UNIVERSITY OF WEST LONDON

Analogue Electronic

Technical Report

Manuel Alberto Suena Galindez

14/12/2018

Table of Contents

Abstract

this assignment was divided into 4 main tasks for which calculations and simulation had to be done. The first main task required to find the voltage across a loud in a voltage divider circuit, this was solved by measuring the voltage with a voltmeter for all the given resistor values. The second task was an RC circuit for which the capacitance had to be calculated and to show how it behaves like a low pass filter, the capacitance was calculated by finding the time constant and the low pass behaver was shown by measuring its values at different frequencies. The third task was to show the V-I characteristics of two diodes a know diode and an unknown, the V-I characteristics were obtained by making a circuit with both diodes and let current flow in both forward and revers bias by increasing the voltage of the power supply, the know diode was silicon junction diode and the unknown was a 2V silicon Zener diode. the last task was a design task were an oscillator and a summer amplifier circuit had to be designed, this design was done by calculating all components then running a simulation of the design the confirm the results.

Introduction

The aims of this assignment were to test the fundamental knowledge of analogue electronic and their applications, by assigning as series of design task that require the application of this knowledge in the simulation, construction and analysis of this circuits. this assignment was divided in two parts the first part were laboratory task that required the circuits to be built and required practical measurements. The second part were design task that required more simulation and theoretical value obtained by calculations.

LAB 1

Test 1 find load value:

For the first circuit, the task was to find and calculated the voltage of a load, connected to a voltage divider circuit. The resistances used for this circuit were given in a table of values for which we had to find the values of the load by calculating with the voltage divider formula and for the practical by measuring with a voltmeter.

Testing

To get the measurement after calculating the values, a breadboard mucks up of the circuit in figure 1 was build following the example given in the report brief, for which the values of the resistor were changed according to the values given on the table of values, then by using a voltmeter and probing the load in parallel the values of the load for all different resistor values can be obtain. after getting all the calculated and measured values they were compered to check the disparity between values and to check which diverged more form the norm, the diverges is caused by the internal resistance pose by the voltmeter which has to be calculated and determine.

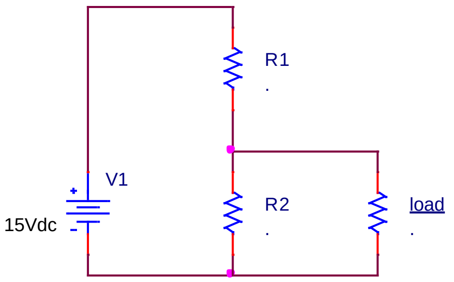

Figure 1: show voltage divider circuit

Calculation

$$V_l = V \times \frac{R_2}{R_1 + R_2}$$

Results

R1	R2	VM1 mes	VM1 calc
4,7kΩ	6,8kΩ	8.84V	8.86V
3,3kΩ	4,7kΩ	8.78V	8.81V
6,8kΩ	3,3kΩ	4.88V	4.9V
2,7MΩ	6,8MΩ	9.59V	10.74V

Table 1

Voltmeter internal resistance value

In this task the internal resistance of the voltmeter had to be calculated by using the calculated values of the previous experiment in table 1

Calculation

The calculation below shows the formulas used to calculate the internal resistance of the voltmeter. First the resistance value for parallel sum of R2 and the voltmeter was calculated using the invers formula of the voltage divider formula, then the resistance value of the voltmeter was calculated by using that value and the invers formula for the addition of two parallel resistors by rearranging the formula for the resistance value of the voltmeter.

$$V_{measured} = Vs\left(\frac{R_{2||m}}{R1 + R_{2,m}}\right) \rightarrow R_{2||m} = \frac{\left(\frac{V_{meas}}{V_s} \times 1\right)}{\left(1 - \frac{V_{meas}}{V_s}\right)} = \frac{\left(\frac{9.59}{15} \times 2.7 \times 10^6\right)}{\left(1 - \frac{9.59}{15}\right)} = 4.786\ M\Omega$$

$$R_{2||m} = \left(\frac{R_m \times R_2}{R_m + R_2}\right) \rightarrow R_m = \frac{\left(R_2 \times \frac{R_{2||m}}{R_2}\right)}{\left(1 - \frac{R_{2||m}}{R_2}\right)} = \frac{\left(6.8 \times 10^6 \times \frac{4.786 \times 10^6}{6.8 \times 10^6}\right)}{\left(1 - \frac{4.786 \times 10^6}{6.8 \times 10^6}\right)}$$

$$R_m = 16.159\ M\Omega$$

Test 2 calculating capacitance

For the second test an RC integrator circuit was given for which we had to calculate the capacitance by knowing the resistance of the resistor and using an oscilloscope with a square wave signal generated by a function generator.

Testing and methodology

to calculate the capacitance of the circuit shown in figure 2 a breadboard muck up of the circuit with a 47kΩ resistor and a unknown capacitor. By plugging the function generator in to the circuit and letting the signal go through the circuit and the measuring the signal outputted by the capacitor and displaying it in an oscilloscope. Then by calculating the time constant by using the curser in oscilloscope and the output wave of the capacitor as shown in figure 3, the capacitance can be calculated by using the invers formula of t = R C

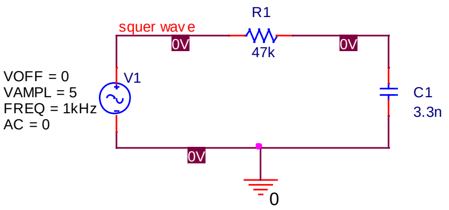

Figure 2: RC circuit

Result

$$C = \frac{\tau}{R} = \frac{160 \ * 10^{-6}}{47 * 10^{3}} = 3.3 * 10^{-9} F$$

Analysis

In figure 3 shows the output signal going through the RC circuit displayed in channel 2 and channel 1 that is the input signal. The cursers of the oscilloscope show the time in which the capacitor is charged to 63% of its total charge or the Time constant of the capacitor which in this case was 160µs, this value was used to find the capacitance of the capacitor in the calculations.

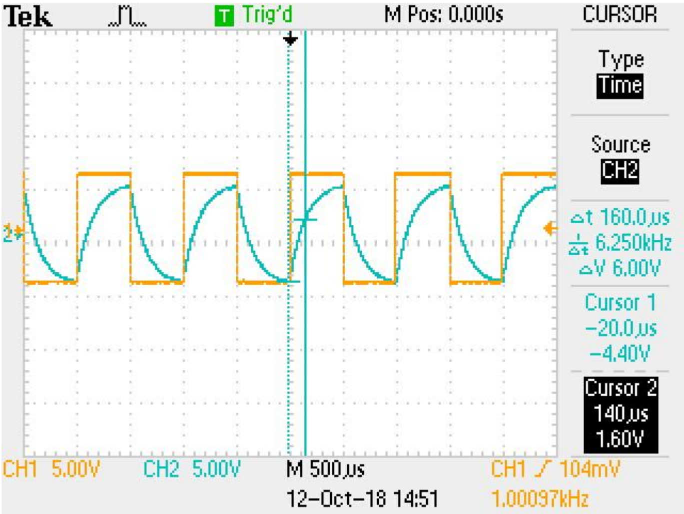

Figure 3: oscilloscope measurement of RC circuit

Experiment 3 RC circuit

For this experiment the circuit is the same RC circuit in figure 2, but for the first experiment the input is a 5V pick voltage sine wave with 1kHz of frequency, as shown in figure 4. Then to continue the experiment the frequency of the signal and measuring voltage, gain and time lag then calculated the period and phase lag of the output signal and record all the results in a table of values. After measuring all the values, the experiment requires to find the break frequency by using the measured values.

