

# ENGS 32 Final Project Report

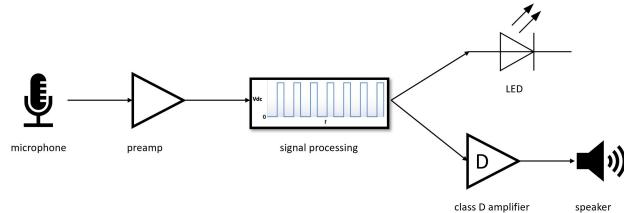
Walden Marshall

February 2021

## I Project Overview

For this project, I was tasked with designing an analog circuit to monitor the heartbeat signal from a stethoscope. The stethoscope signal had to be processed so that it could drive a speaker and turn on an LED when a heartbeat was detected. It was necessary that the circuit detect heartbeats in the range of 60-100 bpm. It was also necessary that it detect each heartbeat only once, and miss no heartbeats.

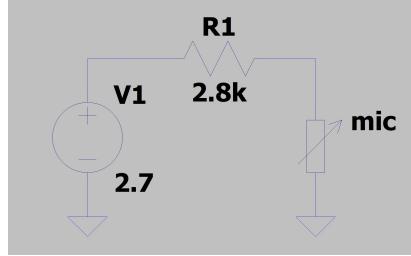
The circuit I designed to accomplish this task is based on the following high level block diagram.



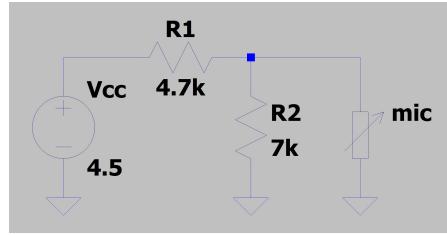
## II Technical Details

### II.1 Microphone

The role of the microphone subsystem is to output a very small signal proportional to the vibration caused by the heartbeat (with a DC offset). The microphone is used as a load attached to a Thevenin circuit. The microphone contains charged plates whose separation varies due to the vibration from the heartbeat. This causes the load impedance to change, and hence the output voltage as well. The circuit we were given to properly operate the microphone is shown below.



The Thevenin portion of this circuit has an open circuit voltage of  $V_{oc} = 2.7$  V and short circuit current of  $I_{sc} = 964 \mu\text{A}$ . Since I did not have a 2.7 V source, I implemented a circuit with the same response to a load using the +4.5 V power rail as the voltage source.

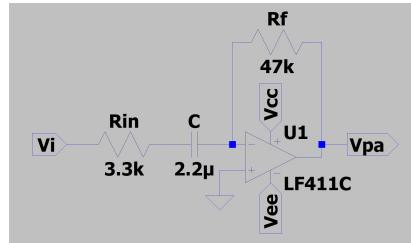


This circuit has Thevenin characteristics of  $V_{oc} = 2.69$  V and  $I_{sc} = 957 \mu\text{A}$ , which are extremely close to the specified values.

## II.2 Preamplifier

The microphone subsystem outputs a very small amplitude signal that has a DC offset. The role of the preamplifier subsystem is to remove the DC offset of the microphone signal and provide gain so that the signal is large enough to be processed effectively.

To achieve these tasks, a first order active highpass filter was implemented.



The transfer function of the circuit is

$$H(s) = \frac{V_{pa}}{V_i} = -\frac{R_f}{R_i} \left( \frac{s}{s + 1/R_i C} \right)$$

We were told that the sound of a heartbeat is roughly 50 Hz, so it was important

to place the pole,  $1/R_i C$ , much below that so as not attenuate the frequency of interest. Using component values  $R_i = 3.3 \text{ k}\Omega$  and  $C = 2.2 \mu\text{F}$ , the pole occurs at  $(1/2\pi)(1/R_i C) = 21.9 \text{ Hz}$ .

The resistance  $R_f$  had to be varied experimentally, since I wasn't sure what the amplitude of the microphone's signal was. In order for the next subsystem to function, I desired a peak value of  $\max(V_{pa}) \approx 1 \text{ V}$ . This was achieved with  $R_f = 47 \text{ k}\Omega$ .

Assuming an output signal during the pulse of a 50 Hz sinusoid of 1 V amplitude, the slew rate,  $\max(\frac{dV_{pa}}{dt})$ , would be  $100\pi \text{ V/s}$ , or  $3.14 \times 10^{-4} \text{ V}/\mu\text{s}$ , which is well within the LF411's allowable slew rate of  $13 \text{ V}/\mu\text{s}$ . The 50 Hz signal is well within the specification of the gain bandwidth product of the LF411, which is 3 MHz. When powered by  $\pm 4.5 \text{ V}$ , the LF411 can only output amplitudes up to 3 V, but the output signal desired is only 1 V. At 50 Hz, the capacitor's impedance is about  $j1.5 \text{ k}\Omega$ , which makes the magnitude of the input impedance  $\sqrt{1500^2 + 3300^2} \approx 3.6 \text{ k}\Omega$ . This means  $F = 71 \text{ mV/V}$  and  $N = -929 \text{ mV/V}$ . The LF411 has an open loop gain of  $A = 200 \text{ V/mV}$ . The perturbation from ideal conditions is

$$\varepsilon = \frac{1}{1 + AF} = 7 \times 10^{-5}$$

which is very small compared to 1, so we can ignore the finite gain. In summary, the LF411 will perform well in this subsystem.

Since the characteristics of the specific microphone I had in my position were integral to this subsystem working, LTspice simulations would not be sufficient. I ensured this worked by measuring the signal. The scope capture below shows the microphone signal in yellow and the preamp signal in blue.



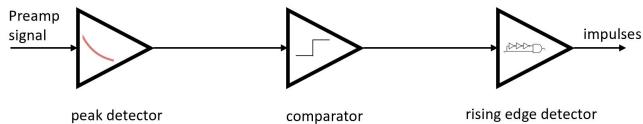
Since the microphone and preamp subsystems contain such few components, I have included their bill of parts together.

Subsystem	Part	Quantity	Price/item	Total	Subsystem Total
Microphone	Resistor	4	\$0.01	\$0.06	
	Stethoscope	1	\$18.99	\$18.99	
Preamplifier	Resistor	2	\$0.01	\$0.02	
	Aluminum Electrolytic Capacitor	1	\$0.05	\$0.05	
	LF411CP Amplifier	1	\$1.84	\$1.84	\$20.96

### II.3 Signal Processing

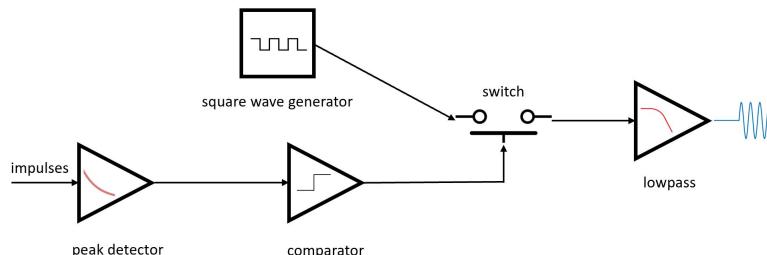
In order to turn the output of the preamp into a signal that can be passed to a power amplifier, the Signal Processing block must perform two tasks.

First, it must eliminate the "dub" portion of the heart beat. This was achieved in Lab 7 with the following block diagram.



The peak detector and comparator combine to create a digital signal whose response to a heartbeat pulse is to remain high for some set amount of time. The rising edge detector outputs a high voltage for a very short duration as the comparator switches from low to high.

Next, the signal processing must turn the impulses from the output of the block diagram above into a bursts of audio frequency sinusoidal oscillation. This was achieved in Lab 6 with the block diagram below.

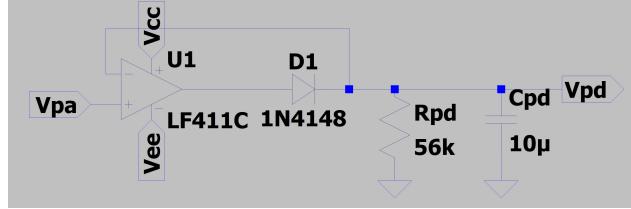


The combination of the peak detector and comparator respond to an impulse with a digital signal that stays high for some fixed amount of time. This digital signal controls a switch to let through the signal from a square wave generator. A lowpass filter then turns this square wave into a sine wave approximation.

I will first go through the Lab 7 portion, which ignores the "dub", Then I will go through the Lab 6 portion, which creates the pulsed sine wave.

### II.3.1 Ignore "Dub"

First is the peak detector.



To analyze this circuit, assume the diode needs a full 0.7 V voltage drop across it to turn on. Also, in the MoHAT analysis, treat the resistor as an open circuit, since we are examining the circuit at a single instant and not considering the effects of the  $RC$  time constant yet. We must consider this circuit case by case.

In the first case, assume:  $V_{pa} > 0$ , the capacitor has some charge on it such that it has a voltage  $0 < V_{pd} < V_{pa}$ , and the diode is on. If the diode is on, then there is a path from the output of the amplifier to the inverting terminal, so the Golden Rule of Feedback applies, and we can see that  $V_+ = V_-$ . If the diode is on, then there is a 0.7 V drop from the output of the amplifier to  $V_-$ , so therefore the amplifier output goes to  $V_{pa} + 0.7$  V and sources current to charge the capacitor up to  $V_{pd} = V_{pa}$ . Therefore current flows through the diode in the forward direction, and we find that the assumption that it was on is correct. Furthermore, since  $V_+ = V_{pa}$  and  $V_- = V_{pd}$ , the output of the filter is equal to the input.

In the next case, assume: charge is accumulated on the capacitor so that  $V_{pd} > 0$ , the filter input satisfies  $0 < V_{pa} < V_{pd}$ , and the diode is on. If the diode is on, then inverting feedback is present, so  $V_+ = V_-$ . But  $V_+ = V_{pa}$  and  $V_- = V_{pd}$  and the initial assumption was that  $V_{pd} > V_{pa}$ . In order to satisfy  $V_+ = V_-$ , the amplifier would have to sink current to let the capacitor discharge. The amplifier cannot sink current because of the diode, so there is a contradiction, so the diode must be off. If the diode is off, no inverting feedback exists, and  $V_{pd}$  remains constant.

