Lab 10

LTSpice Analysis of Active Filters

Student Name: Sean Balbale
Instructor: Dr. Iman Salama
Lab Partner Name: Krish Gupta
Date: November 15, 2024

1 Introduction

Operational amplifiers (op-amps) have served as essential components in electronic circuit design, particularly in sensing and signal processing applications. This lab focused on constructing active filters with op-amps, which are critical for biomedical applications such as electrocardiogram (EKG) signal measurement. These filters were designed to amplify small signals while selectively filtering out noise, thereby enhancing signal quality by rejecting common-mode interference and removing unwanted frequency components. Through LTSpice simulations, low-pass and high-pass filter designs were examined to analyze their frequency responses, cutoff frequencies, and time-domain performance. This approach provided insights into the role of active filters in real-world signal processing, forming a foundation for practical applications in biomedical and other electronic systems.

2 Results

2.1 LTSpice Modeling of Op Amp Circuits

2.1.1 Low Pass Filters

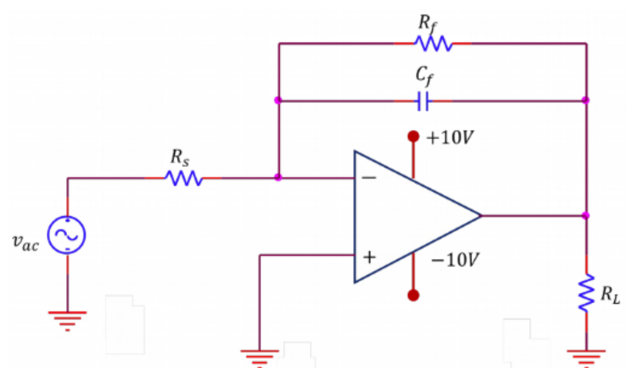

Figure 1: First Order Active Low Pass Filter

A first order low pass filter was constructed using an op-amp, resistors (R_f, R_s, R_L), and capacitor (C_f). Following the circuit design (Figure 1), the circuit was simulated in LTSpice with $\pm 10\,\mathrm{V}$ DC power supplies, $R_f = 100\,\mathrm{k\Omega}$, $R_s = 20\,\mathrm{k\Omega}$, $R_L = 1\,\mathrm{k\Omega}$, and $C_f = 10\,\mathrm{nF}$. The voltage input was a AC input with a 0.1 V amplitude and a frequency sweep from 1 Hz to 1 MHz.

The frequency cutoff can be calculated using the formula:

$$\begin{aligned} f_c &= \frac{1}{2\pi R_f C_f} \\ &= \frac{1}{2\pi \times 100 \times 10^3 \times 10 \times 10^{-9}} \\ &= 159.1\,\mathrm{Hz} \end{aligned}$$

This results in a cutoff frequency of 159.1 Hz.
This value is what is expected to be seen since there is a time constant of 1 ms. Which was calculated using the formula:

$$
\begin{aligned}
\tau &= R_f \times C_f \\
&= 100 \times 10^3 \times 10 \times 10^{-9} \\
&= 1\,\text{ms}
\end{aligned}
$$

These theoretical values are confirmed by the LTSpice simulation results shown in Figure 2.

