STETHOSCOPE LAB REPORT

Marta Masramon Muñoz (CID:01579866)

Abdullah Rehman (CID:01569288)

Introduction

The aim of this report is to outline the process of building a functioning electronic stethoscope that amplifies heartbeat sounds while ignoring other sources of noise. We aimed to pick up the 2 main sounds made by the heart caused by the atrioventricular and semilunar valves closing. To do so, we designed the circuit using PSpice (OrCAD) and modelled its performance before moving on to the physical construction of the stethoscope. Finally, we tested the circuit by connecting a microphone and headphones to listen to an actual heartbeat signal. When building the electrical circuit we used LM324 Op-Amps, a variety of resistors and capacitors, an LED, a microphone and headphones, and a breadboard to place and test the components on.

1. Amplifiers

To get a clear output signal from the heartbeat sound we first had to amplify the input. From an input signal of around 1-5mV we aimed to get an output signal of around 2-3V (as the requirements stated [1]), which makes the heartbeat sound easily discernable. Since the maximum gain for an op-amp is 200, we needed two op-amps to have enough amplification.

We chose to use inverting amplifiers, opposed to non-inverting, for various reasons. Firstly, they have lower internal impedance to the signal. They also use a fewer number of components, making them more cost effective and efficient. The fact that they invert the signal is made irrelevant because we use two of them in series.

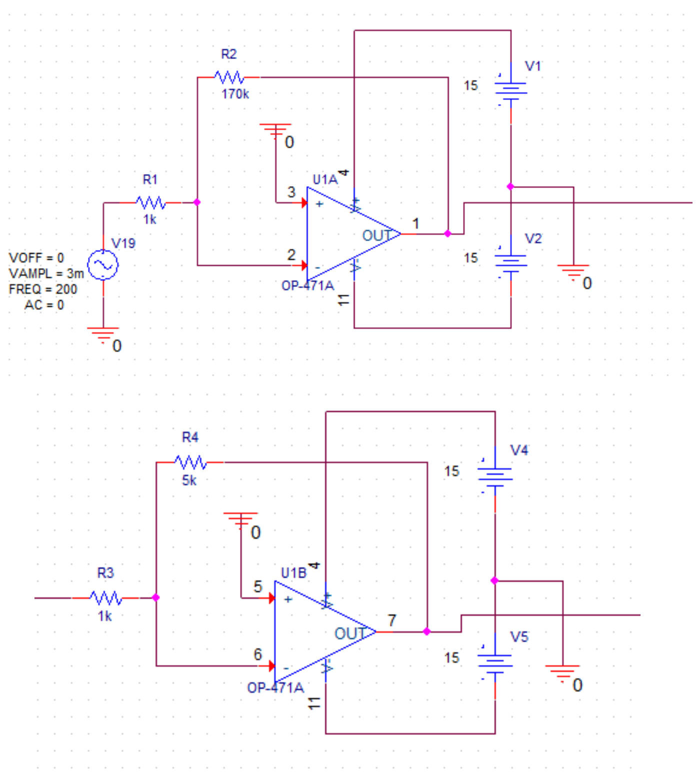


Figure 1. Design of amplifying Op-Amps in OrCAD

Knowing that the gain of inverting amplifiers is given by the following: $gain = \frac{V_{out}}{V_{in}} = \frac{R_2}{R_1}$, we decided to amplify once by 170 and then by 5. Therefore, we used a 170$\text{k}\Omega$ and 1$\text{k}\Omega$ resistors in the first op-amp and a 5$\text{k}\Omega$ and 1$\text{k}\Omega$ resistors in the second. Using these values, $total\ gain = \frac{R_2}{R_1} \cdot \frac{R_2'}{R_1'} = \frac{170k}{1k} \cdot \frac{5k}{1k} = 850$. Figure1 shows the design of the amplifiers.

We tested the design with an input voltage of 3mv and aimed to get an output of about 2.5V after both op-amps. We confirmed the design worked by running an AC analysis over a large range of frequencies (see Figure2).

