EE313 Analog Electronic Laboratory 2017-2018 Fall Term Project FMCW Based Distance Measuring System

1st Halil TEMURTAS *2094522* *halil.temurtas@metu.edu.tr*
2nd Erdem TUNA *2167419* *erdem.tuna@metu.edu.tr*

***Abstract*—Design of a Frequency Modulated Continuous Wave (FMCW) Based Distance Measuring System**

***Index Terms*—Radar, Oscillator, Amplifier, Fmcw, Mixer, Filter**

I. Introduction

In this project, it is aimed to design a frequency modulated continuous wave (FMCW) purposed on measuring distance. FMCW radar concept is used in wide range of applications such as cruise control, crash mitigation and pre-crash sensing [1]. Utilizing from waves with modulated frequencies, distance measurement is possible by finding the frequency difference between transmitted and received waves. There are several blocks constructing this radar system. Overall project diagram is presented in *Figure 1*. Each block will be discussed in detail in related sections.

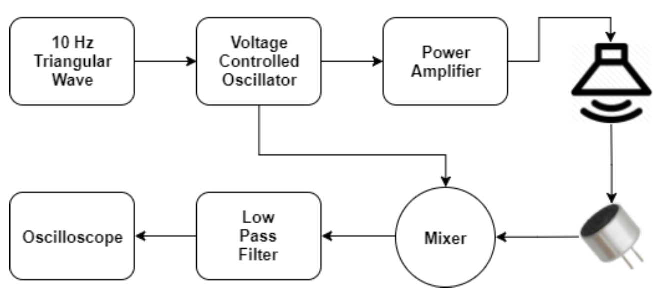


Figure 1: The Overall Block Diagram

II. Transmitter

A. Voltage Controlled Oscillator

Voltage controlled oscillator (VCO) is a voltage-to-frequency mapper. VCO outputs variable frequency voltage as the input voltage changes. The used VCO circuit in this project can be seen in *Figure 2*. To be able to generate frequency modulated signal that is the basic function of VCO, the main input to the circuit is a triangular wave. The triangular wave provides varying input voltage so that output frequency changes gradually. The modulated triangular wave is observed at the V_{ModTri} node. Charging and discharging phenomena of capacitor C_1 is the main reason of the oscillation at the V_{ModTri} node. While capacitor charges, modulated wave climbs down and as capacitor discharges through R_4, modulated wave climbs up. Capacitor charging-discharging cycles do effect the output frequency, however, are not the real reason of modulated triangular wave at the V_{ModTri} node. If there were a constant DC input, the output would be just an oscillating triangular wave with a certain frequency. The main reason for modulated frequency is the changing input voltage. The output frequency depends on three factors. The first one is the capacitance. As capacitance of C_1 increases, the ability of holding charge of the capacitor increases. This results in longer cycles meaning lower frequency at the V_{ModTri} node. Secondly, the resistance R_1 sets a barrier for current flow through C_1. In other words, as R_1 increases, the time for C_1 to charge up becomes longer. Hence longer wave cycles are observed again. Third and last factor is the input voltage value. A higher input voltage implies higher current through C_1. Thus, the time for C_1 to charge up becomes shorter. As a result, higher frequencied cycles are observed at the V_{ModTri} node.

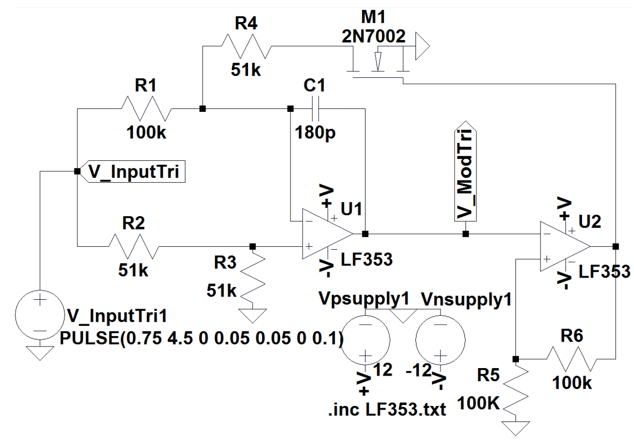
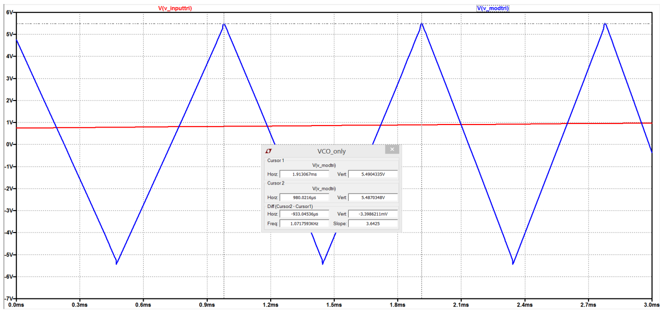
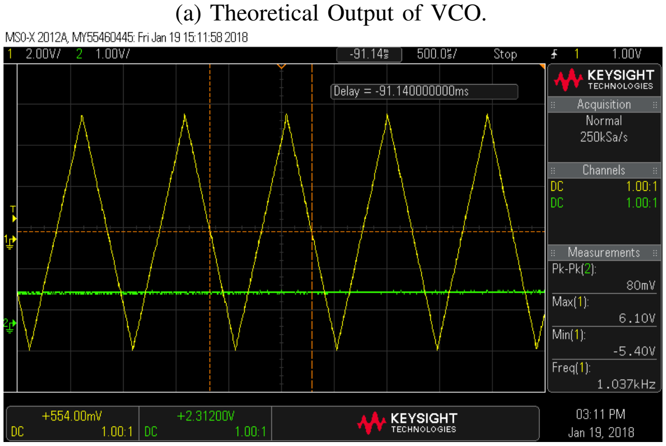