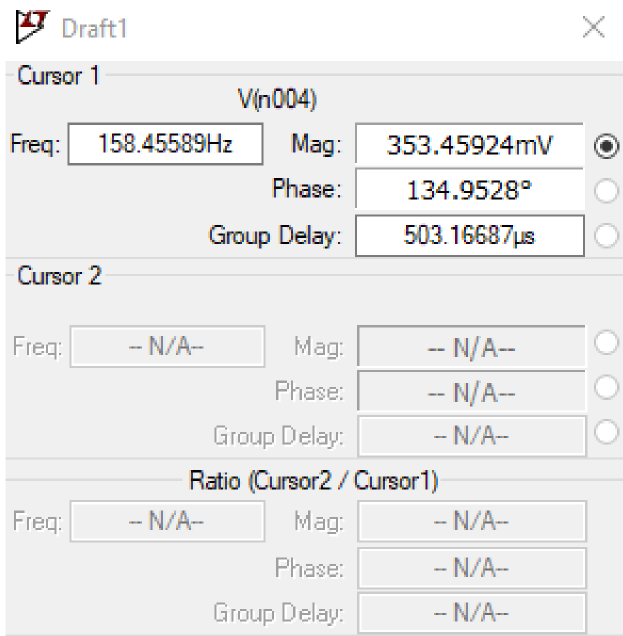

Figure 2: LTSpice Simulation Results

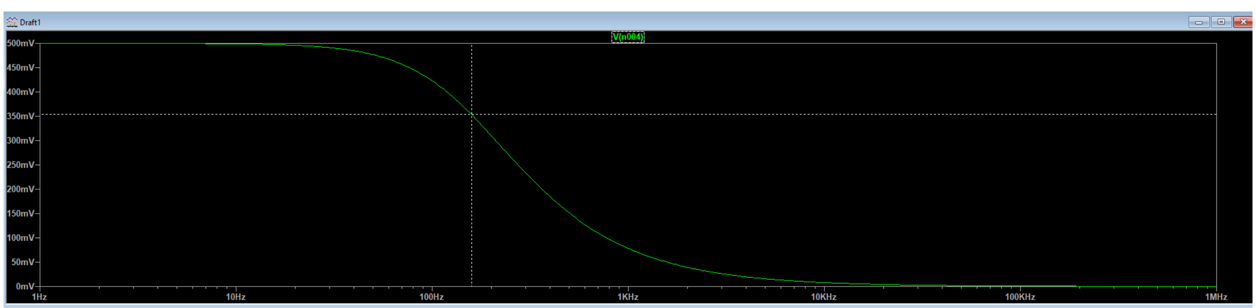

Figure 3: LTSpice Simulation

These results can also be calculated by hand from Figure 3. Next, the amplitude of the input sinusoid was increased to 0.5 V. The simulation is shown in Figure 5 and the results are shown in Figure 4.