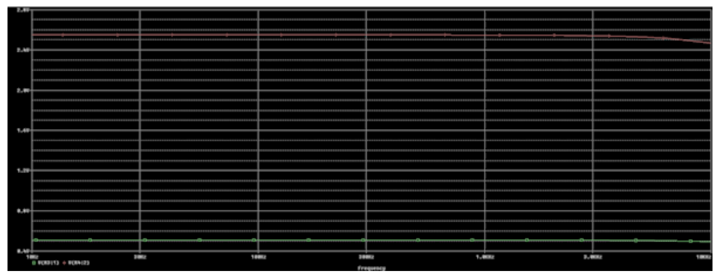


Figure 2. Simulated output of both amplifications

Due to unavailability of certain resistors when building the amplifiers, we had to use a different combination of resistors than in the design (making sure that they still gave a similar gain). We also realized that we needed to consider the internal impedance of the circuit and might need to calculate resistor values that predict a larger gain so that the final amplification is in the range desired. With this in mind:

$$total\ gain \leq \frac{R_2}{R_1} \cdot \frac{R_2'}{R_1'} = \frac{20\text{k}}{111} \cdot \frac{2.4\text{k}}{470} = 918$$

We tested the performance of the circuit using an oscilloscope and measured a gain of 158 after the first amplifier and a total gain of 852 after the second one, very close to the predicted value. We can see this in Figure3.

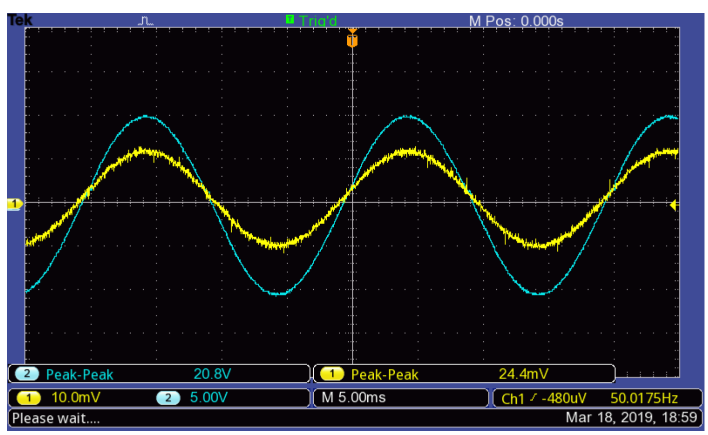


Figure3. Analysis of circuit made with an oscilloscope after the amplifiers

2. Filters

To make sure that we only amplified sounds that corresponded to the heartbeat, we had to filter out noise. We know that frequencies from a heartbeat range from 30 to 70Hz [2]. Therefore, the high-pass filter must eliminate values below 30Hz, corresponding to noises that typically come from movements of the subject or the device.

We chose an arbitrary value of 100nF for the capacitors and calculated the corresponding values of the resistors:

$$C_1 = C_2 = 100nF; R_1 = R_2 = \frac{1}{2\pi \cdot f_c \cdot C_1} = \frac{1}{2\pi \cdot 30 \cdot 10^{-7}}$$
$$= 53051\Omega \cong 53k\Omega; R_3 = \frac{R_1}{2} = 27k\Omega$$

Since most background noise occurs at around 500Hz, we eliminated it by setting the cut-off frequency of the low pass filter to 400Hz. Again, we chose an arbitrary value of 100nF for the capacitors:

$$C_1 = C_2 = 100nF; R_1 = R_2 = \frac{1}{2\pi \cdot f_c \cdot C_1} = \frac{1}{2\pi \cdot 400 \cdot 10^{-7}}$$
$$= 3978\Omega \cong 4k\Omega; R_3 = 2R_1 = 8k\Omega$$

Figure 4 shows the design of both filters. Running various simulations (see Figure5), we saw that the output voltage (yellow) is only above half of the amplified signal (red) between 30Hz and 400Hz, showing that the filters work as expected. What is also noticeable is that the maximum output voltage is still less than the amplified signal which shows that filtration process has some inherent impedance.