Of course in reality, the capacitor can discharge through the resistor, so this comparison between  $V_{pd}$  and  $V_{pa}$  occurs continuously.

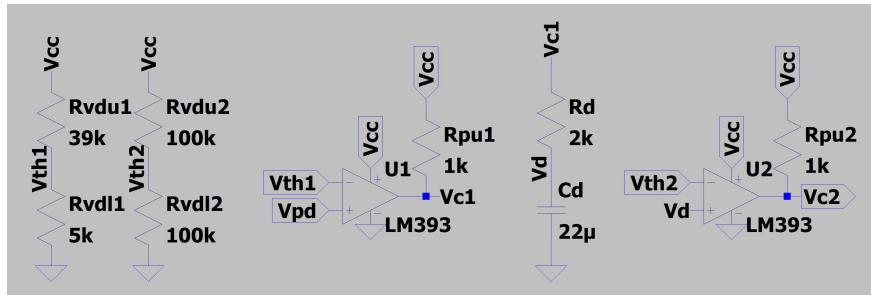
The peak detector response to a very short pulse of amplitude 1 is  $V_{pd}(t) = e^{-(\frac{1}{R_{pd}C_{pd}})t}$ , which has a half-life of  $t_{1/2} = \ln(2)R_{pd}C_{pd} = 390$  ms. This decays slowly enough that the peak detector ignores the slightly lower amplitude "dub".

The main concern for the amplifier in this circuit is whether it will be able to source the necessary current to keep up with a rising  $V_{pa}$ . If  $V_{pa}$  is rising, then  $V_{pa} = V_{pd}$ . If  $I_o$  is the current supplied by the amplifier, then the transfer function from  $V_{pa}$  to  $I_o$  is

$$\frac{I_o}{V_{pa}} = C_{pd}s + 1/R_{pd}$$

Assuming  $V_{pa}$  is a 1 V amplitude 50 Hz sinusoid, the maximum current sourced by the amplifier is about 3.1 mV. Unfortunately, neither the LF411 or LF412 include the output short circuit current on their datasheet, but the Texas Instruments' LM741 has an output short circuit current of 25 mA. Of course it is not ideal to assume the LF411 is similar to the LM741, but since the necessary parameter is not included on the datasheet, this assumption combined with the knowledge that I have previously used the LF411 to drive much smaller loads makes it a relatively safe choice.

After the peak detector comes the comparator. As discussed earlier, the purpose of this block is to provide a digital signal to open and close the switch, which can be implemented with a single comparator. However, I have actually used two comparators to create a delay between the peak detector crossing the comparator threshold and the output to the switch changing. This delay was necessary for a previous design that I abandoned, but I chose to keep the delay because I already had it assembled and it did not detract from my final design.



Using  $1\text{ k}\Omega$  pull-up resistors keeps the current sunk to ground by the comparator to 4.5 mA, which is safely under its maximum allowable sink current of 16 mA. Therefore we are using the comparators safely.

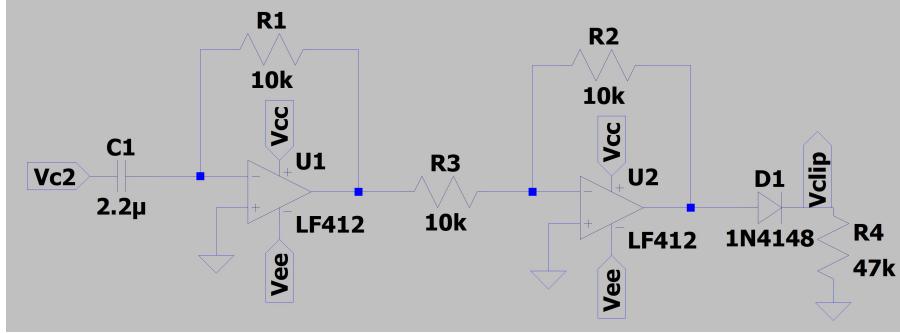
The first voltage divider creates a threshold of  $V_{th1} = 511\text{ mV}$ , which is approximately half the amplitude of the peak detector, so the comparator will trigger after one half life of the peak detector, or 390 ms. When it triggers, it closes the switch, allowing  $C_d$  to charge up through  $R_{pu1}$  and  $R_d$ . This charge up has the equation

$$V_d(t) = 4.5(1 - e^{-\frac{t}{\tau}})$$

where  $\tau = (R_{pu1} + R_d)C_d = 66\text{ ms}$ . Since the second comparator is set to trigger at half the amplitude of the saturating exponential's final value, the delay between when the first and second comparator trigger on is equal to the half life of the exponential decay:  $t_{1/2} = \ln(2)\tau = 46\text{ ms}$ .

Rather than implementing an edge detector with the typical sequence of digital gates, I implemented one with a differentiator, clipping diode, and comparator. The differentiator creates pulses at edges of the digital signal, the clipping diode removes the negative pulse from the falling edge, and the comparator offers a better way to control the amplitude of these pulses than changing the internal workings of the differentiator circuit.

The first part of the edge detector is the differentiator with clipping shown below.

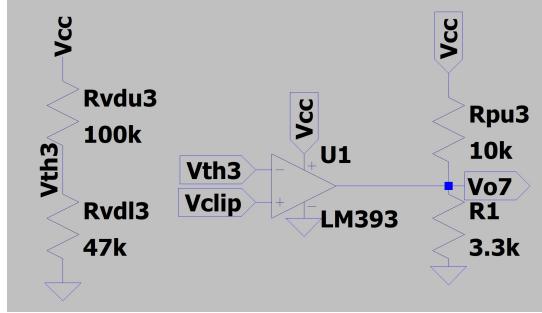


It is important to note that although the model of the comparator signal is a 4.5 V step input, what actually happens is  $V_{cc}$  becomes tied to  $C_1$  through the pull up resistor when the comparator's output switch opens. Therefore we must consider a resistor  $R_{pu} = 1 \text{ k}\Omega$  in series with  $C_1$ . This means the first op amp stage is no longer a pure differentiator, but a filter with a zero at the origin and a pole. It has transfer function

$$H(s) = -\frac{R_1 C_1 s}{R_{pu} C_1 s + 1} = -\frac{R_1}{R_{pu}} \left( \frac{s}{s + 1/R_{pu} C_1} \right)$$

Note that the larger  $R_1$  is compared to  $R_{pu}$ , the closer the circuit is to an ideal differentiator. The LM393 lists the switch time as  $V_{cc}/300 \text{ ns}$ , so I could calculate the theoretical amplitude of the differentiator's impulses, but as I will discuss in Section III, this circuit's functionality relies on certain nonidealities which cannot be obtained from the datasheet. Therefore, for the most part, the values in the first op amp stage were determined by simulation and experimental trial and error, with the goal of creating an output signal of large enough amplitude to reliably trigger a comparator. The second stage is merely an inverter, which is not strictly necessary, but I was using the dual LF412 op amp so it was more convenient to add a second stage and have the sign of the output signal the same as the sign of the derivative being measured. The load consists of a diode and resistor. The diode of course is used to remove the pulse due to the falling edge from the output. 47 kΩ was chosen because it is a fairly large resistance, relative to some nonlinear but small impedance from the diode. If the diode when on is modeled as a small resistor, this creates something of a voltage divider, where the attenuation is fairly small. The  $V_{clip}$  signal ends up being roughly 2.5 V at the rising edges, meaning the amplitude is much larger than the threshold of the diode, so current is being let through, and the diode is on where we intend it to be. Slew rate limitation is a very real possibility in this circuit, but since the actual derivative of the comparator output was not calculated theoretically, slew rate calculations were not performed either. In fact it is very possible that the amp is slewing, but as long it reaches the amplitude intended, whether it is slewing or not is of no concern.

Since the differentiator's amplitude is tuned experimentally, it seemed unwise to use its amplitude as a critical parameter in the circuit. Therefore, an output comparator is used to ensure that the edge detector outputs an impulse of 1 V. Shown below is the output comparator.



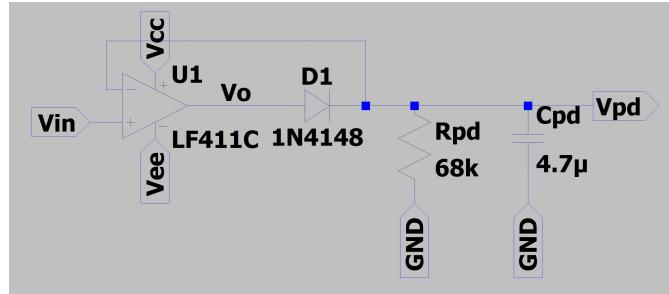
Since the output of the edge detector was roughly 2.5 V, I set the threshold to 1.44 V with the voltage divider involving  $R_{vdu3}$  and  $R_{vdl3}$ . When the comparator output is low,  $R_1$  is shorted and  $V_{o7} = 0$ , and when the comparator is high,  $R_{pu3}$  and  $R_1$  form a 1.12 V voltage divider of  $V_{cc}$ .

The Lab 6 portion of the circuit was designed with its input modeled as a single 1 V impulse per heartbeat. Therefore, the section designed to ignore the "dub" has provided the right output for the next section, the pulsed sine wave generator.

### II.3.2 Pulsed Sine Wave

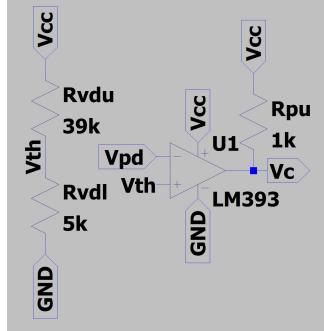
In order to create a pulsed sinusoidal waveform, I used the control signal of the heartbeat to turn on and off the signal from an oscillator. This control was achieved using a peak detector and comparator combination.

The peak detector circuit is the same as used previously, just with different values for the resistor and capacitor, so all the same MoHAT analysis applies here.