Experimental procedure

To get the measurements for this experiment both channels of the oscilloscope where used in combination with a function generator and the circuit muck, shown in figure 4. the measurements of the output signal were measured by the oscilloscope and recorded in a table of values. For the break frequency Excel was used to plot the values measured in Gain vs frequency and phase vs frequency to crosscheck the calculated values with 3 dB method and the alternative 45 degrees method shown in the report.

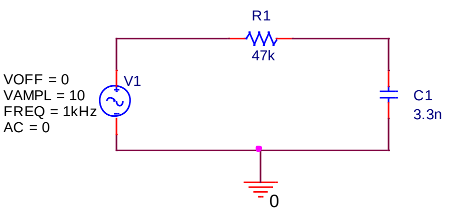

Figure 4: sine wave RC circuit

Results

The table 2 shows the results of the measurements of the circuit and some result that were calculated with the measured results and the formulas shown below, like Gain and Phase lag.

$$Phase\ lag = \frac{Time\ lag}{T}$$

$$Gain = 20\log\left(\frac{Vout}{Vin}\right)$$

F (kHz)	Vout (V)	Gain (dB)	T (ms)	time lag (µs)	phase lag (degree)
0.10	9.80	-0.18	0.01	100.00	3.60
0.20	9.60	-0.35	5.00	140.00	10.08
0.50	8.60	-1.31	2.00	160.00	28.80
1.00	7.00	-3.10	1.00	136.00	48.46
2.00	4.60	-6.74	0.50	80.00	57.60
5.00	3.00	-10.46	0.20	48.00	86.40
10.00	1.44	-16.83	0.10	25.00	90.00
20.00	0.68	-23.10	0.05	13.00	93.60
50.00	0.29	-30.46	0.02	7.50	135.00
100.00	0.12	-38.42	0.01	5.40	194.40

Table 2

Analysis

Figure 5 shows the output wave going through the RC circuit with 1kHz of frequency in channel 2, by analysing this wave two thigs can be noted first that the amplitude of the was decreased and the wave has a shift compered the original. This same effect can be seen in all the values in the table 2, as the frequency increases the wave gest attenuated more and the shift gets bigger. The RC circuit in this experiment is behaving has a low pass filter by attenuating and removing the high frequency input signals.

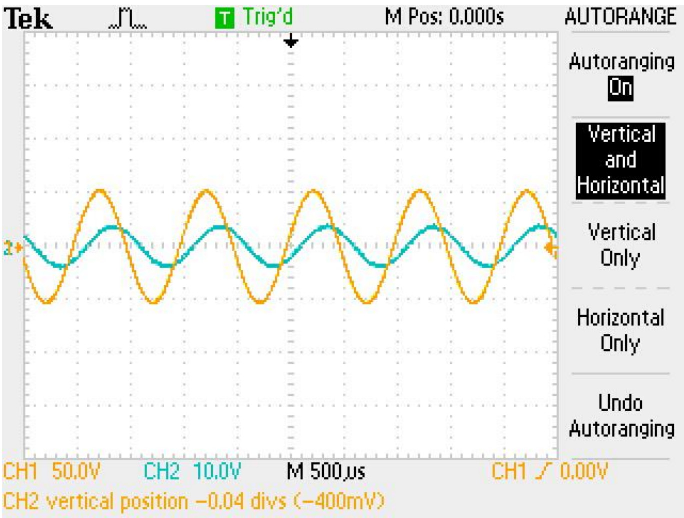

Figure 5: 1kHz RC circuit oscilloscope output

Break frequency analysis

In the figures 6 and 7 show the gain and the frequency plotted in a graph, this was done to fine the break frequency of this circuit, in this first tow graphs the method used was the 3 dB method were by plotting gain and frequency in a graph the break frequency can be found by looking a the point in which the gain is -3dB, in this case the break frequency was around 1kHz. In the last two figures, figure 8 and 9, the break frequency was found by using an alternative method were the break frequency is the phase shift is 45, and here again the break frequency is around 1kHz, so this agrees with the previous method.

