Lab 1: RC Circuits – Section AC

Taylor Fryett, Jake Garrison, Christyan Brown, Chinh Bui
EE 233 Circuit Theory
Department of Electrical Engineering, University of Washington, Seattle, 98195

Abstract – **In this lab, our main objectives are familiarizing with the lab instruments and observing the time characteristics of RC circuits. This is done by running two types of sources through the RC circuit and collecting and analyzing the data. By varying the parameters of the input signals, the effects on the output signals are observed then compared the theoretical data. Our findings show that our experimental data indeed lines up with the theoretical calculations, but there is error associated with the experimental procedure.**

I Introduction

In Lab 1, our main objectives include:

1) Familiarizing with test bench instruments such as the power supply, multimeter, function generator, oscilloscope, and the EE233 toolkit.

2) Observing the characteristics of the RC circuit with varying input functions and stages.

Using step-function and sinusoidal signals as inputs, we vary their parameters and observe the changes in output signals. The varying parameters of input signals include magnitude, frequency, and phase. The observed data of the output signal include waveform graphs, raise time, fall time, delay time, and time constant. We then compare the collected/measured data with the theoretical value from the prelab.

II Lab Procedure

In order to characterize the RC circuit under different input conditions, we used Tektronix Function generator and oscilloscope to alter the input parameters (magnitude, frequency and phase) and observe the resulting RC behavior. We used Channel 1 of the oscilloscope to monitor the voltage across the positive and ground nodes of the circuit. This provided information on the input signals (Vin). We used Channel 2 of the oscilloscope to monitor voltage across the 2 ends of the capacitor. This provided information on the output signals of the capacitor (Vout).

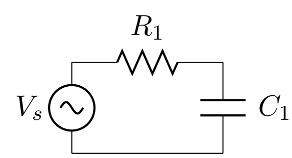


Figure 1: A simple RC circuit.

For the RC response to a Step Function, we first built the RC circuit according to Figure

1 and connected the function generator to the circuit's input. We then used the oscilloscope to collect the required data including: waveform graphs of Vin and Vout using the graphic display; period T, Vout(max), Vout(min) using the information provdided; time value of 10 percent, 50 percent, and 90 percent points of Vout using the sensor function; rise, fall, and delay times using the measurement function; 10 points for the rise and fall time curves to calculate time constant using sensor function. We also experimented with how different stages (see Figures 2 and 3) of RC circuits affected the output voltage

For the RC response to Sinusoidal Function, we used the same circuit in Step Function response but with different parameters and collected the waveform graph of Vin and Vout. We then varied the frequency of the input signals (10Hz-1MHz) by adjusting this parameter in the function generator and recorded how the amplitude of the output signals across the capacitor and across the resistor changed.

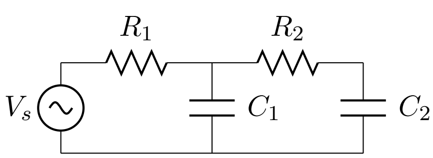


Figure 2: A 2 stage RC circuit.

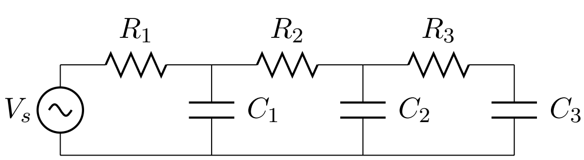
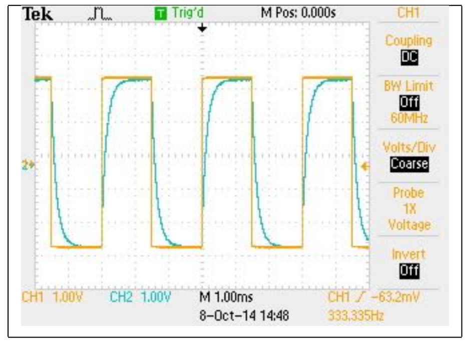


Figure 3: A 3 stage RC circuit.

III Experimental Procedure and Analysis

Question 1:
The waveforms are identical with the exception of a slight deviance in signal amplitude.



We began the lab by testing the circuit in figure 1. The first thing we had to ensure was that the period of our input square wave signal was large enough to be much larger than the RC time of the circuit.

Input Period (ms)
3.000 ± 0.001

Similarly we also had to make sure that the input voltage was always positive as to avoid putting a negative bias across the capacitor. As per the instructions we made sure that the square wave input signal had an amplitude of 5V, but also biased such that it never went negative.

Input Voltage Measurements	
Maximum Voltage	5.12 ± 0.01 V
Minimum Voltage	0.00 ± 0.01 V

Which gave rise to the corresponding output voltages that were read across the capacitor as placed in Figure 1.

Output Voltage Measurements	
Maximum Voltage	5.00 ± 0.01 V
Minimum Voltage	0.00 ± 0.01 V

We next wanted to characterize the response to the input waveform by measuring the rise time, fall time, and delay time. The rise time is given by the time that elapses between when the output signal reaches 10% of its maximum and when it reaches 90% of its maximum value. Likewise the fall time is the time it takes the signal to fall from 90% of the maximum value to 10% of the maximum value. The delay time on the other hand is the time between when the input signal reaches 50% of its maximum value, and when the output signal reaches 50% of its maximum value. Each of these times were measured on the oscilloscope by using the cursor function to find the location on the curve that corresponded to each of these values, which would allow the oscilloscope to display the time value that that point corresponded to.

Output Rise Time	
10% Time	13 ± 1 μs
90% Time	323 ± 1 μs
Calculated Time	310 μs
Theoretical Time	220 μs
Measurement Error	40.9%

Output Fall Time	
10% Time	314 ± 1 μs
90% Time	9.7 ± 0.1 μs
Calculated Time	304.3 μs
Theoretical Time	220 μs
Measurement Error	38.3%

Output 50% Delay Time	
Calculated Time	84.00 ± 1 μs
Theoretical Time	69.32 ± 1 μs
Measurement Error	21.2%

Question 2:
The tables shown above contain the measured values, and the calculated and theoretical rise, fall and delay times. The measurement error averages to around 30% . Possible sources for error reside in the component value tolerances, human error assiciated with moving the curser on the oscilloscope, as well as calibration error associated with the oscilloscope.

To gain an understanding of the automatic processes in the oscilloscope we also used the provided functions within the oscilloscope to compute the rise, fall, and delay times.

Oscilloscope Function Results	
Rise Time	285 μs
Fall Time	385 μs
t_{PHL}	1.492 ms
t_{PLH}	N/A

Question 3:
The following table is the rise time and fall time of V(out) obtained from: the oscilloscope, theoretical value, and our calculation from 4.1 part 2.

