NANJING UNIVERSITY OF INFORMATION SCIENCE & TECHNOLOGY

LAB REPORT

Course Name ______________________________

______ Institute __________ Major ____ Grade ____ Class

Name ________ Student ID ________ Tutor ________

Waterford Institute

December, 2024

Lab1 Potentiometers Part1

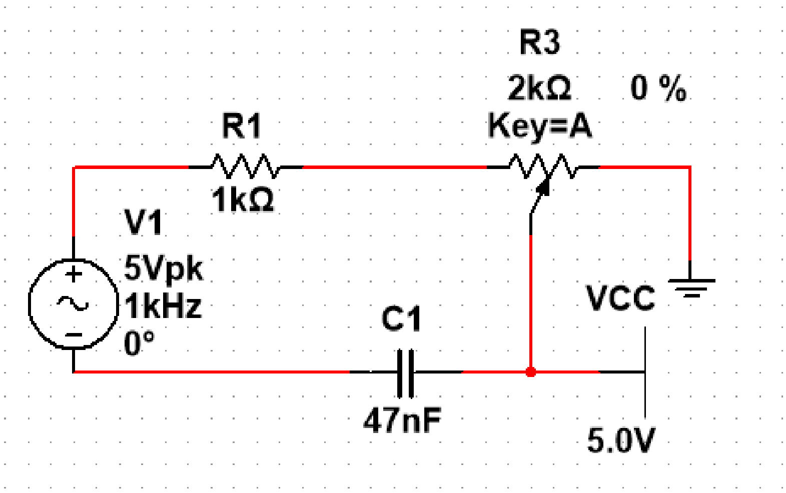
Jairui Huang(黄家睿)
202283890036

Introduction and Aim

This experiment explores the behaviour of a simple RC (Resistor-Capacitor) circuit and understand its response to changes in input voltage. To achieve this, we will use a linear poten@ometer as a variable resistor in the circuit. A potentiometer, oBen referred to as a pot, is a three-terminal resistor with an adjustable tap that allows us to change the resistance value along its length. In this experiment, we will u@lize a linear poten@ometer to vary the resistance in the RC circuit. By adjus@ng the poten@ometer, we can control the rate at which the capacitor charges and discharges, thus influencing the @me constant and the behaviour of the circuit.

Circuit Diagram

The circuit diagram is created in Multisim.



Data_Table

kHZ	Potentiometer perc	Vp	Vc	Vp/Vc
1kHZ	0%	4.43V	7.47V	0.59
1kHZ	20%	3.75V	7.90V	0.47
1kHZ	40%	2.97V	8.37V	0.35
1kHZ	60%	2.09V	8.82V	0.23
1kHZ	80%	1.09V	9.22V	0.11
1kHZ	100%	0.69V	9.58V	0.07
2kHZ	0%	5.80V	4.89V	1.18

kHZ	Potentiometer perc	Vp	Vc	Vp/Vc
2kHZ	20%	5.15V	5.43V	0.94
2kHZ	40%	4.33V	6.09V	0.35
2kHZ	60%	3.23V	6.82V	0.47
2kHZ	80%	1.82V	7.69V	0.23
2kHZ	100%	1.24V	8.60V	0.14
5kHZ	0%	6.50V	2.19V	2.96
5kHZ	20%	5.96V	2.51V	2.37
5kHZ	40%	5.21V	2.93V	1.77
5kHZ	60%	4.15V	3.50V	1.18
5kHZ	80%	2.57V	4.34V	0.59
5kHZ	100%	2.02V	5.59V	0.36
10kHZ	0%	6.61V	1.12V	5.90
10kHZ	20%	6.08V	1.28V	4.75
10kHZ	40%	5.37V	1.51V	3.55
10kHZ	60%	4.36V	1.84V	2.36
10kHZ	80%	2.78V	2.34V	1.18
10kHZ	100%	2.31V	3.19V	0.72
20kHZ	0%	6.66V	0.56V	11.8
20kHZ	20%	6.14V	0.64V	9.59
20kHZ	40%	5.44V	0.76V	7.15
20kHZ	60%	4.42V	0.93V	4.75
20kHZ	80%	2.83V	1.19V	2.37
20kHZ	100%	2.40V	1.66V	1.44
50kHZ	0%	6.66V	0.23V	28.9
50kHZ	20%	6.15V	0.26V	23.6
50kHZ	40%	5.45V	0.31V	17.6
50kHZ	60%	4.44V	0.38V	11.7
50kHZ	80%	2.85V	0.48V	5.93
50kHZ	100%	2.44V	0.67V	3.64
100kHZ	0%	6.66V	0.11V	60.5

kHZ	Potentiometer perc	Vp	Vc	Vp/Vc
100kHZ	20%	6.15V	0.13V	47.3
100kHZ	40%	5.45V	0.15V	36.3
100kHZ	60%	4.44V	0.18V	24.6
100kHZ	80%	2.86V	0.24V	11.9
100kHZ	100%	2.44V	0.34V	7.17

Discussion

From the data above, it can be observed that as the proportion of the potentiometer increases,Vp gradually decreases, Vc gradually increases, and the ratio of Vp to Vc continuously decreases. As the frequency of the signal generator increases, the Vp/Vc ratio at the same potentiometer proportion also increases. For instance, at 1 kHz with a potentiometer proportion of 20%, the Vp/Vc ratio is 0.47, whereas at 10 kHz with the same potentiometer proportion, the Vp/Vc ratio increases to 4.75, which is 100 times larger compared to 0.47. However, with further increases in the signal generator frequency, Vp no longer changes. From the data at 50 kHz and 100 kHz, it can be seen that Vp remains the same for each potentiometer proportion.

kHZ	Potentiometer perc	Vp	Vc	Vp/Vc
50kHZ	0%	6.66V	0.23V	28.9
50kHZ	20%	6.15V	0.26V	23.6
50kHZ	40%	5.45V	0.31V	17.6
50kHZ	60%	4.44V	0.38V	11.7
50kHZ	80%	2.85V	0.48V	5.93
50kHZ	100%	2.44V	0.67V	3.64
100kHZ	0%	6.66V	0.11V	60.5
100kHZ	20%	6.15V	0.13V	47.3
100kHZ	40%	5.45V	0.15V	36.3
100kHZ	60%	4.44V	0.18V	24.6
100kHZ	80%	2.86V	0.24V	11.9
100kHZ	100%	2.44V	0.34V	7.17

AC Sweep simulation

Jiarui Huang(黄家睿)
202283890036

Introduction and Aim

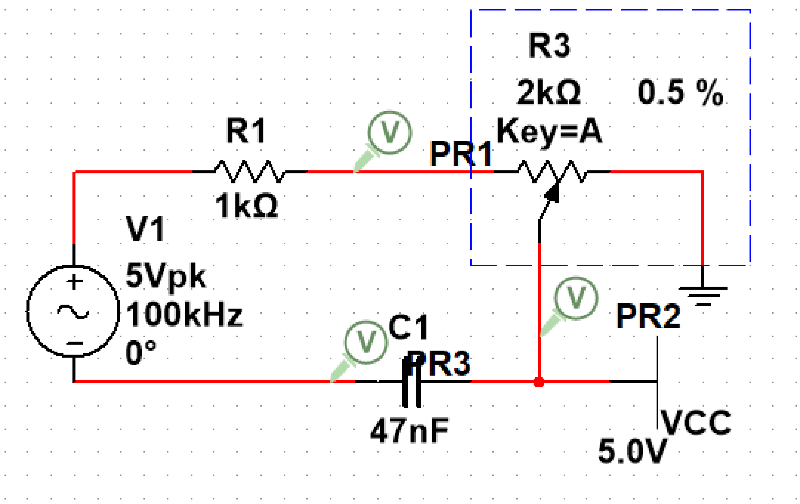
This experiment explores the behaviour of a simple RC (Resistor-Capacitor) circuit and understand its response to changes in input voltage. To achieve this, we will use a linear poten@ometer as a variable resistor in the circuit. A potentiometer, oBen referred to as a pot, is a three-terminal resistor with an adjustable tap that allows us to change the resistance value along its length. In this experiment, we will utilize a linear poten@ometer to vary the resistance in the RC circuit. By adjusting the potentiometer, we can control the rate at which the capacitor charges and discharges, thus influencing the time constant and the behaviour of the circuit.

Theory

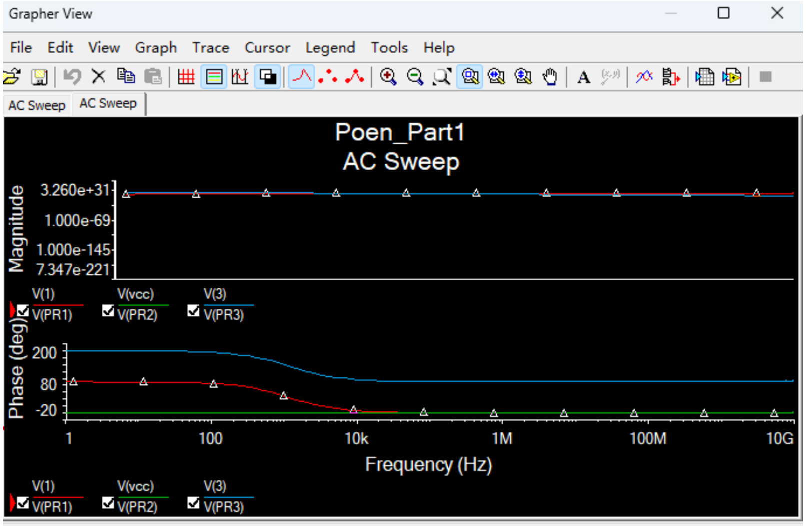
the cut-off frequency fc is defined the point at which the output voltage is equal to -3 dB (or 70%) of the input voltage, where:

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

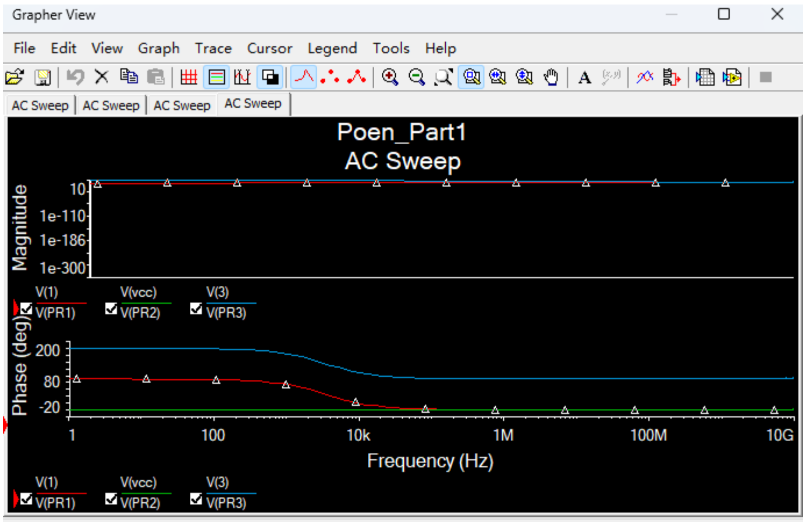
Circiut Diagram



The output of the AC sweep simulation This diagram show the the circuit's response and determine the cut-off frequency with the potentometer of 0.5%.



This diagram show the the circuit's response and determine the cut-off frequency with the potentometer of 100%.



