Lab 13 – Notes

Spring 2024

JOHN GERGUIS

General Notes

$$\omega = \boldsymbol{2\pi} * f$$

Task 1

Task 13.4.1: Design of an active filter

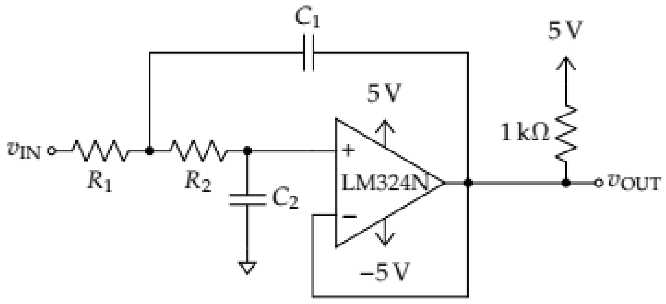


Figure 13.10: Low Pass Filter

1. *Compute* the values of R_1 and R_2 necessary to build a second order Sallen-Key low pass filter with a cutoff frequency of 20 kHz and Q of $\frac{1}{\sqrt{2}}$. Use $C_1 = 10\,\text{nF}$ and $C_2 = 1\,\text{nF}$.
2. **EC:** *Simulate* the response of the filter using AC SPICE analysis. Only use resistor values you can make using resistors in your kit. Calculate the error between the simulated cutoff frequency and the ideal cutoff frequency.
3. *Construct* the circuit using your designed values.[a]
4. *Measure* the gain and phase response of the filter from 100 Hz to 50 kHz.
5. *Capture* an oscilloscope screenshot showing v_{IN} and v_{OUT} at the cutoff frequency.
6. *Calculate* the error between the ideal cutoff frequency and the cutoff frequency of the constructed circuit.

[a] The 1 kΩ resistor is necessary due to limitations of the output stage of the LM324N [2], but is not necessary for all opamp circuits.

** Make sure to add the 1kΩ resistor.

1. Check your values of R1 & R2 with a TA before building the circuit.
2. This is an extra credit step, simulate the circuit on LT spice.

4. Use the Network tool to plot the frequency response. You can measure the cut-off frequency the same way that you measured the BW of the amplifiers before (Go to the point which has the max gain in dB – 3, then measure the corresponding frequency and phase).

5. You are expecting the output at cut-off should have $v_{out_{PP}} = \frac{1}{\sqrt{2}} v_{in_{PP}}$

6. Calculate the error between the measurements and the designed value.

Task 13.4.2: Single supply active filters

Task 2

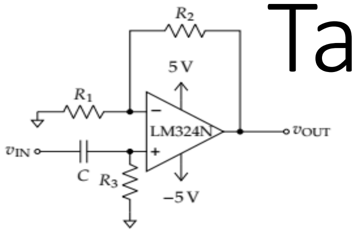


Figure 13.11: High Pass Filter

1. *Calculate* the cutoff frequency and gain of the filter in figure 13.11 when $R_1 = 1\,\text{k}\Omega$, $R_2 = 33\,\text{k}\Omega$, $R_3 = 330\,\text{k}\Omega$, and $C = 1\,\mu\text{F}$.
2. *Construct* the filter in figure 13.11.
3. *Measure* the frequency response of the filter for $0.2\,\text{Hz} < f < 20\,\text{Hz}$. Use a $0.1\,V_{p-p}$ input. You will need to do this manually on the ADALM, as the network tool does not work below 1 Hz.
4. **EC:** *Replace* the −5 V supply with ground. *Apply* a $0.1\,V_{p-p}$ 2 Hz sine wave to v_{IN} and *capture* an oscilloscope plot of v_{IN} and v_{OUT}. Does the filter still work properly?
5. **EC:** In order to bias the opamp correctly, we need to set the DC value of the inputs to a value inside the common mode input range. *Construct* the revised circuit in figure 13.12. Set the potentiometer so that V_{ref} is in the center of the common mode input range of the op amp.
6. **EC:** *Apply* a $0.1\,V_{p-p}$ 5 Hz signal to the input and record the gain. Why is the gain different than the original high pass filter circuit?

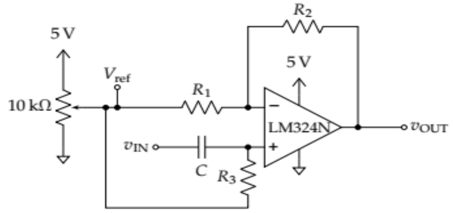


Figure 13.12: Single Supply High Pass Filter

7. **EC:** *Construct* the additional buffer circuit for V_{ref} shown in figure 13.13.