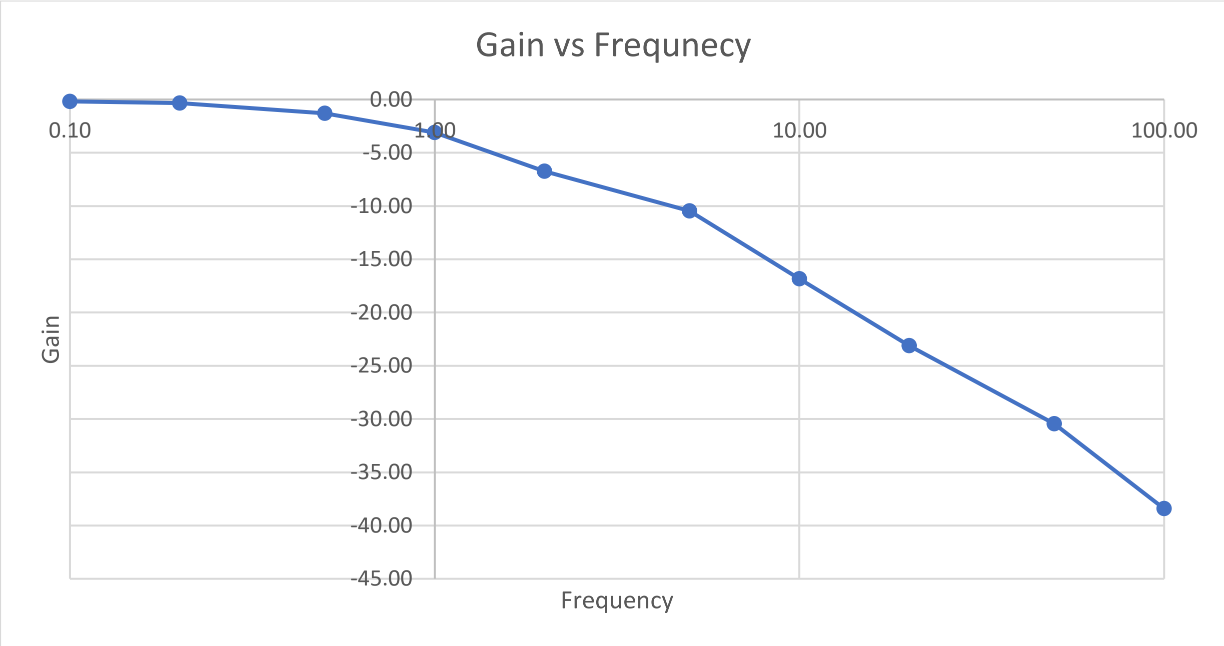

Figure 6: Gain vs frequency

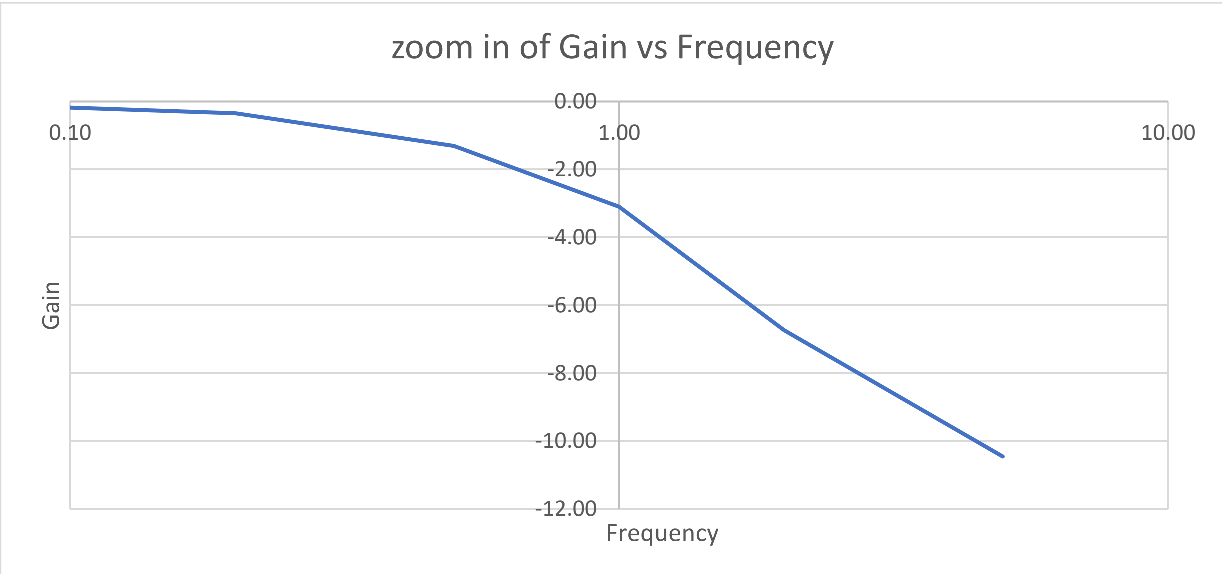

Figure 7: zoomed Gain vs frequency

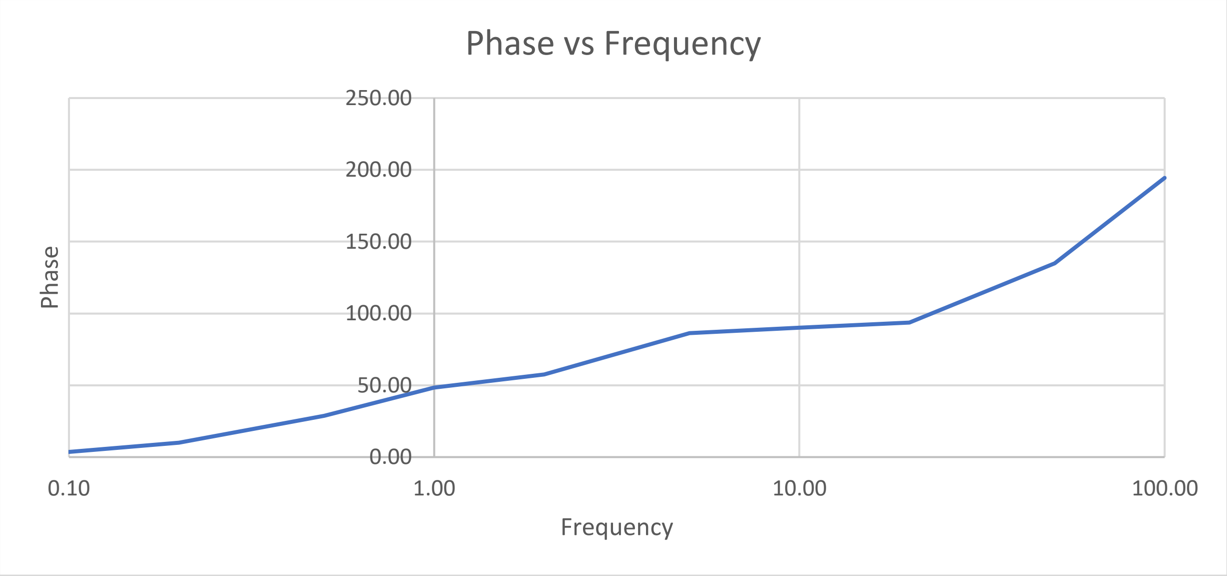

Figure 8: phase vs frequency

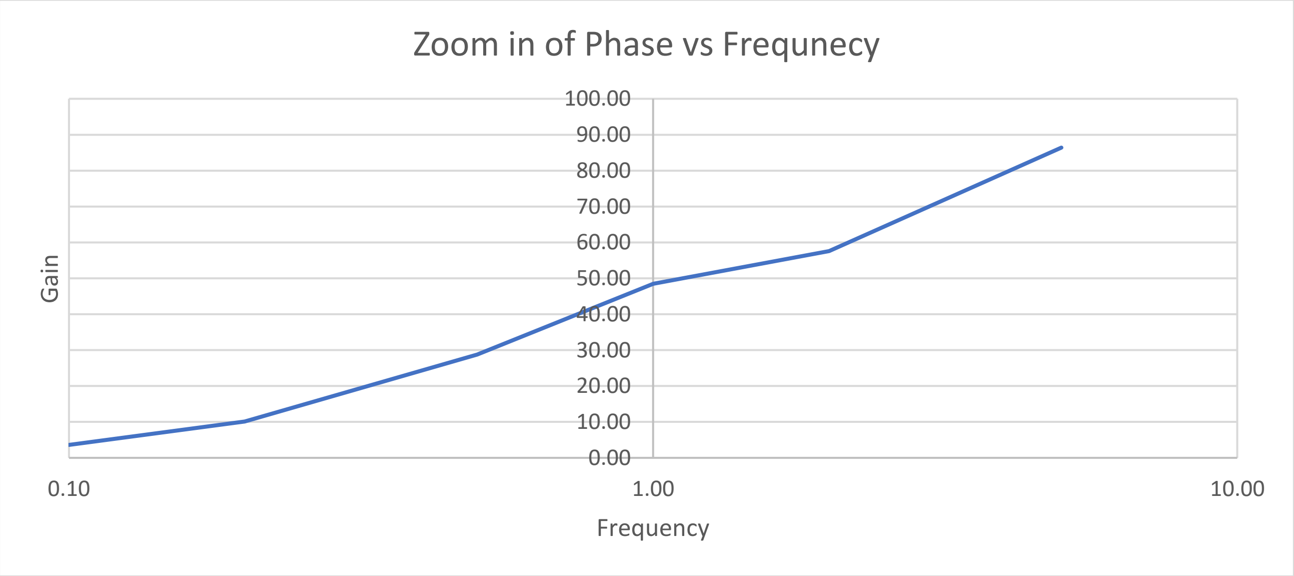

Figure 9: zoomed phase vs frequency

Calculation of break frequency

This calculation shows the exact value of the break frequency and proves that the methods used since all methods result were around 1kHz.