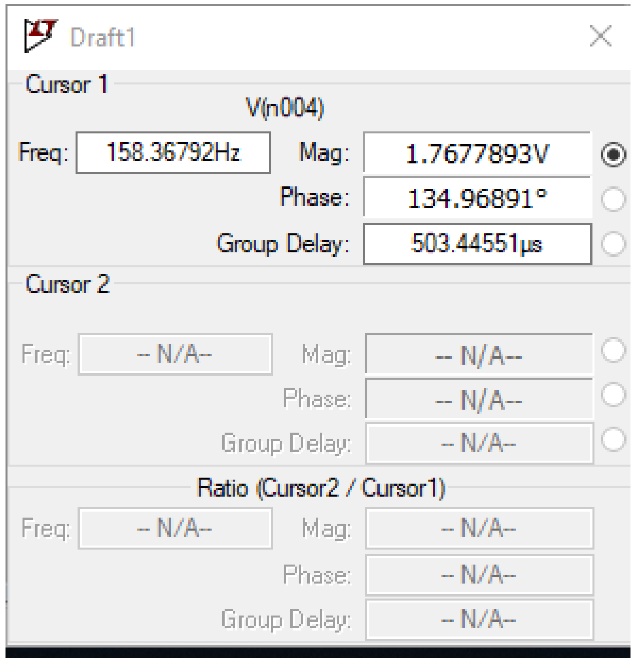

Figure 4: LTSpice Simulation Results

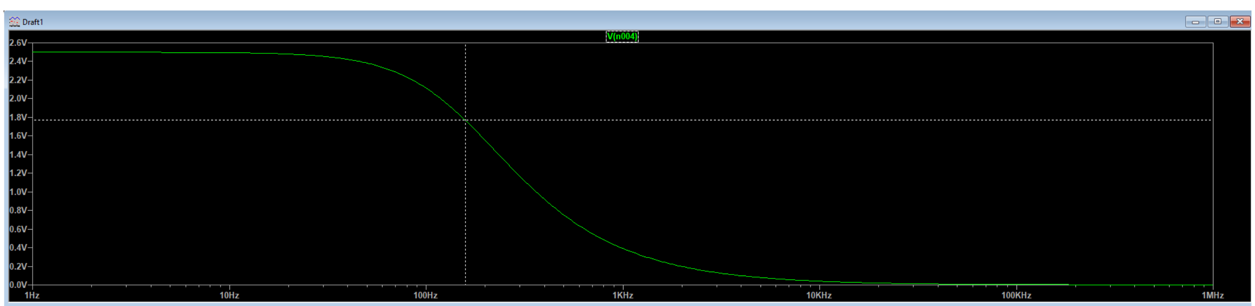

Figure 5: LTSpice Simulation

The frequency cuttoff and time constant are the same as the previous simulation. This should have been clear from the formulas for the cutoff frequency and time constant. The amplitude of the input sinusoid was increased to 0.5 V, but the cutoff frequency and time constant are not dependent on the amplitude.

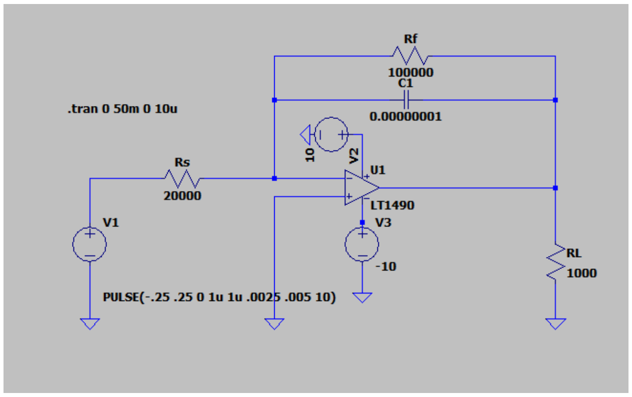

Figure 6: LTSpice Circuit

A new circuit was constructed for a first order high pass filter. The circuit is shown in Figure 6. The input and output voltages were probed and the simulation was run. The results are shown in Figure 7.

