EEE 202 Hardware Report #2
Ayhan Okuyan
21601531
EEE202-03

Hardware Report 2

Introduction

The main objective of this laboratory assignment is to be able to create and use various kinds of OPAMPS and their relations with RC circuits in order to create a desired waveform. The design expects the input of a step function converting from 5V to 0V or a square wave between the same voltages. Furthermore, the output is expected to be a trapezoidal shape given in the figure below. Thus, the objective is this assignment is to design and implement a circuit that meets the expectations with the elements that we are to use.

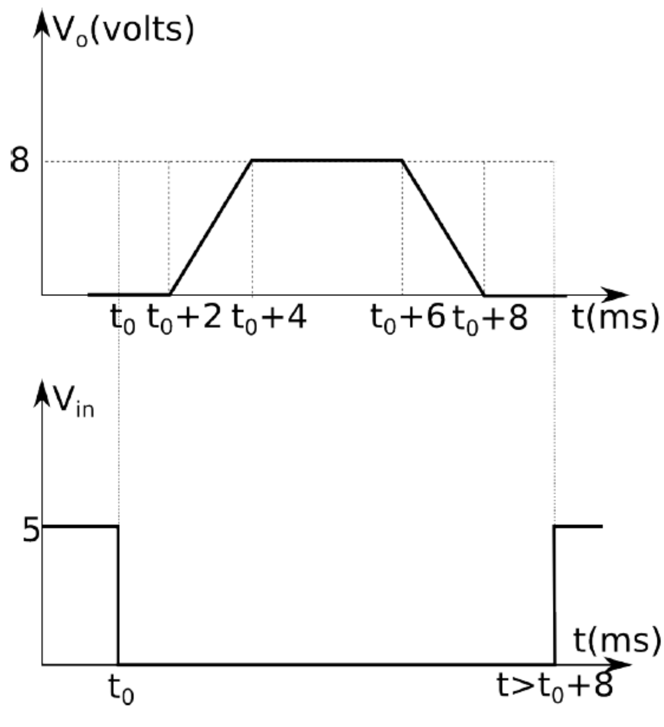

Figure 1 – The expected design limitations.

In order to be able to design the intended circuitry, I needed to do some research on some of the concepts that I thought I needed to use. The research topics are the OPAMP (Operational Amplifiers) types and RC circuits.

OPAMPS are electronic voltage amplifiers that can be set up to do various types of operations on the input waveform. In the assignment, and through this course, we are given LM324 coded OPAMPS which are low-quad (DIY) operational amplifiers. The basic idea behind each OPAMP is multiplying the voltage difference between its positive and negative sides and applying the operated voltage as an output to the output pin. Many arithmetical operations can be done using the OPAMPs Operational Amplifiers, such as inverting, subtracting, differentiating and integrating. Also, there are two applied voltages on the OPAMP which are Vcc and Vee that determine the saturation voltages of the OPAMP circuit. These are also the voltage supplies for the physical component.

When we move onto the other topic, which are RC Circuits. RC circuits here in this assignment is to create the desired delays, together with creating linearly increasing and decreasing waveforms. Thus, further investigation on the RC time constant is made to understand the concept better. However as, we only use one capacitor, the time constant will be degraded almost all the time to the total resistance of the RC Circuit multiplied with the capacitance.

After these investigations are made, the next part is to create a design scheme and try to achieve what is intended slowly by trying different configurations on LTSpice. As we do in each software lab, we will be using LTSpice to simulate the behavior of our circuit.

After the calculations and the necessary tests are commenced, the next stage is to create the circuit physically and test the design. Therefore, for the hardware laboratory, I created the circuit below in order to connect it and test it. The picture of the circuit is given below.

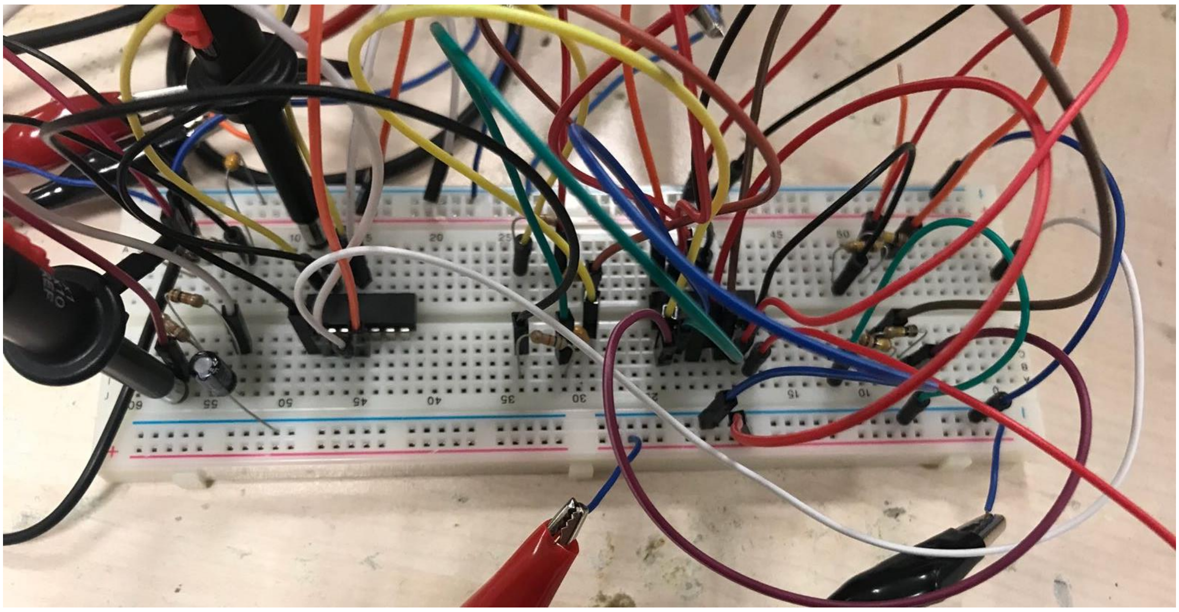

Figure 2 – The Circuit Schematics

The circuit is exactly the same when compared with the designed circuit. Through the left, a probe is seen to be put on the input in order to see through the input voltage at all times while a second probe is used to see the needed voltages. Furthermore, the DC generators and the signal generator used is seen below.

