Final Design Report: Lab ABB SP24

Natali Varela, Apoorva Sharma, Daniel Duffy, Alexandra Ying

Abstract (net-ID: daduffy)

The oscilloscope is a powerful tool in the realm of electrical engineering, as it allows one to measure the electrical potential overtime at any part of a circuit. When graphed, a consistent signal will produce a wave that represents the voltage at that part of the circuit and, more importantly, how it changes. The properties of these waves can provide insight into aspects of the circuit by visualizing how different components change the signal as it travels through them. Given a final output, it is therefore possible to reverse engineer a circuit at a high level by looking at the input and outputs. For example, to generate a clipped sine wave given a DC power supply input, the circuit must have a component that converts the signal to AC as well as a component that limits the signal amplitude to clip it. This thought process helped guide our project as we completed the modules and learned about different subcircuits.

Introduction (net-ID: nataliv2)

For our final project, we were tasked with designing a circuit whose output would be a sine wave with an adjustable amplitude between 10-12Vpp, an adjustable frequency between 400-800Hz, and adjustable top and bottom clipping. Through the knowledge we acquired by completing the modules provided for us, as well as doing our own research, we were able to come up with a circuit that checked off all of these requirements by building a circuit that included a 555 timer IC, 4 low-pass filters, a voltage divider, and 3 operational amplifier IC that were constructed in that order as described in the block diagram below.

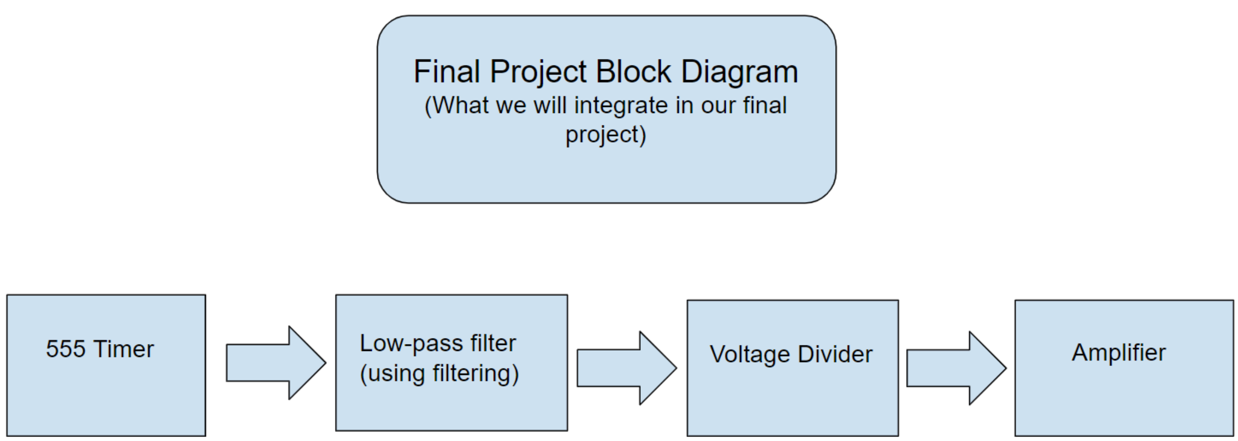


Figure #1: Block Diagram depicting the components of the final project (in order)

In theory, we had thought of using components of all the original modules provided for us to complete our final project, however, through our own research and intuition we came up with the idea of using 3 sub-circuits, which was less than the sub-circuit ideas that were provided for us, to complete the project in a more efficient way. We kept the 555 timer and low-pass filtering sub-circuits more or less the same as how it was explained to us in the modules, however, we modified the amplifier and clipping subcircuits and merged it into one. The main purpose of this sub-circuit was to amplify the sine waves, although 2 extra op-amps were connected to this circuit to perform the clipping function. We encountered a few obstacles with building this circuit and connecting all sub-circuits in unison, nevertheless, we were able to complete the circuit and accomplish all the required tasks.