Output Rise Time	
Oscilloscope Time	285 ± 5 μs
Calculated Time	310 μs
Theoretical Time	220 μs

Rise time error percentage between oscilloscope time and calculated time: 8.06%
Rise time error percentage between oscilloscope time and theoretical time: 22.8%

Output Fall Time	
Oscilloscope Time	305 ± 5 μs
Calculated Time	304.3 μs
Theoretical Time	220 μs

Fall time error percentage between oscillo-

scope time and calculated time: 0.23%
Fall time error percentage between oscilloscope time and theoretical time: 27.9%

Similar to the case in the previous question, we believe that error source is associated with the calibration of the oscilloscope. Since the timeframe is in micro second, there's a possibility that the oscilloscope's measurement is off. Our calculated value from 4.1 part 2 also support this theory. Since our calculation uses the time value at 10% and 90% obtained from the oscilloscope, the error percentage between our calculation and oscilloscope measurement is relatively small (8.06% and 0.23%) compared to the error percentage between theoretical value and oscilloscope measurement (22.8% and 27.9%) We, however, didn't obtain the two delay times t(PHL) and t(PLH), but we believe the result would have similar error percentage to that of rise time and fall time.

Question 4: From the ten data points we collected along a charging cycle we were able to use natural logs to turn the exponential equation into a linear equation from which we used a least squares curve to find the slope which is one over the RC time.

RC Time	
$\tau_{one-stage}$	368.2726 μs

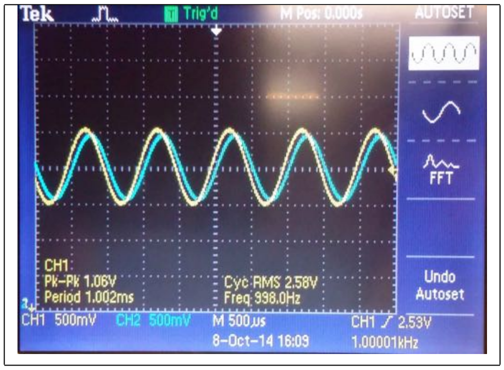
This result is nearly three times the predicted value of 0.1 ms (an error of about 200%). The first few data points had very large signal to noise ratios, and when plotted clearly were not consistent with with the rest of the data, which likely lead to a slope that was not representative of the actual curve.

Question 5: The theoretical values we should have measured are 3RC and 6RC for the two and three stage RC circuits, respectively. The theoretical RC time is 0.1 ms for the one stage circuit. Again we see errors that are upwards of 100%, and again the error was associated with the points with poor signal to noise ratios, which threw off the fit.

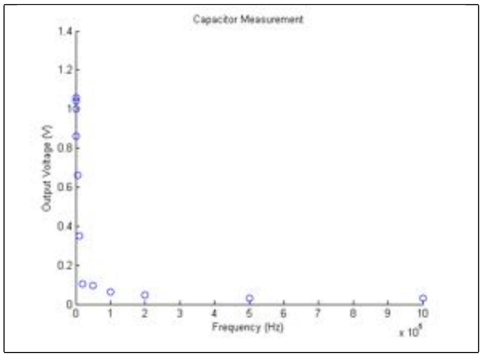
RC Time	
$\tau_{two-stage}$	190.9885 μs
$\tau_{three-stage}$	332.2723 μs

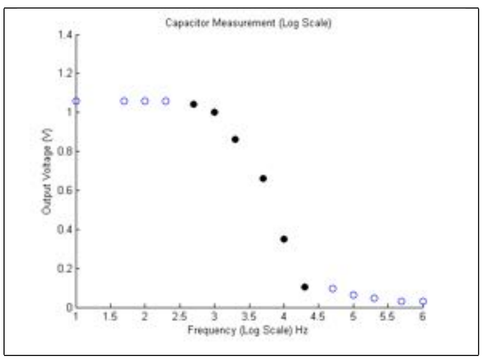
Section 4.2

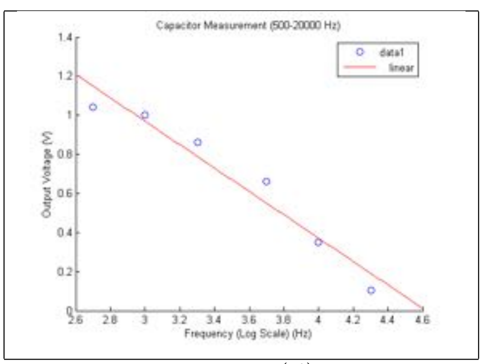
We use the same circuit from section 4.1 with R= 10k Ohm and C= 0.01 micro F, but set the function generator to provide sinusoidal wave input of amplitude 1V and frequency 1 kHz. Then we measure the voltage across the input and the voltage across the capacitor and record their waveform.



We then started changing the frequency of the input signal and recorded the changes in amplitude of the output voltage across the capacitor

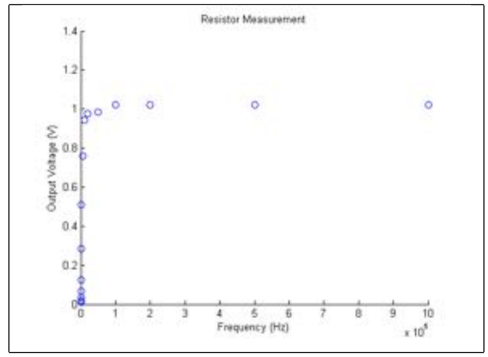


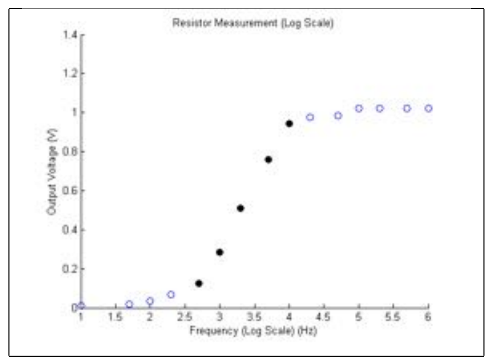


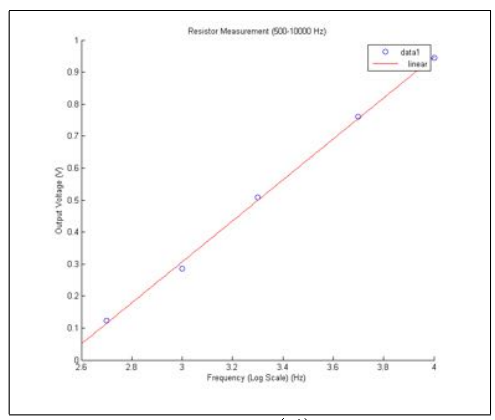


$$V = -0.598 \log{(f)} + 2.76$$

We then switch to measuring the output voltage across the resistor. We use the same circuit and method, only now channel 2 of the oscilloscope reads voltage across the resistor. Then we record the changes in amplitude of the output signal.







$$V = 0.641 \log{(f)} - 1.62$$

IV Conclusion

In this lab, we built and analyzed a simple RC circuit in order to gain a better understanding on RC circuits, the lab testing gear, and the general procedure for conducting and generating a lab report. We tested our circuit with both a step and sinusoidal source, and altered various source parameters such as frequency and amplitude, and analyzed the impact on the circuit by probing the output. The measurements we obtained from our testing were used as the foundation for our calculations to help better understand RC circuits. These calculations included rise, fall and delay time, time constant derivation and the error associated with them. We then compared our calculated results with our theoretical results and assessed why they differed. The overall error between our theoretical and measured values averaged to around 30% and the explanation for this essentially boiled down to the component tolerances and the precision of the testing gear. In addition we experimented with altering the stages of the circuit by analyzing the behavior of stage 2 ans 3. We then experimented with altering the frequency of the sinusoidal source and record how the magnitude of the output voltage changes. We record the output voltage for the capacitor and the resistor separately. From the collected data, we plot the graphs of amplitude of output signal vs frequency of input signal. The experiment graphs match with the our prelab graphs.

