Table of Contents

Prerequisite Knowledge for Experiments

Safety Rules in the Laboratory

A. General Precautions

(1) Follow the instructions of the experiment instructor/teaching assistant carefully.
(2) Familiarize yourself with the experiment's purpose, principles, and precautions before starting.
(3) Maintain quiet in the laboratory and refrain from unnecessary actions.
(4) When conducting experiments with two or more people, cooperate with each other during the experiment and do not adopt a bystander attitude.
(5) Record measurement results systematically.
(6) After completing wiring, perform a verification procedure before turning on the power switch.
(7) After the experiment, tidy up as instructed by the teaching assistant and turn off the power to all equipment.

B. Pre-Experiment Precautions

(1) Fully understand the experiment's content and accurately grasp its purpose to be prepared to begin immediately.
(2) Before starting the experiment, familiarize yourself with the operation of the equipment to be used.

C. Precautions During the Experiment

(1) Record all occurrences during the experiment in detail.
(2) Operate equipment with care and do not apply excessive force to moving parts.
(3) If an experiment goes wrong, stop it immediately, review the procedure, and identify the cause of the error.
(4) Record measurements using tables.

D. Precautions During Wiring

(1) Before beginning wiring, ensure the power supply and function generator are turned off.
(2) When connecting wires to the terminals of experimental equipment, always verify that the correct terminal is being used.

(3) Use appropriate wires for the circuit, considering the current flowing through it and voltage drops.
(4) Verify that the circuit connections are correct before turning on the power supply.

E. Preventing Electrical Accidents

The risk of electric shock depends on the amount of current, the type of power source, the duration of exposure, the path of current through the body, and the condition of the person. Human electrocution occurs when a part of the body comes into direct contact with a live wire or touches a device with a leakage current. While the situation may vary depending on how the body contacts the live circuit, the effects of electrocution are classified as follows based on the magnitude of the current flowing through the body.

- 1 mA: Perceptible sensation of electricity
- 5 mA: A noticeable flow of electricity
- 10 mA: Pain difficult to endure
- 20 mA: Severe muscle contraction makes voluntary movement impossible
- 50 mA: Dangerous condition
- 100 mA: Can cause fatal consequences

The above figures are approximate values, and the severity of shock varies depending on a person's constitution, age, gender, and health condition. The impact of electric shock on the human body is related not only to the magnitude of the current but also to the duration it flows. While the lethal current is 165mA per second, the limit drops to about 500mA for 0.1 seconds. The magnitude of current flowing through the human body varies depending on the body's condition. For example, in a 200V circuit, assuming skin resistance of the hand is 2500 ohms, resistance between the foot and shoe is 1500 ohms, and resistance between the shoe and ground is 700 ohms, the total resistance is approximately 5000 ohms. This results in a current of about 40mA flowing through the body, which is extremely dangerous. Furthermore, if the hands are damp or the shoes are wet, the resistance decreases significantly. Consequently, even a 100V voltage could cause death within about 0.3 seconds. Therefore, strict adherence to safety rules is essential when handling electrical appliances.

Handling Measurement Values

A. Significant Digits Processing

The number of significant digits is crucial in handling and calculating measured values. When expressing a measured value numerically, the last digit is often obtained by rounding the digits below it, and we can assume the digit immediately preceding the rounded last digit is accurate. In this case, the rounded last digit is included as part of the significant digits. For example, if the number 1.2345678 is rounded to 1.235, the four digits including the rounded last digit 5 are considered the significant figures.

When performing operations such as addition, subtraction, multiplication, and division using measured values, it is necessary to consider the number of significant digits. For addition and subtraction, the number of significant digits in the result is limited to the number of significant digits with the fewest decimal places. For example, representing the result of 27.19 + 10.275 as 37.465 is meaningless. Although the last digit of 10.275 is the third decimal place, the significant figures of 27.19 extend only to the second decimal place. Therefore, adding the third decimal place to obtain the final digit '5' in the result is meaningless. In this case, limiting the decimal places to two and expressing the result as 37.47 is correct. If the same calculation were 27.190 + 10.275, the result could be expressed as 37.465. For multiplication and division, the number of significant digits in the result is limited to the smallest number of significant digits. For example, in 12.34 × 1.23 = 15.1782, the result should be expressed as 15.2.

The above content can be applied to the following example in circuit measurement. For instance, consider calculating the combined resistance value when two resistors are connected in series. Assuming one resistor measures 100.2 ohms and the other measures 10.34 ohms, the combined resistance value of the two resistors connected in series should be expressed as 110.5 ohms, not 110.54 ohms. Consider another example where the resistance value is calculated by measuring the voltage across the resistor and the current flowing through it. If a voltage of 35V is applied to a resistor and the measured current is 13mA, the calculated resistance value is

$$R = \frac{35}{13 \times 10^{-3}} = 2692.3077\Omega,$$

but it is correct to express it as 2.7 by limiting the number of significant digits to two.

B. Types of Error

When measuring any physical quantity, no matter how precise the method used, it is impossible to read the true value, and errors can always occur. There are various causes of error, which can be classified into the following types:

- Theoretical error
- Instrumental error
- Personal error
- Negligent error
- Random error

(1) Theoretical error arises from the theoretical basis, such as when certain assumptions are made in the measurement theory or relationship equation, or when certain terms are omitted.
(2) Instrumental error occurs due to abnormalities in the equipment used and can be reduced by repairing or calibrating the measuring instrument.
(3) Personal error arises from the habits of the measurer. It can be reduced by increasing the number of observers and taking the average, or by conducting repeated experiments.
(4) Negligence error occurs when readings are misread or records are incorrectly written. Since it results from carelessness or oversight, it can be detected by carefully reviewing the measurements or by plotting graphs.
(5) Random errors stem from causes difficult to arbitrarily control, such as variations in measurement conditions or the observer's distracted attention. Therefore, they are difficult to simply eliminate. However, random errors are known to occur according to a pattern following Gauss's law. This law holds under the following three assumptions:

- Errors in positive and negative values always occur uniformly.
- Small errors occur more frequently than large errors.
- Errors larger than a certain threshold do not occur.

While it is difficult to artificially reduce random errors, analyzing their impact on results becomes possible by leveraging the fact that they occur in patterns such as the one described above.

Lab 1. Multimeter Usage and Basic Circuit Theory

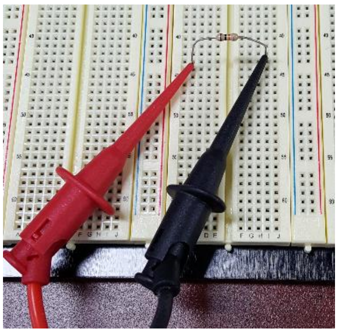
● Multimeter Usage

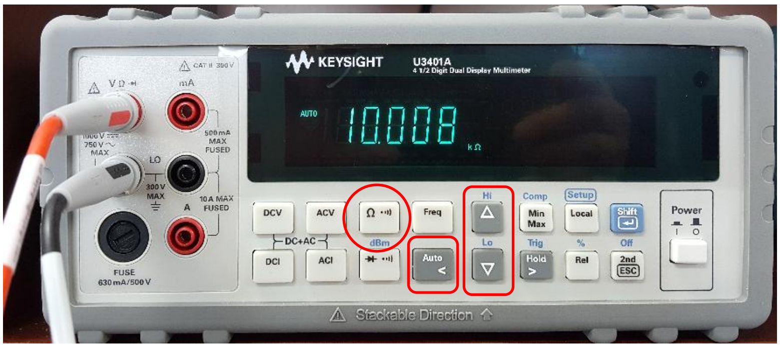
A multimeter is a measuring instrument capable of measuring resistance, voltage, current, etc. Its front panel is as shown in the following figure.

To measure resistance, voltage, current, etc., using a multimeter, insert the probes as shown in the following figure into the front panel of the multimeter. Typically, the black probe is inserted into the black terminal (COM) on the front panel, and the red probe is inserted into the red terminal. However, since there are multiple red terminals, insert the probe into the appropriate terminal depending on the value or signal being measured. In some cases, using alligator clips connected as shown in the right diagram below allows measurement without touching the device, which can be convenien.

● Measuring Resistance

To measure resistance with a multimeter, connect the black probe to the left black terminal (COM) and the red probe to the left red terminal marked V-Ω, as shown in the figure below. Next, press the button labeled among the selection buttons. Touch the probes to both terminals of the resistor you wish to measure, then read the displayed value. For accurate resistance measurement, ensure nothing else touches both ends of the resistor besides the multimeter probes.



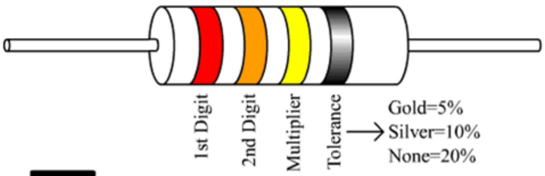


When measuring with a multimeter, there are two methods for setting the measurement range: automatic and manual. Each press of the Auto key on the front panel toggles between automatic and manual mode. When set to automatic, the display shows the letter “AUTO.” The image above shows the display when set to automatic. In manual mode, you can use the Hi and Lo buttons to set the measurement range to a higher or lower value. Pressing the Hi or Lo button while in auto mode switches the setting to manual. While auto mode is convenient, it can sometimes be inaccurate. Therefore, manual setting is recommended for precise measurements.

Resistor values are typically indicated by a color code, with the code values for each color shown in the following table.

Color	Digit	Multiplier	Tolerance (%)
Black	0	10^0 (1)	
Brown	1	10^1	1
Red	2	10^2	2
Orange	3	10^3	
Yellow	4	10^4	
Green	5	10^5	0.5
Blue	6	10^6	0.25
Violet	7	10^7	0.1
Grey	8	10^8	
White	9	10^9	
Gold		10^{-1}	5
Silver		10^{-2}	10
(none)			20

Using the color code shown in the table above, the resistance value and tolerance range are indicated by four bands of color as shown in the following diagram. For example, if the three bands from the left are red, orange, and yellow, as shown in the diagram below, the first two digits are the two-digit number (2 and 3 in this case), and the last digit is the multiplier (in this case, 4, meaning 10^4). Therefore, the resistance value is 230,000Ω, or 230KΩ. If the last band is gold, the tolerance is 5%; if silver, the tolerance is 10%.

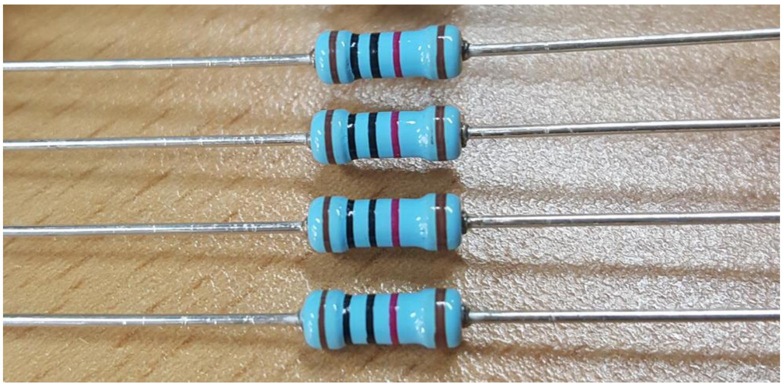


	1st Digit	2nd Digit	Multiplier
Black	0	0	x1
Brown	1	1	x10
Red	2	2	x100
Orange	3	3	x1,000
Yellow	4	4	x10,000
Green	5	5	x100,000
Blue	6	6	x1,000,000
Violet	7	7	x10,000,000
Gray	8	8	x100,000,000
White	9	9	-

(Figure) Resistance value indication method using colors

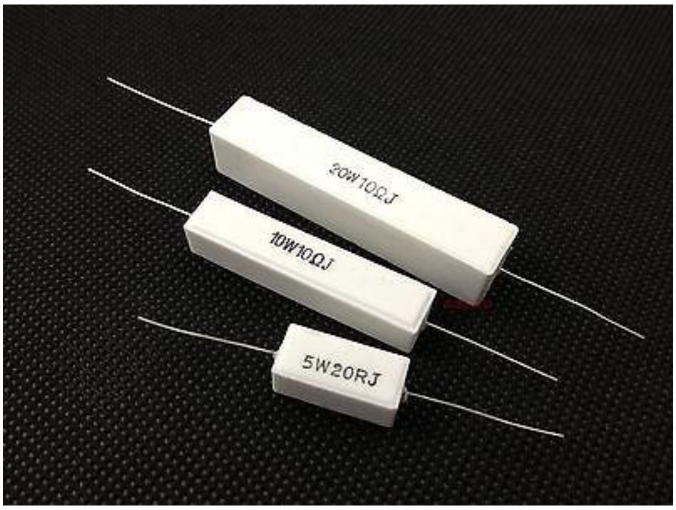
(Figure) Carbon Film Resistor

When low-error resistors are required, precision resistors such as metal oxide film resistors can be used. For precision resistors, an additional band indicating the color code number may be present. For example, in the case of the 1% metal oxide film resistor shown below, there are bands for brown, black, black, and red stripes. The first three stripes represent the three digits (1, 0, 0 in this case), and the last stripe indicates the multiplier (2 in this case, meaning 10^2). Therefore, the resistance value is 10,000Ω, or 10KΩ. The brown band on the far right indicates this is a 1% tolerance resistor.



(Figure) 1% tolerance metal film resistor

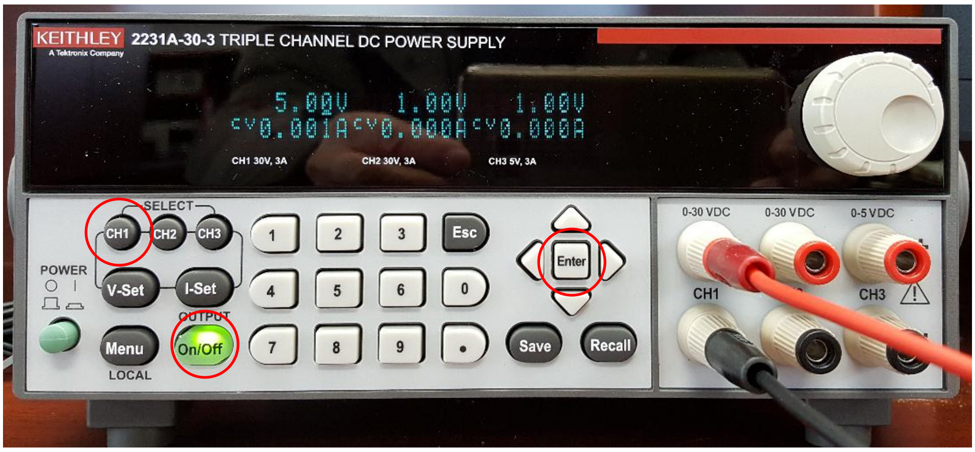
When selecting a resistor, you must first predict the current flow, calculate the power dissipation, and then choose a resistor rated for that power. Commonly used resistors in electronic circuits, such as carbon film resistors and metal oxide film resistors, typically have maximum power ratings of 1/8, 1/4, 1/2, or 1 watt. Cement resistors, like the one in the following figure ( ), are used when power dissipation needs exceed 5 watts.



(Figure) Cement Resistor

- **Voltage Measurement**

Before measuring voltage with a multimeter, adjust the power supply to output the desired voltage. The front panel of the power supply used in this experiment is shown in the following figure. This device has three channels. CH1 and CH2 output variable voltages ranging from 0 to 30 volts, while CH3 outputs a fixed voltage of 5 volts. For the test measurement, set CH1 to output 5 volts and then measure this with the multimeter. First, press the green power button at the bottom left to turn on the power. Next, press the gray CH1 button, then press the number 5 on the numeric keypad, and finally press the white Enter key. Next, press the green button labeled On/Off. The green button will illuminate, and a 5-volt voltage will be output to the CH1 terminal.,



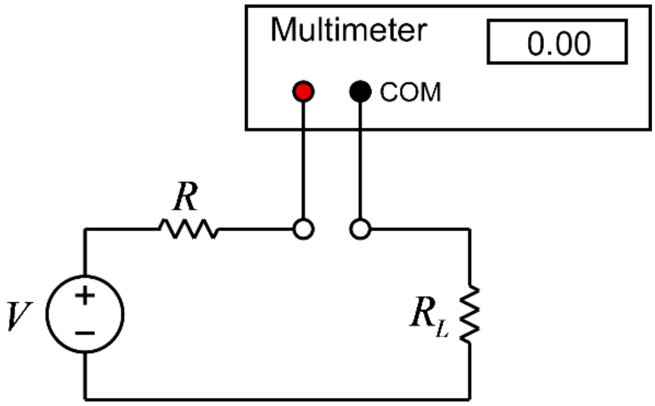
To measure voltage with a multimeter, just as with resistance measurement, insert the black probe into the left black terminal and the red probe into the left red terminal marked V-Ω. Next, press the button labeled DCV among the selection buttons. When measuring voltage, touch the black probe to the lower voltage terminal and the red probe to the higher voltage terminal, then read the displayed value. The displayed voltage is the voltage value at the red terminal relative to the black terminal. Therefore, if you reverse the red and black terminals for measurement, the displayed voltage value will be negative.

- **Current Measurement**

The multimeter terminals for measuring current comprise a low-current measurement terminal (maximum current 500mA; may be lower depending on the multimeter model. Check the maximum current marked on the terminal) and a high-current measurement terminal (maximum current 10A).

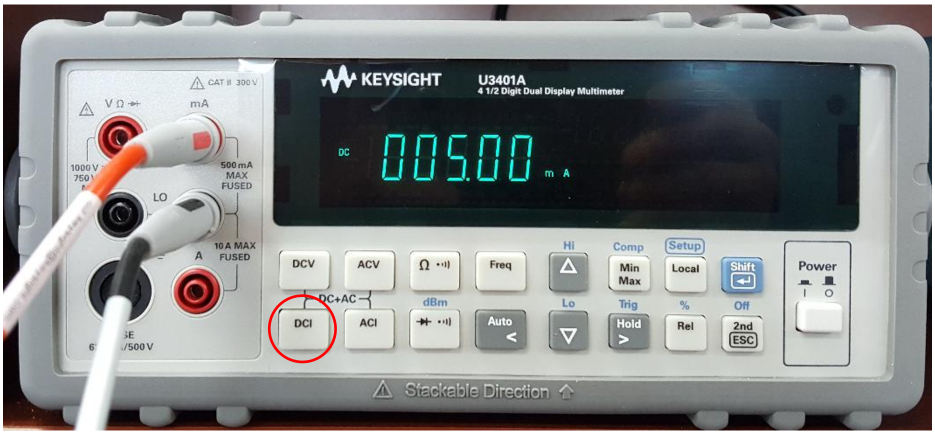
For current measurement, the black probe is inserted into the black terminal, but the red terminal must be inserted into the appropriate terminal depending on the range of the current value being measured. Current values in laboratory experiments are mostly 500mA or less, so the 500mA terminal is frequently used. If the current value to be measured exceeds the maximum rating, the multimeter's fuse may be damaged, so caution is required. Therefore, before connecting the multimeter for current measurement, it is necessary to calculate the approximate current value to ensure it does not exceed the maximum current rating.

Current is measured by connecting a multimeter across the circuit where the current flows. Taking the circuit shown in the following diagram as an example, to measure the current flowing through the resistor, the circuit carrying the current is interrupted and the multimeter is connected across it for measurement. Strictly speaking, as the multimeter itself has a resistance value, connecting it to the original circuit as shown below does affect the current value being measured. However, the resistance value between the multimeter's current measurement terminals is very small and is therefore typically disregarded.



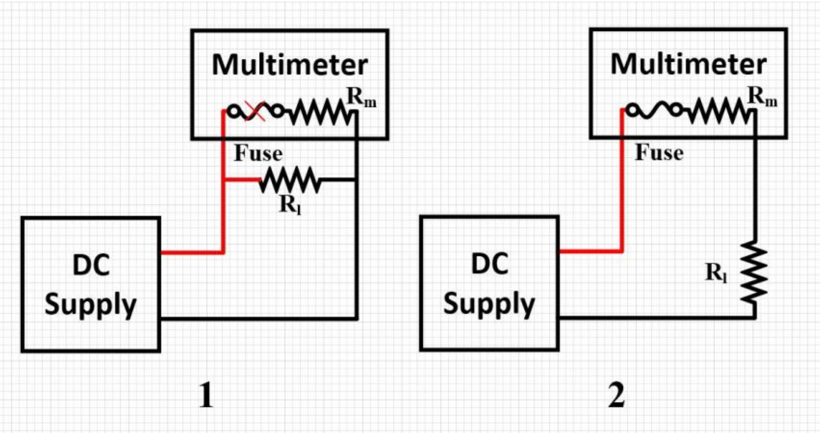
(Figure) Connection of a multimeter for current measurement

Connect the probe to the multimeter's current measurement terminals as shown in the diagram below, then configure the circuit as described above. Press the button labelled DCI among the selection buttons, then read the displayed value. The direction of current is indicated as positive (+) when it flows into the red terminal and out of the black terminal. As current has direction, swapping the positions of the black and red probes will display the same current value but with the opposite sign.



<u>(Important Notice) When measuring current with this multimeter, the current value may be inaccurate if set to AUTO. Therefore, always set it manually for measurement and press the Hi or Lo button to find the range where the correct value is displayed.</u>

Precautions when measuring current using a multimeter:



The diagram above shows a circuit for measuring the current flowing through a resistor. When measuring current using a multimeter, connection 2 in the diagram is the correct method, whereas connection 1 is incorrect. That is, if the current measurement terminals are connected across both ends of the resistor in the same manner as for voltage measurement (connection 1), a very large current will flow through the multimeter, causing the fuse to blow. The resistance between the multimeter's current measurement terminals is extremely low. Consequently, even a small voltage applied across the terminals causes a substantial current to flow. To prevent damage to the instrument, the fuse blows. Therefore, for correct current measurement, the connection must always be made as shown in diagram 2 above.

Instruments and Components Used

- Multimeter: 1
- Power supply: 1
- Resistors: 1KΩ × 1, 2.2KΩ × 1, 3.3KΩ × 1, 4.7KΩ × 1, 10KΩ × 1

Experimental Methods and Procedures

A. Measuring resistance values using a multimeter

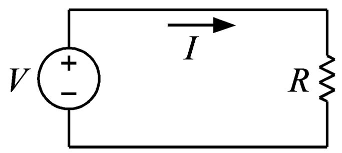
Using a multimeter, measure the given resistance value and record it in the table. Compare the indicated resistance value with the measured resistance value and record the error (%).

Error (%) = 100 × (measured value - displayed value) / displayed value

For example, if the measured value of a resistor marked as 10KΩ is 10.2KΩ, the error is 2%.