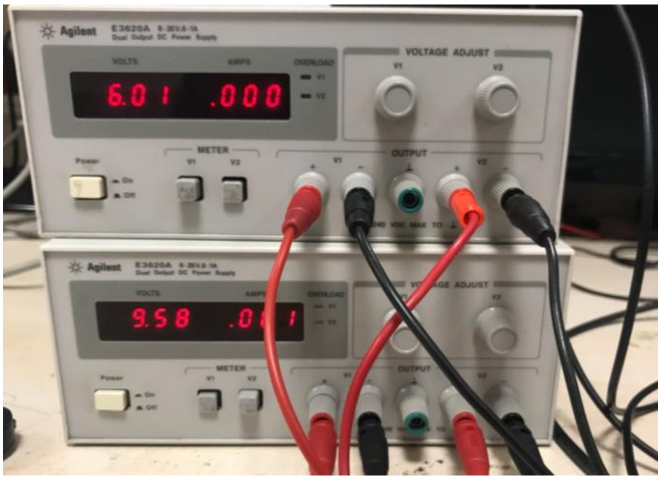

Figure 3 – The DC Generators

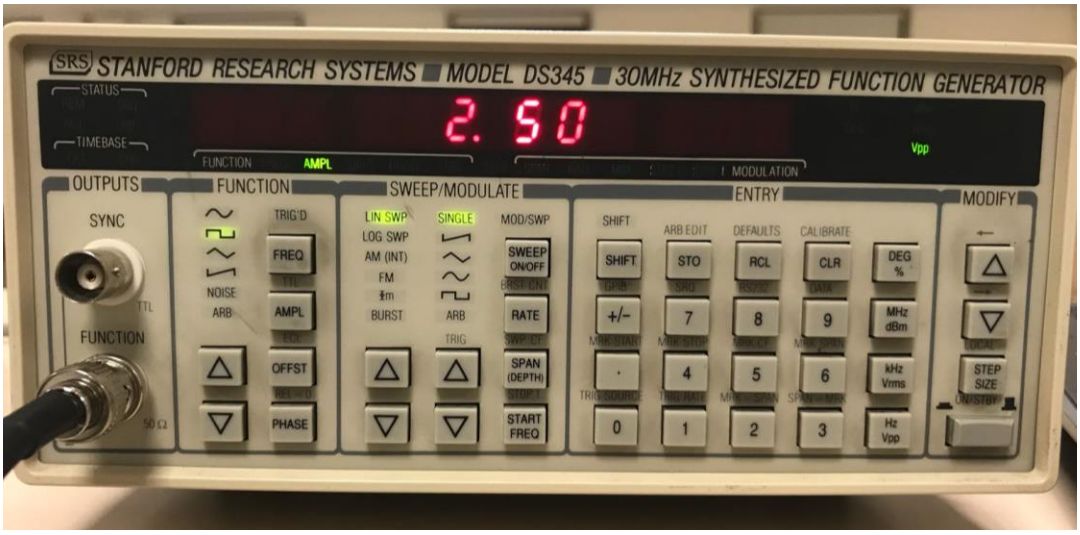

Figure 4 – The Signal Generator

Design

After certain trials, the final conclusion on the shape of the circuit is a two-branch circuit, one is to be able to increase the voltage as pleased and another one to bring it down. I will be explaining the functionality of one branch and then implement the results to the other one directly.

<u>Branch Design</u>

- **Structure of the Comparators**

In order to create a delay form $t=t_0$ to $t=t_2$ we need to use a comparator OPAMP which contains an RC circuit. If we look at figure 1, we can easily see that we need a 2ms of delay until the rise of the trapezoid shaped circuit. The circuit schematics for the comparator circuit is given in figure 3, however, for all the voltage sources and their schematics that will be used through this lab assignment is given below in figure 2.

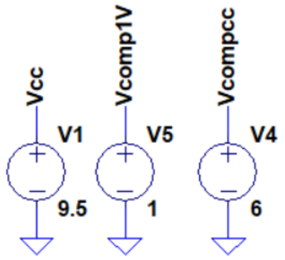

Figure 5 – The Voltages that will be used throughout the circuit labeled.

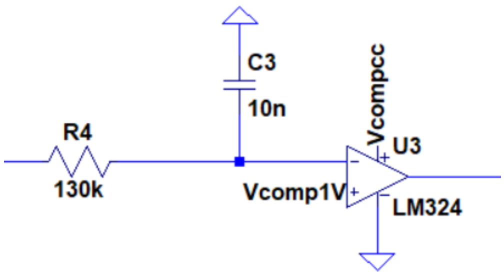

Figure 6 – Comparator Circuit for the First Branch

In this circuit, the reference voltage is decided as 1V. When the capacitor is discharged to 1V, the output voltage becomes equal to the supply voltage. In which overall gives us the opportunity to create any amount of delay based on the RC time constant. The equation for a comparator OPAMP is given below.

$$V_c = V_{in} * e^{-\frac{t}{R*C}}$$

As V_c is equal to the comparator voltage, we can simply give it 1V. V_{in} on the other hand is 5V as it is the input voltage that is applied by the generator. Thus, the equation becomes

$$1 = 5 * e^{-\frac{2*10^{-3}}{R*C}}$$

$$R * C = 1242.67 * 10^{-6}$$

However, there is one important fact that we must not dismiss. The OPAMP that we are going to use is naturally not an ideal OPAMP, Hence, the calculations are not exactly matching with the calculations and the values assigned to the simulation circuits have some miner alterations that differentiate the values from the theoretical calculations, but the base reasoning are the theoretical calculations. Thus, the values chosen for the R and C are 130KΩ and 10nF respectively. When these values are given, the input and output voltage graph for the comparator circuit is given below.