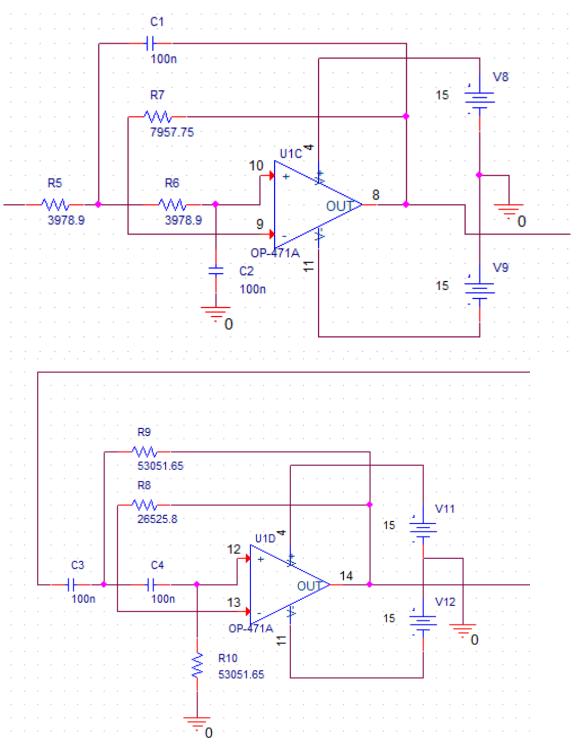


Figure4. Design of filters made using OrCAD. Low-pass (top) and High-pass (bottom).

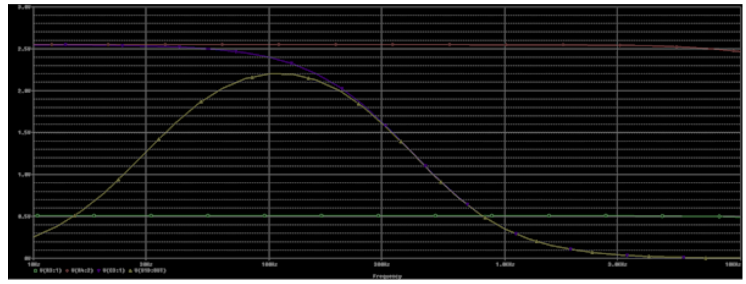


Figure5. Circuit performance: expected output after filters (made using OrCAD)

When building the filters, we used a combination of resistors in series to obtain the required values of resistance.

The first test we ran on the circuit was not consistent with the expected output, since the first filter didn't seem to affect the input in any way. Using a multimeter, we found that this was because of incorrect positioning of a capacitor: having both pins in the same column, it created a short circuit (see Figure6). A second test revealed a constant output that did not depend on the input signal. This was due to another mistake: connecting one of the pins of a capacitor into the wrong op-amp terminal.

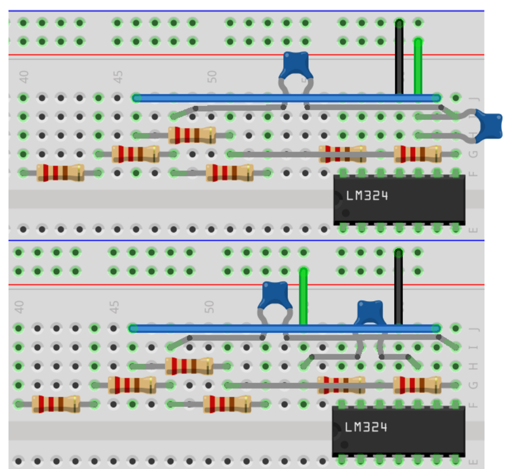


Figure6. Circuit schematics of high pass filter: wrong (top) and correct (bottom). Made using Fritzing.

After dealing with these issues, we got the same output we had predicted with the simulation. This is illustrated in Figure7. We can see that there is less noise in the signal of the output than in the input, which indicates that the filters worked correctly. Nevertheless, there is a significant drop in voltage. We attributed this to the internal resistance found the circuit which was higher in practice rather than in the design. An additional possibility is also that the physical build consisted of more components than the design which could accumulate to further impedance.

We also have to note that there was some noise coming from the mains (50Hz). Since it was inside our range of frequencies, it was also amplified, but we neglected it by assuming that it had a voltage that is several orders of magnitude lower than the heartbeat and therefore does not significantly affect our results.