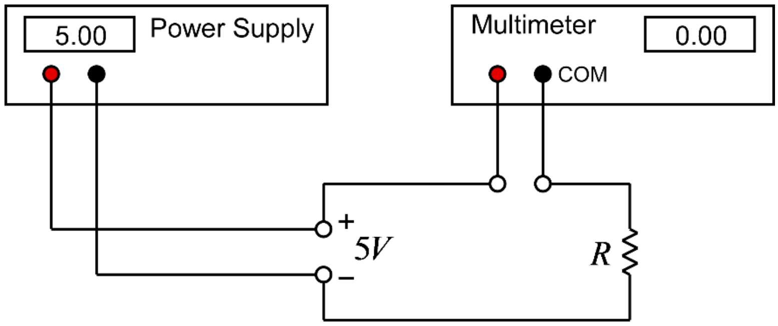
B. Ohm's Law

Using the resistor employed in the preceding section, <Measuring Resistance Values Using a Multimeter>, construct the circuit shown in the diagram below and measure the value of the current flowing through the resistor. Adjust the power supply unit's voltage setting to 5V. When adjusting the power supply unit's voltage, note that the voltage indication on the unit is often inaccurate; therefore, use a multimeter to verify that the voltage value is correct.



(Figure) Ohm's Law Measurement Circuit

The circuit above is actually configured as shown in the following diagram..



(Figure) Ohm's Law Measurement Circuit Configuration

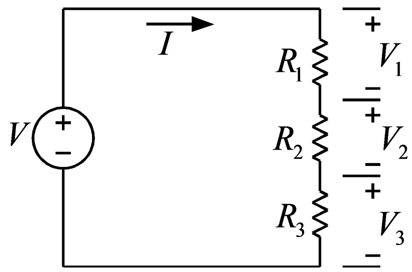
Measure the current flowing through the resistor in the circuit above and record it in the table. Also, calculate the power consumed by the resistor and record it in the table.

$$\text{Power: } P = VI = I^2R = \frac{V^2}{R}$$

Since the resistors typically used in experiments are 1/4 watt resistors, verify that the calculated power is 1/4 watt or less.

C. Kirchhoff's Voltage Law (KVL)

To experimentally verify Kirchhoff's Voltage Law, assemble the circuit shown in the following diagram. Select three resistors from those used in the previous experiment. Set the power supply voltage to 5V, as in the previous experiment.



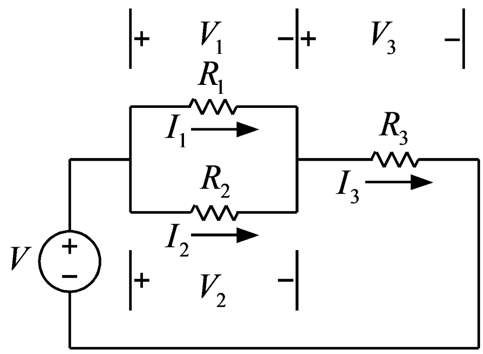
(Figure) KVL Measurement Circuit

Applying Ohm's Law and KVL to the above circuit yields the following relationship. Experimental measurements confirm that the following relationship holds for the above circuit. To verify this, measure the current and voltage, fill in the table, and then confirm that the following equation holds.

$$V_1 = I \times R_1$$
$$V_2 = I \times R_2$$
$$V_3 = I \times R_3$$
$$V = V_1 + V_2 + V_3$$

D. Kirchhoff's Current Law (KCL)

To experimentally verify Kirchhoff's Current Law (KCL), assemble the circuit shown in the following diagram. Select three resistors from those used in the previous experiment. Set the power supply voltage to 5V, as in the previous experiment..



(Figure) KCL Measurement Circuit

Applying Ohm's Law and KCL/KVL to the circuit above yields the following relationships. Experimental measurements confirm that the following relationships hold for the circuit above. To verify this, measure the current and voltage, fill in the table, and then confirm that the following equation holds.

$$V_1 = I_1 \times R_1$$
$$V_2 = I_2 \times R_2$$
$$V_3 = I_3 \times R_3$$
$$I_3 = I_1 + I_2$$
$$V_1 = V_2$$
$$V = V_1 + V_3 = V_2 + V_3$$

● Review of Experimental Results and Discussion

<Table> Resistance Value Measurement

Resistance	Marked resistance value	Measured resistance value	Error (%)
R_1			
R_2			
R_3			
R_4			
R_5			

<Table> Ohm's Law

Resistance	Measured resistance value	Current value calculated from the measured resistance value I=V/R	Measured current value	Power consumption (watt)
R_1				
R_2				
R_3				
R_4				
R_5				

<Table> Kirchhoff's Voltage Law (KVL)

	Marked resistance value	Measured resistance value	Measured current value	Resistance value × Current value	Measured voltage value
R_1					
R_2					
R_3					
V					

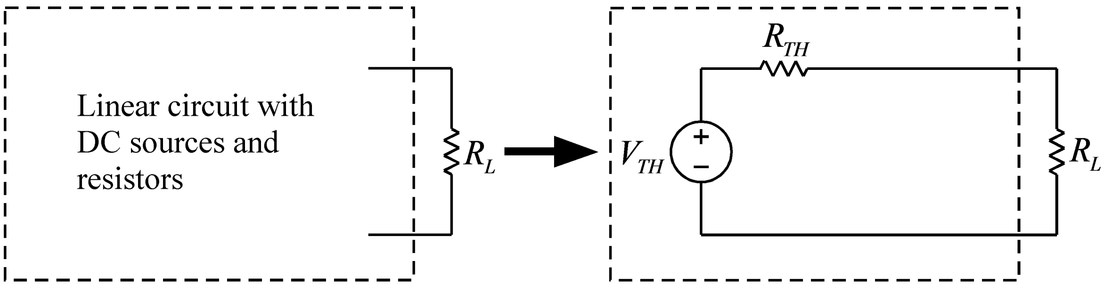
<Table> Kirchhoff's Current Law (KCL)

	Marked resistance value	Measured resistance value	Measured current value	Resistance value × Current value	Measured voltage value
R_1					
R_2					
R_3					
V					

Lab 2. Thevenin's Theorem

● Thevenin Equivalent Circuit

Thevenin's theorem states that any circuit consisting of multiple DC power sources and multiple resistors can be replaced by a single power source and a single resistor connected in series. That is, no matter how complex a linear circuit may be, it can be replaced by a simple equivalent circuit as shown in the following diagram. This is a highly useful theorem that can be employed with great convenience in the analysis of complex circuits. Whilst Norton's theorem forms a complementary pair with Thevenin's theorem, their content is relatively similar; therefore, this experiment will only cover experiments related to Thevenin's theorem.



(Figure) Thevenin equivalent circuit

To determine the Thevenin equivalent circuit, the equivalent voltage V_{TH} and equivalent resistance R_{TH} must be obtained. Equivalent voltage V_{TH} is the open-circuit voltage measured at the terminals with the load resistor removed. The equivalent resistance R_{TH} is the resistance value measured at the two terminals where the load resistance was connected, after removing the load resistance and setting the values of all internal independent power sources to zero. In this experiment, we shall calculate and measure the Thevenin equivalent circuit for the given circuit and compare the results.

- **Instruments and Components Used**

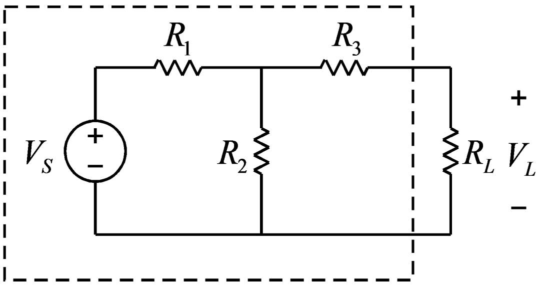
- Multimeter: 1
- Power supply: 1
- Reisitors: 10KΩ 1 piece, 2.2KΩ 2 pieces, 3.3KΩ 2 pieces, 4.7KΩ 2 pieces

- **Experimental Methods and Procedures**

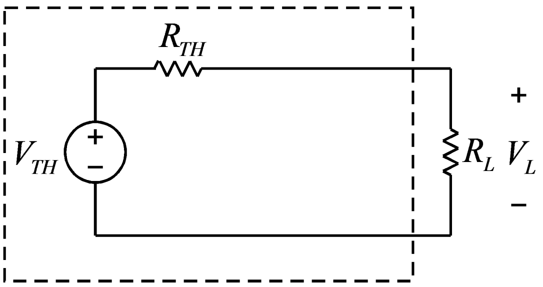
A. Thevenin Equivalent Circuit1

First, use a multimeter to measure the resistance values below and record them in the table. In all subsequent calculations involving resistance values, use the measured values rather than the marked values. Use these resistors to construct the circuit shown in the following diagram. Set the power supply voltage to 5V.

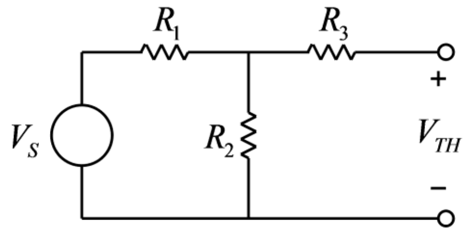
$$R_1 = 2.2K\Omega, R_2 = 4.7K\Omega, R_3 = 3.3K\Omega$$
$$R_L = 10K\Omega$$



Measure the voltage V_L across the load resistor R_L in the above circuit and record it in the table. For the above circuit, we will calculate and measure the Thevenin equivalent circuit as shown in the following figure and compare the results.



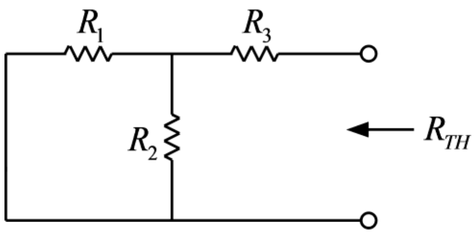
To find the Thevenin equivalent voltage V_{TH}, the circuit below is constructed with the load resistance R_L removed.



Calculate the Thevenin equivalent voltage V_{TH} using the following equation and record it in the table. Measure the actual circuit's voltage V_{TH} using a multimeter and record it in the table.

$$V_{TH} = \frac{R_2}{R_1 + R_2} V_S$$

To find the Thevenin equivalent resistance R_{TH} , disconnect the power supply and configure the circuit below.



Calculate the Thevenin equivalent resistance R_{TH} using the following equation and record it in the table. Measure the actual circuit resistance R_{TH} using a multimeter and record it in the table.

$$R_{TH} = \frac{R_1 R_2}{R_1 + R_2} + R_3$$

Using V_{TH} and R_{TH} obtained in the above process, the voltage across the load resistance in the Thevenin equivalent circuit can be calculated as shown in the equation below. This voltage value can then be compared to the voltage V_L measured across the load resistance in the original circuit.

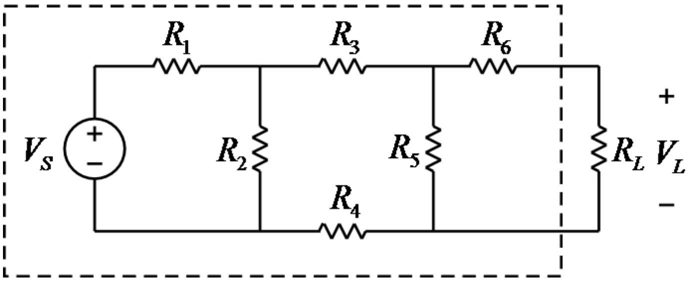
$$V_L = \frac{R_L}{R_{TH} + R_L} V_{TH}$$

In the above equation, since both calculated and measured values exist for V_{TH} and R_{TH} , calculate both cases and enter them in the table.

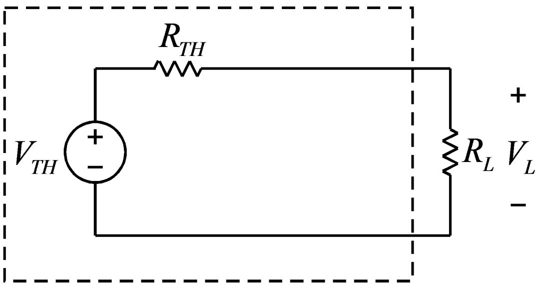
B. Thevenin Equivalent Circuit 2

First, use a multimeter to measure the resistance values below and record them in the table. For all subsequent calculations involving resistance values, use the measured values, not the marked values. Use these resistors to construct the circuit shown in the following diagram. Set the power supply voltage to 5V.

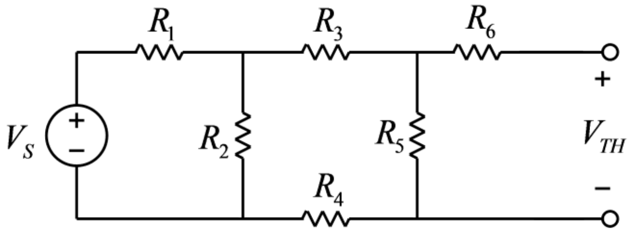
$R_1 = 2.2K\Omega, R_2 = 4.7K\Omega, R_3 = 3.3K\Omega$
$R_4 = 2.2K\Omega, R_5 = 4.7K\Omega, R_6 = 3.3K\Omega$
$R_L = 10K\Omega$



Measure the voltage V_L across the load resistor R_L in the above circuit and record it in the table. For the above circuit, we will calculate and measure the Thevenin equivalent circuit as shown in the following figure and compare the results.

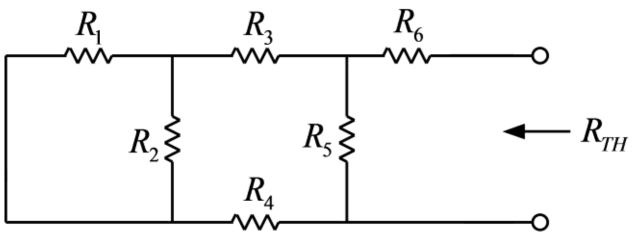


To find the Thevenin equivalent voltage V_{TH} , the circuit below is constructed with the load resistance R_L removed.



Calculate the Thevenin equivalent voltage V_{TH} and record it in the table. Measure the actual circuit's voltage V_{TH} using a multimeter and record it in the table.

To find the Thevenin equivalent resistance R_{TH} , disconnect the power supply and configure the circuit below.



Calculate the Thevenin equivalent resistance R_{TH} and record it in the table. Measure the actual circuit resistance R_{TH} using a multimeter and record it in the table.

Using V_{TH} and R_{TH} obtained in the above process, the voltage across the load resistance in the Thevenin equivalent circuit can be calculated as shown in the equation below. This voltage value can then be compared to the voltage V_L measured across the load resistance in the original circuit.

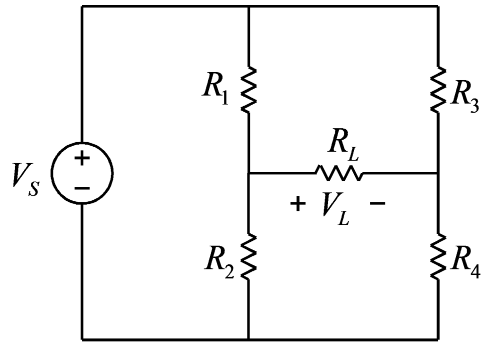
$$V_L = \frac{R_L}{R_{TH} + R_L} V_{TH}$$

In the above equation, since both calculated and measured values exist for V_{TH} and R_{TH} , calculate both cases and enter them in the table.

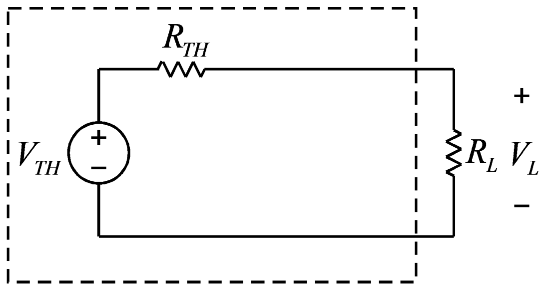
C. Thevenin Equivalent Circuit 3

First, use a multimeter to measure the resistance values below and record them in the table. For all subsequent calculations involving resistance values, use the measured values, not the marked values. Use these resistors to construct the circuit shown in the following diagram. Set the power supply voltage to 5V.

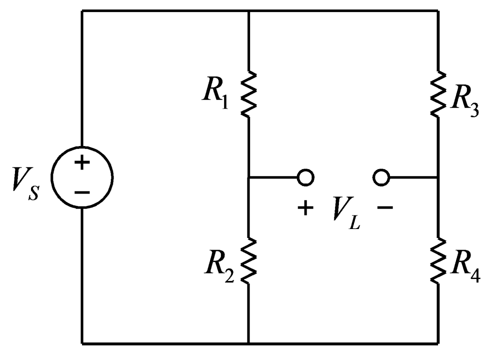
$$R_1 = 2.2K\Omega, R_2 = 4.7K\Omega, R_3 = 3.3K\Omega, R_4 = 1K\Omega$$
$$R_L = 10K\Omega$$



Measure the voltage V_L across the load resistor R_L in the above circuit and record it in the table. For the above circuit, we will calculate and measure the Thevenin equivalent circuit as shown in the following figure and compare the results.

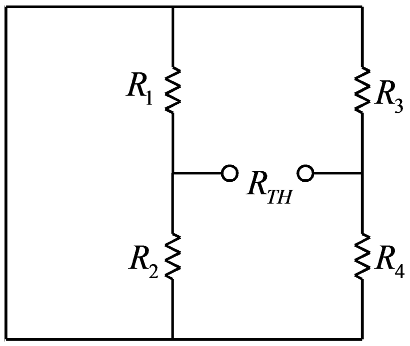


To find the Thevenin equivalent voltage V_{TH}, the circuit below is constructed with the load resistance R_L removed.



Calculate the Thevenin equivalent voltage V_{TH} and record it in the table. Measure the actual circuit's voltage V_{TH} using a multimeter and record it in the table.

To find the Thevenin equivalent resistance R_{TH}, disconnect the power supply and configure the circuit below.



Using V_{TH} and R_{TH} obtained in the above process, the voltage across the load resistance in the Thevenin equivalent circuit can be calculated as shown in the equation below. This voltage value can then be compared to the voltage V_L measured across the load resistance in the original circuit.

$$V_L = \frac{R_L}{R_{TH} + R_L} V_{TH}$$

In the above equation, since both calculated and measured values exist for V_{TH} and R_{TH}, calculate both cases and enter them in the table.

Review of Experimental Results and Discussion

<Table> Thevenin Equivalent Circuit 1

Resistor	Marked resistance value	Measured resistance value
R_1	$2.2K\Omega$	
R_2	$4.7K\Omega$	
R_3	$3.3K\Omega$	
R_L	$10K\Omega$	

<Table> Thevenin Equivalent Circuit 1

	Calculated value	Measurement value
V_{TH}		
R_{TH}		

<Table> Thevenin Equivalent Circuit 1

	Measurement value	Calculated value 1 (Using calculated values of V_{TH} , R_{TH})	Calculated value 2 (Using measured values of V_{TH} , R_{TH})
V_L			

<Table> Thevenin Equivalent Circuit 2

Reistor	Marked resistance value	Measured resistance value
R_1	$2.2K\Omega$	
R_2	$4.7K\Omega$	
R_3	$3.3K\Omega$	
R_4	$2.2K\Omega$	
R_5	$4.7K\Omega$	
R_6	$3.3K\Omega$	
R_L	$10K\Omega$	

<Table> Thevenin Equivalent Circuit 2

	Calculated value	Measurement value
V_{TH}		
R_{TH}		

<Table> Thevenin Equivalent Circuit 2

	Measurement value	Calculated value 1 (Using calculated values of V_{TH} , R_{TH})	Calculated value 2 (Using measured values of V_{TH} , R_{TH})
V_L			

<Table> Thevenin Equivalent Circuit 3

Resistor	Marked resistance value	Measured resistance value
R_1	$2.2K\Omega$	
R_2	$4.7K\Omega$	
R_3	$3.3K\Omega$	
R_4	$1K\Omega$	
R_L	$10K\Omega$	

<Table> Thevenin Equivalent Circuit 3

	Calculated value	Measurement value
V_{TH}		
R_{TH}		

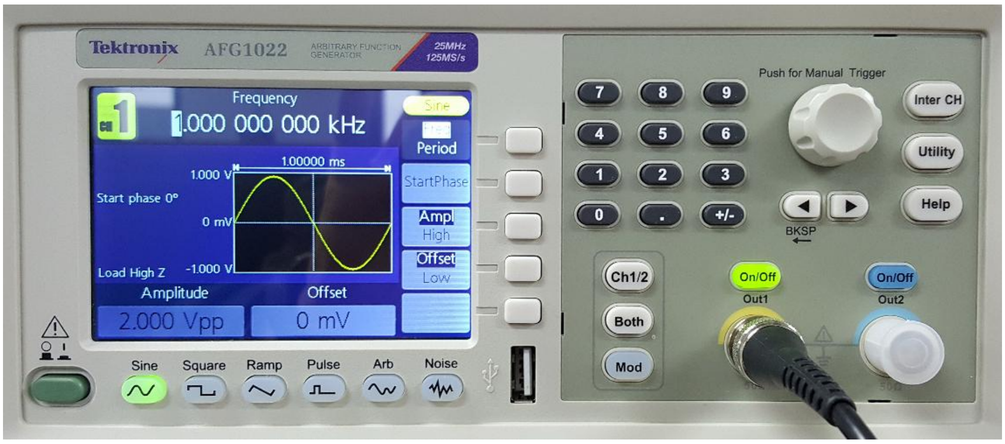
<Table> Thevenin Equivalent Circuit 3

	Measurement value	Calculated value 1 (Using calculated values of V_{TH} , R_{TH})	Calculated value 2 (Using measured values of V_{TH} , R_{TH})
V_L			

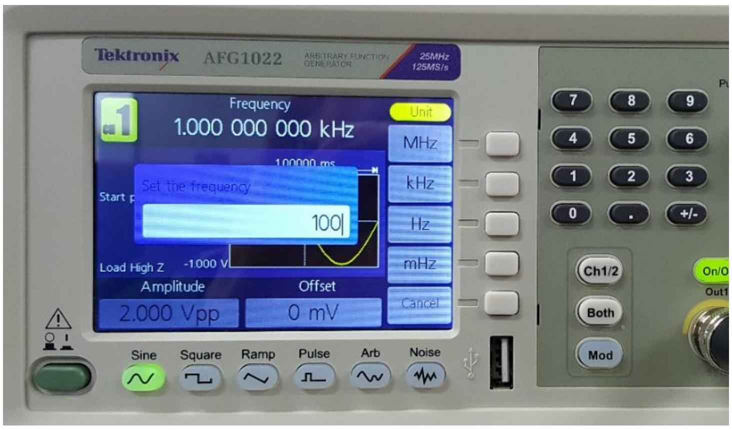
How to Use an Oscilloscope and Function Generator

- ### How to Use the Function Generator법

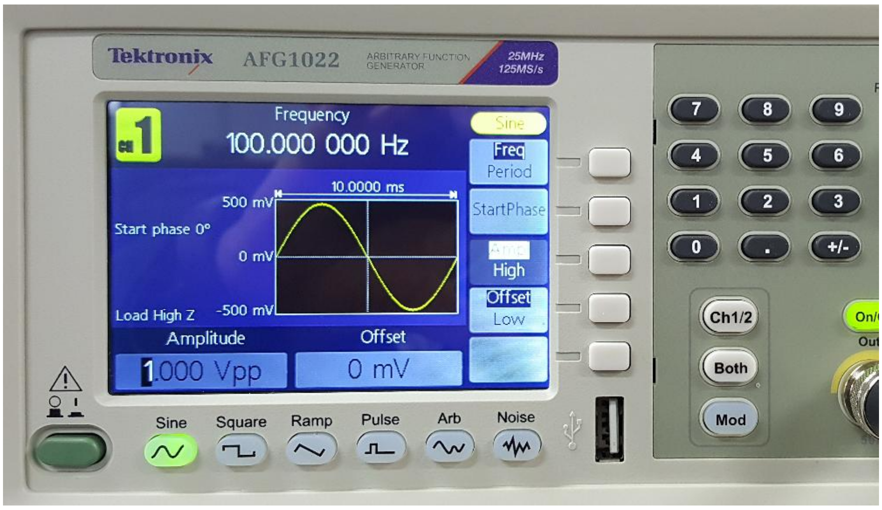
A function generator is a device that generates various waveforms and outputs them as voltage signals. The waveforms it can generate include sine, square, ramp, and pulse. The front panel is as shown in the following figure.