Team Roles	
Activity	Student Name
Prelab / Circuit Analysis	Christyan and Jake
Prelab / Simulations	Taylor and Chinh
Prelab / answer questions	Chinh and Christyan
Circuit construction	Jake and Taylor
Data Collection	Christyan and Chinh
Data Analysis	Taylor and Jake
Writing Lab report	Everyone

Lab 2: Amplifiers – Section AC

Taylor Fryett, Jake Garrison, Christyan Brown, Chinh Bui
EE 233 Circuit Theory
Department of Electrical Engineering, University of Washington, Seattle, 98195

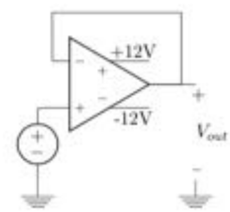
Abstract **– In this lab, our main objectives are to learn how to incorporate integrated circuit specification sheets, and how to use Spice MultiSim simulations to predict our circuit's behavior. Our ultimate goal is to start working towards our final project, making a complete audio mixer.**

I Introduction

In Lab 2, our main objectives include: learning how to use integrated circuit data sheets, as well as learning how to use Spice simulations such as MultiSim. In particular we are exploring the follower and summer circuits, which will be critical for the construction of our audio mixer.

II Lab Procedure

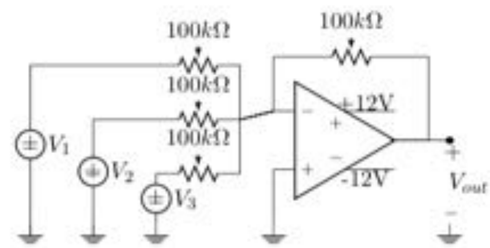
The general procedure for this lab was to build our circuit on a solder-less circuit board, provide the input with our Tektronix function generator, power our op-amp V_{cc} and V_{ee} inputs with a DC power supply, and measure the input and output waveforms on a Tektronix oscilloscope. For section 4.1 we made the following follower circuit



Circuit 1: The Follower

to test the limitations of the operational amplifier's slew rate, and how that would affect a range of signals. We drove a 50% duty cycle square wave with an amplitude of 10 V at 3 kHz, so we could easily measure the slew rate. Similarly we tested this circuit with a sine wave with a 3V amplitude to measure the lowest frequency where the signal is distorted. Our last test of the slew rate was to use a 100 mV sine wave to find the 3 dB point along the frequency range of our wave form generator.

For section 4.2, we used a basic summing amplifier.



$Circuit2 : The Summing Amplifier$

We also supply the amplifier with DC power. For the first part we set the function generator to provide V1 with signal V1 = cos(2π x 1000 x t). Then we sweep the frequency of input signal from 10Hz to 1MHz in 1-2-5 pattern while keeping the amplitude of the input signal the same and record the change in amplitude of the output signal. For the second part, we provide V1,V2, and V3 with 3 audio input signals and record the display of the output signal on the oscilloscope. Finally, we adjust the four potentiometers and observe how it affects the output sounds.

III Experimental Procedure and Analysis

SECTION 4.1

Part 1: In part one of 4.1, after setting up the circuit, we display the follower output voltage on the oscilloscope. After confirming that it's a follower voltage, we use the cursor of the oscilloscope to measure the time interval for the output voltage to reach the steady state and calculate the slew rate to compare with our theoretical value. We also compare the wave form obtained from the oscilloscope with the one we got from SPICE program

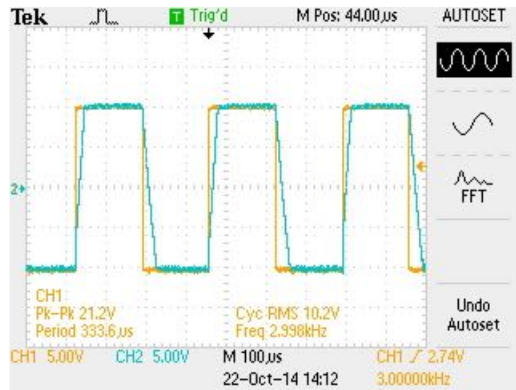


Figure 1: Voltage Follower output

Question 1:

$\Delta V = 20.2V$
$\Delta t = 22.4\mu s$
Slew Rate: 0.902 $\frac{V}{\mu s}$
Specification Slew Rate: 0.500 $\frac{V}{\mu s}$
Error Percentage: 44.6 %

The output waveform obtained from the oscilloscope shows similar characteristic with the waveform from the datasheet and from Multisim which is that of a voltage follower function

Part 2: In part two of 4.1, we change the input signal to a sine wave function. After confirming the output voltage is a follower one, we increase the frequency of the input signal until the output signal just start getting distorted and record the frequency to compare with our theoretical value

Question 2:

The frequency at which the output signal starts getting distorted is: 26 kHz

Using slew rate calculated in 4.1 part 1: 0.902 $\frac{V}{\mu s}$, max amplitude 6V, our theoretical max frequency before the signal getting distorted is: 24 kHz

Error percentage: 8.33 %

Part 3: In part 3 of 4.1, we change the input to sine wave. After confirming the output signal is a voltage follower function, we increase the input frequency while keeping input amplitude the same until voltage gain decreases to exactly half of low-frequency gain and record this high frequency

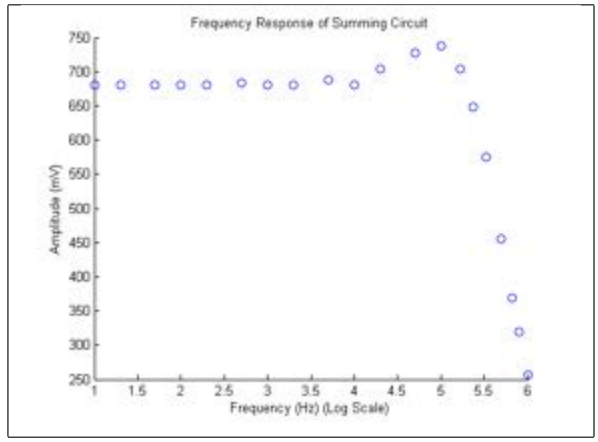
Question 3:

We measured the low frequency gain to be 1.08 by setting the input sinusoid to have a frequency of 10 Hz. Likewise we increased the frequency until our signal halved and found the -3 dB point to be at 2.1 MHz. Our answers from the pre-lab gave the -3 dB point as 800 kHz and low frequency gain of 1. The low frequency gain error might have been something as small as a poor connection which shifted the resistances of the leads by a couple of percent. The -3 dB point, however, is so far off that it is likely that we used a broken amplifier.