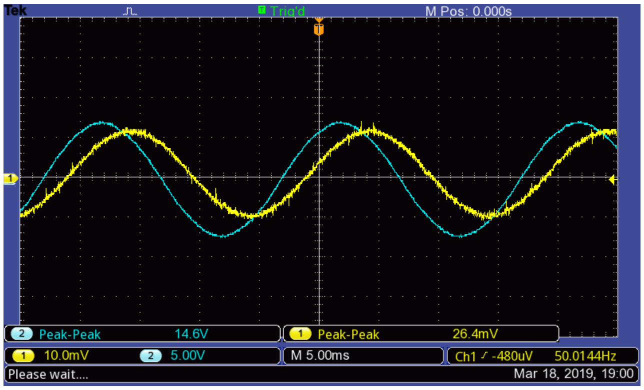


Figure7. Analysis of circuit made with an oscilloscope after the filters

3. Additional components

We included a Schmitt Trigger in the circuit so that we could convert the sinusoidal output into a clean square wave. This effectively converts the output from analogue to digital which has many advantages. By setting the threshold voltage to nearly 0, we can make it so that when the output signal is just positive (the stethoscope picks up a signal) the Schmitt trigger gives a high value and otherwise a low one. We used the following equation to get our values:

$$V_{threshold} = \frac{R_2}{R_1 + R_2} \times V_{saturation}$$

Where $V_{saturation}$ is the saturation of the Op-Amp, typically 13-15V [3].

The high gain required in the circuit meant that a small DC offset in the input could be amplified and saturate the signal, so we added a Summing Inverting Amplifier to eliminate the offset and make sure this did not happen. The datasheet of the LM324 Op-Amp [3] indicates that the maximum input DC offset is of 3mV, so taking into account the amplification of 850, we had to compensate for an offset of about 2.5V.

We designed the Summing Inverting Amplifier in a way that it would do 2 main things. Firstly, it would attenuate the square wave input from the Schmitt trigger to be between -2.5V and 2.5V and then add a DC bias of 2.5V so that the signal was exactly between 0 and 5V. This means that the output from the stethoscope would meet the set requirements and also could be used with, for example, an Arduino, which could give many added benefits such as the ability to count and keep track of the heartbeats. We used the following equation to get our values:

$$V_{out} = -R_3\left(\frac{V_1}{R_1} + \frac{R_1}{R_2}\right);\ R_3 = R_4$$

Figure8 shows the design of the Schmitt trigger and the summing amplifier, and Figure9 shows the simulated output after both of these components. We can see that the Schmitt trigger successfully converts the signal from analogue to digital and that the summing inverting amplifier successfully attenuates it and biases it to be between 0 and 5V.

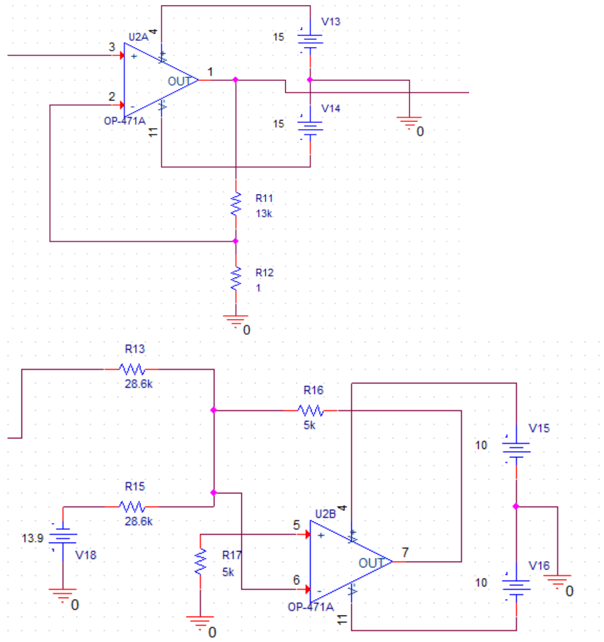


Figure8. Design of Schmitt trigger (top) and summing amplifier (bottom). Made using OrCAD.

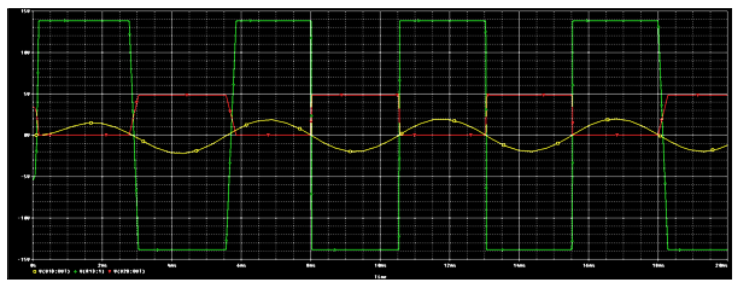


Figure9. Simulated output after the Schmitt trigger (green) and summing amplifier (red)

