

μ Theremin

Final Design Review

Timothy Alexander, Chase Bourg, Greg Hurst, Subol Shrestha

11/29/2011

Table of Contents

1 Executive Summary	4
1 Functional Requirements.....	5
1.1 Portability.....	5
1.2 Usability	5
2 Sensor Stage.....	6
2.1 Design Requirements.....	6
2.2 Proposed Design.....	6
2.3 Testing and Verification	7
2.4 Design Changes	10
2.5 Outcomes	12
2.6 Lessons Learned	13
3 Mixing and Filtering Stage.....	14
3.1 Design Requirements.....	14
3.2 Proposed Design.....	16
3.3 Testing and Verification	21
3.4 Design Changes and Outcomes	24
3.5 Lessons Learned	28
4 Microprocessor Stage	29
4.1 Design Requirements.....	29
4.2 Proposed Design	29
4.3 Flow Control.....	31
4.4 Testing and Verification.....	32
4.6 Outcomes.....	37
4.7 Lessons Learned.....	37
5 Audio Amplifier Stage	38
5.1 Design Requirements.....	38
5.2 Proposed Design.....	38
5.3 Testing and Verification I	39
5.4 Design Changes	39
5.5 Testing and Verification II	40

5.6 Outcomes	44
5.7 Lessons Learned	44
6 Power Stage	45
6.1 Design Requirements.....	45
6.2 Proposed Design.....	45
6.3 Design Changes	46
6.4 Testing and Verification	46
6.5 Outcomes	49
6.6 Lessons Learned	49
7 Enclosure.....	50
8 Performance Outcomes	52
9 Printed Circuit Board	53
10 Design Economics	56
10.1 Budget Analysis	56
10.2 Reproducibility	58
2 Service Learning	59
3 Appendix	61

Table of Figures

Figure 1: Clapp Oscillator.....	7
Figure 2: Transient Response	7
Figure 3: Frequency Response.....	8
Figure 4: Clapp Oscillator Testing.....	8
Figure 5: Clapp Oscillator Waveform.....	9
Figure 6: Redesigned Oscillator Circuit Schematic.....	10
Figure 7: Redesigned Oscillator Results	10
Figure 8: CMOS Oscillator Circuit Diagram	11
Figure 9: CMOS Oscillator Testing	12
Figure 10: CMOS Test 1.....	13
Figure 11: CMOS Test 2.....	13
Figure 12: Frequency Mixer Behavioral Model.....	14
Figure 13: Frequency Modulation Illustration	14
Figure 14: Demodulation Behavioral Model	15
Figure 15: Single JFET Mixing Circuit Schematic.....	16
Figure 16: Single JFET Mixer Small Signal Model.....	16
Figure 17: Mixer Simulation Results. Inputs(Purple/Gold) and Output(Blue).....	17
Figure 18: Envelope Detector Circuit Schematic.....	18
Figure 19: Butterworth Low Pass Filter Circuit Schematic.....	19
Figure 20: Transient Simulation, VFO Output(150kHz) vs. Output(17kHz).....	19
Figure 21: Butterworth Filter Bode Plot (Frequency Response).....	20
Figure 22: Clamping Circuit Schematic	20
Figure 23: Envelope Detector, Filter, and Clamping Circuit Schematics.....	21
Figure 24: Mixer/Envelope Detector/Filter/Clamper Construction	22
Figure 25: Mixing Circuit Simulation Purple,Gold = Inputs; Blue = Output.....	22
Figure 26: Inputs/Outputs of Mixing Stage (a) Vref, (b) Vvar, (c) Vmod	23
Figure 27: Filter Simulation	23
Figure 28: Filter Results	23
Figure 29: Mixer Input Signals and Envelope Oscillator Result.....	24
Figure 30: Low Pass Filter Circuit Schematic	25
Figure 31: Results after Filter	25
Figure 32: DC Level Shifter with Gain Modification	26
Figure 33: Whole Mixer and Filter Schematic	26
Figure 34: Output Signal of Mixing/Filtering Stage	27
Figure 35: Soldering Mixing/Filtering Circuit.....	27
Figure 36: Results of Soldered Circuit	28
Figure 37: Software Flow Control	31
Figure 38: DAC Output.....	34
Figure 39: Time Domain Detection.....	36
Figure 40: Proposed Amplifier Circuit	38
Figure 41: Audio Output	39
Figure 42: Revised Amplifier Circuit.....	40
Figure 44: Amplifier Noise Output.....	40
Figure 45: Soldered Audio Amp.....	41

<i>Figure 46: Voltage Offset Shifter Circuit</i>	42
<i>Figure 47: Hidden Voltage Shifter</i>	43
<i>Figure 48: Proposed Power Circuit</i>	45
<i>Figure 49: Revised Power Circuit</i>	46
<i>Figure 50: Power Circuit Tested Results</i>	47
<i>Figure 51: Charge Pump Circuit</i>	47
<i>Figure 52: Soldered Power Circuit</i>	48
<i>Figure 53: Front Face Logo</i>	50
<i>Figure 54: Back Grill Logo</i>	50
<i>Figure 55: Top Face Logo</i>	51
<i>Figure 56: Completed Box</i>	51
<i>Figure 57: Mixer Schematic</i>	53
<i>Figure 58: PCB Layout</i>	54
<i>Figure 59: Diode Pad Layout</i>	55
<i>Figure 60: Robot Arm Competition Playing Field</i>	59

1 Executive Summary

The project described herein was undertaken in response to a solicitation calling LSU Engineering students to promote their discipline and field of study in High School classrooms. There was no specific roadmap for tackling this challenge and the actual implementation was left entirely up to the students.

When considering this project, much thought was put into what type of material would be appropriate for inciting curiosity in students. After researching this, the determination was made that a presentation incorporating various principles of electromagnetics and electrical engineering principles would be the most realistic way to generate interest in a classroom. Continuing along with this idea, having a project that was not only interesting by its theory of operation but also created with the fundamental principle of direct interaction with student would be the best solution to accomplishing these project goals.

To satisfy the demands of the project, a Theremin device was considered as a solution. This is an electronic instrument originally invented at the dawn of the 20th century as a military proximity sensor. Leon Theremin took this design and created an instrument that is controlled strictly by the position of the player's hands. The player's hands contribute a capacitance to the input oscillator circuit, and the capacitance created varies the frequencies that are generated and carried throughout the system.

Theremins have come from early vacuum tube incarnations crossing over into the solid state era with transistor iterations using digital signals. For this project several different types of technologies were used in order to increase the educational potential of the device as well as the difficulty of the design. The system that was created uses both electronic components and a microcontroller that regulates all the action taking place within the circuits. The electronic systems incorporate the principles of signal generation, modulation, demodulation, and filtering. The microcontroller will gather its inputs from the electronic circuits and be able to distinguish the frequencies that are being read in. Based on these varying frequencies, the microcontroller will define each combination of input frequencies as a certain musical note. Once that task is performed, the microcontroller will send these notes to an audio amplifier, which is attached to a speaker that will output the resulting sound.

1 Functional Requirements

1.1 Portability

Because the Theremin is the cornerstone of presentations, the overall weight and dimension of the device and its components need to be easily portable by the presenter. In order to have measurable restrictions the entirety of the device and its accessories was designed to fit in a Bankers Box and weigh no more than 5 pounds. The majority of the weight of the device comes from the acrylic case, as the circuit board is extremely light.

1.2 Usability

Because one of the main goals of the project is to generate interest from high school students, the apparatus should be highly interactive. The Theremin as an instrument needs to be playable not only by the instructor but by any students that would use the device as part of the presentation. Because of this need, it is helpful to have a player's manual that can be carried along with the Theremin. This manual will instruct the player how the Theremin operates, and will have minor instructions on how to adjust some of the Theremin components if the need arises.

2 Sensor Stage

2.1 Design Requirements

The sensor stage of the Theremin is the only stage of the Theremin that the user can manipulate. The purpose of the sensor stage in the Theremin is to receive the signal from antenna and pass it to the mixer circuit. Therefore sensor stage is the most essential stage of the Theremin as it is the only way we can get the signal required for the Theremin.

So, we need to design an oscillator for our sensor stage that could detect and receive signal from antenna at various range and transmit it to the mixing stage. Moreover we need to design an oscillator that can perform on various frequencies and produce periodic output voltage signals such as sine or square waves.

2.2 Proposed Design

Initially Clapp Oscillator was chosen, as it is a better variable frequency oscillator. These oscillators are constructed from a transistor and a positive feedback network using the combination of an inductor with a capacitor to determine frequency. Therefore, it is also called LC oscillator. Clapp oscillator provides better stability to the system with variable frequency. Plus, oscillation can be achieved over a period of desired range.

To build a Clapp oscillator for the project, different parameters were assumed that would remain constant in our circuit. First of all, desired center frequency (f) of 320 kHz was assumed. Apart from that, the total capacitance of the circuit was chosen to be 450 pF. Also output voltage was fixed to 3V.

To calculate the inductance of the inductor (L), resonating frequency formula was used as follows,

$$\omega = \frac{1}{\sqrt{LC_T}}$$

$$L = 549.7 \mu H$$

The capacitance for the capacitor (C_0) which is in series with the inductor was assumed to be of 30 pF. Then the coil loss (R_s) was calculated using the quality factor equation. And the quality factor (Q_0) was assumed to be 200. The ideal value of the quality factor for antenna is usually in the range of 200 to 300.

$$Q_0 = \frac{X_L}{R_s}$$

$$R_s = 77.34 \Omega$$

At resonance, the source terminal of the MOSFET has the voltage

$$V_s = Ri_d$$

Also,

$$V_{gs} = \frac{C_1}{C_2} V_s = \frac{C_1}{C_2} Ri_d$$

And

$$L = \frac{i_d}{V_{gs}} = \frac{C_1}{R_s C_2}$$

$$g_m > \frac{C_1}{R_s C_2}$$

Then,

$$\frac{C_1}{C_2} = LR_s = 42513.7 \times 10^{-6}$$

$$\frac{g_m}{\omega_0^2 R_s C_1 C_2} > 1$$

Therefore

$$C_2 = 6.4158nF$$

$$C_1 = 5.93nF$$

2.3 Testing and Verification

Based on the above information, the circuit was built in pspice to check whether the circuit can perform as expected or not. Since the components values did not match with the components that were available in Digi-key's inventory, components with close values were used. The pspice simulation can be seen below.

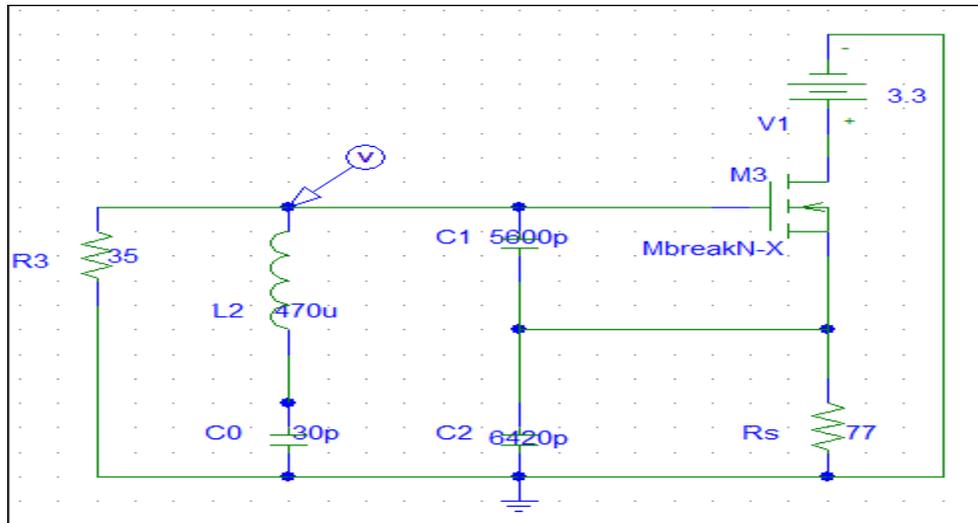


Figure 1: Clapp Oscillator

The output responses from the pspice simulation can be seen below.

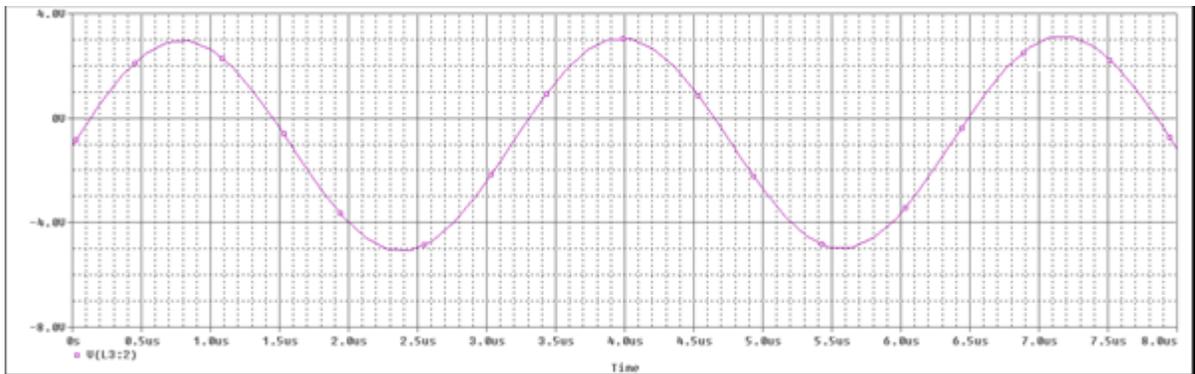


Figure 2: Transient Response

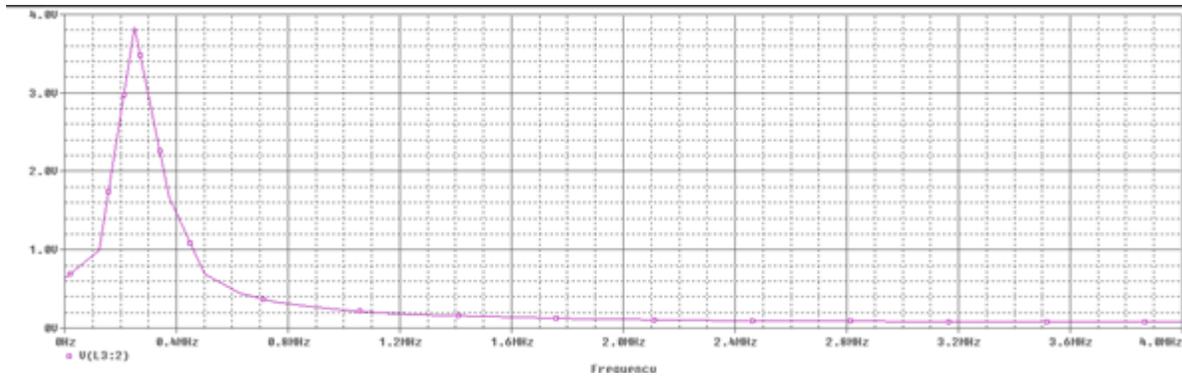


Figure 3: Frequency Response

Once the circuit was simulated with satisfactory results, the parts were ordered from Digi-key and the Circuit was built. The oscillator was constructed in the ERAD 326 Lab, as seen in Fig 4. Power was supplied by a DC Power Supply and measurements were taken with an Oscilloscope. A Solderless Breadboard was used to verify the design before permanently constructing the circuit. A bare piece of solid 24 AWG wire was used as a simple antenna.

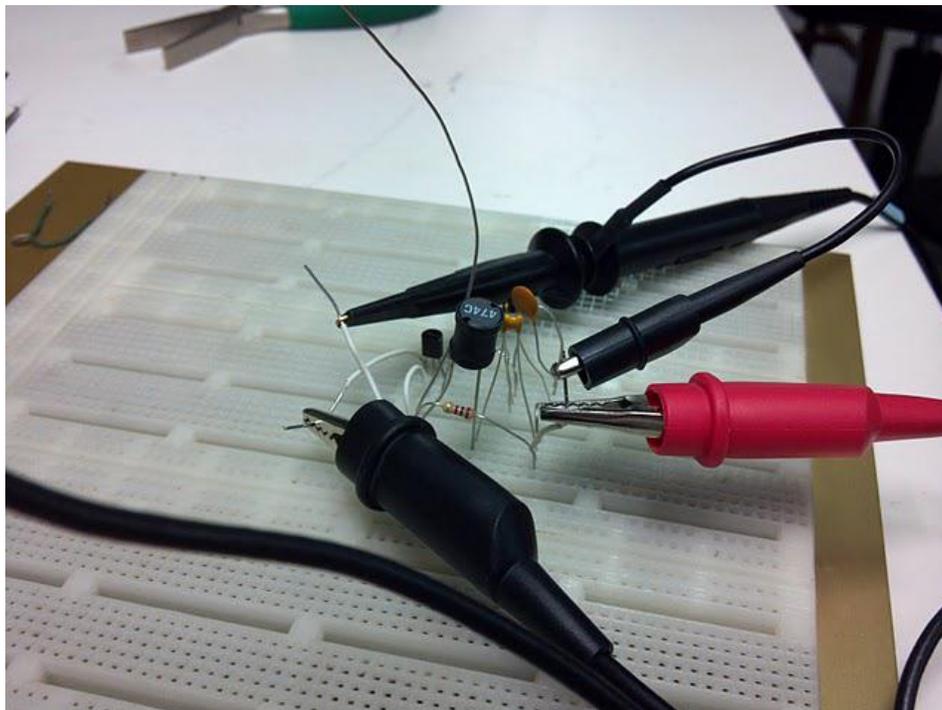


Figure 4: Clapp Oscillator Testing

This circuit did not perform as expected. Despite troubleshooting the Oscillator multiple times, the oscillator was still unstable and the Oscilloscope was unable to get a steady reading on it. There was response in the form of noise from hand position near the antenna but it was larger than the chaotic signal that was being viewed on the scope. A screenshot of the waveform is shown in Figure 5.



Figure 5: Clapp Oscillator Waveform

Many attempts were made to construct this circuit successfully. The original design specification was centered on achieving oscillation conditions within the detection window of the ADC. Believing such an approach to be at fault the circuit calculations were redone with the goal of finding the appropriate frequency of oscillation. Despite these efforts no appreciable results came to fruition.

In order to get over this impasse, assistance was sought from Dr. Feldman. Suggestions given to improve the design were:

- Increase Capacitance or Inductance to drive frequency higher
- Configure the amplifier circuit with higher gain in order to fix the amplitude issues

Using this guidance the circuit was yet again reconfigured. The capacitor values were increased by several orders of magnitude. Oscillations were achieved on the order of several kilohertz. Despite this the amplitude of the oscillations was a few hundred millivolts, which allowed noise ruin any signal fidelity.

2.4 Design Changes

Two approaches were taken to solve the issue with the oscillator. First, the design that was worked on last semester was reviewed. A second type of oscillator using Hysteresis is also being considered. While testing the updated design, a new prototype was made that had better results than the original design. This modified circuit is shown below.

