volt (peak to peak value of 2.0 volts) and 0 volt offset. Set the **AC** channels to **200 mv/div** (the signals will fill up the screen) and **20 ms/div.** Adjust the variable resistor so the input and output signals have the same amplitude.
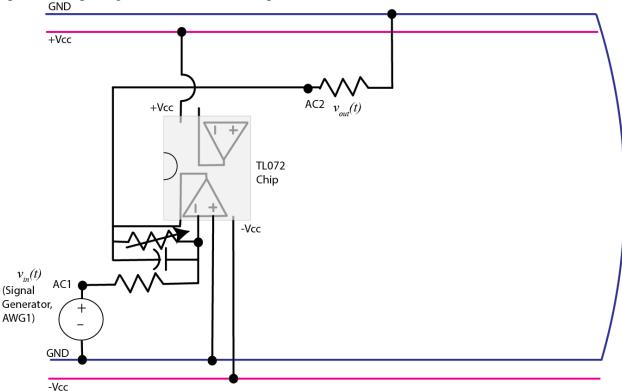

Figure 2. Lowpass filter.

The transfer function for this lowpass filter can be shown to be

$$G_{lp}(s) = \frac{-K_{low}\omega_{low}}{s + \omega_{low}}$$

where $K_{low} = \dfrac{R_a}{R}$ and $\omega_{low} = \dfrac{1}{R_a C}$

2) We will now construct the magnitude portion of the Bode plot for this subsystem, just as we did in Lab 9.

a) For input frequencies of 10, 20, 50 100, 200, 250, 500, 600, 700, 800, 1000, 1500, 2000, 2500, and 3000 Hz, measure the amplitude of the output signal (assuming the input remains at a 1.0 volt amplitude). It is probably easiest to use the **Measure** icon, then right click in the Measurements window and add the amplitude measurement. Note that as your input frequency increases, you will need to decrease the time/div setting. Try and keep the second channel to 200 mV/div. Enter your data into the program **process_data_low.m.** In this file, each row corresponds to a frequency. The three entries in the row are the frequency (measure in Hz), the amplitude of the input signal (measured in volts), and the amplitude of the output signal (measured in volts).

b) In the Matlab command window type **data = process_data_low;**

c) Now we will fit the data to our model using the program **model_low.m**. The arguments to this file are the data array from part b, the estimated amplitude of the gain, and the estimated value of the cutoff frequency. In the Matlab command window type **model_low(data, 0.3, 500*2*pi);** You should get results similar to that shown in Figure 3. Click on the **Datatip** icon (just below Desktop) to identify points on the graph. Once you have identified the maximum value, you may have to use a right click and create a new data tip (it usually helps if you right click when you are on a current datatip). You can fine tune your datatip locations by using the cursor keys on your keayboard. Your data points should be from the peak value to a value 3 dB lower. _Include this graph in your memo. Be sure your graph has two datatip points._
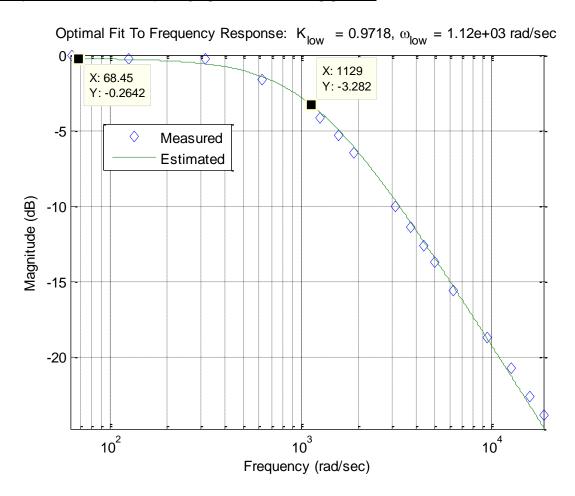

Figure 3. The lowpass filter with a bandwidth of approximately 1130 rad/sec (180 Hz).

3

PART 3: Highpass Filter

3) Now build the highpass filter as shown in Figure 4. We have again included a load resistor to measure the output voltage. Do not connect the two subsystems yet. Adjust the variable resistor so for an input signal (sine wave) at 1000 Hz with an amplitude of 1.0 volts (2.0 volts peak to peak) the output signal has the same amplitude as the input signal.
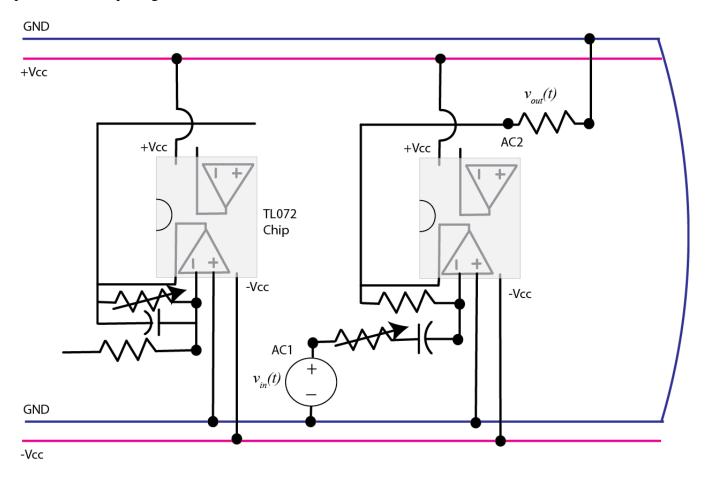

Figure 4. Highpass filter.