As a simple example, let's generate a 100Hz sine wave. The peak-to-peak voltage of the generated sine wave is 2V, with an offset of 0V. That is, the lowest voltage of the sine wave is -1V and the highest voltage is 1V. First, confirm that the sine button is selected and its indicator lamp is lit, as shown in the figure above. Next, to select the frequency, press the button labeled Freq/Period on the right side of the five white buttons next to the screen (the topmost button). Each press of this button alternates the input mode between frequency (Freq) and period (Period). The screen above shows the frequency selection screen. While in frequency selection mode, pressing a number on the numeric keypad switches to the number input screen as shown below. On this screen, press the number 100, then press the button to the right of the Hz display to complete the frequency selection.

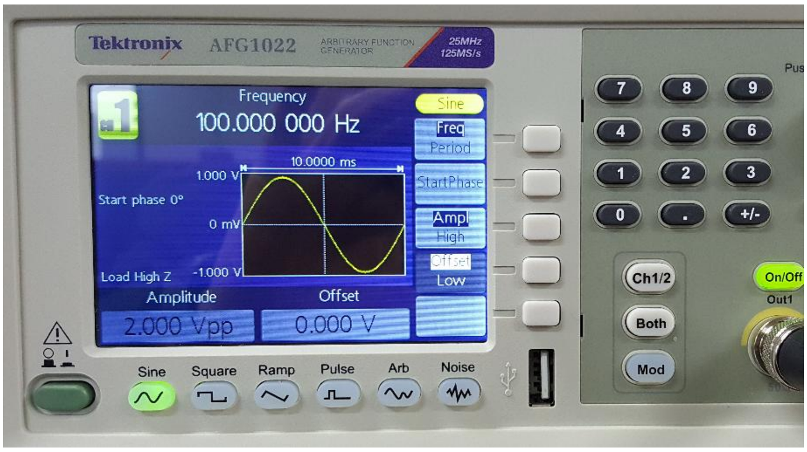


Once frequency selection is complete, the screen shown in the following figure appears. In this state, pressing the button to the right of the Ampl display (the third button from the top) allows you to select the signal magnitude. This is the menu for selecting the signal magnitude, which can be chosen in two ways. The first method is selecting by Amplitude and Offset, where Amplitude is selected as the peak-to-peak voltage magnitude. Offset refers to the DC voltage, meaning the average value. The second method involves selecting the signal's maximum value (High) and minimum value (Low). Pressing the button to the right of Amplitude cycles through these two methods alternately with each press. The figure below shows the screen selected using Amplitude and Offset.

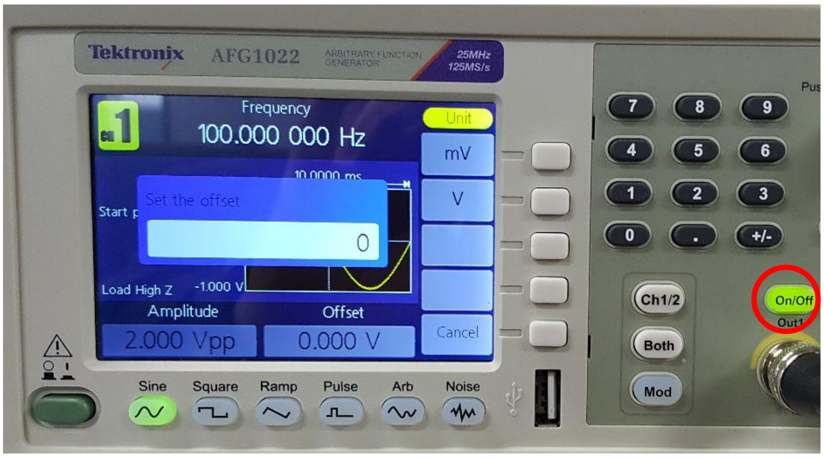


With Amplitude selected, pressing the numeric keypad displays the number input as shown below. After pressing the number 2, pressing the button to the right of Vpp sets the peak-to-peak voltage to 2V.

Once the Amplitude selection is complete, the screen shown below appears. In this state, pressing the Offset button enables Offset selection.



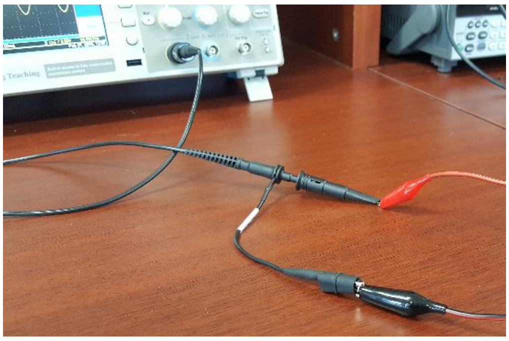
Press the number 0 on the Offset selection screen, then press the button to the right of the V mark to complete the Offset voltage setting.

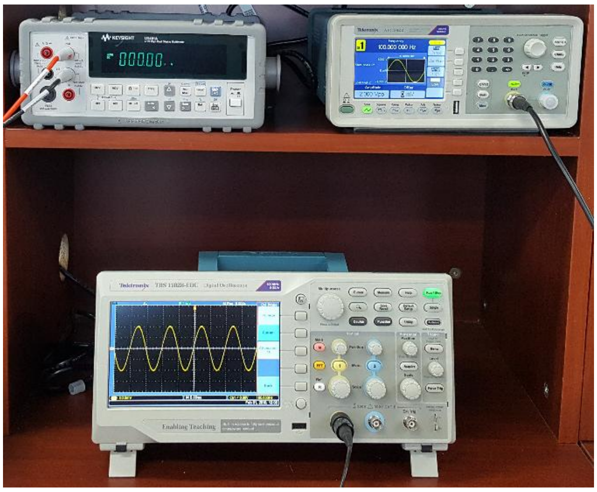


Verify that the On/Off button above the Out1 terminal of the function generator is illuminated. If it is not illuminated, press the button to turn on the button lamp. If this button is not illuminated, no signal will be output.

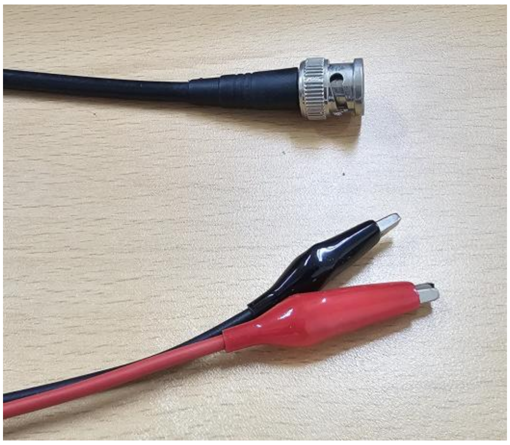
How to Use an Oscilloscope

Use an oscilloscope to verify that the signal output from the function generator matches the set configuration. First, connect the function generator cable to the Output terminal of the function generator. Next, connect the oscilloscope cable to the CH1 terminal of the oscilloscope. Then, connect the cable's ground wire and signal wire separately. Confirm that the Output button on the function generator is illuminated.





Caution: Since the connectors (BNC connectors) for function generator cables and oscilloscope cables are identical in shape, take care not to mistakenly swap the two cables. The left side in the figure below shows the function generator cable, and the right side shows the oscilloscope cable.



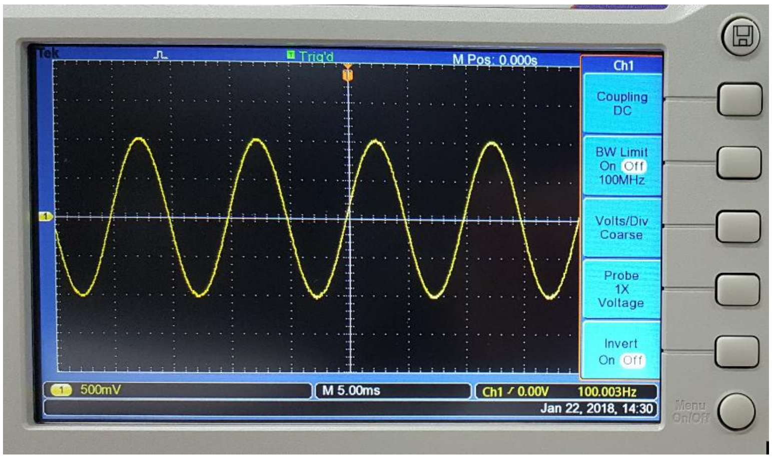
Function generator cable

Oscilloscope Cable

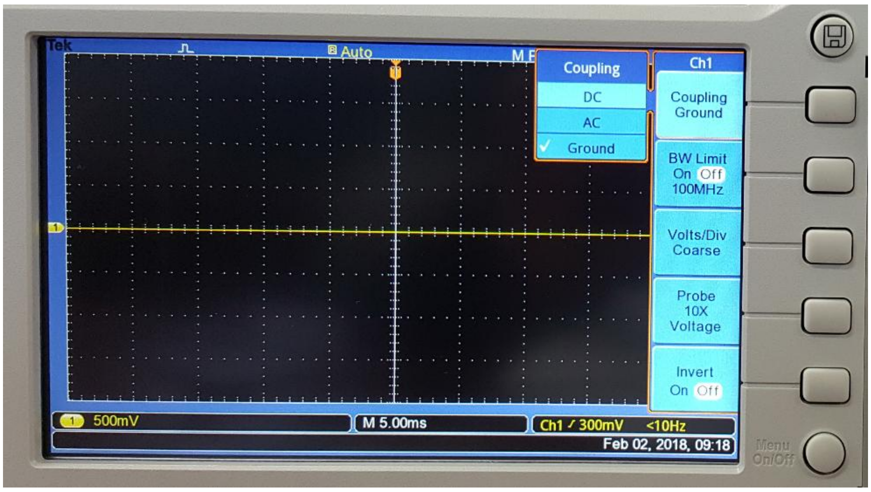
The front panel of the oscilloscope is as shown in the following figure. The most frequently used knobs on the oscilloscope panel are the Vertical Scale knob (VOLTS/DIV knob), which adjusts the voltage per division on the vertical axis, and the Horizontal Scale knob (SEC/DIV knob), which adjusts the time per division on the horizontal axis. Additionally, there is a Position knob for adjusting the vertical and horizontal display positions. While there is an AutoSet button that automatically sets these parameters, it is important to develop the habit of adjusting them manually rather than relying on this button. This practice is crucial for mastering the skillful use of an oscilloscope.

■ **CHANNEL MENU**

First, since a signal is connected to the CH1 terminal, press the yellow CH1 Menu button. Each time this button is pressed repeatedly, the signal indicator line on the screen appears or disappears.

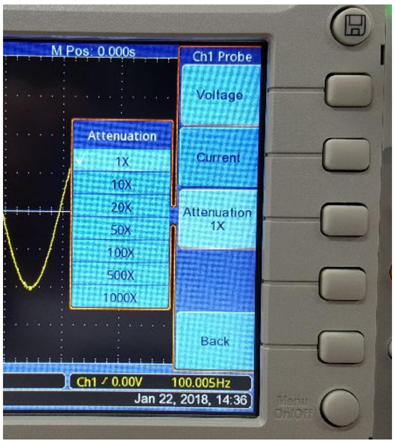


Pressing the CH1 MENU button displays the selection menu on the right side of the screen as shown in the figure above. First, pressing the button to the right of Coupling at the top of the menu displays the Coupling Sub-Menu as shown in the next figure. In this menu, turning the Multipurpose dial cycles through DC, AC, and Ground selections, and pressing the Multipurpose dial confirms the selection. The screen below shows the display when Ground is selected. Ground selection is typically used for adjusting the vertical position. When measuring signals, select either DC or AC. The difference between AC and DC is that AC displays the signal with its DC component (constant voltage) removed. Unless there is a specific reason, this experiment will mostly use DC Coupling for measurements.

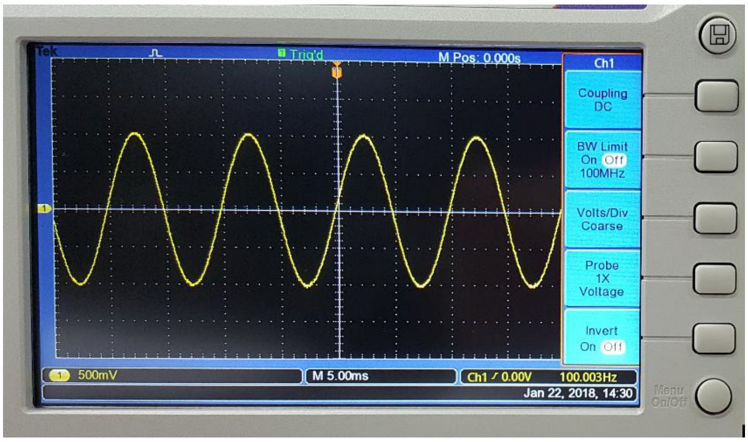


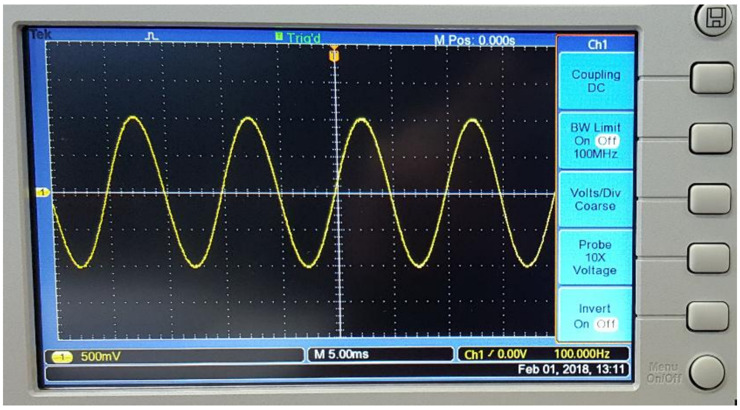
With Ground selected, the Ground position (0V position) can be moved up or down using the vertical POSITION knobs. The figure above shows the Ground position placed on the center line, but this is not mandatory. To use the screen efficiently, the Ground position may sometimes be placed elsewhere. However, in such cases, the exact Ground position must be remembered throughout the experiment. After setting the Ground position, ensure Coupling is reset to DC. Also set the Probe scale in this menu. Observe the oscilloscope probe: it may allow selection of 1x or 10x scale, or the scale may be fixed as shown in the figure below.

Set the probe magnification based on the magnification value of the probe currently in use. Pressing the right button on the probe display opens the Probe Magnification Selection menu shown below. In this menu, turning the Multipurpose dial changes the magnification, and pressing the Multipurpose dial confirms the magnification selection..



The images below show the screen when using a 1X probe and when using a 10X probe..

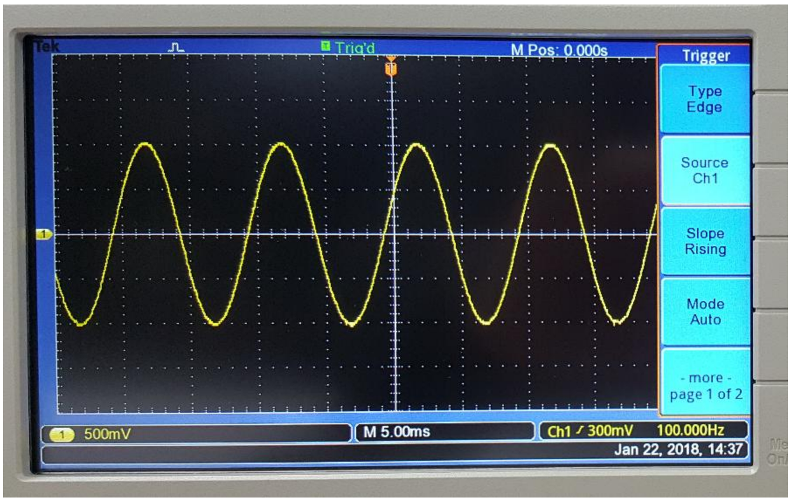




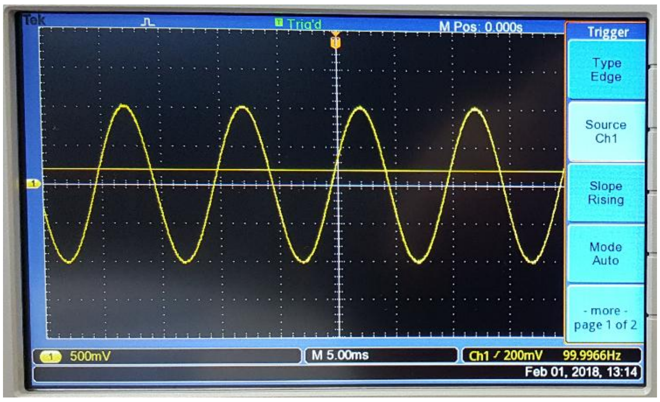
In the figure above, you can see CH1 500mV at the bottom. This indicates that each division represents 500mV, and this value can be changed by turning the Vertical Scale (VOLTS/DIV) knob. In the figure above, you can confirm that the peak-to-peak voltage is 2V, as set on the function generator. Additionally, you can see 5.00ms at the bottom center of the screen. This indicates 5 milliseconds per division on the horizontal axis, and this value can be adjusted using the Horizontal Scale (SEC/DIV) knob. Looking at the sine wave above, one full cycle is 10 milliseconds, corresponding to a frequency of 100Hz.

- **TRIGGER MENU**

If the screen displays a continuous signal flowing left or right instead of a stable static image, this indicates the trigger level is not set correctly. The trigger determines the starting point at which the oscilloscope acquires data and repeatedly displays the waveform. To configure the trigger settings, press the Trigger Menu button on the right side of the panel to access the trigger menu, as shown in the figure below.

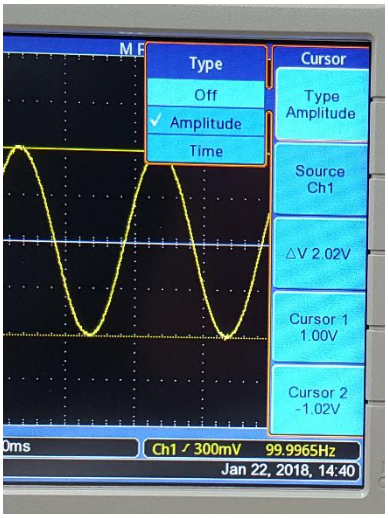


Verify that the trigger menu is configured as shown above, especially ensuring that the Source is set to Ch1. Additionally, turning the trigger Level knob will show the trigger level moving up and down as depicted in the figure below. Adjusting this knob to position the trigger level between the signal's maximum and minimum values will allow you to observe a stable freeze frame..



■ **CURSOR MENU**

A useful feature on a digital oscilloscope is the ability to read values using cursors. Reading values using only the scale can be difficult, but cursors make value reading easy. The cursor menu can be accessed by pressing the Cursor button.



The image above shows the cursor menu with Type set to Amplitude. In this case, two horizontal lines are visible, and these two horizontal lines can be moved up and down by turning the Multipurpose dial. Pressing the button to the right of the Cursor 1, Cursor 2 display allows you to select the cursor you wish to move. The image below shows the screen with Cursor 1 selected.

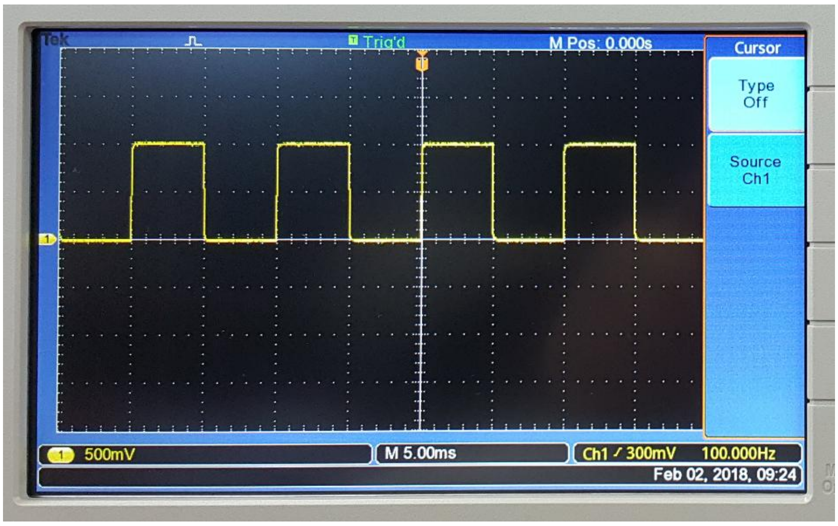
And depending on the position of each horizontal cursor, it displays the voltage value corresponding to that position, while ΔV shows the difference between the two values. Here, we can see that the ΔV value is 2V, confirming it matches the value set on the function generator.

The above image shows the cursor menu with Type set to Time, displaying two vertical lines. The position of these two vertical lines can be adjusted by rotating the Multipurpose dial. It shows the time value corresponding to each vertical line's position, and △ represents the difference between the two values.