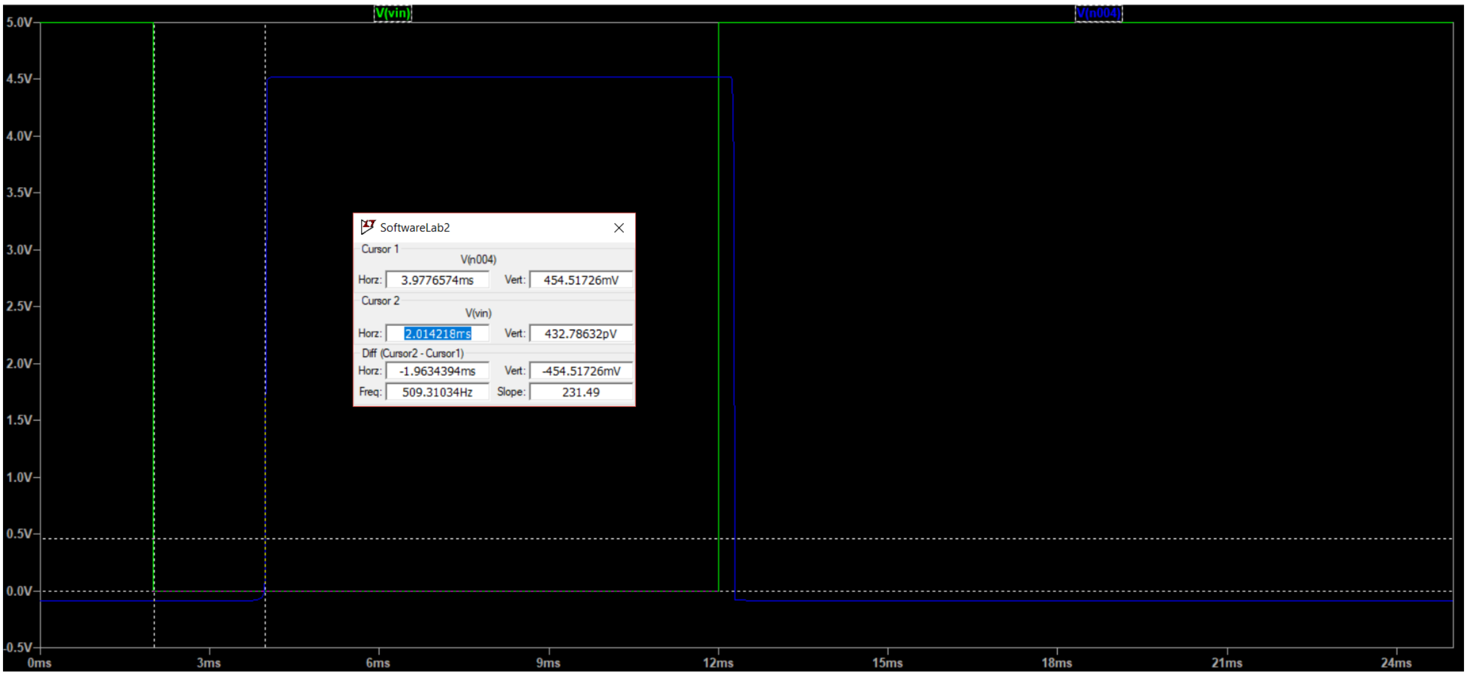

Figure 7 – The Graph of the Comparator Circuit in First Branch

The green line shows the input voltage while the blue line is the comparator output. We can easily see that, with the help of the cursors, the delay that we were able to create was 2.01 seconds, which is exactly as we desired.

Hardware Implementation of 1st Comparator

The hardware implementation of the first comparator went pretty smoothly as foreseen. The oscilloscope view of the output of the comparator and the input voltage is shown below.

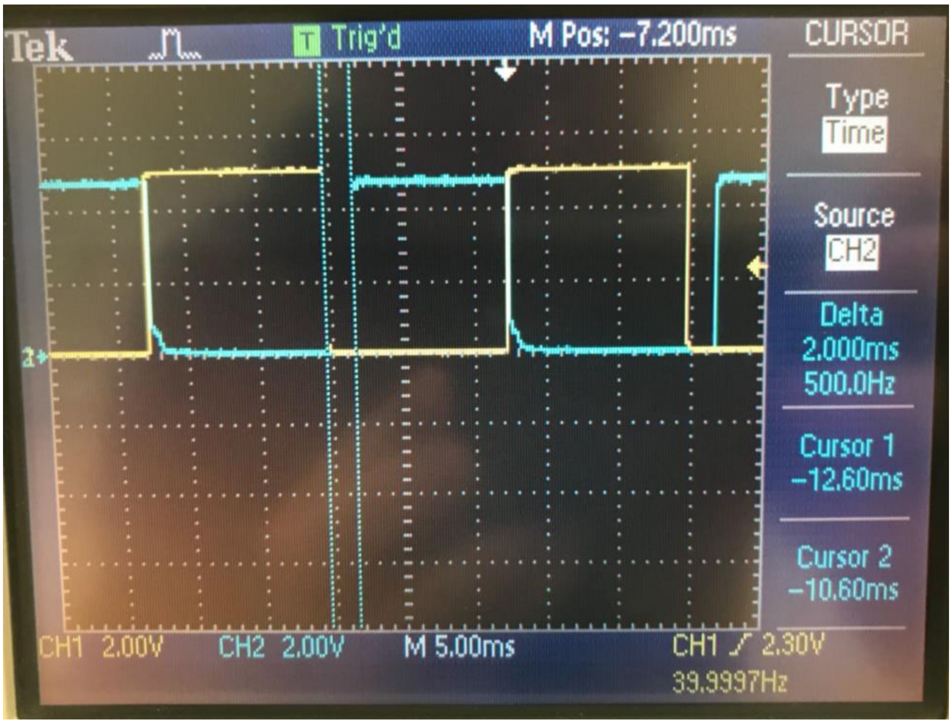

Figure 8 – The Oscilloscope View of the 1st Comparator

For further analysis, the comparator of the second branch needs a delay that will last from $t=t_0$ to $t=t_8$, 8ms to be precise. In order to do that, we run the same calculations that we have previously, and found the R and C to be 123KΩ and 33nF. The circuitry is given below.

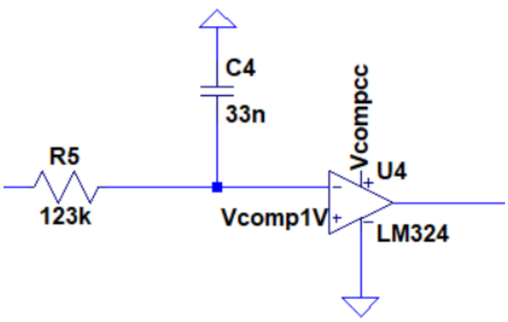

Figure 9 – The Comparator Circuit for the Second Branch

The voltage characteristics graph is given below, in figure 6.