Figure 2: The Voltage Controlled Oscillator Circuit

To handle these frequent voltage changes, high slew rated LF353 [2] opamps are used. Also for biasing purposes, a N-MOS with low open voltage provides lower DC offset at the input $V_{InputTri}$. 2N7000 [3] model N-MOS suits for this application. Simulation and theoretical results for two corner frequencies are presented in *Figure 3* and *Figure 4*. VCO outputs are revealed as expected in theoretical results. The resistor and capacitor values are chosen initially by observing examples on websites [4] and later by trial and error. The ratio of $\frac{R_5}{R_5+R_6}$, which is the threshold voltage of the Schmitt

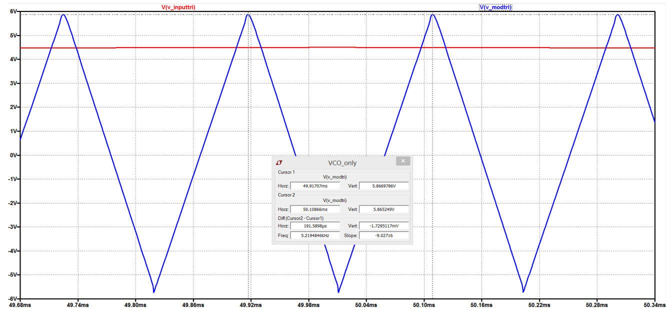


(a) Theoretical Output of VCO.

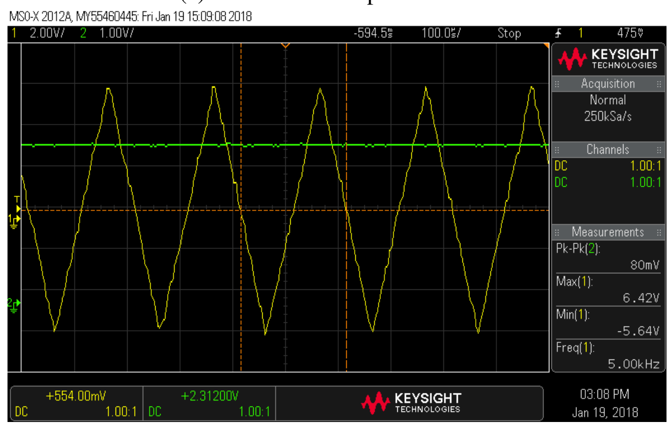


(b) Practical Output of VCO.

Figure 3: VCO Outputs for 1kHz.



(a) Theoretical Output of VCO.



(b) Practical Output of VCO.

Figure 4: VCO Outputs for 5kHz.

Trigger represented with U_2, sets the peak voltage of the V_{ModTri}. This voltage is $\sim 6V$ in our design.

B. Power Amplifier and Speaker

This part is aimed to transmit the frequency modulated signal, which is the output of the VCO, to a medium by using a power amplifier. A Class AB amplifier is utilized for this purpose. The amplifier is basically composed of two stages that are common emitter driver and AB amplifier.

The main transistor of common emitter driver is BDN_1. This transistor sets the DC biasing of BDN_2 and BDP_1. It may be considered as a current source. Also, the input signal is output as inversely polarized at the collector of the BDN_1.

The class AB amplifier mainly consists of two transistors one of which is a NPN and the other is PNP. The diodes D_1 and D_2 are to provide Q point stability of the transistors. Also they provide constant voltage difference between the bases of BDN_2 and BDP_1. When the voltage in positive cycle, BDN_2 amplifies the signal whereas BDP_1 amplifies the signal in negative cycle. These transistors operate cooperatively according to the cycle of the signal. Lastly, feedback resistor R_1 reduces the distortion of the output signal by introducing a negative feedback to the input signal in shunt-shunt topology. On the other hand, the DC biasing of BDN_1 is improved with the use of R_1.

The simulation results for power amplifier regarding current through and voltage across speaker is shown in *Figure 6*. The rms of triangular wave can be found by $\frac{V_{peak}}{\sqrt{3}}$. With this in mind, power consumed by the speaker is simply $\frac{4.3V \times 0.53A}{3} \approx$ 0.76 Watts. In breadboard, however, the circuit didn't function properly. In simulation, transistors were all in forward active region and BDN_2 and BDP_1 were satisfying $12V - 6V - 0V$ DC voltage biasing as in *Figure 7*.

III. Receiver

A. Microphone & Microphone Driver

To receive the sound signal coming from the speaker and turn it into a electrical signal, some kind of speaker should be