$$f_0 = \frac{1}{2\pi RC} = \frac{1}{2\pi * 47k\Omega * 3.3nf} = 1.03kHz$$

LAB 2

Diode characteristic

For this part of the assignment two diodes were given for which the V-I characteristics had to be made, one of the diodes was known to be a regular bipolar silicon diode and the other was unknow. for the unknow diode was needed to find the type of diode it was.

Experimental procedure

To obtain the values for voltage and current of diodes a circuit with the diodes was made by following the diagram in figure 10, then the circuit is supplied with an increasing voltage value and with a voltmeter the voltage across the diode and the current is measured for all the voltage values. This was done for both diodes in forward and revers bias.

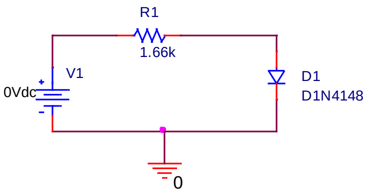

Figure 10: diode circuit

Calculations

The calculation below shows how the resistor value was calculated by setting a value for current and voltage supply, for this experiment the resistor value was approximated to 1kΩ, and the formula used to calculate the current values used in table 3, 4, 5 and 6

$$R = \frac{V_s - V_d}{I} = \frac{9 - 0.7}{5 \times 10^{-3}} = 1.660k\Omega$$

$$I = \frac{V_s + V_d}{R}$$

Results

Known diode

forward	Vs (V)	Vd (V)	I mes (mA)	I calc (mA)	I sim (mA)
	0	0.11	0	0.11	0.11
	0.2	0.18	0	0.02	0.02
	0.5	0.44	0	0.06	0.06
	1	0.56	0	0.44	0.44
	2	0.62	0	1.38	1.38
	5	0.68	4.3	4.32	4.32
	10	0.68	9.2	9.32	9.28
	15	0.74	14.2	14.25	14.25
	20	0.76	19.2	19.2	19.2
	25	0.77	24.2	24.23	24.23
	30	0.78	29.2	29.22	29.22

Table 3

Revers	Vs (V)	Vd (V)	I mes (mA)	I calc (mA)	I sim (mA)
	0	-0.08	0	-0.08	-0.08
	0.2	-0.02	0	0	0
	0.5	-0.48	0	-0.02	-0.02
	1	-0.99	0	-0.01	-0.01
	2	-1.97	0	-0.03	-0.03
	5	-4.97	0	-0.03	-0.03
	10	-9.95	0	-0.05	-0.05
	15	-14.93	0	-0.07	-0.07
	20	-19.92	0	-0.08	-0.08
	25	-24.8	0	-0.02	-0.02
	30	-29.8	0	-0.02	-0.02

Table 4

Unknown diode

forward	Vs (V)	Vd (V)	I mes (mA)	I calc (mA)
	0	0.12	0	0.11
	0.2	0.19	0	0.01
	0.5	0.48	0	0.02
	1	0.67	0.3	0.33
	2	0.71	1.26	1.29
	5	0.71	4.22	4.25
	10	0.75	4.23	9.25
	15	0.75	4.22	14.25
	20	0.75	4.17	19.25
	25	0.75	4.17	24.25
	30	0.75	4.18	29.25

Table 5

Revers	Vs (V)	Vd (V)	I mes (mA)	I calc (mA)
	0	-0.09	0	0
	0.2	-0.2	0	0
	0.5	-0.49	0	-0.01
	1	-0.99	0	-0.01
	2	-1.79	-0.21	-0.21
	5	-2.49	-2.48	-2.54
	10	-2.69	-4.86	-7.31
	15	-2.69	-4.86	-12.31
	20	-2.69	-4.86	17.31
	25	-2.69	-4.86	-22.31
	30	-2.69	-4.86	-27.31

Table 6

Analysis

By looking at the V-I characteristic shown in figure 11 that shows the unknown diode the diode given was a 2V silicon Zener diode, 2V because in revers bias around 2V current started to flow through and silicon because in forward bias the current flowed at around 0.7V. for the known diode in figure 12 it was confirmed by the V-I characteristics that it was a regular silicon diode.