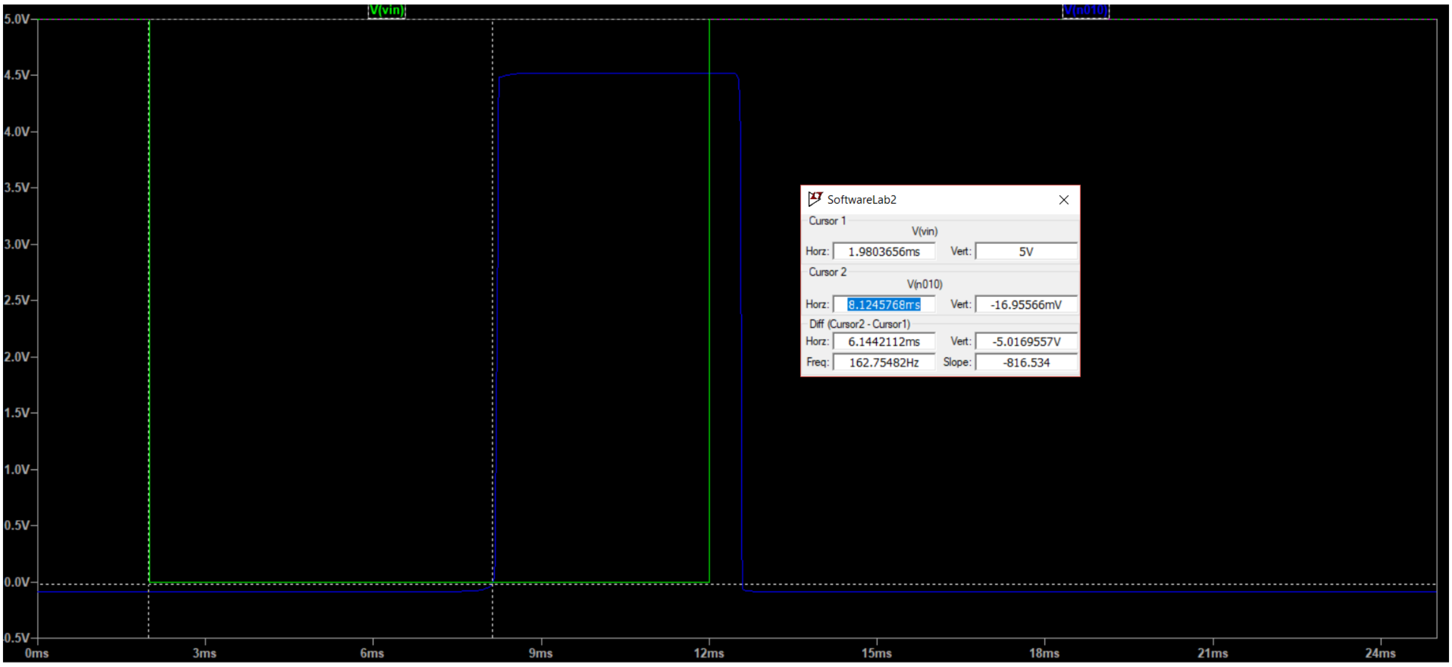

Figure 10– The Graph of the Comparator Circuit in Second Branch

Similar with the first branch, we have found the delay to be equal to 8.12ms which is nearly equal to the desired value as the green voltage represent the input voltage while the blue represents the output. This concludes the design of the comparators. The next stage is to secure the linear increase after the delay, for that, we should use an integrator OPAMP circuit.

Hardware Implementation of 2nd Comparator

The hardware implementation of the second comparator also went well. The oscilloscope view of the output of the comparator and the input voltage is shown below. As can be seen there is 6ms delay from the beginning of the cycle, which is what we were looking for before.

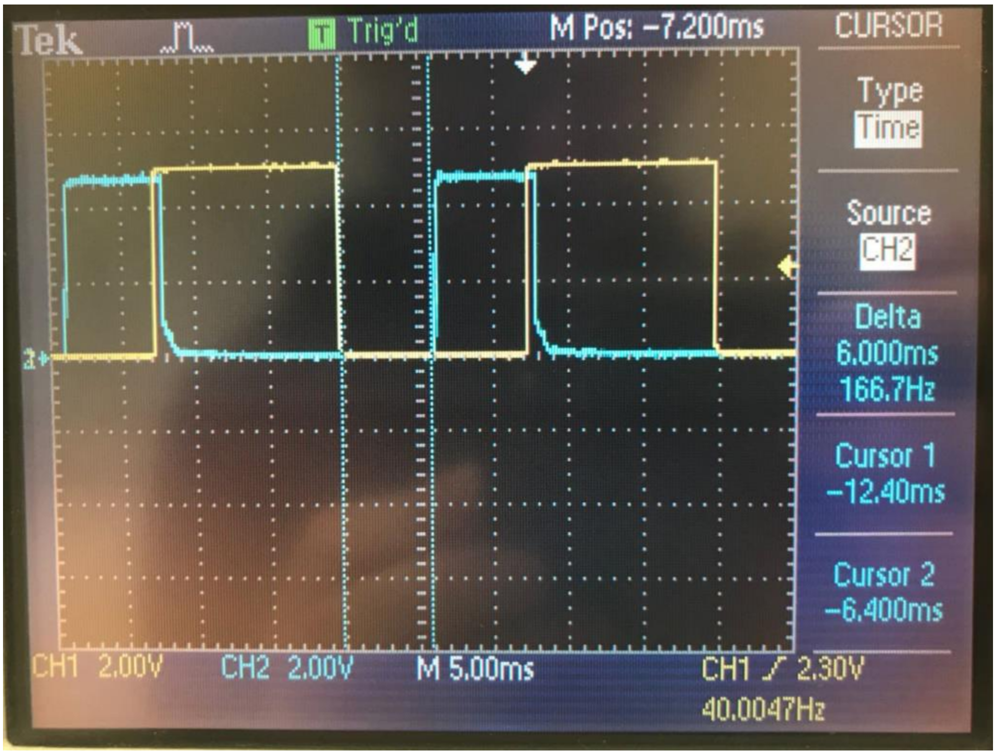

Figure 11 – The Oscilloscope View of the 2nd Comparator

- **Structure of the Integrators**

As I have mentioned, the integrator is the part of the circuit that creates the linearly increasing or decreasing voltage waveform. In order to achieve this, we need to investigate how it works. The schematics for an integrating OPAMP is given below.