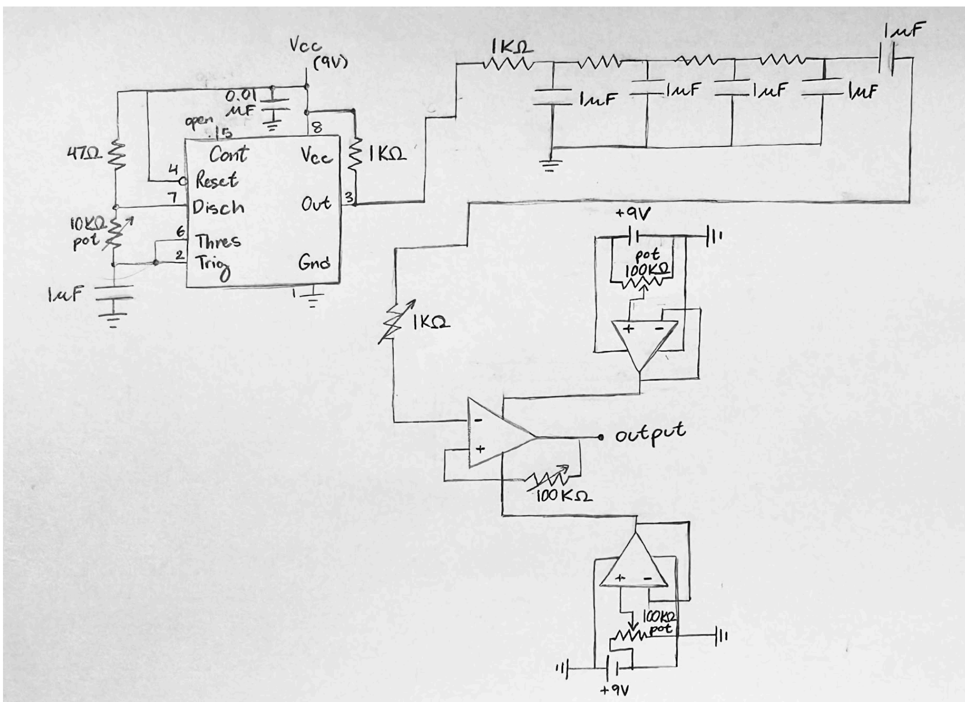


Figure #2: Final circuit diagram

555 Timer IC (net-ID: agying2)

The first step of creating a clipped signal generator is by creating an adjustable signal. The 555 Timer can be used to construct a voltage-controlled oscillator, where a voltage-control signal will cause a change in the oscillating frequency. For our project, it creates an oscillating square wave, and changing the resistor or capacitor values changes the frequency of the wave. This subcircuit creates the signal that varies between 400-800 Hz, and provides a foundation for the following subcircuits to adjust.

In order to have the duty cycle of the square wave to be close to 50%, there needs to be a large difference between R_A and R_B, since duty cycle = $(R_A+R_B)/((R_A+2R_B)*100$. We set R_A to 47 ohms, since this is the smallest resistance value we have in the lab kit. R_B was set as a 10 k ohm potentiometer, which was a large enough resistance value to have the duty cycle be close to 50% when using the equation above.

We left R_B as a potentiometer in order to have an adjustable frequency between the values of 400-800 Hz. This changes the duty cycle slightly, but not enough to distort the final sine wave. The same effect without the change in duty cycle could be achieved by adjusting the capacitor, but in order to avoid swapping components, a potentiometer needs to be used. We used a 1 microfarad capacitor for the timer, and left this constant when changing the wavelength.

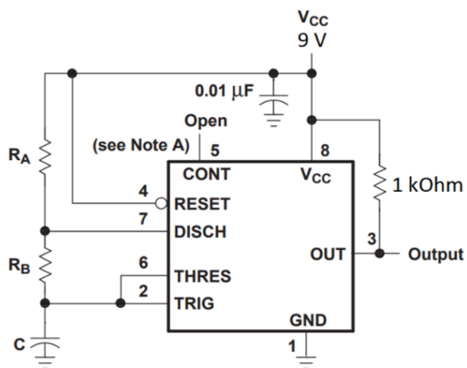


Figure #3: 555 Timer circuit diagram

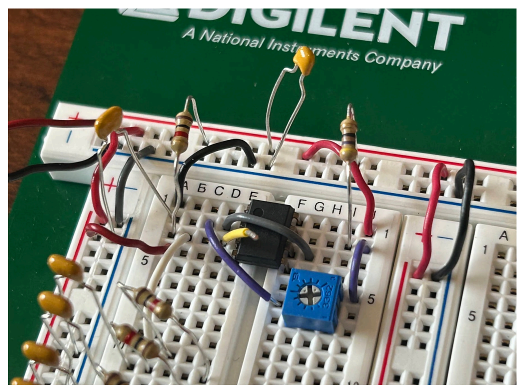


Figure #4: 555 Timer circuit

The 555 timer was tested for proper functionality by measuring the output of pin 3 with the oscilloscope, which produced a square wave as expected. The potentiometer was turned to confirm that the wavelength of the wave could be manipulated within the required range.