RC Circuit measurement

Jiarui Huang(黄家睿)
202283890036

Introduction and Aim

This experiment explores the behaviour of a simple RC (Resistor-Capacitor) circuit and understand its response to changes in input voltage. To achieve this, we will use a linear potentiometer as a variable resistor in the circuit. A potentiometer, oBen referred to as a pot, is a three-terminal resistor with an adjustable tap that allows us to change the resistance value along its length. In this experiment, we will utilize a linear poten@ometer to vary the resistance in the RC circuit. By adjusting the potentiometer, we can control the rate at which the capacitor charges and discharges, thus influencing the time constant and the behaviour of the circuit.

Theory

An RC circuit is a basic circuit unit composed of a resistor R and a capacitor C , used for signal processing, filtering, time delay, etc.

1.Time Constantτ

The time constant describes the dynamic response of an RC circuit and is defined as: $\tau = R \times C$

2. Cutoff Frequencyf_c

For an RC filter, the cutoff frequency is the frequency where the circuit starts to significantly attenuate the signal, defined as:

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

Low-pass filter:$f \leq f_c$will pass through, while signals with $f \geq f_c$will be attenuated. High-pass filter: $f \geq f_c$will pass through, while signals with$f \leq f_c$will be attenuated.

3.Impedance

In AC circuits, the total impedance of an RC circuit is the combination of the resistance R and the capacitive reactance X_C :

$$Z_T = Z_R + Z_C = R\angle 0\omega + X_C\angle - 90\omega = R - jX_C$$

4.Voltage-Related Parameters

4.1Power Supply Voltage $V_S

This is the input voltage provided to the circuit, typically supplied by a signal generator or a DC power source. V_S is a known external parameter. $V_s$is a known external parameter.

4.2Voltage Across the Resistor V_R

According to Ohm's law, the voltage across the resistor is:

$$V_R = I \cdot R$$

where I is the current in the circuit and R is the resistance.

4.3 Voltage Across the Capacitor V_C

The voltage across the capacitor depends on its charging or discharging state: During charging:

$$V_C(t) = V_S \cdot (1 - e^{\frac{-t}{\tau}})$$

During discharging:

$$V_C(t) = V_c0 \cdot e^{\frac{-t}{\tau}}$$

Here, $\tau = R \cdot C$ is the time constant, and v_C0 is the initial voltage.

Time Period T and Frequency f

If the circuit uses an AC power source, the time period and frequency are determined by the input signal: Time Period:

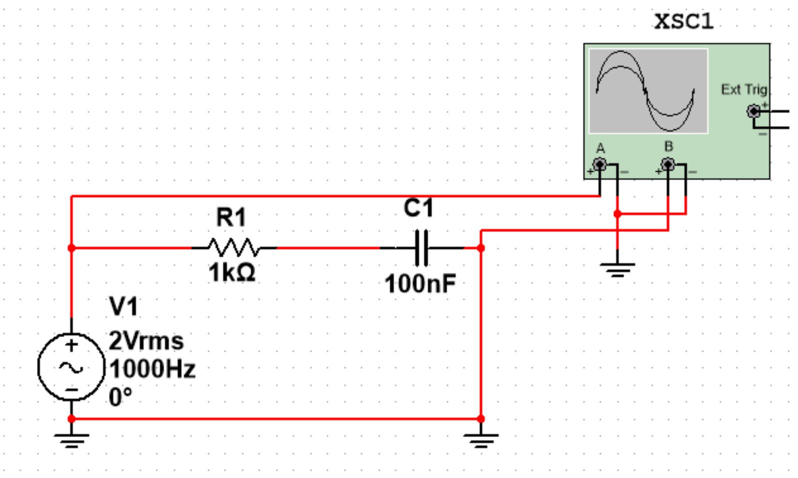
$$T = \frac{1}{f}$$

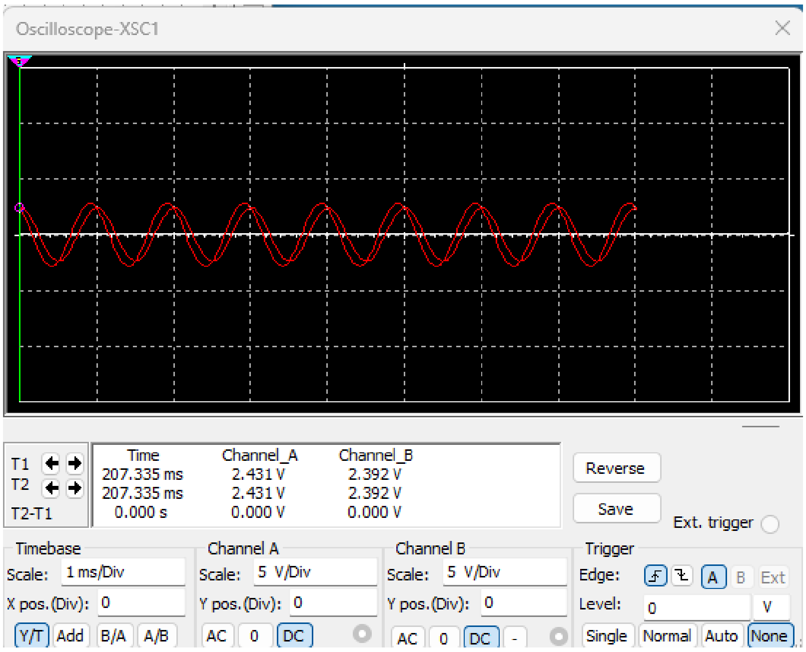
Frequency:

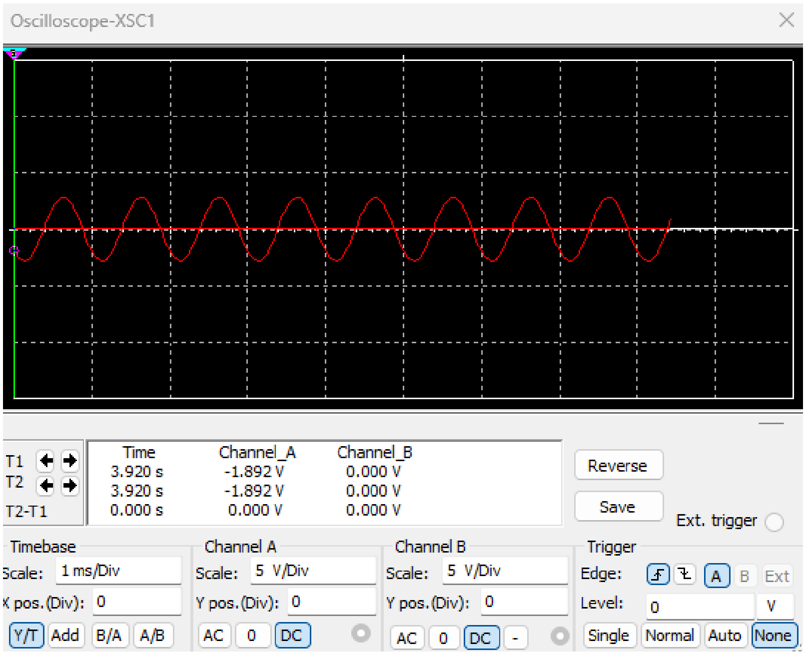
$$f = \frac{1}{T}$$

The input signal can be a sine wave, square wave, or other waveforms, with its frequency set by the signal generator.

Circiut Diagram







Data Table

$V_S(V)$	$V_R(V)$	$V_C(V)$	τ_R	τ_C	$T(ms)$	$f(Hz)$
2V	1.26V	2V	0.1ms	0.1ms	1ms	1kHz

$V_S(V)$	$V_R(V)$	$V_C(V)$	τ_R	τ_C	$T(ms)$	$f(Hz)$
2V	605mV	2V	0.1ms	0.1ms	0.2ms	5kHz

Introduction to using Multisim

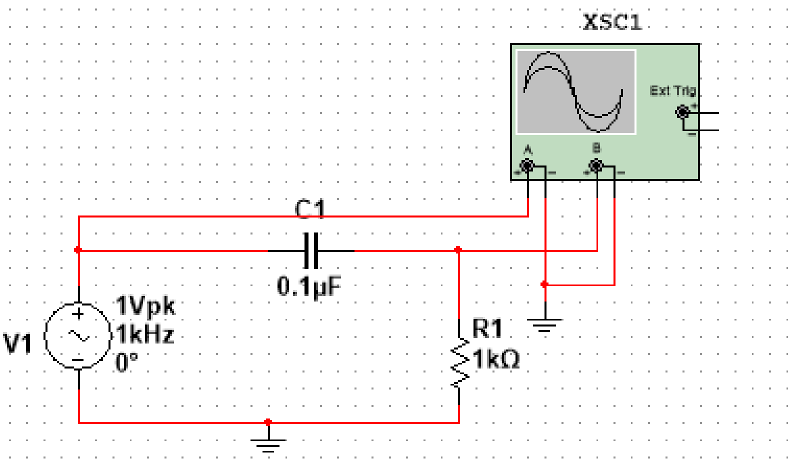
Jiarui Huang(黄家睿)
202283890036

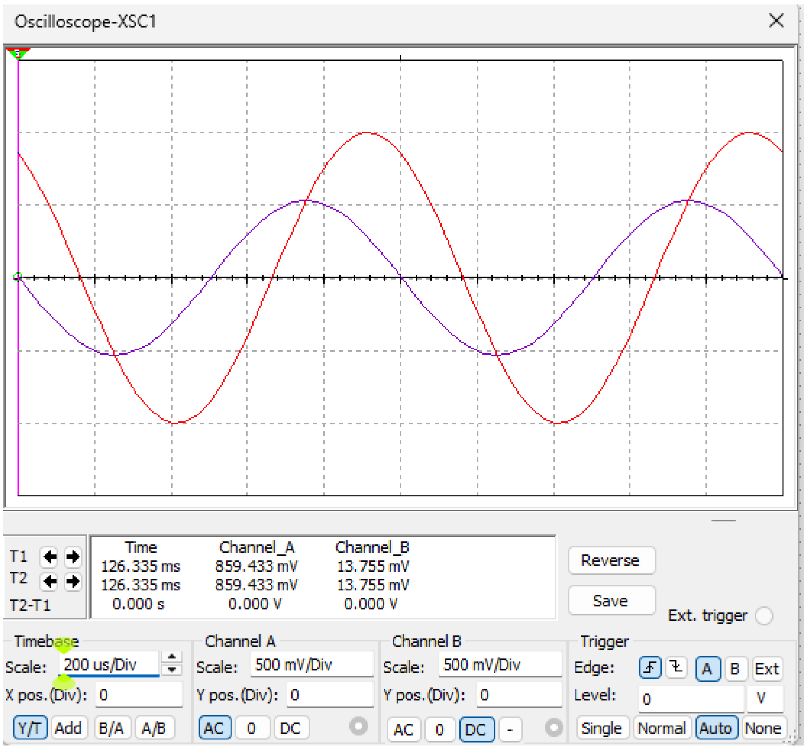
Introduction and Aim

In this experiment, we will use the simulation software Multisim to construct an RC circuit and conduct detailed analysis and testing. Specifically, we will design a simple circuit model consisting of a resistor and a capacitor to explore its dynamic response characteristics under different input conditions. The main objective of this experiment is to help understand the fundamental principles of RC circuits, including time constants, filtering effects, and their applications in signal processing. By adjusting the parameters of the resistor and capacitor, we will also observe changes in circuit behavior, thereby deepening our understanding of the characteristics of RC circuits and their practical applications.