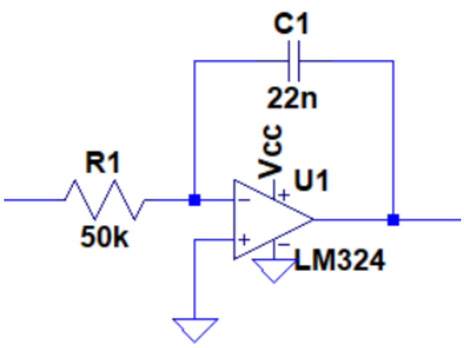

Figure 12 – The Integrator Circuit for the First Branch

As we can observe, there also is an RC Circuit embedded in the structure. Hence, we can write the equation for this circuit as shown below.

$$V_{out} = -\frac{\int V_{in} * dt}{R * C}$$

However, this equation in this form can not be considered as useful enough for us to get the needed results. In order to, make it more useful, we take the derivative of both sides with respect to t and the equation becomes

$$\frac{d(V_{out})}{dt} = -\frac{V_{in}}{R * C}$$

Thus, this means that the slope of the integrated circuit is given in the right side of the equation. The Vout is obviously the voltage that we should obtain after the operation is done which is 8V as can be seen from Figure 1. Vin is the input voltage which is 5V, and the time difference from t_{0+2} to t_{0+4}, which is 2ms. Then we can calculate the RC time constant as

$$\frac{-8}{2 * 10^{-3}} = -\frac{5}{R * C}$$

$$R * C = 1.25 * 10^{-3}$$

Using the found time constant and some minor trial errors, we found the R and C values as 50KΩ and 22nF. Then we simulate the circuit in LTSpice, and the graph is given below.

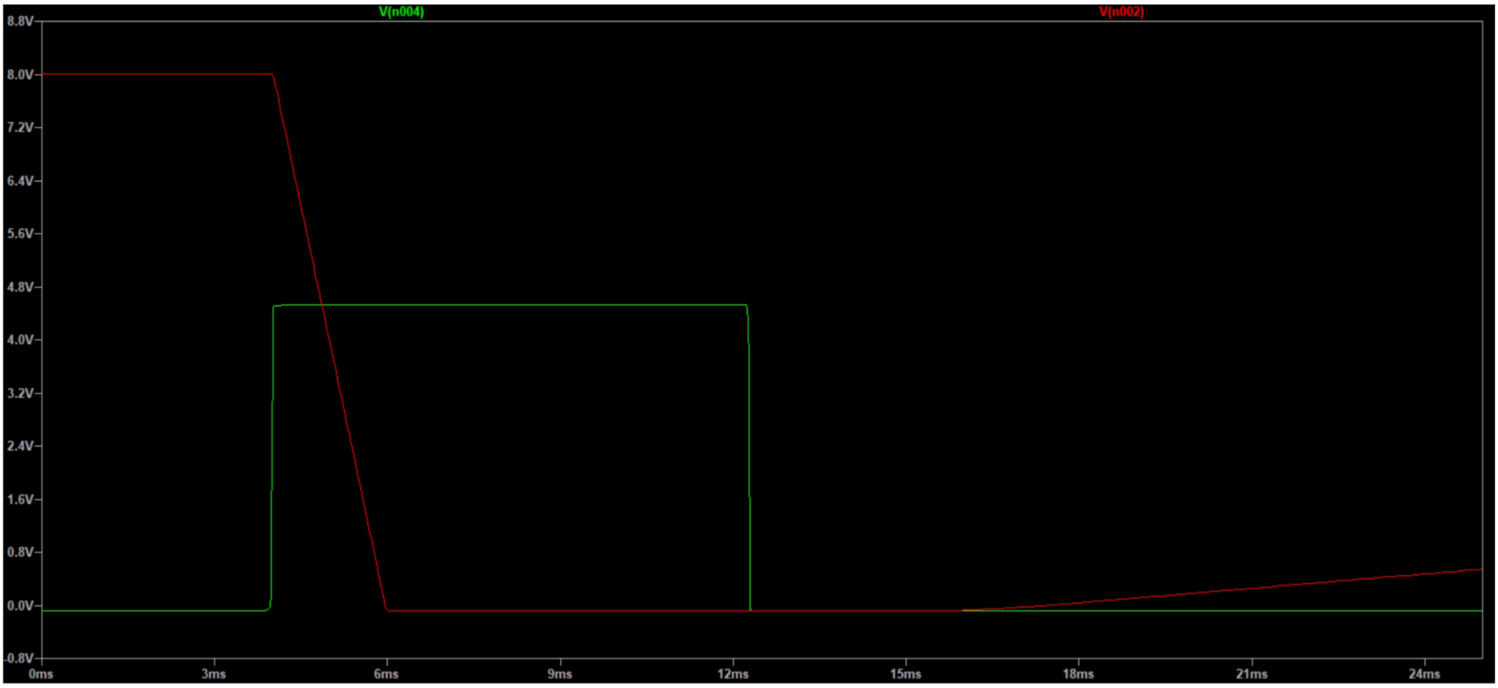

Figure 13 – The Graph for the İnput and Output Voltages of the Integrator Circuit

In this graph, green line shows the input voltage, which is also the output of the comparator circuit and the red line represents the output voltage. As thought, we see a change of 7.95V in 2ms. The real value for the time difference is 1.97ms as it was found by using cursors.

Hardware Implementation of the 1st Integrator

The integrator circuit for the first branch also seemed to come out as designed in the LTSpice. The cursors below show that the width of the decline to be 2ms as it drops 4V which is what was expected. The oscilloscope view is given below.