- **Generation and Verification of Square Waves**

Generate a square wave using a function generator and verify it with an oscilloscope. The frequency is 100Hz, with the low voltage set to 0V and the high voltage to 1V. That is, the peak-to-peak voltage is 1V, and the offset is 0.5V. The method is nearly identical to before, with the difference being that the Square waveform is selected, and the peak-to-peak voltage and offset values are different. The image below shows the screen after selection is complete.

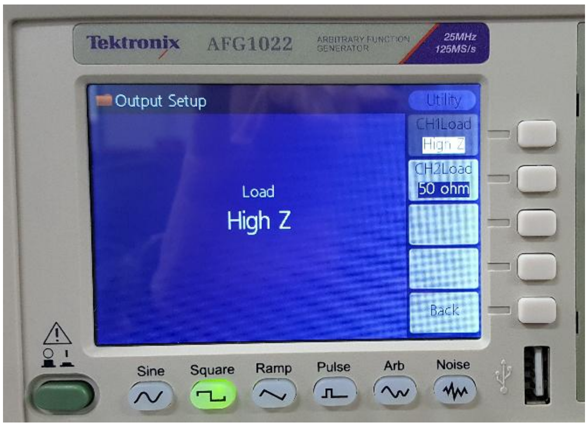
Using an oscilloscope, you can verify that a square wave is generated as shown in the figure below. At this point, confirm that the trigger level indicator arrow is positioned between 0V and 1V. If it is not, adjust the trigger LEVEL knob to position the trigger level between 0V and 1V, ensuring a stable waveform is displayed.



- **Impedance matching of the function generator (fixing the issue where the voltage output doubles)**

Sometimes, all voltage values from the function generator output double the set voltage value. This is an issue related to the function generator's impedance matching. The function generator's output impedance is 50 ohms, and when shipped from the factory, the load impedance is also matched to 50 ohms. Therefore, to apply a 1V voltage to the load, the actual output voltage is doubled to 2V. However, devices like oscilloscopes or multimeters have significantly higher input impedance values. Consequently, measuring the output with an oscilloscope in this state will show a reading twice the set value. To correct this, configure as follows: First, press the Utility button to select the next screen.

Press the Output Setup button on the Utility screen to select it, and the following screen will appear. Each time you press the button next to CH1Load, the value changes; set this value to High Z. Once the setting is complete, press the Utility button to return to the original screen.



Lab 3. Characteristics of Capacitors and Inductors – Transient Characteristics

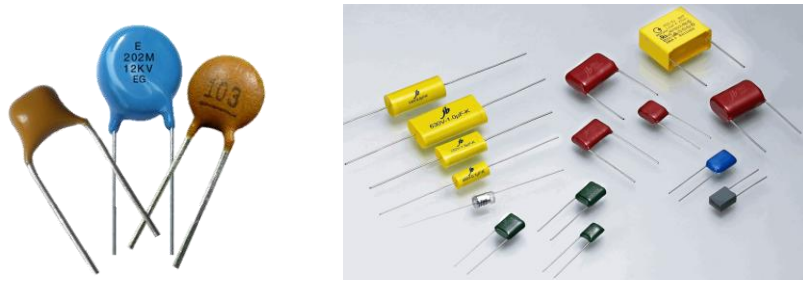
● Types of Capacitors

A capacitor is a component consisting of two opposing electrode plates separated by a dielectric material, serving to store electrical charge. (In Korea, it is often called a condenser instead of a capacitor.) Capacitors come in various types depending on their structure and materials, and their characteristics may differ by type, making it important to select and use the appropriate type. The primary classification of capacitors is into polarized and non-polarized types. Among polarized types, electrolytic capacitors (electrolytic condensers) are common, and their typical shape is as shown in the following figure.

(Figure) Electrolytic Capacitor

Electrolytic capacitors are polarized, so as shown in the figure above, the negative terminal is marked with a minus (-) sign on its surface. Electrolytic capacitors can achieve large capacitance in a relatively small size, but caution is required as there is a risk of explosion if voltage is applied in the reverse direction. Electrolytic capacitors are primarily used in circuits where the applied voltage direction does not change, such as DC power supplies. **In experiments using capacitors from this textbook, both positive and negative voltages may be applied across the capacitor terminals. Therefore, electrolytic capacitors (electrolytic condensers) with the shape shown above are not used.**

Non-polar capacitors include ceramic capacitors, film capacitors, and others, and the figure below shows their typical shapes.



(Figure) Ceramic capacitor, film capacitor

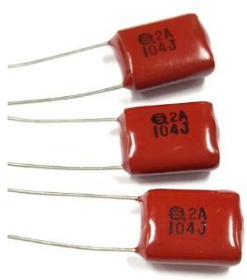
The unit of capacitance for a capacitor is F (Farad), but since this value is extremely large in practice, the units commonly used for capacitance are as follows:

$$1\ \mu F = 10^{-6}\,F$$
$$1\ nF = 10^{-9}\,F$$
$$1\ pF = 10^{-12}\,F$$

As shown in the figure above, the capacitance of a non-polar capacitor is denoted by a three-digit number in pF (picofarad), the smallest unit of capacitance. The first two digits represent the significant figures, while the last digit indicates the power of 10. The following are several examples of capacitor capacitance notation.

$$103 \rightarrow 10\times 10^{3}\ pF = 0.01\ \mu F$$
$$104 \rightarrow 10\times 10^{4}\ pF = 0.1\ \mu F$$
$$105 \rightarrow 10\times 10^{5}\ pF = 1\ \mu F$$
$$225 \rightarrow 22\times 10^{5}\ pF = 2.2\mu F$$
$$684 \rightarrow 68\times 10^{4}\ pF = 0.68\mu F$$

The capacitor most frequently used in this experimental manual is the $0.1\ \mu F$ polyester film capacitor (also called a Mylar capacitor) shown in the following figure. Polarized electrolytic capacitors (electrolytic capacitors) are not used in any of the experiments described in this manual.



(figure) $0.1\ \mu F$ Polyester Film Capacitor

Types of Inductors

An inductor is a component made by winding copper wire multiple times around a core, such as ferrite. Depending on the circuit in which it is used, there are various types, including power supply circuits, general circuits, and high-frequency circuits. The following figure shows various forms of inductors.

(Figure) Various Types of Inductors

The unit of inductance is H (Henry). Since 1 H is an extremely large value, smaller units such as $\mu H = 10^{-6}\ H$ or $mH = 10^{-3}\ H$ are typically used to represent inductance values. The notation method uses three-digit numbers for the μH unit value. The first two digits are the significant figures, and the last digit indicates the power of 10. For example, an inductor labeled 104 has the following value:

$$10?10^{4}\ \mu H = 100\ mH$$

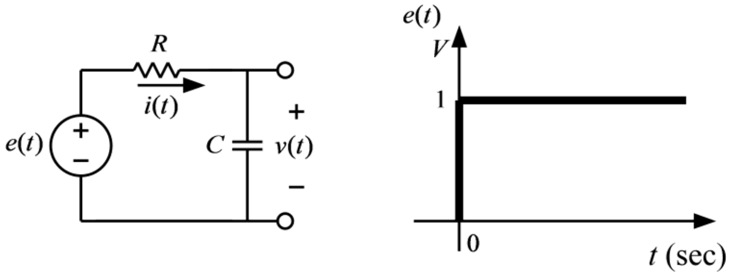
In this experiment, a 100mH inductor will be primarily used, typically labeled as 104 in the form shown in the following figure.

(Figure) The form of the inductor used in this experiment

● Transient Phenomena in RC Circuits

To analyze the transient behavior of a circuit containing a resistor and a capacitor, consider the circuit shown in the figure. The input to this circuit is a voltage signal $e(t)$. When the input signal is a unit step function as shown in the figure below, we determine the voltage $v(t)$ across the capacitor. In the figure below, the current $i(t)$ flowing through the resistor is equal to the current entering the capacitor, and the capacitor current is given by:

$$i(t) = \frac{e(t) - v(t)}{R} = C\frac{dv(t)}{dt}$$



(Figure) RC Circuit

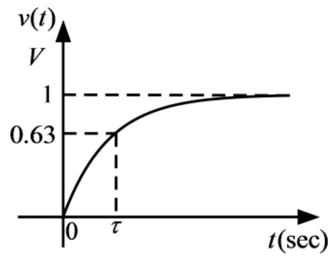
Rearranging the above equation yields the following first-order differential equation:

$$RC\frac{dv(t)}{dt} + v(t) = e(t)$$

Assuming the initial voltage of the capacitor is zero, solving the above differential equation yields the following solution:

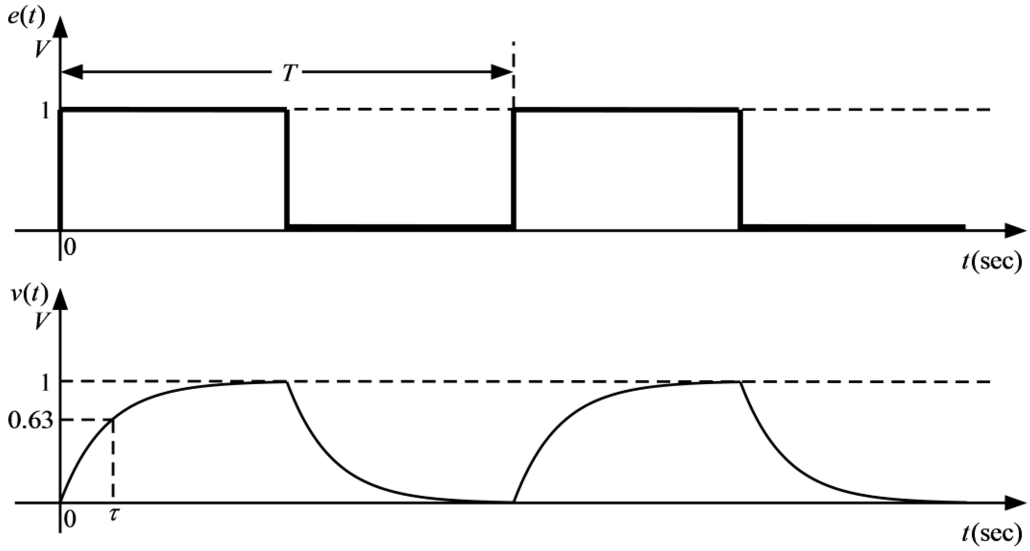
$$v(t) = 1 - e^{-\frac{t}{RC}} = 1 - e^{-\frac{t}{\tau}}, t \geq 0$$

In the above equation, $\tau = RC$ is the time constant of the circuit, and when $t = \tau$, the voltage across the capacitor is $v(t) = 1 - e^{-1} = 0.63$. The waveform of $v(t)$ is shown in the figure below.



(Figure) Unit step response

To observe the above transient phenomenon using an oscilloscope, inputting a square wave as the input signal allows observation of the output waveform shown in the figure below. From this output waveform, the time constant (τ) can be measured.

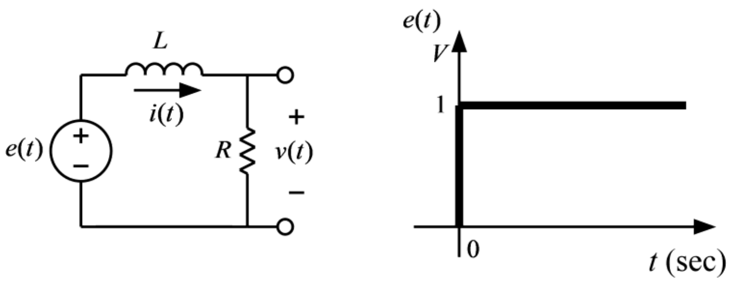


(Figure) Input and Output Waveforms

- **Transient phenomena in RL circuits**

To analyze the transient behavior of a circuit containing a resistor and an inductor, consider the circuit shown in the figure. The input to this circuit is a voltage signal $e(t)$. When the input signal is a unit step function as shown in the figure below, we determine the current $i(t)$ flowing through the inductor. In the figure below, the voltage induced across the inductor is $L(di(t)/dt)$, so applying KVL yields the following first-order differential equation:

$$e(t) = L\frac{di(t)}{dt} + v(t) = L\frac{di(t)}{dt} + Ri(t)$$



(Figure) RL Circuit

Although the above differential equation can be solved to determine the current waveform, since the observable signal on an oscilloscope is the voltage signal, we reformulate the differential equation with the voltage across the resistor as the variable. The following relationship holds in the figure above:.

$$i(t) = \frac{v(t)}{R}$$

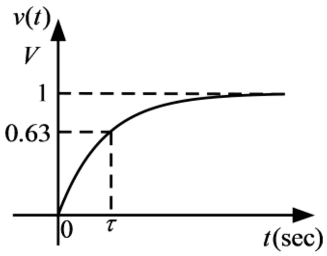
Therefore, substituting this into the above differential equation yields the following differential equation:

$$e(t) = \frac{L}{R}\frac{dv(t)}{dt} + v(t)$$

Assuming the initial current through the inductor is zero, the initial voltage across the resistor is also zero. Therefore, solving the above differential equation yields the following solution:

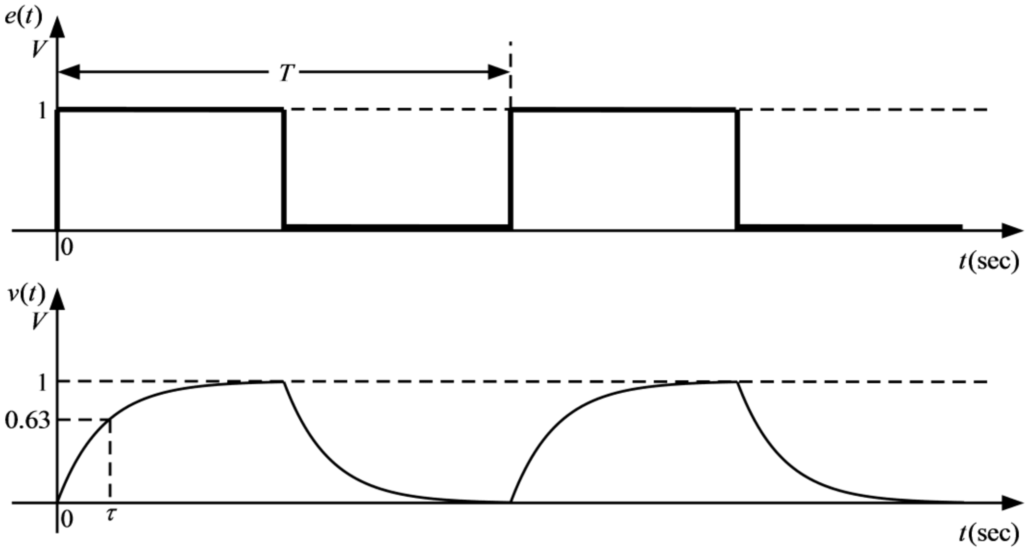
$$v(t) = 1 - e^{-\frac{R}{L}t} = 1 - e^{-\frac{t}{\tau}}, t \geq 0$$

In the above equation, $\tau = L / R$ is the time constant of the circuit, and when $t = \tau$, $v(t) = 1 - e^{-1} = 0.63$. The figure below shows the waveform of $v(t)$.



(Figure) Unit step response

To observe the above transient phenomenon using an oscilloscope, inputting a square wave as the input signal allows observation of the output waveform shown in the figure below. From this output waveform, the time constant (τ) can be measured..



(Figure) Input and Output Waveforms

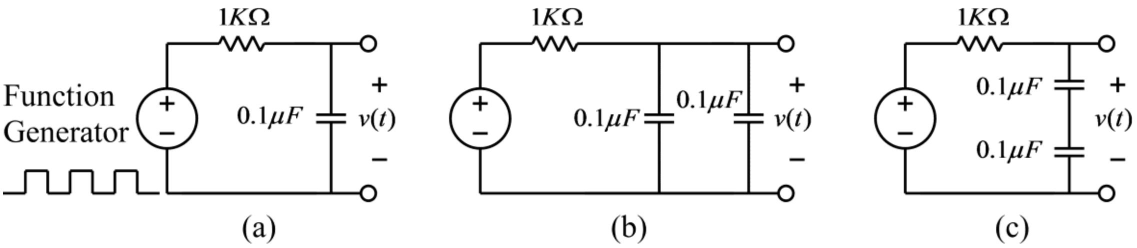
Instruments and Components Used

- Oscilloscope: 1
- Signal generator: 1
- Capacitor: $0.1\ \mu\mathrm{F}$ (polyester film capacitor) x2

- Inductor: $100\ \mathrm{mH}$ x2
- Resistor: $1\ \mathrm{K\Omega}$ x1

Experimental Methods and Procedures

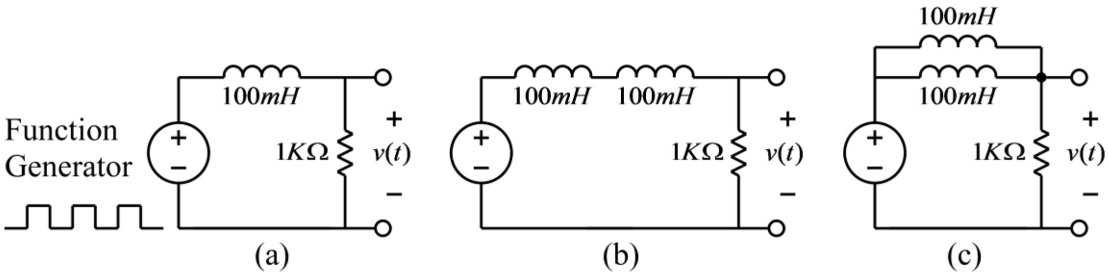
A. Measurement of the time constant in an RC circuit



(Figure) RC Circuit Time Constant Measurement Circuit

(1) Construct the circuit shown in (a) of the figure above, connecting a function generator to the input side and an oscilloscope to the output side.
(2) Set the function type of the function generator to square wave, with a frequency of approximately 500Hz (period 2msec) and a voltage amplitude of 1 volt. That is, set the difference between the low voltage and high voltage of the square wave to 1 volt; the low voltage does not necessarily need to be 0 volts. When adjusting the function generator, always verify the correct settings by measuring the output waveform with an oscilloscope.
(3) Next, measure the time constant of the RC circuit using the method described above, then compare it with the theoretical value.
(4) Repeat the above experiments for circuits (b) and (c) in the figure above.

B. Measurement of RL Circuit Constants



(Figure) RL Circuit Constant Measurement Circuit

(1) Construct the circuit shown in (a) of the figure above, connecting a function generator to the input side and an oscilloscope to the output side.

(2) Set the function generator to a square wave, with a frequency of approximately 500Hz (period 2msec) and a voltage amplitude of 1 volt. That is, set the difference between the low and high voltages of the square wave to 1 volt; the low voltage does not necessarily need to be 0 volts. When adjusting the function generator, always verify the correct settings by measuring the output waveform with the oscilloscope.

(3) Next, measure the time constant of the RL circuit using the method described above and compare it with the theoretical value.

(4) Repeat the above experiments for circuits (b) and (c) in the figure above.

Important Note: When using two inductors, they must be placed on the breadboard with a distance of at least 3 cm between them. If the distance between the inductors is too close, mutual inductance effects may influence the overall inductance value.

● Review of Experimental Results and Discussion

<Table> Measurement of Time Constant in RC Circuit

Circuit	R	C	Time Constant (theoretical value)	Time Constant (measured value)
(a)	$1K\Omega$			
(b)	$1K\Omega$			
(c)	$1K\Omega$			

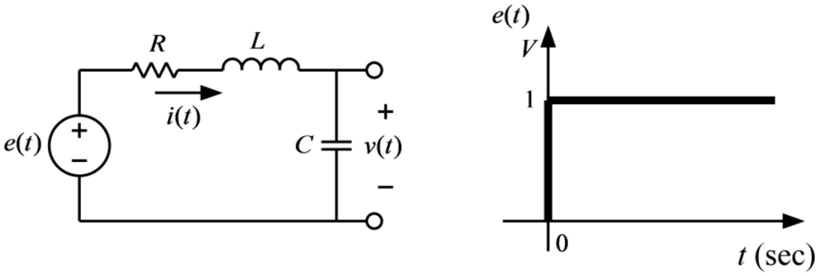
<Table> Measurement of RL Circuit Constants

Circuit	R	L	Time Constant (theoretical value)	Time Constant (measured value)
(a)	$1K\Omega$			
(b)	$1K\Omega$			
(c)	$1K\Omega$			

Lab 4. Transient Characteristics of RLC Circuits

- **Transient parameter measurement of a series RLC circuit**

To analyze the transient behavior of an RLC circuit, consider the circuit shown in the figure below. The input to this circuit is a voltage signal $e(t)$. Determine the voltage $v(t)$ across the capacitor when the input signal is a unit step function, as shown in the figure below.