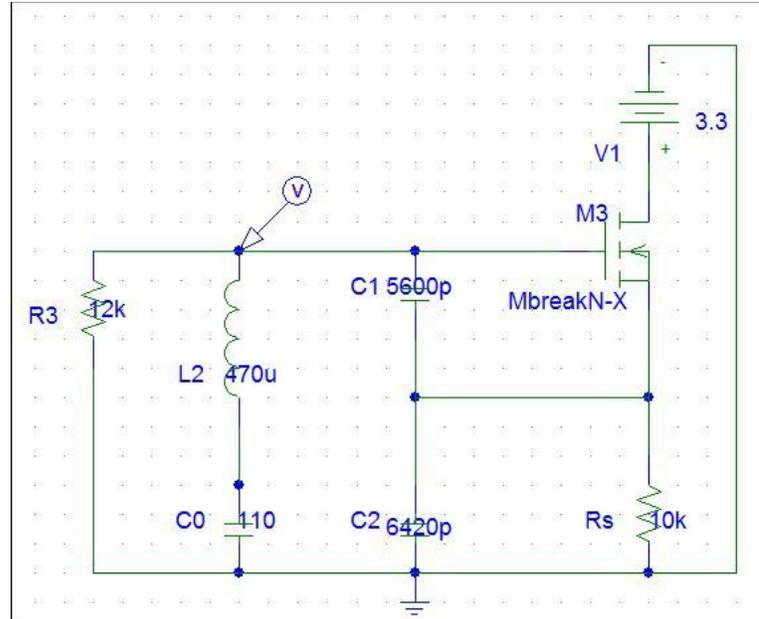


Figure 6: Redesigned Oscillator Circuit Schematic

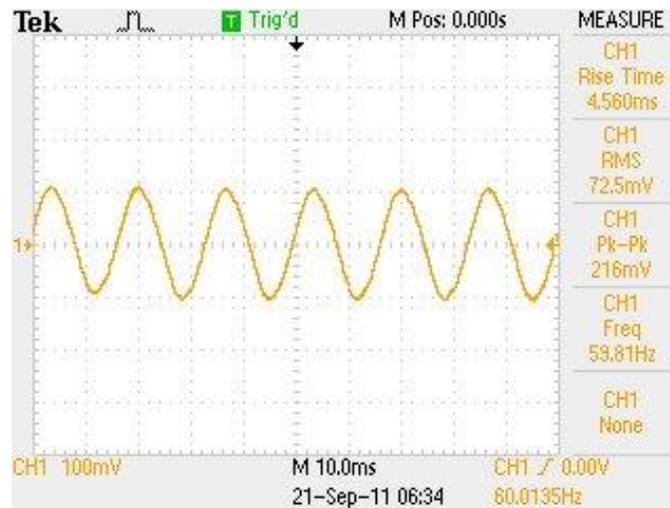


Figure 7: Redesigned Oscillator Results

This design was assembled on solderless breadboard and tested using the equipment in ERAD 326. The results of the test are shown in the figure above. This circuit produced a stable waveform but the frequency was well below the desired value and the circuit did not show any reaction to hand position when the antenna was attached.

As described in the preceding section the amount of effort and time that was being devoted to the LC oscillator was becoming a burden on the manufacturing effort as a whole. Because the Oscillators are the cornerstone of the entire Theremin a proven design using digital CMOS technology chips to create square waves was used.

The basis for the oscillation of this chip is known as Hysteresis, plainly when the input voltage of the chip reaches a certain threshold of the Supply the output will switch states between Supply and Ground Voltages.

After several attempts it became obvious that the time spent working on this design was eating into other areas of the project and a decision was made by the team to scrap the design and use a different method. The one that was chosen is a CMOS Oscillator, which operates using a NAND Schmitt trigger and an RC feedback network to set the frequency.

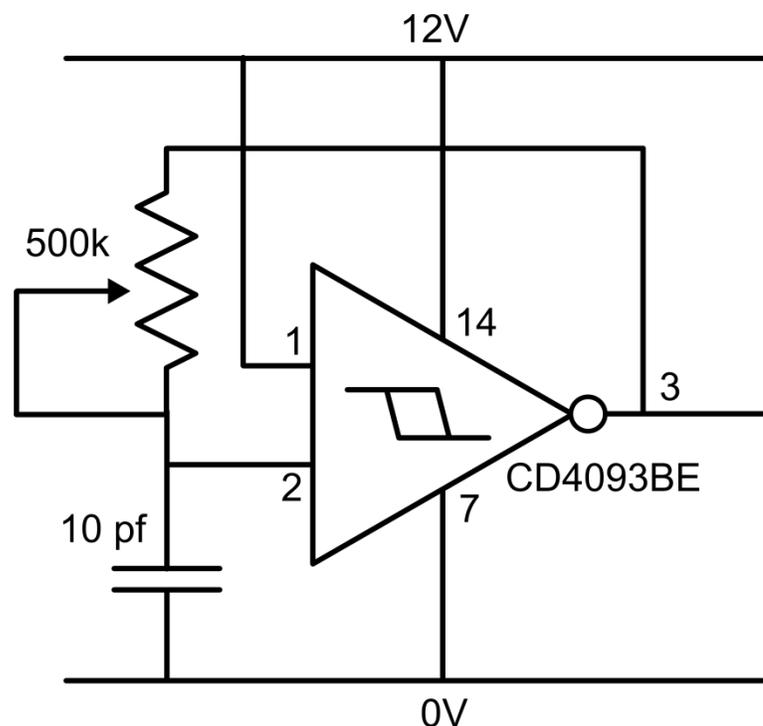


Figure 8: CMOS Oscillator Circuit Diagram

The oscillator was constructed in the ERAD 326 Lab. Power was supplied by a DC Power Supply and measurements were taken with a Oscilloscope. A Solderless Breadboard was used to verify the design before permanently constructing the circuit. A bare piece of solid 24AWG wire, and then a Metal Plate was used as an Antenna. The results of the testing with component values are below

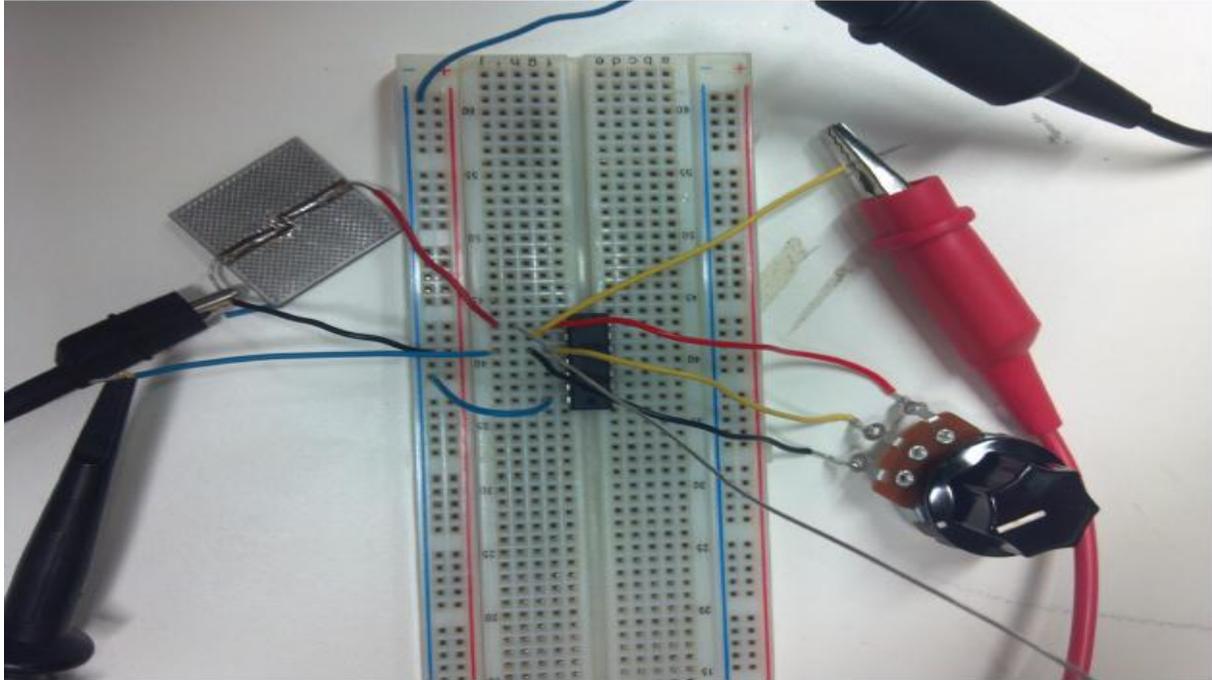


Figure 9: CMOS Oscillator Testing

Based on the CM4039 datasheet, the formula for determining the center frequency of the oscillator is

$$f = \frac{1.2}{RC}$$

However it was noted that constructing a circuit with different values of R and C trying to obtain the same center frequency was not always successful. Choosing the 10pF for the capacitor was done so that hand capacitance could have the greatest impact on the center frequency. Because of variation in parts using a potentiometer to set the center frequency precisely was useful and incorporating a precise multi-turn potentiometer into the final design for the variable and fixed oscillators will make tuning the device possible.

Vin(V)	C(pF)	R(kΩ)	f(kHz)	Δf(kHz)
12	10	125	400	30
12	10	125	200	15

2.5 Outcomes

The most important observation of testing on this circuit was the range of the frequency variation as the center frequency of the oscillator increased. This gives some prospect for making the device more responsive. However even with the plate antenna the distance required to have the smallest effect on the center frequency was about 20cm.

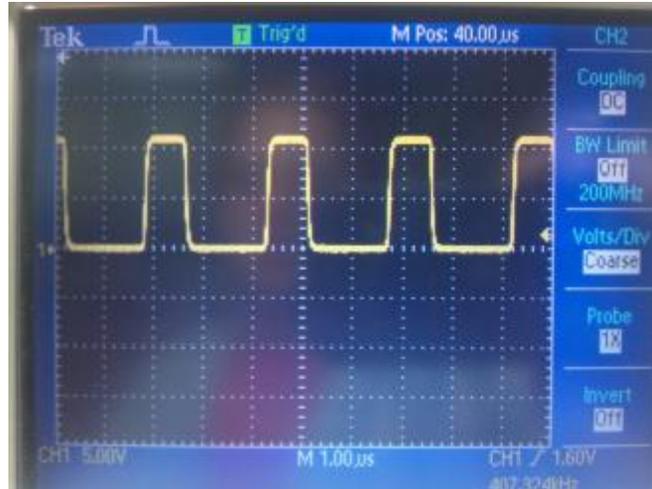


Figure 10: CMOS Test 1

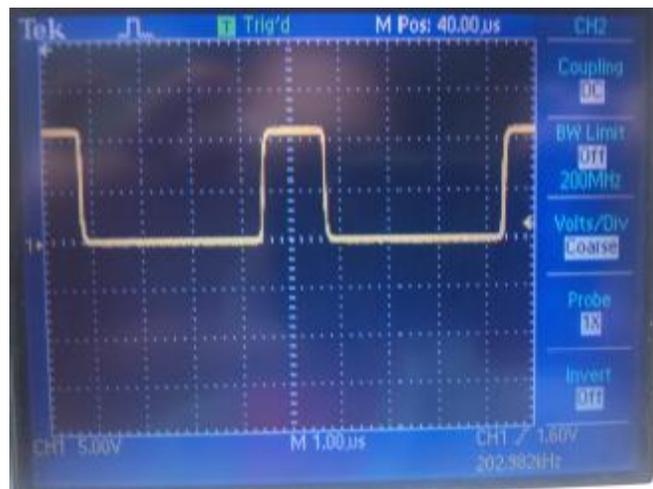


Figure 11: CMOS Test 2

The CMOS oscillator design was much more responsive to the requirements of the Theremin, so therefore, the original design that used the Clapp oscillator was scrapped.

2.6 Lessons Learned

For this stage of the design not enough credence was given to the difficulty in simulation of the advanced circuits.

3 Mixing and Filtering Stage

3.1 Design Requirements

The purpose of the mixing stage of the Theremin is to heterodyne, or mix, signals that are being produced by the variable oscillator and the reference oscillator. This will have to be done for both the tone dual oscillator circuit and the octave dual oscillator circuit. To do this procedure, the two input signals from each dual oscillator circuit must be multiplied into each other, thus giving one single output signal for the tone circuit, and one output signal for the octave circuit. During this process, the frequencies will be multiplied together.

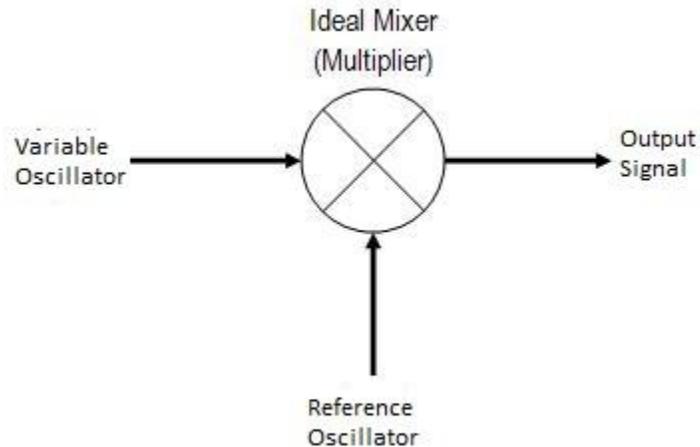


Figure 12: Frequency Mixer Behavioral Model

The relationship for this procedure is shown with the equation:

$$A\sin(x)B\sin(y) = \frac{1}{2}AB[\cos(x - y) - \cos(x + y)]$$

where A and B are the gain of each of the input signals, and x and y are their respective frequencies. The output modulated signal is basically a combination of the sum, the difference, and both input frequencies of the input signals.

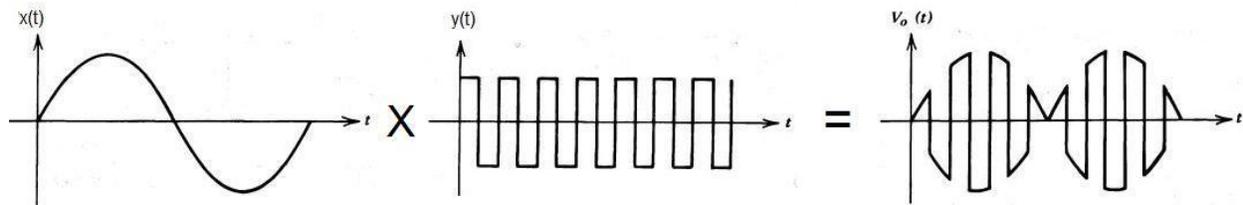


Figure 13: Frequency Modulation Illustration

Working under the consideration that the input signal $y(t)$ is the reference oscillator signal, it is clear that the modulated (multiplied) output signal should be a mixture of the amplitudes and frequencies of $y(t)$ and the variable oscillator, $x(t)$.

Because the reference oscillators only exist to serve as a reference to any input signals the Theremin may see, it does not contain any important input information. Therefore, the modulated signals produced by the mixing stage will contain much information that was supplied by the reference oscillators that is not pertinent to the creation of the musical notes. Because of this, it is beneficial to remove the contents of the modulated signal that belong to the reference oscillators so that going forward, only the necessary information that is contained by the variable oscillator is carried on. To do this, it will then be necessary to extract the difference between the frequencies, $x - y$, from the modulated signal, which contains the input information carried by the variable oscillators' signals. The unnecessary reference oscillator information is contained within the $x + y$ frequency, so that portion of the modulated signal will be eliminated by passing the signal through an envelope detector that demodulates the signal.

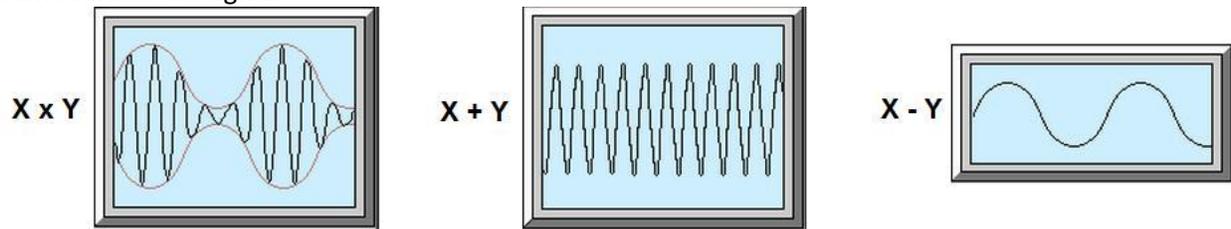


Figure 14: Demodulation Behavioral Model

Once the difference between the two frequencies is extracted, it is also beneficial to eliminate all of the frequencies that lie outside of the average human range of hearing, which tends to be from 20 Hz to 20 kHz. While a band pass filter could be applied to eliminate all frequencies below 20 Hz and all frequencies above 20 kHz, the frequencies below 20 Hz will not affect the signal as much as the frequencies above 20 kHz will. So in this case, a low pass filter with a cutoff frequency at 20 kHz will be utilized. Also, the input signals for the upcoming microprocessor stage must be a 0-3.3 V sinusoidal signal, based on the specifications of the chip. To do this, some sort of DC-level shifting and amplification will most likely need to be done.

3.2 Proposed Design

There are several ways to perform this heterodyning stage of the Theremin circuit. Mixers can consist of any combination of diodes, operational amplifiers, transformers, bipolar junction transistors (BJTs), field effect transistors (FETs), and other basic circuit parts, such as resistors, capacitors, and inductors. When choosing design one aspect that had to be kept in mind was size. Because of that it was better to eliminate the idea of using transformers in the mixer design. After much research, it was discovered that there were several effective BJT and FET mixers. Wanting to satisfy the requirement of using little power throughout the circuit, a single n-channel JFET mixer was chosen. The input signals, from each of the reference oscillators and variable oscillators, will be fed into the circuit. When the signals meet at the common node before the n-channel JFET, they are mixed, and once they are sent through the JFET, they are multiplied. Another capacitor is added at the output of the mixer to serve as a DC-level shifter to have a more manageable signal. Using LTSpice, the JFET mixer circuit was simulated using two expected input signals.

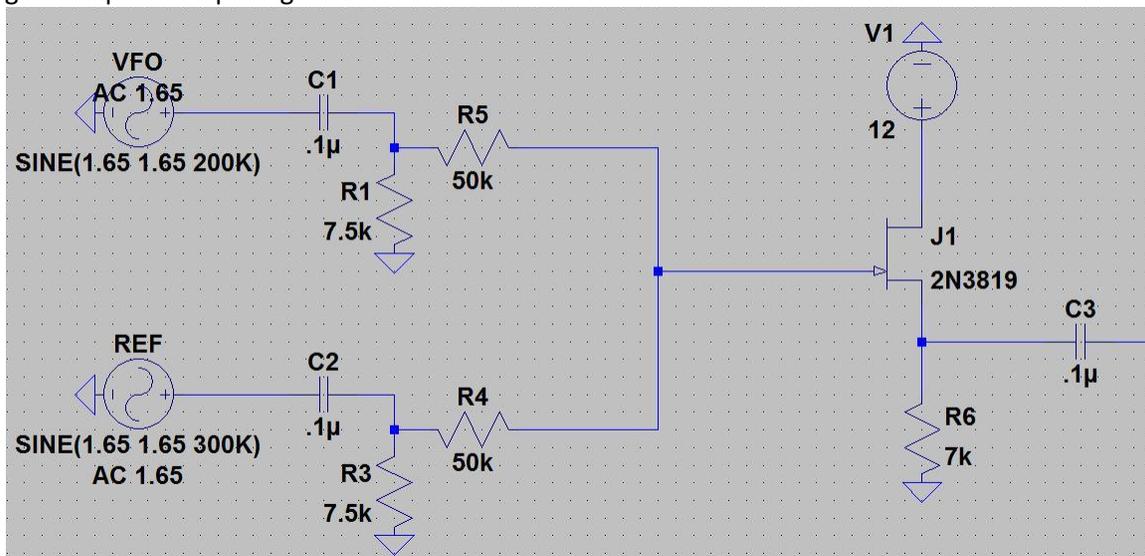


Figure 15: Single JFET Mixing Circuit Schematic

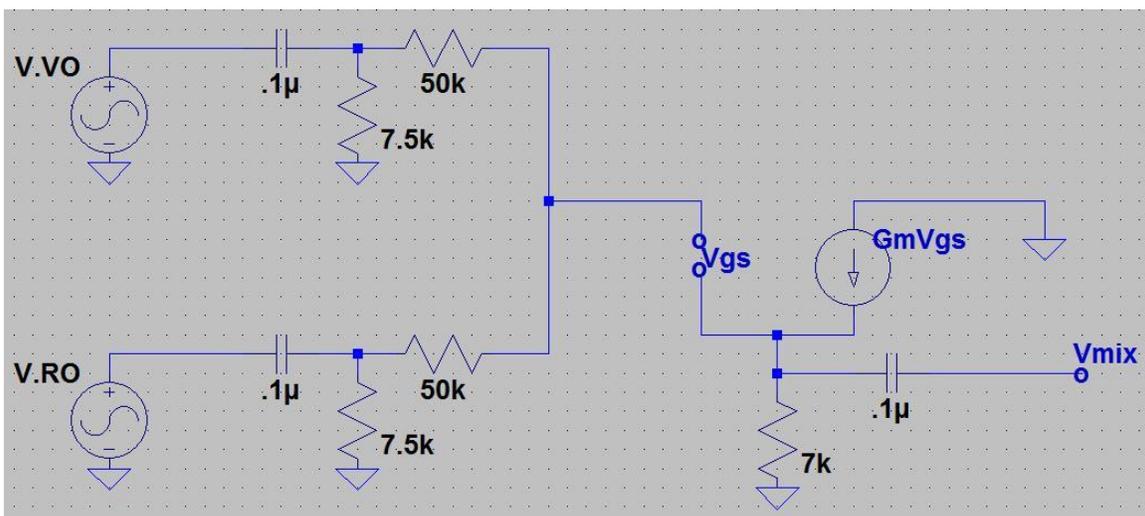


Figure 16: Single JFET Mixer Small Signal Model

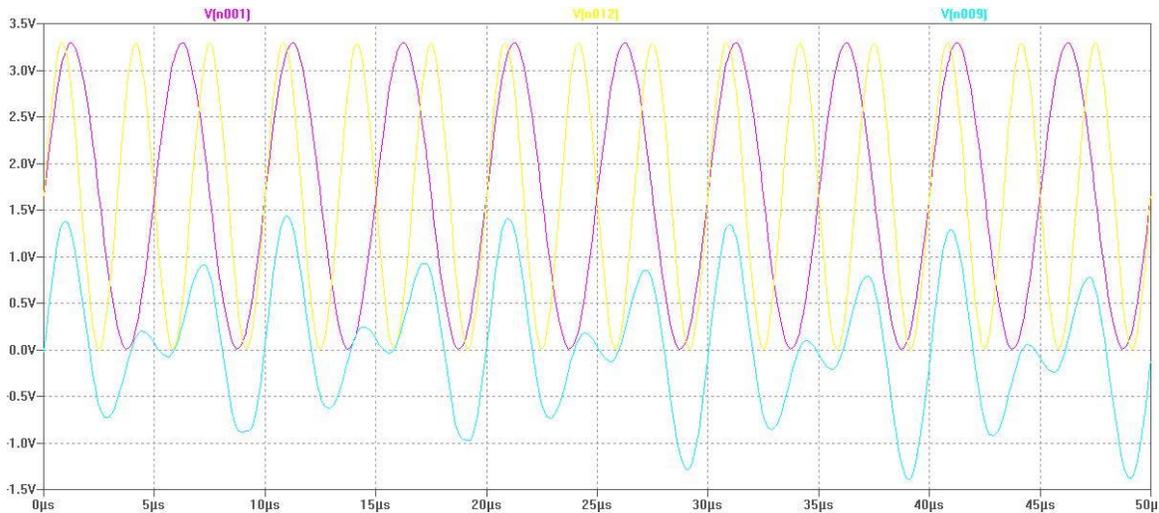


Figure 17: Mixer Simulation Results. Inputs(Purple/Gold) and Output(Blue)

The two input signals are mixed through the FET mixer, and the output signal is a combination between the two input frequencies, their sum and their difference. The result has then been shifted down by the output capacitor. So, according to this simulation, the mixing circuit works according to plan. This output signal will then be carried to the next stage of the Theremin circuit, the demodulation and filtering stage.