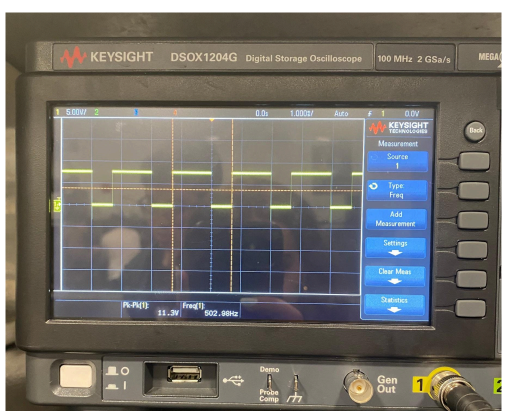


Figure #5: 555 Timer oscilloscope output

Low-pass filters (net-ID: nataliv2)

The following step in our circuit design was adding low-pass filters to transform the square waves generated by the 555 Timer IC into sine waves which would then become amplified and then clipped. In the filtering module, we discovered that low pass filters allow higher gain for lower frequencies, and the low frequencies create the form of the sine wave, so we chose to incorporate these types of filters in our project. Using Falstad's circuit simulator, we decided that at least three low-pass filters were necessary to convincingly convert a square wave to a sine wave. We also added a fourth filter for good measure since we noticed that adding another filter modified the shape of the sine wave and made it look more like its expected result. Our design utilized four filters in a row with a cutoff frequency below the 555 timer's minimum frequency. This ensures that the waveform conversion works at all realistic frequencies.

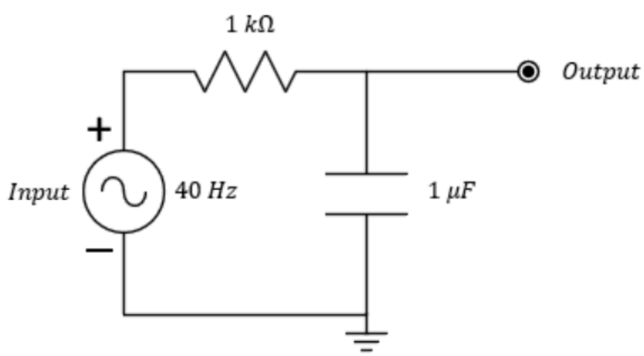


Figure #6: Circuit diagram for low-pass filter

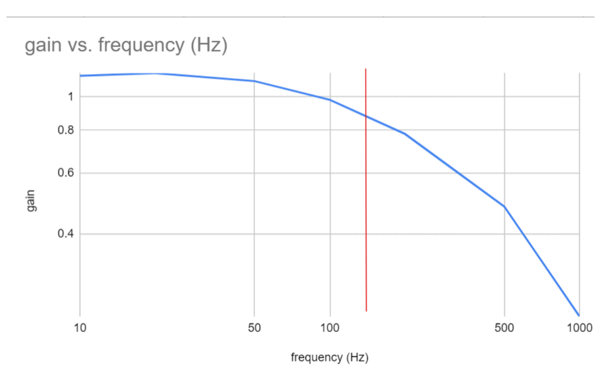


Figure #7: Gain vs Frequency graph of Low-pass filter derived from Filtering Module

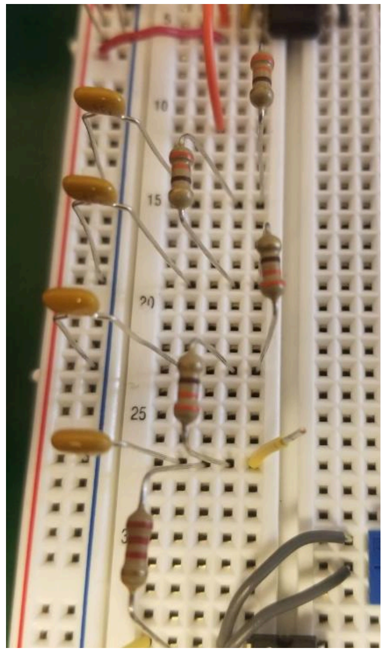


Figure #8: Filtering Circuit

We would test to make sure that the filters were exhibiting their expected behavior by measuring the input and output voltages and using the oscilloscope to see what the graph looked like. Below, we have the result shown on the oscilloscope which was produced after measuring the output of the filters once the 555 timer was tested and complete, as the two subcircuits were connected together.