(Figure) Series RLC Circuit

Applying KVL to this circuit yields the following:

$$Ri(t) + L\frac{di(t)}{dt} + v(t) = e(t)$$

Meanwhile, the current flowing through the capacitor is as follows:

$$i(t) = C\frac{dv(t)}{dt}.$$

Eliminating the current from the above two equations yields the following second-order differential equation.

$$LC\frac{d^2v(t)}{dt^2} + RC\frac{dv(t)}{dt} + v(t) = e(t)$$

The characteristic equation of the above differential equation is as follows:

$$s^2 + \frac{R}{L}s + \frac{1}{LC} = 0$$

This characteristic equation can be expressed in the following standard form:

$$s^2 + 2\alpha s + \omega_0^2 = 0$$

In the above equation, α is the damping coefficient, ω_0 is the natural frequency, and from the two equations above, the following relationship can be derived:

$$\alpha = \frac{R}{2L}, \omega_0 = \frac{1}{\sqrt{LC}}$$

The roots of the characteristic equation expressed in standard form are as follows:

$$s = -\alpha \pm \sqrt{\alpha^2 - \omega_0^2}$$

The roots of the characteristic equation fall into three categories:

1) Two real roots: $\alpha > \omega_0$ (overdamped)
2) A double root: $\alpha = \omega_0$ (critically damped)
3) Two complex roots: $\alpha < \omega_0$ (underdamped)

Solve the differential equation for the underdamped case among the three cases above to determine the response to a unit step input. In this case, the roots of the characteristic equation are as follows:

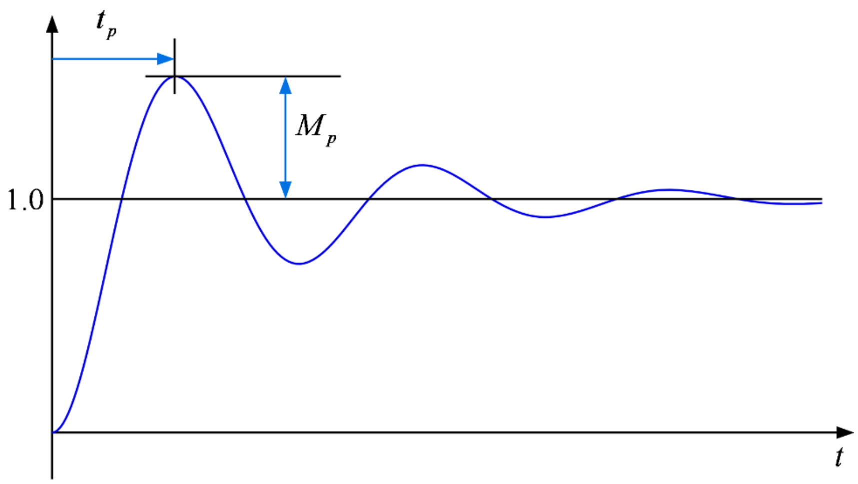
$$s = -\alpha \pm j\sqrt{\omega_0^2 - \alpha^2} = -\alpha \pm j\omega_d$$

In the above equation, $\omega_d = \sqrt{\omega_0^2 - \alpha^2}$ is damped natural frequency. In this case, the response to a unit step input is given by the following equation, and the general form of the response is as shown in the figure below.

$$v(t) = -\frac{1}{\sqrt{1-\left(\frac{\alpha}{\omega_0}\right)^2}} e^{-\alpha t} \cos\left(\omega_d t - \phi\right) + 1, \quad t \geq 0$$

In the above equation, the phase angle ϕ is defined as follows:

$$\phi = \tan^{-1} \frac{\left(\frac{\alpha}{\omega_0}\right)}{\sqrt{1-\left(\frac{\alpha}{\omega_0}\right)^2}}$$



(Figure) Unit step response

In the figure above, t_p is referred to as peak-time, and M_p is referred to as overshoot. Their values are calculated as follows:

$$t_p = \frac{\pi}{\omega_d}$$

$$M_p = e^{-\alpha\pi/\omega_d}$$

From the above equation, the following equations can be obtained.

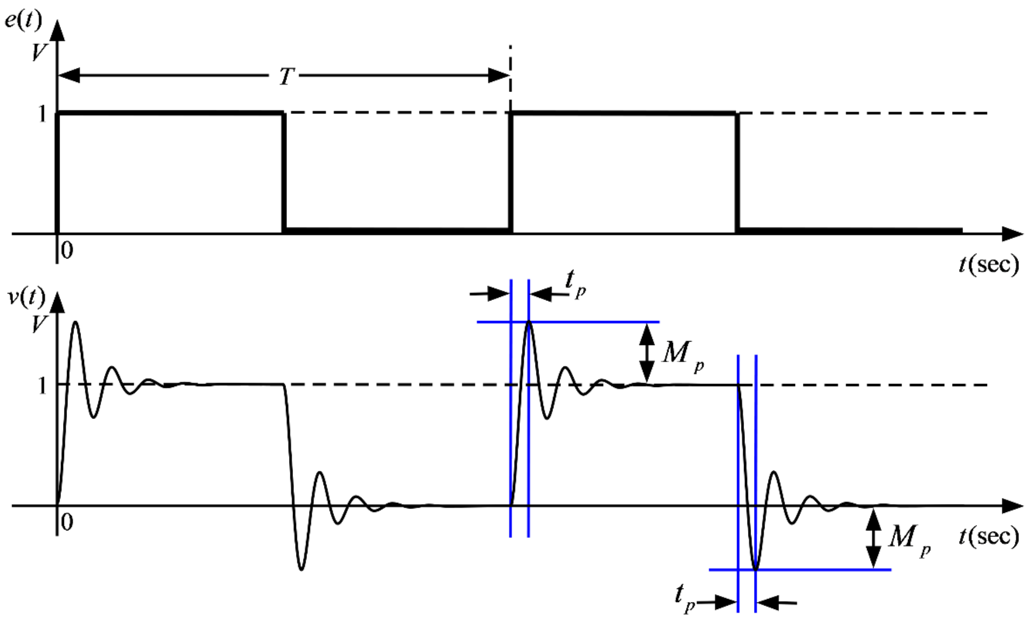
$$\omega_d = \frac{\pi}{t_p}$$

$$\alpha = -\frac{\omega_d}{\pi}\ln M_p$$

Therefore, by measuring the peak time and overshoot value of the unit step response, we can determine α and ω_d using the above equation, and we can also determine ω_0 from the relationship below.

$$\omega_0 = \sqrt{\omega_d^2 + \alpha^2}$$

To observe the above transient phenomena using an oscilloscope, inputting a square wave as the input signal allows observation of the output waveform shown in the figure below. From this output waveform, the parameters of the transient phenomena can be measured.



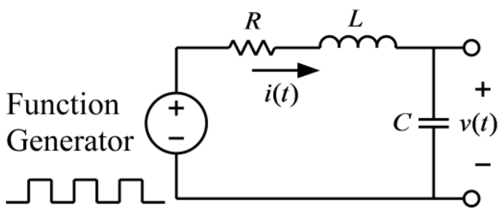
(Figure) Input waveform and output waveform

Instruments and Componets Used

- Oscilloscope: 1
- Function generator: 1
- Capacitor: $0.1\ \mu\text{F}$ x2
- Inductor: $100\ \text{mH}$ x1
- Resistor: $1\ \text{K}\Omega$ x1, $100\ \Omega$ x2

Experimental Methods and Procedures

A. Parameter Measurement of a Series RLC Circuit



(Figure) Parameter measurement circuit for an RLC circuit

(1) Construct the circuit shown in the figure above, connecting a function generator to the input side and an oscilloscope to the output side. The values of each component are as follows:

$$R = 100\ \Omega,\ L = 100\ \text{mH},\ C = 0.1\ \mu\text{F}$$

(2) Set the function type of the function generator to square wave, with a frequency of approximately 100Hz (period 10msec) and a voltage amplitude of 1 volt. That is, set the difference between the low voltage and high voltage of the square wave to 1 volt; the low voltage does not necessarily need to be 0 volts. When adjusting the function generator, always verify the correct settings by measuring the output waveform with an oscilloscope.

(3) Next, using the method described above, measure the peak time and overshoot of the RLC circuit, then compare them with the theoretical values.

(4) For the circuit shown in the figure above, change the component values as follows and repeat the above experiments.

$$R = 200\Omega, L = 100mH, C = 0.1\mu F$$
$$R = 1K\Omega, L = 100mH, C = 0.1\mu F$$
$$R = 100\Omega, L = 100mH, C = 0.2\mu F$$
$$R = 200\Omega, L = 100mH, C = 0.2\mu F$$
$$R = 1K\Omega, L = 100mH, C = 0.2\mu F$$

(5) Compare the experimental results with the results obtained from the Pspice simulation of the

above circuit, which are included in the results report.

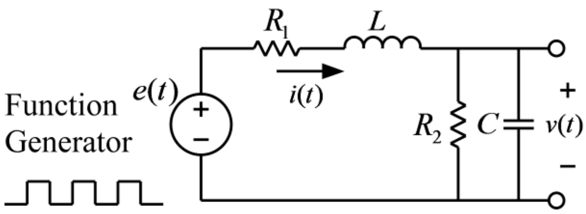
Note 1) Unlike an ideal inductor, actual inductors used in practice contain both inductive and resistive components in the coil. In this experiment, this resistance value cannot be ignored. Therefore, the resistance value of the inductor must be measured using a multimeter and added to the resistance value in the circuit above for calculation.

Note 2) Since this circuit is a linear system, the output is linearly proportional to the magnitude of the input signal. Therefore, the experiment above is possible even if the square wave from the function generator is not necessarily set to 1 volt. When the input square wave amplitude is not 1 volt, the measured overshoot must be scaled by dividing the overshoot value read on the oscilloscope by the square wave amplitude. For example, if a 2-volt square wave is applied and the overshoot on the oscilloscope is 0.6 volts, $M_p = 0.6/2.0 = 0.3$. That is, if a 1-volt square wave is applied to the same circuit, the overshoot magnitude would be 0.3 volts.

Note 3) The frequency of the approved square wave does not necessarily need to be fixed at 100Hz. The frequency can be increased or decreased as needed. However, if the square wave frequency is excessively high, the period becomes too short, potentially causing the voltage to change before reaching steady state. Therefore, the frequency must be adjusted to ensure the input voltage changes only after steady state is achieved.

B. Modified RLC Circuit

(This experiment is for reference only and should not be conducted unless otherwise instructed.)



(Figure) Modified RLC Circuit

(1) Construct the circuit shown in the figure above, connecting a function generator to the input side and an oscilloscope to the output side. The values of each component are as follows.

$$R_1 = 100\Omega, L = 100mH, C = 0.1\mu F$$
$$R_2 = 1K\Omega$$

(2) Observe the difference in transient response between the circuit without the resistance R_2 and the circuit with the resistance R_2, and measure and compare the parameter values. Include

in the report a theoretical explanation of how adding the R_2 resistance affects the damping coefficient and natural frequency. Compare the theoretical results with the experimental results and describe them in the final report.

Hint) When there is an R_2 resistor, the differential equation is as follows.

$$R_1 i(t) + L\frac{di(t)}{dt} + v(t) = e(t)$$

$$i(t) = C\frac{dv(t)}{dt} + \frac{v(t)}{R_2}$$

Let us derive the damping coefficient and natural frequency from the above differential equation.

Review of Experimental Results and Discussion

<Table> Transient Response Parameters of RLC Circuits

R	L	C	R^*	Calculated value			Measured value				
				ω_d	α	ω_0	t_p	M_p	ω_d	α	ω_0
100Ω	$100mH$	$0.1\mu F$									
200Ω	$100mH$	$0.1\mu F$									
$1K\Omega$	$100mH$	$0.1\mu F$									
100Ω	$100mH$	$0.2\mu F$									
200Ω	$100mH$	$0.2\mu F$									
$1K\Omega$	$100mH$	$0.2\mu F$									

Note) R^* is total resistance value including the inductor's resistance.

Lab 5. Characteristics of Capacitors and Inductors – AC Circuits

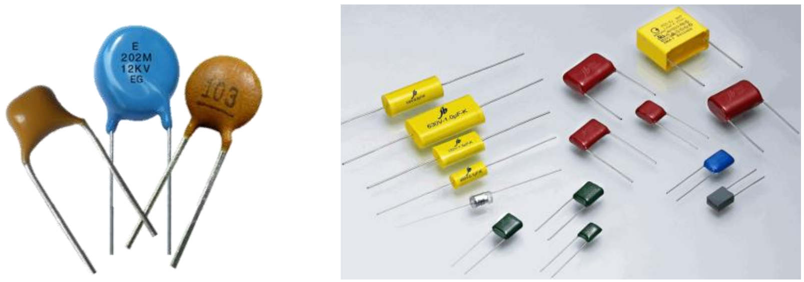
● Types of Capacitors

A capacitor is a component consisting of two opposing electrode plates separated by a dielectric material, serving to store electrical charge. (In Korea, it is often called a condenser instead of a capacitor.) Capacitors come in various types depending on their structure and materials, and their characteristics may differ by type, making it important to select and use the appropriate type. The primary classification of capacitors is into polarized and non-polarized types. Among polarized types, electrolytic capacitors (electrolytic condensers) are common, and their typical shape is as shown in the following figure.

(Figure) Electrolytic Capacitor

Electrolytic capacitors are polarized, so as shown in the figure above, the negative terminal is marked with a minus (-) sign on its surface. Electrolytic capacitors can achieve large capacitance in a relatively small size, but caution is required as there is a risk of explosion if voltage is applied in the reverse direction. Electrolytic capacitors are primarily used in circuits where the applied voltage direction does not change, such as DC power supplies. **In experiments using capacitors from this textbook, both positive and negative voltages may be applied across the capacitor terminals. Therefore, electrolytic capacitors (electrolytic condensers) with the shape shown above are not used.**

Non-polar capacitors include ceramic capacitors, film capacitors, and others, and the figure below shows their typical shapes.



(Figure) Ceramic capacitor, film capacitor

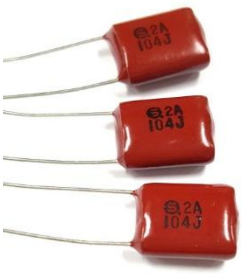
The unit of capacitance for a capacitor is F (Farad), but since this value is extremely large in practice, the units commonly used for capacitance are as follows:

$$1\ \mu\text{F} = 10^{-6}\,\text{F}$$
$$1\ \text{nF} = 10^{-9}\,\text{F}$$
$$1\ \text{pF} = 10^{-12}\,\text{F}$$

As shown in the figure above, the capacitance of a non-polar capacitor is denoted by a three-digit number in pF (picofarad), the smallest unit of capacitance. The first two digits represent the significant figures, while the last digit indicates the power of 10. The following are several examples of capacitor capacitance notation.

$$103 \rightarrow 10\times10^{3}\ \text{pF} = 0.01\ \mu\text{F}$$
$$104 \rightarrow 10\times10^{4}\ \text{pF} = 0.1\ \mu\text{F}$$
$$105 \rightarrow 10\times10^{5}\ \text{pF} = 1\ \mu\text{F}$$
$$225 \rightarrow 22\times10^{5}\ \text{pF} = 2.2\mu\text{F}$$
$$684 \rightarrow 68\times10^{4}\ \text{pF} = 0.68\mu\text{F}$$

The capacitor most frequently used in this experimental manual is the $0.1\ \mu\text{F}$ polyester film capacitor (also called a Mylar capacitor) shown in the following figure. Polarized electrolytic capacitors (electrolytic capacitors) are not used in any of the experiments described in this manual.



(figure) $0.1\ \mu\text{F}$ Polyester Film Capacitor

● Types of Inductors

An inductor is a component made by winding copper wire multiple times around a core, such as ferrite. Depending on the circuit in which it is used, there are various types, including power supply circuits, general circuits, and high-frequency circuits. The following figure shows various forms of inductors.

(Figure) Various Types of Inductors

The unit of inductance is H (Henry). Since 1 H is an extremely large value, smaller units such as $\mu H = 10^{-6}$ H or $mH = 10^{-3}$ H are typically used to represent inductance values. The notation method uses three-digit numbers for the μH unit value. The first two digits are the significant figures, and the last digit indicates the power of 10. For example, an inductor labeled 104 has the following value:

$$10 ? 10^4 \ \mu H = 100 \ mH$$

In this experiment, a 100mH inductor will be primarily used, typically labeled as 104 in the form shown in the following figure.

(Figure) The form of the inductor used in this experiment

- **Capacitance Measurement of Capacitors**

In this experiment, we investigate the relationship between voltage and current in a capacitor when an alternating current (AC) power source is connected to a circuit containing the capacitor, under steady-state conditions. The relationship between current and voltage for a capacitor with capacitance C is given by the following equation:.

$$i_C = C\frac{dv_C}{dt}$$

Under steady-state conditions, the voltage across the capacitor is assumed to be as follows:

$$v_C = V_C \sin \omega t$$

Substituting the above voltage into the voltage-current relationship equation yields the following equation:

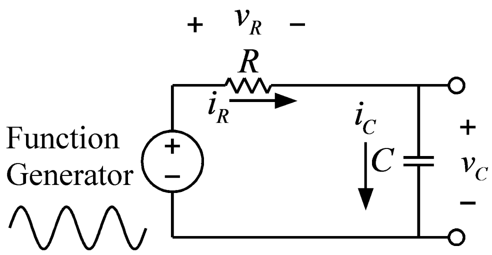
$$C\frac{dv_C}{dt} = CV_C\omega \cos \omega t = I_C \cos \omega t = i_C$$

Therefore, the following relationship can be derived from the above equation.

$$I_C = CV_C\omega$$

$$C = \frac{I_C}{V_C\omega}$$

That is, knowing the magnitude of the current and voltage flowing through the capacitor allows us to calculate the capacitance value. To do this, we construct the following circuit.



(Figure) RC Circuit

In the circuit above, we measure the voltage and current across the capacitor to calculate its capacitance value. For DC circuits, voltage and current can be measured using a multimeter. However, for AC circuits like the one above, an oscilloscope is used. While multimeters also have functions to measure voltage and current in AC circuits, this function is intended for use with 60Hz commercial AC power sources and cannot be used for AC circuits at arbitrary frequencies. AC voltage is measured by reading the peak-to-peak voltage of the sine wave using an oscilloscope. For example, if an AC voltage like $V \sin \omega t$ is measured using an oscilloscope to read the peak-to-peak value, that value will be twice V . Therefore, dividing the peak-to-peak value read by the oscilloscope by two gives the V value. For current, since an oscilloscope cannot directly read current, the voltage across a resistor is measured and divided by the resistance to obtain the value. In this experiment, the current flowing through the resistor and the capacitor are equal. Therefore, the voltage across the resistor is measured using an oscilloscope, and then divided by the resistance value to obtain the capacitor's current value.

Furthermore, the current through a capacitor is obtained by differentiating the voltage, and when the voltage is a sine function, the current is a cosine function. The cosine function leads the sine function by 90 degrees in phase. In this experiment, by simultaneously comparing the voltage across the resistor and the voltage across the capacitor using two channels of an oscilloscope, we confirm that the current through the capacitor leads the voltage by 90 degrees.

● Measuring the Inductance of an Inductor

This experiment investigates the relationship between voltage and current in an inductor when an alternating current (AC) power source is connected to a circuit containing the inductor, under steady-state conditions. The relationship between current and voltage for an inductor with inductance L is given by the following equation.

$$v_L = L\frac{di_L}{dt}$$

Under steady-state conditions, the current through the inductor is assumed to be as follows:

$$i_L = I_L \sin \omega t$$

Substituting the above voltage into the voltage-current relationship equation yields the following equation:

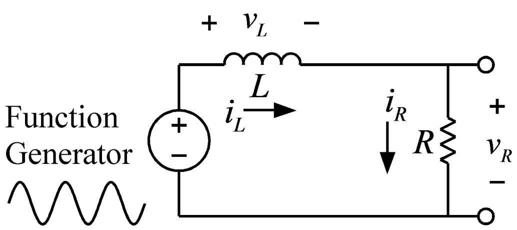
$$L\frac{di_L}{dt} = LI_L\omega\cos\omega t = V_L\cos\omega t = v_L$$

Therefore, the following relationship can be derived from the above equation.

$$LI_L\omega = V_L$$

$$L = \frac{V_L}{I_L\omega}$$

That is, knowing the magnitude of the current and voltage flowing through the inductor allows us to calculate the value of the inductance. To do this, we construct the following circuit.



(Figure) RL Circuit

In the circuit above, we measure the voltage and current across the inductor to calculate its inductance value. For DC circuits, voltage and current can be measured using a multimeter. However, for AC circuits like the one above, an oscilloscope is used. While multimeters also have functions to measure voltage and current in AC circuits, this function is intended for use with 60Hz commercial AC power sources and cannot be used for AC circuits at arbitrary frequencies. AC voltage is measured by reading the peak-to-peak voltage of the sine wave using an oscilloscope. For example, if an AC voltage like $V\sin\omega t$ is measured using an oscilloscope, the peak-to-peak value read will be twice the V value. Therefore, dividing the peak-to-peak value read by the oscilloscope by two gives the V value. For current, since an oscilloscope cannot directly read current, the voltage across a resistor is measured and divided by the resistance to obtain the value. In this experiment, the current flowing through the resistor and the current flowing through the inductor are equal. Therefore, the voltage across the resistor is measured using an oscilloscope, and then divided by the resistance value to obtain the current value through the inductor.

Furthermore, the voltage across an inductor is obtained by differentiating the current. When the current is a sine function, the voltage is a cosine function. The cosine function leads the sine function by 90 degrees in phase. In this experiment, we simultaneously compare the voltage across the resistor and the voltage across the inductor using two channels of the oscilloscope, confirming that the current through the inductor lags the voltage by 90 degrees in phase.

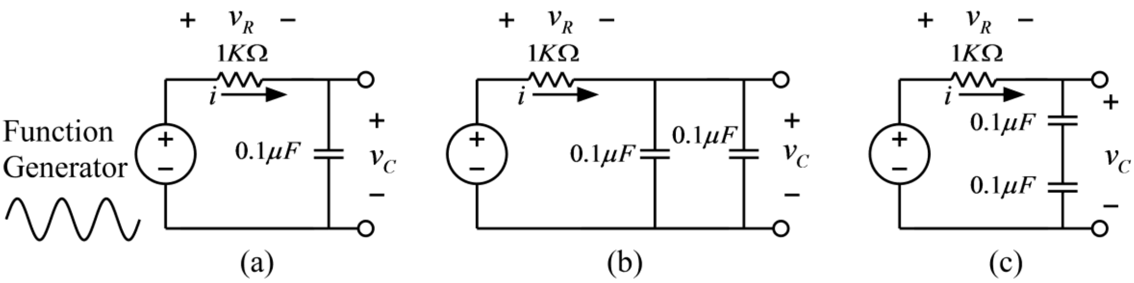
Instruments and Components Used

- Oscilloscope: 1
- Function gegenrator: 1
- Capacitor: $0.1\mu F$ (polyester film capacitor) x2

- Inductor: $100mH$ x2
- Resistor: $1K\Omega$ x1

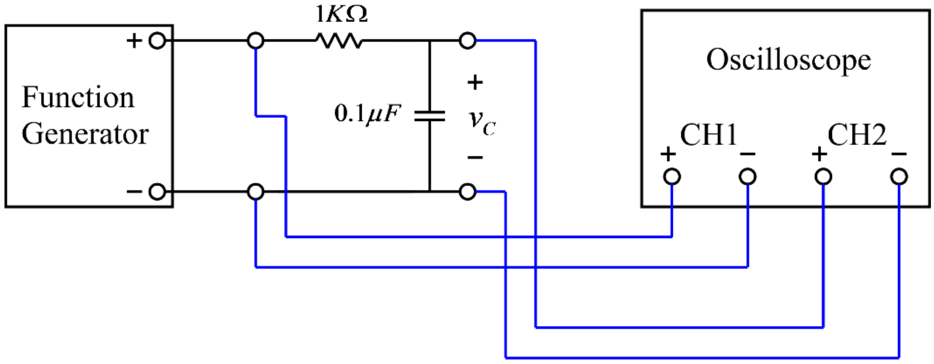
Experimental Methods and Procedures

A. Capacitance Measurement



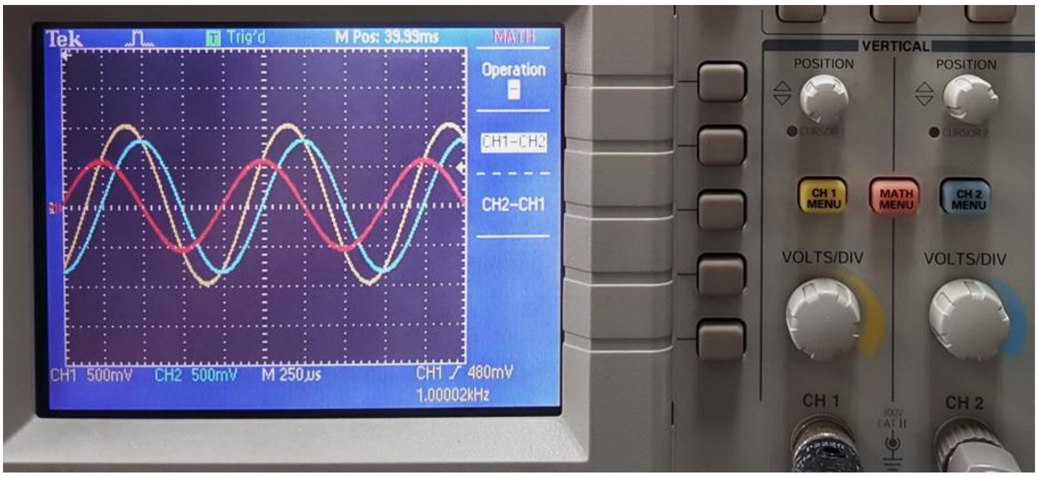
(Figure 1) Capacitance Measurement Circuit

(1) Assemble the circuit shown in Figure 1(a) and connect a function generator to the input side.
(2) Set the function type of the function generator to sine wave, with a frequency of 1000Hz (period of 1msec) and a voltage amplitude of 2 volts (peak-to-peak). That is, ensure the difference between the lowest and highest voltages of the sine wave is 2 volts. When adjusting the function generator, always verify the correct settings by measuring the output waveform with an oscilloscope.
Note: When measuring the amplitude of a sinusoidal voltage, since you are measuring the voltage amplitude V_m in the function $v(t) = V_m \sin \omega t$, it is convenient to read the difference between the lowest and highest voltages (peak-to-peak) on the oscilloscope and divide it by two.
(3) Measure the voltage across the resistor and the voltage across the capacitor, then fill in <Table 1> to determine the measured capacitance value. Connect the capacitor and resistor as shown in Figure 2 to measure their voltages. At this time, use Oscilloscope Channel 1 to measure the function generator output and Channel 2 to measure the voltage across the capacitor.