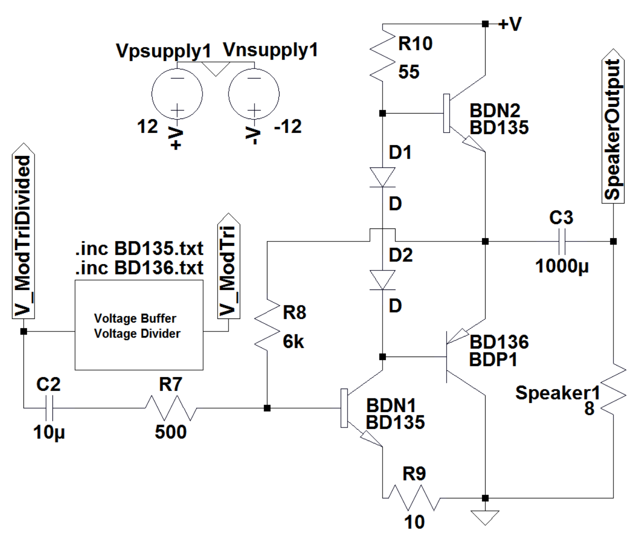


Figure 5: Power Amplifier Circuit Driving the Speaker

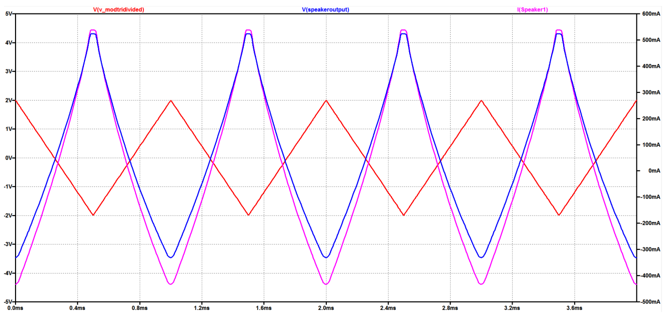


Figure 6: Power Amplifier Voltage Waveforms Across Speaker

Name:	q$bdp1	q$bdn2	q$bdn1
Model:	bd136	bd135	bd135
Ib:	-6.82e-03	1.55e-02	7.78e-04
Ic:	-7.23e-01	7.15e-01	8.63e-02
Vbe:	-7.27e-01	8.10e-01	4.29e-01
Vbc:	5.24e+00	-5.22e+00	-3.94e+00
Vce:	-5.97e+00	6.03e+00	4.37e+00
BetaDC:	1.06e+02	4.62e+01	1.11e+02
Gm:	1.75e+01	1.81e+01	2.97e+00

Figure 7: Power Amplifier Circuit DC Biasing Parameters

used. For this purpose, an electret microphone will be used.

To drive the electret microphone, a driver circuit is designed. The design should include feeding for the transistor inside the microphone and amplify the output signal since the output signal would be too small to be used. But before amplifying the signal, we shall remove possible dc offsets and noises by a passive high pass filter. The designed driver can be seen at *Figure 8*.

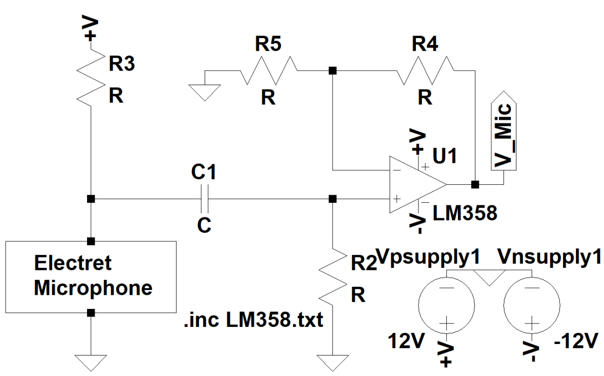


Figure 8: Driver Circuit for Electret Microphone

By node analysis, the lower cutoff frequency of the highpass filter can be found as

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

If we take $R_2 = 10k\Omega$ and $C_1 = 0.1\mu F$, cutoff frequency f_c can be found as

$$f_c = \frac{1}{2\pi * 10\ k\Omega * 0.1\ \mu F} \approx 159Hz$$

The resulting f_c will be enough to remove noise and DC offsets.

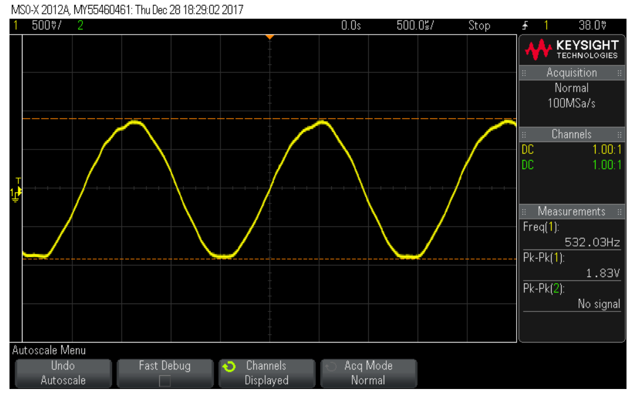


Figure 9: Output Signal of Microphone Circuit at Laboratory

The Output waveform for the microphone circuit for given sound signal at the laboratory can be seen at *Figure 9*.

B. Mixer

Mixer is the device that accepts two input signals and gives a output signal that consists of two distinct signals with different frequencies. While, one of these signals has a frequency that is equal to the difference between the frequencies of first signal and second signal, other signal has a frequency that is summation of the frequencies of first and second signal. In other words, if we assume input signal 1 has a frequency f_1 and input signal 2 has a frequency f_2. The output signal would look like

$$O/P \quad Signal \ = \ A(f_1 + f_2) + B(\mid f_1 - f_2 \mid)$$

where A and B are the signals having frequencies

$$f_A \ = \ f_1 + f_2 \ \ \&$$
$$f_B \ = \ \mid f_1 - f_2 \mid$$

respectively.