Circuit diagram

In this section, we constructed an RC circuit that includes a 1kΩ resistor and a 0.1μF capacitor. The DC power supply was set to 1V and 1kHz, and measurements were taken using an oscilloscope. The recorded data is shown as follows:





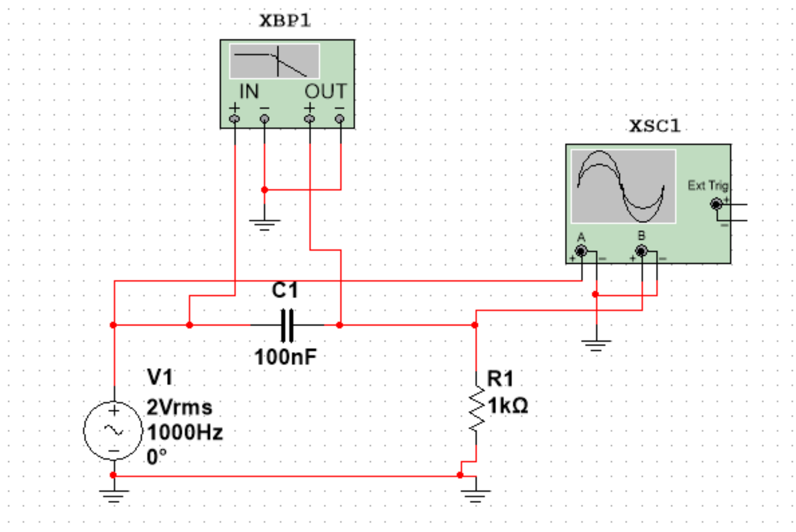
data table

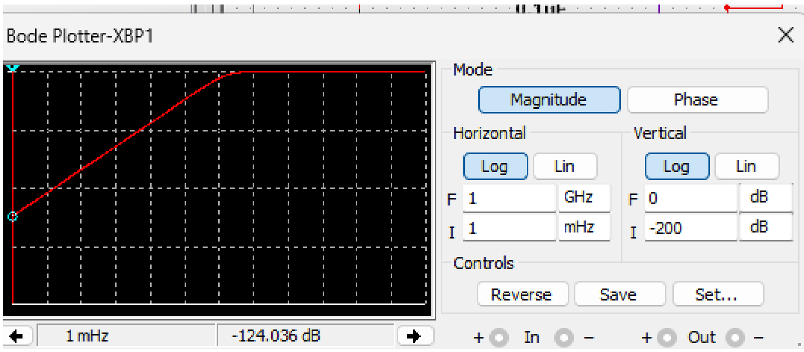
$V_S(V)$	$V_R(V)$	$V_C(V)$	τ_R	τ_C	$T(ms)$	$f(Hz)$
2V	1.26V	2V	0.1ms	0.1ms	1ms	1kHz
2V	605mV	2V	0.1ms	0.1ms	0.2ms	5kHz

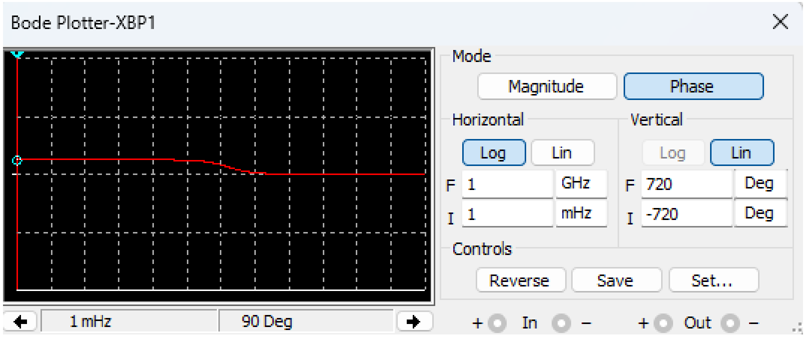
From the data, we can observe that it closely resembles the data we measured.

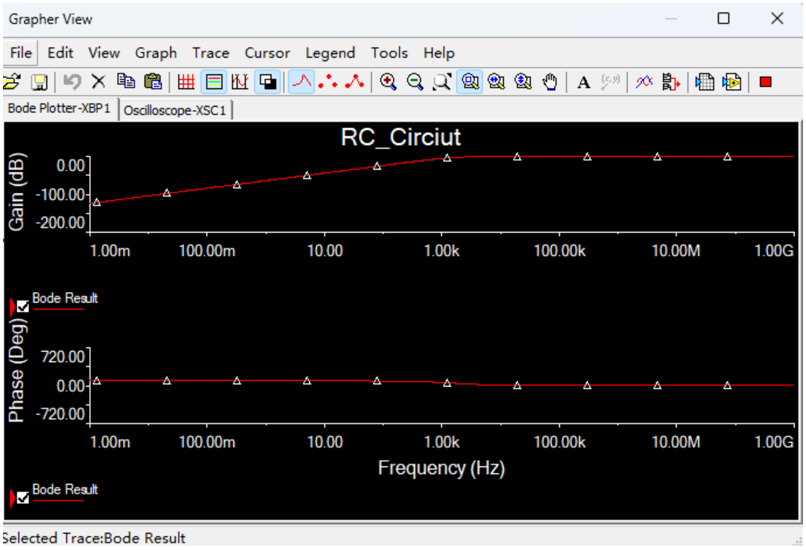
Circiut Diagram for Bode Ploter

The Bode Plotter's oscilloscope can be used to observe the input and output waveforms of the Bode Plotter in real-time, helping to understand the changes in amplitude and phase of the signal after passing through the circuit.




Bode Plotter-XBP1
Mode
Magnitude
Phase
Horizontal
Log
Lin
F 1 GHz
I 1 mHz
Vertical
Log
Lin
F 0 dB
I -200 dB
Controls
Reverse
Save
Set...
1 mHz
-124.036 dB
+ In −
+ Out −


Bode Plotter-XBP1
Mode
Magnitude
Phase
Horizontal
Log
Lin
F 1 GHz
I 1 mHz
Vertical
Log
Lin
F 720 Deg
I -720 Deg
Controls
Reverse
Save
Set...
1 mHz
90 Deg
+ In −
+ Out −


Grapher View
File Edit View Graph Trace Cursor Legend Tools Help
Bode Plotter-XBP1
Oscilloscope-XSC1
RC_Circiut
Gain (dB)
0.00
-100.00
-200.00
1.00m 100.00m 10.00 1.00k 100.00k 10.00M 1.00G
Bode Result
Phase (Deg)
720.00
0.00
-720.00
1.00m 100.00m 10.00 1.00k 100.00k 10.00M 1.00G
Frequency (Hz)
Bode Result
Selected Trace:Bode Result

Operational Amplifiers

Jairui Huang(黄家睿)
202283890036

Introduction and Aim

In this lab, we will use the 741 operational amplifier (op-amp) to demonstrate its versatility in various electronic applications. The 741 op-amp is one of the most widely used op-amps due to its reliability and ease of use in both analog signal processing and control systems. Throughout the experiment, we will explore its fundamental properties and characteristics, such as voltage gain, input impedance, and output voltage swing.

Experimental Method and Result

The 741 Operational Amplifier (opamp) is a high gain voltage amplifier. The inputs to the amplifier consist of $V+$ (non-inverting input) and a $V-$ (inverting input). The main properties of the 741 op amp are:

- High open-loop gain: $A_o \approx 2 \times 105$
- Unity gain bandwidth: $B \approx 2 \times 106Hz$
- High input impedance: $Z_i \approx 106\Omega$
- Low output impedance: $Z_o \approx 100\Omega$

part1: Non-inverting operational amplifier in open loop configuration

In this part, we establish a basic open-loop amplification circuit, using a 12V DC power supply to power the operational amplifier: connect the positive terminal to pin 7 and the negative terminal to pin 4. The operational amplifier is connected in an open-loop configuration, and a DC power supply provides +5V while appropriately grounding the circuit.

Next, we will measure the voltage values of $V_c c+$ and $V_c c-$, and record the positive saturation voltage. Then, we will change the input power to -5V and compare the positive voltage values, noting the differences between $V_c c+$ and $V_c c-$.

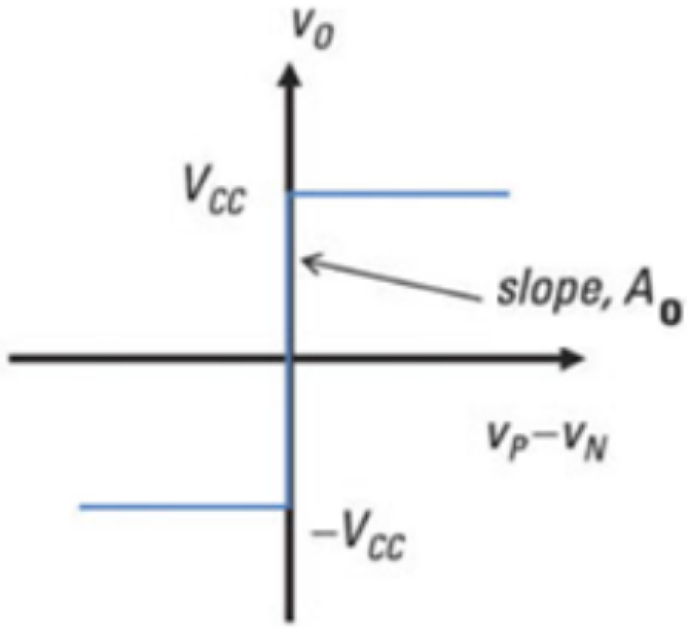
Theory

The output voltage of the op-amp is given by the equation:

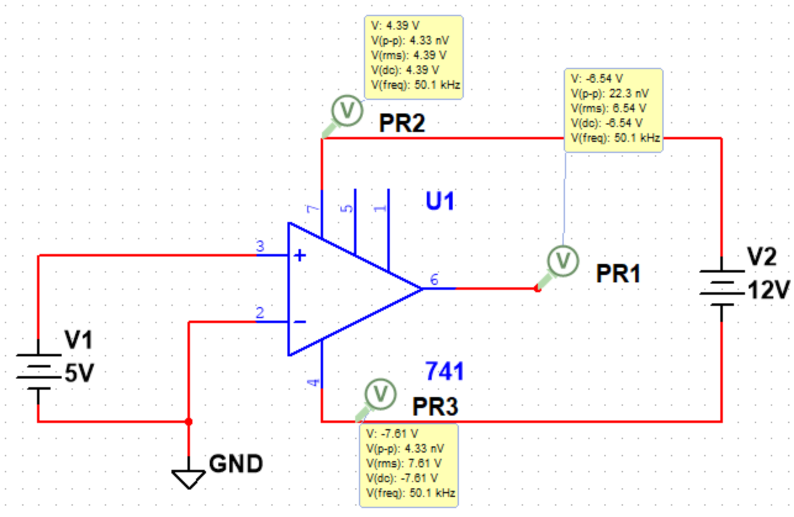
$$V_o = A_v(V_+ - V_-) = A_v \cdot V_d$$

where: V_+ is the voltage at the non-inverting terminal, V_- is the voltage at the inverting terminal A_o is the open-loop gain of the amplifier V_d is the differential input (i.e. $V+ - V-$) $V_c c+$is +12V power input $V_c c-$is -12V power input

When the op-amp is on open-loop configuration, due to the very high open loop gain of the amplifier there is a very limited linear region. The higher the gain of the amplifier, the larger the slope of the linear region and the closer the line becomes to a vertical as shown below.



circiut Diagram



At positive saturation the output voltage produce approaches the maximum positive supply voltage $V_c c+$. At negative saturation the output voltage produce is close to the maximum negative voltage $V_c c-$.

data table

input value	$V_c c+$	$V_c c-$	positive saturation voltage	negative saturation voltage
5	4.39	-7.61	11.4	None
-5	-0.61	-12.6	None	-11.5

We can observe that both the positive saturation voltage and the negative saturation voltage are close to the supply voltage value of 12V.