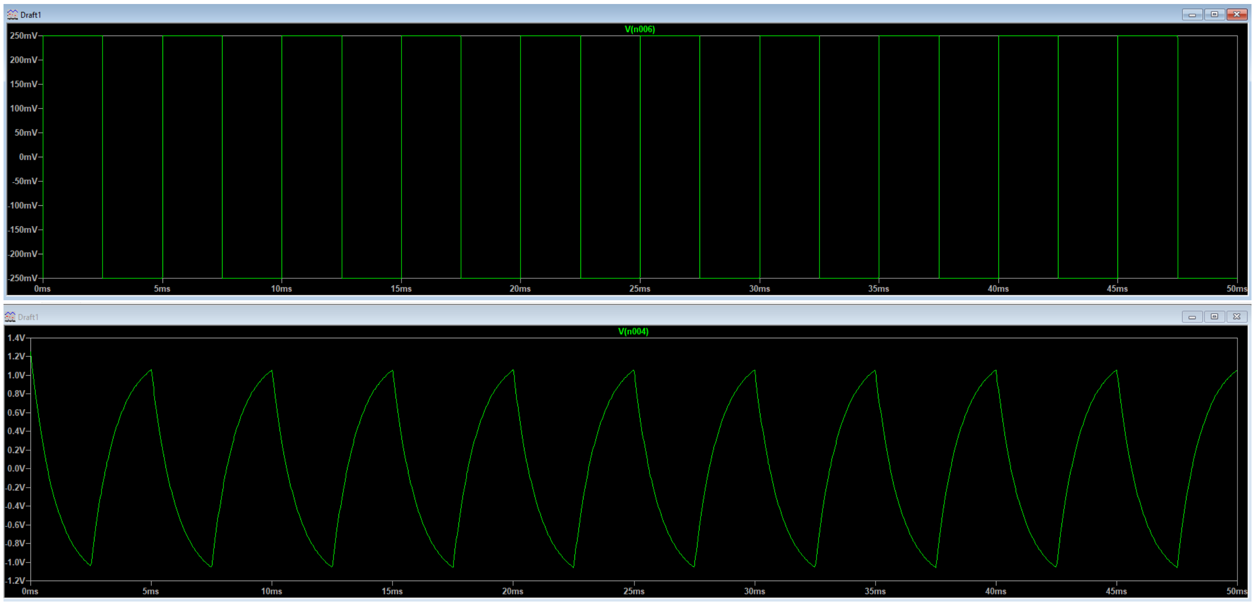

Figure 7: LTSpice Input (Top) and Output (Bottom) Simulation Results

The input is a regular square wave. The output is also a square wave but the capacitor is charging and discharging. This results in more of a point at the top of a logarhythmic curve.

2.1.2 High Pass Filters

Now a first order high pass filter was constructed using an op-amp, resistors (R_f, R_s, R_L), and capacitor (C_f). Following the circuit design (Figure 8), the circuit was simulated in LTSpice (Figure 9).

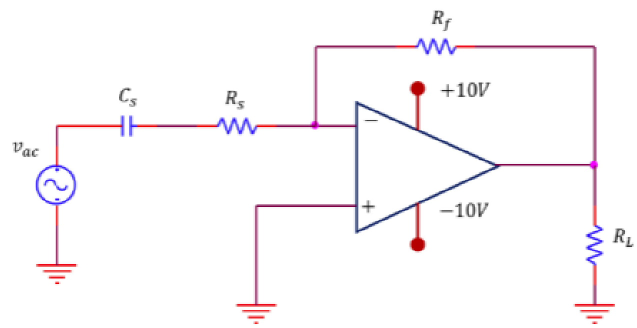

Figure 8: First Order High Pass Filter Circuit

The circuit was simulated with $\pm 10\,\mathrm{V}$ DC power supplies, $R_f = 200\,\mathrm{k\Omega}$, $R_s = 100\,\mathrm{k\Omega}$, $C_s = 10\,\mathrm{nF}$. The voltage input was a AC input with a $0.1\,\mathrm{V}$ amplitude and a frequency sweep from $1\,\mathrm{Hz}$ to $1\,\mathrm{MHz}$.

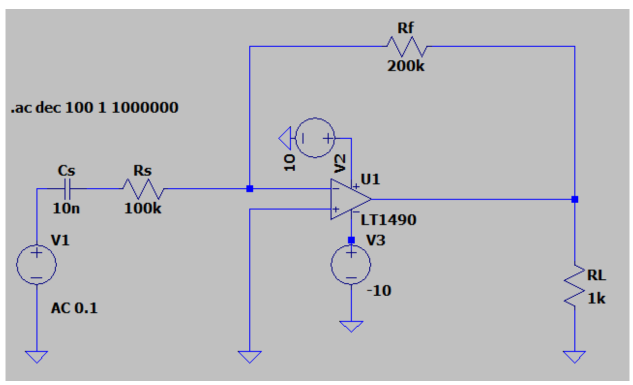

Figure 9: LTSpice Circuit

The frequency cutoff can be calculated using the formula:

$$
\begin{aligned}
f_c &= \frac{1}{2\pi R_s C_f} \\
&= \frac{1}{2\pi \times 100 \times 10^3 \times 10 \times 10^{-9}} \\
&= 159.1\,\mathrm{Hz}
\end{aligned}
$$

The in-band gain can be calculated using the formula:

$$
\begin{aligned}
Gain &= -\frac{R_f}{R_s} \\
&= -\frac{200 \times 10^3}{100 \times 10^3} \\
&= -2
\end{aligned}
$$

The simulation is shown in Figure 11 and the results are shown in Figure 10. These results support the theoretical calculations for the cutoff frequency.