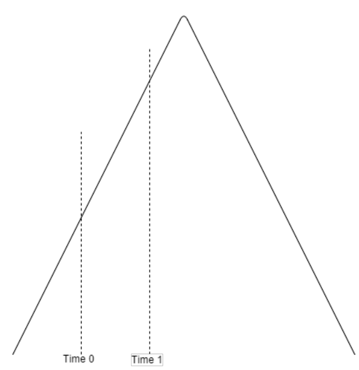


Figure 10: One Period of Triangular Wave

We also know that the distance between source and receiver causes a time delay proportional to the distance for the received signal in comparison to the transmitted signal. Thanks to triangular wave and the voltage controlled oscillator we used at the beginning of the project, the distance between the

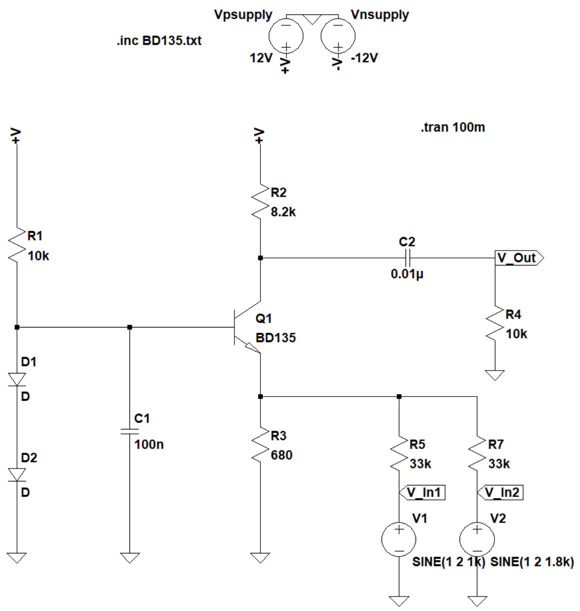


Figure 11: A Mixer Circuit

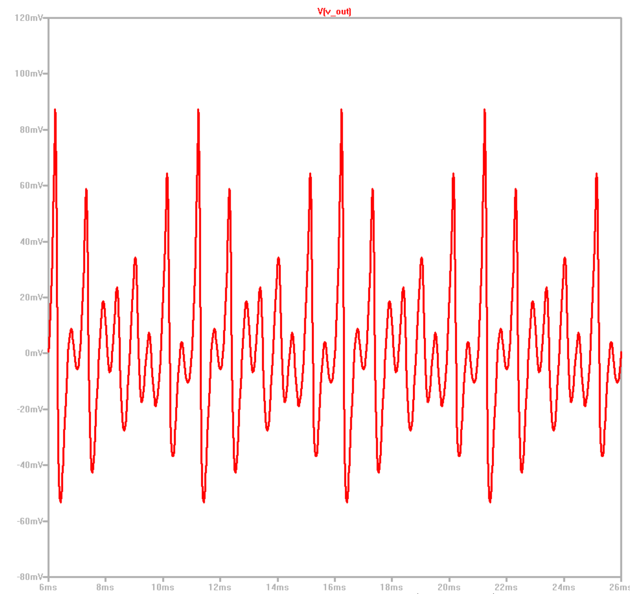


Figure 12: The Output Waveform of the Mixer Circuit

source and receiver is also proportional to the the frequency difference between original source signal and received signal by receiver. This can be understood from the basic principle of VCO. Assume that the first signal left the VCO and entered the mixer at Time 0 and has a frequency f_1 proportional to the magnitude of triangular wave at Time 0. Similarly second signal entered the Mixer at Time 1 and has a frequency f_2 proportional to the magnitude of triangular wave at Time 1. One period of the input triangular wave and "Time 0" & "Time 1" on top of it can be seen at *Figure 10*.

Therefore, the time shift " $Time\ 1 - Time\ 0$ " is proportional to the "$|\ f_2 - f_1\ |$". Thus, if we can find the frequency difference between this signal, we can easily find the desired distance since the distance can be found by

$$d \ = \ (Time\ 1 - Time\ 0) * v_s \ = \ K * |\ f_1 - f_2\ |$$

where K is a constant and v_s is the speed of sound.

The value of the constant $'K'$ can be found easily by considering the time that takes for the voltage controlled oscillator to span the half of the rectangular wave. In other words, the wave spans the 4 kHz frequency distance at half period, that is $\frac{1}{20}\ second$ for 10 Hz. Therefore, for any frequency difference to happen, the time takes can be found to be as

$$t \ = \ \frac{f * T}{2} * \frac{1}{4\ k} \ = \ \frac{f}{80} * 10^{-3}\ second$$

Thus, the distance between source and receiver can be by considering the time takes for any frequency difference and the distance sound wave can cover at that frequency, that is

$$d \ = \ v_s \ * \ t \ = \ \frac{v_s * f}{80} * 10^{-3} \ = \ \frac{343 * f}{80} * 10^{-3}\ meter$$

$$d \ = \ 4.2875 * f * 10^{-3}\ meter$$

Thus, the constant $'K'$ can be found as

$$K \ = \ 4.2875 * 10^{-3}\ meter$$

and the distance measured can be found as

$$d_{measured} \ = \ K * f_B \ = \ 4.2875 * f_B * 10^{-3}\ meter$$

where f and f_B are the unitless numerical values of the frequency difference.

For that reasons, a mixer circuit is a very crucial part of our project to be designed. A basic mixer circuit we have used can be seen at *Figure 11*.