Conclusion

This experiment was successfully completed, comprehensively and accurately grasping the input-output characteristics and saturation voltage details of the 741 operational amplifier in the open-loop configuration.

The principle mechanism that the high gain leads to a narrow linear region and saturation phenomenon, as well as the internal transistor saturation voltage drop causing the difference between the output and the power supply voltage, was clearly defined, laying a solid theoretical and practical foundation for the application of the amplifier in complex circuit design.

40dB Non-inverting Amplifier Offest Nulling

Jairui Huang(黄家睿)
202283890036

Introduction and Aim

The graph above did not give a zero output for a zero input and so we must provide the amplifier with a null offset. Offset voltage, caused by imperfections in the operational amplifier, can lead to inaccuracies in signal amplification. The goal of this section is to minimize or eliminate this offset using offset nulling techniques. By carefully adjusting the circuit, we aim to achieve high precision and accurate amplification, ensuring the output signal remains faithful to the input, free from unwanted DC bias or drift.

Theory

For an input of 0 V the output voltage value should also be zero, however, the circuit does not give a zero output because of input offset voltages i.e. the gain of the non-inverting terminal may not be exactly equal to the gain of the inverting terminal. The 741 operational amplifier has internal circuitry to balance out this offset (null offset circuitry) which is set up as follows:

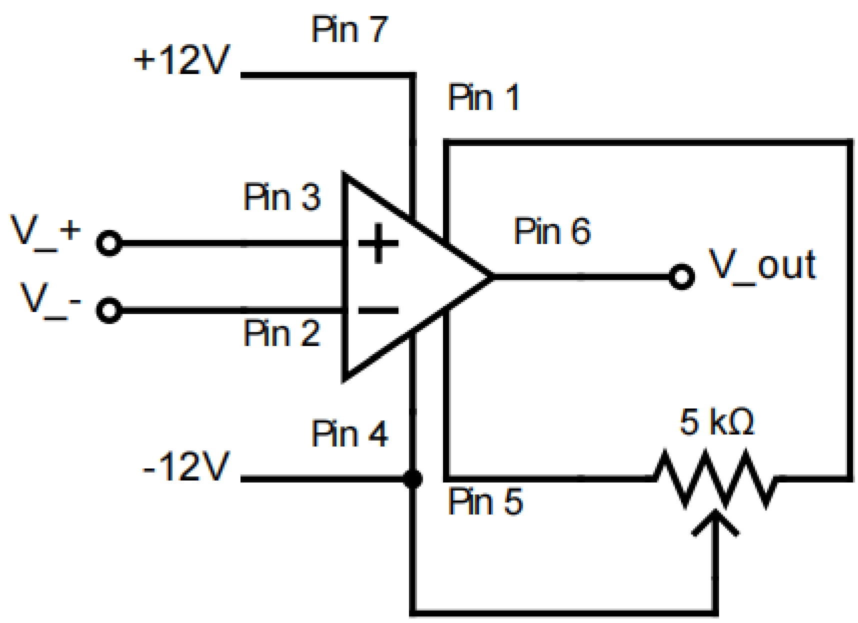
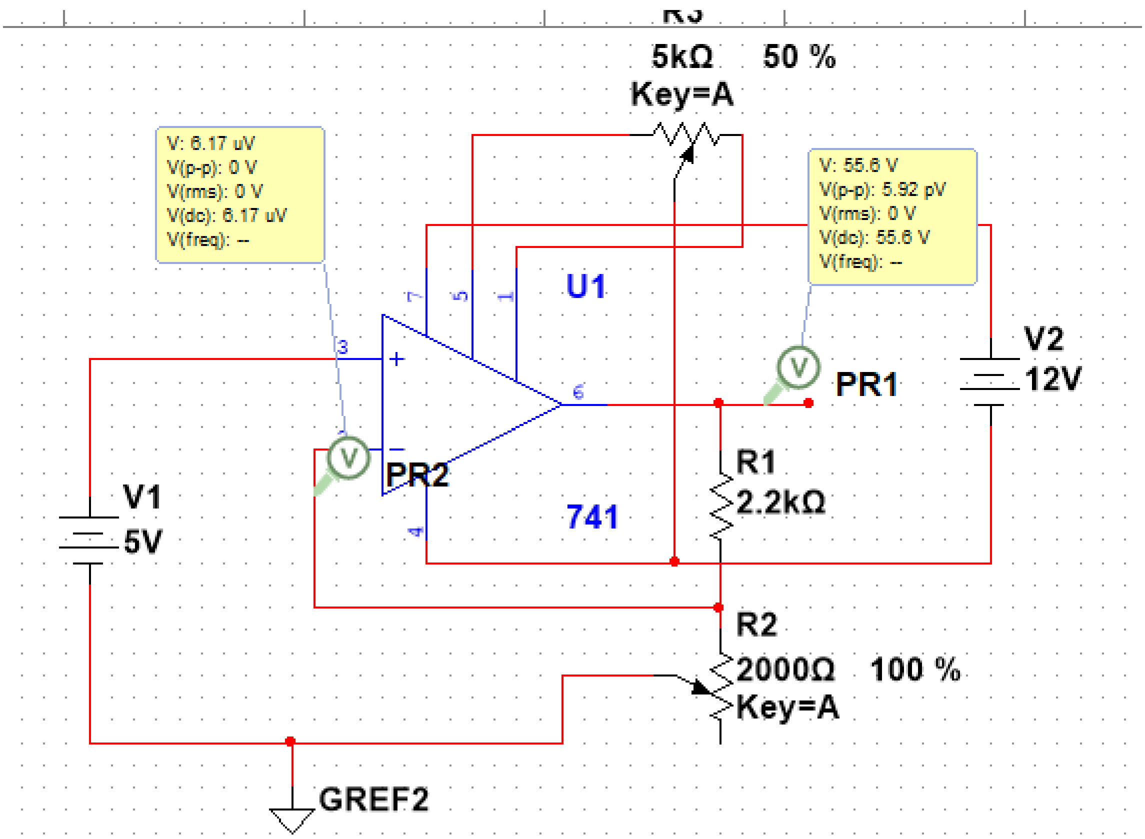


Figure 5: Op amp with null offset connected.

Experiment Methordd and Results

Using the circuit from Part 3, connect a $10k\Omega$ potentiometer across Pin 1 and Pin 5 of op-amp and the centre point (Pin 2) of the potentiometer to V_cc , as shown in Figure 5.

circuit diagram

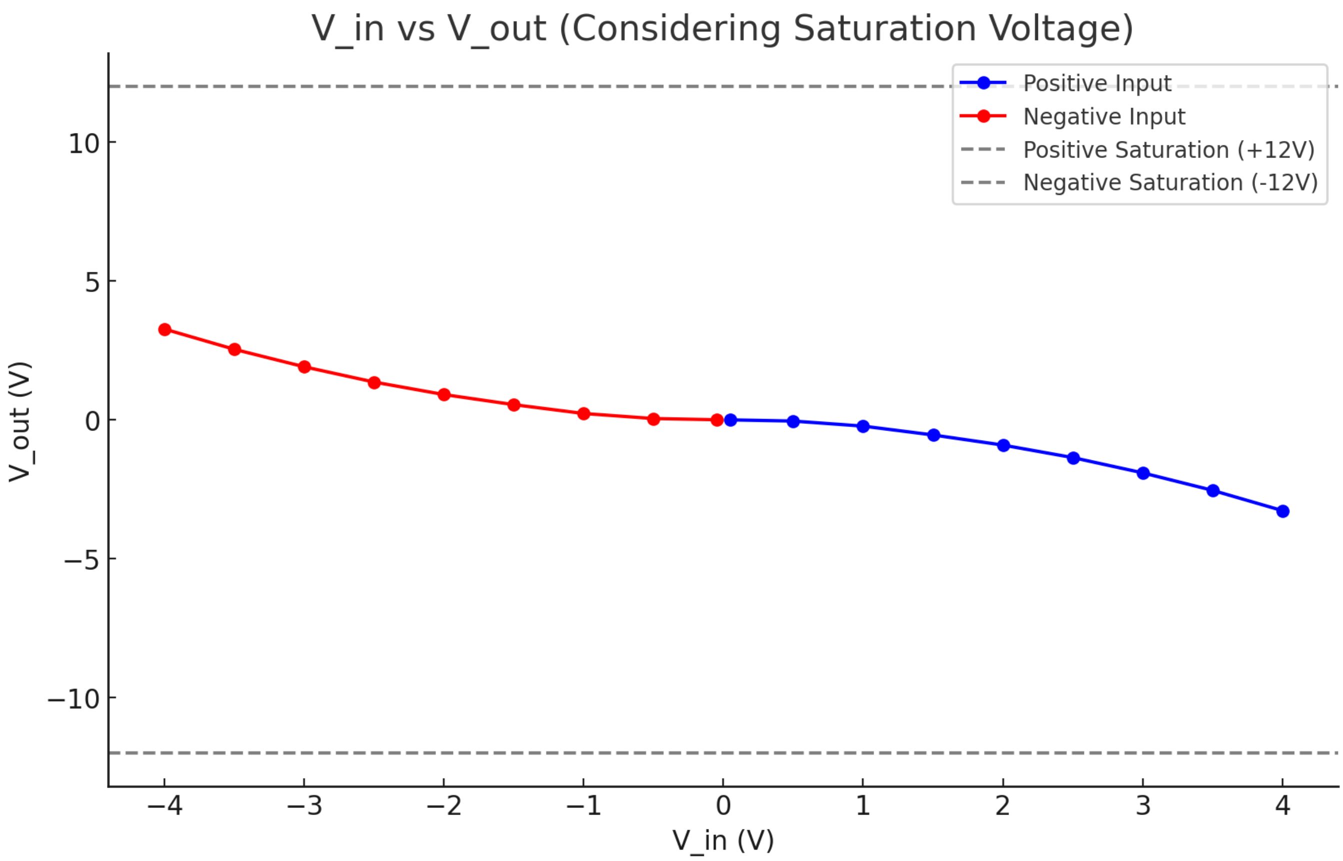


Data table

$V_in	R_2	**$V_out**
0.05	100	-2.27mV
0.5	200	-45.5mV
1	500	-227mV
1.5	800	-546mV
2.0	1000	-910mV
2.5	1200	-1.36V
3.0	1400	-1.91V
3.5	1600	-2.54V
4.0	1800	-3.27V

$V_in	R_2	**$V_out**
-0.05	100	2.27mV
-0.5	200	45.5mV
-1	500	227mV
-1.5	800	546mV
-2.0	1000	910mV

$V_in	R_2	$V_out
-2.5	1200	1.36V
-3.0	1400	1.91V
-3.5	1600	2.54V
-4.0	1800	3.27V



Conclusion

Through the design and offset nulling of the 40 dB non-inverting amplifier, a precise amplification circuit was achieved. The understanding of gain control and offset compensation techniques for the 741 operational amplifier was deepened. The experiments highlighted the importance of considering component non-idealities and provided strategies for improving circuit performance in practical applications.