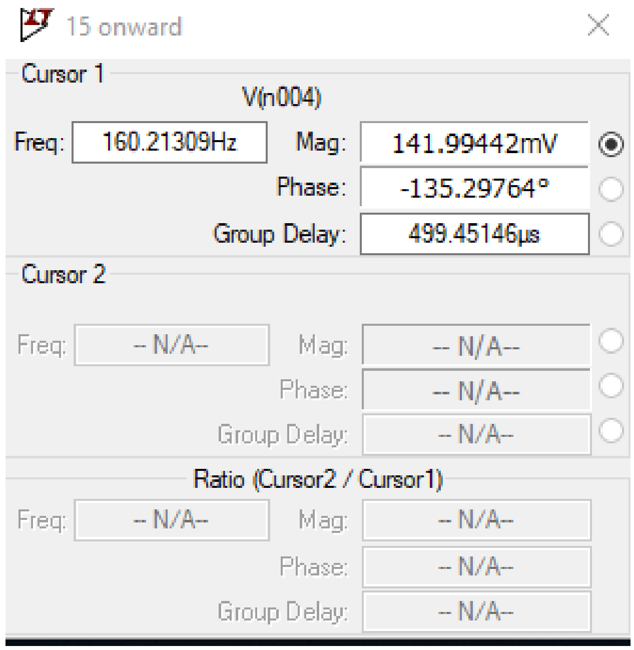

Figure 10: LTSpice Simulation Results

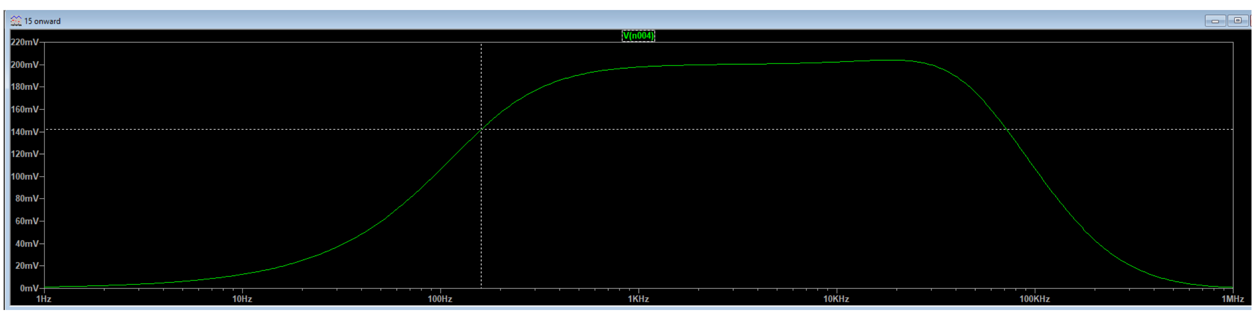

Figure 11: LTSpice Simulation

In the simulation, the output rolls off at higher frequencies. This behavior can be seen in Figure 11. This occurs because the input frequency exceeds the maximum op-amp frequency. This results in the avalanche effect seen in the graph.

3 Discussion and Conclusion

This lab successfully demonstrated the design and analysis of active low-pass and high-pass filters using LTSpice. The simulations confirmed theoretical predictions for cutoff frequencies and gain, and the filters performed as expected in both frequency and time domains. The low-pass filter effectively attenuated high frequencies, with a confirmed cutoff frequency of 159.1 Hz, while the high-pass filter attenuated low frequencies with a matching cutoff. Additionally, the observed roll-off at high frequencies in the high-pass filter simulation illustrated the op-amp's bandwidth limitations, a crucial consideration in real-world filter applications.

Through these exercises, this lab highlighted the critical role of component selection in filter design, the effect of op-amp bandwidth on filter performance, and the practical applications of active filters in biomedical signal processing. Overall, the experiment reinforced the value of LTSpice as a simulation tool and provided foundational insights for implementing filters in various signal processing applications.

4 References

[1] Dr. Iman Salama. "Lab 10 – LTSpice Analysis of Active Filters" Northeastern University. 11 November 2024.