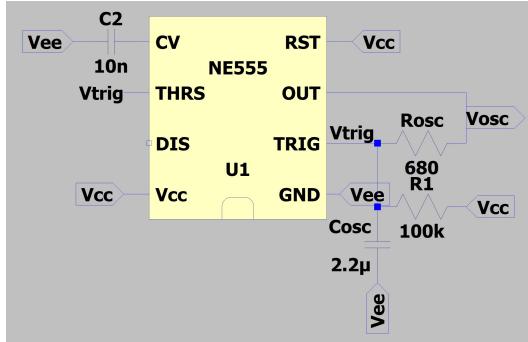
The half-life of this exponential decay is  $\ln(2)R_{pd}C_{pd} = 220$  ms. Since the impedance of its RC network is higher than that of the previous peak detector, we need not worry about the finite current limitation of the LF411.

In the next section, the peak detector voltage triggers an inverting comparator.



Setting the threshold of the comparator to 511 mV, which is roughly half the amplitude of the peak detector, we ensure that the time the comparator is on is roughly equal to the half-life of the RC decay, 220 ms. Since this signal ultimately controls the tone burst, we have set the time the speaker will be on per heartbeat to about 220 ms. Again, 1 k $\Omega$  has been determined to be a safe value for the pull-up resistor.

Before we examine the switching circuitry, examine the square wave generator. This 50% duty cycle pulse generator<sup>1</sup> is implemented with the LM555.



The frequency of the oscillator is given by

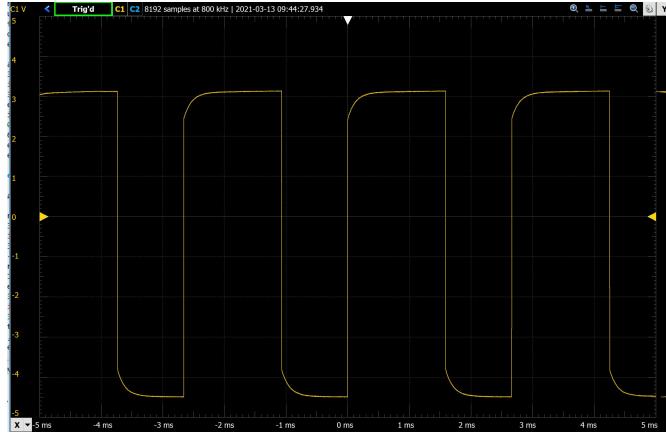
$$f = \frac{1}{2 \ln(2) R_{osc} C_{osc}}$$

For  $R_{osc} = 680 \Omega$  and  $C_{osc} = 2.2 \mu\text{F}$ , a frequency of 482 Hz is achieved, which is well within the audible range. One important thing to note with this circuit is that the schematic in the cited website includes connections to ground, but the circuit I implemented includes connections to  $V_{ee}$  instead. This is because the LM555 data sheet calls for a 5 V power supply, but we only have  $\pm 4.5$  V. While running some integrated circuits (like the LF411) slightly under their specified

---

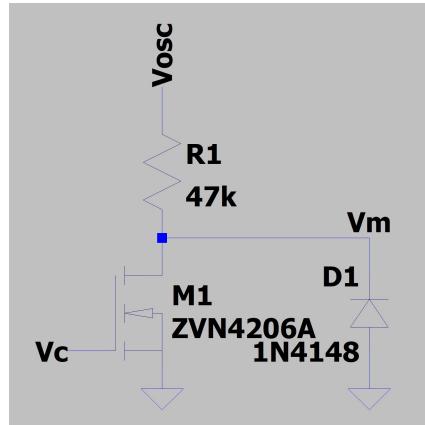
<sup>1</sup>[https://www.electronics-tutorials.ws/waveforms/555\\_oscillator.html](https://www.electronics-tutorials.ws/waveforms/555_oscillator.html)

power supply is not problematic, when I did this for the LM555, I saw the duty cycle become extremely asymmetrical. Powering the chip with  $\pm 4.5$  V created a much more symmetrical pulse, though it was still not the ideal square wave expected.



The high value of the square wave does not go all the way to  $V_{cc}$ , but to about 3 V instead. The pulse is on for longer in the high state than the low state, so the time average value of the wave is still fairly close to 0 V.

This oscillator constantly created a square wave as long as it was powered, but this square wave was only let through to the next section of the circuit by the opening of an nMOS switch. The configuration for this switch is shown below.



The control signal,  $V_c$  can be either 0 V or 4.5 V and the oscillator signal,  $V_{osc}$ , can be either 3 V or -4.5 V. Therefore to determine the output,  $V_m$  of this circuit, we must consider four cases.

First consider the case that  $V_c$  is high and  $V_{osc}$  is high. The ZVN4206A data sheet says that  $V_{th} = 1.3$  V. Though the oscillator signal's high value

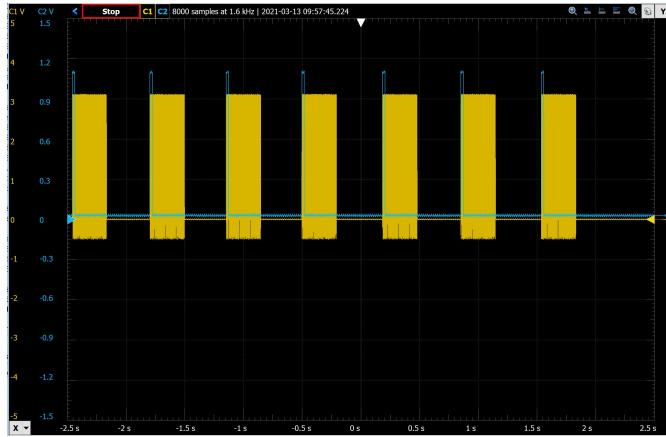
was supposed to be 4.5 V, I measured only about 3 V, so  $V_{DS} = 3$  V. If the comparator high goes all the way to 4.5 V, then  $V_{GS} - V_{th} = 3.2$  V >  $V_{DS}$ , so the transistor would be in the saturation. However, according to the data sheet,  $R_{DS} = 1.5 \Omega$  at  $V_{GS} = 5$  V, so the 4.5 V control signal will cause the transistor to behave approximately like a  $1.5 \Omega$  resistor. This means that  $V_{DS}$  will be the voltage division of the high value of  $V_{osc}$  by  $R_1$  and  $R_{DS}$ . Since  $R_1 \gg R_{DS}$  it follows that  $V_{DS} \approx 0$  V, and therefore the transistor is in the linear region. The diode will of course be off since it is reverse biased.  $R_1$  and  $R_{DS}$  form a voltage divider but since  $R_{DS}$  is tiny compared to  $R_1$ , the output voltage will be  $V_m \approx 0$  V.

Next consider the case that  $V_c$  is high and  $V_{osc}$  is low. While  $V_{GS}$  is large enough to turn the transistor on,  $V_{DS} < 0$ . The transistor is not designed to let current flow in this direction, so with the addition of the diode, we can treat the transistor as an open circuit in this case. The voltage drop from the anode of the diode to  $V_{osc}$  is 3 V, so the diode is forward biased. With a 0.7 V drop across the diode, 49  $\mu$ A would be allowed to flow from ground to  $V_{osc}$ , so the diode is indeed on. However, examining the VI curve on the datasheet of the 1N4148, we see that at 49  $\mu$ A, the voltage drop across the diode is only about 475 mV. Of course the ratio of voltage to current is nonlinear in the diode, so it is hard to calculate the exact voltage across the diode, but because  $R_1$  is large enough to significantly limit the current through the diode, it is actually only partially on, and the output voltage will be  $0 \text{ V} < V_m < 0.7 \text{ V}$ .

Next consider the case that  $V_c$  is low and  $V_{osc}$  is high. Since  $V_c$  is low, the transistor is off, and the circuit becomes 3 V across  $R_1$  and a reversed biased diode. Since the diode is reverse biased, it is off. No current flows, so  $V_m = V_{osc} = 3$  V.

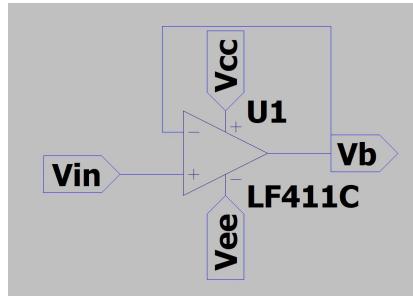
Next consider the case that  $V_c$  is low and  $V_{osc}$  is low. As in the second case the diode is forward biased, so the output of the switching circuitry is  $0 \text{ V} < V_m < 0.7 \text{ V}$ .

So we see that the output oscillates from 3 V to a small voltage when the the control signal is low and stays stays relatively close to 0 V when the control signal is high. This creates noise bursts as intended. Testing the switch showed that it worked properly. The yellow signal in the scope capture below is the output of the switch and the blue signal is the input to the output from the Ignore "Dub" System.



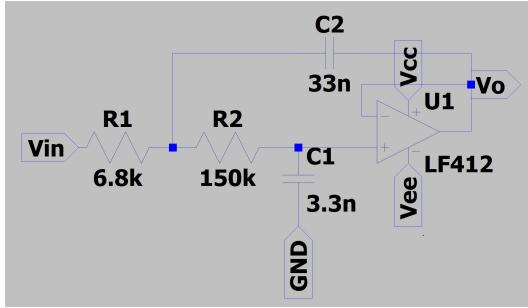
While the switch output does not go as high as expected, this is not problematic, since that signal would have had to be attenuated anyway. The reason the switch output does not go as high as expected is because the output of the 555 timer is measured as only going up to 3 V.

Since these noise bursts are square shaped, they must be processed further in order to be a useful audio signal. This was achieved by buffering the signal, lowpass filtering it, then removing the DC offset. The buffer used is shown below.



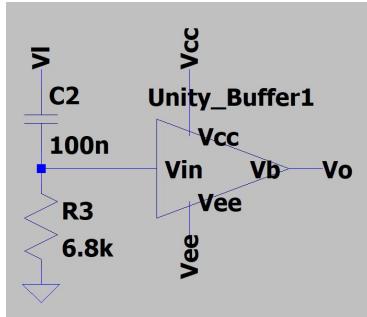
The  $V_m$  signal has a very high output impedance, so feeding it directly to a filter would make the filter much harder to design, hence a buffer is used to give the filter a more ideal input source. It was predicted that the input to the buffer would go up to 4.5 V, but the LF411 can only go up to 3 V output. I determined that since I was going to process the signal further anyway, it was not particularly important whether the signal got clipped, though I would have to take this into consideration when designing my filter. However, since the output of the switch only ends up going up to 3 V, if the buffer clips the signal at all, it is very minimal.