Due to a shortage of time, we only built the Schmitt trigger, leaving out the summing amplifier. This allowed us to add an LED which acted as a helpful visual cue blinking alongside the heartbeat, as the square wave created by the Schmitt trigger could easily be interpreted as an on-off command (on being positive voltage and off being 0 or negative voltage). It is noticeable that the Schmitt Trigger output inverses the stethoscope output, this is why a summing inverting amplifier was also chosen as the next Op-Amp, it inversed the output a final time.

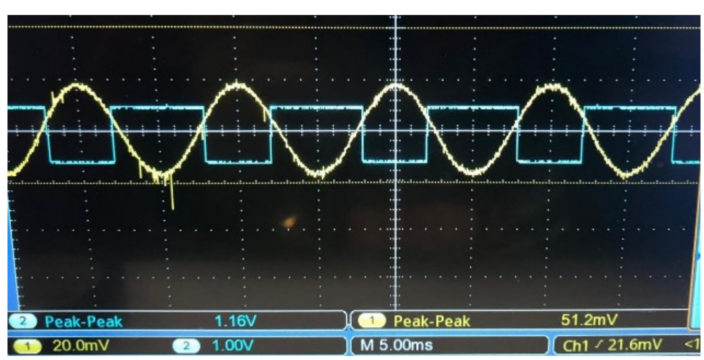


Figure10. Analysis of circuit after the Schmitt trigger made with an oscilloscope

4. Microphone and headphones

After building the circuit and testing its performance using an input wave from a signal generator, we connected a microphone and headphones to show that it also worked as expected on more realistic conditions. To make sure we didn't surpass the maximum power that our headphones could withstand (60mW) or the maximum output current of the Op-Amps (40mA)[3], we introduced a resistor (39Ohm) and a capacitor (470uF) in series with the headphones directly after the output from the filtration. We also added resistors (5.6kOhm and 4.3kOhm in series) and a capacitor (22nF) at the beginning of the circuit to ensure the correct functioning of the microphone. In both cases we used different values to those indicated in the handbook [4], but we made sure to maintain the ratio between the values of the components.

Results and Conclusion

We ran tests with the microphone and headphones on ourselves to try and pick up our own heartbeat, on a dummy that models heart sounds, and on our phones as they played heartbeat simulations.

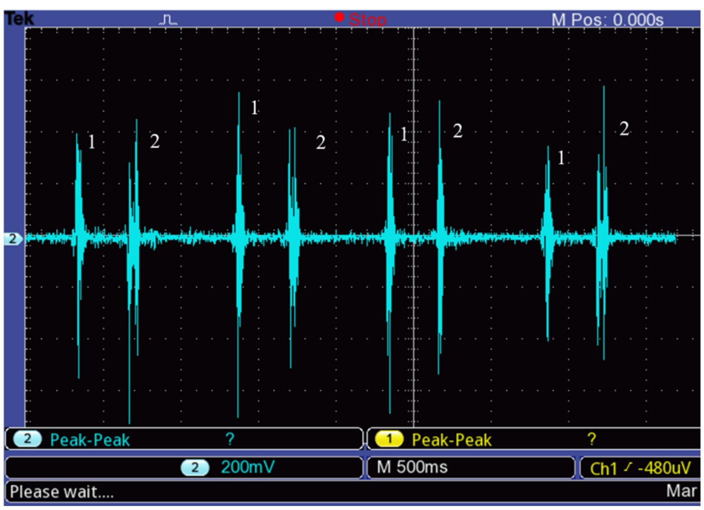


Figure11. Heartbeat sounds from an audio recording processed with our circuit and analysed using an oscilloscope.

The heartbeat sounds were correctly picked up by the microphone and amplified into the headphones with very little noise. We had problems when the amplitude of the input signal was too small, since it wasn't properly registered by the microphone. However, by making appropriate changes to the resistor and capacitor connected for the microphone we were able to consistently pick up heartbeat signals (Figure11) where the "Lub" and "Dub" sounds were clearly discernable, and have been labelled as 1 and 2 respectively.