Most methods of envelope detection used in signal demodulation discovered during research were found to be very simple designs. The most commonly used envelope detection circuit in the RF-circuit design field seems to be the simple diode envelope detector. This circuit consists of a 1N4148 diode connected between the input and output of the detector. A grounded resistor and grounded capacitor are placed in parallel formation at the output of the detector circuit. The diode will be placed facing right to left to act as a half wave rectifier for the lower half of the frequencies. This rectified signal is then filtered by the capacitor, which smoothes the negative side of the envelope by eliminating most of the reference oscillator frequency components. The resistor serves purely as a load to the rectifier. This demodulator circuit will extract the difference between the two input frequencies. The resulting envelope of this demodulated wave can be expressed by the equation:

$$V(t) = V_m(1 + m_a \cos \omega_m t)$$

Where V_m is the amplitude of the modulated carrier signal, m_a is the modulation factor, and $\omega_m t$ is the frequency of the modulated signal.

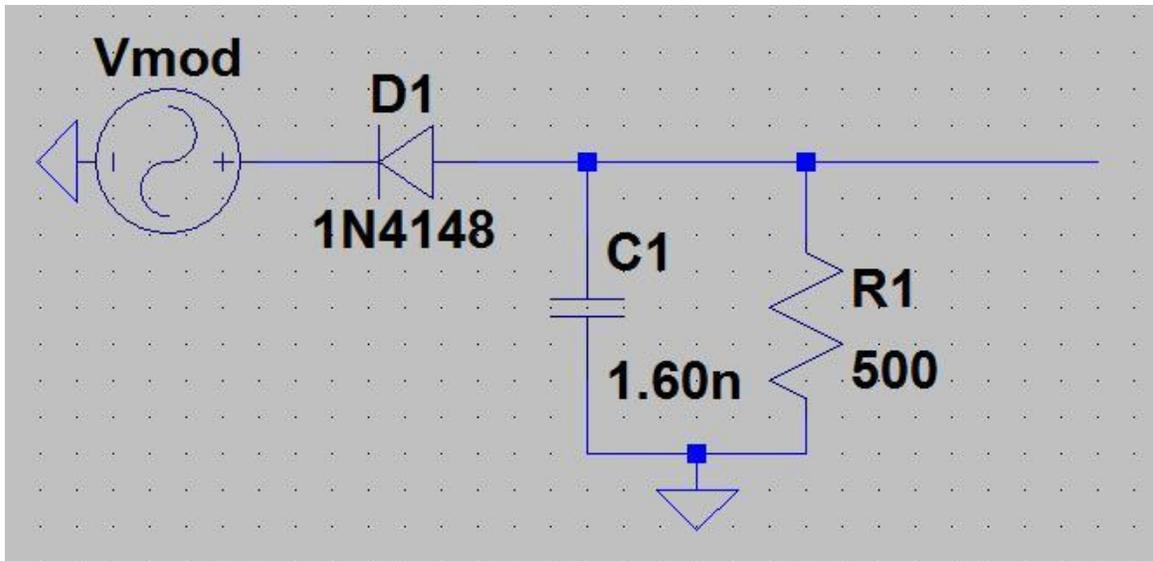


Figure 18: Envelope Detector Circuit Schematic

The values of the resistor and capacitor are determined by the following expression:

$$2\pi F_m < \frac{1}{RC} \ll 2\pi F_c$$

Where f_m and f_c are the modulated and carrier frequencies (in Hertz), respectively. To ensure that the maximum range of input frequencies was covered, f_c was chosen to be 300 kHz, and f_m was chosen to be 100 kHz. To meet the time constant conditions, the midpoint of the two frequencies was found, which came out to be 1.885×10^6 radians. Then, choosing an arbitrary resistor value of 500Ω , the complementary capacitor was calculated to be approximately 1.60 nF. The output of this circuit is then fed to the low pass filter, to eliminate all unnecessary frequencies that are outside of the human range of hearing.

While a simple passive low-pass filter could have been used to extract the necessary frequencies, that method leaves little room for gain adjustment and DC level shifting. Therefore, while originally designing this stage of the Theremin, it was beneficial to apply an active low pass filter, whose gain and shifting can be modified with the use of capacitors and resistors. The active model that was chosen for this stage is a Butterworth low pass filter, because the Butterworth filter is designed to output as at a frequency response as possible in the pass band range. This will help ensure that that signal is stable all throughout the pass band range. A Butterworth filter with Sallen-Key topology consists of an LT1001 operational amplifier, a pair of resistors, and a pair of capacitors. There is also an extra pair of resistors used to perform gain modification.

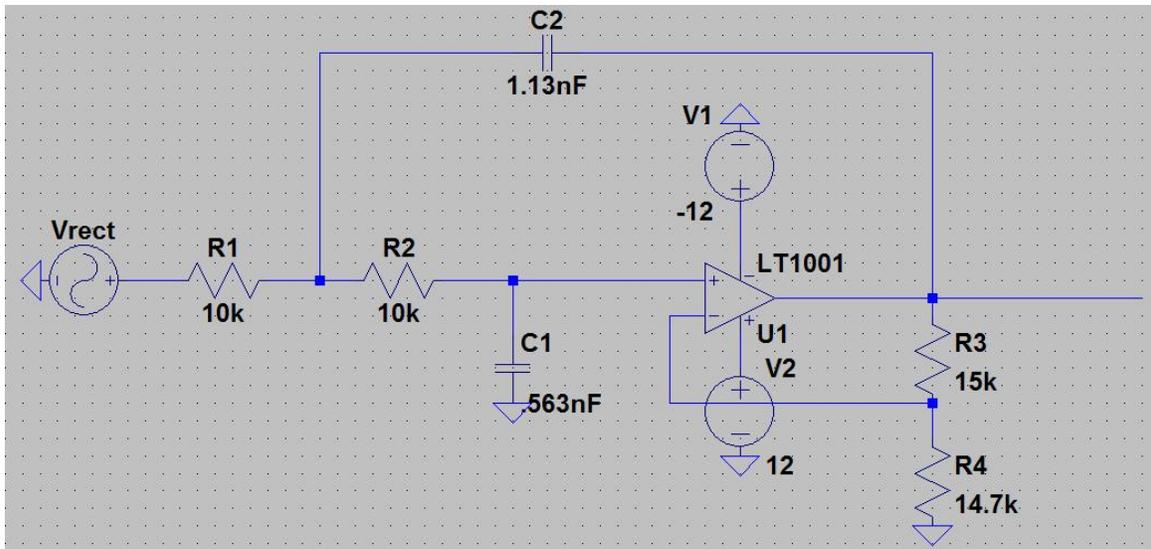


Figure 19: Butterworth Low Pass Filter Circuit Schematic

The cutoff frequency, f_c , of the Butterworth Low Pass Filter with Sallen-Key topology is defined by the equation:

$$f_c = \frac{1}{2\pi R_2 C_1 \sqrt{2}}$$

With a cutoff frequency of 20 kHz, an arbitrary resistor value of 10 k Ω was chosen, thus giving a complementary capacitor value of 0.563 nF. For maximum stability, the resistor values of the filter were matched, and C2 was given a value of twice that of C1. The final step of this filtering circuit was to control the gain of the signal. Keeping in mind that an output signal of 3.3 V_{pp} sinusoidal is desired, resistor values for the gain were chosen based on the following gain equation of the op-amp:

$$\frac{V_{out}}{V_{in}} = \frac{R_3 + R_4}{R_4}$$

Using this equation, an approximate gain of 2.02 was desired to make up for the amplitude of the signal that had been lost throughout the mixing and filtering stages. After choosing a resistor value of 15 k Ω , the matching resistor was found to be 14.7 k Ω . Using a computer program named LTspice IV, a variant of Pspice, the filtering stage was simulated using a signal that was carried throughout.

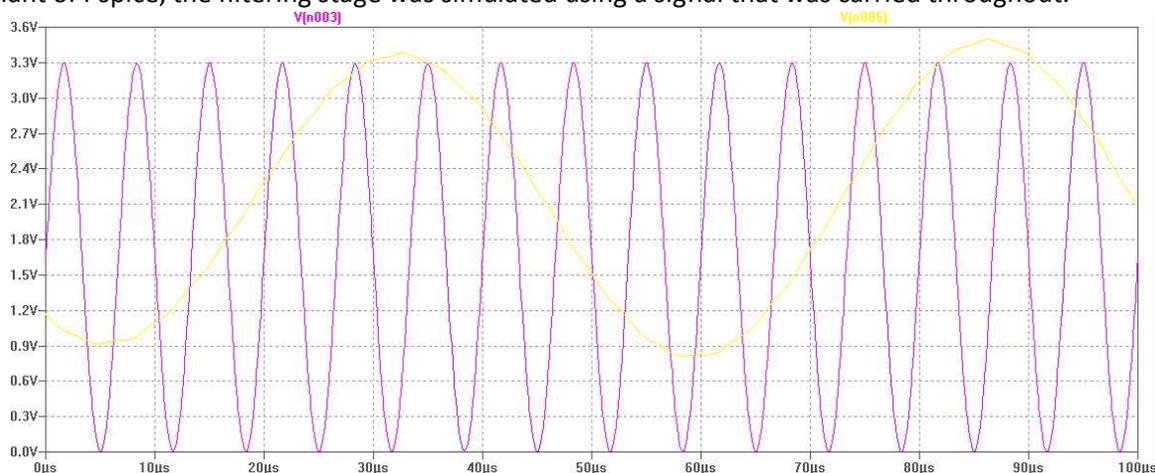


Figure 20: Transient Simulation, VFO Output(150kHz) vs. Output(17kHz)

The data shows that the original variable frequency oscillator input had now been filtered to eliminate all frequencies above 20 kHz, and the output has a peak voltage of roughly 3.3 V. This output signal may be shifted down if needed with the addition of a capacitor at the output node. The frequency response Bode plot was also simulated using LTspice.

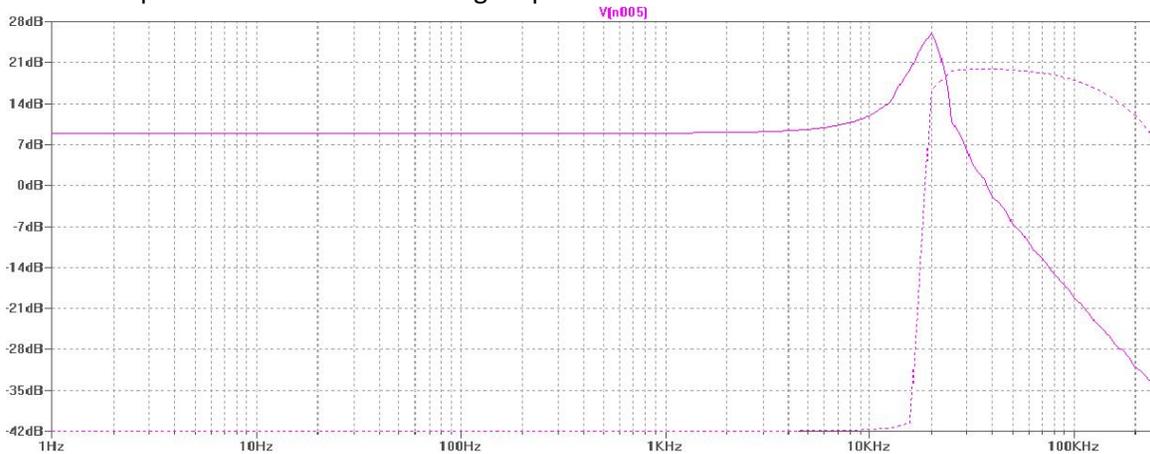


Figure 21: Butterworth Filter Bode Plot (Frequency Response)

As the simulation shows, the filter allowed all frequencies below 20 kHz to pass through the circuit. The spike at 20 kHz is what is normally known as the 3 dB point. The steep decline in dB/decade after 20 kHz shows that any information carried above the 40 kHz threshold would not be retained in the signal.

The next step that had to be taken was to design a circuit that would shift the signal of whatever the output of the filter would be, so that the bottom peak of the signal was situated at 0 V. The circuit used to perform this task is known as a clamping circuit. Its construction simply consists of a diode, a capacitor, and a resistor. The resistor and capacitor were chosen to the same specifications of the resistor/capacitor combination at the beginning of the mixer circuit. This circuit is a well proven method to shift signals up above the 0 V threshold.

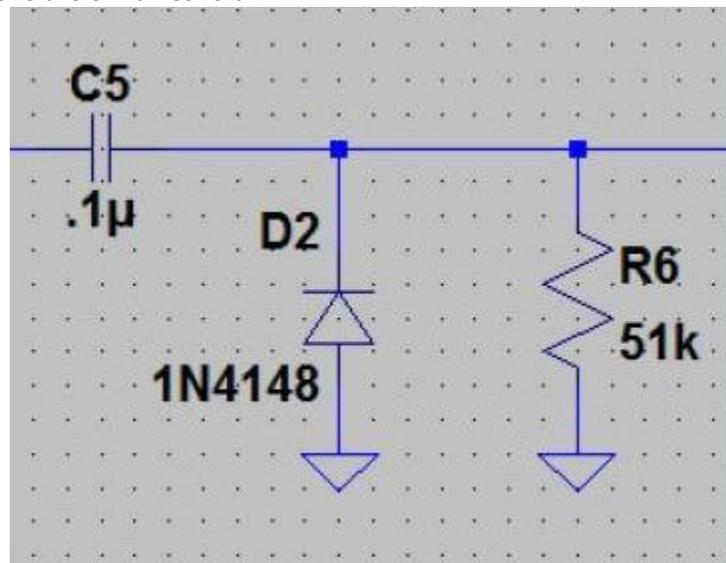


Figure 22: Clamping Circuit Schematic

3.3 Testing and Verification

When ordering parts on DigiKey, one problem that was encountered was that DigiKey did not have several of the exact values of the parts that were supposed to be ordered based on the design and simulations. Because of this, the design had to be altered to include the closest possible values to each component, based on what was available from DigiKey. This in no way affected the actual design of the circuits. After finding which component values were available on DigiKey, the final designs of the mixing and filtering circuit from the PDR were reconstructed. The Mixing sub circuit is shown in the figure directly below and the Envelope Detector, Low Pass Filter, and Clamping Circuit is shown in the next figure.

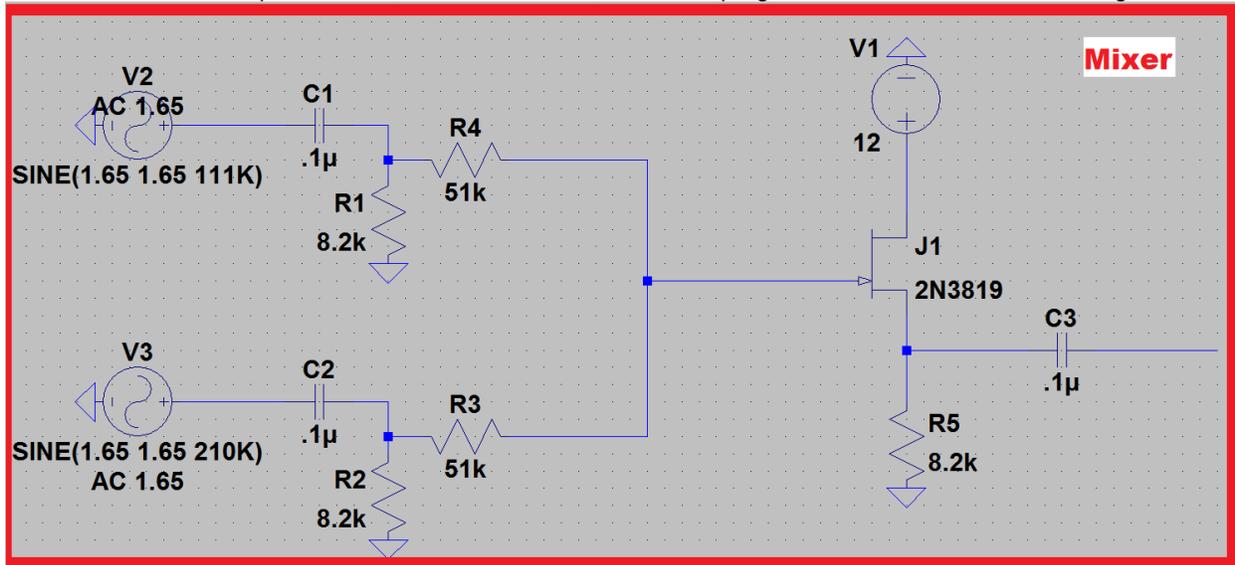


Figure 21: Mixing Circuit Schematic

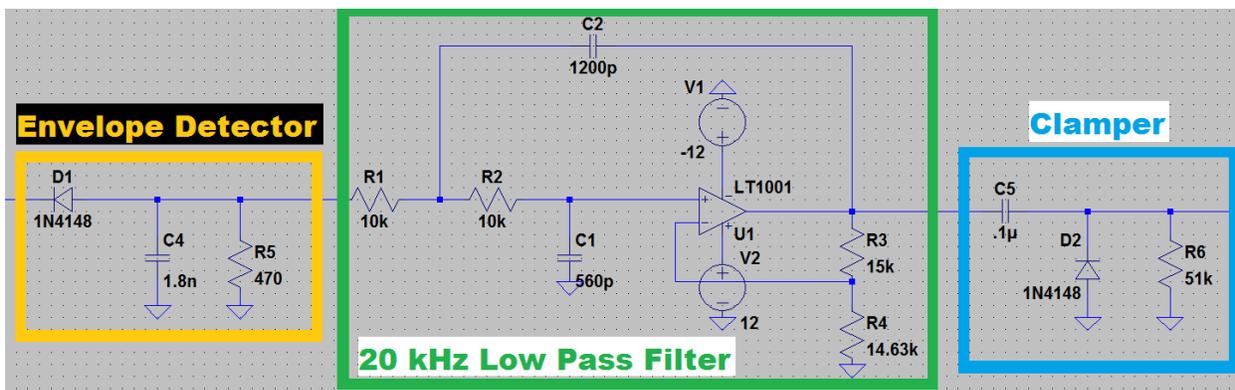


Figure 23: Envelope Detector, Filter, and Clamping Circuit Schematics

To test the mixing circuit, it was constructed on a solderless breadboard using the components received from DigiKey. This was done in room 326 of the ERAD building. Function generators were used to supply the input signals of the mixer, and a DC power supply was used to power on the transistor. After testing the results of the mixing circuit, the envelope/filtering/clamping circuit was constructed on the same breadboard. The input signal for this stage was the output of the mixing circuit. The DC power supply was used to power the op-amp. Oscilloscopes were used to read or record any signals.