Basic principal of mixer circuit can be understood from 'Fourier Transform'. Let us assume two periodic signal $x_1(t)$ with angular frequency w_0 and $x_2(t)$ with angular frequency w_0' that can be represented with 'Fourier Series Representations' like

$$x_1(t) = \sum_{n=0}^{\infty} a_n e^{jnw_0 t}$$

$$x_1(t) = \sum_{n=0}^{\infty} b_n e^{jnw_0' t}$$

that have Fourier Transforms $X_1(\Omega)$ and $X_2(\Omega)$

$$X_1(\Omega) = \sum_{n=0}^{\infty} a_n \delta(w - nw_0)$$

$$X_2(\Omega) = \sum_{n=0}^{\infty} b_n \delta(w - nw_0')$$

Assuming the output signal $y(t)$ equals to the multiple of two input signal, that is

$$y(t) \ = \ x_1(t) * x_2(t)$$

that has a Fourier Transform $Y(\Omega)$

$$Y(\Omega) \;=\; X_1(\Omega) \circledast X_2(\Omega)$$

where $\circledast$ is a basic convolution operation in jw domain. Thus, $Y(\Omega)$ can be written as

$$Y(\Omega) = \sum_{n=0}^{\infty} \sum_{n'=0}^{\infty} a_n b'_n [\delta(w - nw_0 - n'w'_0) + \delta(w - nw_0 + n'w'_0)]$$

The fact that the output signal has both summation of frequency information and difference of frequency information can be easily observed from the $Y(\Omega)$ expression. Therefore, if we investigate the Fourier Transform of the output signal, we can easily understand the validity of the designed circuit. In other words 'FFT' operation can be applied to the output signal. And if we observed impulses at the frequency difference and at the total frequency, we can conclude that mixer circuit is indeed working.

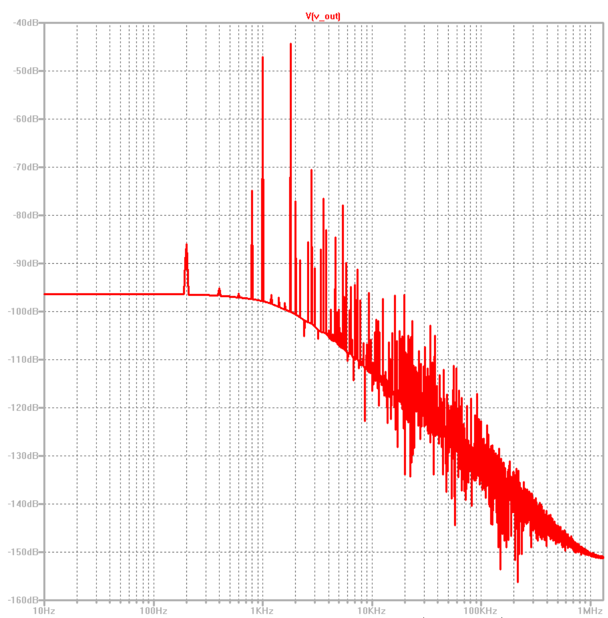


Figure 13: The Output FFT Spectrum of the Mixer Circuit

To test our design, we have used LTSpice for simulations. Applying two sinusoidal inputs from $33k\Omega$ resistor with $1\ kHz$ frequency & $1.8\ kHz$ frequency, we observed the output waveform at the *Figure 12*. To understand whether it works or not, the 'FFT Spectrum Analysis' was used. As can be seen from the *Figure 13*, we have observed impulses at the input frequencies, at difference frequency , at total frequency and their second and higher harmonics. Since we would use low-pass filter after the mixer, we were not worried about the higher order harmonic since they would be eliminated at the output anyway.

To test the design at the laboratory, we have used similar methods. By using two signal generators, we supplied the mixer circuit with two sinusoidal signal with different frequencies. The output waveform after that procedure can be seen at *Figure 14*.

After finding the proper mixer circuit, the only step-back from measuring the measuring the distance was designing a proper low pass filter. Since we wanted the filter to have sharp frequency response we have used second order low pass filter. The following section will explain the design steps of the filter.

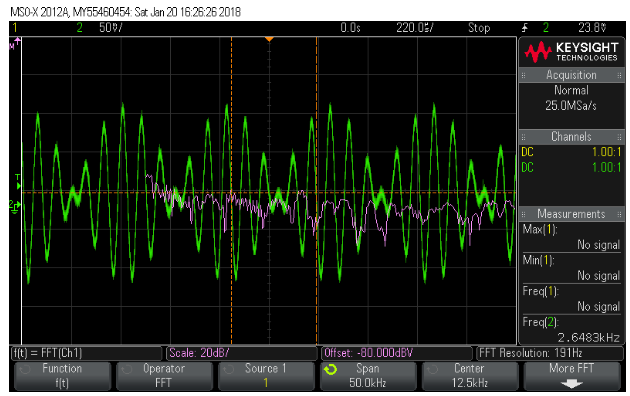


Figure 14: The Practical Output Waveform of the Mixer Circuit

C. Low Pass Filter

Unfortunately, the output signal of the Mixer did not only carry the frequency difference information but also frequency summation information. Since we are only interested in the difference between the frequencies only, a low pass filter was used to extract the wanted signal.