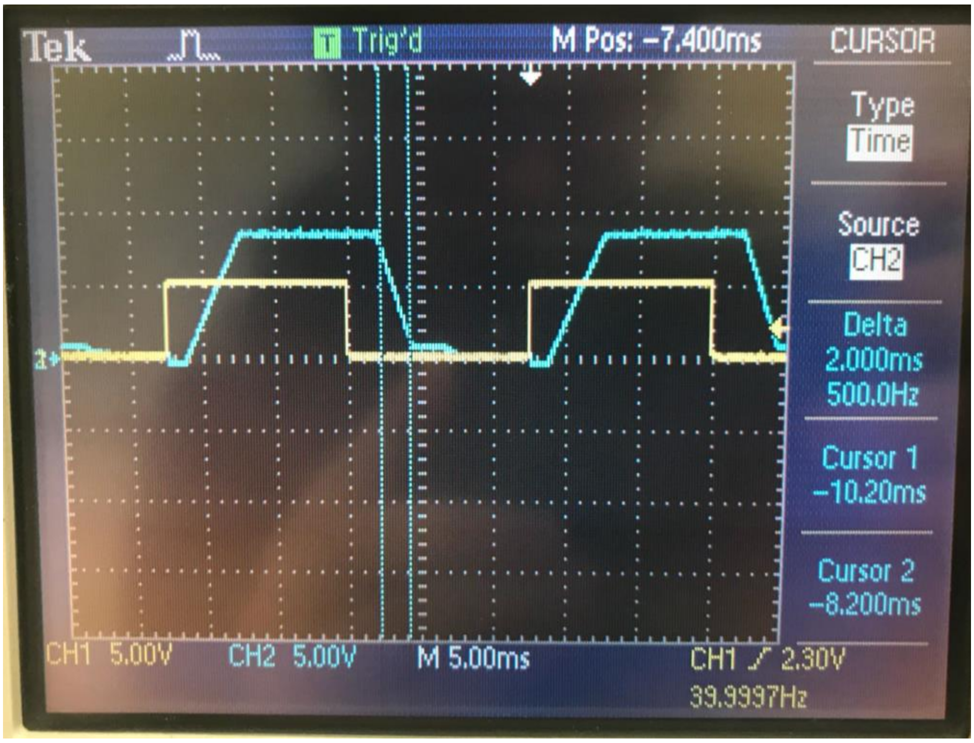

Figure 14 – The oscilloscope view for the first integrator

As easily seen the desired and actual waveforms show great resemblance.

Through the same inspection, for the second circuit, the schematics for the second branch integrator is given below.

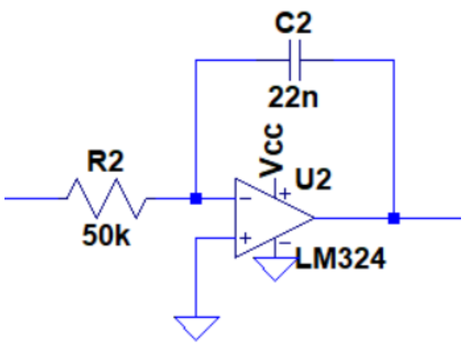

Figure 15 – Circuitry for the Integrator of the Second Branch

As seen, the circuit structure is the same as the one of the first branch. However, with the delay in the second branch the graph of the voltages in the integrator is also delayed. The graph f it is shown below.

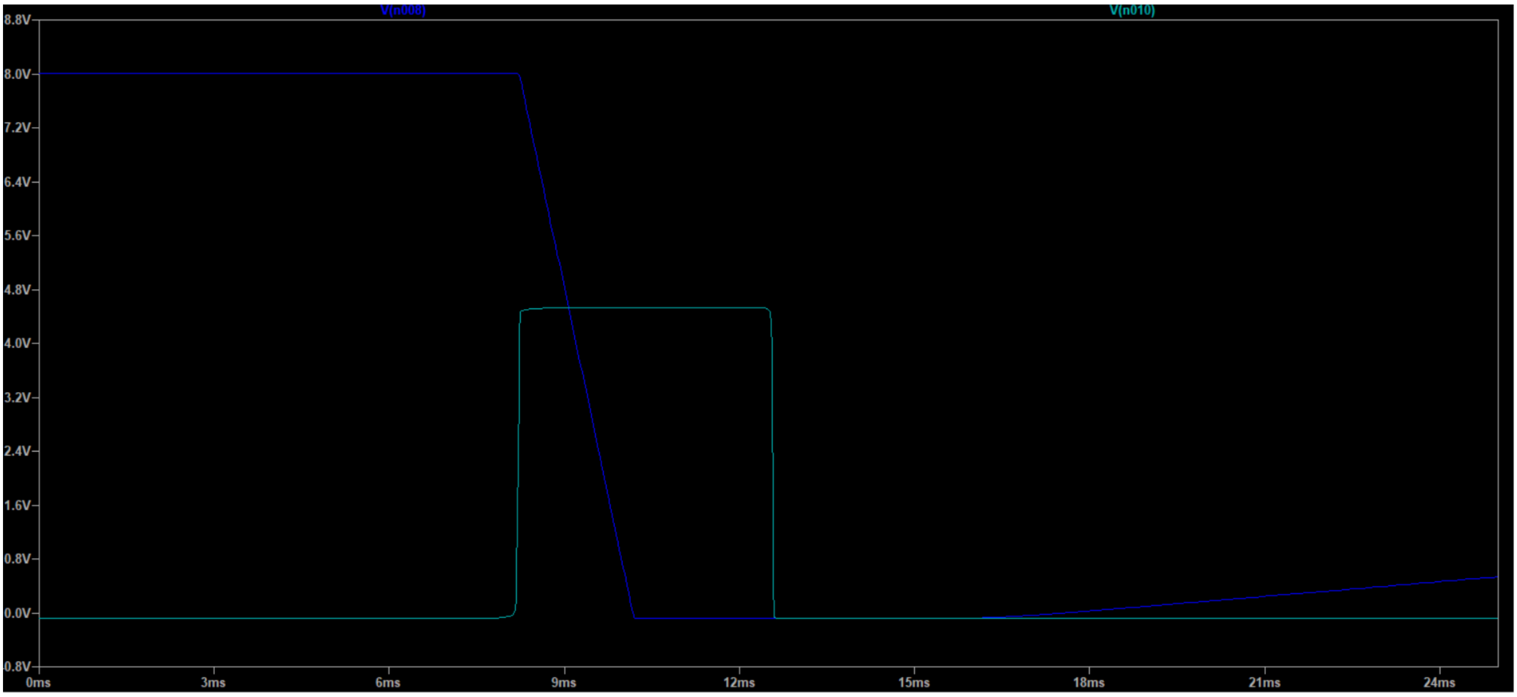

Figure 16 – The Graph of the Second Integrator

This is exactly the same with Figure 7, however the waves are all delayed for 2ms when compared to it. The values of R and C are the same with the 1st branch as the voltage difference to obtain and the time difference is the same.

Hardware Implementation of the 2nd Integrator

The integrator circuit for the second branch is exactly as designed in the LTSpice. The cursors below show that the width of the decline to be 2ms as it drops 4V which is what was expected but at a later time than the first one. The oscilloscope view is shown below.