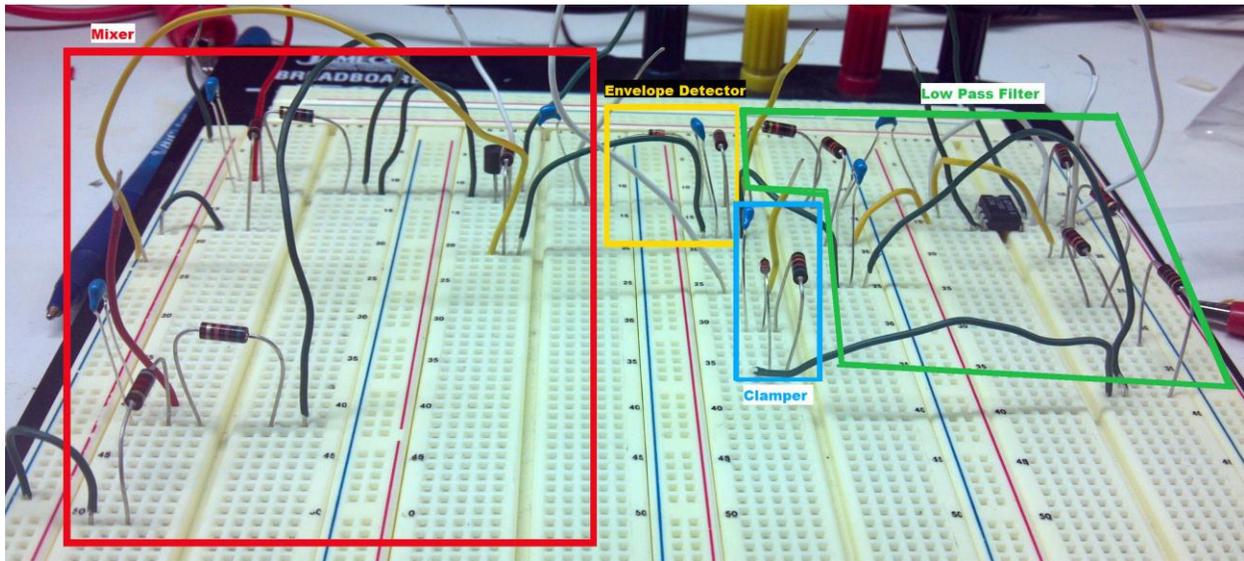


Figure 24: Mixer/Envelope Detector/Filter/Clamper Construction

The results of the circuit pictured in the figure below were taken after the mixing stage, and after the entire envelope/filter/clamper stage.

After redesigning the circuits in LTspice, they had to be resimulated with the new values so that there were updated simulations to compare with. The mixer simulation results are shown in the figure below.

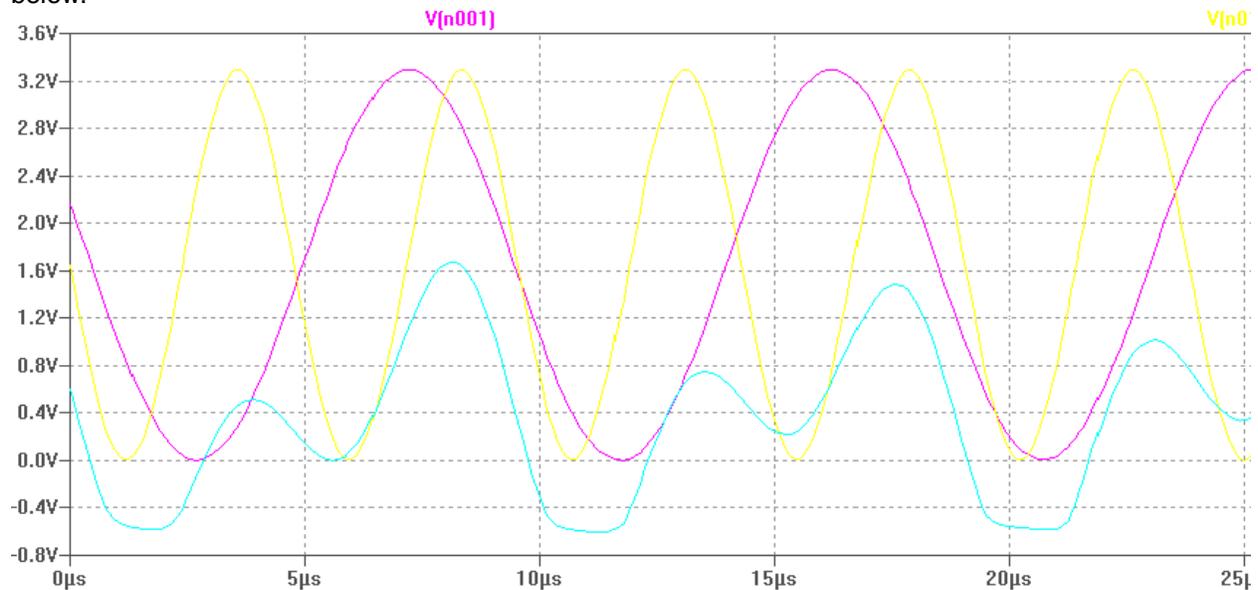


Figure 25: Mixing Circuit Simulation
Purple,Gold = Inputs; Blue = Output

The purple sine wave, at 111 kHz, represents one of the input signals to the mixing circuit. The other input, represented by the gold sine wave, had a frequency of 210 kHz. The output of the mixer is represented by the blue signal. As shown, the output signal is a mixture of the two input signals, with a varying wave instead of a clean sinusoidal form.

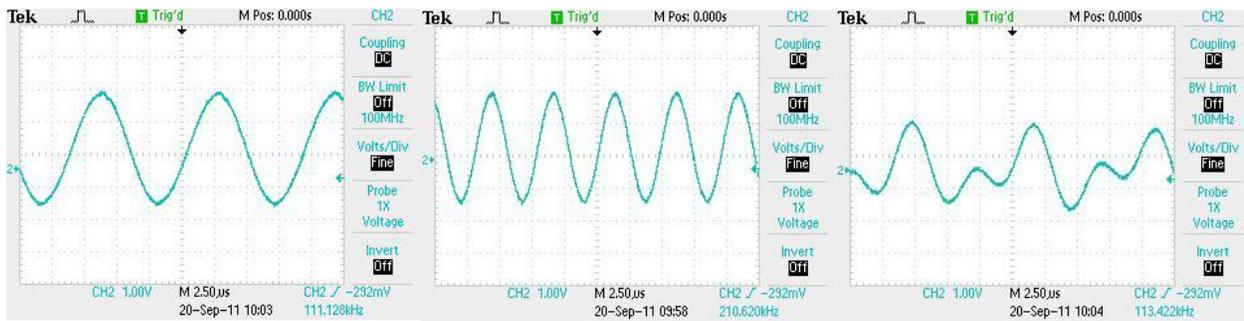


Figure 26: Inputs/Outputs of Mixing Stage
(a) V_{ref} , (b) V_{var} , (c) V_{mod}

As seen the above figure, the results of the mixing stage were a huge success. The two input signals, at 111 kHz and 210 kHz, were modulated into one output signal, represented by the third picture in the series.

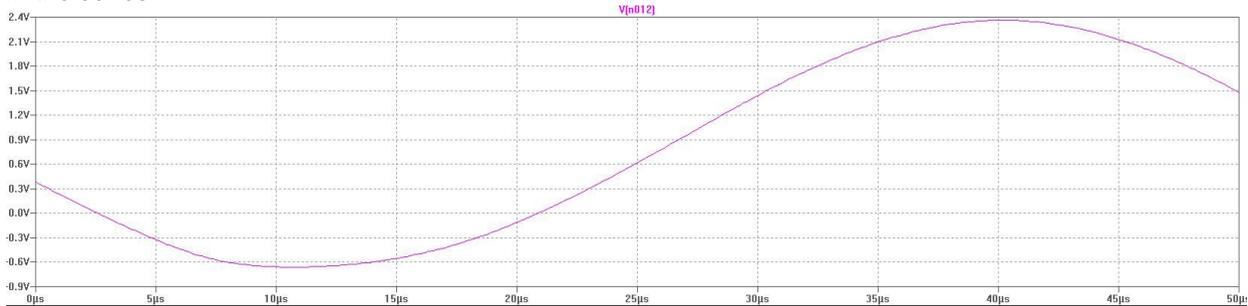


Figure 27: Filter Simulation

The envelope/filter/clamper circuit's simulation is shown in the figure above. As seen, the result was a smooth sine wave at a frequency below 20 kHz. This frequency varies depending on the input to the circuit.

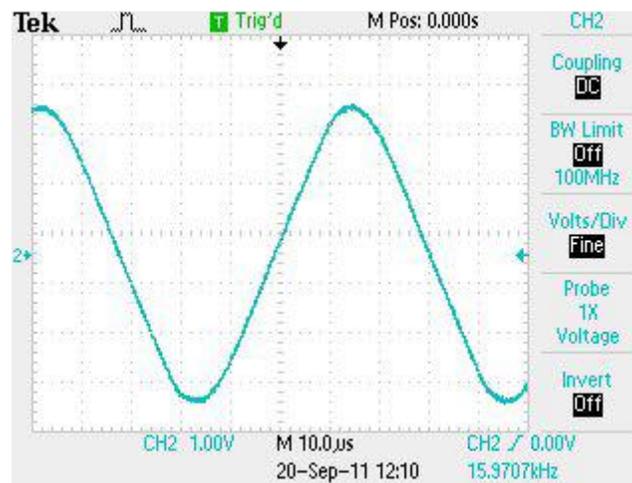


Figure 28: Filter Results

As shown in the figure above, the lab results of the filtering stage were mildly successful. A smooth, somewhat sinusoidal wave is being read at the output. As shown in the picture, the resulting frequency for this signal was right around 16 kHz. However, the gain of this signal is higher than expected. It was supposed to be a 0-3.3 Vpp wave, where as it actually turned out to be a 0-6.6 Vpp wave. This is a problem that can be adjusted by fixing the gain within the circuit.

3.4 Design Changes and Outcomes

When the team decided to go in the direction of using CMOS logic gates at the sensor stage, while trying to accommodate these changes, a few things were discovered. The 20kHz low pass filter that was using an op-amp was being forced to oscillate around 17kHz by the op-amp. No matter what the input signals to the low pass filter were, the result of the stage would be the same smooth sine wave. It was decided that a passive low pass filter could be used in place of the active filter here, with the hopes of performing any gain adjustment within the audio amplification stage. Also, this gain adjustment would most likely actually be a reduction now, since the input square waves to the mixer will be 0-12 Vpp.

After scrapping the low pass filter, it was decided that the envelope detector should be re-tested with the square wave inputs to the system as well. While the mixer still did its job and created a modulated waveform, the envelope detector now appeared to completely lose any amplitude of its input wave. It appeared that the envelope detector was reading the peak voltage of the input, storing it with the use of its capacitor, however not discharging by use of the coupled resistor. Therefore, the full envelope was not being detected, but instead just the more extreme peaks. After consulting with Mr. Scalzo about this problem, he pointed out that the issue here was that the shifting capacitor after the mixing stage, valued at 0.1uF, was a blocking capacitor, and that what it was blocking was the signal being passed through it. He demonstrated that simply increasing the capacitor value to a much larger value, 220uF, was enough to allow the signal to pass through to the envelope detector.

After this adjustment was made, it was then tested in the lab in ERAD. The following figure shows the output signal of the envelope detector, whose input signal is the output of the mixer.

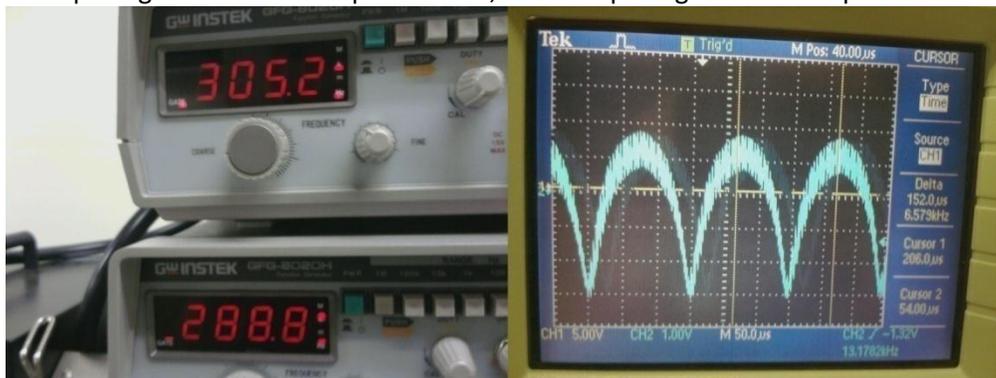


Figure 29: Mixer Input Signals and Envelope Oscillator Result

As seen in the figure above, the envelope detector was working after the adjustment was made. The resulting waveform seen in the second half of the picture was an envelope, or a peak waveform, of the modulated signal that was being fed to the input of the envelope detector. However, it is also apparent that there was a good bit of noise, or ripple, being seen in this signal. This is where the upcoming low pass filter comes in handy. It will extinguish all of the remaining ripple that was being created by the high frequency noise in the output signal above. To do this, a simple passive low pass filter, consisting of only a resistor and capacitor was constructed. Assuming a capacitor value of 1nF, the resistor value to establish a cutoff frequency of about 30kHz (to allow the gain at 20kHz to stay high) was established using the following equation:

$$f_c = \frac{1}{2\pi RC}$$

The resistor value was found to be approximately $5\text{k}\Omega$. The desired brand of resistors on Digikey were out of that value, so the settled upon value was $4.7\text{k}\Omega$. The resulting filter is shown below.

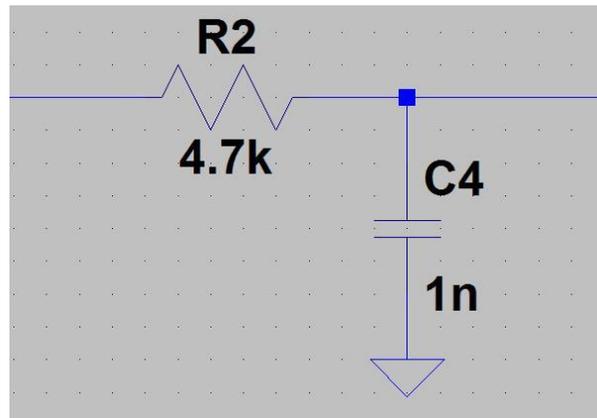


Figure 30: Low Pass Filter Circuit Schematic

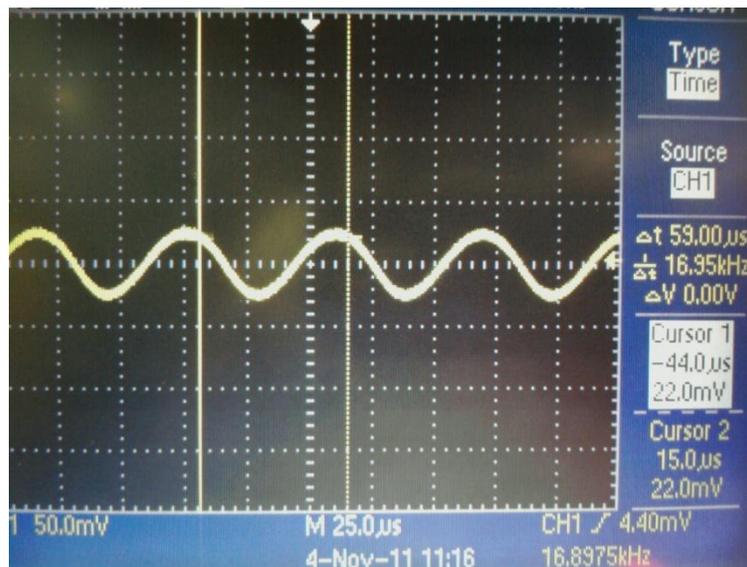


Figure 31: Results after Filter

Since the output signal of the mixing and filtering stage needs to be within a certain range for the input of the microcontroller, $0\text{-}3.3\text{V}$, the next stage that was developed to meet these conditions was a DC-level shifter, with gain control. The design that was decided on, consists of an LT1001 Op-Amp with a resistor combination that determines the DC level, and a resistor combination that determines the gain. The resulting circuit is shown below.

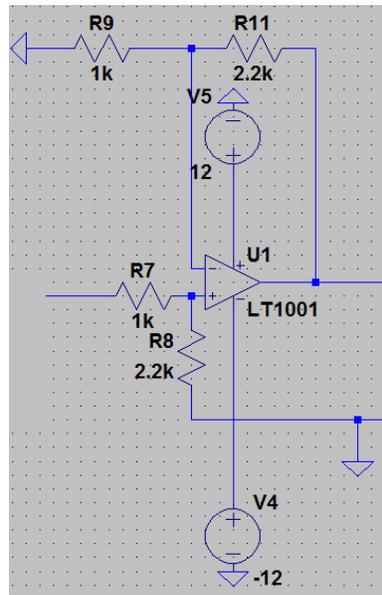


Figure 32: DC Level Shifter with Gain Modification

Assuming $R7 = R9$ and $R8 = R11$, the equations used to determine the resistor values are as follows:

$$Gain = \frac{R8}{R9} = 2.2Offset = \frac{R7}{R9} = 1$$

After seeing these results, it was decided that another passive low pass filter, identical to the previous one, would be added just to make sure the signal was as noise-free as possible. After this, the mixing and filtering stages were completed.

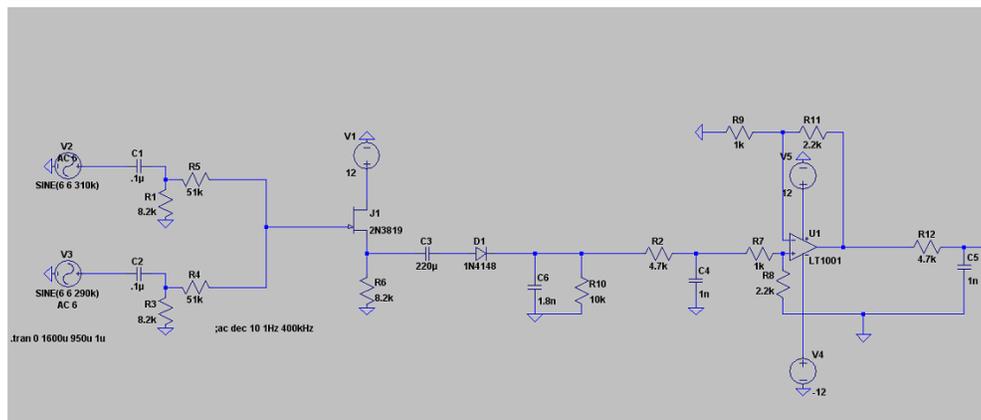


Figure 33: Whole Mixer and Filter Schematic

The circuit pictured above was tested in its entirety and the results were admirable.