The transfer function for this subsystem can be shown to be

$$G_{hp}(s) = \frac{-K_{high}s}{s + \omega_{high}}$$

where $K_{high} = \dfrac{R}{R_b}$ and $\omega_{high} = \dfrac{1}{R_b C}$

4) Now we will construct the magnitude of the Bode plot for this highpass filter.

a) For input frequencies of 10, 20, 50, 75, 100, 200, 300, 400, 500, 750, 1000, 1500, and 2000 Hz, measure the amplitude of the output signal (assuming the input remains at a 1.0 volt amplitude). Enter your data into the program **process_data_high.m.** In this file, each row corresponds to a frequency. The three entries in the row are the frequency (measure in Hz), the amplitude of the input signal (measured in volts), and the amplitude of the output signal (measured in volts).

b) In the Matlab command window type **data = process_data_high;**

4

c) In the Matlab command window type **model_high(data, 0.3, 100*2*pi);** You should get results similar to that shown in Figure 5. Click on the **Datatip** icon (just below Desktop) to identify points on the graph. Once you have identified the maximum value, you may have to use a right click and create a new data tip (it usually helps if you right click when you are on a current datatip). You can fine tune your datatip locations by using the cursor keys on your keayboard. Your data points should be from the peak value to a value 3 dB lower. *Include this graph in your memo. Be sure your graph has two datatip points.*
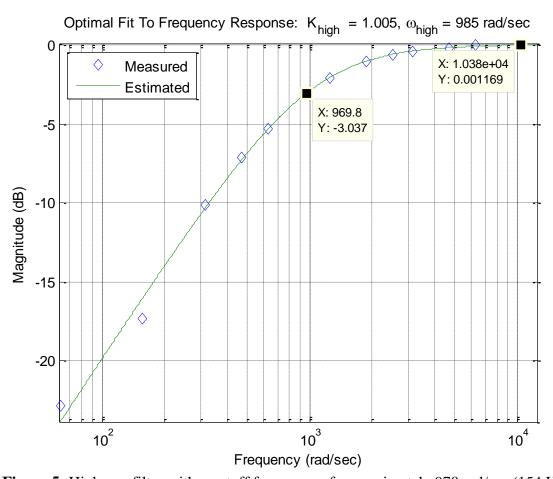

Figure 5: Highpass filter with a cutoff frequency of approximately 970 rad/sec (154 Hz).

PART 4: Allpass Filter

5) Construct the gain subsystem, as shown in Figure 6. Determine the frequency that is the average of the two cutoff frequencies for the lowpass and highpass filters you have constructed. Be sure to be consistent in using either radians/sec or Hz. Adjust the variable resistor so that for an input sine wave at this frequency (it should probably be between 100 and 200 Hz) with a 1 volt amplitude, the output has a 1 volt amplitude (or as close to this as you can get it).

The transfer function for this subsystem can be easily shown to be

$$G_{ap}(s) = -K_{ap}$$
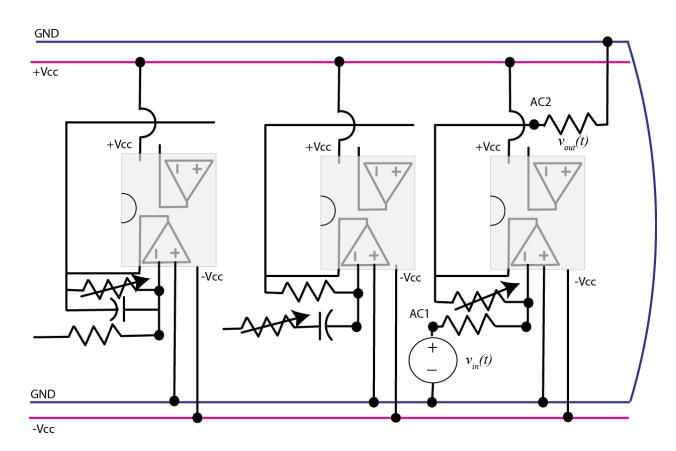
where $K_{ap} = \dfrac{R_c}{R}$

Figure 6. Allpass (gain) subsystem.