(Figure 2) Capacitor Voltage Measurement

To measure the voltage across the resistor in the circuit shown above, use the MATH menu on the oscilloscope. Pressing the MATH button on the oscilloscope displays the MATH menu as shown below. In this menu, select Operation as – (minus), then choose CH1-CH2. The result of subtracting Channel 2 from Channel 1 appears as a red line. In the circuit diagram above, the voltage obtained by subtracting Channel 2 from Channel 1 is the voltage across the resistor. Therefore, the red line on the oscilloscope screen below represents the voltage across the resistor. This allows us to measure the voltage across the resistor. Furthermore, the current through the capacitor is equal to the current through the resistor. Since voltage and current share the same phase in the resistor, the phase difference between the capacitor's voltage and current can be measured on the oscilloscope screen below. Using this method, measure the phase difference and determine whether the capacitor's current or voltage leads.

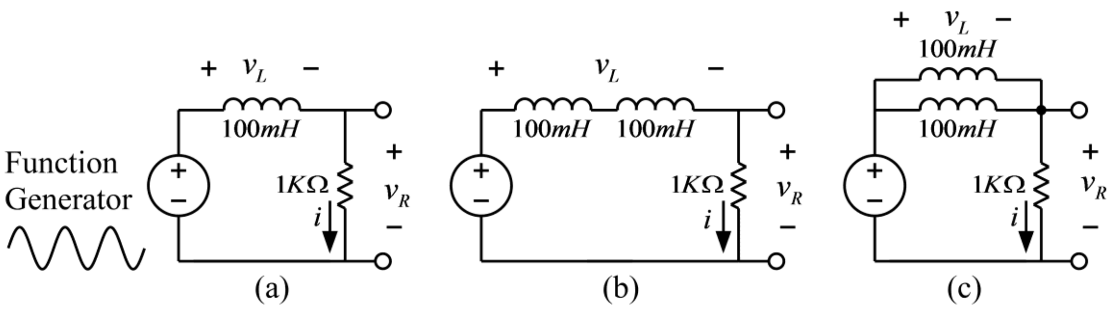


(Figure 3) Measurement using the MATH menu of the oscilloscope

(4) Change the function generator's frequency to 2000 Hz and repeat the above process.

(5) Repeat the above experiments for the circuits shown in Figure 1(b) and (c).

B. Inductance Measurement

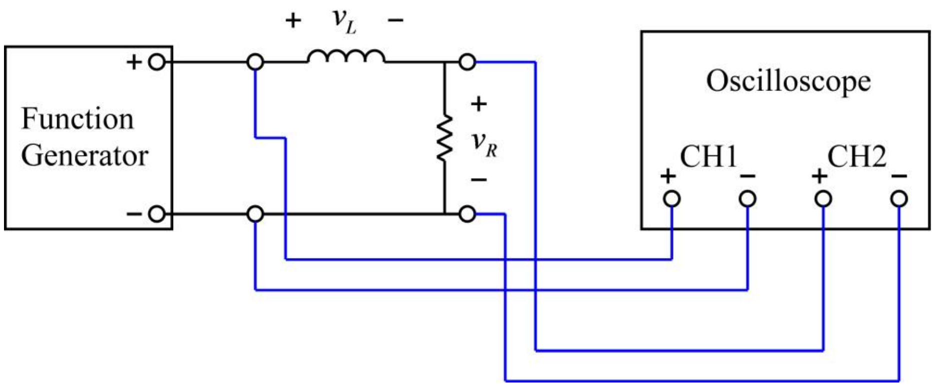


(Figure 4) Inductance Measurement Circuit

(1) Assemble the circuit shown in Figure 4(a) and connect a function generator to the input side.

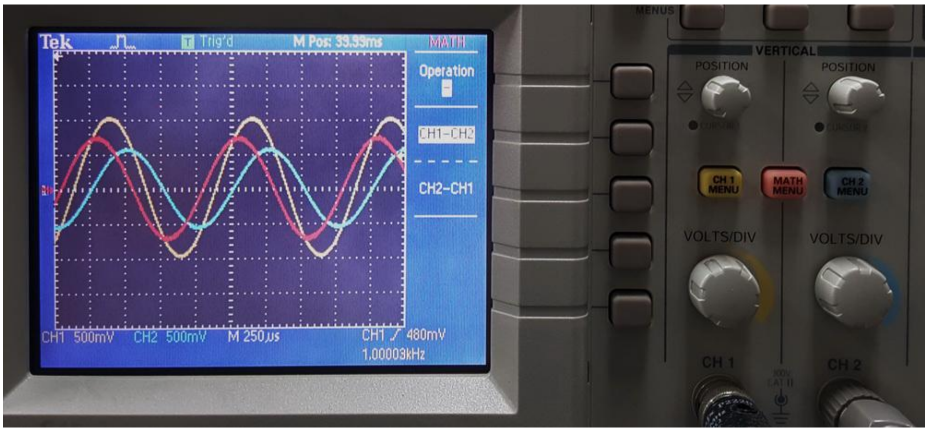
(2) Set the function type of the function generator to sine wave, with a frequency of 1000Hz (period of 1msec) and a voltage amplitude of 2 volts (peak-to-peak). That is, ensure the difference between the lowest and highest voltages of the sine wave is 2 volts. When adjusting the function generator, always verify the correct settings by measuring the output waveform with an oscilloscope.

(3) Measure the voltage across the resistor and the voltage across the inductor, then fill in <Table 2> to determine the measured inductance value. Connect the inductor and resistor as shown in Figure 5 to measure their voltages. At this time, channel 1 of the oscilloscope measures the output from the function generator, and channel 2 measures the voltage across the resistor.



(Figure 5) Inductor Voltage Measurement

To measure the voltage across the inductor terminals in the configuration shown in the figure above, use the MATH menu on the oscilloscope. Pressing the MATH button on the oscilloscope displays the MATH menu as shown below. In this menu, select Operation as – (minus), then select CH1-CH2. The result of subtracting Channel 2 from Channel 1 appears as a red line. In the circuit diagram above, the voltage obtained by subtracting Channel 2 from Channel 1 is the voltage across the inductor. Therefore, the red line on the oscilloscope screen below represents the voltage across the inductor. This allows us to measure the inductor's voltage. Furthermore, the current through the inductor is equal to the current through the resistor. Since voltage and current in a resistor have the same phase, the phase difference between the inductor voltage and inductor current can be measured on the oscilloscope screen below. Using this method, measure the phase difference and determine whether the inductor current or voltage leads.



(Figure 6) Measurement using the MATH menu of the oscilloscope

(4) Change the function generator's frequency to 2000 Hz and repeat the above process.

(5) Repeat the above experiments for the circuits shown in Figure 4(b) and (c).

Important Note: When using two inductors, they must be placed on the breadboard with a distance of at least 3 cm between them. If the distance between the inductors is too close, mutual inductance effects may influence the overall inductance value.

- **Review and Discussion of Experimental Results**

<Table 1> Capacitance Measurement

Circuit	Frequency	C	V_C	V_R	$I = V_R / R$	$X_C = V_C / I$	$C = 1/(2\pi f X_C)$
(a)	1000Hz						
(a)	2000Hz						
(b)	1000Hz						
(b)	2000Hz						
(c)	1000Hz						
(c)	2000Hz						

<Table 2> Inductance Measurement

Circuit	Frequency	L	V_R	$I = V_R / R$	V_L	$X_L = V_L / I$	$L = X_L /(2\pi f)$
(a)	1000Hz						
(a)	2000Hz						
(b)	1000Hz						
(b)	2000Hz						
(c)	1000Hz						
(c)	2000Hz						

Lab 6. RC and RL Circuits – AC Circuits

- **Phasor**

Circuit equations for circuits containing capacitors or inductors take the form of differential equations. In circuits with capacitors or inductors, if the voltage or current sources are sinusoidal (AC) rather than direct current (DC), analyzing voltage and current using differential equations can become quite complex. However, if the frequency of the sinusoidal wave is a known fixed value and the analysis is limited to steady-state conditions, using phasors to determine voltage and current can be greatly simplified. A phasor represents a sinusoidal voltage or current as a complex number. For example, consider the following voltage signal with a frequency of f Hz. Assume the frequency is a fixed, known value.

$$v(t) = V\cos\left(\omega t + \phi\right) = V\cos\left(2\pi ft + \phi\right)$$

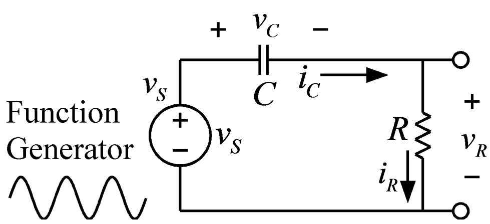
The above equation can be transformed into the following form:

$$v(t) = V\cos\left(\omega t + \phi\right) = \mathrm{Re}\left\{Ve^{j(\omega t+\phi)}\right\} = \mathrm{Re}\left\{Ve^{j\phi}e^{j\omega t}\right\}$$

Since ω is already known in the above equation, the signal can be uniquely specified by only the voltage magnitude V and the phase angle ϕ. The phasor of this signal is defined as the following complex number:

$$V\angle\phi = Ve^{j\phi}$$

As an example of circuit analysis using phasors, consider the following circuit. Assume the power supply voltage is $v_S = V_S\cos\omega t$, and determine the capacitor voltage v_C in steady state.



(Figure) RC Circuit

The circuit equation for the above circuit is the following differential equation:

$$RC\frac{dv_C}{dt} + v_C = v_S$$

Since we are limiting ourselves to the steady-state case, we transform the power equation as follows to find the particular solution for this circuit:

$$v_S = V_S \cos \omega t = \mathrm{Re}\left\{V_S e^{j\omega t}\right\}$$

This circuit is a time-invariant linear circuit, and since the power supply is sinusoidal, the solution v_C to this differential equation is also a sinusoidal wave of the same frequency. We can assume it takes the following form:

$$v_C = V_C \cos\left(\omega t + \phi\right) = \mathrm{Re}\left\{V_C e^{j(\omega t + \phi)}\right\} = \mathrm{Re}\left\{V_C e^{j\omega t} e^{j\phi}\right\}$$

In the above equation, the actual signal is the real part of the signal expressed as a complex function. Therefore, instead of using the real function, we can find the solution using the following complex function, and then take the real part to obtain the actual solution.

$$v_S = V_S e^{j\omega t}$$

$$v_C = V_C e^{j(\omega t + \phi)}$$

Substituting the above complex function into the differential equation yields the following expression:

$$j\omega RCV_C e^{j(\omega t + \phi)} + V_C e^{j(\omega t + \phi)} = V_S e^{j\omega t}$$

$$\left(j\omega RCV_C e^{j\phi} + V_C e^{j\phi}\right) e^{j\omega t} = V_S e^{j\omega t}$$

In the above equation, since $e^{j\omega t}$ is multiplied on both sides, it can be eliminated, resulting in the following equation.

$$j\omega RCV_C e^{j\phi} + V_C e^{j\phi} = \left(j\omega RC + 1\right) V_C e^{j\phi} = V_S$$

The above equation is the phasor equation. Comparing this equation with the original differential equation reveals that the use of phasors transforms the differential equation into an algebraic equation. Furthermore, in the phasor expression, the differential term of the original equation takes the form of $j\omega$ multiplied instead of the differential. This is analogous to the Laplace transform of a differential function being similar to the Laplace transform of the original function multiplied by the Laplace variable s. Rearranging the above equation with respect to $V_C e^{j\phi}$ yields the following:

$$V_C e^{j\phi} = \frac{V_S}{j\omega RC + 1} = \frac{V_S}{\sqrt{\left(\omega RC\right)^2 + 1}} e^{-j\tan^{-1}\omega RC}$$

From the above equation, the following relationship can be derived.

$$V_C = \frac{V_S}{\sqrt{(\omega RC)^2 + 1}}$$

$$\phi = -\tan^{-1}\omega RC$$

Using the above values, the solution to the differential equation involving complex functions can be obtained as follows:

$$v_C = V_C e^{j(\omega t+\phi)} = \frac{V_S}{\sqrt{(\omega RC)^2 + 1}} e^{j\left(\omega t - \tan^{-1}\omega RC\right)}$$

Taking the real part of the above expression, the particular solution to the original differential equation can be obtained as follows:

$$v_C = \frac{V_S}{\sqrt{(\omega RC)^2 + 1}} \cos\left(\omega t - \tan^{-1}\omega RC\right)$$

● Impedance

The impedance of an AC circuit is defined as the voltage phasor divided by the current phasor. Since phasors are complex numbers, impedance is also complex. As a simple example, let's define the impedance of a capacitor. The general relationship for a capacitor is as follows:

$$i_C = C\frac{dv_C}{dt}$$

Voltage v_C is assumed to be as follows.

$$v_C = V_C \cos\omega t = \mathrm{Re}\left\{V_C e^{j\omega t}\right\}$$

The function above is a real-valued function, but for convenience, we assume v_C is a complex-valued function as follows.

$$v_C = V_C e^{j\omega t}$$

Substituting the above equation into the capacitor's relationship yields the following equation:

$$i_C = C\frac{dv_C}{dt} = C\frac{dV_C e^{j\omega t}}{dt} = j\omega C V_C e^{j\omega t}$$

Furthermore, we assume the complex function of the current as follows:

$$i_C = I_C e^{j(\omega t+\phi)}$$

Using the above equation, we can derive the following relationship.

$$i_C = I_C e^{j(\omega t+\phi)} = I_C e^{j\phi} e^{j\omega t} = j\omega C V_C e^{j\omega t}$$

Eliminating $e^{j\omega t}$ from both sides of the above equation yields the following expression:

$$I_C e^{j\phi} = j\omega C V_C$$

The above equation is the relationship between the current phasor and the voltage phasor, from which the impedance of the capacitor can be obtained.

$$Z_C = \frac{V_C}{I_C e^{j\phi}} = \frac{1}{j\omega C}$$

Similarly to the above process, we define the impedance of an inductor. The general relationship for an inductor is as follows:

$$v_L = L\frac{di_L}{dt}$$

The current i_L is assumed to be as follows:

$$i_L = I_L \cos \omega t = \mathrm{Re}\left\{I_L e^{j\omega t}\right\}$$

Although the above function is a real-valued function, for convenience we assume i_L to be a complex-valued function as follows.

$$i_L = I_L e^{j\omega t}$$

Substituting the above expression into the inductor's relationship yields the following equation:

$$v_L = L\frac{di_L}{dt} = C\frac{dI_L e^{j\omega t}}{dt} = j\omega L I_L e^{j\omega t}$$

Furthermore, we assume the complex function of voltage as follows:

$$v_L = V_L e^{j(\omega t+\phi)}$$

Using the above equation, we can derive the following relationship.

$$v_L = V_L e^{j(\omega t+\phi)} = V_L e^{j\phi} e^{j\omega t} = j\omega L I_L e^{j\omega t}$$

Eliminating $e^{j\omega t}$ from both sides of the above equation yields the following expression.

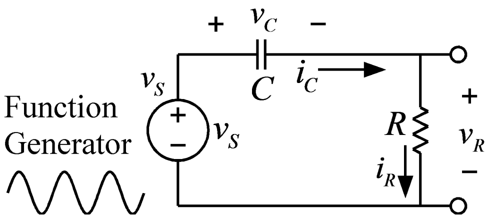
$$V_L e^{j\phi} = j\omega L I_L$$

The above equation is the relationship between the current phasor and the voltage phasor, and from this, the impedance of the inductor can be obtained.

$$Z_L = \frac{V_L e^{j\phi}}{I_L} = j\omega L$$

- **Impedance of RC circuits**

Next, we find the impedance of the following RC circuit.



(Figure) RC Circuit

The impedance of the above circuit, where a resistor and capacitor are connected in series, can be calculated by dividing the voltage phasor by the current phasor. The power supply for this circuit is assumed to be as follows:

$$v_S = V_S \cos \omega t$$

Since the phase of the above equation is 0, the phasor is the real number V_S. Dividing this value by the current phasor yields the impedance of the circuit. The current in this circuit can be found using the capacitor voltage v_C obtained earlier from the following relationship:

$$i_C = C \frac{dv_C}{dt}$$

The phasor of the capacitor voltage is as follows, as previously determined.

$$V_C e^{j\phi} = \frac{V_S}{j\omega RC + 1} = \frac{V_S}{\sqrt{(\omega RC)^2 + 1}} e^{-j\tan^{-1}\omega RC}$$

The phasor of the derivative of a signal is obtained by multiplying the phasor of the original signal by $j\omega$. Therefore, the current phasor can be obtained by multiplying the voltage phasor $V_C e^{j\phi}$ by $j\omega C$.

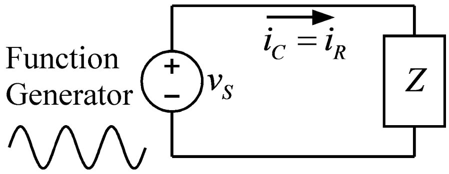
$$j\omega C V_C e^{j\phi} = \frac{j\omega C V_S}{j\omega RC + 1}$$

The impedance of this circuit is the phasor of the power source divided by the current phasor, and can be calculated as follows. Since the phase of the power source is assumed to be zero in this equation, the phasor of the power source is the real-valued V_S.

$$Z = \frac{V_S}{\left(\dfrac{j\omega C V_S}{j\omega RC + 1}\right)} = \frac{j\omega RC + 1}{j\omega C} = R + \frac{1}{j\omega C}$$

From the above equation, we can see that the impedance of this circuit is equal to the sum of the impedance of the resistor and the capacitor.

In this experiment, we measure the impedance value of an RC circuit and compare it with the calculated impedance value obtained from the above equation. The impedance of an RC circuit can be measured as follows. First, apply a signal of voltage $v_S = \cos\omega t$ to the input of the circuit below.



(Figure) Impedance Measurement of an RC Circuit

Next, by measuring the current in the above circuit with an oscilloscope, the following equation for the current signal can be obtained.

$$i_C = I_C \cos(\omega t + \phi)$$

Current measurement using an oscilloscope is possible by measuring the voltage across a resistor and dividing it by the resistance value. In this equation, ϕ is the phase difference between the power supply voltage signal v_S and the current signal i_C. In the above equation, the phase of the power supply voltage is assumed to be 0 for convenience, but since the phase difference between voltage and current is actually important, assuming it is not 0 yields the same result. The magnitude of the current I_C can be obtained by reading the peak-to-peak value of the current and dividing it by two. The phasor of the current is as follows:

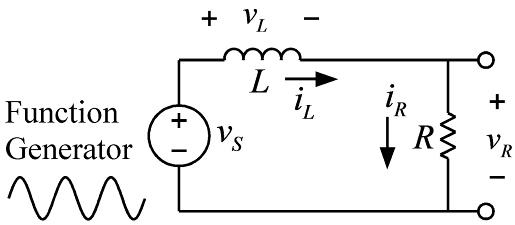
$$I_C\angle\phi = I_C e^{j\phi} = I_C \cos\phi + jI_C \sin\phi$$

The magnitude of the input power voltage is 1, and the phase is 0 degrees, so the phasor is 1. Therefore, the impedance can be obtained as follows:

$$Z = \frac{1}{I_C e^{j\phi}} = \frac{1}{I_C \cos\phi + jI_C \sin\phi}$$

- **Impedance of RL circuit**

Next, we calculate the impedance of the following RL circuit.



(Figure) RL Circuit

The impedance of the above circuit, where a resistor and inductor are connected in series, can be calculated by dividing the voltage phasor by the current phasor. The power supply for this circuit is assumed to be as follows:

$$v_S = V_S \cos \omega t$$

Since the phase of the above equation is 0, the phasor is the real number V_S. Dividing this value by the current phasor yields the impedance of the circuit. The current in this circuit can be determined from the following circuit differential equation:

$$L\frac{di_L}{dt} + Ri_L = v_S$$

To find the current in this circuit at steady state, let the phasor of the current signal i_L be denoted as $I_L e^{j\phi}$. Converting the above differential equation into a phasor equation yields the following. Note that, as in the case of an RC circuit, the differential term in the differential equation has changed to a term multiplied by $j\omega$.

$$j\omega L I_L e^{j\phi} + RI I_L e^{j\phi} = \left(j\omega L + R\right) I_L e^{j\phi} = V_S$$

From the above equation, the impedance can be calculated as follows.

$$Z = \frac{V_S}{I_L e^{j\phi}} = j\omega L + R$$

As with RC circuits, the total impedance is the sum of the resistance value and the impedance value of the inductor. The process of determining the impedance of an RL circuit through experimental measurement is identical to that for an RC circuit and is therefore omitted.