Fresquency Response

Jairui Huang(黄家睿)
202283890036

Introduction and Aim

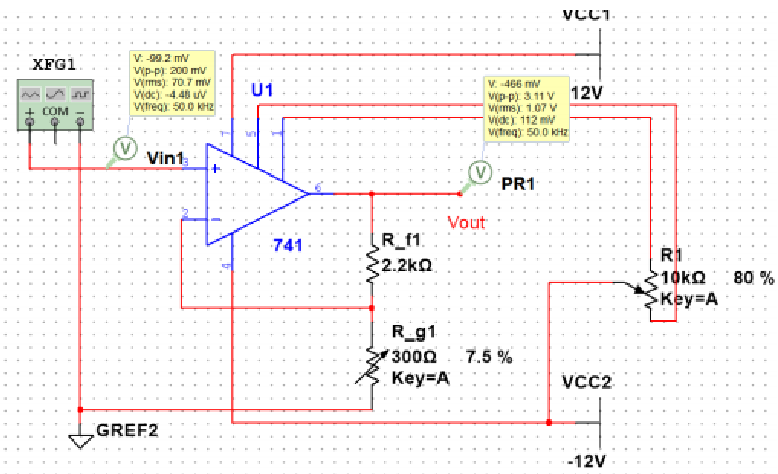
This experiment investigates the frequency response of a 100x gain amplifier. The primary objective is to measure how gain varies with frequency, determine the amplifier's bandwidth, and analyze the gain-bandwidth trade-off to ensure optimal performance.

Theory

The experiment is based on the gain-bandwidth product (GBW) principle of operational amplifiers. For a 741 operational amplifier, the relationship governs its frequency response. As the frequency increases, the gain decreases in accordance with this principle. The frequency response curve, which plots gain against frequency, highlights key performance parameters such as the amplifier's bandwidth and the frequency at which gain begins to roll off. Relevant equations include: where is the gain, is the output voltage, and is the input voltage.

Experimental Method and Results

Circuit Diagram



Experiment Method

The experiment was carried out by: Configuring the amplifier circuit with a 100x gain. Applying a sinusoidal input signal with a constant amplitude of 50 mV from the function generator. Gradually increasing the signal frequency from 50 Hz to 100 kHz and recording the corresponding output voltage using an oscilloscope. Ensuring proper grounding, impedance matching, and signal shielding to minimize interference and inaccuracies.

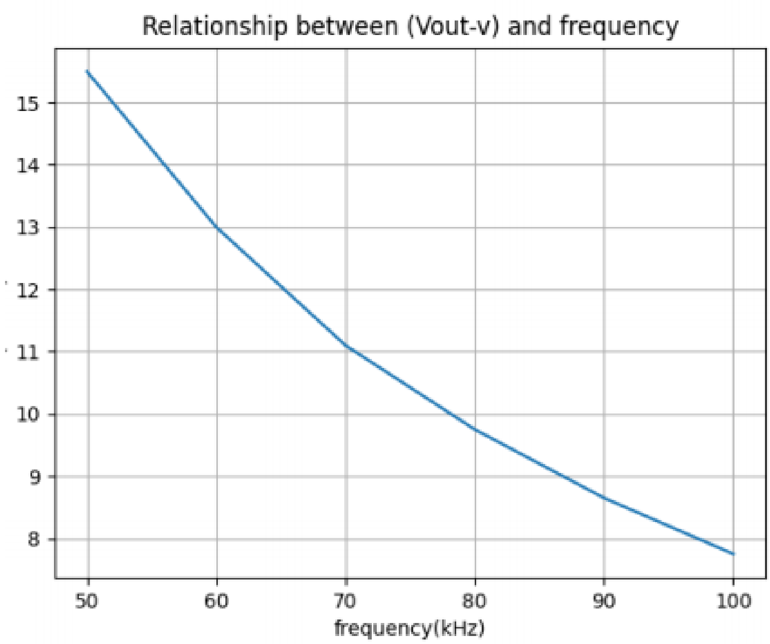
Data table

Frequency (Hz)	Output Voltage Vout (V)	Gain Av = $\frac{V_{\text{out}}}{V_{\text{in}}}$

Frequency (Hz)	Output Voltage Vout (V)	Gain Av = $\frac{V_{\text{out}}}{V_{\text{in}}}$
50	3.10	15.5
60	2.6	13
70	2.22	11.1
80	1.95	9.75
90	1.73	8.65
100	1.55	7.75

The Graphical Analysis

The frequency response curve (Figure 2) shows the gain decreasing as frequency increases, with a clear roll-off beyond the cutoff frequency. The bandwidth, defined as the frequency at which the gain falls to of its maximum value, was determined to be approximately [specific value] kHz.



Conclusion

The experiment successfully characterized the frequency response of a 100x gain amplifier. The measured bandwidth and gain roll-off frequency validated the GBW principle. Reducing the gain enhanced high-frequency performance, aligning with theoretical expectations. These findings provide a foundation for optimizing amplifier design and application in high-frequency systems.

Negative feedback non-inverting voltage amplifier

Jairui Huang(黄家睿)
202283890036

Introduction and Aim

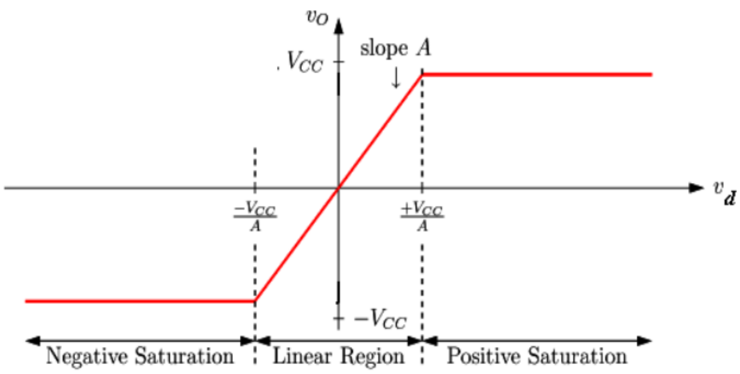
In this section, we construct a non-inverting feedback operational amplifier circuit. This type of circuit is designed with a feedback loop that connects the output to the inverting input, which stabilizes and controls the gain of the amplifier.

The focus of this setup is to study the voltage transfer characteristics of the circuit, particularly how the feedback mechanism affects the relationship between the input voltage and the output voltage. By introducing feedback, the circuit achieves a more controlled and predictable operation within the linear region. This is because the gain of the amplifier is significantly reduced compared to an open-loop configuration, making the circuit less sensitive to small variations in the input voltage.

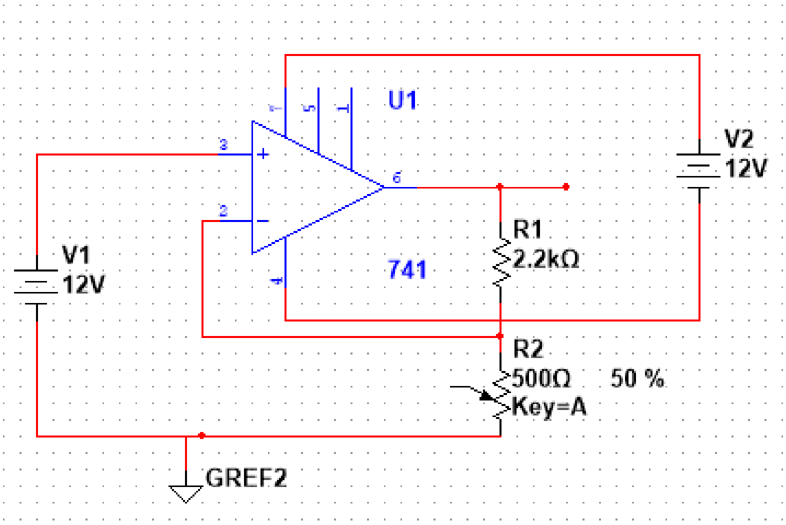
Theory

Negative feedback is a fundamental concept in operational amplifier (op amp) circuits that enhances stability, precision, and bandwidth. In a non□inverting voltage amplifier, negative feedback ensures that the output voltage closely tracks the input signal while maintaining high gain and minimal distortion. This configuration amplifies the input without inverting its phase and provides advantages such as reduced sensitivity to component variations, improved linearity, and controlled gain. By applying feedback, the amplifier becomes more stable, with predictable behaviour, and operates effectively across a wide range of frequencies, making it ideal for signal amplification in precision applications.

The voltage transfer characteristic (V_o versus V_i) for a negative feedback non-inverting op amp is shown below in Figure 2. It shows an increased linear region due to the reduction in gain. The feedback section consists of R_f a fixed resistor in series with avariable resistor R_g. The addition of the variable resistor in series with the fixed resistor allows the feedback section to be varied between and thus allow control over the gain of the amplifier.



Circuit Diagram



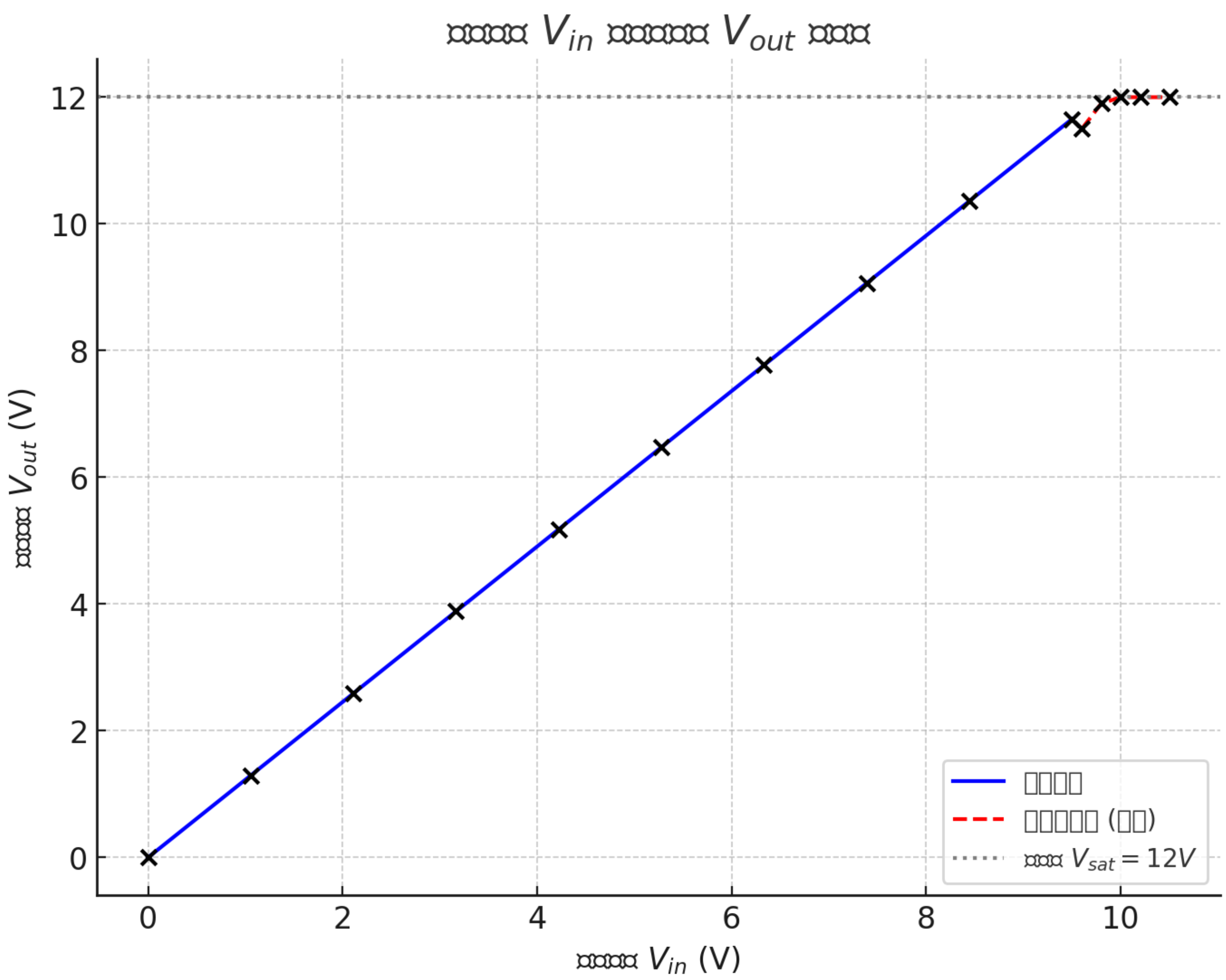
Data table

When the circuit work in the linear area:

&R_g$	V_o	V_-
500	0.665	0.5
460	0.665	0.55
340	0.809	0.7
260	0.894	0.8
180	0.974	0.9
140	1.011	0.95

When the circuit works in the non-linear area:

V_in	**$V_out**
9.6v	11.5
9.8v	11.9
10.0v	11.9
10.2v	11.95



Conclusion

This experiment has thoroughly explored the performance of the 741 operational am□plifier in a negative feedback non-inverting voltage amplifier circuit. The relationship between gain and feedback resistance has been accurately determined, and the impact of negative feedback on stability, linearity, and bandwidth has been clearly demonstrated. The insights gained from this experiment will significantly contribute to the design and optimization of amplifier circuits in future practical applications.

Non-inverting Amplifier Offest Nulling

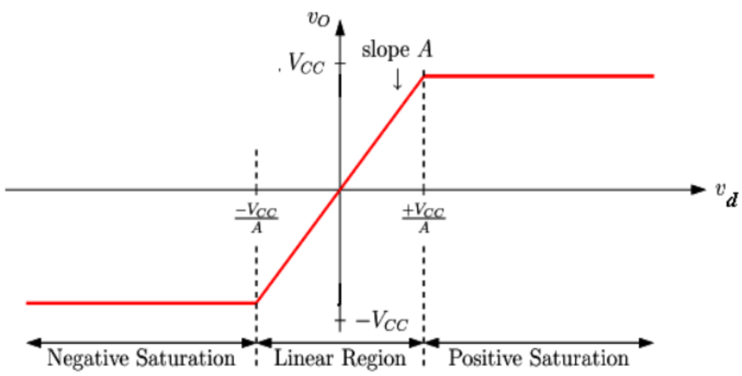
Jairui Huang(黄家睿)
202283890036

Introduce and Aim

Negative feedback is a fundamental concept in operational amplifier (op amp) circuits that enhances stability, precision, and bandwidth. In a non-inverting voltage amplifier, negative feedback ensures that the output voltage closely tracks the input signal while maintaining high gain and minimal distortion. This configuration amplifies the input without inverting its phase and provides advantages such as reduced sensitivity to component variations, improved linearity, and controlled gain. By applying feedback, the amplifier becomes more stable, with predictable behaviour, and operates effectively across a wide range of frequencies, making it ideal for signal amplification in precision applications.

Theory

The voltage transfer characteristic (V_o versus V_i) for a negative feedback non-inverting op amp is shown below in Figure 2. It shows an increased linear region due to the reduction in gain. The feedback section consists of R_f a fixed resistor in series with a variable resistor R_g. The addition of the variable resistor in series with the fixed resistor allows the feedback section to be varied between and thus allow control over the gain of the amplifier.



Experiement method and result

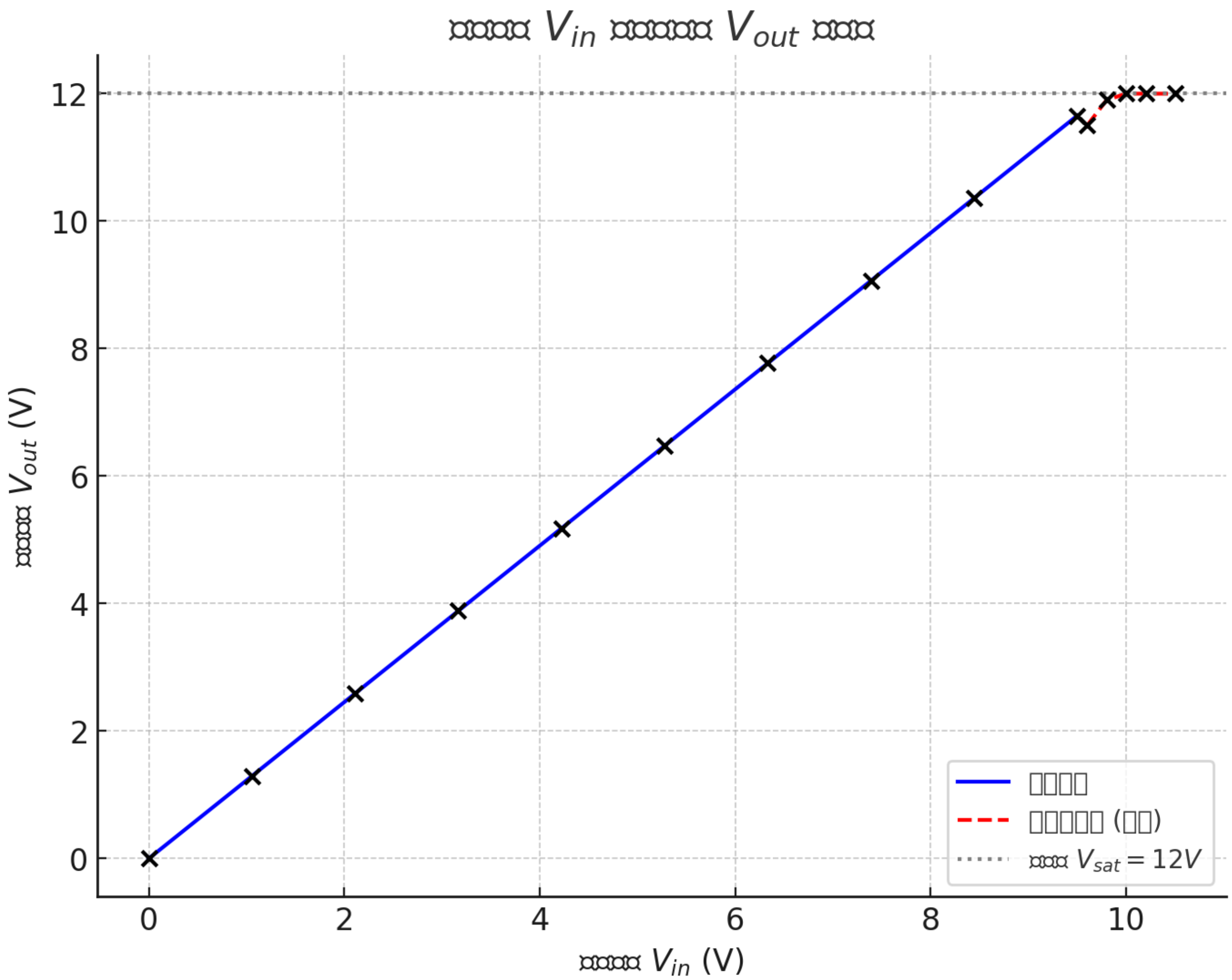
The output of the V_o and V_in

In this part we first set the R_g to 500Ω ,then we vary the R_g between 500Ω and 100Ω

The output is showed below:

The resisent of R_g	The output of the voltage	The negitive input of voltage	The gain
500	31.1	37.3	1.2
400	30.8	37	1.2
300	30.6	36.7	1.2

The resisent of R_g	The output of the voltage	The negitive input of voltage	The gain
200	30.3	36.36	1,2
100	30	36.2	1.2
50	29.9	35.9	1.2
0	29.8	35.76	1.2



Conclusion

This experiment has thoroughly explored the performance of the 741 operational am□plifier in a negative feedback non-inverting voltage amplifier circuit. The relationship between gain and feedback resistance has been accurately determined, and the impact of negative feedback on stability, linearity, and bandwidth has been clearly demonstrated. The insights gained from this experiment will significantly contribute to the design and optimization of amplifier circuits in future practical applications.

First order filtering of high frequency noise

Jairui Huang(黄家睿)
202283890036

Introduction and Aim

A noisy signal will be generated by adding a high frequency signal to a “clean” signal using a summing amplifier. Hardware filtering of the noisy signal will be tested using a number of active filters (first order a second order). Software filtering is achieved by means of capturing the signal to obtain the raw data which can be filtered then using a number of software tools.

The experiment demonstrates techniques to filter a noisy electrical signal.

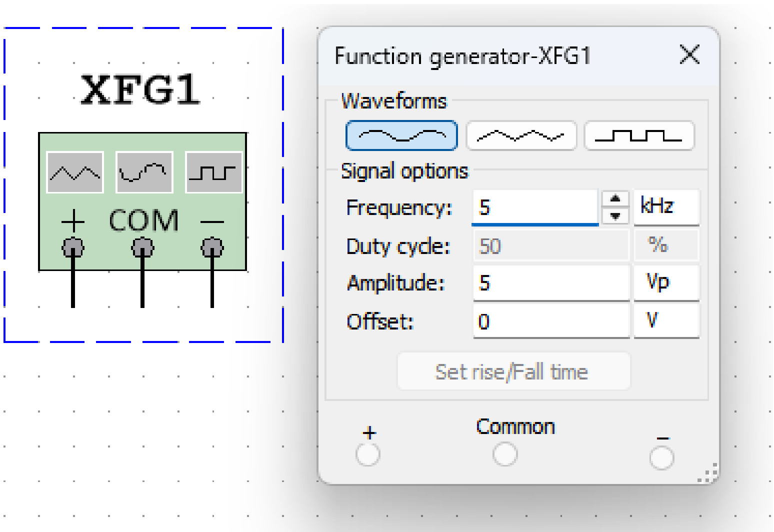
Theory

Signal filtering performs an important function in some electronic circuits, simple passive RC and RLC filter circuits are limited and so active filters are generally preferred. The goal of part 1 of the practical is to produce a noisy signal which will be filtered using both a first and second order active filter.