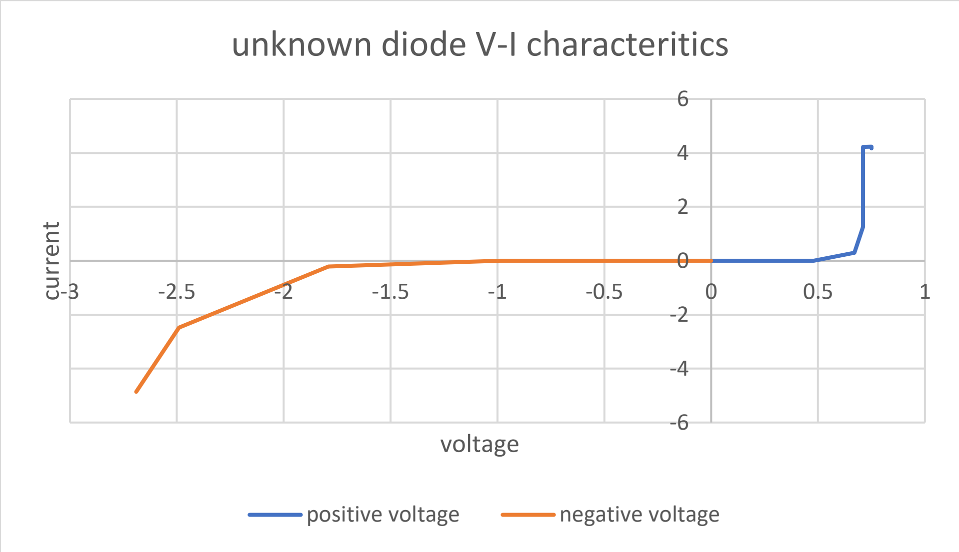

Figure 10: unknow diode V-I characteristics

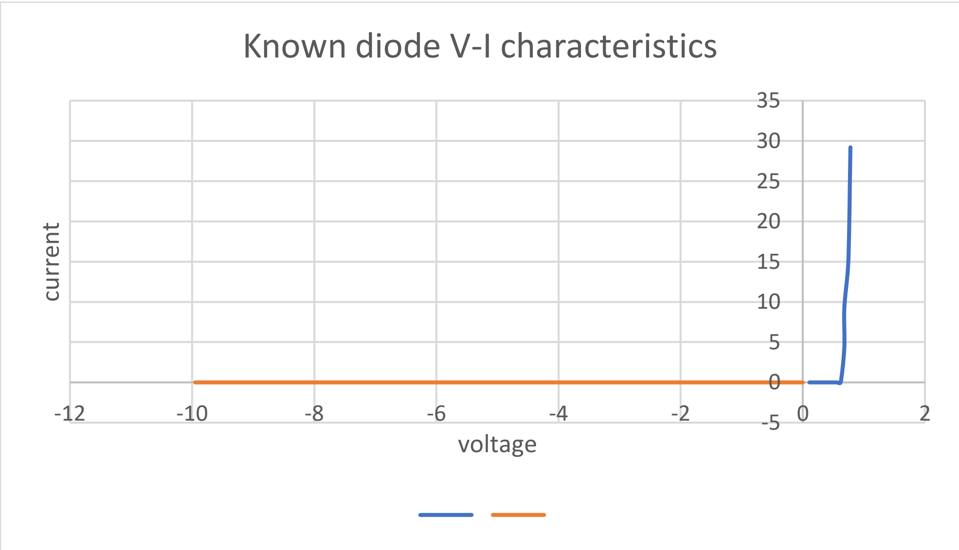

Figure 11:known diode V-I characteristics

Design task 1

For this task a Wein bridge oscillator with 200Hz of frequency and a 200Hz square wave oscillator had to be designed. All oscillator had to maintain a constant oscillation and a way to make the square wave into a triangle wave had to be found.

Calculations

This calculation shows how the value for the capacitor was calculated the resistor value was assigned the value of 5kΩ, the formula used was an inverse formula of the frequency. The second calculation shows the rate of change of the output wave. The resistor and capacitor value where found by trial and error since there was no specification for the output change rate and no invers formulas could be made to find the values without knowing at least three of variables in the formula.

$$f = \frac{1}{2\pi RC} \rightarrow C = \frac{1}{2\pi Rf} = \frac{1}{2\pi \times 5000x200} = 159nF$$

$$\frac{dVout}{dt} = \frac{1}{RC} \times Vin = \frac{1}{5000(1 \times 10^{-3})} \times 7.4771 = 1.5\, V/ms$$

$$C = \frac{1}{R \times \frac{dVout}{dt}} \times Vin = \frac{1}{5000 \times 1.5} \times 7.4771 = 0.997\, mF$$

Analysis

In figure 13 show the first design of the circuit, in this circuit the purpose was to design a Wein bridge oscillator with constant oscillations, and as seen in figure 14 the oscillations are constant.in the second design in figure 15 the wave had to be a square wave, this was achieved by connecting a 9V 200Hz sin wave power supply to the circuit, the output in figure 16 shows the resulting square wave. The third design in figure 17 had to turn does square wave into triangular waves, as shown in figure 18 the output of the design were triangular wave, this was done by putting a RC integrator circuit in the output.