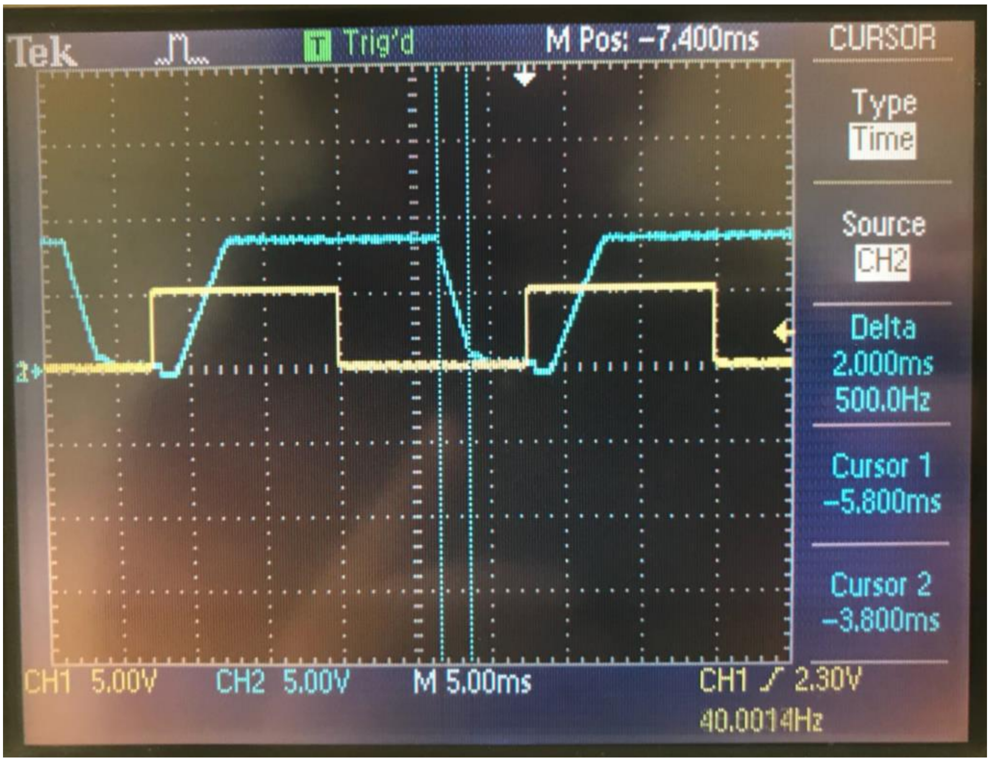

Figure 17 – The oscilloscope view for the first integrator

As easily seen the desired and actual waveforms are pretty similar.

These two circuits placed in series construct a branch and obviously we use two of them to construct the given circuit. Then the problem comes how to unite these branches to get the final waveform. In order to achieve that, we need to use subtractor OPAMP that we will be discussing in the next section.

Subtractor Design

One of the most important parts of the circuits is the subtractor and it completes the assignment as a whole. In order to achieve the desired waveform, we need to use a subtractor OPAMP as we see that the difference of the voltages that we have obtained through the outputs of the integrator OPAMPs give it. The schematics of the subtractor OPAMP is given below.

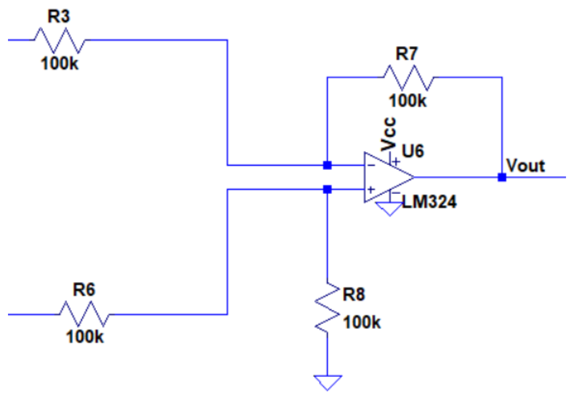

Figure 18 – The Schematic of the Subtractor Circuit

The important part is that, the resistance values actually don't matter however, their ratio are pretty important.

Here the resistance values are not really important but the ratios of them are important. To find the ratio between them we have used a set of equations for the subtractor OPAMP circuits. Hence if we were to write the equations for this type of OPAMP;

$$V_+ = V_-$$

$$V_+ = V_{lower} * \frac{R_8}{R_6 + R_8}$$

Using these two equations, we can obtain,

$$V_- = V_{lower} * \frac{R_8}{R_6 + R_8}$$

Furthermore, if we look at the upper branch now, we see,

$$\frac{V_- - V_{upper}}{R_3} = \frac{V_{out} - V_-}{R_7}$$

If we solve the third equation with the fourth, we find that,

$$V_{lower} - V_{upper} = V_{out}$$

This is the important equation that we will be using. Hence, we can conclude that if the resistance values at the subtractor are chosen to be the same, we will be obtaining our desired output voltage. I have arbitrarily chosen the resistance values to be 100KΩ. The simulation of the circuit is given below.

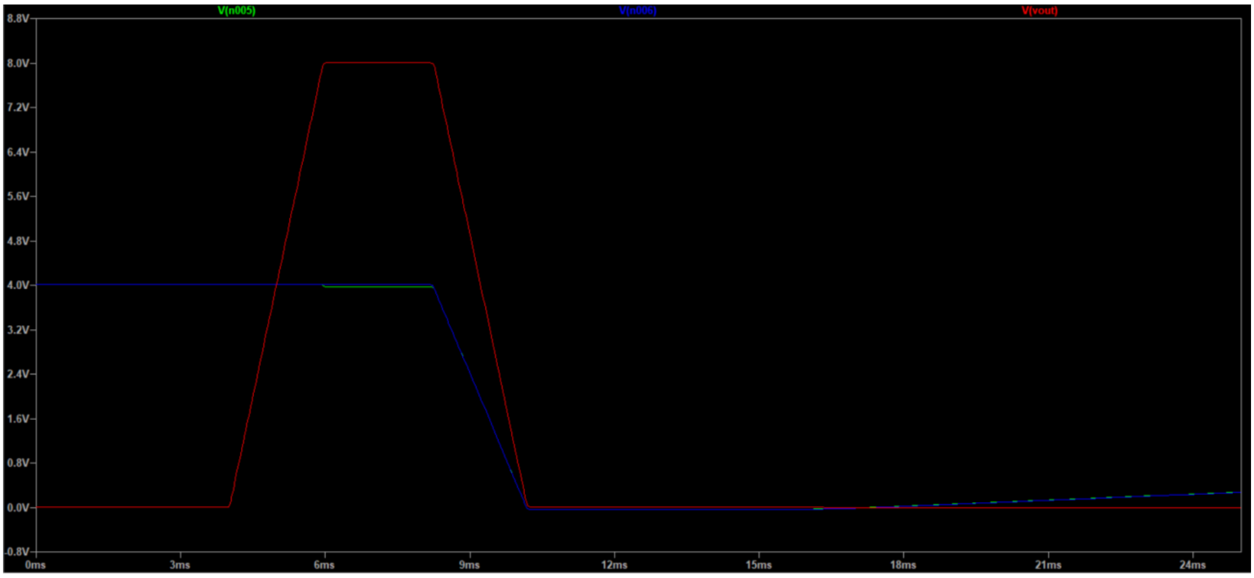

Figure 19 – The Graph for the Subtractor Circuit

The blue and green line which are exactly identical are the input voltages and naturally, the red line gives the output, which is the final waveform. As we see, we got the final waveform to be correct.