SECTION 4.2

Part 1: In the first part of 4.2, we change the frequency of the input signal in 1-2-5 pattern while keeping the amplitude the same. We record the output amplitudes with the correspondent input frequencies and plot a graph showing the relationship between them

Question 4:



In general, the graph we got from our data and the theoretical graph are identical. However, the graph from our data has a section where the amplitude raise a little before it start to decrease while the theoretical graph doesn't.

Part 2: In the second part of 4.2, we test the summing amplifier circuit with 3 audio sources as inputs (using three audio devices) then observe the output sounds from the speaker and output signals from the oscilloscope

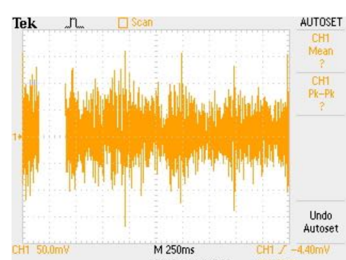


Figure 2: Audio waveform

Question 5:
The sounds of the audio signals from all three sources are mixed and played together in the speaker with a little bit of audible noise. The display of these signals looks like thousands of sine waves with different frequencies and amplitudes overlap each other. When we use the function generator to provide a 1kHz sine wave, we only get a single tone sound from the speaker

Part 3: In the third part of 4.2, we only adjust the four potentiometers and hear the change in output sound as we adjust each of them
Question 6:

Three of the four potentiometers control the input volume of the sources since they are wired in series with their respective signals. The fourth potentiometer controls the output volume of the three mixed sources. As the resistance of each of each input potentiometer is increased, the output of that signal is decreased; as the resistance of the output potentiometer is increased, the output for all of the signals is increased.

IV Conclusion

In this lab, we have learned how to use integrated circuit data sheet and learned using Multisim to build simulation of our circuit to predict its behaviors. We later experiment the characteristics of circuits built with op-amps (voltage follower circuit and summing amplifier circuit) by building the circuit, varying the input parameters, and collecting the output data then compare it with our theoretical values. Through the experiment with voltage follower circuit, we have learned about the slew rate, slew rate limitation, gain, and the relationship between frequency, amplitude, and gain. Through the experiment with summing amplifier circuit, we have learned about its frequency response as well as testing and confirming that the circuit works as a three channel mixer. We also experiment with the function of each potentiometer in the circuit. Our lab result, however, has high percentage of error most likely due to the malfunction of our op-amp.

Team Roles	
Activity	Student Name
Prelab / Circuit Analysis	Taylor and Chinh
Prelab / Simulations	Christyan and Jake
Prelab / answer questions	Taylor and Jake
Circuit construction	Jake and Taylor
Data Collection	Chinh and Christyan
Data Analysis	Christyan and Chinh
Writing Lab report	Everyone

Lab 3: Simple Filters

Taylor Fryett, Jake Garrison, Christyan Brown, Chinh Bui
EE 233 Circuit Theory
Department of Electrical Engineering, University of Washington, Seattle, 98195

***Abstract* – The purpose of this lab is to start preparing the equalizer for our final project, and to get more familiar with operational amplifier circuits.**

I Introduction

Our first project for this lab is to build the equalizer for our audio mixer preamp. (shown below)

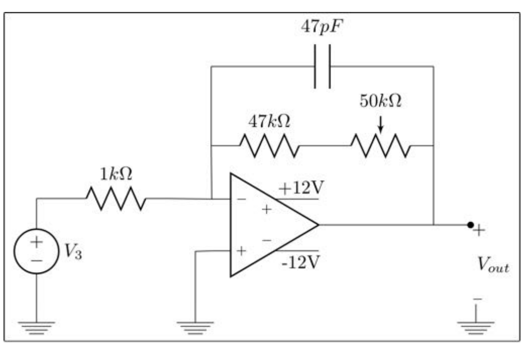


Figure 4.1

By performing a frequency sweep, while the potentiometer was set to be 0 $k\Omega$, on this circuit we generated a bode plot, which will help us understand our audio mixer's frequency response. Next we set the potentiometer to be its maximum value (50 kΩ), and repeated the same measurements to create a companion bode plot with the new resistance.

The second part of our lab was to build and characterize the summing amplifier. (shown below)

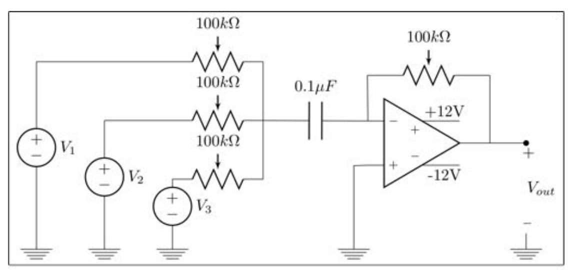


Figure 4.2

Our first task with this circuit was to set all of the potentiometer to 100 kΩ and again measure the frequency response, and make a Bode Plot from the data. Next we turned the input resistance of one of the channels to zero and put a low amplitude signal into it. By adjusting the resistor in the feedback loop we observed the limits that we can push our summing amplifier to without distorting our our output waveforms.

II Lab Procedure

In part 1 of section 4.1, we first build a preamplifier circuit according to the diagram of Figure 4.1, and the components of the circuit are used as directed in lab handout. We set the potentiometer to be 0 and provide the circuit with and input of 100mV amplitude. We sweep the frequency of input signal starting from 10Hz up to 1MHz in 1-2-5 pattern while keeping its amplitude the same. We record the amplitude of output signal and the time delay between input and output signals to calculate the phase change of output signal.

In part 2 of section 4.1, we use the same circuit of part 1 but apply an input of amplitude 300mV and frequency 300Hz. We display the input and output waveforms on the oscilloscope, adjust the time base to display 3 complete cycles, and get a hard copy of the oscilloscope screen.

In part 3 of section 4.1, we use the same circuit of part 1 but with the potentiometer set to 50kΩ. We provide the circuit with and input of 100mV amplitude. We sweep the frequency of input signal starting from 10Hz up to 1MHz in 1-2-5 pattern while keeping its amplitude the same. We record the amplitude of output signal and the time delay between input and output signals to calculate the phase change of output signal.