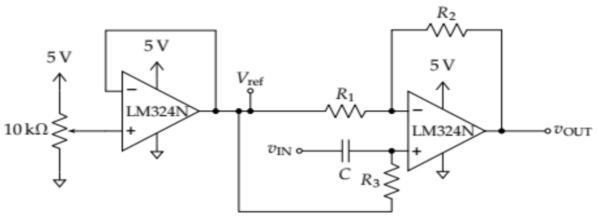


Figure 13.13: Corrected Single Supply High Pass Filter

8. **EC:** *Adjust* the potentiometer so that V_{ref} is still at the desired bias voltage.
9. **EC:** *Measure* the gain of the circuit with the buffer added at 5 Hz. You will need to reduce the amplitude of the input until the output is no longer clipping. *Capture* an oscilloscope screenshot showing v_{IN} and v_{OUT}.

** $f_c = \frac{1}{2\pi R_3 C}$.

3. The frequency response on the Network tool will take a long time, so before running it, check first that your circuit is correct. You can do 2 checks:

   a. Use the wavegen and the scope, put a signal at freq >> fc and check that the gain $(\frac{v_{out}}{v_{in}})$ is as expected. Please note that the gain is so high, so make sure to put a small voltage at the input.

   b. Now, repeat the same things but put the frequency to be the fc, and make sure that the gain has dropped by a factor of $\frac{1}{\sqrt{2}}$.

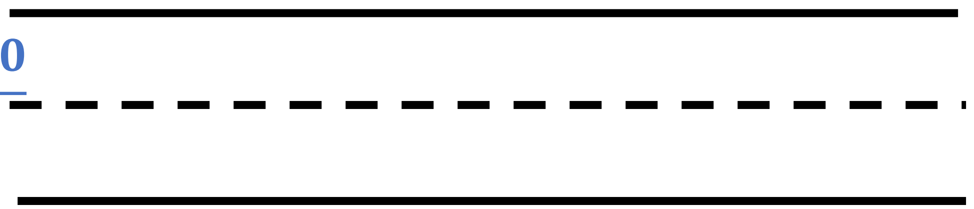
4-9) Those are extra credit steps which build toward using one supply (+5v & gnd) instead of using dual supply (+/- 5v). There is an extra credit part in the project which builds on this.

4) The circuit will not be able to output –ve voltages, i.e. it clips the –ve part.

5/6) Add offset to the signal to put it within the common mode of the circuit. This will solve the clipping issue, but the gain will decrease, why?

$v_{max} = 5 - 1.5$

$\mathbf{v_{ref}} = \frac{3.5+0}{2}$



$v_{min} = 0$

7-9) Adding the buffer should fix both issues, now the gain should be close to the expected gain and output has an offset DC voltage (what you have added).

Task 3

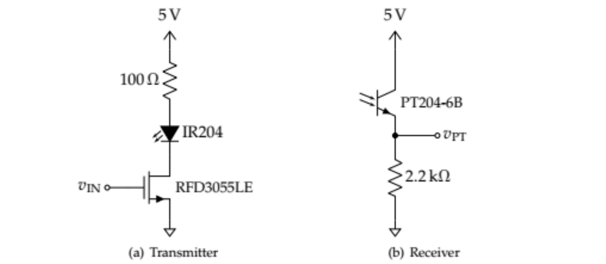


Figure 13.14: Infrared communication building blocks

1. *Construct* the infrared transmitter and receiver circuits in figure 13.14 on opposite sides of your breadboard. Use an infrared LED (IR204) [3] as the transmitter and an infrared phototransistor (PT204-6B) [4] as the receiver.[a]
2. *Apply* a 0 V to 5 V 10 kHz square wave to v_{IN}.
3. *Aim* the transmitter LED at the receiver phototransistor.
4. *Capture* an oscilloscope printout showing v_{IN} and v_{PT}.