Next, the buffer signal is lowpass filtered.

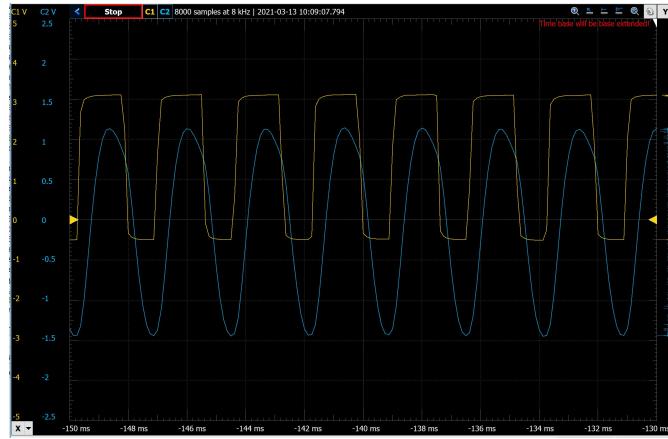


I observed through measurement that the buffer signal was approximately 3 V in amplitude, which was acceptable, so I didn't bother adding gain or attenuation. Rather I used the Analog Filter Wizard to create second order lowpass with a corner frequency of about 600 Hz. This would put the fundamental frequency, 482 Hz, safely in the passband while attenuating the upper harmonics. The frequency I wanted to preserve, 482 Hz, is well within the LF412's gain bandwidth product of 4MHz, and  $\max(\frac{dV_o}{dt}) = 9 \text{ kV/s}$  or  $9 \times 10^{-3} \text{ V}/\mu\text{s}$ , which is well within the LF412's allowable slew rate of  $15 \text{ V}/\mu\text{s}$ .

Next we need to remove the DC offset with a highpass filter. A single passive RC followed by a buffer was all that was necessary to accomplish this task.



The filter has a zero at the origin and a pole at  $\frac{1}{2\pi R_3 C_2} = 234 \text{ Hz}$ , putting the fundamental frequency of 482 Hz safely in the passband. The buffer uses the same topology before, though this one is an LF412. This lowpass and highpass filter combine to create a suitable sine wave approximation as shown below. The yellow signal is the output of the switch and the blue signal is the output of the cascaded filters.



Note that the peaks of the sinusoidal approximation are not perfectly symmetrical about the  $x$ -axis: The high peak is about 1.4 V and the low peak is about -1.2 V. While this is not a huge concern at the moment, it ended being a large design consideration in the Class D Amplifier. However, the implications of this will be discussed more in relation to the other components of the Class D Amplifier.

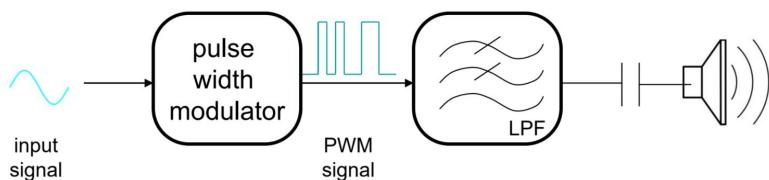
This portion of the circuit, from Lab 6, provides an approximation of a sine wave with peak-to-peak of roughly 2.6 V and no DC offset. While it ended up being modified slightly, this signal is theoretically appropriate to drive a power amplifier.

The bill of parts for the Signal Processing system is shown below, broken down by subsystem. The order of subsystems is also the same as the order of subsystems in the signal chain.

Subsystem	Part	Quantity	Price/item	Total	System Total
Peak Detector 1	LF411CP Amplifier	1	\$1.84	\$1.84	
	1N4148 Diode	1	\$0.04	\$0.04	
	Resistor	1	\$0.01	\$0.01	
	Aluminum Electrolytic Capacitor	1	\$0.05	\$0.05	
Comparator (Delay)For PD1	LM393AP Comparator	1	\$0.40	\$0.40	
	Aluminum Electrolytic Capacitor	1	\$0.05	\$0.05	
	Resistor	9	\$0.01	\$0.09	
Edge Detector	Aluminum Electrolytic Capacitor	1	\$0.05	\$0.05	
	Resistor	7	\$0.01	\$0.07	
	LF412CN Amplifier	1	\$1.47	\$1.47	
	LM393AP Comparator	1	\$0.40	\$0.40	
Oscillator	LM555CN Timer	1	\$1.17	\$1.17	
	Resistor	2	\$0.01	\$0.02	
	Aluminum Electrolytic Capacitor	1	\$0.05	\$0.05	
	Ceramic Capacitor	1	\$0.10	\$0.10	
Peak Detector 2	LF411CP Amplifier	1	\$1.84	\$1.84	
	1N4148 Diode	1	\$0.04	\$0.04	
	Resistor	1	\$0.01	\$0.01	
	Aluminum Electrolytic Capacitor	1	\$0.05	\$0.05	
Comparator For PD2	LM393AP Comparator	1	\$0.40	\$0.40	
	Aluminum Electrolytic Capacitor	1	\$0.05	\$0.05	
	Resistor	3	\$0.01	\$0.03	
Switch	Resistor	1	\$0.01	\$0.01	
	ZVN4206A nMOS	1	\$0.68	\$0.68	
	1N4148 Diode	1	\$0.04	\$0.04	
Bandpass Filter	LF412CN Amplifier	1	\$1.47	\$1.47	
	Resistor	4	\$0.01	\$0.04	
	Ceramic Capacitor	3	\$0.10	\$0.30	
					\$10.78

## II.4 Class D Amplifier

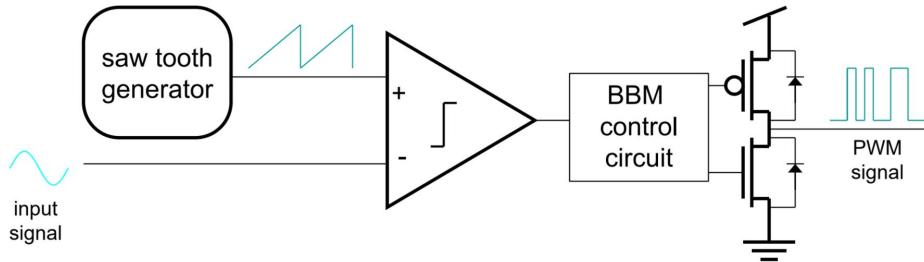
Next is the power amplifier, whose purpose is to provide the necessary current to drive a speaker. The power amplifier block diagram<sup>1</sup> is as follows.



The bulk of the components used come from the pulse width modulator, which has a block diagram<sup>2</sup> shown below.

<sup>1</sup>From "classD.pdf" by Prof. Kofi Odame

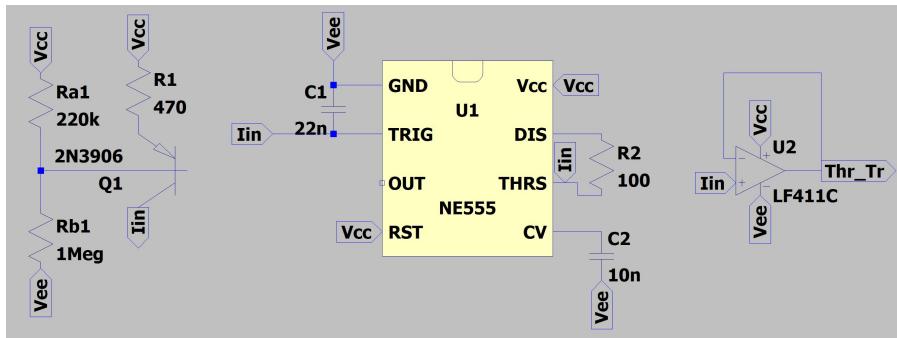
<sup>2</sup>Also from "classD.pdf"



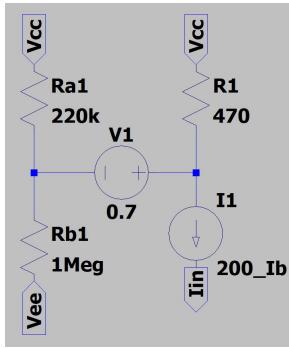
By feeding a comparator a high frequency saw tooth wave and a low frequency sine wave, we get a digital output whose localized time-average voltage is roughly sinusoidal. This digital signal goes to a break-before-make circuit that ensures the two transistors are never on at the same time. The two transistors then provide the current necessary to drive the speaker, after the high frequency and DC components have been removed from the signal.

#### II.4.1 Pulse Width Modulator

The sawtooth generator uses a PNP transistor and an LM555 to create an approximation of a sawtooth wave. The circuit is shown below.



The first portion of this circuit is a current source created with a PNP transistor. Assuming the transistor is in the forward active region, the circuit becomes the following:



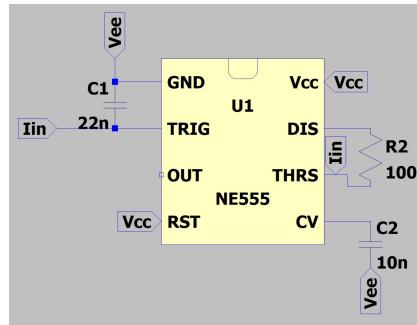
The transistor's current gain is  $h_{fe} = 200$ . If we let  $I_a$  be the current through  $R_{a1}$  and let  $I_{bias}$  be the current through the 0.7 V source, we can generate a  $2 \times 2$  system of equations:

$$(R_{b1})I_a + (201R_1 + R_{b1})I_{bias} = V_{cc} - 0.7$$

$$(R_{a1} + R_{b1})I_a + (R_{b1})I_{bias} = V_{cc}$$

Solving this system gives  $I_{bias} = 4.1 \mu\text{A}$ , so the dependent current source outputs  $820 \mu\text{A}$ . As long as the capacitor connected to the current source never charges up to within 0.2 V of  $V_{cc} - R_1(1 + h_{fe})I_{bias} = 4.46 \text{ V}$ , the transistor will remain in the forward active region. My process for actually picking these component values was to make the assumption that  $I_{bias}$  was small compared to the current in the voltage divider, which allows for me to make an estimate of an appropriate  $R_1$ . Then I was able to pick voltage resistors for the voltage divider and fine tune their values based on the fact that the presence  $I_{bias}$  actually does affect the voltage divider.

