EECE 2510 – Circuits and Signals: Biomedical Applications

Lab 10, LTSpice Analysis of Active Filters

INTRODUCTION:

As discussed in class, Op-amps are useful building blocks in many sensing and measurement applications. To measure the EKG signal, we will be using them to amplify small signals, to reject common-mode signals, and to filter out unwanted low and high-frequency noise and interference.

PART 1: LTSPICE MODELING OF OP AMP CIRCUITS

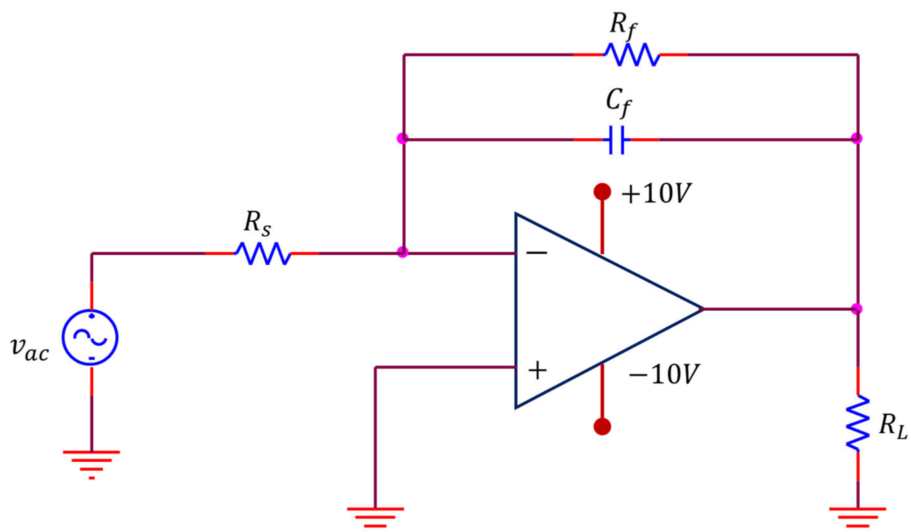

Figure 1: First order active low pass filter for part 1

1.1 In LTSpice, use the **LT1490 op-amp** to build an active **low pass** filter shown in figure 1. Use $\boldsymbol{R_f = 100\ k\Omega, C_f = 10\ nF, and\ R_s = 20\ k\Omega}$. Use $+/-\ \boldsymbol{10V\ DC}$ power supplies for the op-amp and use an **AC source** to the input of the filter. The load resistance $R_L = 1k\Omega$

Some tips on starting/using PSpice:

1. Run LTSpice
2. New schematic
3. Save as, specify name and location – somewhere you have permission to write (desktop, for example).
4. OK

5. Special note: m is 10^{-3}, M is also 10^{-3}, Meg is 10^{6}, meg is also 10^{6} or 1e6 is 10^{6}.
6. Add a standard voltage source, right click to edit, click advanced and enter the AC amplitude of the source. There is no need to change any other parameters.

1.2 Test the amplifier with an **AC input** with $0.1V$ amplitude and create a semilog plot of the response from 1 Hz to 1 MHz using the AC Sweep mode with a logarithmic frequency sweep. Set up the simulation as follows (using the simulate button and "edit simulation command" on Windows or by going to draft>spice directive on Mac). Set up the simulation for an ac analysis, by decade, 100 points/decade, from 1 Hz to 1,000,000Hz. The command shown on the schematic should be

.ac dec 100 1 1000000

(hint – use a voltage probe at the output of the op-amp to display the output voltage). You will need to right click on the left vertical axis to change to linear scale. Right click on the right vertical axis to remove the phase plot. Note that this is the transfer function, $H(\omega)$, except that it is multiplied by 0.1 because the input is 0.1V. **Q1: What is the cutoff frequency, f_c, (the half power point, or where the voltage is 0.707 times the maximum output voltage)? Is this what you expect, considering the value of the RC time constant?**

1.3 **Q2: What happens if the amplitude of the input sinusoid is changed to $0.5V$?**

1.4 Time domain view of the filter response: Now a use a pulsed source to produce a **200 Hz square wave** with $0.5V$ p-p and connect to the filter input. Plot both the input and output voltages. Use transient analysis in this case, choose the interval of the simulation to be about 10 cycles of the square waveform. A reasonable choice for the properties of the Pulse source is as follows

```
PULSE(-.25 0.25 0 1u 1u 2.5ms 5ms 10)
```