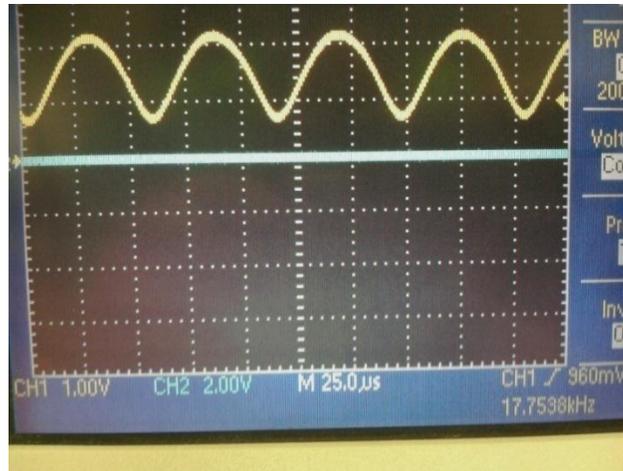


Figure 34: Output Signal of Mixing/Filtering Stage

As seen above, the signal that is produced by the mixer, and passed through the shifter and filters, is a clean sinusoidal wave that falls within the 0-3.3V range.

After confirming that this stage worked on a solderless breadboard, the next step was to solder the circuit to the final board that would be placed inside of the Theremin case.

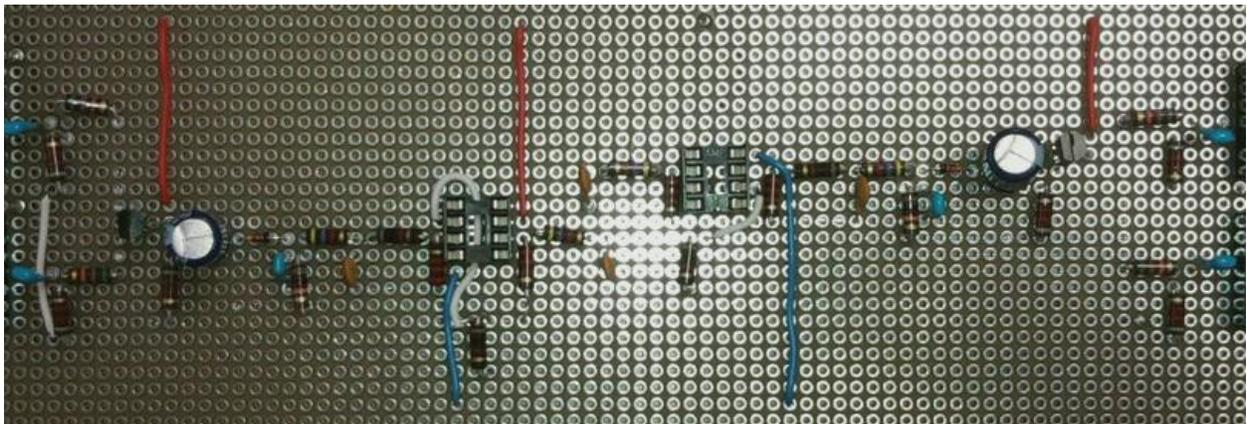


Figure 35: Soldering Mixing/Filtering Circuit

Of course, the only remaining step was to test this soldered circuit and compare it to the results that were produced by on the solderless breadboard. The same lab equipment was used for these tests.

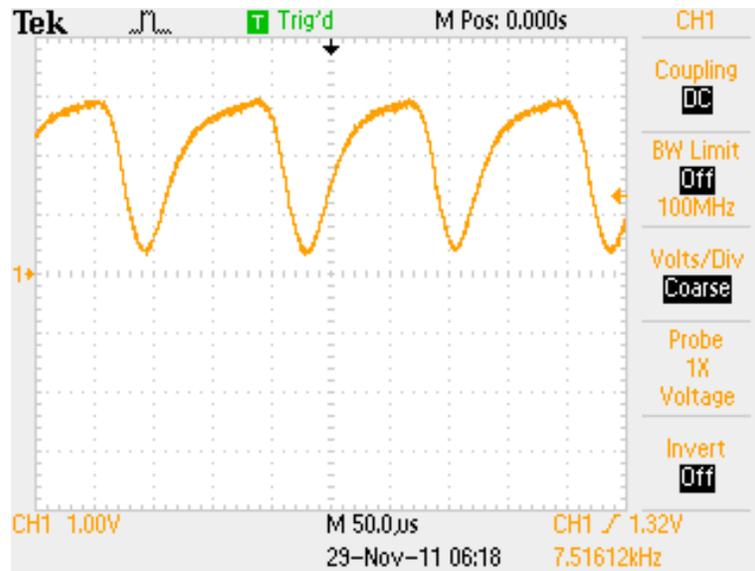


Figure 36: Results of Soldered Circuit

As seen, the results of this circuit produced on the soldered board were very successful, but not quite as good as they were on the solderless board. There is a small bit of noise being seen in the signal. The magnitude of this noise is approximately 50mV. This could be because of one or two reasons. The first of which is that maybe there was some faulty soldering. Nobody is perfect at soldering, so there may have been some shoddy connections that were made that created noise within the system. Another possible problem is that most of the resistors used in the circuit were carbon composition resistors. It may have been more effective to use carbon film resistors instead, which would have produced slightly less noise.

3.5 Lessons Learned

The issues encountered throughout the design and construction of this stage of the Theremin circuit were, for the most part, minor and easily avoidable. It's basic electronics knowledge that a certain value of capacitor will block a signal. Also, it was very easy to replace the active low pass filter with a passive one. Overall, the problems that were faced were much easier to deal with than could have been seen. This shows that sometimes the problem that you can't figure out can be the simplest of mistakes, like having the wrong value of component, or trying to do too much in one step when you can just break it up and have it do the same intended job.

4 Microprocessor Stage

4.1 Design Requirements

The Microprocessor is the brain of the Theremin. It acts as an intermediary between the Sensor/Mixing Stage front end of the device and the Audio Amplifier output stage. Traditional Theremins have no such stage. For their operation the frequency output band of the mixing stage is tuned to the human spectrum of hearing.

In order to make the device easier to play, the output would be relegated to different tones of sound instead of continuously varying output in a traditional method. This output scheme means that the Microcontroller will look for the player's hands to be in certain ranges instead of playing a unique tone for every possible hand position.

To accomplish this feat there must be a component that can parse the frequency driven input and create said output. The most realistic candidate for this task is a Microcontroller. MCU's are scaled down processors designed for specific embedded applications instead of general purpose computing in a PC. With this in mind they are typically highly connective, meaning they feature various forms of communication to and from other digital and analog devices. For a Theremin the most important devices are Analog to Digital Converters (ADC) and Digital to Analog Converters (DAC).

ADC's and DAC's functions are mostly self-descriptive. The ADC will take an analog signal at its input and create a digital representation of that signal at a given time using binary values. The rate at which an ADC can do this operation is known as its Sampling Rate. This value is only as important as the input window that the device can convert. Peripheral ADC's which are integrated onto the MCU chip itself are restricted to a window from Logic 0 (0V) to Logic 1 (3.3v or 5v). In order to determine an effective sampling rate when choosing an ADC the guiding principle is the Nyquist Theorem. Plainly this states that the sampling rate needed to fully reconstruct an analog signal is 2x that of the sampled signal.

Digital to Analog Converters are capable of creating voltages at precise intervals/steppings within their output window. The larger the amount of bits the DAC has the more precise. Creating periodic signals with these devices is done by updating the output voltage at such a time step that the output is a digital facsimile of an analog waveform. The number of steps per wave required to create a satisfactorily smooth signal is entirely subjective. Old Nintendo and Sega video game systems were famous for their 8-bit sound output.

4.2 Proposed Design

Selection of a Microprocessor was made by looking at the design requirements and the hardware implementation hurdles that would have to be overcome to use the MCU. Performance among MCU's varies from devices meant to do little more than run an alarm clock to chips powerful enough for complex DSP calculations in real-time. Overall all MCU's are available in a few different formats

- Discrete Product, a single(or pack of) chips that must be soldered to a board
- Evaluation Board, usually just the discrete product with a power regulation circuit and breakout
- Prototyping Platform, much more well-rounded product, all of the above with unique software

After considering the alternatives using a prototyping platform was the path that was chosen for the Theremin. Because of the inherent complexity of the hardware on the Theremin itself adding hand wired circuits to run the MCU seemed to add complexity only for its own sake and create potential roadblocks to getting a functioning design.

At the time of the PDR the MCU that seemed to most likely candidate to be used in the Theremin was the STM32F103 made by STM Electronics and available as a prototyping kit via a number of manufacturers. A number of features on this board made it stand out

- Powerful 32-bit ARM Cortex M3 Architecture
- Built in 10 Bit ADC unit
- Built in 10 Bit DAC unit
- Low Power Consumption (<100mA)

However after researching the issue of selection a processor over the summer it became clear that the STM32 had a deficiency: Software Support. The Software Toolchain is a term used to refer to the serial chain of tools and processes that are required to take a piece of HLL(High Level Language) code in C++ and create a binary image file that can be run by the MCU. The plan of action at the time of the PDR had been to use the Code Sourcery G++ Toolchain which was free to use. After doing more research the complexity of using this Toolchain was looming.

A new Microcontroller candidate for the Theremin was found while researching the original option. The Mbed microcontroller is a Prototyping Platform created with the intention of allowing rapid prototyping by users. The Mbed itself is a breakout board for the NXP LPC1768 MCU. The clear advantage that the Mbed has over the STM32 and any of its competition was the community support and the compiler ease of use. A compiler is the program that converts HLL to a binary image file. For the Mbed the compiler is online, along with the entire community at Mbed.org. This allows access to your code from any location that has internet.

The community however was a real motivational factor for choosing the Mbed. The majority of the libraries for the Mbed are used created and user supported. Community driven development of both the software and hardware of the Mbed greatly softens the learning curve of programming.

With regards to hardware, the Mbed has nearly a 60% increased Clock speed at 100 MHz compared to the 60 MHz of the STM32. It uses the same 32bit ARM Cortex M3 core as the STM32 and is coupled with comparable Peripherals:

- 12 Bit Analog to Digital Converter with 200Khz Max Sampling Rate
- 10 Bit Digital to Analog Converter
- 0-3.3V Logic Levels.
- 512kb Programming Flash vs 256kb on the STM32

4.3 Flow Control

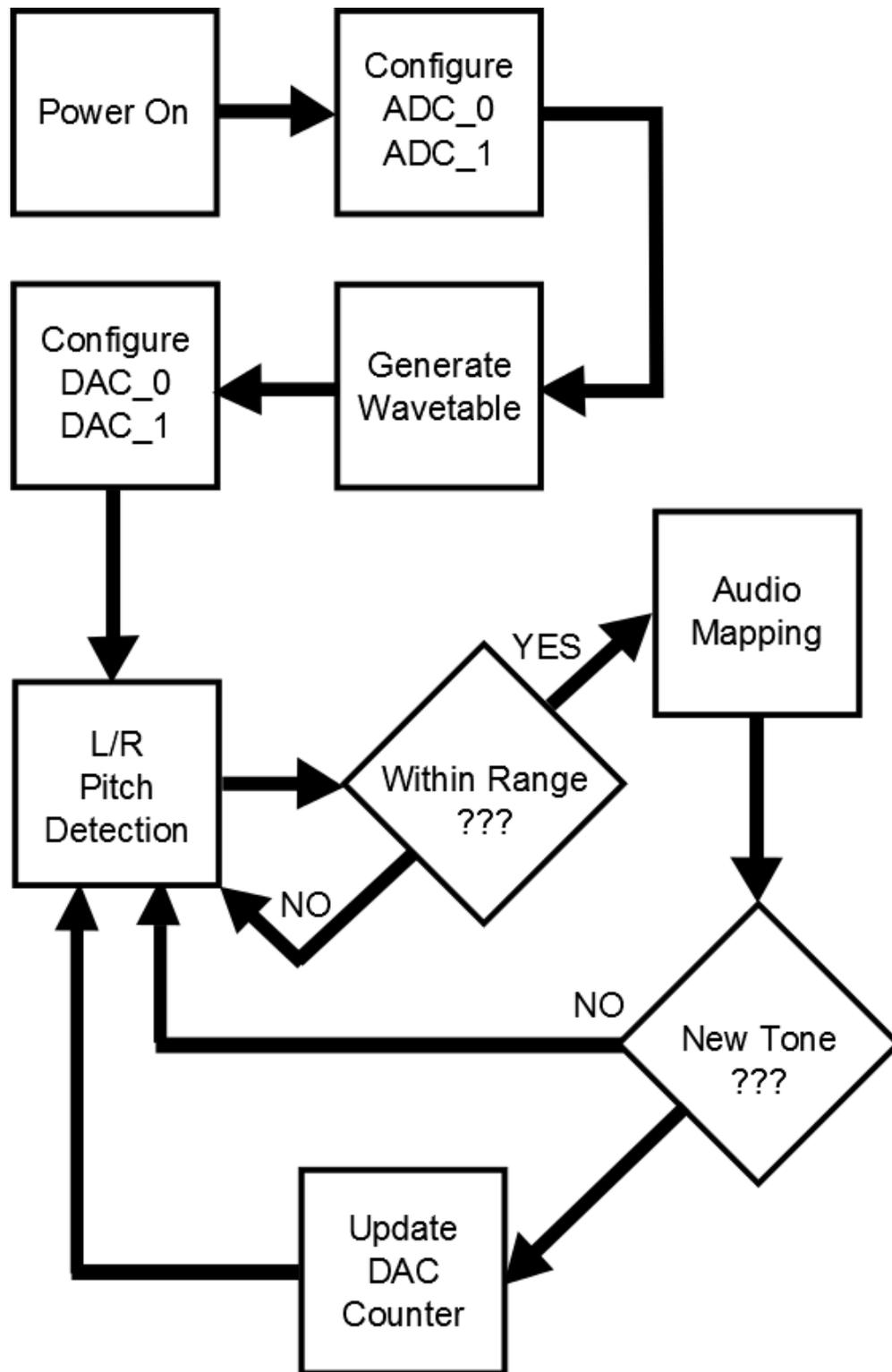


Figure 37: Software Flow Control

This flow control diagram is a revised version from the Preliminary Design Review. The major difference in the flow control is the change in pitch detection. In the original concept this operation was performed on each hand before every update of the audio waveform. In this more streamlined design only one ADC channel is sampled per each loop of the control program.

There are several potential benefits to this

- Doubling the Sampling Time/Memory Space to get better accuracy
- Performing more computationally complex detection algorithm only once per cycle
- Getting faster device response since computation is invariably $\frac{1}{2}$ of previous version

4.4 Testing and Verification

For the first milestone of the project the goal was to have a naïve frequency detection algorithm running. To get the most efficiency in development speed a function generator from the lab would serve as the input during testing. This allowed development in parallel with the hardware.

Using the peripherals in software on the Mbed can be done in a few different fashions. Using internal timers on the device to trigger ADC captures is the most “naïve” way to get a timed ADC sample. This method incurs a large amount of CPU overhead and can be subject to failure if the timing operation is triggered and offset by internal interrupts on the Microcontroller.

The more correct way to use the peripherals on the MCU involved using the GPDMA (General Purpose Direct Memory Access) Controller. The core functionality of this device is that it allows data transfers to and from peripherals and memory without intervention of the CPU. Having 8 logical channels that can be preconfigured to run in the background allows for precisely timed ADC capture rates while creating very little overhead.

Accessing the GPDMA to load configurations can be done in 2 ways: Direct Manipulation of registers that are API accessible in the mbed.h header as pointer references, Use of community created libraries that accomplish the bit bashing for you while allowing the configuration to be as semantically pleasing as possible. The latter method was chosen as adding multiple configurations becomes a large issue with maintaining system state without having some type of automation.

Because of the learning curve associated with understanding the GPDMA functionality and implementation the Milestone 1 goal was reduced to getting the ADC Peripheral and the GPDMA working in unison to sample voltages. The test configuration for Milestone 1 included

- DC Power Supply adjusted from 0 to 3.3V
- Mbed configured to take samples from ADC using GPDMA
- Output verified by having Mbed print the converted ADC values to the serial terminal on connected PC

Results for the first Milestone were satisfactory; the Mbed was able to successfully read voltage values past several decimal places. While frequency detection was not implemented the learning curve with understanding the GPDMA controller took its place in difficulty.

For Milestone 2 the main focus was put on getting DAC output working. Focus was shifted away from ADC conversions due to changes in the oscillator and mixing circuit designs. Because the detection algorithm in such a small device cannot be context free, meaning it will always take advantage of knowing specifics of the input waveform; there was little sense in continuing development until details of the waveform were completely understood.

The DAC Peripheral can be configured with the GPDMA to be fed values for output directly from memory. Outputting a waveform on the DAC required two basic elements

- Wavetable Array to hold the voltage values of a Sinusoid
- Note Array to hold precomputed values to output specific frequencies.

The Output Frequencies for the Theremin are modeled on a traditional Piano, excluding the 2 Highest Octaves. This gives us a frequency range from 27Hz at the lowest key to ~1kHz at the highest key, in steps as close as possible to the Piano. In practice 27Hz is untenable due to the frequency response of common speakers being non audible until 300Hz. Piano Keys are calculated to the 3 decimal place but a Digital Device is restricted to non-fractional frequencies. Calculations of these frequencies are done by the following equation:

$$f(n) = 440(\sqrt[12]{2})^{n-49}$$

Where n is the key number and $f(n)$ is the frequency of that key

To get a sinusoid output at a specific frequency f each value of the wavetable of size n requires a total of $f * n$ updates per second. The DAC in the Mbed has 10 bits of output resolution. For a wavetable this allows up to 1024 indexes on the table. For mathematical simplicity the wavetable will be size 360. To set the specific frequency of output there are two elements that can be modified:

- PCLK Register to divide the DAC clock by binary intervals of its main 24Mhz frequency
- DACNTVAL register which is a counter that resets after it reaches 0

PCLK only has to be divided to get more precision out the DAC or to hit different ranges of frequencies. For the desired spectrum discussed above it does not have to be modified. The formula for determining the DACNTVAL value for frequency f with wavetable size n is:

$$DACNTVAL = \frac{24,000,000}{fn}$$

To get smoother output a technique known as Double Buffering was used. In practice the DMA fetches single memory addresses per cycle. There are conditions which can force the audio output to be interrupted during operation. To prevent this stutter two DMA channels are used. The first channel grabs index n of the wavetable, with the second channel prefetching $n + 1$. Once the transfer of n has completed the second channel is immediately ready to send the subsequent value. Upon completing this transfer the first channel is ready with the next value and this process iterates continuously.

The test configuration for testing DAC output was as follows:

- Mbed plugged into PC for power
- Oscilloscope probing Pin 18 of Mbed (This is the only Audio Out Pin)
- Mbed generated a 0-3V sinusoid @ 435 Hz for the 440Hz target frequency.

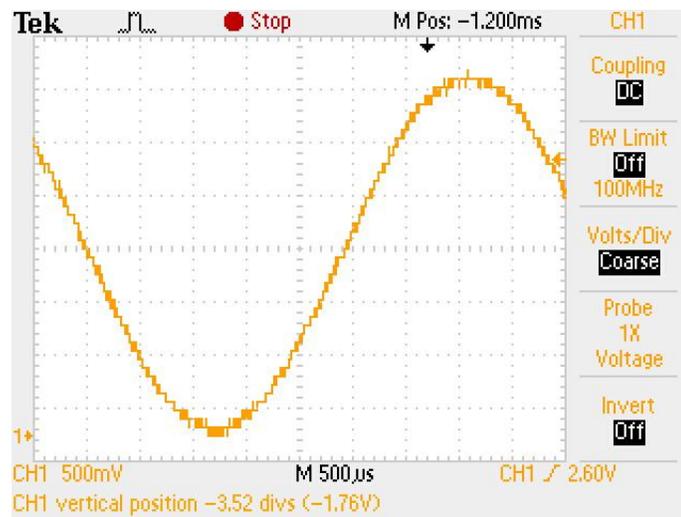


Figure 38: DAC Output

Note that the wave position has been shifted on the scope to get a better view of the voltage steps. In general most of the frequencies were slightly lower than what was predicted using the formula. To compensate for this they were adjusted manually until tuned properly. It is likely that the Oscillator that is assumed to be 24Mhz has some percentage of error in oscillation frequency that is causing this issue.

After completing the DAC Output Algorithm the next logical step was to return to the Frequency/Pitch detection algorithm.

Some success was seen by having pure square waves trigger edge interrupts on the Digital I/O Pins of the Mbed device. Using a Timer to count the time between interrupts the frequency could be estimated. This method however is flawed due to the fact that square waves that trigger the interrupt will not reach the device with the filtering mechanisms in the circuit.