Simulation

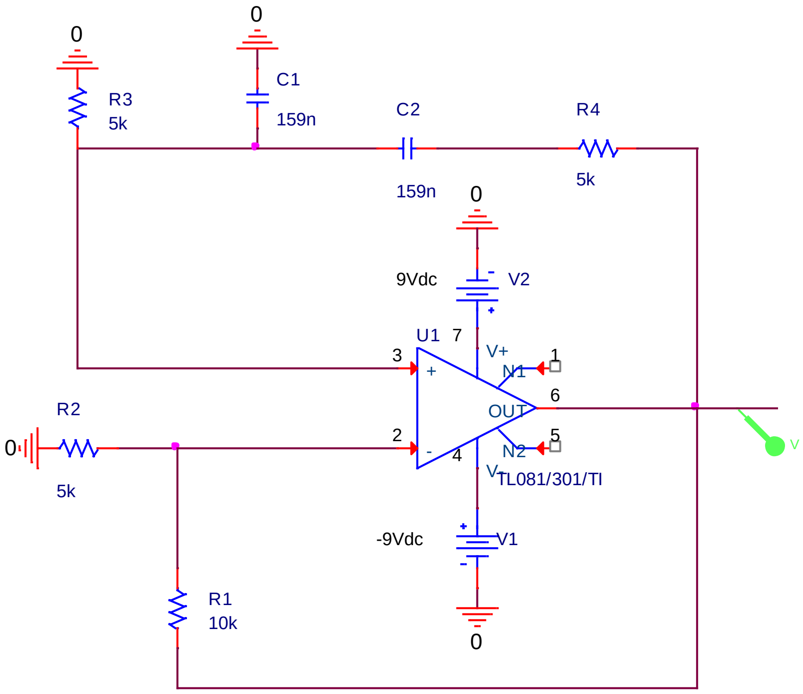

Figure 12: Wein bridge oscillator circuit

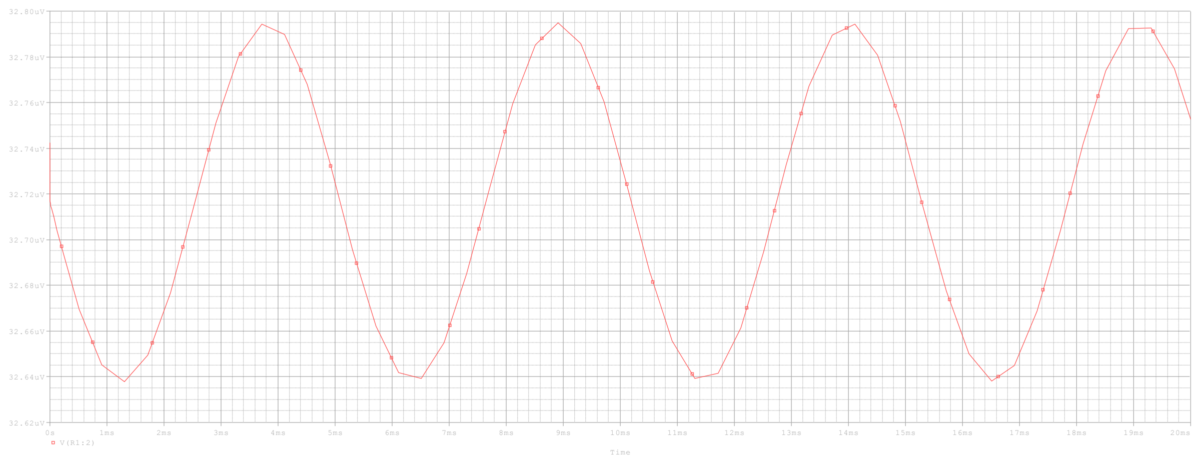

Figure 13: constant oscillation

Square waveform simulation

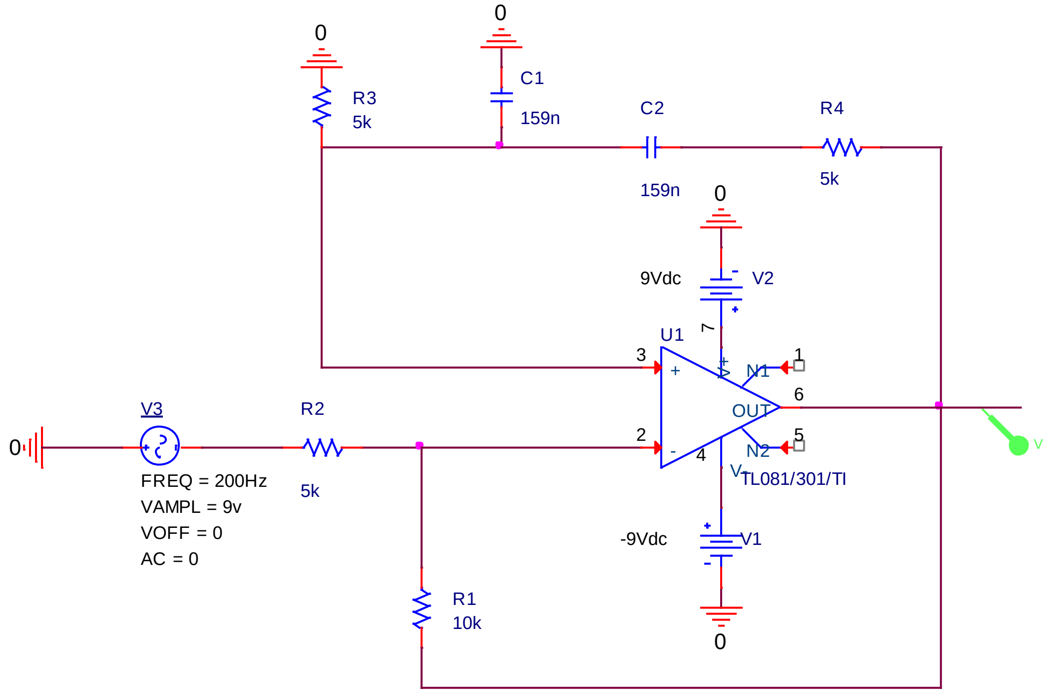

Figure 14: square wave design

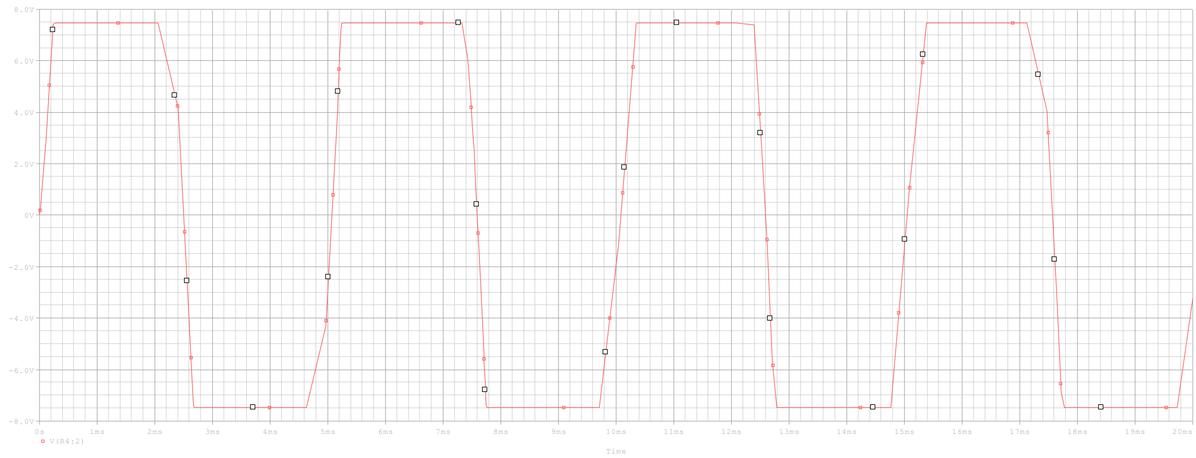

Figure 15: square wave result

Triangle waveform simulation

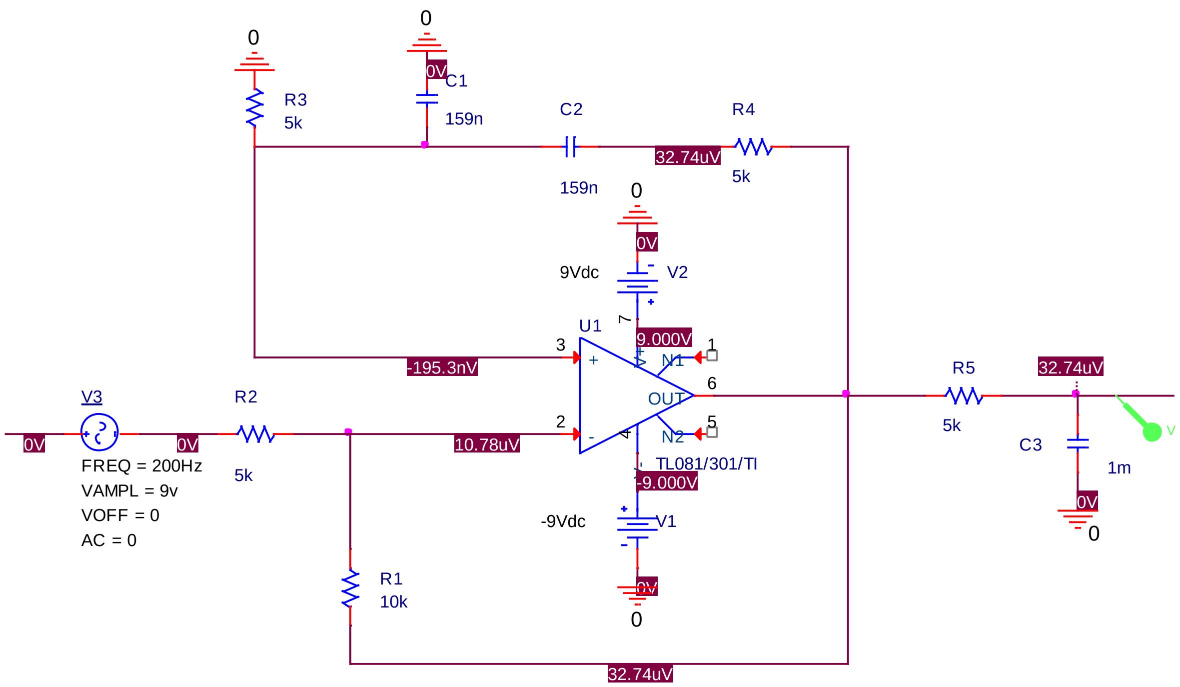

Figure 16: triangle wave design

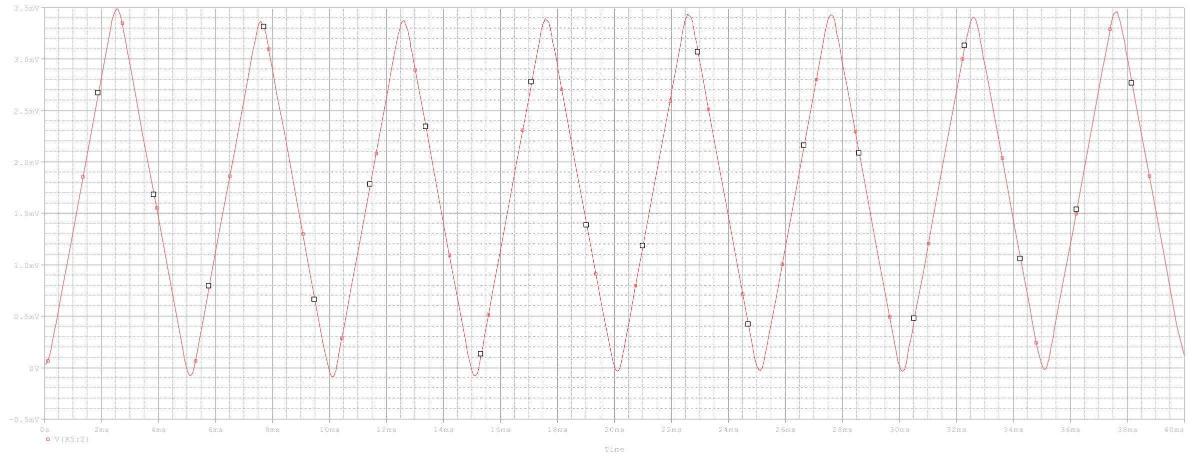

Figure 17: triangle wave result

Design task 2

The task for task 2 was to design and build a summer amplifier circuit to add two DC signal 1V and 4V, the design had to be made with a total impedance of 10kΩ, and the offset voltage of the Operational amplifier set to a minimum. Then one of the DC supplies of the design had to be replaced with an1V, 10kHz sine wave power supply, to see the effects of the designed amplifier circuit on the wave.

Methodology

For the practical measurements of this task a circuit was built by following the diagram in figure 19 then by first connecting the two 9V power supply for the amplifier gain the offset was measured with a voltmeter. By setting the voltmeter pins in pin 3 and 6 of the amplifier, then with a screw driver the potentiometer was turned until the voltage measured by the voltmeter was set to as close as possible to 0. After the offset was set to 0 the other two power supplies that had to be added were turn on and the output result of the sum was recorded by the oscilloscope. For the maximum offset design instead when measuring the offset, it was set to the maximum value by turning the potentiometer to in the opposite direction.

Calculation

The calculation below shows how the resistors values were calculated and the calculation for the Vout, by using the summer amplifier formula.

$$R_{1,2,3} = \frac{R_T}{Number\ of\ resistors} = \frac{R_T}{3} = 3.3k\Omega$$

$$Vout = -R_1 \times \left(\frac{Vin_1}{R_2} + \frac{Vin_2}{R_3}\right) = -(3.3 \times 10^3) \times \left(\frac{1}{3.3 \times 10^3} + \frac{4}{3.3 \times 10^3}\right) = -5V$$

Analysis

In figure 19 shows the simulation of the designed circuit, it can be noted that e Vout value of the simulation agrees with the calculated value, that also the same values of the practical measurements in figure 20 that show the output voltage as recorded by the oscilloscope. The practical results were taken by a breadboard build of the circuit, in that design the offset voltage was kept to a minimum by changing the value of the potentiometer connected to pin 1 and 5. In figure 21 instead the offset was set to its maximum value and the difference in the voltage value is noticeable