In part 1 of section 4.2, we first build a summing amplifier circuit according to the diagram of figure 4.2, and the components of the circuits are used as directed in lab handout. We set all the potentiometers to 100kΩ and apply an input of amplitude 500mV to the circuit. We then proceed with the same procedure of 4.1.1 and 4.1.3 to record the amplitude and time delay of the output signal.

In part 2 of section 4.2, we use the same circuit of 4.2 part 1 but with R1 set to 0 and Rf set to 50kΩ. We apply an input of amplitude 300mV and frequency 300Hz. We display the input and output waveforms on the oscilloscope, adjust the time base to display 3 complete cycles, and get a hard copy of the oscilloscope screen.

In part 3 of section 4.2, we set R1 to 1kΩ and apply an input of amplitude 1V and frequency 1kHz and display the output signal on the oscilloscope. We then increase Rf until the output wave becomes distorted to investigate this behavior.

In part 4 of section 4.2, we use the same circuit and apply a low frequency to V1. We connect Vout to a speaker and listen to the sound in the speaker. We later remove the capacitor of the circuit and compare the changes in sound of the speaker when the capacitor is removed.

III ExperimentalProcedure and Analysis

Section 4.1

In part 1 of section 4.1, we calculate the phase change of the output signal using the recorded delay time and the equation

Phase change $= \frac{timedelay}{T} * 360$

With T calculated from the frequency that corresponds to the time delay.

Preamp Bode Measurements (0K)		
Freq. (Hz)	Amp. (V)	Phase (°)
10	4.64	183.6
20	4.64	181.44
50	4.64	180
100	4.64	180
200	4.64	181.44
500	4.64	176.4
1k	4.64	176.4
2k	4.71	172.8
5k	4.56	162
10k	4.00	144
20k	2.96	126.72
50k	1.52	100.8
100k	0.880	90
200k	0.560	77.76
500k	0.160	72.0
1M	0.064	54.0

Question 1:
The following Bode plot agrees well with the Bode plot and formula we generated in section 3.2.5 and 3.2.10. We measured a low frequency amplitude of 4.64 V while we predicted an amplitude of 46.98 which is well within the tolerances of the devices we are using.The phases likewise are in agreement.

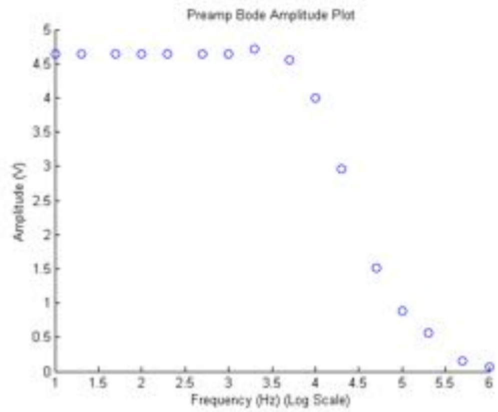


Figure 1: Preamp Amplitude (0Ω)

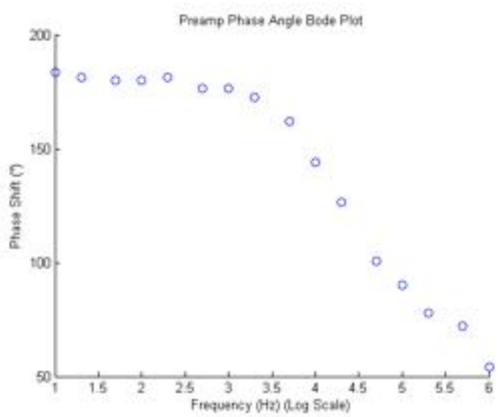


Figure 2: Preamp Phase Delay (0Ω)

Question 2:
In part 2 of 4.1, we took a hard copy of the oscilloscope screen.

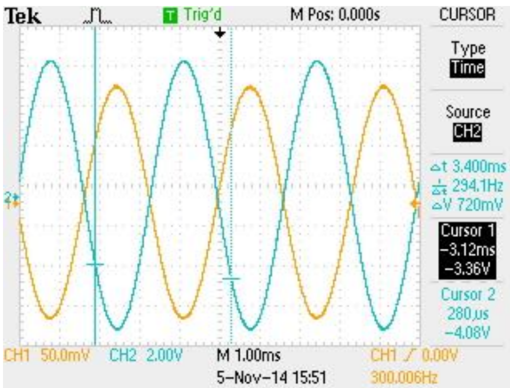


Figure 3: Preamp signal output

We confirm that circuit is an integrator as the sine wave function of the input voltage becomes a cosine wave function of the output voltage.

Preamp Bode Measurements (47K)		
Freq. (Hz)	Amp. (V)	Phase (°)
10	10	180
20	10	184.32
50	10	176.4
100	10	180
200	10	178.56
500	10	176.4
1k	10.2	176.4
2k	9.8	164.16
5k	8.4	144
10k	6.2	122.4
20k	3.52	109.44
50k	1.6	93.6
100k	0.880	90
200k	0.416	83.52
500k	0.164	72
1M	0.048	54

In part 3 of section 4.1, we repeat the step from part 1 to calculate phase change then make bode plot of amplitude vs frequency and phase change vs frequency

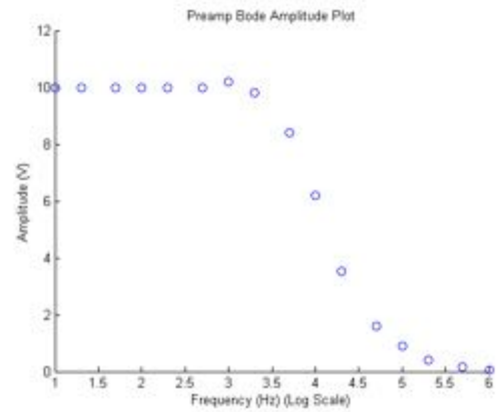


Figure 5: Preamp Amplitude (47Ω)

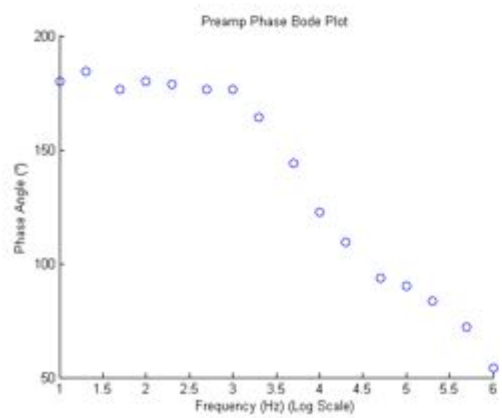


Figure 5: Preamp Phase Delay (47Ω)

Question 3:
The potentiometer makes the signal amplitude much higher and makes the knee in the amplitude come at a lower frequency. This makes sense as the potentiometer effectively increases the feedback resistor which also increases the amplification of the circuit. Furthermore since the feedback resistor is now effectively larger than the relative magnitude of the capacitor becomes much smaller at a smaller frequency.