Some of the Time Domain detection methods were more promising. As shown below in figure 36

The main method of detection was based on Peak Detection. Knowing that a sine wave has periodic behavior certain assumptions can be made that allow for easy detection. At the absolute peak of a sine wave the slope of the tangent line is 0. This point is essentially undetectable through DSP because the waveform as it exists in system memory is reduced to a number of finite points. To get a rough estimate of where the peak is Derivative Testing was used to get Peak Estimation. The essential component is that the first derivative test in discrete time between two consecutive points is the same as subtracting them. By continually doing this test and checking for the change in the derivative you can get a rough estimation of where the peak or valley of the waveform is. Because it is known that the waveform coming into the microcontroller is symmetrical, only $\frac{1}{2}$ the waveform needs to be seen. Once a peak and a valley is detected then an estimate of the frequency can be made.

There are some noticeable downsides to this technique. First it requires sampling far over the Nyquist Rate. The Mbed using DMA for ADC control can sample at a maximum of 126kHz. Sampling at 120kHz gives 6 points of estimation on a 20kHz signal, this is just enough to perform this test. Also the computational complexity might be higher than it seems. Because operations are being performed on every points 2 times to test for derivatives the complexity is $2n$. In the tests performed this method worked reasonably well at Low Frequency(5kHz) but suffer at higher frequencies. If this method were pursued further it may be beneficial to use external ADC's with large sampling rates that give to program more advantage.

Zero Crossing is a Time Domain technique that resembles the Derivative Test. Instead of searching for the Derivative sign change, instead one is looking for a sign change in the values of the waveform itself. Because the Mbed ADC range of input is 0 to 3.3V the point of interest for zero crossing is approximately 1.65V. To do this comparison 1.65V was converted into the raw value that the ADC reads, and if it two consecutive samples in the buffer crossed over that unconverted value then that was considered a zero crossing point. Like the Derivative Test technique this relies on heavy oversampling to get more accurate results, but still only requires that $\frac{1}{2}$ of the total waveform be seen to get an estimate of the frequency.

A graphic showing the Spatial Domain Techniques is shown below.

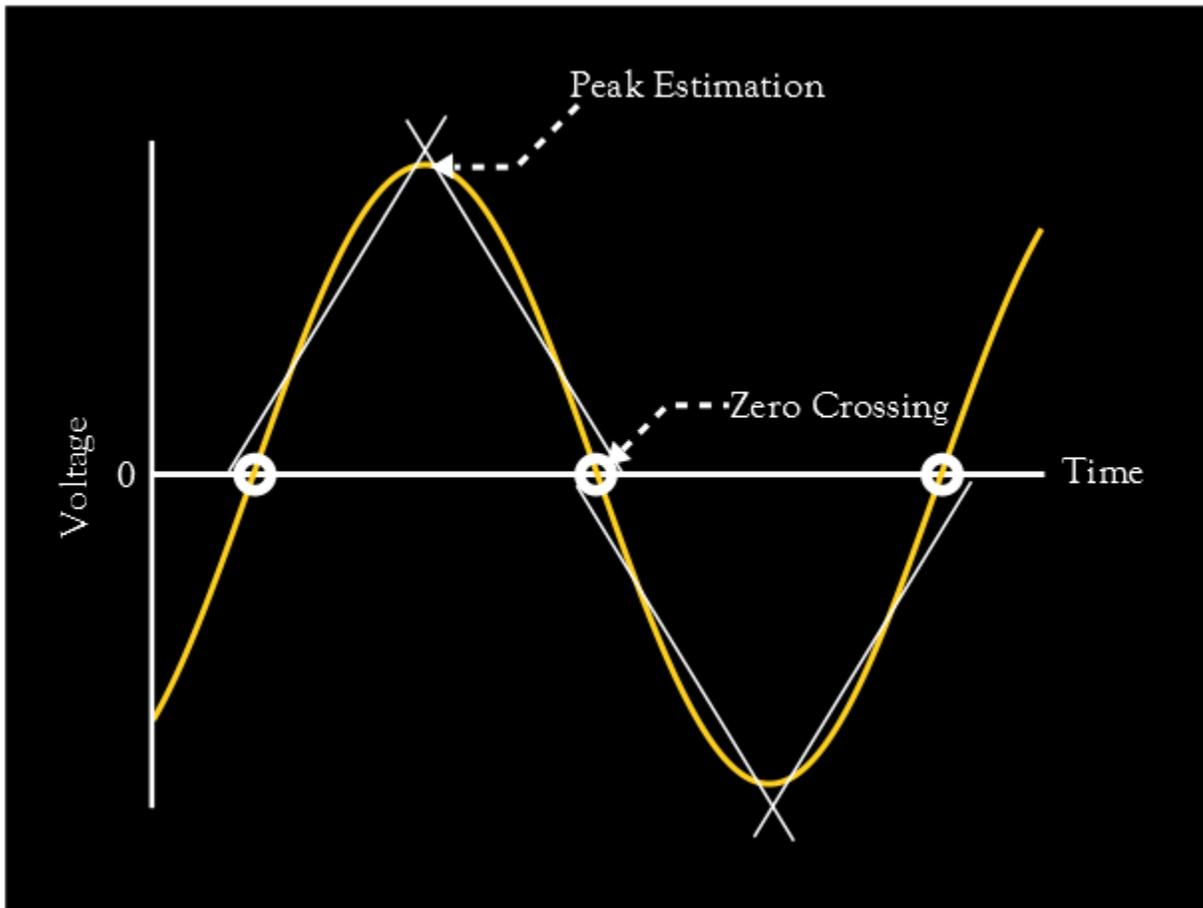


Figure 39: Time Domain Detection

The final technique that was tried unsuccessfully was a Frequency Domain Technique known as the FFT(Fast Fourier Transform). The function was available as a library function from the company who manufactures the chips on the Mbed(NXP). Implementation of this function required knowledge that was too much to pick up as late in the semester as this technique was tried. The motivation for this technique being implemented was based on the introduction of noise into the circuit after it was soldered together on Perforated Board. To use the spatial techniques several bits would have to be truncated from the sampled waveforms to compensate and there was less accuracy in the functions themselves at that point.

4.6 Outcomes

The Outcomes of the Microprocessor Stage are essentially 2 or 3 distinct programs that serve different parts of the design goals that were set. No one program was created that could do both the frequency detection and the audio output due to time constraints and issues with hardware that put the microprocessor on the backburner until they were fixed. Due to continuing hardware issues up until the end of the semester little more effort was put into completing the algorithms as troubleshooting the perforated board was occupying hours of time from every member.

4.7 Lessons Learned

The most important lesson that can be had from this experience is that many types of algorithms that work with ideal inputs will simply not work well with realistic ones. Assumptions were made that did not compensate for noise in circuits that rendered 2 algorithms useless, and the third algorithm could not be attempted because it was thought the entire semester that it would not be needed. For future such project it would be prudent to make worst case assumptions and assume that it will never get better. Microcontroller interfaces continues to be one of the most difficult types of programming as interfacing with external components is simply more difficult than software only design.

5 Audio Amplifier Stage

5.1 Design Requirements

The amplifier is the last stage of the audio conversion process. After the microprocessor outputs the final waveform, it is the audio amplifier's job to adjust the volume of that waveform to make it louder or quieter based on the input of the user. The volume adjusted wave is then output to a speaker and heard by the audience.

There were only two requirements that were required from the audio amplifier. First, the volume had to be adjustable by the player by a case mounted volume knob. Second, the maximum volume of the device had to be loud enough so it could fill a room with sound and could be heard clearly by an audience.

5.2 Proposed Design

The proposed design, that was presented last semester in the PDR, utilized the LM380 integrated circuit chip, a specialized class AB power audio amplifier, widely used in consumer application such as TV sound systems and AM-FM radios. The LM380 features a wide supply voltage range (10V-20V), low distortion, and a fixed voltage gain of 50. The circuit that was presented is shown below.

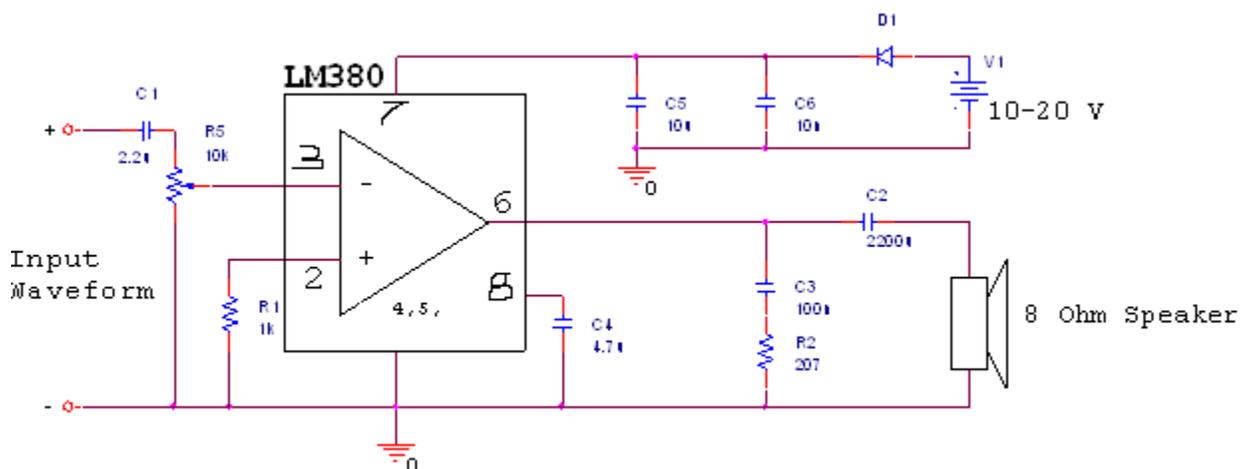


Figure 40: Proposed Amplifier Circuit

Capacitor 1 (C1) is the input coupling capacitor, which blocks any DC voltage that might be present at the input. Capacitor 2 (C2) represents the output coupling capacitor which is required to block the DC level (half supply voltage) that is present at the amplifier output, from reaching the speaker. Capacitors 5 and 6 (C5, C6) provide power supply filtering, and Capacitor 4 (C4) provides an internal supply bypass for extra supply decoupling. Resistor 2 (R2) and Capacitor 3 (C3) make up a filter for high frequency load stability. The potentiometer at Pin 3 of the chip represents the adjustable volume knob that will be implemented into the design.

5.3 Testing and Verification I

The proposed circuit was built and tested on a solder less breadboard in the lab. After a few minor mistakes with pin labels, the following output waveform was measured with the oscilloscope.

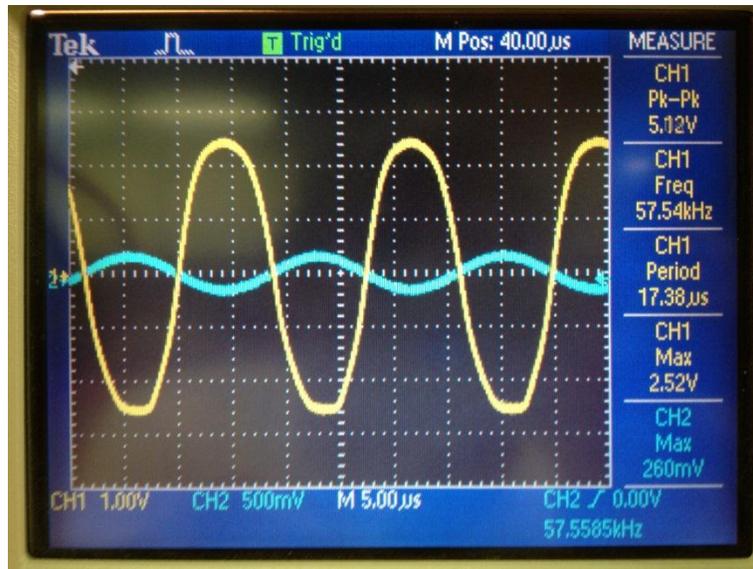


Figure 41: Audio Output

The blue wave represents the input waveform that was represented with the help of the function generator which measures 0.26 V. The output of the waveform represented in yellow is measured at 2.52 V, giving a gain of about, $(\frac{V_{out}}{V_{in}}) = 9.69$. This measurement was taken with the volume knob potentiometer turned at a quarter of maximum potential.

This is where the testing went wrong. After the first measurement above, no more further measurements were recorded. This problem was due to an accidental short between two leads of the capacitors on the breadboard and while trying to figure out what was causing the problems, the chip was inexplicably fried, thus halting our testing.

5.4 Design Changes

Some design changes occurred after the accident with the previous design. Radioshack sold a similar model IC chip called the LM386, which has the exact same features but varies by having variable gains. Purchasing this chip locally was cheaper than buying online, so it was decided to scrap the old design and use the LM386 as our audio amplifier.

Like it was previously said, the LM386 is very similar to the LM380. The only difference is that a change in gain occurs when coupling a capacitor between two pins of the LM386. The second proposed design of the audio amplifier is shown below.

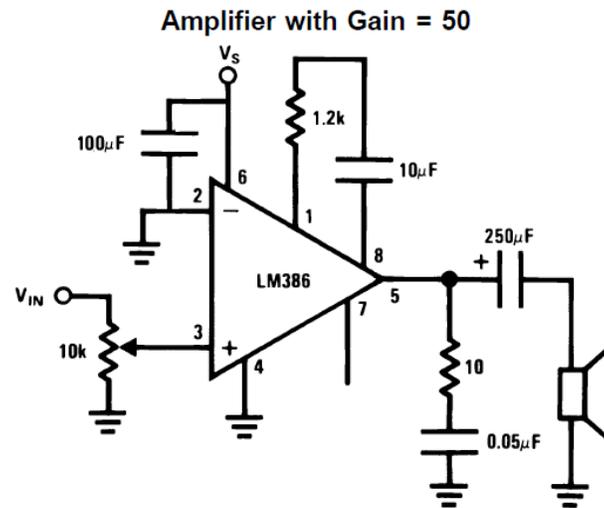


Figure 42: Revised Amplifier Circuit

Evident from the shown diagram, the revised circuit has fewer components than the original, another benefit of the changed design. The variation in gain comes from adding the capacitor between pins 1 and 8. With pin 1 and 8 open the internal gain is set at 20. While adding a $10\ \mu\text{F}$ between the pins increases the gain to 200. A 50 gain was also suggested in the datasheet by keeping the capacitor and adding a $1.2\ \text{k}\Omega$ resistor in series. This is the circuit that was chosen for the final design, but if this gain proves to be too little, then a simple adjustment to the circuit will be able to increase the maximum gain to the desired range.

5.5 Testing and Verification II

The new circuit was built and tested in the same way as the first, in the lab on a breadboard. However this one proved to be more difficult to work with than the previous design. When this circuit was tested, the output waveform was incorrect, it was noisy and nowhere near the desired output. An example of what was measured with the oscilloscope is shown in the figure below.

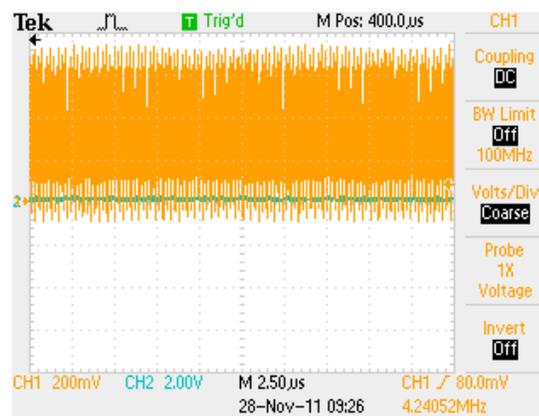


Figure 43: Amplifier Noise Output

It wasn't until a tireless search of the internet for solutions was the real problem. Originally, the datasheet suggested the revised circuit to be built without the capacitor coupled between pins 6 and 2. It wasn't until an updated datasheet was found did it say that there were problems with source noise affecting the output. The company suggested a 100 μF capacitor to help keep that noise down after implementation the correct output was viewed.

After the correct waveform was viewed from the oscilloscope, the speaker of the Theremin was hooked up to the output of the amplifier. A waveform within the desired hearing range of the Theremin was hooked up to the input and we received the desired amplification. The sound of the output was significantly higher than the input of the device. The potentiometer volume control was also tested and found to work as well, having a very fine tuning adjustment to find the correct volume desired by the player.

Next the microcontroller was implemented as the input of the amplifier. The program that was demonstrated in the Milestone II presentation, that swept the output frequency range of the Theremin, was run with successful results as well. The volume control also completed its job successfully varying the output sound, while also cutting off the sound when the knob was turned all the way down. Now that testing on the breadboard was successful it was time to solder the parts to the perf-board and implement it into the final design circuit.

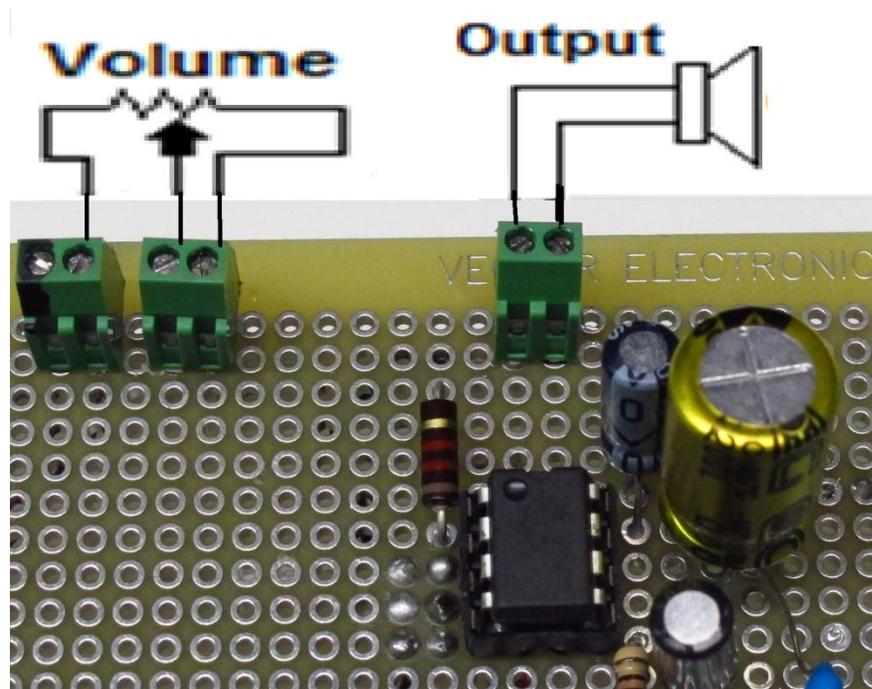


Figure 44: Soldered Audio Amp

After we soldered the audio amplifier to the breadboard, more problems seemed to occur, both minor and major. The minor problems were worked out with simple troubleshooting, for example a lead wasn't connected properly, the major problems, however, were more time consuming. It seemed like every time we reconnected the circuit together, and different result was achieved, both successful and

unsuccessful. No matter what the result, when measuring the output with the oscilloscope, it never looked right but when they speaker was used as the output is sounded fine and the desired volume adjustment was achieved. Even though the amplification can't be shown and verified with the oscilloscope the proof is in the volume of the sound.

During testing it occurred to the group that the initial offset of the output waveform coming out of the microcontroller may have an effect on the audio amplifier. In order to eliminate any problem that may occur with that a voltage offset shifter was built using an LM741 operational amplifier to shift the wave coming out of the DAC of the microcontroller down so that the offset will be zero. This circuit was built using simple components and is shown in the figure below.

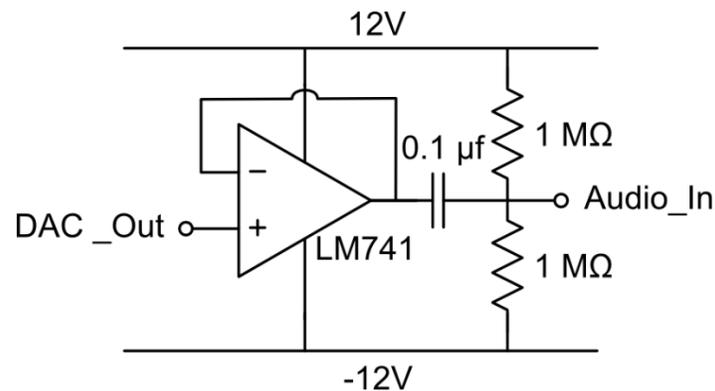


Figure 45: Voltage Offset Shifter Circuit

This circuit was the first circuit built in the project that worked without any problems, so we soldered it to the board. During initial planning this circuit was never needed, so the diagram wasn't implemented into the final circuit diagram that we etched on the top of the box. Instead of leaving the circuit components out in the open, where students could get confused about which part of the circuit this actually is, the decision was made to hide the circuit to prevent any confusion. This tiny circuit was just small enough to fit under the mounted microcontroller, so that is where it was hidden. A picture of the hidden circuit soldered to the board without the microcontroller can be seen below.