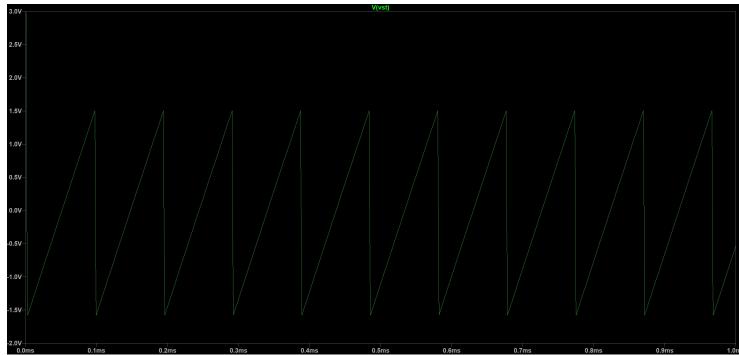
Next we consider the 555 portion of the sawtooth generator.



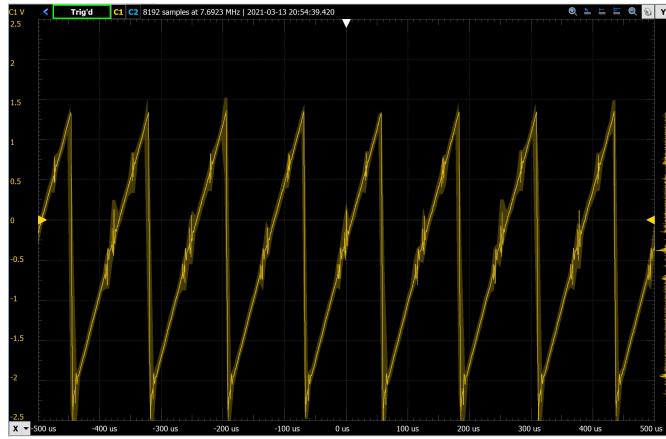
Again, notice that the 555 timer is powered from  $-4.5 \text{ V}$  to  $+4.5 \text{ V}$ . In this case, this is not only because the chip performs poorly with lower voltages across it, but also because the sawtooth wave generated goes from  $\frac{1}{3}V_{cc} = 1.5 \text{ V}$  to  $\frac{1}{3}V_{ee} = -1.5 \text{ V}$ , which is centered around zero. The capacitor  $C_1$  gets charged up by the current source, so  $\frac{dV_{trig}}{dt} = \frac{I_{in}}{C_1}$ . A 14 kHz sawtooth is about half a

decade above the frequency of the pulsed sine wave, so it should be an acceptable signal for the modulator. A 14 kHz sawtooth of amplitude 3 V has  $\frac{dV}{dt} = 42$  kV/s during its rise. Setting  $C_1 = 22$  nF achieved  $\frac{dV_{trig}}{dt} = 37$  kV/s which means the sawtooth frequency is about 12.3 kHz, which is close enough. The value for  $R_2$ , 100  $\Omega$ , was determined experimentally. Theoretically,  $R_2$  should just control the speed at which  $C_1$  discharges, but I found that the resistor's value actually had an effect on the final value of the RC decay. Namely, the lower the resistor was, the lower the bottom corner of the sawtooth wave went. In order to get the bottom corner up to the desired -1.5 V, I had to use a 3.3 k $\Omega$  resistor, which caused much too slow of an RC decay. I tried adjusting the values of the capacitor, current source, and resistor in hopes of achieving a fast RC decay while maintaining the a 12 kHz signal from -1.5 V to 1.5 V, but was unsuccessful. The solution I ended up opting for was to create a sawtooth with the fastest decay at the cost of having its amplitude go from -2.5 V to 1.5 V. This meant that I would have to modify the sinusoidal voltage form in the PWM, which will be discussed more briefly. The buffer is of course included to prevent the sawtooth wave generator from being loaded by the next subsystem. It is the same topology as I have used numerous times before but its input signal is much higher frequency, so its bandwidth and slew rate had to be considered. As calculated earlier, the slope of the sawtooth rise is 37 kV/s, which is well within the LM411's allowable slew rate of 13 V/ $\mu$ s. While a sawtooth wave contains relatively high amplitude harmonics, its fundamental frequency is so far below the LF411's gain bandwidth product of 3 MHz that any attenuation of the upper harmonics will be inconsequential.

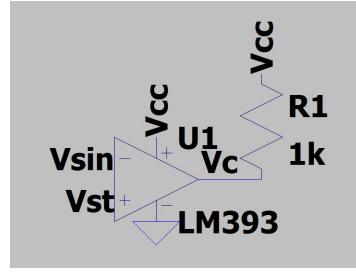
Below is shown an LTspice simulation of the sawtooth generator.



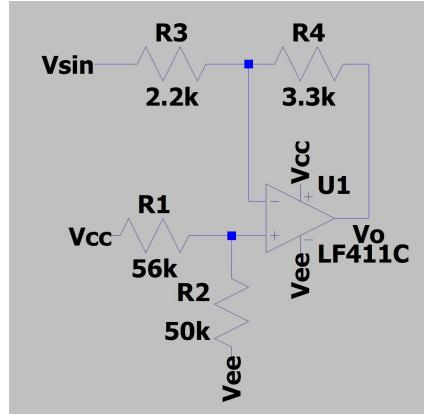
The LTspice simulation predicts that the sawtooth generator works exactly as I have designed it to, and the lowest voltage in the wave form was independent of the value of  $R_2$ . However, as with the other 555 timer circuit, there were unforeseen deviations between simulation and measured results. The diagram below shows the signal from the 555 timer when  $R_2 = 100 \Omega$ .



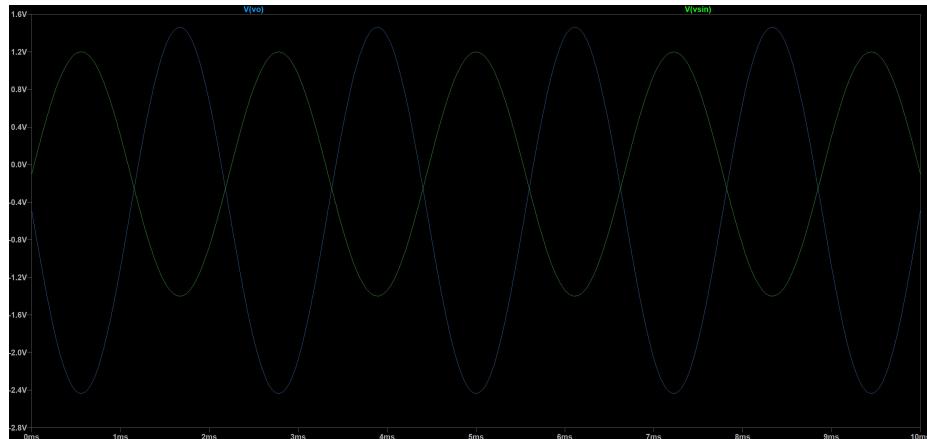
Next is in the pulse width modulator is the comparator.



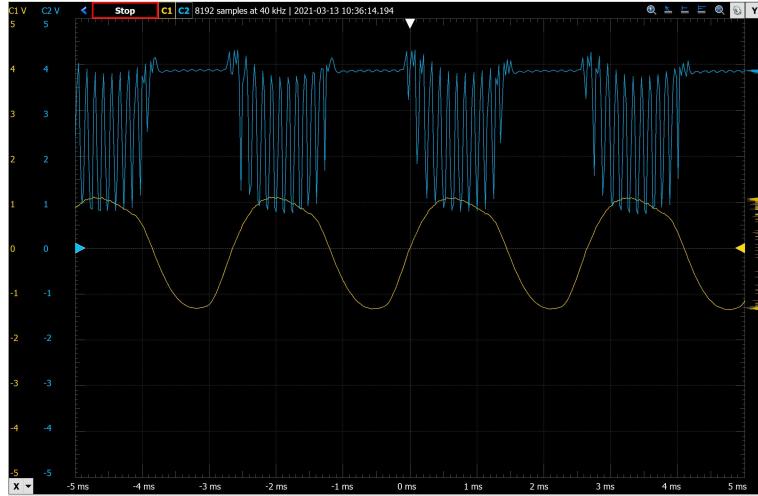
This very simple subsystem merely compares the low frequency sinusoid and the high frequency sawtooth wave. The pull up resistor used is  $1\text{ k}\Omega$  as in all previous comparators. However, the sinusoidal signal must be modified in order to work with the sawtooth wave. Specifically, I hoped to create a sinusoid that went from -2.5 V to 1.5 V. As discussed at the end of the Pulsed Sine Wave section, the sinusoidal approximation I created went from -1.4 V to 1.2 V. To get this sine wave to have the desired peak values, I needed to amplify it by approximately 1.5 V/V and add a DC offset of about -400 mV. The circuit below achieves this.



The gain of the circuit is exactly  $-1.5 \text{ V/V}$  and the voltage at the noninverting terminal is about  $-250 \text{ mV}$ , so this circuit should provide an additional  $-380 \text{ mV}$  of DC offset, creating a roughly sinusoidal waveform whose peaks are roughly the same as those of the sawtooth. The LTspice simulation below shows that acceptable peak values are achieved.

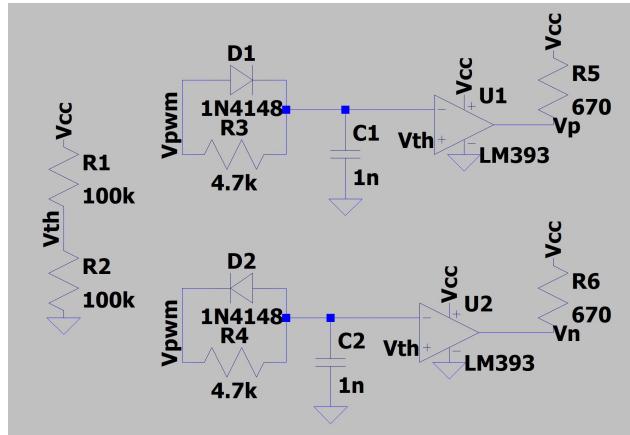


With this new sinusoid, the comparator should be able to function properly. The scope capture below shows the output of the comparator and the sinusoidal modulator signal.