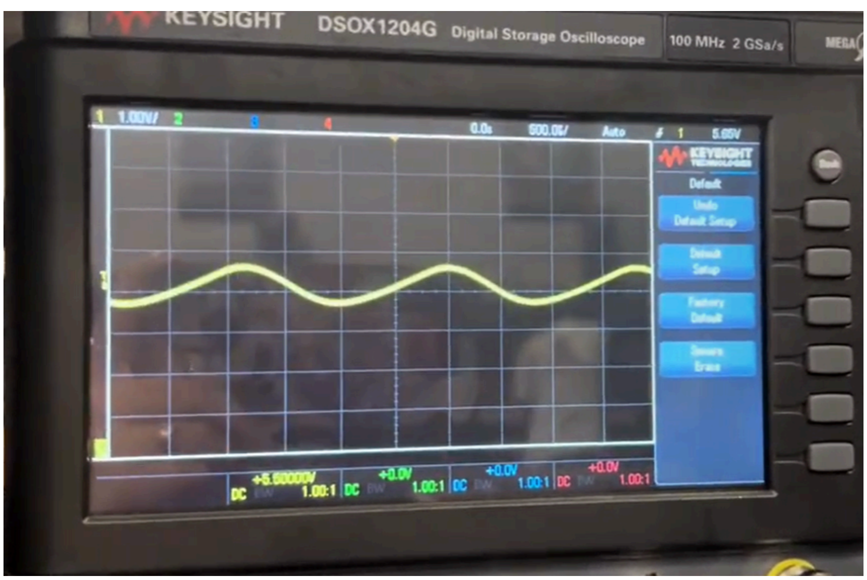


Figure #9: Sine waves as viewed from the oscilloscope

The frequency corresponding to the RC time constant determines the frequency that the filter will allow to pass through, and what frequencies it will attenuate depending on if the filter is low pass or high. Because we know how to solve for this specific frequency using 1/(2π*R*C), the equation for cutoff frequency, we can solve for values R and C for a desired frequency that we want to allow up to and filter. The lowest frequency the 555 timer needs to output is 400Hz. Using the equation for a low-pass filter cutoff frequency, we calculated that a 1kΩ resistor and 1μF capacitor would create a cutoff frequency of about 160 Hz. Having a cutoff frequency lower than the frequency produced by the 555 timer creates the desired sine wave output. This value uses components in our lab kit while also being under the 555 timer minimum frequency. In addition to our sub-circuit, we added an extra capacitor that connects the low-pass filter subcircuit to the amplifier subcircuit which removes the DC offset voltage so the wave could go from -9 V to 9 V. Overall, using these low-pass filters was a crucial and efficient way to complete our final project because it was the simplest way to transform the square wave into the waveform we needed.

Amplifier (net-ID: as204)

The next logical step is to amplify the signal, as the sine wave produced by the 555 timer and low pass filter has a very small amplitude. Also, since one of the requirements of this project requires the output sine wave to have an adjustable amplitude between 10 to 12, this can also be achieved by amplifying the signal. We used the below schematic diagram to gain an idea of how the amplifier circuit is built using an LM358 op-amp.

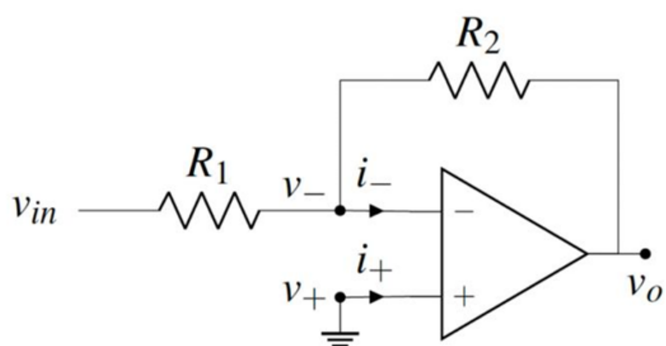


Figure #10: Circuit diagram for amplifier

In the above circuit diagram, there is a resistor, R_1, between the input Vin and the inverting terminal of the op-amp (V-). There is another resistor, R_2, that provides feedback from the output to the inverting terminal. The current flowing through the inverting terminal is i_-. The non-inverting terminal (V+) is connected to the ground and the current flowing through it is i_+. Since V+ is directly connected to the ground, V+ will be 0V, making i_+= 0A. V- will also be 0V because V+ = V- (approximations provided in the op-amp doc). Due to V- being 0V, i_- will be

0A.