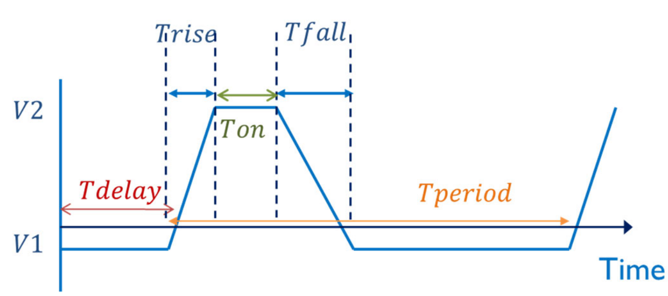

Figure 2, periodic pulse wave form in LTSpice

$V1$ (Vinitial)	*the voltage level at the start of the pulse*
$V2$ (Von)	*the voltage level at the peak of the pulse*
$Tdelay$	*the time delay before the start of the pulse*
$Trise$	*the pulse rise time*
$Tfall$	*the pulse fall time*
Ton	*the pulse width at peak voltage (V2)*
$Tperiod$	*the pulse period*
$Ncycles$	*Number of cycles (periods) to display*

V1 and V2 should be $-0.25V$ and $+0.25$ or $0V$ and 0.5V for a $0.5V$ p-p square wave. $Tdelay$ can be set to zero. $Trise$ and $Tfall$ should be significantly smaller than the period, perhaps 1-10 microseconds. If they are too small, LTSpice could take a long time to simulate, or it might not find a solution. $Tperiod$ is the total period of the pulse which should be double the value of Ton for a square wave that has equal times spent in the high and low voltage states. As before, use the simulation command on Windows or the spice directive on Mac to set up the simulation to transient analysis.

Q3: Use probes to plot both the input and the output voltages. Examine the output signal, save/print the output or sketch it in your lab-book and try to explain why the output wave looks as it does, rather than just being a square wave. Try thinking in both the time and frequency domains when coming up with your answer. It may be helpful to use the FFT function to display the frequency components for both the input and output waves (you may have to do this in two windows). Change both the vertical and horizontal axis to linear scale and limit the horizontal axis if you like – displaying the FFT up to 5 kHz is about right.

Hint – it may help you see what is happening if you try using square waves with higher and lower frequencies to see what the filter does (10x to 100x higher or lower, maybe).

1.5 Now, change your circuit so that it is an active high pass filter shown in Figure 3 with $R_S = 100\ k\Omega$, $C_S = 10\ nF$, $R_f = 200\ k\Omega$ and plot the filter response to an AC input from 1 Hz to 1 MHz. **Q4: What are the in-band gain and f_c of the circuit? Q5: Why does the output roll off at high frequencies, if this is a high-pass filter??**

1.6 <u>If you have time:</u> Now use a pulsed source to produce a **200 Hz square wave** with $\mathbf{0.5V}$ amplitude and connect this to the filter input. Plot both the input and output voltages. **Q6: Examine the output signal, save the output plot or sketch**

it in your lab-book and try to explain why the output wave looks as it does, again thinking in both the time and frequency domains.
Hint – again it may help you see what is happening if you try inputting square waves with higher and lower frequencies (10x to 100x higher or lower, maybe).

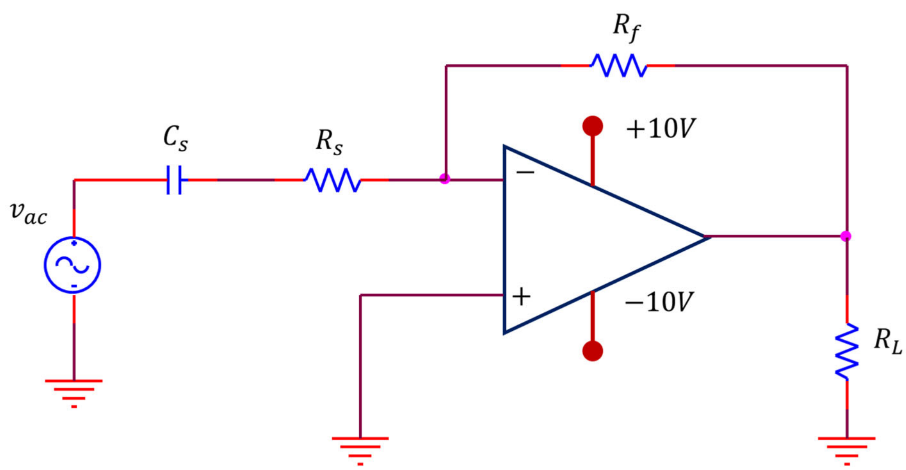

Figure 3: First order active high pass filter for part 1

INSTRUCTIONS FOR THE WRITE-UP…

Please follow instructions for writing lab reports available on Canvas

IMPORTANT: BEFORE YOU LEAVE THE LAB:

- **a.** Place all of the components that your removed from the red tool box back in that box and return it to the cabinet that houses them
- **b.** Collect all used components and wires from your bench and place them in your group's reusable plastic container. If you are not going to use these components or wires again please discard them in the trash bin.
- **c.** Turn off all of the equipment you have used on your workbench.
- **d.** Make sure you return your protoboard, the equipment wires and your reusable container to the front window.
- **e.** Make sure to have your notebook signed by an instructor before you leave the lab.

Department of Electrical Engineering, Northeastern University.

Last updated: 8/23/2021/ I. Salama;2/28/16, N. McGruer;