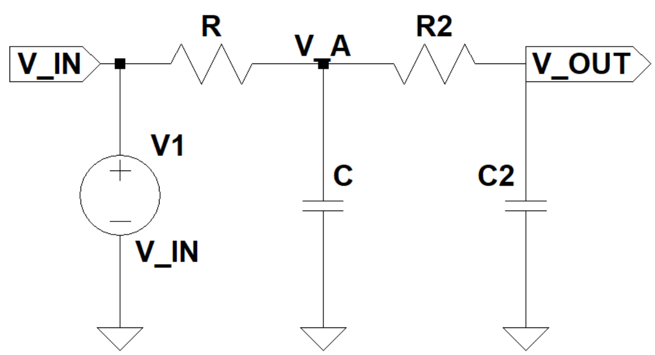


Figure 15: A Passive Second Order Low-Pass Filter

Low pass filters are the type of filters that pass the signals having the frequencies lower than the desired frequencies also known as the cut-off frequencies of the filter. Other signals having higher frequencies would be eliminated at the output of the low pass filter. Low pass filters can be categorized by their included capacitor number. For instance, if one low pass filter includes only one capacitor and resistor, it can be named as first order low pass filter. Similarly, a filter with two capacitor can be considered as second order low pass filter. In our design we preferred to use a second order low pass filter in order to get sharp enough frequency response. A basic second order low-pass filter can be seen at *Figure 15*

Low pass filters can also be categorized into two main type that are active filters and passive filters. Active filters are the ones that can amplify the desired signals and eliminate otherwise. These type of filters generally includes an op-amp for that purpose. A simple active second order low-pass filter can be seen at *Figure 16*. Whereas the passive low pass filters can only supply desired output signal with maximum unity gain. A general passive second order low-pass filter can be seen at *Figure 15*.

After considering the active one, we have decided using a passive second order low pass filter at *Figure 15*. For choosing proper resistance and capacitance values, the some KCL and KVL operations can be conducted on the circuit at S-Domain.

$$V_A = \frac{V_{in} * R}{R + \frac{1}{sC}//(R_2 + \frac{1}{sC_2})}$$

$$V_{out} = \frac{V_A * \frac{1}{sC_2}}{R_2 + \frac{1}{sC_2}}$$

Transfer function being

$$H(s) = \frac{V_{out}}{V_{in}}$$

Cut-off frequency can be found by rearranging terms into wanted form. If we equate $R = R_1$ & $C = C_1$ for the simplicity, after some effort the cut-off frequency can be found as

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

The resistance values can be found from there by assuming the capacitor values for desired frequency. For

$$f = 1.5\ kHz = 1.5 * 10^3\ Hz$$

$$\&$$

$$C = 10\ nF = 10^{-8}\ F$$

Resistance R can be found as

$$R = \frac{1}{2\pi * 1.5 * 10^3 * 10^{-8}} \approx 10\ k\Omega$$

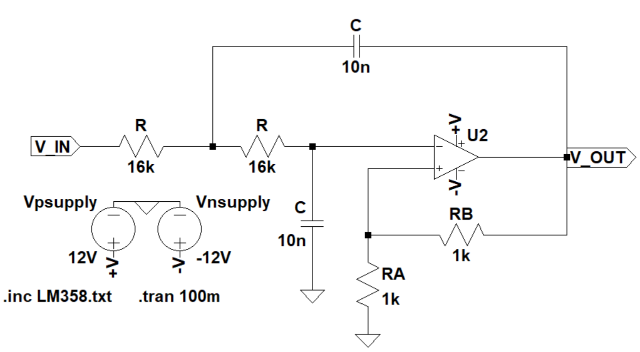


Figure 16: An Active Second Order Low-Pass Filter

To test the design of the low pass filter, firstly LTSpice was used as a simulation tool. As can be seen from the *Figure 15*, a voltage input applied as an input and output is observed. For three different sinusoidal input, the input and output waveforms can be seen at *Figures 17, 18, 19*.

Similar methodology was used at the laboratory. By giving sinusoidal inputs with different frequencies from signal generator, the outputs were observed through oscilloscope. The *Figures 22, 20 and 21* shows the input and output waveforms for the low pass filter having approximately $4\ kHz$ cutoff frequency that we designed earlier at the design step. Unfortunately, we forgot to take similar screenshots for the latest version of our filter.