Section 4.2 In part 1 of section 4.2, we repeat the same step of 4.1.1 and calculate the phase change of this summing circuit. We then make the bode plot of amplitude vs frequency and phase change vs frequency

Equalizer Bode Measurements		
Freq. (Hz)	Amp. (mV)	Phase (°)
10	244	237.6
20	368	221.76
50	520	198
100	515	190.8
200	512	181.44
500	512	183.6
1k	512	180
2k	520	178.56
5k	520	180
10k	504	176.4
20k	512	178.56
50k	520	172.8
100k	544	165.6
200k	560	135.36
500k	240	82.8
1M	114	61.2

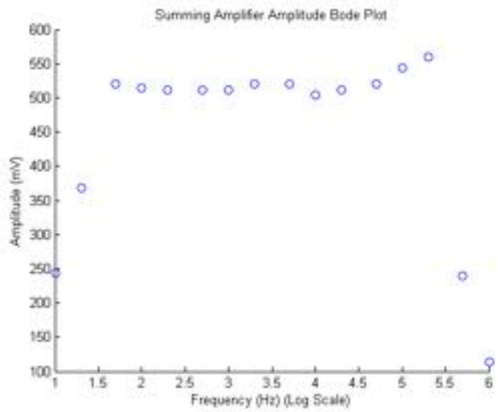


Figure 6: Summing Amplifier Amplitude

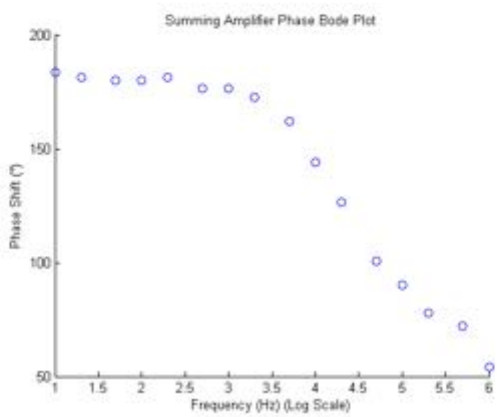


Figure 7: Summing Amplifier Phase Delay

Question 4:
The Bode plot shows this circuit to be a band pass filter. This is made obvious by the dip in output amplitude both at 10 Hz and 1 MHz.

Shown below is a screenshot of the input and output waveform of the summing amplifier

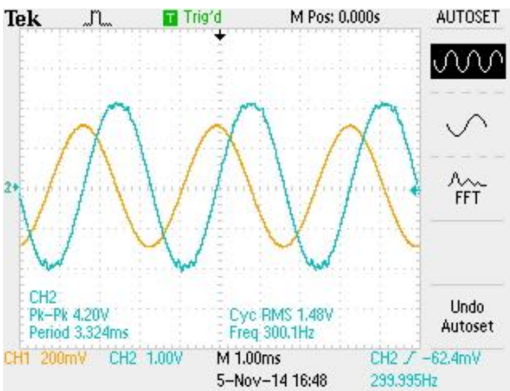


Figure 2: Equalizer signal output

Question 5:
From the picture of the input/output waveform, we confirm that this circuit is a differentiator. By observation, we see that as Vin reaches its peak, Vout would approach 0. When Vin's slope is steepest, Vout would reach its peak. Thus differentiating Vin would give us kVout.

Question 6:
Max of distorted waveform: 11V Min of distorted waveform: -10.2V This is the max of the op amp capabilities given the 12 V rail supply and the non ideal nature of the op amp. Possible reasoning for our min and max differing in magnitude is likely due to error in our DC power supply

Question 7:
The capacitor acts as a low pass filter on the signal, so when it is installed, low frequencies are attenuated

Question E:
The noise is due to the DC offset associated with the microphone power source. This offset shifts the signal in the vertical direction, and causes unnecessary noise. To counter the bias, a filter capacitor can be used to remove the DC offset from the signal and subsequently reduce the noise. Since the microphone can detect 50 to 5k Hz the filter can be designed to attenuate frequencies outside this noise.

IV Conclusion

In this lab, we have learned about the preamplifier circuit and the summing amplifier circuit of an equalizer, as well as the microphone.

We determined that the preamplifier circuit is an integrator after examining the input/output waveform. We also learned about the response and phase difference of the output voltages at different frequencies and how the potentiometer in this specific preamplifier circuit affects the output amplitude.

Finally, we concluded that the summing amplifier circuit is a differentiator and a band-pass filter according to the bode plots of its output response. We then find the maximum output capacity of the op-amp which can't be larger than its power supply. If the output is larger than the power supply, the output voltage will be clipped at the maximum capacity of the op-amp, causing a distortion. We also find that the capacitor in the summing amplifier acts as a low pass filter by listening to and observing the output sound of the circuit.

In addition we experimented with using the microphone as an input. Due to the DC offset associated with powering the mic, extra noise was introduced to our signal. To mediate this noise, a filter capacitor is used to attenuate frequencies no necessary to our amplifier.

Team Roles	
Activity	Student Name
Prelab/Circuit Analysis	Christyan and Jake
Prelab/Simulations	Taylor and Chinh
Prelab/Answer questions	Chinh and Christyan
Circuit construction	Jake and Taylor
Data Collection	Christyan and Chinh
Data Analysis	Taylor and Jake
Writing Lab report	Everyone

Lab 4: Filters
Lab Report

Taylor Fryett, Jake Garrison, Christyan Brown, Chinh Bui
EE 233 Circuit Theory
Department of Electrical Engineering, University of Washington, Seattle, 98195

***Abstract* – The goal of this lab is to gain experience with making filters and incorporating three filters into our final audio mixer circuit. The purpose of these filters are to give our mixer some control over the output frequencies.**

I Introduction

We will be constructing the 1 kHz band pass filter first, then move on the the 250 Hz and 4 kHz band pass filters. These filters will be incorporated into our audio mixer's output buffer - the only remaining component of the audio mixer that has yet to be built. We will be doing the final testing of all of the components from our other lab and then testing the way they work together.

II Lab Procedure

For the first part of the lab we built 3 band pass filters (BPF) that is centered near 250 Hz, 1 kHz, and 4 kHz according to the circuit in Figure 2 in lab manual. The components' values are selected according to the profile tables in Experimental Procedure and Analysis section. We then use the Spectrum Analyzer to measure the gain of each filter in the range of frequency from 10Hz to 5kHz.

For the second part of the lab, we combine the circuits from the previous labs with the 3 filters to build an audio mixer. The schematic of the mixer can be seen in Figure 11 of our report. We use the function generator to generate inputs with small amplitude at 250Hz, 1kHz, and 4kHz to test the mixer with filter. Lastly, we use the Spectrum Analyzer to measure the gain and frequency response of the mixer.