V_{in} is applied to R_1, so as per Ohm's law, current V_{in}/R_1 will flow towards v_-, however, this current cannot flow through v_-, so it will take the path containing R_2. Thus, the current flowing through R_2 will be V_{in}/R_1. On applying nodal analysis for R_2, we notice that the voltage on the left side of R_2 will be 0V and the voltage on the right side will be V_o. So,$(0-V_o)/R_2=V_{in}/R_1$. Therefore, $Vo =- Vin(R2/R1)$.
This is the voltage gain formula. (The use of this formula is stated below).

In our circuit, the power supply of the IC was configured from +9V to -9V so that the amplifier could use the full voltage range with both the positive and negative amplitudes of the wave. The original resistors R_1 and R_2 of the above circuit diagram were replaced by potentiometers in our circuit. The output of the 4th low-pass filter was given as the input to the amplifier. Since the amplitude of this incoming signal was quite low, we needed our voltage gain to be high to achieve the required output.

As derived above, the voltage gain formula of an amplifier is $V_o= -V_{in}(R_2/R_1)$, so in order to increase the amplitude, R_2 should be greater than R_1 so, we used a 104 potentiometer as R_2 and 102 potentiometer as R_1. Below, we have provided a picture of our amplifier circuit built on the breadboard.

Below, it can be seen that on adjusting the values of the potentiometer R_2, the input signal is getting amplified and we are able to adjust the output between 10-12 V_{pp} as required.