Overall the design of the circuit is done, and the overall circuit is shown below in Figure 12.

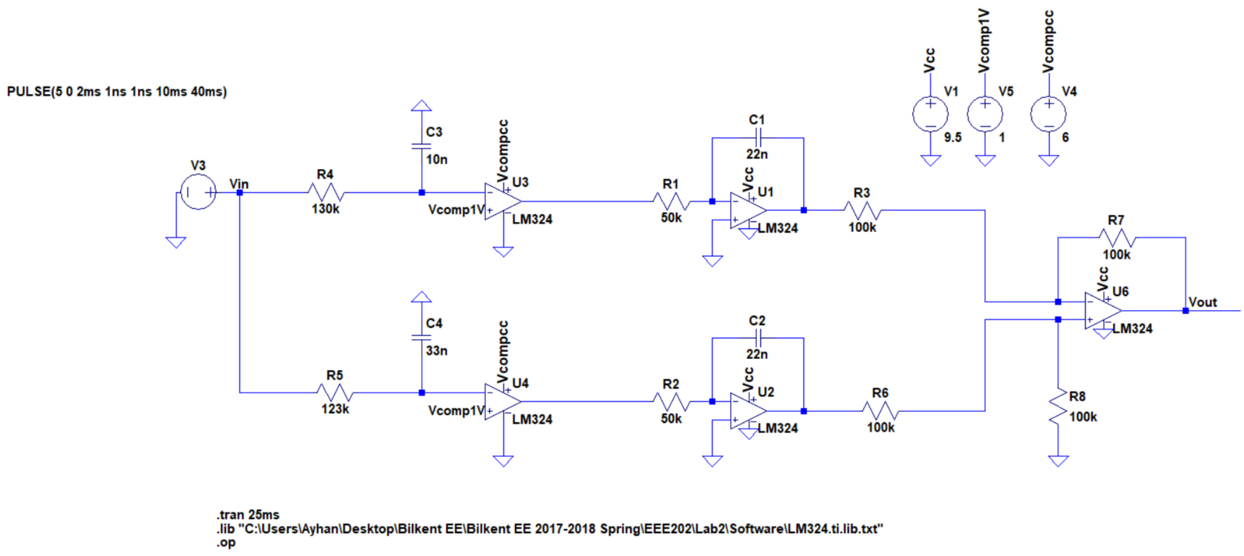

Figure 20 – The Whole Design

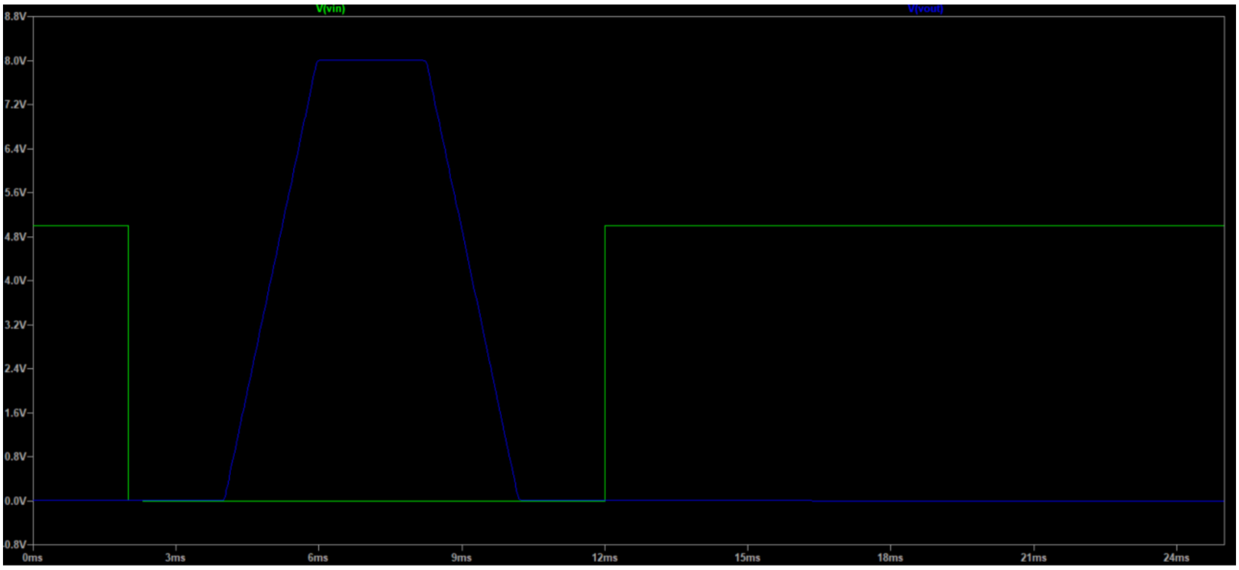

Figure 21 – The Graph for the Input and Output Voltages

Also, if we look at the input and output voltage through the whole circuit which is above in Figure 13, we will see this graph, which is very close to the one that is provided in Figure 1.

Hardware Final Results

The overall circuit looks to be working near perfectly fine. When the output and input is observed, the graph appearing on the oscilloscope matches with the simulation also is given below.

Figure 22 – The final view on the oscilloscope

Conclusion

Through this lab assignment, I have understood that OPAMPs and RC Circuits combined can be used in many different voltage applications. Here, we used it to get a trapezoidal waveform, with all of its attributes changeable. In total we have used 5 OPAMPs in order to make this circuitry.

The hardware part of the lab was fairly easy when compared with the design stage. However, it was too time consuming to build the circuit on the breadboard and one should be careful while doing it. In the end, it was an educating lab assignment in order to understand OPAMPS and RC circuits more.