Simulation

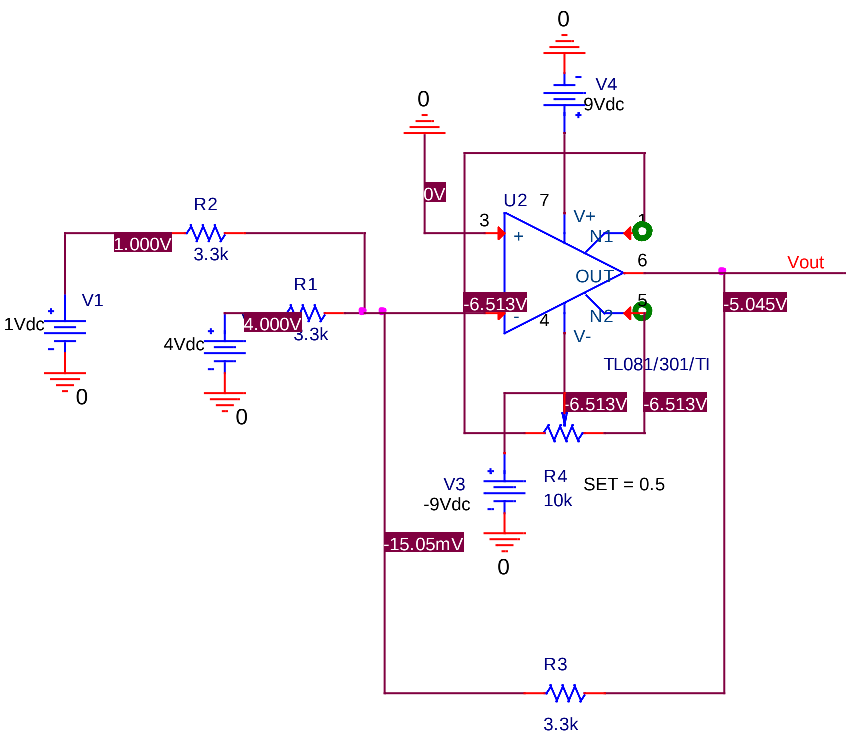

Figure 18: summer amplifier circuit

Practical measurements

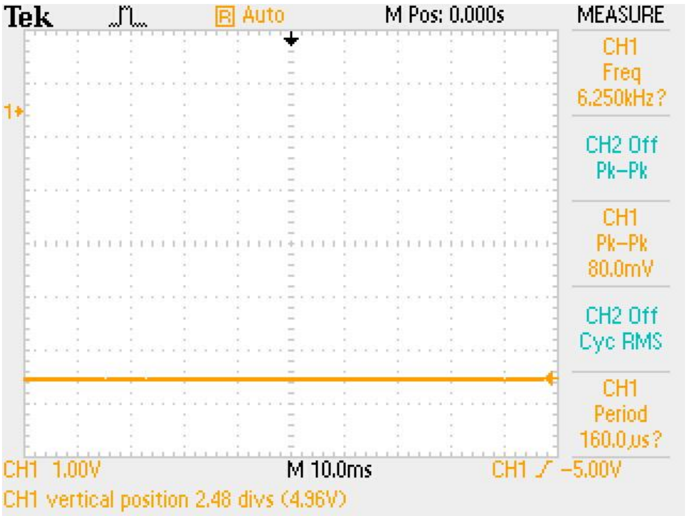

Figure 19: minimum offset value

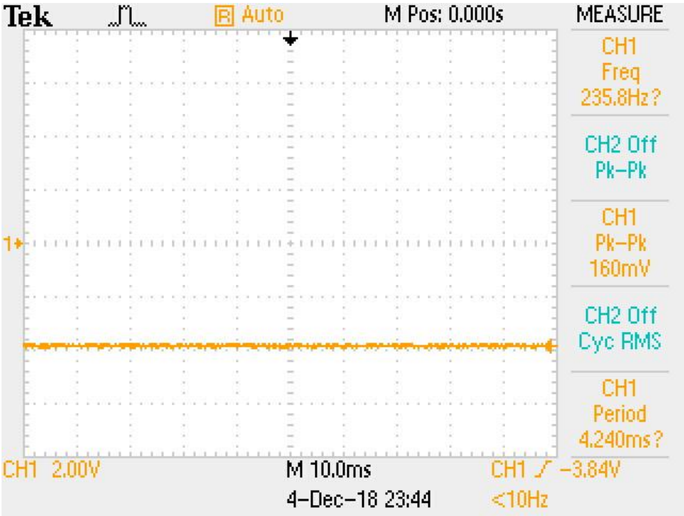

Figure 20: maximum offset value

Analysis for the second design

the diagram shown in figure 22 was supplied with a 1V and 10kHz frequency sine power supply, used to analyze the effects that this operational amplifier has on the wave. The effects that the opamp has on the wave can be more clearly observed in figure 24 that shows the practical measurements of this circuit and compares the output signal with the input signal, by comparing the signal it can be noted that the wave sims shifted compared to the original signal, this sims to be the only effect since amplitude and frequency were unchanged.

Sine wave supply simulation

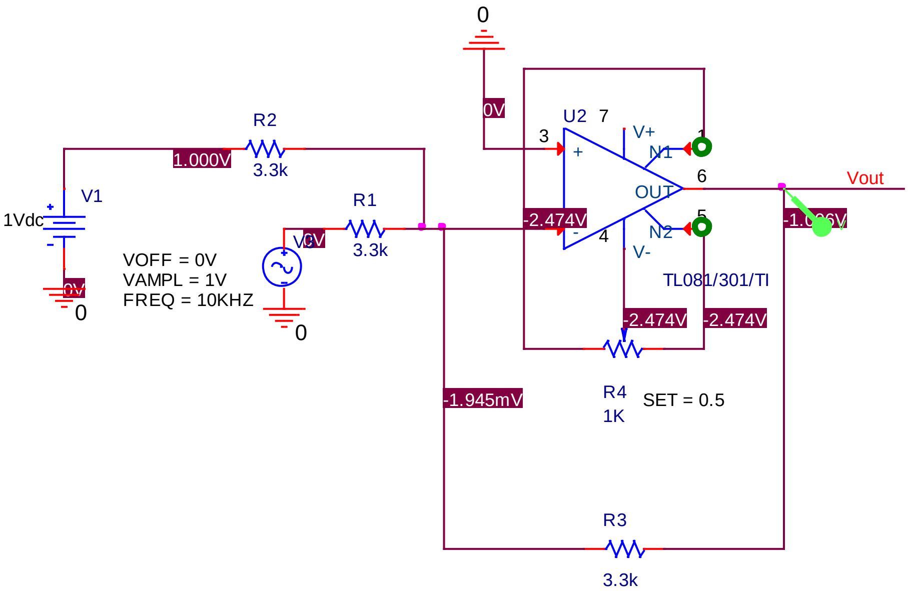

Figure 21: sine design

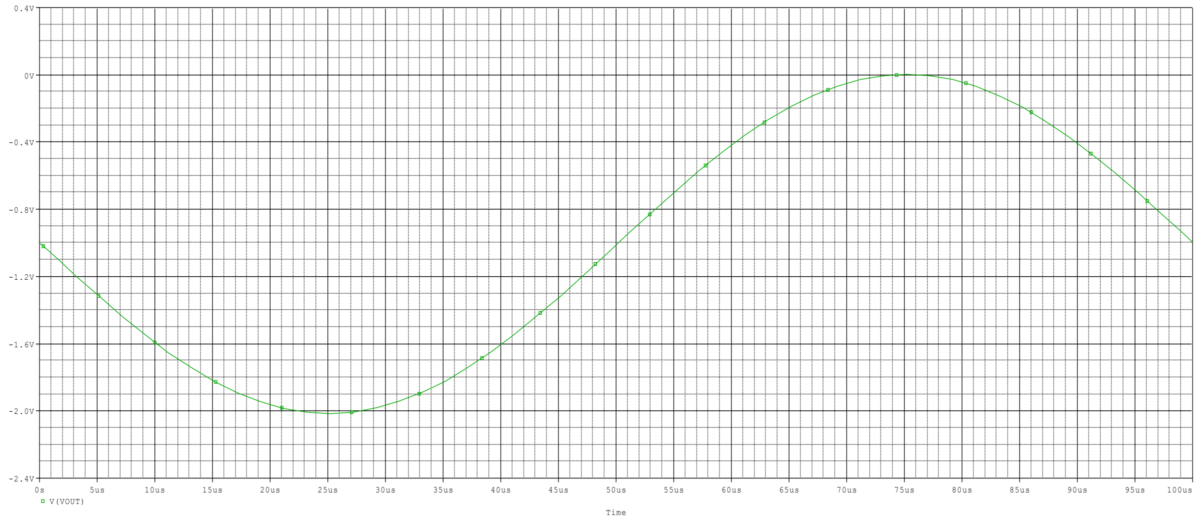

Figure 22: sine wave output

Practical measurement for the second design

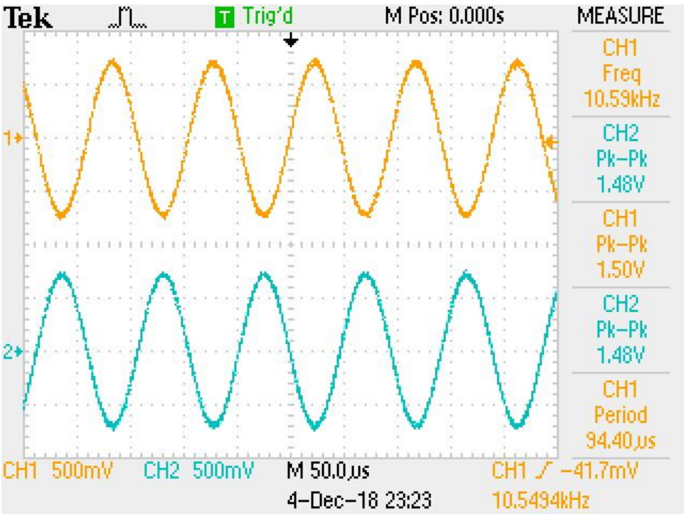

Figure 23: oscilloscope measurement of the sine wave output