While the signal is not pretty, it is good enough to operate the power amplifier with.

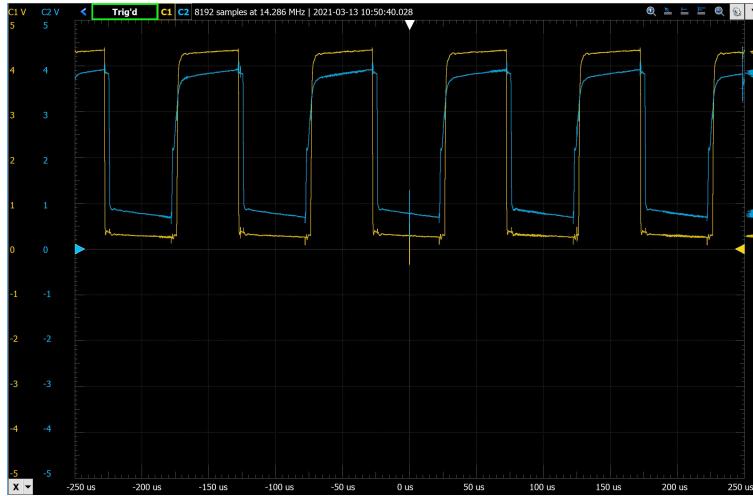
Next is the break-before-make (BBM). This circuit protects the power MOSFETs in the next section by making sure that only one of them is turned on at a time. It does so by outputting two digital waveforms,  $V_p$  and  $V_n$ . When the input to the BBM changes state,  $V_p$  falls before  $V_n$  does and  $V_n$  rises before  $V_p$  does.



When  $V_{pwm}$  goes from low to high, the upper comparator circuit can charge up its capacitor through  $D_1$ , which is forward biased. Since  $D_2$  is reverse biased, the lower comparator must charge its capacitor up through  $R_4$  though, so there is a much larger time before the lower comparator switches. Likewise, when  $V_{pwm}$  goes from high to low, the upper comparator must discharge its capacitor through the  $R_3$  while the lower comparator can discharge its capacitor through  $D_2$ , which is forward biased. Since the threshold of each comparator is half the

amplitude of the step input, the time delay is the half-life of the RC. As we have seen before,  $t_{1/2} = \ln(2)RC = 3.26 \mu\text{s}$ . This value was chosen because the smallest pulse widths I measured from the PWM circuit were about  $8 \mu\text{s}$  wide, and a delay of  $3.26 \mu\text{s}$  is close to the maximum delay which would still allow these pulses to happen at all. While it is only the product,  $RC$ , that controls the delay, in this particular case, it was important to make the resistor as large as possible. This is because even when the diode is on, it has some resistance, so some current would go through the parallel resistor. Using a large resistor and small capacitor for some fixed  $RC$  helps ensure that when the diode turns on it acts as much like a short circuit as possible compared to the resistor. While I have used  $1 \text{ k}\Omega$  for all other pull-up resistors, I used  $670 \Omega$  in this case. I did this because I wanted to be extra sure that the  $RC$  time constant for the charge up of the gate of the FETs was as small as possible. The gate capacitance of the transistors is roughly  $700 \text{ pF}$ , so the  $RC$  time constant with a  $670 \Omega$  resistor is about  $0.5 \mu\text{s}$ , which is certainly safe, since the pulses are about  $8 \mu\text{s}$  long at minimum. The  $670 \Omega$  resistors will create sink currents of  $6.7 \text{ mA}$ , which is safe given the LM393's maximum sink current of  $16 \text{ mA}$ .

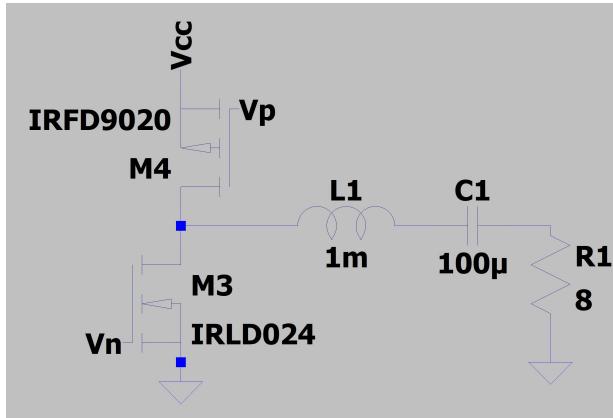
To test that the BBM circuit was working I fed it a  $10 \text{ kHz}$  square wave and measured  $V_p$ , yellow, and  $V_n$ , blue.



There is a noticeable delay at both the rising and falling edge, so the BBM circuit is working. We do notice that the  $V_n$  comparator is not going all the way low or all the way. Unfortunately, nothing can be done about this, since it is not a problem with the circuit design but with the comparator itself.

#### II.4.2 Power Amplifier

The power transistors provide high current to drive the speaker, which can be modeled as an  $8 \Omega$  resistance. A capacitor and inductor filter out the DC offset and high frequency portion of the signal.



The RLC bandpass filter has transfer function

$$H(s) = \frac{RCs}{LCs^2 + RCs + 1}$$

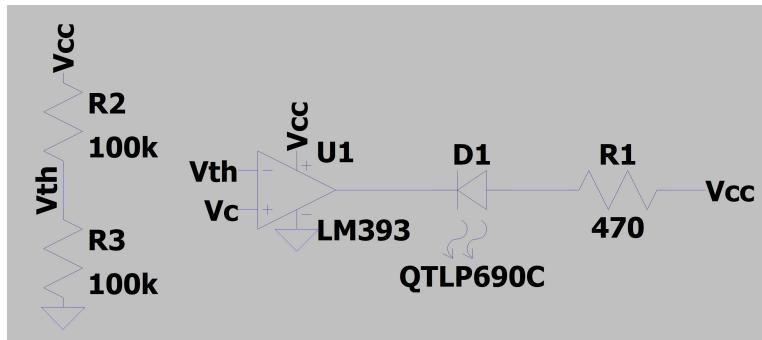
The resistance is of course fixed, since it is the speaker. To choose values for L and C, I made the approximation that the lower passband corner frequency occurs at  $\frac{1}{2\pi RC}$  and that the upper passband corner frequency occurs at  $\frac{R}{2\pi L}$ . We want to pass only the 450 Hz signal, so placing the lower pole at about 200 Hz and the upper pole at about 1 kHz should be sufficient, let  $L = 1$  mH and let  $C = 100 \mu\text{F}$ . The actual poles with these component values occur at 250 Hz and 1 kHz. The magnitude of this transfer function evaluated at  $s = j\omega = j2\pi(482 \text{ Hz})$  is 0.33, which should produce a sine wave of roughly 1.5 V amplitude, and provide the speaker about 140 mW of power assuming a RMS speaker voltage of  $1.5/\sqrt{2}$ . The speaker is rated for 3 W, so this should be safe.

The bill of parts for the Class D Amplifier system is included below, organized by subsystem, in order of place in the signal chain.

Subsystem	Part	Quantity	Price/item	Total	System Total
Sine Modifier	LF411CP Amplifier	1	\$1.84	\$1.84	\$11.07
	Resistor	4	\$0.01	\$0.04	
PWM	LM555CN Timer	1	\$1.17	\$1.17	\$11.07
	Ceramic Capacitor	1	\$0.10	\$0.10	
	Resistor	4	\$0.01	\$0.04	
	Film Capacitor	1	\$0.55	\$0.55	
	2N3906 PNP transistor	1	\$0.13	\$0.13	
BBM	LM393AP Comparator	1	\$0.40	\$0.40	\$11.07
	1N4148 Diode	2	\$0.04	\$0.08	
	Resistor	6	\$0.01	\$0.06	
	Film Capacitor	2	\$0.55	\$1.10	
Power Amplifier	LM393AP Comparator	1	\$0.40	\$0.40	\$11.07
	IRLD024 Power nMOS	1	\$0.99	\$0.99	
	IRFD9020 Power pMOS	1	\$1.29	\$1.29	
	Inductor	1	\$0.31	\$0.31	
	Aluminum Electrolytic Capacitor	1	\$0.05	\$0.05	
	Speaker 8 Ohm 1 W	1	\$2.52	\$2.52	\$11.07

## II.5 LED

The goal of this subsystem is simple: turn on an LED simultaneously with the speaker. To do so, I have used a comparator which operates off of a digital control signal.



The digital signal in question,  $V_c$ , is the same signal that controls the gate of the MOSFET in Signal Processing which lets through or blocks the  $V_{osc}$  signal.  $V_c$  goes low to turn on a sound burst, and when  $V_c$  is low, the output of the LED comparator is also low, allowing current to flow to ground through the LED. Assuming a voltage drop of 2 V across the LED, the  $470 \Omega$  resistor allows 5.3 mA to flow through the LED, which is enough to turn it on while still being a safe amount for the comparator to sink.

The bill of parts for the LED system is shown below.

Subsystem	Part	Quantity	Price/item	Total	System Total
LED	Resistor	6	\$0.01	\$0.06	
	LM393AP Comparator	1	\$0.40	\$0.40	
	LED	1	\$0.14	\$0.14	
					\$0.60

The bill of parts for the entire heart rate monitor is shown below.

Subsystem	Price
Mic + Preamp	\$20.96
Signal Processing	\$10.78
Class D Amplifier	\$11.07
LED	\$0.60
<b>Grand Total</b>	<b>\$43.41</b>

## III Experimental Results

### III.1 Techniques

Though very few techniques were used to empirically determine the functionality of the project, having multiple techniques available was key.

The first technique used was LTspice transient simulations. This was used for every system and nearly every subsystem. Since it takes much less time to construct and run an LTspice simulation than it does to build and test a physical circuit, a transient simulation was used in every location except those where the result was not worth testing (such as creating a threshold with a voltage divider). It was hugely advantageous to have this tool because it offered a very quick way to change component values and check results. For the most part, all the calculations in the design could be done by hand, but in some scenarios it was simply not practical given time constraints (such as determining the waveform of a lowpass filtered square wave), and in these cases, LTspice again proved to be a crucial resource.