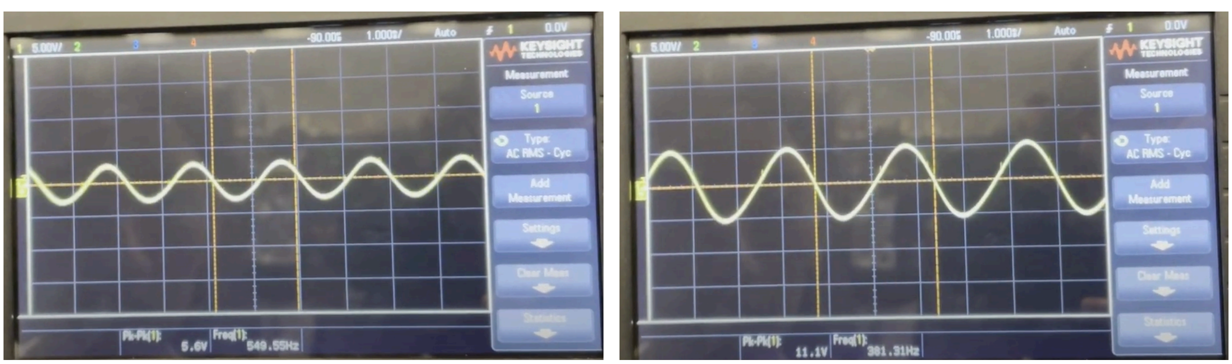


Figure #12: Adjustable sine wave outputs of the amplifier

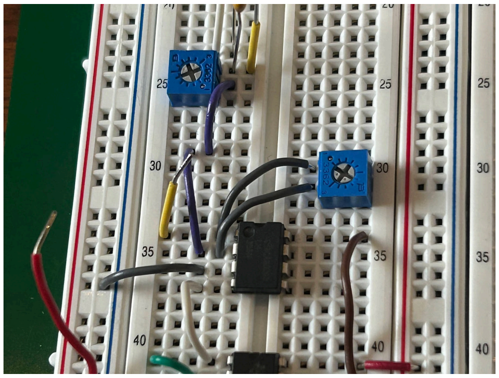


Figure #11: Amplifier Circuit

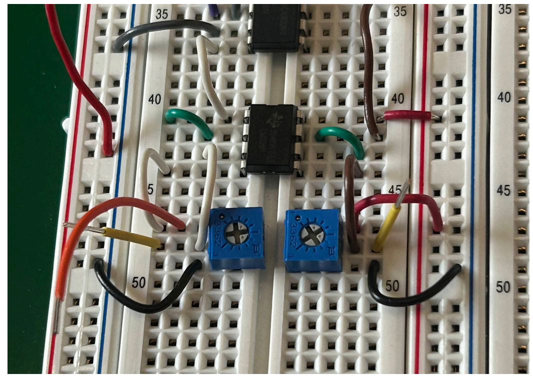
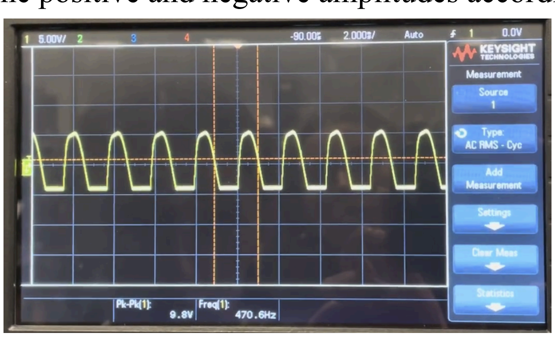


Figure #12: Clipping buffer

Clipper (net-ID: daduffy)

Lastly, the amplified sine wave needs to be clipped. Rather than an explicit subcircuit solely utilized to clip the signal, it is possible to use the inherent properties of the amplifier circuit. Since the op-amp can only boost an input signal up to the limits of its source voltage, it will independently clip the signal at the amplitude equal to the high and low source voltages. By default, these values are +9V and -9V respectively, but if these are adjusted, it will allow the amplifier to double as an adjustable signal clipper. This can simply be accomplished by using voltage dividers with one caveat: Since these signals serve as the source for the op-amp, directly connecting the voltage dividers to the amplifier circuit will cause the op-amp to draw current through the voltage dividers. A voltage divider will only work properly when two resistors are in series, which by definition means the same current flows through both of them. The output of the voltage divider is in between the two resistors, so even without knowing the exact current draw of the IC, the fact that the voltage divider resistors are no longer in series means the voltage divider is no longer reliable. To fix this situation, a buffer or voltage follower in the form of another op-amp IC can be used to observe the voltage divider. Since an op-amp has a high impedance, it will not draw current from its inputs but will still be able to provide a current at its output because it is a powered IC. With this solution implemented at each of the two voltage dividers, the amplitude for the amplifier circuit can be independently controlled by the positive and negative limits. This was tested by independently turning the potentiometers to divide the high and low-voltage sources. This functionality was reflected on the oscilloscope by clipping the positive and negative amplitudes according to their respective dividers.



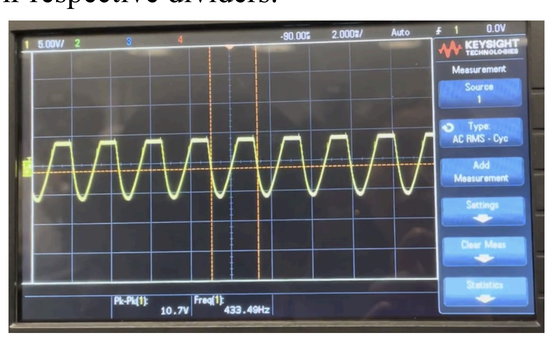


Figure #13: Clipping the negative amplitude. *Figure #14: Clipping the positive amplitude*

Conclusion (net-ID: as204, agying2)

The overall goal of this project was to build a circuit that would produce a sine wave with an adjustable amplitude between 10-12 Vpp, an adjustable frequency between 400-800 Hz, and adjustable top and bottom clipping as output. To successfully achieve this, we made use of a 555 Timer, low-pass filters, an LM358 configured as an amplifier, and another LM358 configured as a buffer. The 555 Timer was used to produce an oscillating square wave which was then converted into a sine wave using the low-pass filters. To amplify and clip the sine wave as per the requirements of the project, we used an amplifier. Finally, the buffer was used to guarantee the correct working of our circuit. We used 3 of the 5 modules provided, which is the result of us choosing not to use a separate subcircuit to accomplish the clipping. This simplified the final circuit and reduced the number of additional components.

If we were to revisit this project, we could explore the usage of the BJT clipping and integrator modules to determine the specific advantages and disadvantages of using those subcircuits to accomplish the clipping. Also, we could experiment with using 3 low pass filters instead of 4, since when we exceeded 3 filters, the additional filters refined the sine wave less compared to the previous ones.

An interesting expansion of this project could include an LED indicator sub-circuit. This would prompt the LED to switch on when the desired clipping and amplification level has been reached.