PART 5: Bandpass Filter

6) Connect all of the subsystems together as shown in Figure 7. Be sure to remove some of the load resistors we have been using.
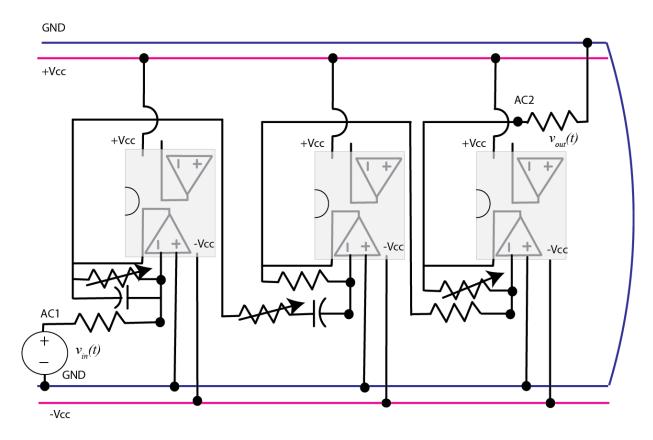

Figure 7. The bandpass system composed of lowpass, highpass, and all pass subsystems.

The transfer function for this system is then the product of the transfer functions of the subsystems,

$$G_{bp}(s) = G_{lp}(s)G_{hp}(s)G_{ap}(s) = \left(-\frac{K_{low}\omega_{low}}{s+\omega_{low}}\right)\left(-\frac{K_{high}s}{s+\omega_{high}}\right)\left(-K_{ap}\right)$$

We can simplify this as

$$G_{bp}(s) = -\frac{Ks}{s^2 + (\omega_{low} + \omega_{high})s + \omega_{low}\omega_{high}}$$

Since we are only interested in the magnitude, we can ignore the negative sign (but you will see it on the output since it will be out of phase with the input).

7) Now adjust the gain of the allpass filter (the last stage) so the out is equal to the input at the frequency that the output is the maximum. Start with the frequency you used in step 5 and look at frequencies above and below it to try and find the frequency with the maximum output.

8) Finally we will construct the magnitude portion of the Bode plot for this simple bandpass filter.

a) For input frequencies of 10,15, 25,40, 50, 75, 100, 200, 300, 400, 500, 600, 700, 800, 900, 1000, 1500 and 2000 Hz, measure the amplitude of the output signal (assuming the input remains at a 1.0 volt amplitude). Also include data near the frequency you estimated to have the maximum output (plus one frequency 5 Hz below and one frequency 5 Hz above.) Enter your data into the program **process_data_bandpass.m.** In this file, each row corresponds to a frequency. The three entries in the row are the frequency (measure in Hz), the amplitude of the input signal (measured in volts), and the amplitude of the output signal (measured in volts).

b) In the Matlab command window type **data = process_data_bandpass;**

c) Look at your previous graphs to determine the lowpass cutoff and the highpass cutoff freuquencies. For my graphs these are approximately 970 and 1130 radians per second. Use these in the next step.

d) In the Matlab command window type **model_bandpass(data, 1, 970, 1130);** You should get results similar to that shown in Figure 8. Note that your peak amplitude may be different depending on how much you could adjust the gain on the final stage. At this point we mostly are looking for the same general shape. Click on the **Datatip** icon (just below Desktop) to identify points on the graph. Once you have identified the maximum value, you may have to use a right click and create a new data tip (it usually helps if you right click when you are on a current datatip). You can fine tune your datatip locations by using the cursor keys on your keayboard.Your data points should be from the peak value to a value 3 dB lower on both sides. _Include this graph in your memo. Be sure your graph has three datatip points._ Note that the program may confuse the low and high frequencies, since they are symmetric there is no way to tell them apart. _Estimate the bandwidth of your system._

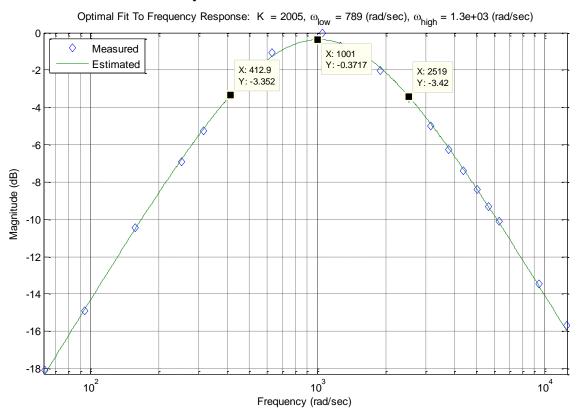**Send me your memo in an e-mail and you are done!**

Figure 8. Bandpass filter with both low and high cutoff frequencies identified. The bandwidth of this system is approximately 2106 rad/sec (335 Hz). Note that this is not a very good bandpass filter.