While LTspice is an exceptional tool to validate that a circuit is ready for construction, the most important test was of course the measurement of the breadboarded circuit. Most of the time this involved using the oscilloscope to measure some voltage signal, though if I wanted to measure a current or a static voltage, I used the multimeter instead. For the most part, I measured the output of a subsystem when it was connected in the signal chain as it was supposed to be, but for some subsystems, like the BBM, I used the function generator instead. It wasn't until a subsystem passed these tests that I would move on to building or designing the next subsystem.

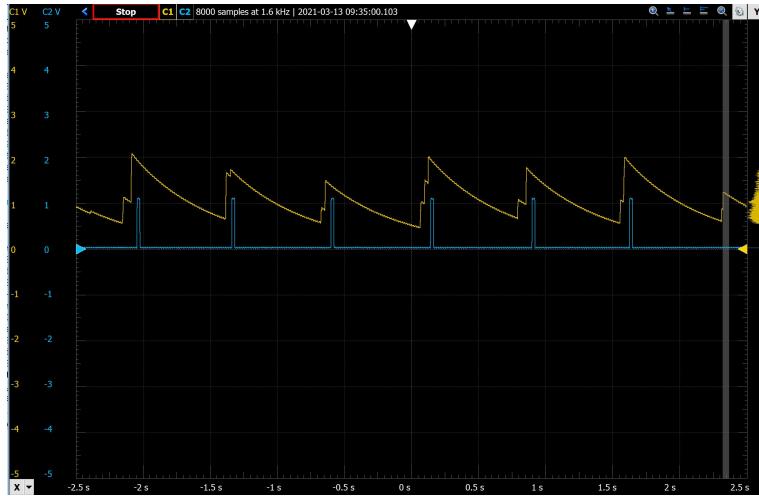
Another test that was not used as frequently but was very important was sensory tests. For instance, when I wanted to make sure my power amplifier was

working as expected, I listened to the speaker. When I wanted to make sure my LED comparator was working, I watched the LED. This was a better way of determining functionality than with the oscilloscope in these subsystems.

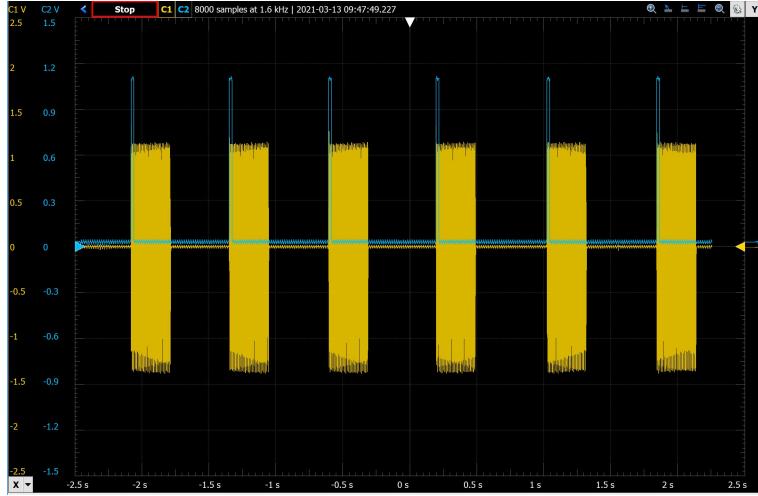
### III.2 Important Measurements

I have shown many scope captures and transient simulation results in my Technical Details section, so I will briefly describe some of the most important measurements I made to validate my predictions, then discuss some of the key ways that the experimental results differed from my predictions and how this effected my overall circuit design.

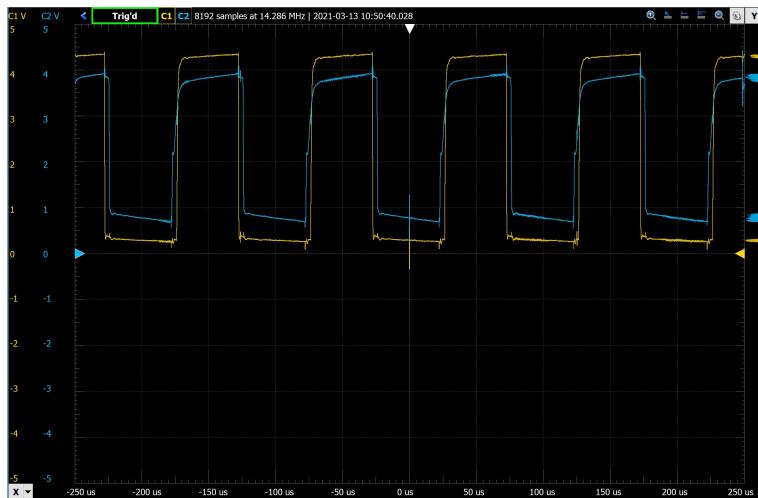
While I made many measurements in the signal chain prior to this one, the first very important measurement is the one that shows I have tuned the peak detector and comparators correctly such that the "dub" is ignored, and a short 1 V is created. The signal below shows that the "dub" from my heart is actually small enough to not even trigger the peak detector (yellow), and that only one impulse occurs per heartbeat (blue).



The next very important measurement shows that these short impulses are capable of creating longer tone bursts. The signal below shows that each short impulse (blue) triggers a tone burst (yellow).

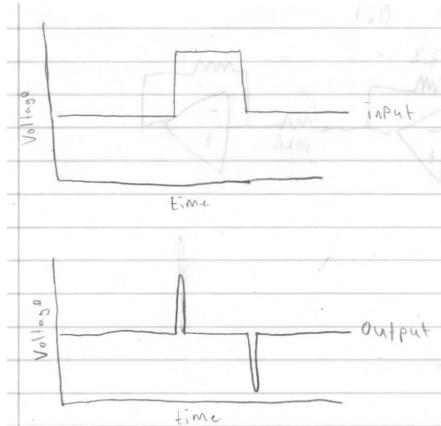


Perhaps the most important measurement I made was the one showing that the BBM circuit worked correctly. This was very important to confirm empirically because if it didn't work as expected, it could cause huge amounts of power to flow through the MOSFETs and melt them. While the scope capture below actually shows that the nMOS will be partially on even when it is supposed to be fully off, it also shows that there is enough delay between the rising edges and two falling edges to ensure that both transistors are never fully on at the same time.

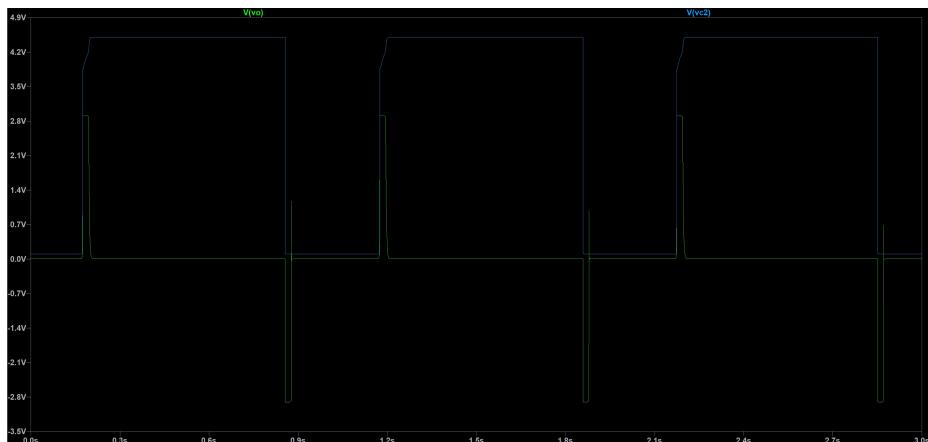


The first place that I encountered significant deviations between calculations, simulations and measurements was in my implementation of an edge detector using a differentiator. As discussed in Technical Details, rather than relying on the data sheet's switch time for the comparator, I decided to tune the values of

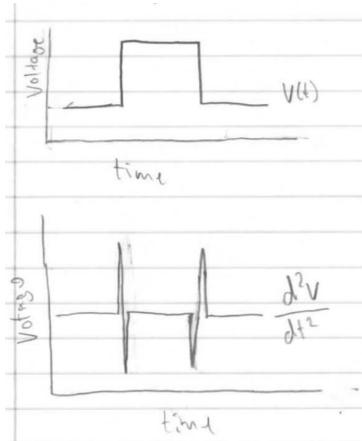
the differentiator experimentally so that I achieved sufficiently large pulses. I expected the output of the differentiator to look something like this:



But when I ran LTspice simulations, I found this output instead.

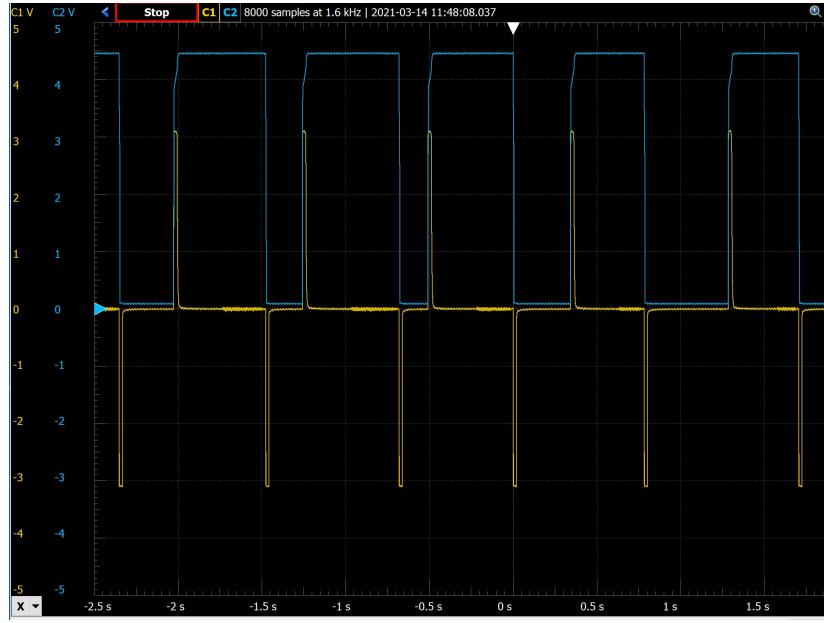


When the comparator signal falls, the output shows a negative impulse as expected, but it is followed by a small positive impulse. I believe the explanation for this is that somehow the differentiator has had second order effects introduced, such that it outputs becomes some linear combination of the derivative of the input and the second derivative of the input. The second derivative response to a square wave takes the following form:



While the first derivative has large magnitude at edges, the second derivative has large magnitude at corners. If the comparator signal was a more ideal square wave, we would expect to see a response more like the one sketched above. However, as we can see from the LTspice simulation, the filter actually loads the comparator and causes one of the corners of the rising edge to round out. This is why we see no visible second order effects on the rising edge. The falling edge, however, has sharp enough corners that we do see a small positive peak following the large negative peak.