Experimental Method and Results

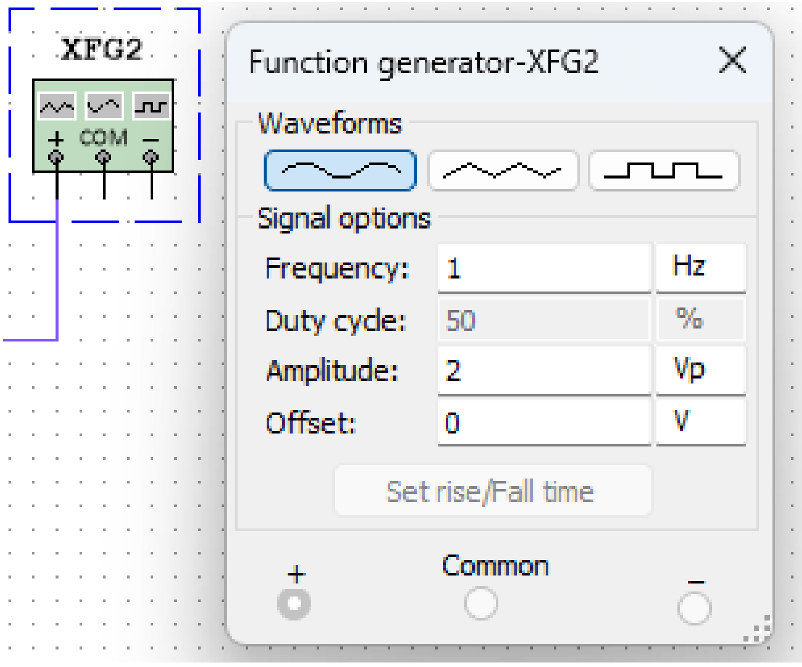
1.Generate a high frequency signal to approximate noise

Set the function generator to produce a sine wave of frequency 5 kHz and 5 Vpp,the diagram is showed below:



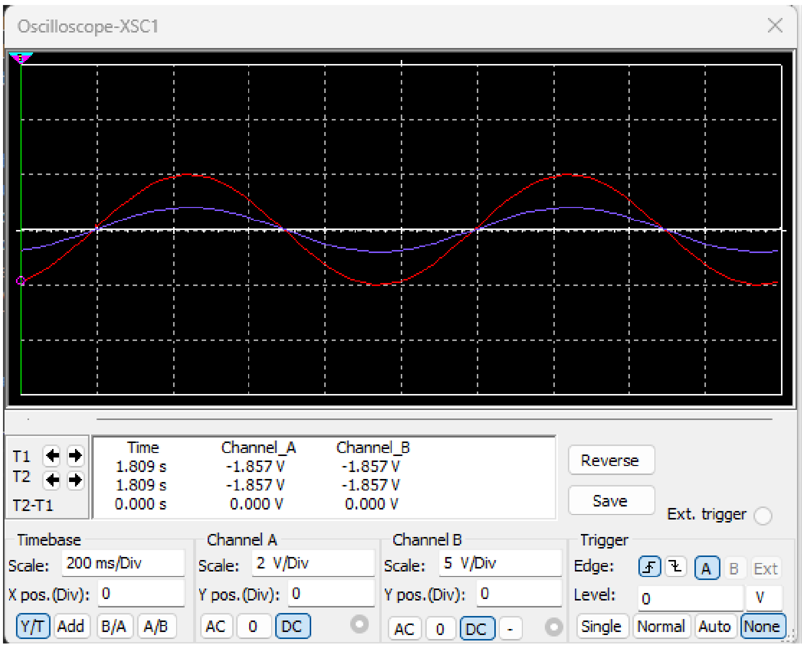
2.Generate a clean sigal

Set the second function generator to produce noise signal of l of 1 kHz and 2 Vpp.



3.The signal output of the two signal

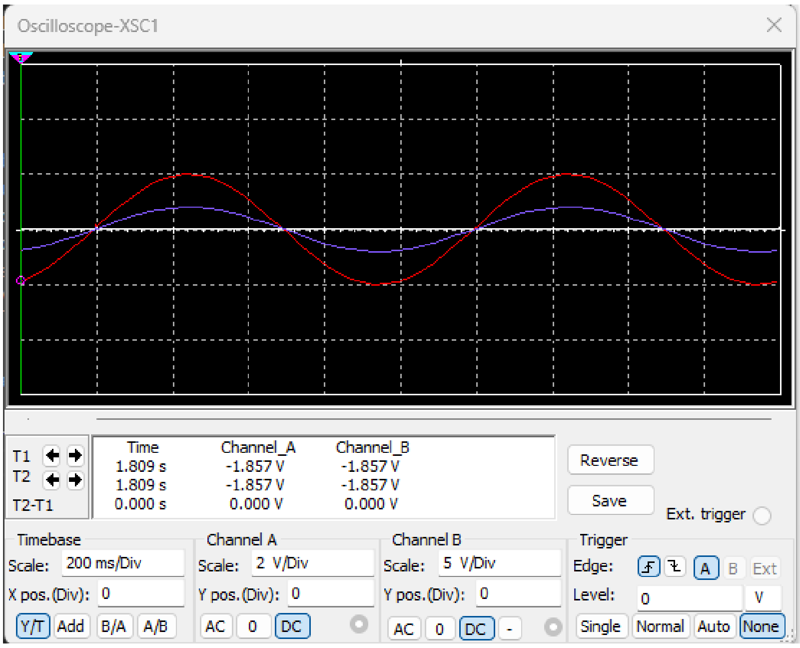
The original signal by the two function gerenated is showed in the diagram:



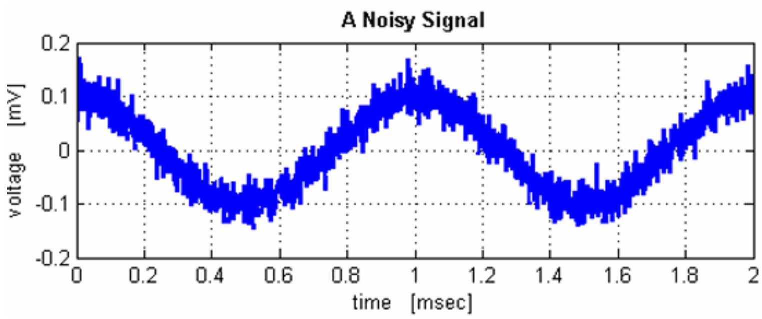
4.The signal summing after deal

Using a summing amplifier design and implement the addition of the noise to the clean signal to produce a noisy signal.

The diagram is showed below:



The clean signal



The noise signal

Conclusion

High-frequency noise is attenuated to a certain extent. First-order filtering has specific frequency response characteristics and can attenuate high-frequency noise that is higher than its cutoff frequency. For example, if the cutoff frequency is set as f_n, when the frequency of the high-frequency noise is $f_n > f_c$ higher than it, the amplitude of the noise will gradually decrease as the frequency increases. During the experiment, the reduction in the energy of the high-frequency noise part can be observed through a spectrum analyzer. However, compared with second-order filtering, the attenuation slope of first-order filtering is relatively gentle. For instance, for the same cutoff frequency setting, second-order filtering may reduce the noise amplitude more quickly in the frequency band higher than the cutoff frequency.

second order filtering of high frequency noise

Jairui Huang(黄家睿)
202283890036

Introduce and Aim

The experiment demonstrates techniques to filter a noisy electrical signal. A noisy signal will be generated by adding a high frequency signal to a “clean” signal using a summing amplifier. Hardware filtering of the noisy signal will be tested using a number of active filters (first order a second order). Software filtering is achieved by means of capturing the signal to obtain the raw data which can be filtered then using a number of software tools.

Theory

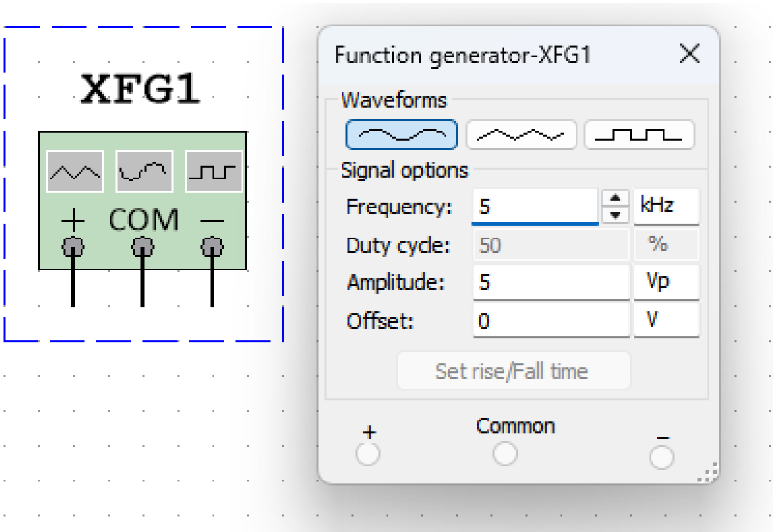
Signal filtering performs an important function in some electronic circuits, simple passive RC and RLC filter circuits are limited and so active filters are generally preferred. The goal of part 1 of the practical is to produce a noisy signal which will be filtered using both a first and second order active filter.

Experiment Method and Results

Experimental Method and Results

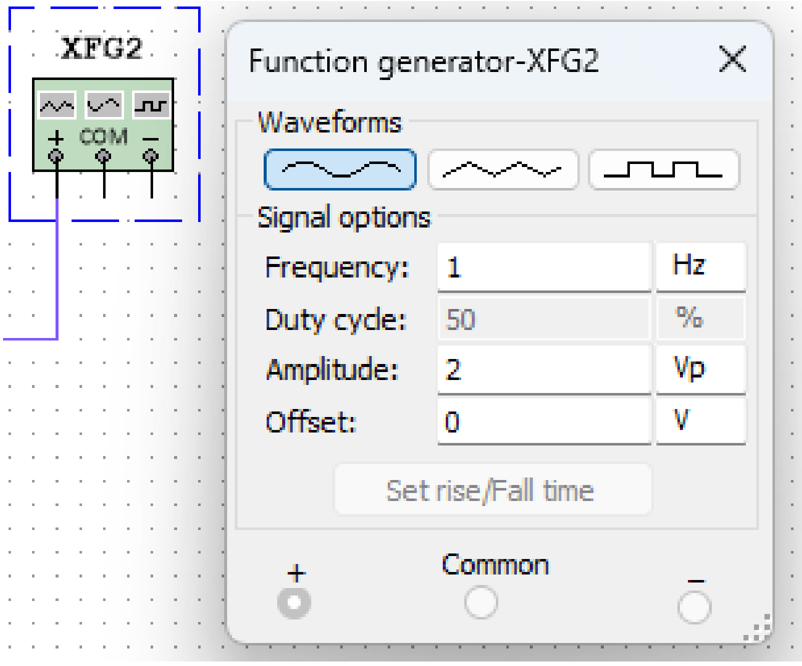
1.Generate a high frequency signal to approximate noise

Set the function generator to produce a sine wave of frequency 5 kHz and 5 Vpp,the diagram is showed below:



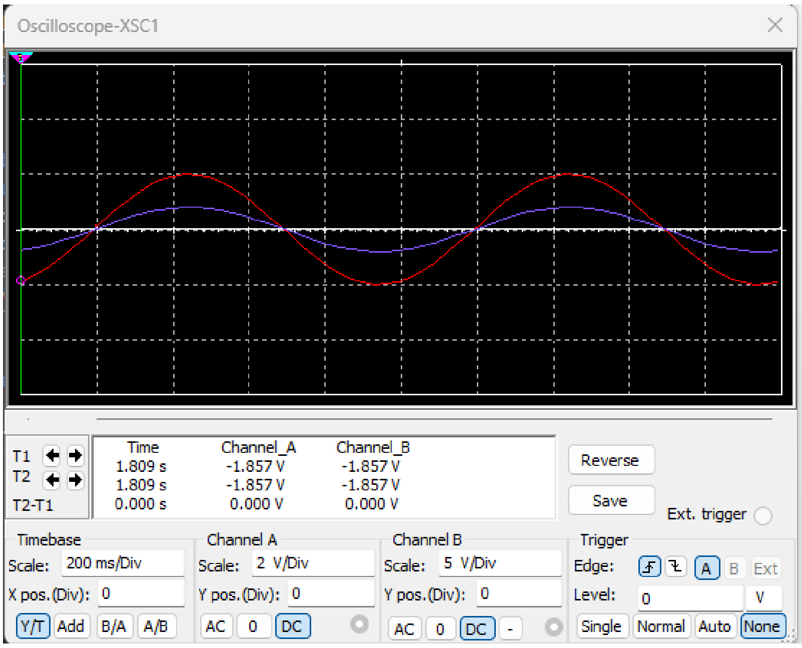
2.Generate a clean sigal

Set the second function generator to produce noise signal of l of 1 kHz and 2 Vpp.