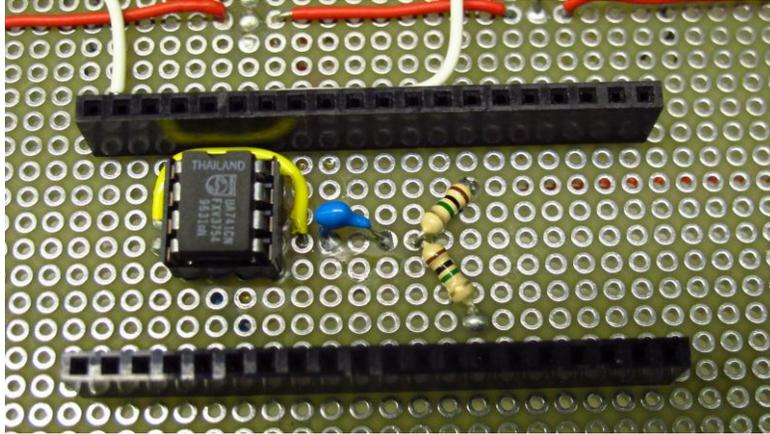


Figure 46: Hidden Voltage Shifter

After implementing the voltage shifter between the microcontroller and audio amplifier, there is a slight difference in maximum volume that is output out of the speaker. This occurs because the waveform now has an extra +3.3V range of space that it can amplify within, therefore making the sound louder.

5.6 Outcomes

The outcome of the audio amplifier proved to be quite successful. The amp takes in the, now shifted, waveform of the microcontroller and increases the volume of the sound by a certain degree. This degree can be adjusted by the user by means of the volume knob that is mounted on the side of the box. It is with this, that both of our requirements of the audio amplifier are satisfied.

5.7 Lessons Learned

Certainly there are plenty of lessons to be learned here. When testing, it's always a good idea to turn off the power supply when making adjustments to the circuit. Always seek out up to date datasheets before building the circuit, as a crucial component to circuit functionality could be missing. A backup plan is always smart to have just in case of accidents.

6 Power Stage

6.1 Design Requirements

The power supply of the Theremin is the heart of the entire device, without it the other stages won't have power to function properly. It takes the supply voltage and converts it down through a series of voltage regulators to a specific voltage needed somewhere else in the circuit.

The only requirement for this stage of the Theremin was it had to be run off a standard American wall socket, a supply of 120 V AC 60 Hz. The task was ours to design a efficient power supply that would take that socket voltage step it down and power the rest of the circuit.

6.2 Proposed Design

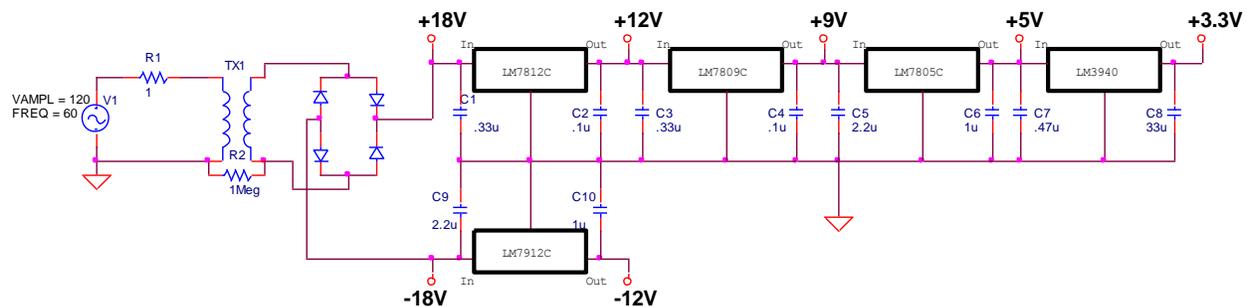


Figure 47: Proposed Power Circuit

The circuit shown above is the proposed power supply to the Theremin demonstrated in the PDR. It provides the various voltages needed to power the different components of the circuit. The input of the circuit is the 120 V 60 Hz AC voltages coming from a standard American wall socket. A transformer is used to step down the 120 V AC wall socket voltage to 18 V AC. After the voltage step down, the waveform needs to be converted to DC and a standard full-wave bridge rectifier is used for this purpose. Coupled with a 2200 μ F capacitor in parallel with the output, the result is a smooth DC voltage of 18 V with little to no ripple. After some research, it was found that using a commercially available AC to DC wall mounted power supply would be best for our project. Seeing as the transformer uses magnetic field induction to actually step down the voltage, there might be some magnetic interference with the antennas that could arise with having a transformer mounted inside the box. It also keeps the box less cluttered with components and more visually appealing to the audience viewing the device.

In order to achieve the ± 12 VDC rail voltage required by the op-amps in the mixing circuit, a series of complimentary voltage regulators are required. The first set of regulators step down the 18VDC supply to both positive and negative 12 VDC using two linear voltage regulators. Next the voltage has to be adjusted down again to 9V to power the audio amplifier. Another regulator is used to change the +12V to 9VDC, which is required to bias the transistors in the amplifier circuit. The final regulator steps down the +9VDC supply to 5VDC and then to 3.3VDC, which will supply the microcontroller.

6.3 Design Changes

Due to certain changes in the original Theremin design, two regulators in the circuit could be cut out. The microprocessor that was originally chosen for our project needed a 3.3 V supply. The microprocessor that is in use currently can accept a voltage of 5V, so the last regulator supplying the 3.3V could be eliminated. Since there was a switch to square wave oscillators, that accepts a supply of ± 12 V, the 9V regulator could also be eliminated as well.

Permission was given to use any parts that were found in the ERAD labs for the project, so out of curiosity a search for anything that would be beneficial to the project design occurred. A 19.5V laptop battery charger was found in the search and after examination of the datasheets of the regulators an AC-DC transformer was selected for the project. The only stipulation required by the regulators that was found, was that the input voltage had to be at least 2V higher than the desired output voltage. For example for the 12V regulator, a supply of at least 14V or higher has to be supplied. This decision was for the best because it not only saved money on the project, but it removed any problems that may arise from building AC to DC transformer. The new revised power supply circuit is shown below in the Figure.

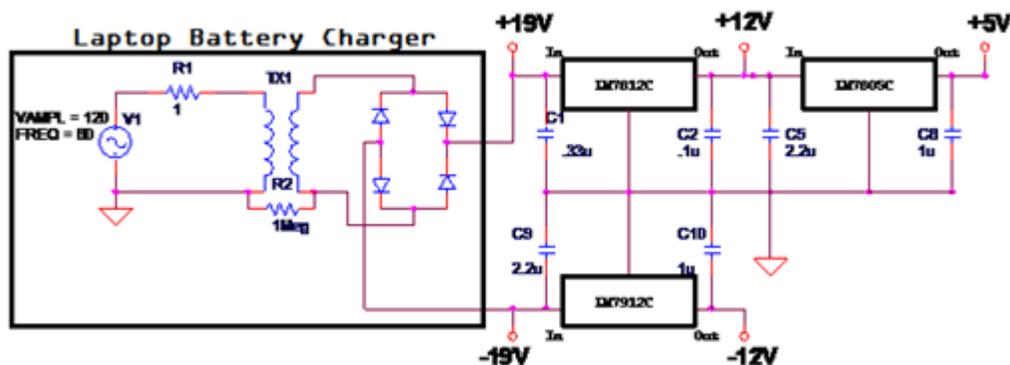


Figure 48: Revised Power Circuit

6.4 Testing and Verification

During construction of the power supply, there were several problems that occurred. The circuit as shown above is shown to have three input terminals: positive supply, negative supply, and a ground. When testing the circuit it was found that having a common ground between all four regulators would result in the wrong output voltages. The culprit of this problem was believed to be the -12V regulator because unlike the positive regulators it needed a negative voltage supply and the only way to supply this was through hooking it up to the laptop charger reverse bias. This didn't work because there was a theoretical short in the entire circuit with having the ground of the top half of the circuit being the supply of the bottom which in turn output the wrong voltages. After testing different designs that proved unsuccessful, it was decided to leave the two circuits separate and supply the two halves with a

power and ground rail fed from the laptop charger. The final constructed circuit is shown below. Each regulator has been labeled with the voltage it supplies (White) and the measured voltage recorded in the lab during testing (Red).

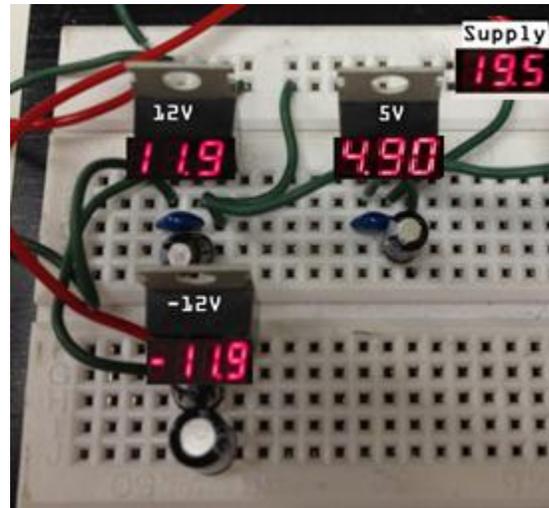


Figure 49: Power Circuit Tested Results

Unfortunately the problems didn't end there, when potential difference across the -12 V and +12V regulator was measured, 5V was the measurement received. Theoretically this voltage should be in the range of 24V, a major difference from the measured output. This was cause for concern and it was figured this would be a potential problem when integrating circuits together. Instead of spending hours troubleshooting, it was decided that a design change was needed.

The proposed design change was to scrap the troublesome -12 V regulator and instead use a charge pump IC (TC962), a standard DC to DC converter. This IC has a specialized circuit that can take in a voltage in the 3V to 18V range and converts it to a negative voltage. The plan was to run the output of +12 V regulator to the input of the charge pump (TC962), thus outputting the desired -12 voltage. The circuit for the TC962 is shown below.

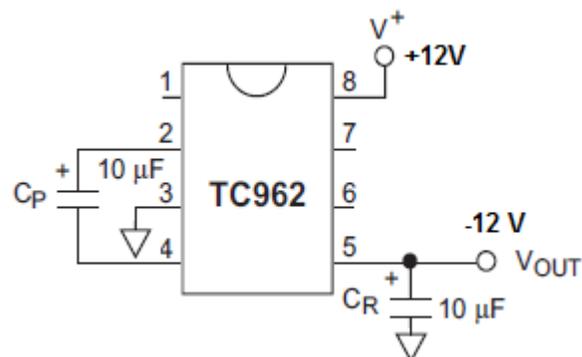


Figure 50: Charge Pump Circuit

When assembled and tested on the breadboard, the desired -12V was measured, so we moved forward to soldering. A picture of the power supply soldered to the perf-board is shown below

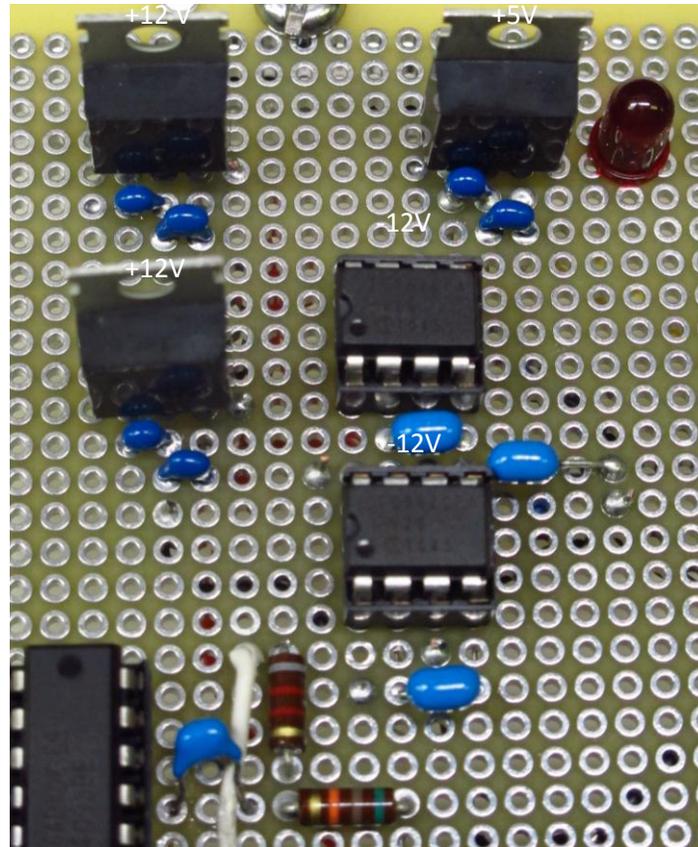


Figure 51: Soldered Power Circuit

Two positive regulators were used in the final design to ensure enough current was supplied to the positive and negative sides of the circuit. The top 12V regulator was used to supply the +12V rail that supplied the oscillators, op-amps, and +5V regulator. The bottom regulator was used exclusively to supply the -12V charge pumps, which in turn supplied the -12V rail on the board. Initial testing proved to have successful results, but after those initial tests the circuit never worked the same again.

This part of the circuit was by far the most temperamental part of the project. Testing this circuit at one point in time would be successful, but at the next it would be failure. The root of the problem stemmed from the circuit drawing unusual amounts of current from the power supply in the range of 0.8 amps. These problems would continue when the increased current draw would unknowingly burn up the charge pump chips, leaving the negative supply of the circuit dead. A solution to this problem unfortunately was not found, and without a properly working power supply, the project

would not be complete. In order to get the Theremin working effectively the laboratory power supply will now supply the negative rail.

6.5 Outcomes

The outcome of the power supply proved to be a failure. The -12V rail never worked and when it did it was only for a short amount of time. There is no way that this supply design could be used effectively in the final product and be expected to work every time. Time proved to be a factor when testing this part of the circuit as this was an unexpected roadblock for finishing our project.

6.6 Lessons Learned

There are many lessons to be learned from this part of the project. A decision was made to hold off on part ordering and testing until after Milestone I for this part of the circuit because it wouldn't be as difficult to get working as the mixer or the oscillator stages. The lesson here is being, the group should have ordered all circuit components immediately giving as much time as possible for testing because problems will always occur. Backup designs are absolutely necessary, because components don't always work how they should. Finally one should always expect the unexpected. The group didn't expect this circuit would cause so many headaches and take so much time as it did, but these are the lessons we learn from failure.

7 Enclosure

The enclosure for the Theremin is made of solid acrylic and houses the components of the Theremin. It measures 18 x 6 x 8 inches and is light enough that anyone who would want to play it can lift and move it if necessary. The .220 inch thick acrylic was cut out of a sheet of 2.5 x 3 ft. with a laser cutter with the help of Mr. Paul Rodriguez, shop manager of the Chemical Engineering machine shop. With Paul's help we were able to quickly and accurately cut out the sides, holes for case mounted components, and etch the artistic designs onto the box. The very precise holes were cut into the case so case mounted components such as volume control, power switch, antennas, and power supply could be accessed easily. The case is meant to be see-through so the audience will be able see the components working as the player plays the Theremin.

Each of the sides of the box has a particular detail to be aware of. The front facing side has the LSU electrical engineering logo in the middle of two tiger eyes that appear to look directly at the audience as you play the device. That design is shown below and was created by one of the group members, Gregory Hurst.

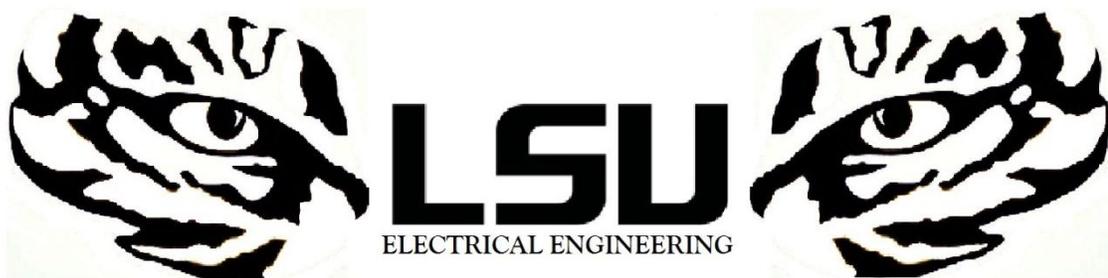


Figure 52: Front Face Logo

The backside of the device, which holds the speaker mounting, has a laser cut grill directly in the center to allow sound to emanate from the speaker mounted inside the box. The grill is a special design logo that was given to us by Mr. Rodriguez. Its main use was to be cut into circular storm drains used around campus, but was donated to the project for use as a speaker grill. A picture of the speaker grill is shown below.



Figure 53: Back Grill Logo

The last detail is on the top face looking down into the enclosure. The overall aim of the project is inspire students and have this be a learning experience for them. The group thought nothing better than having the circuit diagram etched on the top of the box so the students would be able to follow along the circuit diagram while seeing the actual components inside as well. The names of the group members are also etched along the bottom so the creators of the Theremin will go along with the device

wherever it may go. A picture of the top edge design is shown below and was created by another group member, Tim Alexander.

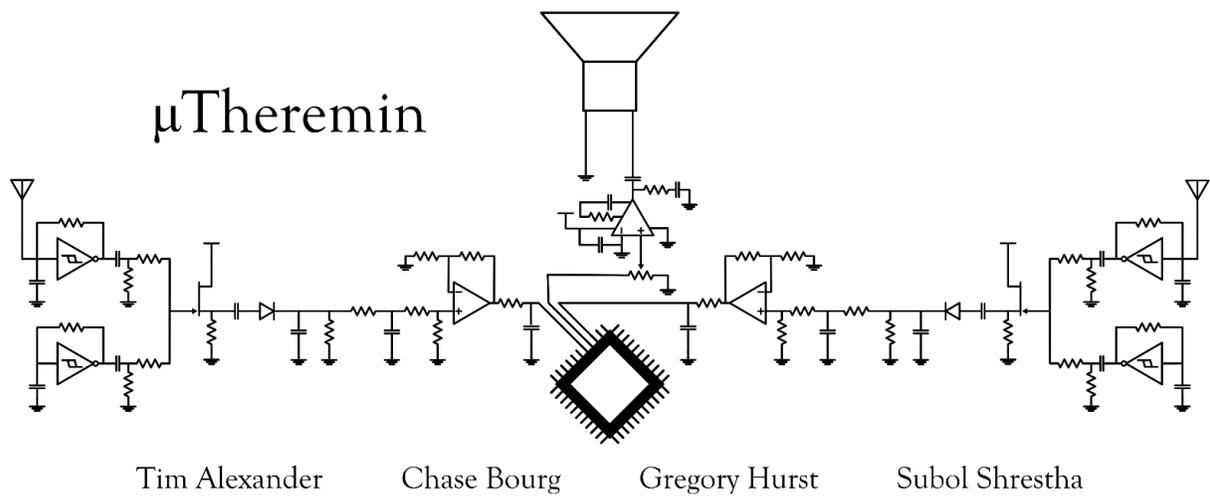


Figure 54: Top Face Logo

After all the sides were cut and etched there were glued together with the help of Mr. James Breedlove, Electrical Engineering shop manager. Special glue was applied to the sides being glued that chemically fused the two sides together creating a solid structure. The top face was left unglued and instead screw holes were drilled and tapped so to allow easy access to the inside circuit components if any troubles arise. After all the sides were glued and the screws were attached the final box came out spectacular and can be seen below.