Considerations and Improvements

The main limitation of our stethoscope is that there is a specific range of input voltage in which it works. If we assume that the Op-Amps saturate at 15V, then the maximum possible input voltage can be around 17mV, as we amplify the signal by 850. We confirmed this experimentally and so kept the input voltage at 15mV. Also, when testing the heartbeat, if the amplitude (volume) of the input was too high, the output signal was cut.

Moreover, although the requirements state that the input voltage must be between 1-5mV, these values gave us a very noisy signal (probably due to the mains) that didn't allow the circuit to work as designed. This limits our stethoscope, since it decreases its precision at small amplitudes or faint heartbeats. A possible improvement could be the use of batteries instead of a mains supply to reduce the noise.

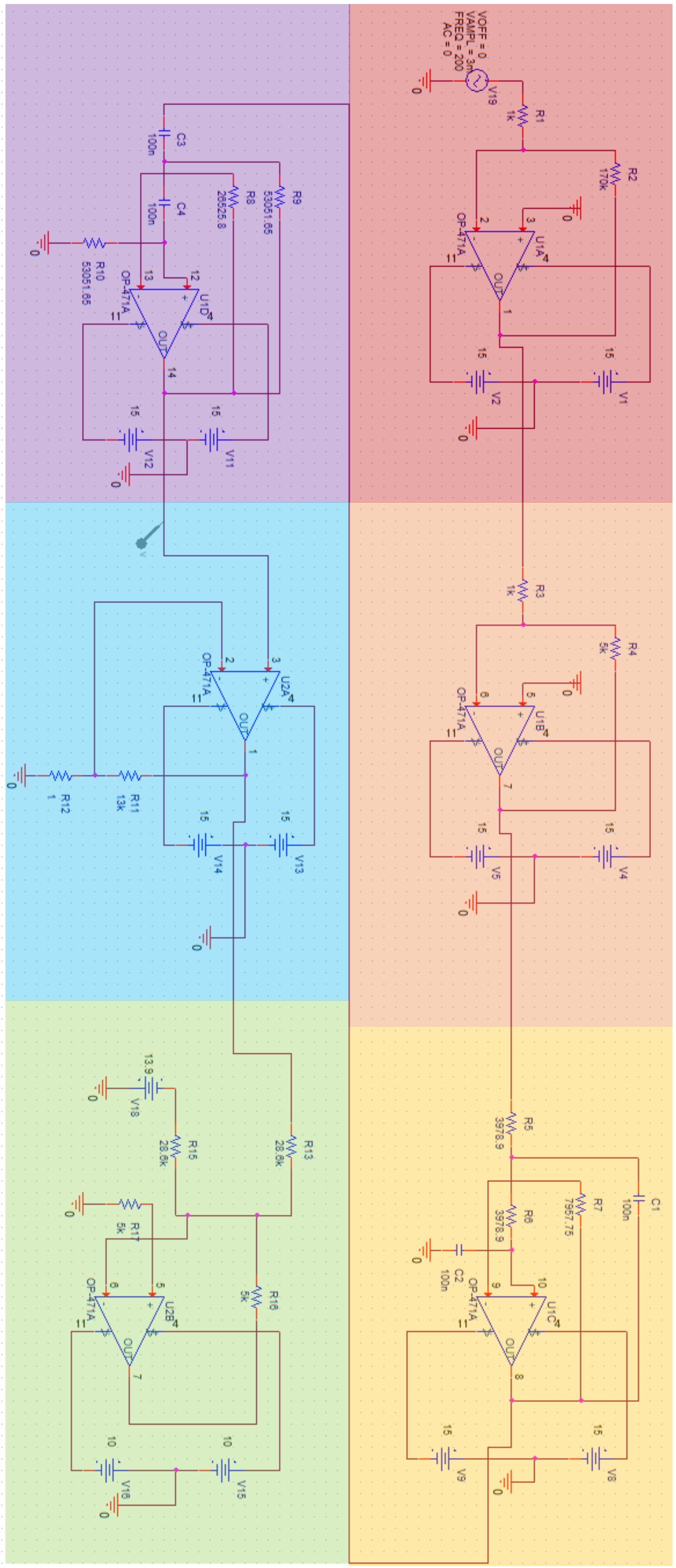
Another problem we came across was the decrease in amplitude after the second filtration. If we were to repeat the design we could try to further compensate for the impedance so that the amplitude does not drop. Additionally, we often had to use a variety of resistors in series instead of a single one due to lack of availability in the lab. This might also introduce further impedance in the circuit and it also increases the experimental error. Ideally, we would use components as efficiently as possible, but this is not always possible in practice.