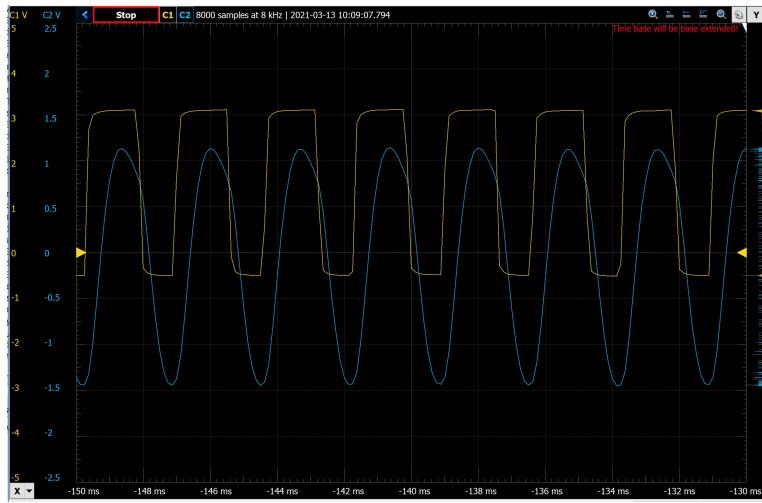
As discussed before, I had already determined the component values for the differentiator would be tuned experimentally, so now in addition to finding the component values which gave an appropriate impulse amplitude, I had to make sure they also minimized the second order effects which might cause unintended impulses with the opposite sign of the expected peaks. When testing different component values for this circuit, I actually did observe second order peaks up to 700 mV, but with the final component value combination,  $C = 2.2 \mu\text{F}$  and  $R = 10 \text{ k}\Omega$ , I was able to eliminate any second order effect completely as shown in the scope capture below.



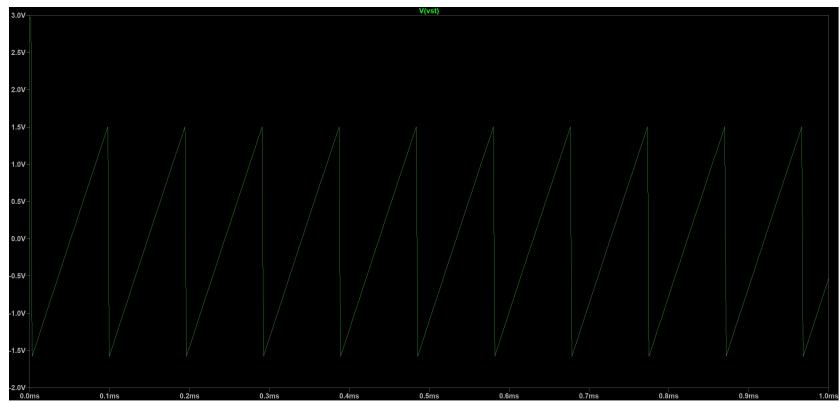
As discussed earlier, I fed the clipped version of this differentiator signal to a comparator, that way I could ensure I was getting a predictable output amplitude. This comparator ended up also being beneficial because I was able set my threshold high enough that I could be certain no second order peaks would be let through, if they began to reoccur for some reason.

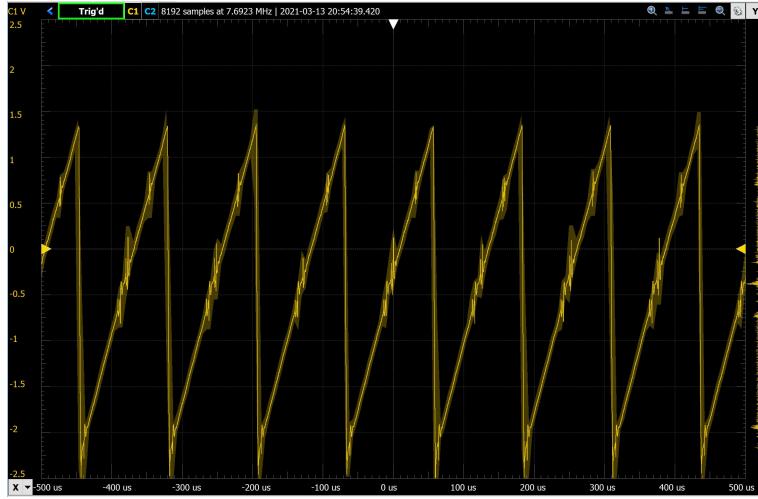
I also had to consider conflicting experimental results and theoretical predictions in both my circuits involving the 555 timer.

The first circuit involving the 555 timer was the 480 Hz oscillator. As discussed in the Technical Details section, my predictions showed a 50% duty cycle pulse from -4.5 V to +4.5 V, but I ended up seeing a duty cycle of significantly more than 50% and a high amplitude of only about 3 V. While this did not hugely effect the functionality of the circuit as a whole, it did cause my sinusoidal approximation to be slightly lopsided. As shown below, both the 555 output and the sine wave approximation are wider and shorter in the positive semicycle than in the negative semicycle.



As discussed in the Technical Details section, the 555 timer used in the sawtooth generator did not behave as predicted either. My understanding of the 555 timer and the LTspice simulation showed the sawtooth wave achieve its bottom corner at  $1/3V_{ee}$  regardless of what resistor was chosen to discharge the capacitor. However, I found that I could shift the low corner of the sawtooth wave by changing the value of the discharge resistor. The first plot below shows the result of an LTspice simulation with a  $100\ \Omega$  discharge resistor, and the second plot shows the measured result of the same circuit.





While LTspice was an exceptionally useful tool, ultimately, oscilloscope measurements were the paramount test.

## IV Discussion

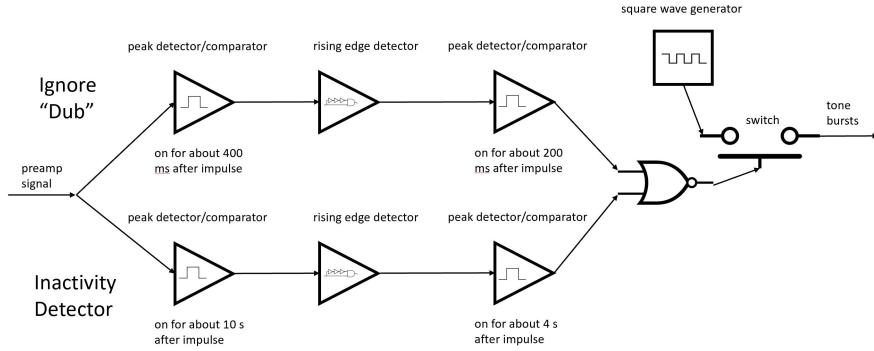
### IV.1 Missing Requirement

An additional requirement for the system was that the first beep after a long period of inactivity be a very long one. I did not satisfy this requirement because given my time constraints, it seemed like a risky proposition to modify any existing circuit elements. The solution I have come up with after the fact would actually add a number of blocks, but would only involve modifying one existing one.

The solution I would have used, Inactivity Detector, given a little more time is nearly identical to the Ignore "Dub" portion of the circuit. The Ignore "Dub" section works by ensuring that a comparator is high for both the "lub" and "dub" portion, but then off during a brief time of inactivity between heartbeats. The comparator turning on again triggers the rising edge detector, which then operates the pulsed sine wave. The solution I would implement would operate similarly, except that rather than having the comparator stay on for the "lub" and "dub" and then turn off between heartbeats, it would stay on for any number of repeated heartbeats and only turn off after several seconds of silence.

The rising edge in the original circuit triggers a peak detector/comparator combination that stays on for roughly 200 ms, the duration of a tone burst. The rising edge detector I would add in Inactivity Detector would trigger a peak detector/comparator combination that would stay on for a few seconds. The Ignore "Dub" section and the Inactivity Detector would run in parallel, and together control the switching mechanism to let through the oscillator signal. An abstracted block diagram for a revised Signal Processing subsystem is shown

below.



The NOR gate is the proper choice because its output is low if either Ignore "Dub" or Inactivity Detector are on, and the square wave is allowed to pass through the switching mechanism only when the nMOS gate is low.

## IV.2 Suggested Improvement

While my system was able to achieve the desired results over the heartbeat range of 60 bpm to 100 bpm, it would not have worked for heartbeats much faster than this. The rising edge detector I used in the Ignore "Dub" section of the Signal Processing subsystem relied on the peak detector/comparator combination resetting about 700 ms after every heartbeat. But when the heartbeats occur more frequently than every 700 ms, the comparator will not reset, and hence the rising edge detector will not trigger. My solution was acceptable in an academic scenario, but for an actual application, the system would have to be more robust. This would likely involve significantly reducing the time the peak detectors were on and introducing a flip-flop to ignore the "dub".

## IV.3 Conclusion

I have worked on several circuit design projects in the past, but none of this magnitude. In all the smaller projects, I did not prioritize breadboard neatness. In this project, though, I was cautious and deliberate as I breadboarded my circuit. Despite the large number of discrete components and wires, the amount of time I had to spend debugging my circuit was much smaller in this project than it has been for any other project. Clearly, breadboard neatness pays off. Also, because of the magnitude of this project, I was much more organized creating a file structure for my LTspice simulations, MATLAB calculations, and Waveform workspaces. Additionally, I learned to use *.subckt* files in LTspice to help keep my circuit schematics much more intelligible, and read as block diagrams rather than hundreds of entangled components. My use of *.subckt* files proved extremely beneficial because I was able to reuse blocks in different

locations by just modifying parameter values. I expect I will be able to use these files in the future for academic and personal projects as well. Overall, I discovered that some of the organization and formatting that is unnecessary in small scale projects can save time in the long run on projects of a larger scale.