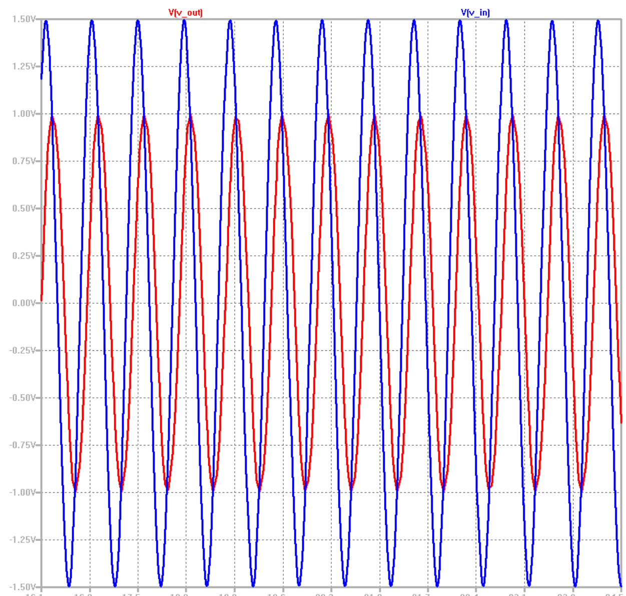


Figure 17: The Input & Output Signals for 1.5 kHz

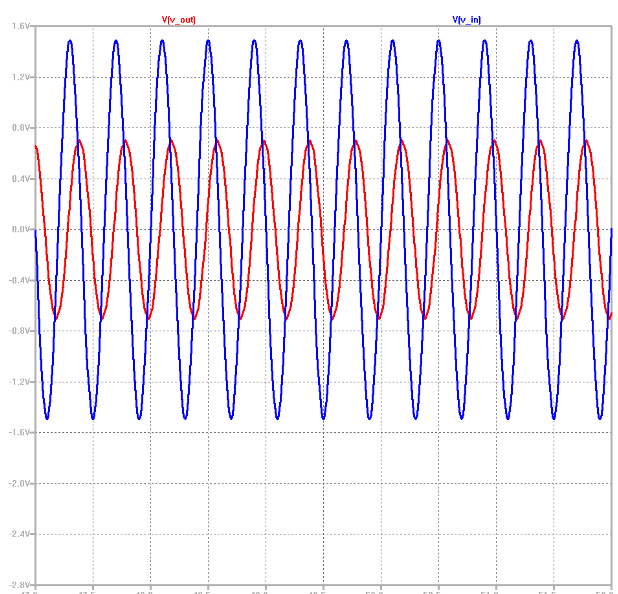


Figure 18: The Input & Output Signals for 2.5 kHz

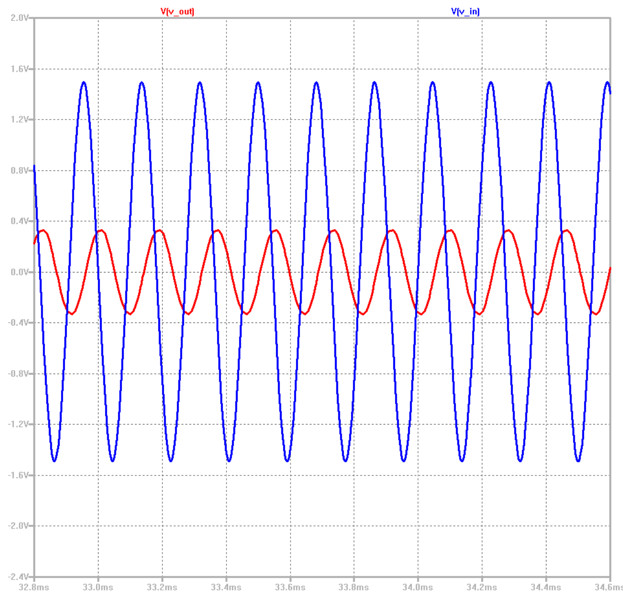


Figure 19: The Input & Output Signals for 5 kHz

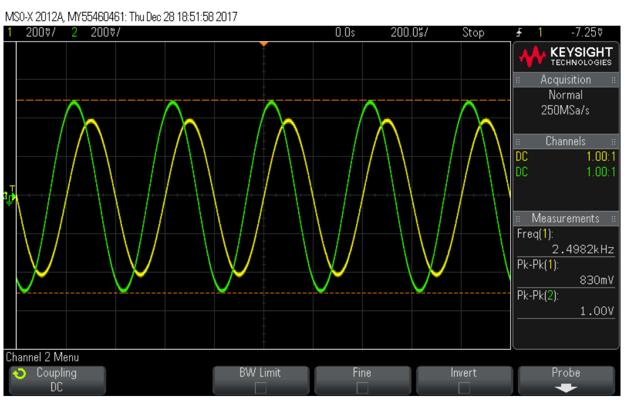


Figure 20: Practical Input & Output Signal for 2.5 kHz

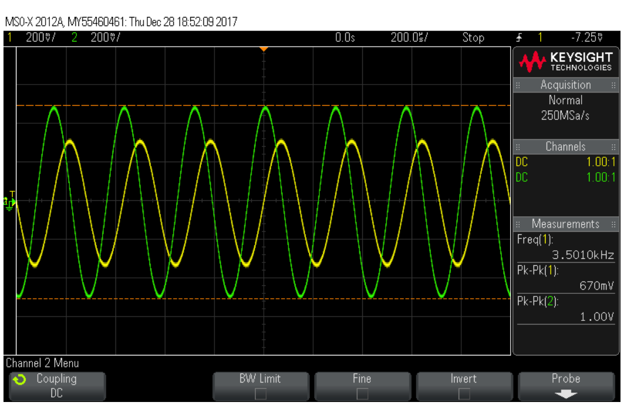


Figure 21: Practical Input & Output Signal for 3.5 kHz

IV. Overall Project

Overall project built at the laboratory on breadboard which has a schematic representation in *Figure 1* can be seen at *Figure 23*.