Further improvements would be the addition of the summing inverting amplifier to allow the connection to an Arduino. At this stage, we could analyze the heartbeat signal to determine the heart rate or analyze the heartbeat itself. We could display the data on the computer and use it to detect dangerous situations or illnesses.

Another useful addition would be using a variable resistor (potentiometer) in the amplifiers, to be able to change the gain of the circuit. The main benefit this would have is that we would be able to compensate for fluctuating input voltages and keep the signal from saturating the Op-Amps. Similarly, we could also control the frequencies filtered in real time, as well as the amplitude of the output (volume). This would allow us to have a greater flexibility and accommodate for all varieties of signals faced in real life scenarios.

References

1. Bioengineering EE Op-Amp lab version 7.2 (2019)
2. Research gate. Amplitude of a normal human heart beat. https://www.researchgate.net/post/What_is_the_approx_frequency_and_amplitude_of_S1_and_S2_sound_of_normal_human_heart_beat3
3. Texas Instruments Incorporated (2015). LMx24-N, LM2902-N Low-Power, Quad-Operational Amplifiers datasheet (Rev. D) http://www.ti.com/lit/ds/symlink/lm324-n.pdf
4. R.Dickinson. Bioengineering EE Practical on Op-Amps v4.8 (2019)



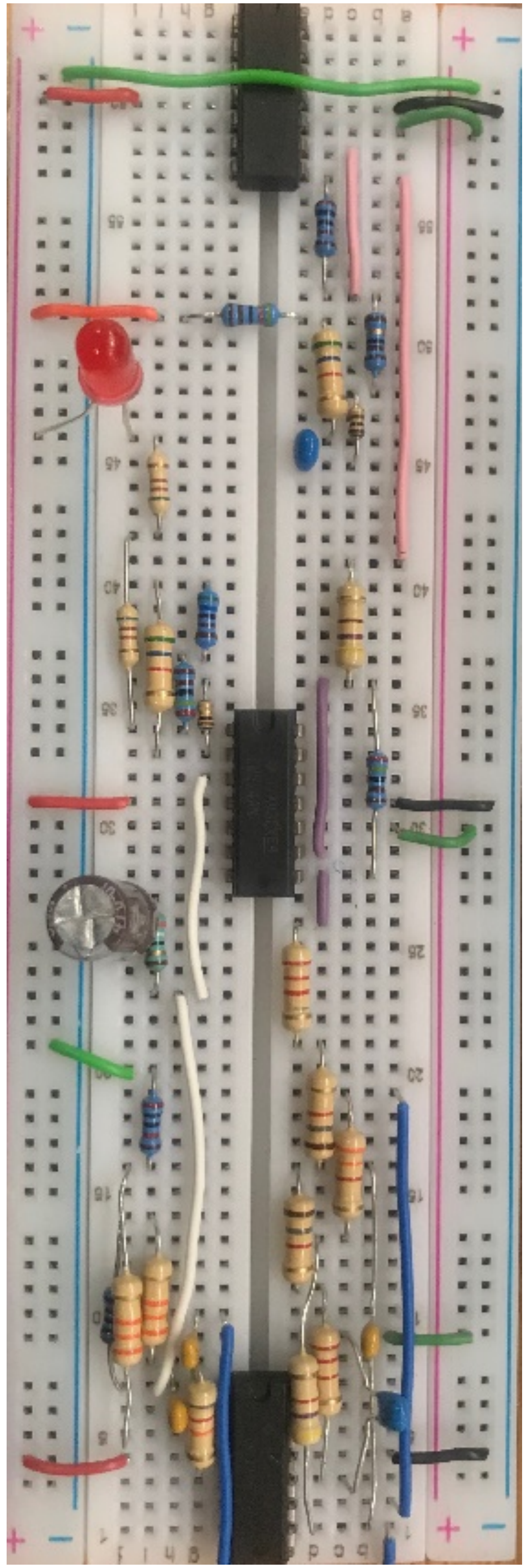


Figure12. Full stethoscope circuit diagram (top) and full electrical circuit (right)