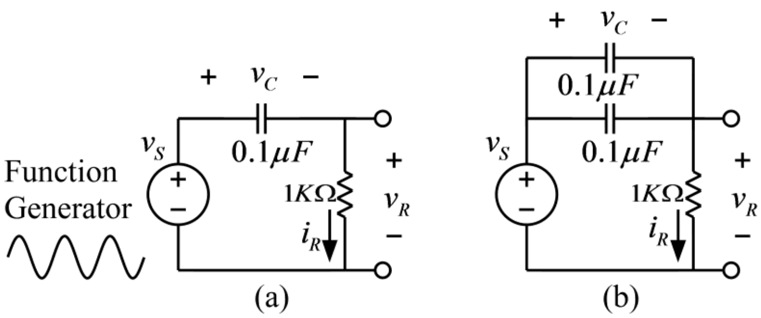
● Instruments and Components Used

- Oacilloscope: 1
- Function generator: 1
- Capacitor: $0.1\ \mu\text{F}$ (polyester film capacitor) x2

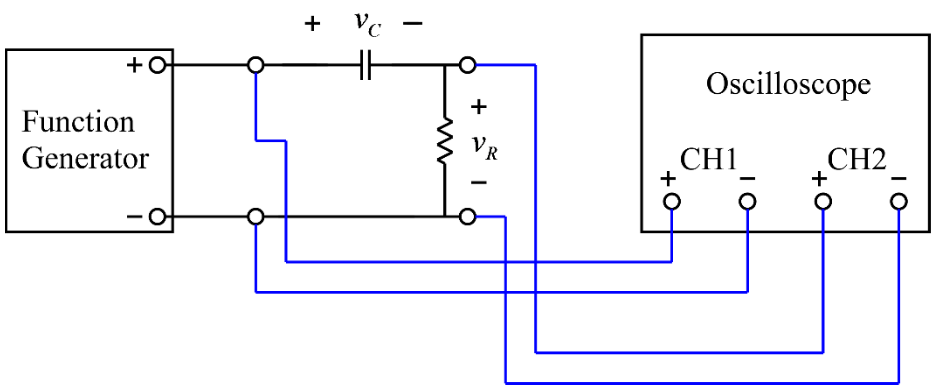
- Inductor: $100\ \text{mH}$ x2
- Resistor: $1\ \text{K}\Omega$ x2

● Experimental Methods and Procedures

A. Impedance Measurement in an RC Series Circuit



(Figure 1) RC Series Circuit



(Figure 2) Connection between the function generator and the oscilloscope

(1) Assemble the circuit shown in Figure 1(a) and connect the function generator and oscilloscope as shown in Figure 2.

(2) Set the function type of the function generator to sine wave, with a frequency of 1000Hz (period

of 1msec) and a voltage amplitude of 2 volts (peak-to-peak). That is, ensure the difference between the lowest and highest voltages of the sine wave is 2 volts. When adjusting the function generator, always verify the correct settings by measuring the output waveform with an oscilloscope.

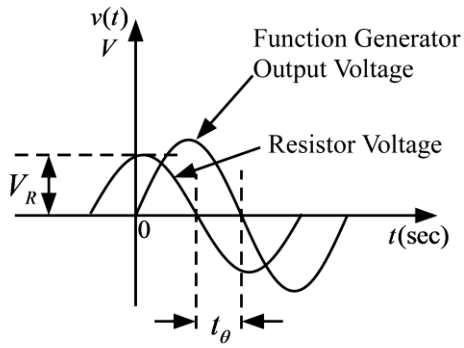
Note: When measuring the amplitude of a sinusoidal voltage, since you are measuring the voltage amplitude V_m in the function $v(t) = V_m \sin(\omega t)$, it is convenient to read the difference between the lowest and highest voltages (peak-to-peak) on the oscilloscope and divide it by two.

(3) Display channels 1 and 2 of the oscilloscope simultaneously on one screen so that the function generator voltage and the voltage across the resistor are visible as shown in Figure 3. In the figure, the magnitude V_R of the voltage across the resistor is obtained by measuring the peak-to-peak voltage across the resistor and dividing it by two. Furthermore, since the phase of the voltage waveform across the resistor and the current waveform are the same, measuring the phase difference between the resistor voltage and the function generator voltage allows the phase of the current relative to the input power source to be determined. The phase difference can be measured by reading the time difference t_θ between the two waveforms in Figure 6.24. Since one cycle of the sine wave used in this experiment is 1 msec, 1 msec corresponds to 360 degrees of phase. Therefore, dividing the measured time difference t_θ between the two waveforms in the figure by 1 msec and multiplying by 360 degrees yields the phase angle. That is, it is as follows:

$$\theta = 360° \times \frac{t_\theta}{0.001} \text{ (degree)}$$

$$\theta = 2\pi \times \frac{t_\theta}{0.001} \text{ (rad)}$$

And, in this case, since the current leads the voltage of the function generator in phase, the phase of the current is greater than zero.

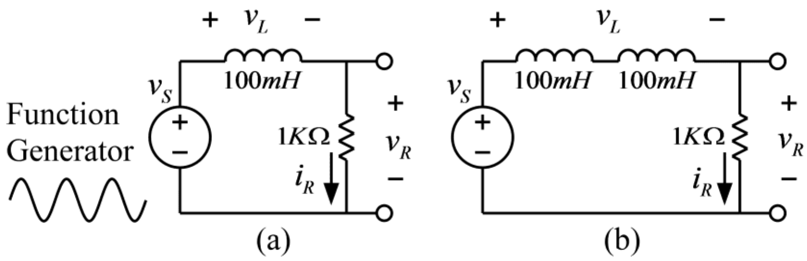


(Figure 3) Measurement of Voltage and Phase Difference

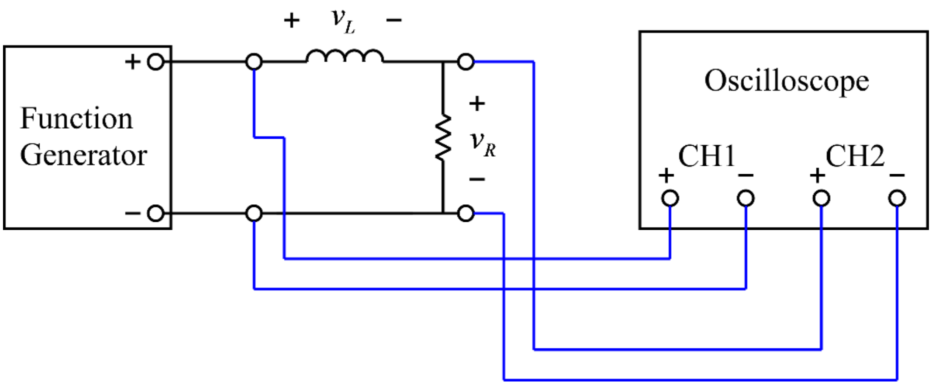
After filling in <Table 1> using the values measured by the above method, compare the calculated impedance value with the measured impedance value. At this time, the input power is a sine wave with a peak-to-peak voltage of 2 volts ($v_S = 1 \cdot \cos \omega t$). Therefore, the phasor for the power source is 1, and the impedance can be calculated by dividing the phasor of the power source by the phasor of the current.

(4) Repeat the above experiments for the circuit shown in Figure 1(b)..

B. Impedance Measurement in RL Series Circuits



(Figure 4) RC Series Circuit



(Figure 5) Connection between the function generator and the oscilloscope

(1) Assemble the circuit shown in Figure 4(a) and connect the function generator and oscilloscope as shown in Figure 5.

(2) Set the function type of the function generator to sine wave, with a frequency of 1000Hz (period of 1msec) and a voltage amplitude of 2 volts (peak-to-peak). That is, ensure the difference between the lowest and highest voltages of the sine wave is 2 volts. When adjusting the function generator, always verify the correct settings by measuring the output waveform with an oscilloscope.

Note: When measuring the amplitude of a sinusoidal voltage, since you are measuring the voltage magnitude V_m in the function $v(t) = V_m \sin(\omega t)$, it is convenient to read the difference between the lowest and highest voltages (peak-to-peak) on the oscilloscope and divide it by two.

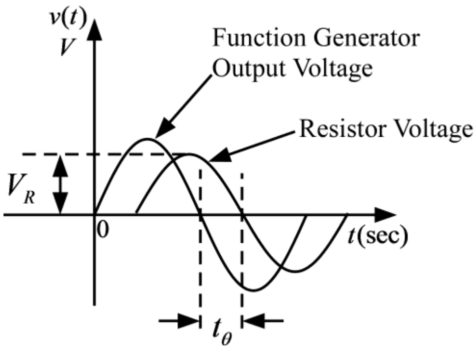
(3) Display channels 1 and 2 of the oscilloscope simultaneously on one screen so that the function generator voltage and the voltage across the resistor are visible as shown in Figure 6. In the figure, the magnitude V_R of the voltage across the resistor is obtained by measuring the peak-to-peak voltage across the resistor and dividing it by two. Furthermore, since the phase of the voltage waveform across the resistor and the current waveform are the same, measuring the phase difference between the resistor voltage and the function generator voltage allows the phase of the current relative to the input power source to be determined. The phase difference can be

measured by reading the time difference t_θ between the two waveforms in Figure 6. Since one cycle of the sine wave used in this experiment is 1 msec, 1 msec corresponds to 360 degrees of phase. Therefore, dividing the measured time difference t_θ between the two waveforms in the figure by 1 msec and multiplying by 360 degrees yields the phase angle. That is, it is as follows:

$$\theta = 360° \times \frac{t_\theta}{0.001}$$

$$\theta = 2\pi \times \frac{t_\theta}{0.001}(rad)$$

And, <u>in this case, the current lags behind the voltage of the function generator, so the phase of the current is less than zero.</u>



(Figure 6) Measurement of Voltage and Phase Difference

Using the values measured by the method described above, fill in <Table 2>. Then, compare the calculated impedance value with the measured impedance value. At this time, the input power is a sine wave with a peak-to-peak voltage of 2 volts ($v_S = 1 \cdot \cos \omega t$). Therefore the phasor for the power source is 1, and the impedance can be obtained by dividing the phasor of the power source by the phasor of the current.

(4) In the case of RL circuits, significant errors can occur. This is because inductors are made by winding copper wire, and the resistance value of the copper wire is not negligible. Measure the resistance value of the inductor using a multimeter. Furthermore, assuming the actual inductor is equivalent to an ideal inductor connected in series with a resistor, the error can be corrected. Correct the results obtained above in this manner and write the corrected results in the report.

(5) Repeat the above experiments for the circuit shown in Figure 4(b).

Important Note: When using two inductors, they must be placed on the breadboard with a distance of at least 3 cm between them. If the distance between the inductors is too close, mutual inductance effects may influence the overall inductance value.

- **Review and Discussion of Experimental Results**

<Table 1> Impedance Measurement in an RC Series Circuit

(Figure 1) Circuit	Calculated value	Measured value				
	$Z = R+1/(j\omega C)$	Resistor voltage V_R	Resistor current $I_R = V_R / R$	Phase difference θ	Current phasor $I = I_R \cos\theta + jI_R \sin\theta$	Impedance $Z = 1/I$
(a)						
(b)						

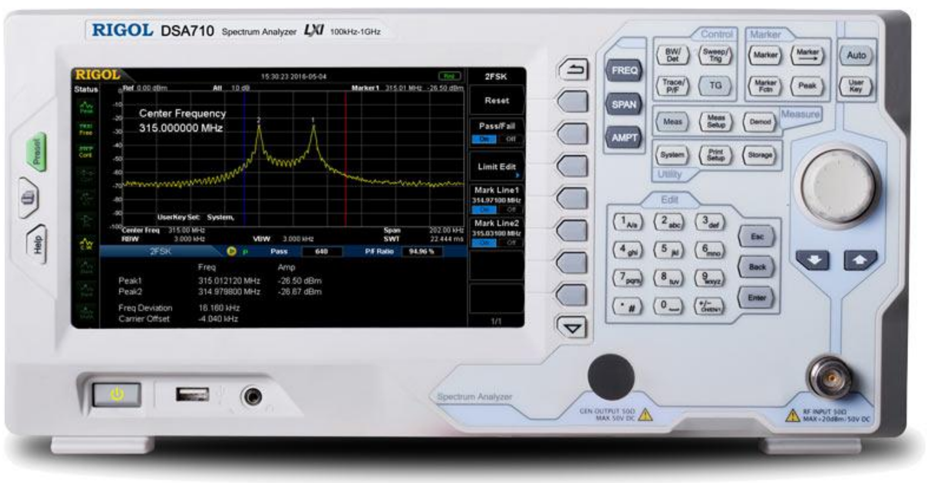
<Table 2> Impedance Measurement in RL Series Circuits

(Figure 4) Circuit	Calculated value	Measured value				
	$Z = R + j\omega L$	Resistor voltage V_R	Resistor current $I_R = V_R / R$	Phase difference θ	Current phasor $I = I_R \cos\theta + jI_R \sin\theta$	Impedance $Z = 1/I$
(a)						
(b)						

Lab 7. Frequency Response of RC and RLC Circuits

- ### Frequency response of the circuit

The concept of frequency response is extremely important in circuit analysis. The transient response experiments conducted in the previous section measured how a circuit changes over time in response to an input signal. In other words, the transient response experiments conducted in the previous section can be considered experiments measuring the circuit's time response. Time response is typically represented graphically, with the horizontal axis of the time response graph representing time. Similarly, frequency response is also typically represented graphically. The horizontal axis of the frequency response graph represents frequency, with units of Hz or rad/sec. The oscilloscope, frequently used in experiments, can be considered a device for measuring time response; the horizontal axis on an oscilloscope screen represents time. Similarly, there are devices for measuring frequency response, called spectrum analyzers. The horizontal axis on a spectrum analyzer, as shown in the figure below, represents frequency.



(Figure) Spectrum Analyzer

Although somewhat inconvenient, it is possible to measure frequency response using an oscilloscope without a spectrum analyzer. In this experiment, we will measure the frequency response of an RLC circuit using an oscilloscope.

When a sinusoidal wave is input to an RLC circuit, measuring the output at steady state yields a sinusoidal wave of the same frequency. (Circuits exhibiting this characteristic belong to the category of linear time-invariant circuits, and all circuits covered in this textbook are linear time-invariant circuits.) At this point, the output sine wave has the same frequency as the input sine wave, but its amplitude and phase differ. Suppose the input signal to a circuit is a sine wave described by the following equation.

$$e(t) = A\cos(\omega t)$$

And when the output signal of this circuit is measured in steady state, it is given by the following equation.

$$v(t) = B\cos(\omega t + \phi)$$

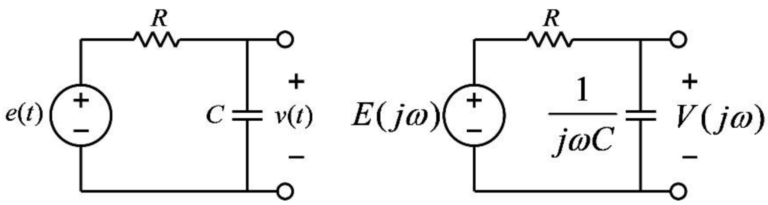
The output signal of this circuit is a sine wave with the same frequency as the input signal, but its amplitude and phase differ from the input signal. The gain of this circuit can be defined as the ratio of the output sine wave amplitude B to the input sine wave amplitude A.

$$Gain = \frac{B}{A}$$

This gain value is constant at a fixed frequency, but varies with the input frequency. ω Changing this also changes the gain value. That is, the gain value is a function of frequency, varying with frequency. Similarly, the phase difference between the input sine wave and the output sine wave ϕ It is a function that varies with frequency. The graph plotting the gain and phase values as they change with frequency is called the frequency response.

- **Network Function**

To define the frequency response using a formula, it is necessary to define the network function. The network function is defined using the phasors of the input signal and the output signal. When the input signal is a sinusoidal wave, the steady-state output signal is a sinusoidal wave with the same frequency as the input signal. The ratio of the phasor of the output signal to the phasor of the input signal is called the network function. Let's find the network function for the following simple RC circuit.



(Figure) RC Circuit

The input signal $e(t)$ to the circuit above is a sinusoidal wave with frequency ω (rad/sec), and the phasor representing this signal is denoted as $E(j\omega)$. This phasor is expressed as a function of ω because the frequency ω can vary, and thus the phasor can change with ω.

Furthermore, the output signal $v(t)$ of this circuit is a sinusoidal wave in steady state, and its phasor is denoted as $V(j\omega)$. Then, the network function of this circuit can be defined by the following equation:

$$G(j\omega) = \frac{V(j\omega)}{E(j\omega)}$$

Since the impedance of a capacitor is $1/(j\omega C)$, the network function of the above circuit can be obtained using the voltage division rule as follows.

$$V(j\omega) = \frac{\frac{1}{j\omega C}}{R + \frac{1}{j\omega C}} E(j\omega) = \frac{1}{j\omega RC + 1} E(j\omega)$$

$$G(j\omega) = \frac{V(j\omega)}{E(j\omega)} = \frac{1}{j\omega RC + 1}$$

- **Definition of Frequency Response**

If a sinusoidal input $e(t) = A\cos(\omega t)$ is applied to the input of a circuit with a network function $G(j\omega)$, the steady-state output is given by the following equation.

$$v(t) = A\left|G(j\omega)\right|\cos\left(\omega t + \angle G(j\omega)\right)$$

That is, the output signal in steady state is a sinusoidal wave of the same frequency as the input signal, and its magnitude is the magnitude of the input signal A multiplied by the value of $\left|G(j\omega)\right|$. Therefore, $\left|G(j\omega)\right|$ acts as a gain, and the magnitude of this gain is a function of the input signal frequency ω. Furthermore, the phase of the output waveform, $\angle G(j\omega)$, is also a function of the input sine wave's frequency ω. The variation in gain and phase with respect to the input sine wave's frequency is referred to as the frequency response of this circuit.

The frequency response can also be defined using the transfer function. The transfer function is defined as the ratio of the Laplace transform of the output signal to the Laplace transform of the input signal. As a simple example, let's find the transfer function for the RC circuit above. Applying Kirchhoff's Voltage Law (KVL) to the RC circuit above yields the following differential equation.

$$RC\frac{dv(t)}{dt} + v(t) = e(t)$$

Applying the Laplace transform to both sides of the differential equation yields the following. When determining the transfer function, it is assumed that all initial conditions are zero.

$$RCsV(s) + V(s) = E(s)$$

In the above equation, $V(s)$ and $E(s)$ are the Laplace transforms of the output voltage $v(t)$ and the input voltage $e(t)$, respectively. From the above equation, the transfer function can be obtained as follows.

$$G(s) = \frac{V(s)}{E(s)} = \frac{1}{RCs + 1}$$

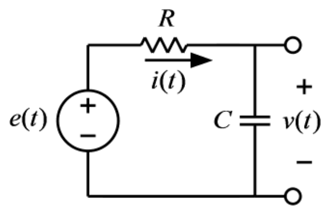
Substituting $s = j\omega$ into the transfer function above yields the same function as the network function:

$$G(j\omega) = G(s)\big|_{s=j\omega} = \frac{1}{j\omega RC + 1}$$

As seen above, the frequency response can be defined using the network function or defined using the transfer function.

- **Frequency Response of a Low-Pass RC Circuit**

To analyze the frequency response of a circuit containing a resistor and a capacitor, consider the circuit shown in the figure below.



(Figure) Low-Pass RC Circuit

The input to this circuit is the voltage signal $e(t)$, and the output is the voltage $v(t)$ across the capacitor terminals. The network function of this circuit is given by the following equation, as shown above.

$$G(j\omega) = \frac{V(j\omega)}{E(j\omega)} = \frac{1}{j\omega RC + 1}$$

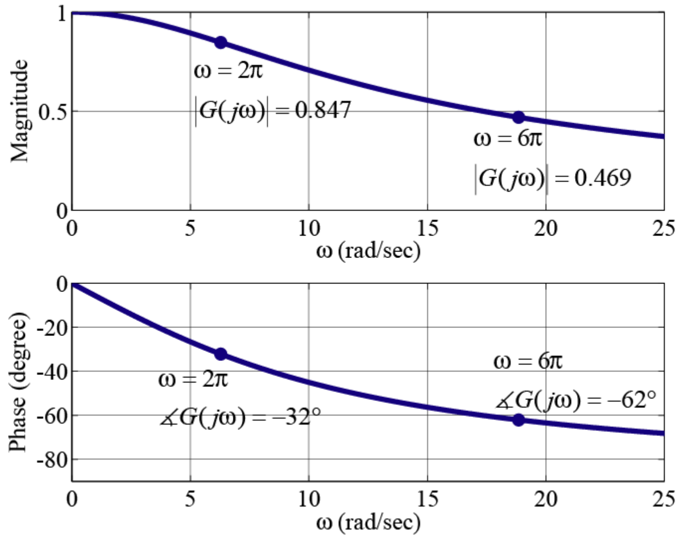
Therefore, the frequency response function of the above circuit is as follows:

$$|G(j\omega)| = \left|\frac{1}{j\omega RC + 1}\right| = \frac{1}{\sqrt{(\omega RC)^2 + 1}}$$

$$\measuredangle G(j\omega) = \measuredangle\left(\frac{1}{j\omega RC + 1}\right) = -\tan^{-1}(\omega RC)$$

For example, substituting the values of the resistor and capacitor in this electrical circuit as follows and plotting the frequency response yields the graph shown below.