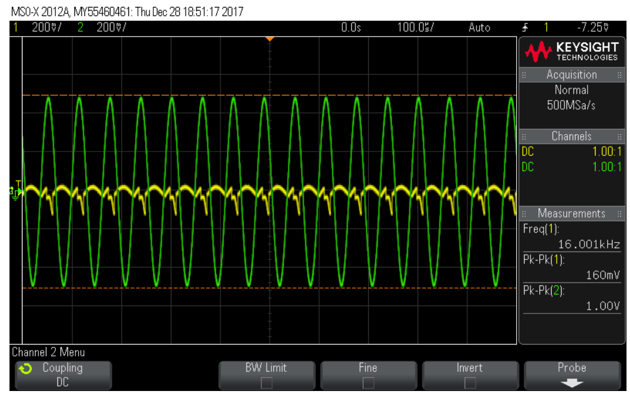


Figure 22: Practical Input & Output Signal for 16 kHz

V. Conclusion

Firstly, by using the 10 Hz triangular input from signal generator as an input, a triangular wave with varying frequency was generated with voltage controlled oscillator. This modulated triangular wave had a frequency varying between 1 kHz and 5 kHz. Then, this signal is sent to a speaker by amplifying it through a power amplifier we designed. Unfortunately this amplifier failed to do its job at the laboratory. The same signal from the VCO also sent as a first input of the mixer circuit. In the second part of our project, the sound signal sent through 8Ω speaker was received by the electret microphone and converted to a meaningful voltage by microphone driver circuit. This voltage signal was used as a second input signal for the mixer circuit we designed. Therefore, there are two inputs with a phase shift to be mixed. The signal at the output of the mixer actually consisted of two distinct signal with two different frequency. One of these signal had a frequency that equals to the sum of two input signals' frequencies in other words $(f_1 + f_2)$ and other signal had a frequency that equals to the difference between the frequencies of the input signals that is $(\mid f_1 - f_2 \mid)$. Since we were interested in only the frequency difference info, a second order low pass filter was used. By observing the signal at the output of low pass filter, we can measure the frequency difference $(\mid f_1 - f_2 \mid)$. Thus, the distance between the speaker and the microphone can be found by simply multiplying that difference with a constant $'K'$ that we calculated at this report.

References

[1] J.-J. Lin, Y.-P. Li, W.-C. Hsu, and T.-S. Lee, "Design of an FMCW radar baseband signal processing system for automotive application," SpringerPlus, vol. 5, no. 1, 2016.

[2] "LF353 Datasheet." [Online]. Available: http://www.ti.com/lit/ds/symlink/lf353-n.pdf. [Accessed: 20-Jan-2018].

[3] "2N7000 Datasheet." [Online]. Available: https://www.onsemi.com/pub/Collateral/2N7000-D.PDF. [Accessed: 20-Jan-2018].

[4] "LF353 Datasheet." [Online]. Available: http://www.falstad.com/circuit/e-vco.html.

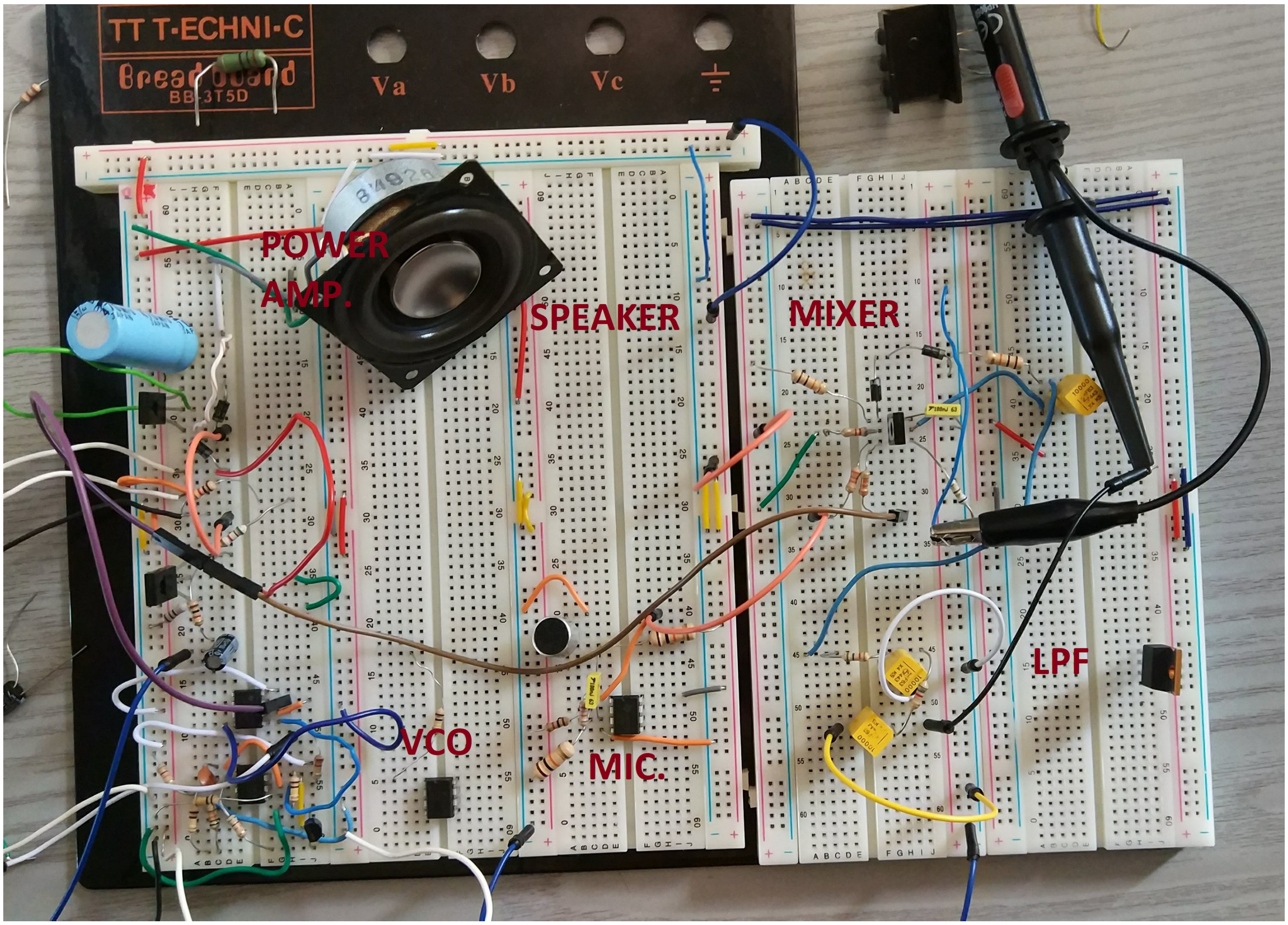


Figure 23: Overall Project