III Experimental Procedure and Analysis

Section 4.1

1 kHz BPF

1k Hz BPF Resistors	
Resistor Name	Resistance
R1 = R2	240k
R3 = R4	2.4k
R5	100k variable

1k Hz BPF Capacitors	
Capacitor Name	Capacitance (F)
C1	22 nF
C2	2.2 nF

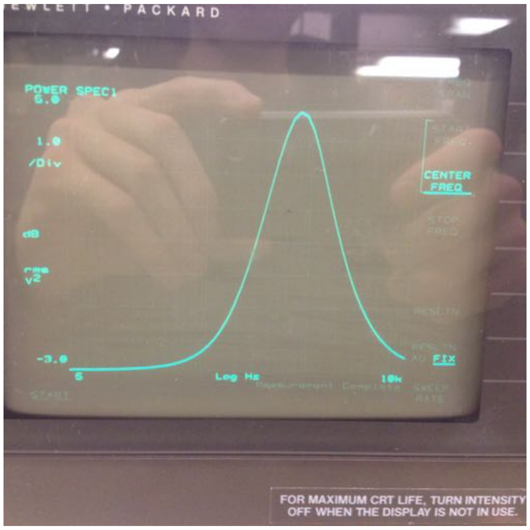


Figure 1: 1k Hz BPF

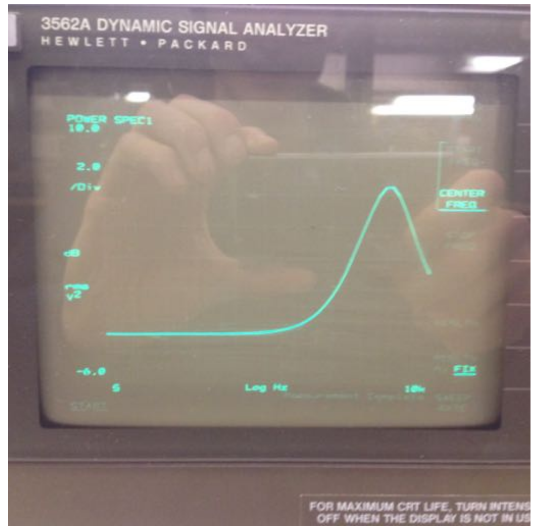


Figure 2: 4k Hz BPF

1k Hz Filter Profile			
Pot %	Center (Hz)	3 dB (Hz)	Gain
25	900	340 , 2.5k	2
50	900	1 , 1	0.97
75	950	350 , 2.5k	0.5

4k Hz Filter Profile			
Pot %	Center (Hz)	3 dB (Hz)	Gain
25	3.5k	1.2k , 10k	1.9
50	3.5k	1 , 1	1.1
75	3.6k	1.3k , 9.8k	0.45

4 kHz BPF

4k Hz BPF Resistors	
Resistor Name	Resistance
R1 = R2	240k
R3 = R3	2.4k
R5	100k variable

4k Hz BPF Capacitors	
Capacitor Name	Capacitance (F)
C1	5.6 nF
C2	0.56 nF

250 Hz BPF

250 Hz BPF Resistors	
Resistor Name	Resistance
R1 = R2	240k
R3 = R3	2.4k
R5	100k variable

250 Hz BPF Capacitors	
Capacitor Name	Capacitance (F)
C1	100 nF
C2	10 nF

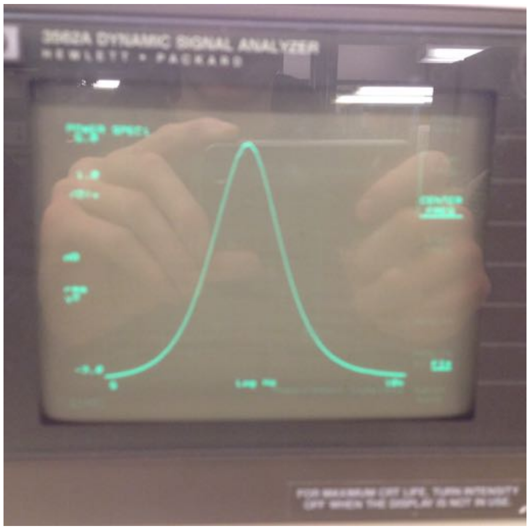


Figure 3: 250 Hz BPF

250 Hz Filter Profile			
Pot %	Center (Hz)	3 dB (Hz)	Gain
25	200	70 , 500	1.8
50	200	1 , 1	0.99
75	190	75 , 550	0.55

Section 4.2

First the signal was set to an amplitude of 100 mV and a frequency of 250 Hz. In Figures 4-9 CH2 is the input signal and CH1 is the output.

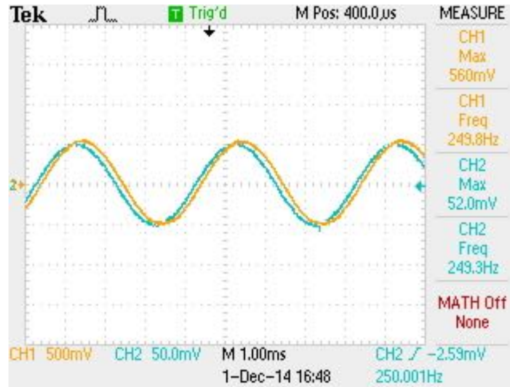


Figure 4: 250 Hz Unfiltered

After taking the initial picture,each of the filters was adjusted in turn. The 1 kHz and 4 kHz filters had marginal effect on the output signal. Figure 5 shows the input and output when the 250 Hz filter is set as a Band Reject Filter.

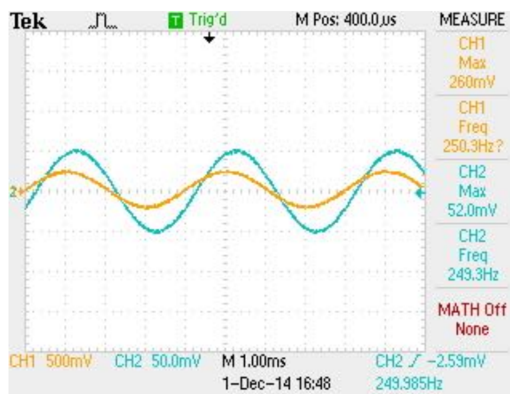


Figure 5: 250 Hz Filtered

For a signal with amplitude 100 mV and frequency 1 kHz the unfiltered input and output are shown in Figure 6.

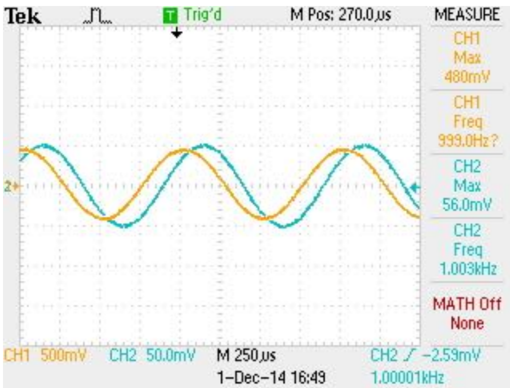


Figure 6: 1 kHz Unfiltered

The 250 Hz filter had a negligible effect on the output. While the 4 kHz filter had an effect, the effect was negligible in comparison with the effect of the 1 kHz filter. Figure 7 shows the input and output when the 1 kHz filter is set as a Band Reject Filter.