Figure 55: Completed Box

8 Performance Outcomes

Goal	Measure of Success	Weight	%	Score
Stable Oscillations	Verifiable stable waveform	0.125		
Mixing/Filtering	Multiple waveform verifications	0.125		
Power Supply	DC voltage measurements	0.125		
Software Development	Frequency detection / Output	0.125		
Audio Amplifier	Amplify sound	0.125		
Service Learning	Assist high school students	0.125		
Enclosure	Build box for Theremin	0.125		
Total		1.0		

9 Printed Circuit Board

Printed Circuit Board was planned to be used because it is highly reliable and easily reproducible. Also they are inexpensive. Even though initial cost is higher than either wire wrap or point-to-point construction, but are much cheaper and faster for high-volume production. So after a thorough research on companies and software packages for PCB, it was decided that Express PCB should be used. It had close match to requirements that was set up. First of all, it is easier to design the layouts in its software package. Also the price of the PCB they offer is not that high compared to other companies with same products. Also they have user support, which assists the user to troubleshoot problems easily and quickly.

The most important thing that was found during the PCB design phase was the need to first make schematics and check for netlist errors before designing the PCB. Plus it was also realized that the schematics library should be vast, otherwise one needs to build the components and waste a lot of time. The sample schematic for mixer circuit from the Theremin is shown below.

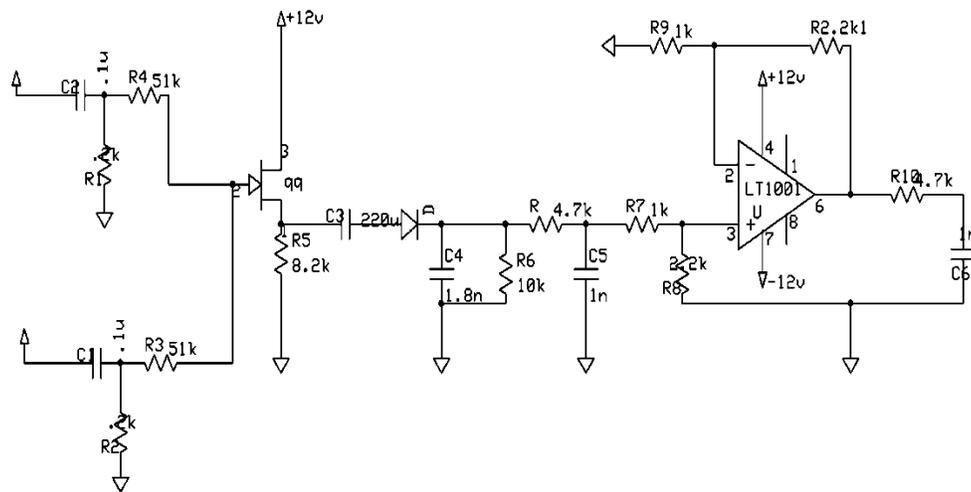


Figure 56: Mixer Schematic

While building this circuit, netlist error occurred continuously because of the JFET used in the circuit. Netlist error prompted to group all the components of the JFET. So to fix the problem, Express PCB customer support was contacted and they provided instructions to fix the problem. Steps to fix the grouping error are listed below.

- First ungroup the JFET using the 'ungroup component' listed in component menu.
- Then select all the terminal points in the JFET.
- Now again, click on the component menu. But now click on 'group to make component'.
- Now run the netlist check to check for errors.

It is easier to build one circuit at a time in schematics and implement it in PCB Express. As there would be a reference circuit for the integration of all the circuits in PCB Express. The vast library of components in Express PCB made the job easier. The final integrated circuit is shown below.

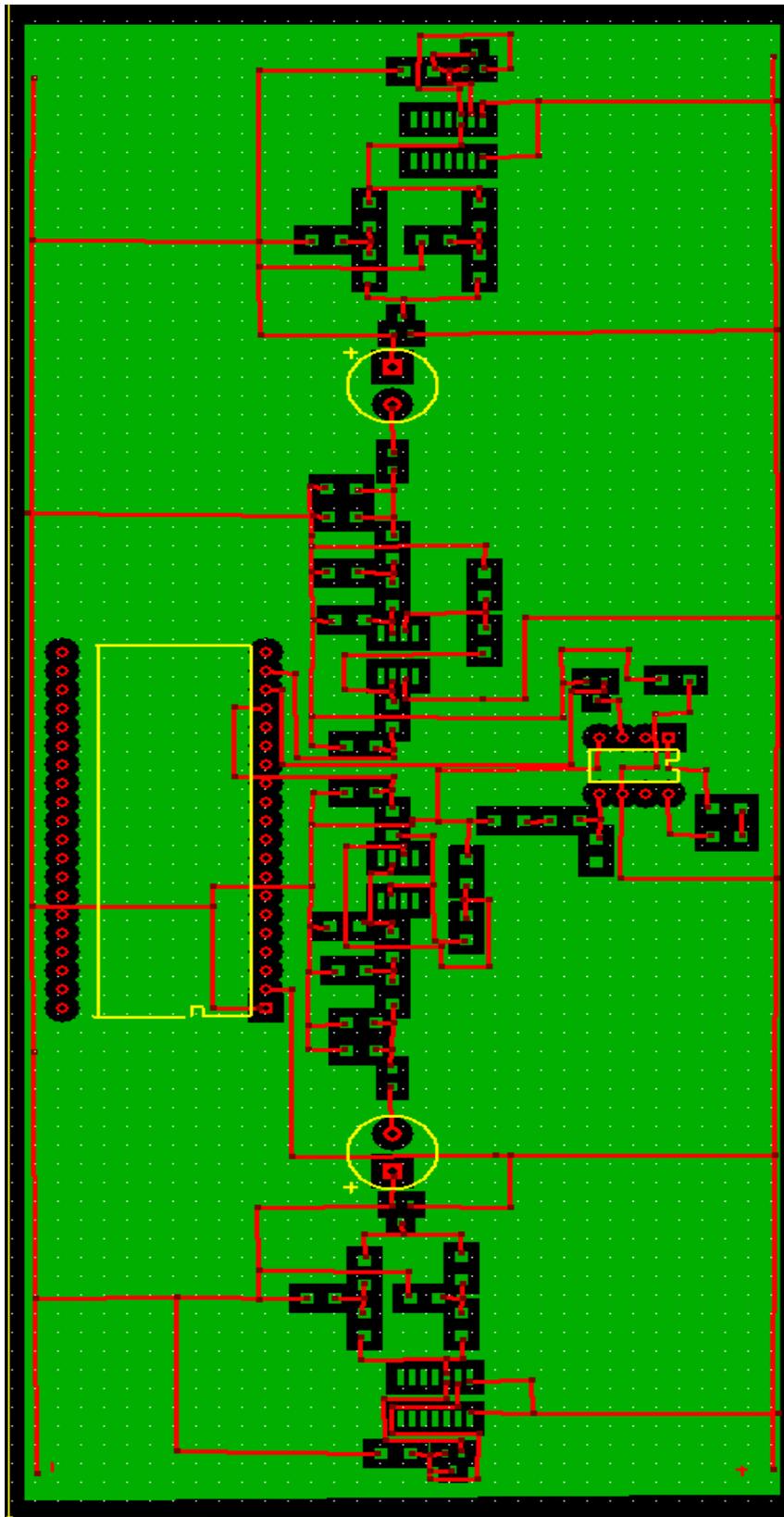


Figure 57: PCB Layout

One of the challenges that were encountered while designing the circuit was drawing lines that led to the components. It was also found that anyone can easily custom design their own components pad using custom pad layout. The custom pad layout was used to design the pads for the diodes used in mixer circuit. Even though Express PCB had similar component configuration in its library, exact match was not found. The diode packaging used in the circuit was SOD-123. So, another SOD package with close dimensions was chosen. Then the dimensions for the diode were configured according to the diode's datasheet.

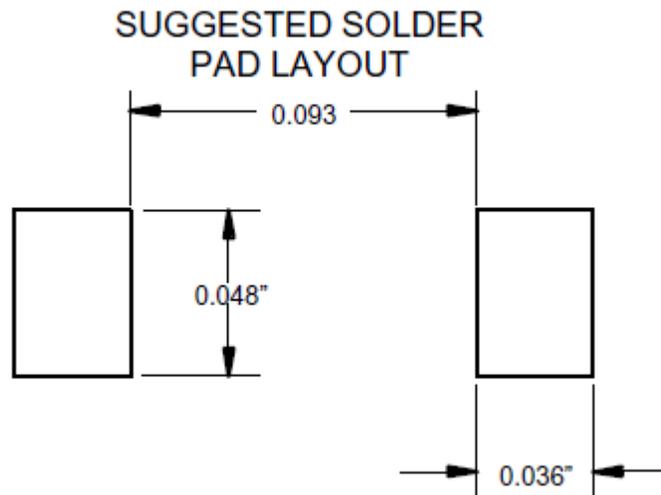


Figure 58: Diode Pad Layout

For instance, the left terminal of the diode that was used was located at 2.3" (X_1) from the x-axis and the right terminal was at 2.425" (X_2). But according to the diode's datasheet the maximum distance between two terminals was only 0.093". So using basic Algebra, the coordinates for the new X_1^* was found out.

Here,

$$X_2 - X_1^* = 0.093"$$

$$X_1^* = 2.332"$$

So, the pad which was at X_1 was moved to 2.332". Thus the pads for the components were easily built using the custom pad layout.

10 Design Economics

10.1 Budget Analysis

The total cost to build the whole Theremin circuit came out to be \$ 169.56. The components that were used to build the Theremin from scratch are listed below along with their price.

Quantity	Part	Value	Vendor	Perf Board Part#	Unit Price	Total
4	Female Header	20-Pin	DIGIKEY	N/A	\$1.60	\$6.40
10	Capacitor	10pF	DIGIKEY	SR151A100CAR-ND	n/a	\$5.79
4	Potentiometer	500K	DIGIKEY	3299Y-504LF-ND	\$2.87	\$11.48
4	Capacitor	1uF	DIGIKEY	478-5178-1-ND	\$0.62	\$2.48
4	Capacitor	2.2uF	DIGIKEY	478-1871-ND	\$0.71	\$2.84
4	Resistor	1.2k	DIGIKEY	OD122JE-ND	\$0.42	\$1.68
1	Power Switch	SPST	DIGIKEY	SW651-ND	\$2.03	\$2.03
8	Capacitor	220uF	DIGIKEY	P10384TB-ND	\$0.09	\$0.72
4	Capacitor	10uF	DIGIKEY	P5134-ND	\$0.20	\$0.80
1	Speaker	N/A	DIGIKEY		\$6.15	\$6.15
4	LM386	1 Watt	DIGIKEY	LM386N-4-ND	\$0.95	\$3.80
6	Screw Terminals	2 Pin	DIGIKEY	z	\$0.78	\$4.68
3	diodes	1N4148***	DIGIKEY	1N4148TACT-ND	\$0.14	\$0.42
4	capacitors	1800pF	DIGIKEY	490-3778-ND	\$0.32	\$1.28
5	resistors	10kOhms	DIGIKEY	OD103JE-ND	\$0.42	\$2.10
5	resistors	4.7kOhms	DIGIKEY	OD472JE-ND	\$0.42	\$2.10
5	resistors	1kOhms	DIGIKEY	OD102JE-ND	0.42	\$2.10
5	resistors	2.2kOhms	DIGIKEY	OD222JE-ND	\$0.42	\$2.10
2	Op-Amps	LT1001***	DIGIKEY	LT1001CN8#PBF-ND	\$3.30	\$6.60
12	capacitors	.1uF	DIGIKEY	490-3810-ND	\$0.32	\$3.84
7	resistors	8.2kOhms	DIGIKEY	OD822JE-ND	\$0.42	\$2.94
5	resistors	51kOhms	DIGIKEY	OD513JE-ND	\$0.42	\$2.10
10	N-channel JFETs	2N3819***	DIGIKEY	2N3819-ND	\$0.63	\$6.30
4	NAND GATE		DIGIKEY	568-5598-ND	\$0.55	\$2.20
1	PC Board		DIGIKEY		\$11.52	\$11.52
2	IC		DIGIKEY	TC962CPA-ND	\$3.48	\$6.96
1	LM7812	12V Regulator	DIGIKEY	LM7812ACT-ND	0.69	\$0.69
1	LM7805	5V regulator	DIGIKEY	LM7805CT-ND	0.69	\$0.69
1	33 μ F Capacitor	33 μ F	DIGIKEY	445-5289-ND	1.67	\$1.67
2	.33 μ F Capacitor	.33 μ F	DIGIKEY	445-5306-ND	0.35	\$0.70
1	.47 μ F Capacitor	.47 μ F	DIGIKEY	445-4807-ND	0.33	\$0.33
2	BNC Terminal	N/A	DIGIKEY	A24530-ND	\$3.38	\$6.76
1	Acrylic	36"X30"X0.22"	Home Depot		56.00	56.00
					Total	\$169.56

While the total cost to build the entire circuit on PCB was estimated to be around \$ 287.61. Also, PCB Express does not ship less than 2 PCB's, so there are actually 2 PCBs' in the list compared to one PC Board for the perforated board design.

Quantity	Part	Value	Vendor	PCB Part#	Unit Price	Total
4	Female Header	20-Pin	DIGIKEY	N/A	\$1.60	\$6.40
10	Capacitor	10pF	DIGIKEY	399-5540-1-ND	\$1.62	\$16.20
4	Potentiometer	500K	DIGIKEY	3269W-1-504LF-ND	\$5.64	\$22.56
4	Capacitor	1uF	DIGIKEY	478-5178-1-ND	\$0.79	\$3.16
4	Capacitor	2.2uF	DIGIKEY	399-5613-1-ND	\$1.09	\$4.36
4	Resistor	1.2k	DIGIKEY	P1.20KFCT-ND	\$0.10	\$0.40
1	Power Switch	SPST	DIGIKEY	SW651-ND	\$2.03	\$2.03
8	Capacitor	220uF	DIGIKEY	PCE3733CT-ND	\$1.46	\$11.68
4	Capacitor	10uF	DIGIKEY	493-2185-1-ND	\$0.53	\$2.12
6	Capacitor	.1uF	DIGIKEY	399-1250-1-ND	\$0.46	\$2.76
1	Speaker	N/A	DIGIKEY		\$6.15	\$6.15
4	LM386	1 Watt	DIGIKEY	LM386N-4-ND	\$0.95	\$3.80
6	Screw Terminals	2 Pin	DIGIKEY		\$0.78	\$4.68
3	diodes	1N4148***	DIGIKEY	1N4148WTPMSCT-ND	\$0.42	\$1.26
4	capacitors	1800pF	DIGIKEY	445-4155-1-ND	\$0.47	\$1.88
5	resistors	10kOhms	DIGIKEY	P10.0KFCT-ND	\$0.10	\$0.50
5	resistors	4.7kOhms	DIGIKEY	P4.70KFCT-ND	\$0.10	\$0.50
5	resistors	1kOhms	DIGIKEY	P1.00KFCT-ND	\$0.10	\$0.50
5	resistors	2.2kOhms	DIGIKEY	P2.20KFCT-ND	\$0.10	\$0.50
2	Op-Amps	LT1001***	DIGIKEY	LT1001CS8#PBF-ND	\$4.10	\$8.20
5	capacitors	.1uF	DIGIKEY	399-1250-1-ND	\$0.46	\$2.30
7	resistors	8.2kOhms	DIGIKEY	P8.20KFCT-ND	\$0.10	\$0.70
5	resistors	51kOhms	DIGIKEY	P51.0KFCT-ND	\$0.10	\$0.50
10	N-channel JFETs	2N3819***	DIGIKEY	MMBFJ310LT1GOSCT-ND	\$0.35	\$3.54
4	NAND GATE	CD4039BE	DIGIKEY	296-3524-5-ND	\$0.50	\$2.00
2	PCB		E- PCB		\$103.82	\$103.82
2	integrated circuit		DIGIKEY	TC962COE-ND	\$3.48	\$6.96
1	LM7812	12V Regulator	DIGIKEY	LM7812ACT-ND	0.69	\$0.69
1	LM7805	5V regulator	DIGIKEY	LM7805CT-ND	0.69	\$0.69
1	33 μF Capacitor	33 μF	DIGIKEY	445-5289-ND	1.67	\$1.67
2	.33 μF Capacitor	.33 μF	DIGIKEY	445-5306-ND	0.35	\$0.70
1	.47 μF Capacitor	.47 μF	DIGIKEY	445-4807-ND	0.33	\$0.33
2	BNC Terminal	N/A	DIGIKEY	A24530-ND	\$3.38	\$6.76
1	Acrylic	36"X30"X0.22"	Home Depot		56.00	56.00
					Total	\$287.61

10.2 Reproducibility

As shown by the tables above, the reproduction of these Theremins isn't necessarily cheap. Producing the circuitry on a PCB Board instead of a standard soldering breadboard adds about \$120 to the cost of production. It appears that it is still a manageable price for any professor or school that would like to buy and construct a few Theremins. Several cost cutting measures were taken during the design process so that the apparatus could be reproduced at a lower cost.

2 Service Learning

At the onset of this project, one of the main goals was to perform some form of community outreach with the local high schools. The goal was to be able to take the Theremin, bring it to some local schools, and demonstrate how it works according to the principles currently being learned in their physics classes. The original method of fulfilling this goal was to create a lesson plan that would accompany the Theremin demonstration. This lesson plan was going to show the transition from physics principals to basic electrical engineering principles. However, throughout the course of this project, it became clear that it was difficult to find a high school physics teacher that wanted to devote a class period to the presentation.

Because service learning was still an important part of this project that needed to be accomplished, when an alternative arose, it was taken. Mr. Scalzo arranged for a new aspect of service learning to be put in place. There is a group of student at Baton Rouge Magnet High School that is competing the Robotic Arm Competition portion of the National Science Olympiad. Each team in this competition has been instructed to design and build a robotic arm that will perform a series of tasks that will accumulate points in a competitive format. The service learning part of this Capstone project is to assist the BRMHS students whenever they need help. If they have any questions or concerns with their designs, then they will be addressed as a part of this project.

There are a set of rules that must be followed while constructing this robotic arm. One main rule is that at the beginning of the competition, the entire arm and its base must be contained in a 30cm x 30cm box, with unlimited height. There are also restrictions on the frequencies at which any remote controls can communicate with the arm. As far as construction goes, there aren't any major restrictions regarding the physical structure or mechanical operations of the arm.

The competition area for this arm will be an 80cm x 80cm square. This square is divided in halves called the North Zone and South Zone. At the centers of each the Northern, Eastern, and Western edge of the competition area, there will be a half of a milk jug, the goals. There will also be a bonus jug in either the Northeast or Northwest corner of the playing field. Along each edge of the 30cm x 30cm box known as the robot arm box, there will be 5 pencils, 5 PVC pipes, 5 nails, and 4 batteries in between the nails.

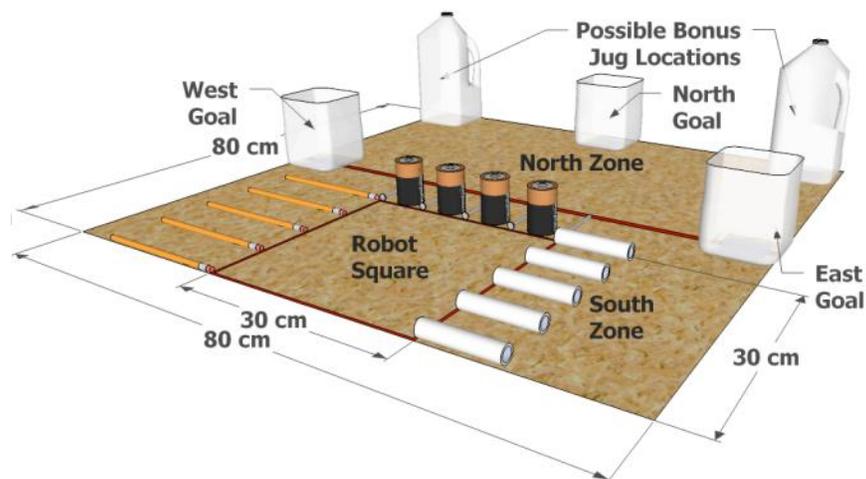


Figure 59: Robot Arm Competition Playing Field

When the competition starts, each team will have 3 minutes to accumulate as many points as possible. This is done by using the robot arm to grab each of the items that are situated around its base, and put them in certain goal boxes. The ideal goal boxes for the pencils, pipes, and nails are the West, East, and North goal boxes, respectively. However, one of each item should be put in the bonus jug. Also, points are given for no more than one battery placed in each of the East, West, and North goal boxes. Another point is given for putting the last battery inside the North Zone. If it comes down to a tie, the tiebreaker will be awarded to the team that has used the least amount of motors, actuators, etc.

3 Appendix