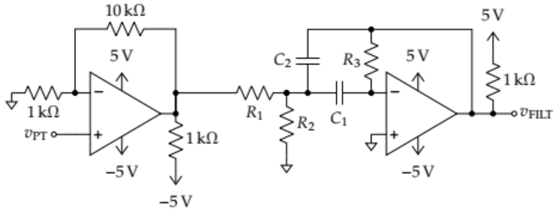


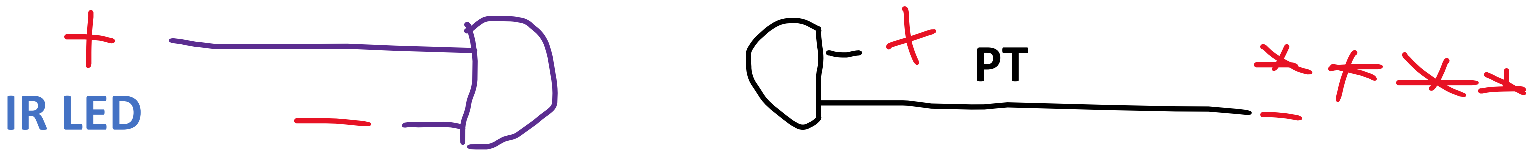
Figure 13.15: Receiver amplifier and filter

5. *Calculate* the values of R_1, R_2, and R_3 necessary in the multiple feedback band pass amplifier to have $k = -5$, $Q = 10$, and $f_0 = 10$ kHz given $C_1 = C_2 = 10$ nF.
6. *Construct* the circuit in figure 13.15.

8. *Measure* the center frequency, gain at the center frequency, bandwidth, and Q of the filter using the magnitude response of the circuit.
9. *Connect* the output of the phototransistor circuit in section 13.4 to the input of the filter circuit in figure 13.15.
10. Once again, *Apply* a 0 V to 5 V square wave to v_{IN} while aiming the LED at the phototransistor. Set the square wave frequency to center frequency you measured in step 8.
11. *Capture* an oscilloscope screenshot of v_{IN} and v_{FILT}.
12. *Record* the peak to peak voltage of v_{FILT} while sweeping the square wave frequency of v_{IN} from 1 kHz to 20 kHz. Record at least one data point per 500 Hz.

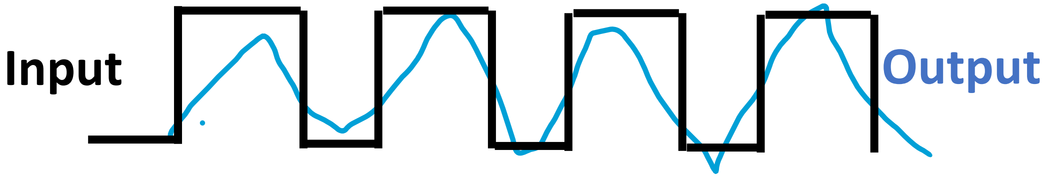
[a] A phototransistor is a device that changes the amount of current flowing through it based on the amount of received light.

** The IR is the blue one and the PT is the black one. PLEASE double check the polarities.



1. Put the IR204 and PT204 to be pointing towards each other (this is different from lab 12) and to have a reasonable distance between them (e.g. on opposite sides of the breadboard).

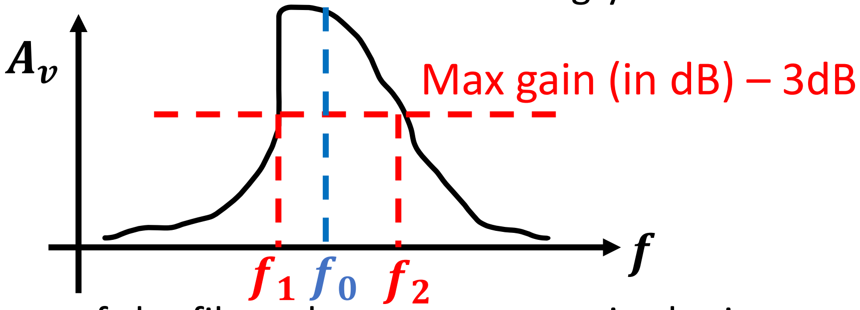
4. Measure the input and the output.



5-8. Check the values of R1, R2 and R3 with a TA before building your circuit. Build the BPF and characterize it.

$$BW = f_2 - f_1$$

$$Q = \frac{f_0}{BW}$$



9-11. Connect the v_{PT} to the input of the filter, then measure again the input and the output of the full system. You should be able to find that the output became a little bit smoother.

12. You need to measure the Vp-p output while sweeping the frequency. You need to do this manually, you CANNOT use the Network tool (or FRA) because you need the input to be a square wave. Take a measurement every 0.5kHz between 1-20 kHz.

You may find more peak(s) around 3.3 or/and 5 kHz