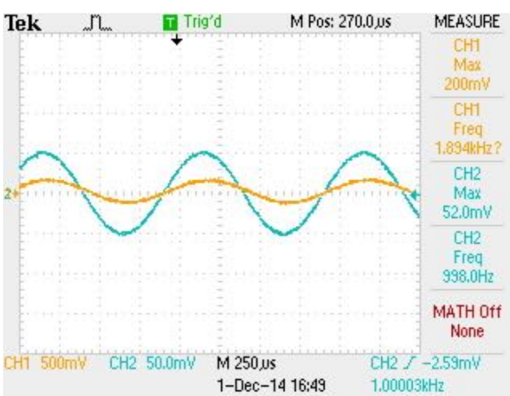


Figure 7: 1 kHz Filtered

For a signal with amplitude 100 mV and frequency 4 kHz the unfiltered input and output are shown in Figure 8.

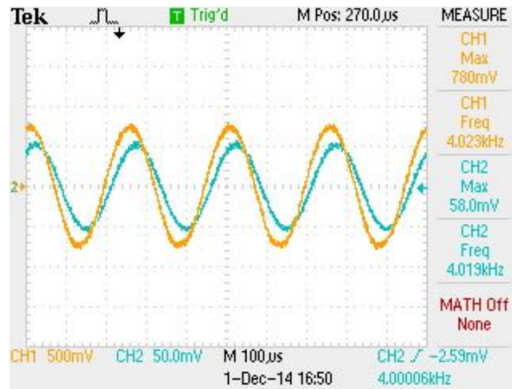


Figure 8: 4 kHz Unfiltered

The 250 Hz and 1 kHz filters had a neglible effect on the output. Figure 9 shows the input and output when the 4 kHz filter is set as a Band Reject Filter

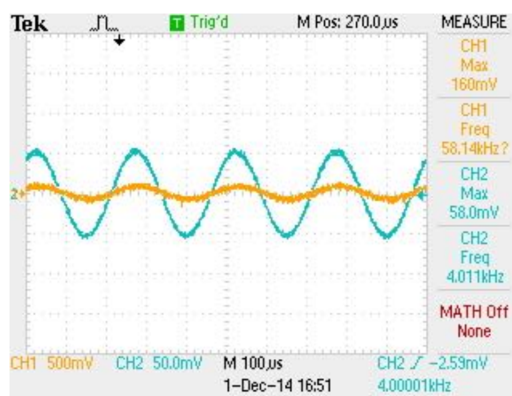


Figure 9: 4 kHz Filtered

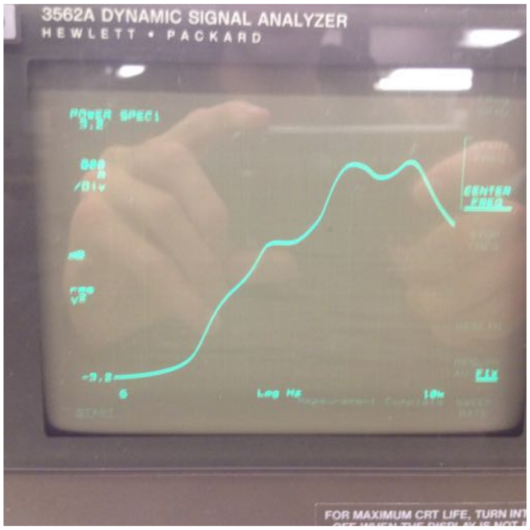


Figure 10: Mixer Gain

IV Conclusion

In this lab, we have learned how to built filters centered at different frequencies as well as how the filter affects the inputs that have the frequency near or equal the center frequency. We also observe how the filter behaves when we change the resistance value of the potentiometer. By changing the resistance value of the potentiometer, the filter can become a BPF or BRF for signals of frequency centered at 250Hz, 1kHz, and 4kHz. Given the broadband nature of these circuits we are given decent control over the general frequency response of our system.

The main part of this lab is combining the filters with the summing amplifiers (integrator and differentiator) to create an audio mixer. After confirming that the audio mixer works, we add the microphone as an input. The microphone is run through a pre-amplifier circuit to boost the signal and filter out high frequency noise. We also add a buffer (voltage follower) after the integrator summing amplifier to reduce distortion caused by the relatively high impedence of our microphone. Lastly, we observe the behavior of the filters in the mixer and confirm that they behave similar to how we have individually tested them. Upon driving the completed circuit through all three available inputs we are able to observe the control provided by the audio mixer and were able to combine three unique signals in a coherent and somewhat appealing manner.

Extra Credit

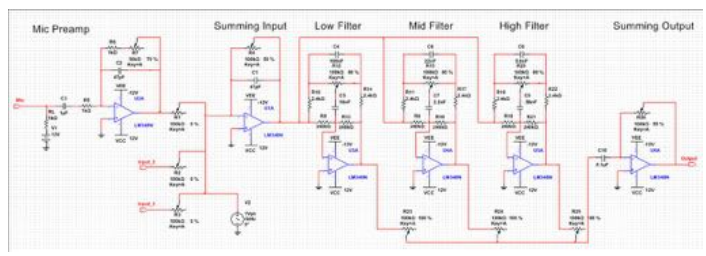


Figure 11: Multisim Complete Schematic

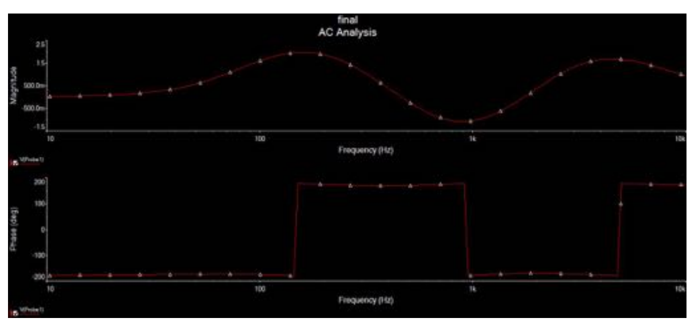


Figure 12: AC Analysis with BPF for low and high and BRF for mid frequencies

Team Roles	
Activity	Student Name
Prelab / Circuit Analysis	Taylor and Chinh
Prelab / Simulations	Christyan and Jake
Prelab / answer questions	Taylor and Jake
Circuit construction	Jake and Taylor
Data Collection	Chinh and Christyan
Data Analysis	Christyan and Chinh
Writing Lab report	Everyone