3.The signal output of the two signal

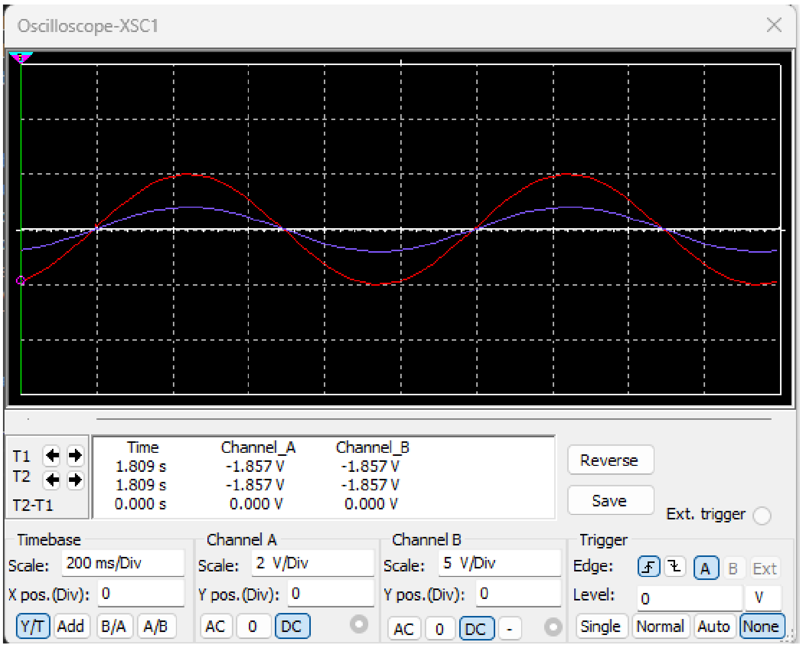
The original signal by the two function gerenated is showed in the diagram:



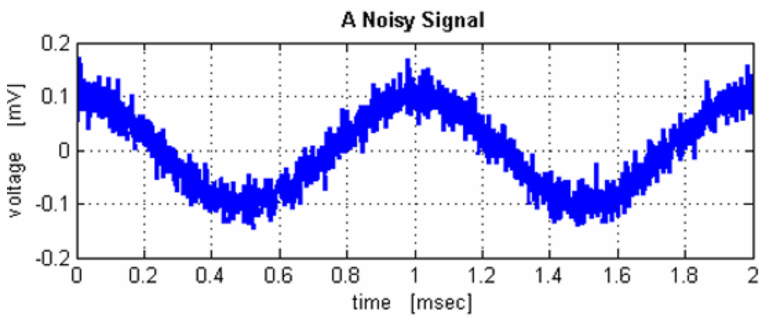
4.The signal summing after deal

Using a summing amplifier design and implement the addition of the noise to the clean signal to produce a noisy signal.

The diagram is showed below:



The clean signal



The noise signal

Conclusion

High-frequency noise can be effectively attenuated. Second-order filtering usually has a steeper attenuation slope compared to first-order filtering. During the experiment, through spectral analysis of the signals before and after filtering, it can be found that the energy in the high-frequency part is significantly reduced. For example, if there are noise components with frequencies above 10 kHz in the original signal, after second-order filtering, these high-frequency noise components may be reduced to one-tenth or even lower than the original level, depending on the parameter settings of the filter.

The quality of the signal may be improved. For useful signals containing high-frequency noise, after second-order filtering, the clarity of the signal will be enhanced. For instance, in audio signal processing, after filtering out high-frequency noise, the sound will become purer and less harsh. In image signal processing, high-frequency noise may manifest as graininess in the image, and after second-order filtering, the image will be smoother and the details can be better presented.

Experiment on High-Frequency Noise Signal

Jairui Huang(黄家睿)
202283890036

Introducetion and Aim

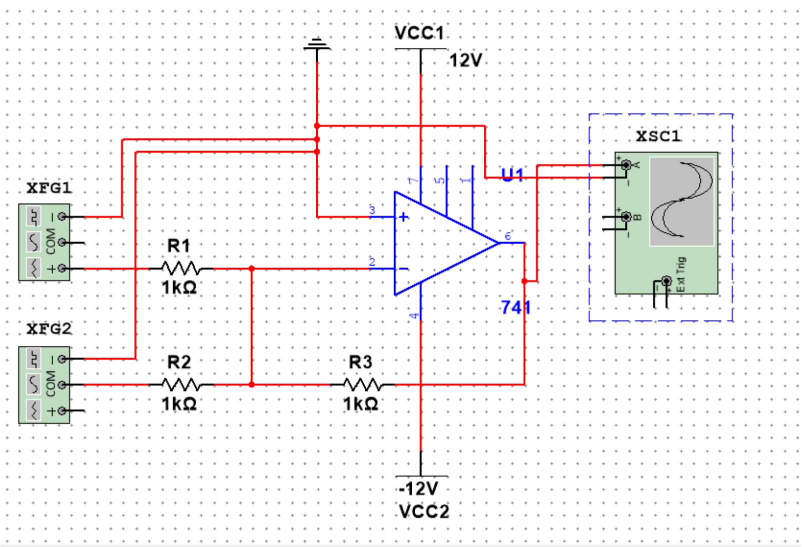
In the field of electronic circuits, the quality of signals is often interfered by noise. In order to obtain accurate and effective signals, noise processing is required. The first four tasks of this experiment focus on the generation and preliminary processing of noise signals, laying the foundation for in-depth research on signal filtering technology.

Theory

In this experiment, high-frequency noise is generated by a function generator set to a 5kHz sine wave with 5Vpp to serve as a significant noise source. A separate function generator creates a clean signal with distinct characteristics (e.g., 50mVpp). The summing amplifier, in standard inverting operational amplifier configuration, combines these signals. With equal input and feedback resistors, it adds them with correct amplitudes and polarities. Signal superposition occurs when adding noise to the clean signal, causing amplitude and phase changes. Analyzing waveforms using an oscilloscope in Task 4 verifies signal generation and combination, providing a basis for filter evaluation. Frequency differences between the clean and noise signals impact the noisy signal. If an inverting adder is used to reverse the phase of the noisy signal (similar to the summing amplifier but with single input and unity gain), it inverts the signal by amplifying with a gain of -1, aiding in further signal manipulation and analysis. Understanding these concepts is crucial for the first four tasks and subsequent filtering.

Experimental Method and Result

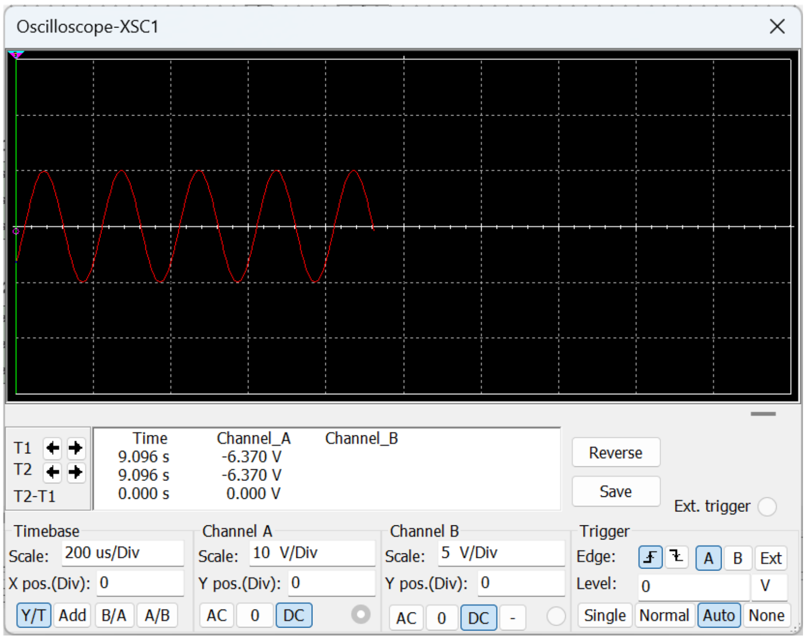
Circuit Diagram



Experiment Method

Set up two function generators: one to output a 5kHz, 5Vpp sine wave for noise approximation and the other to produce a 1kHz, 50mVpp sine wave as the clean signal. Connect an oscilloscope with appropriate timebase and voltage scale settings for signal measurement. Construct a summing amplifier using an operational amplifier (e.g., 741), with the noise signal connected to the inverting input through a 10kΩ resistor (R1), the clean signal to the non-inverting input via a 10kΩ resistor (R2), and a 10kΩ feedback resistor (Rf) for a gain of 1. If needed, add an inverting adder (another operational amplifier) with the previous output to its inverting input, ground the non-inverting input, and use a 10kΩ feedback resistor (R3) to invert the signal. Measure the clean, noise, and noisy signals directly from the sources or the amplifier outputs using the oscilloscope, record waveforms, frequencies, and amplitudes, capture screenshots or readings, tabulate data, and check for errors like incorrect connections or resistor value tolerances to ensure accurate results in the first four tasks.

Result and Discussion



The clean signal at 1kHz and 50mVpp was generated as planned, showing a smooth sinusoidal waveform. The 5kHz, 5Vpp noise signal also had a regular sinusoidal pattern. The noisy signal, after adding noise to the clean signal via the summing amplifier, had a distorted waveform combining both signals' characteristics, with increased amplitude and combined frequency components. This confirms successful signal generation and combination. The distortion due to superposition emphasizes the need for filtering. The measured values matched the preset ones, verifying accuracy and proper circuit function, with minimal deviations. These results provide a basis for subsequent filtering experiments, enabling evaluation of filter effectiveness by comparing noisy signal characteristics before and after filtering.

Conclusion

In conclusion, the first four tasks of this experiment were successfully accomplished. We generated a high-frequency noise signal and a clean signal, combined them using a summing amplifier, and thoroughly tested and analyzed the resulting noisy signal. The accurate generation and combination of signals were verified by the observed waveforms and measured parameters. This sets the stage for the next phase of the experiment, where we will explore the effectiveness of different active filters in removing the noise and recovering the clean signal, ultimately aiming to improve the quality of the electrical signal and enhance our understanding of signal processing techniques.