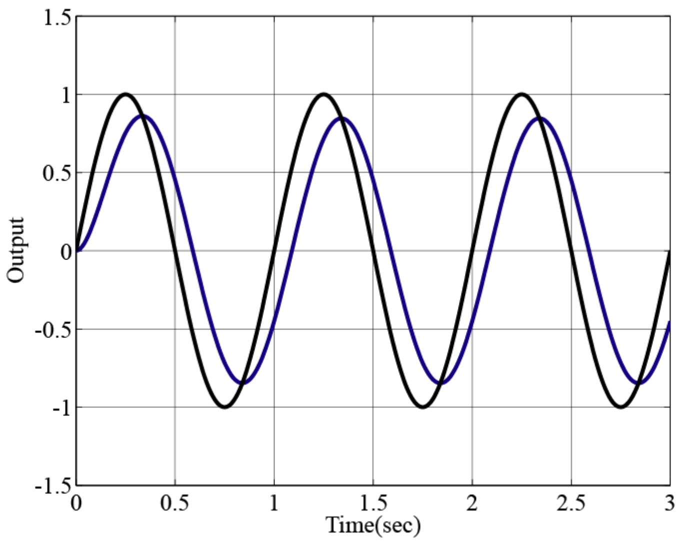
$$R = 100\Omega, C = 0.001F$$



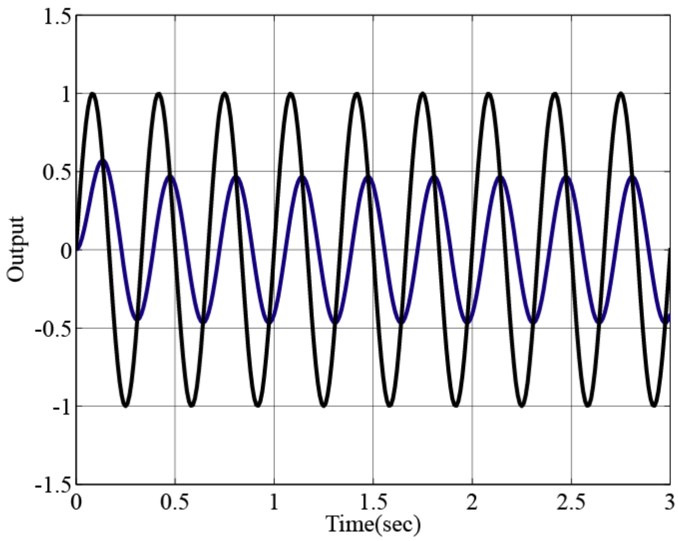
(Figure) Frequency Response

To verify the values of the frequency response in the time response, plotting the time response for the input sine waves at the following two frequencies yields the figures below.

$$f = 1 \text{ Hz}, \ f = 3 \text{ Hz}$$



(Figure) Time response for an input signal of $f = 1$ Hz



(Figure) Time response for an input signal of $f = 3$ Hz

The amplitude of the input sine wave is 1, and for comparison, the input sine wave is plotted alongside each time response. The relationship between frequency $f\,(Hz)$ and radial frequency $\omega(rad/\sec)$ is as follows:

$$\omega = 2\pi f$$

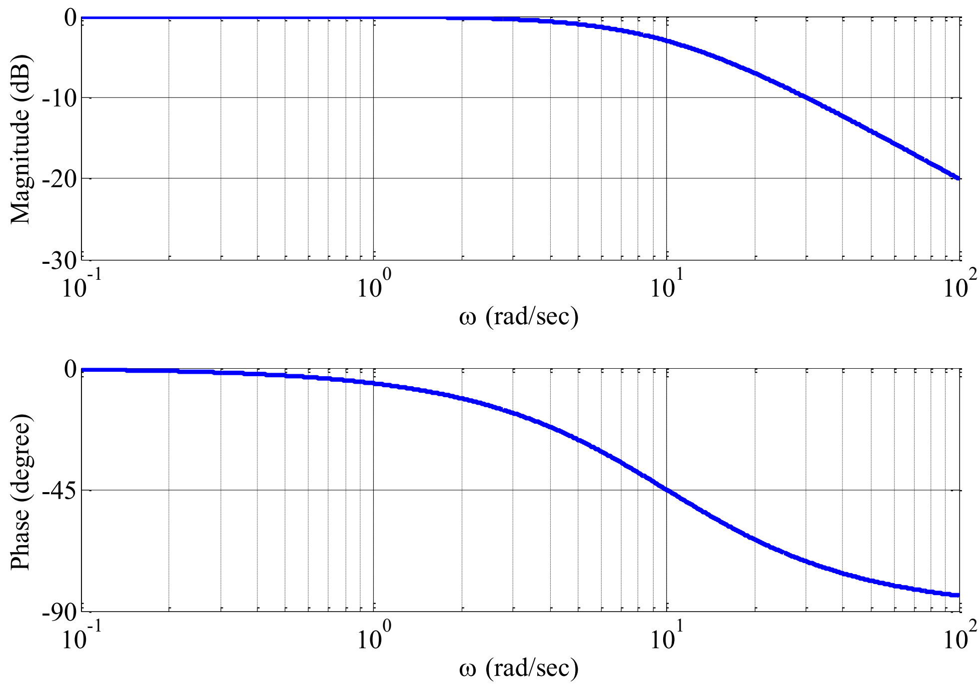
Therefore, reading the magnitude value of the frequency response at the angular velocity frequency $\omega = 2\pi(rad/\sec)$ corresponding to a frequency of 1 Hz yields 0.847, which matches the time response shown in the figure below. Additionally, reading the magnitude value of the frequency response at the angular frequency $\omega = 6\pi(rad/\sec)$ corresponding to the frequency 3 Hz yields 0.469, which matches the time response. While the phase angle value is not easily read precisely from the time response, approximate phase angle values for each frequency can also be determined using the same method.

As shown in the example above, the $|G(j\omega)|$ value of a transfer function where the denominator's degree is greater than the numerator's degree decreases as frequency increases, and $|G(j\omega)|$ converges to zero as $\omega \to \infty$. The value of $|G(j\omega)|$ acts as a gain constant that varies with frequency. Therefore, as the frequency of the input signal increases, the magnitude of the output signal gradually decreases, and at infinite frequency, the signal magnitude becomes zero.

The frequency response is typically plotted in the form of a Bode plot. In a Bode plot, the frequency axis uses a logarithmic scale, while the magnitude axis uses a decibel (dB) scale. The decibel value for magnitude is obtained by taking the common logarithm and multiplying by 20. That is, the magnitude frequency response on a Bode plot is plotted as the graph of the following function.

$$|G(j\omega)|_{dB} = 20\log_{10}|G(j\omega)|$$

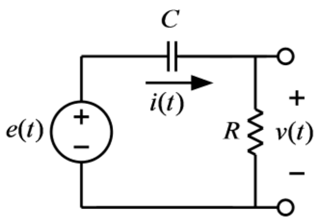
The figure below shows the Bode plot of the frequency response for the circuit above. As seen in the figure, the Bode plot consists of a magnitude Bode plot and a phase Bode plot.



(Figure) Low-Pass Bode plot

- **Frequency Response of a High-Pass RC Circuit**

We analyze the frequency response of the circuit shown below, which swaps the positions of the resistor and capacitor in the circuit considered above.

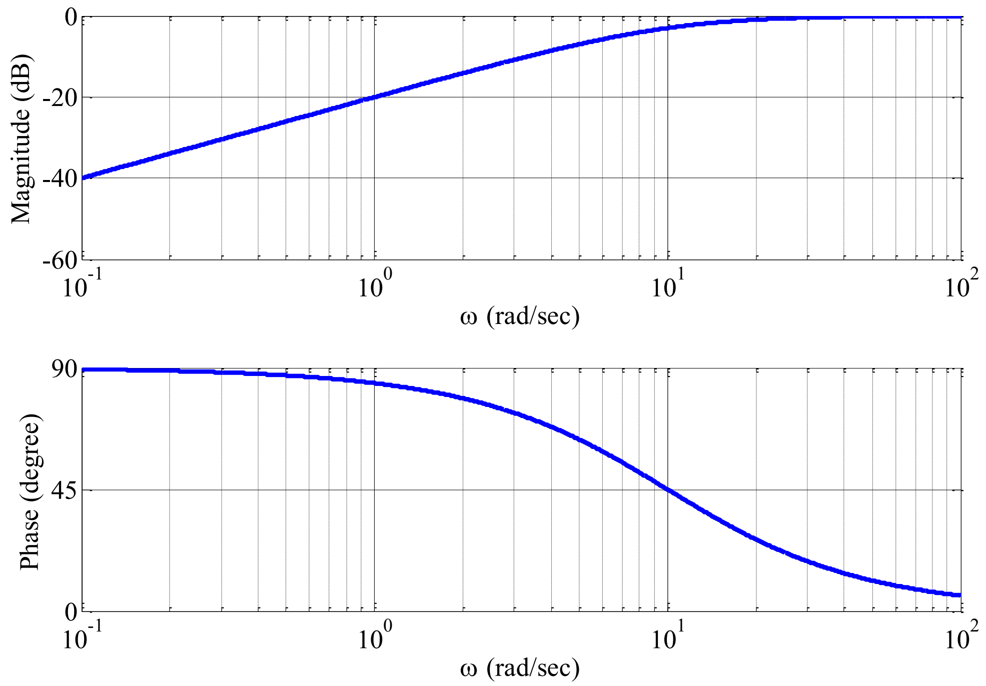


(Figure) High-Pass RC Circuit

The input to this circuit is the voltage signal $e(t)$, and the output is the voltage $v(t)$ across the resistor. Assuming the input signal is a sinusoidal wave, applying phasors and using the voltage division rule yields the following network function:

$$G(j\omega) = \frac{V(j\omega)}{E(j\omega)} = \frac{R}{\frac{1}{j\omega C} + R} = \frac{j\omega RC}{j\omega RC + 1}$$

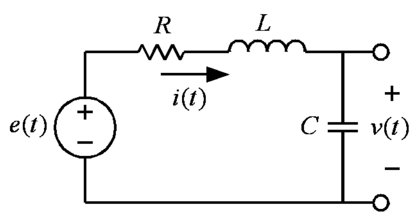
The Bode plot for this circuit is as shown in the figure below.



(Figure) High-Pass Bode plot

- **Frequency Response of a Low-Pass RLC Circuit**

To analyze the frequency response of an RLC circuit, consider the circuit shown in the figure below. The input to this circuit is the voltage signal $e(t)$, and the output is the voltage $v(t)$ across the capacitor terminals.



(Figure) Low-Pass RLC Circuit

The impedance of an inductor is $j\omega L$, and the impedance of a capacitor is $1/(j\omega C)$. Using the voltage division rule, the network function of the above circuit can be determined as follows.

$$G(j\omega)=\frac{V(j\omega)}{E(j\omega)}=\frac{\dfrac{1}{j\omega C}}{R+j\omega L+\dfrac{1}{j\omega C}}$$

The above equation can be simplified as follows.

$$G(j\omega)=\frac{V(j\omega)}{E(j\omega)}=\frac{1}{LC(j\omega)^2+RC(j\omega)+1}=\frac{1/(LC)}{(j\omega)^2+(R/L)(j\omega)+1/(LC)}$$

Meanwhile, since the standard form of a second-order transfer function is

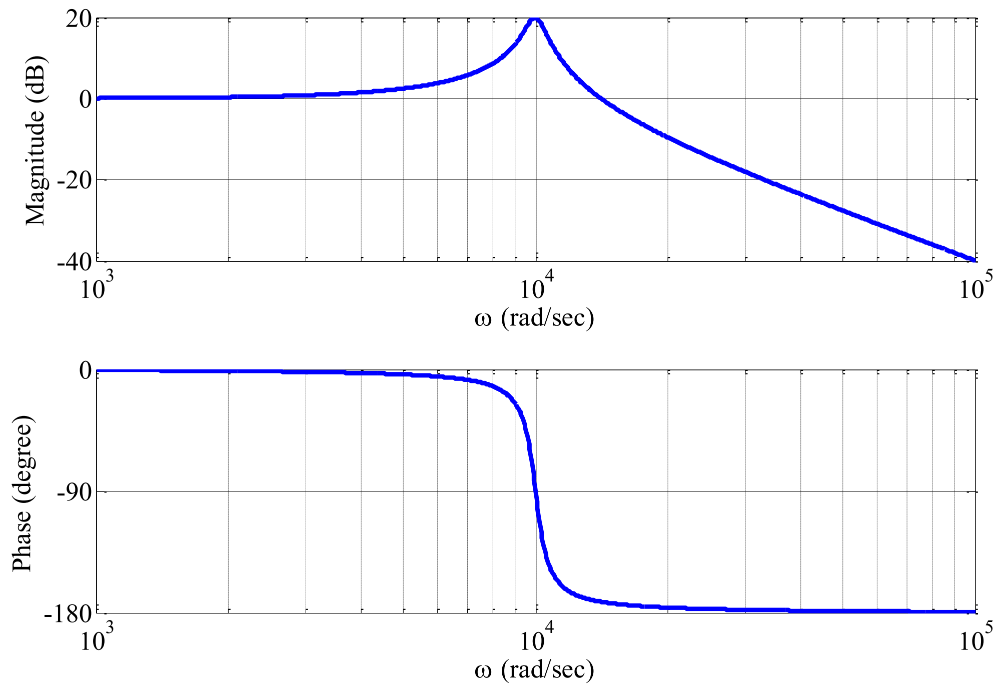
$$G(j\omega)=\frac{\omega_0^2}{(j\omega)^2+2\alpha(j\omega)+\omega_0^2},$$

the relationship between the parameters α, ω_0 of the standard form and the component values of the circuit given above is as follows.

$$\alpha=\frac{R}{2L}, \omega_0=\frac{1}{\sqrt{LC}}$$

For example, if the RLC values in the above circuit are set as follows and the frequency response Bode plot is drawn, it appears as shown in the figure below.

$$R=100\ \Omega,\ L=100\ \text{mH},\ C=0.1\ \mu\text{F}$$

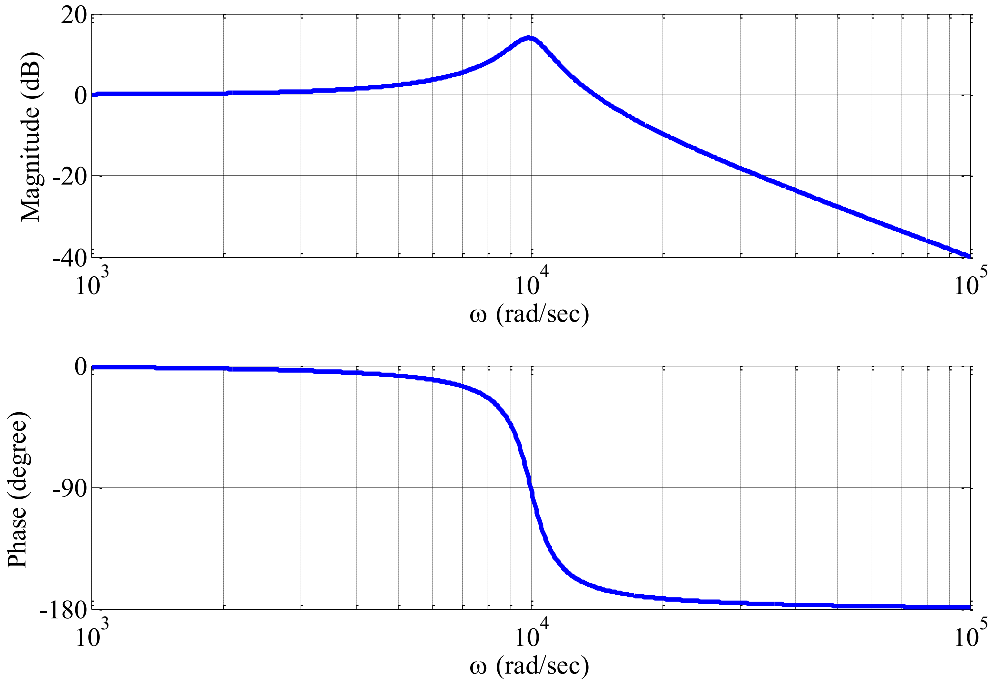


(Figure) Bode plot of the second-odeder circuit, $R=100\Omega, L=100\text{mH}, C=0.1\mu\text{F}$

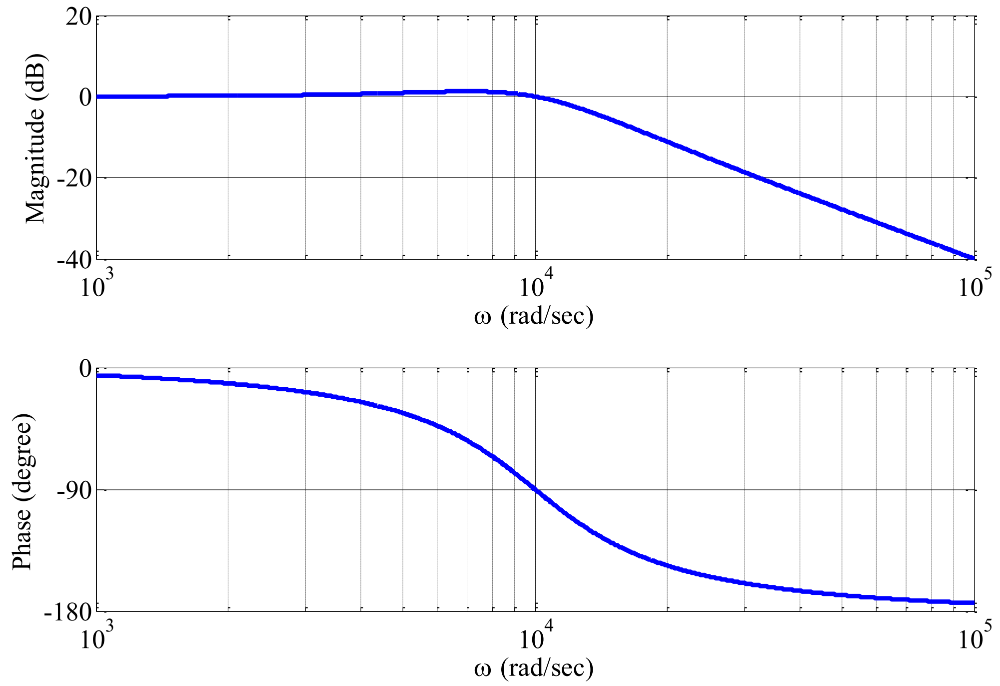
The frequency response of this circuit with a second-order transfer function differs from that of a first-order circuit in that it exhibits a peak value near the natural frequency $\omega_0 = 1/\sqrt{LC} = 10000 \text{ rad/sec}$. The magnitude of this peak value is related to the damping coefficient α. As the value of α increases, the peak value tends to decrease. To determine the frequency characteristics for different α values, the values of L and C are fixed, and the value of R is increased as follows.

$$R = 200\ \Omega,\ L = 100\ \text{mH},\ C = 0.1\ \mu\text{F}$$
$$R = 1\ \text{K}\Omega,\ L = 100\ \text{mH},\ C = 0.1\ \mu\text{F}$$

The Bode plot for the two cases above are as shown in the figure below.



(Figure) Bode plot of the second-order circuit, $R = 200\Omega, L = 100\text{mH}, C = 0.1\mu\text{F}$



(Figure) Bode plot of the second-order circuit, $R = 1\mathrm{K}\Omega, L = 100\mathrm{mH}, C = 0.1\mu\mathrm{F}$

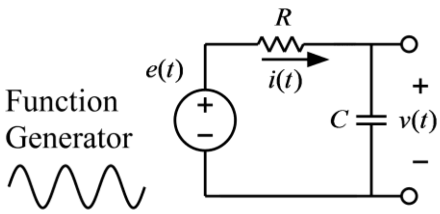
- **Instruments and Componets Used**

- Oscilloscope: 1
- Function generator: 1
- Capacitor: $0.1\ \mu\mathrm{F}$ (polyester film capacitor) x1

- Inductor: $100\ \mathrm{mH}$ x1
- Resistor: $1\ \mathrm{K}\Omega$ x1, $100\ \Omega$ x2

- **Experimental Methods and Procedures**

A. Measurement of the Frequency Response of a Low-Pass RC Circuit



(Figure) Frequency Response Measurement Circuit for a Low-Pass RC Circuit

(1) Construct the circuit shown in the figure above, connecting a function generator to the input side and an oscilloscope to the output side. The values of each component are as follows:.

$$R = 1K\Omega, C = 0.1\mu F$$

(2) Set the function type of the function generator to sine wave, and set the voltage amplitude (peak-to-peak) to 2 volts. That is, set the difference between the low voltage and high voltage of the sine wave to 2 volts.

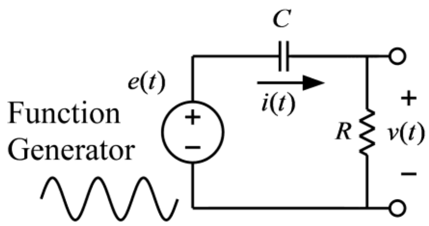
(3) Vary the frequency of the sine wave with the following values, and record the ratio of the output voltage magnitude to the input voltage magnitude.

$$\omega = 1000 \text{ rad/sec}, \ f = 159 \text{ Hz}$$
$$\omega = 4000 \text{ rad/sec}, \ f = 637 \text{ Hz}$$
$$\omega = 8000 \text{ rad/sec}, \ f = 1.27 \text{ KHz}$$
$$\omega = 10000 \text{ rad/sec}, \ f = 1.59 \text{ KHz}$$
$$\omega = 20000 \text{ rad/sec}, \ f = 3.18 \text{ KHz}$$
$$\omega = 40000 \text{ rad/sec}, \ f = 6.37 \text{ KHz}$$
$$\omega = 80000 \text{ rad/sec}, \ f = 12.7 \text{ KHz}$$
$$\omega = 100000 \text{ rad/sec}, \ f = 15.9 \text{ KHz}$$

(4) Using the values measured above, draw an approximate Bode plot

(5) Using PSpice, draw the Bode plot for the above circuit and compare it with the experimental results drawn in (4).

B. Frequency Response Measurement of a High-Pass RC Circuit



(Figure) Frequency Response Measurement Circuit for a High-Pass RC Circuit

(1) Construct the circuit shown in the figure above, connecting a function generator to the input side and an oscilloscope to the output side. The values of each component are as follows.

$$R = 1 \text{ K}\Omega, \ C = 0.1 \ \mu\text{F}$$

(2) Set the function type of the function generator to sine wave, and set the voltage amplitude (peak-to-peak) to 2 volts. That is, set the difference between the low voltage and high voltage of the sine wave to 2 volts.

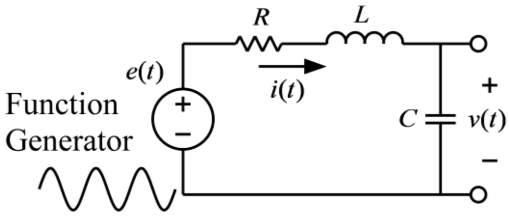
(3) Vary the frequency of the sine wave with the following values, and record the ratio of the output voltage magnitude to the input voltage magnitude.

$$\omega = 1000 \text{ rad/sec}, \ f = 159 \text{ Hz}$$
$$\omega = 4000 \text{ rad/sec}, \ f = 637 \text{ Hz}$$
$$\omega = 8000 \text{ rad/sec}, \ f = 1.27 \text{ KHz}$$
$$\omega = 10000 \text{ rad/sec}, \ f = 1.59 \text{ KHz}$$
$$\omega = 20000 \text{ rad/sec}, \ f = 3.18 \text{ KHz}$$
$$\omega = 40000 \text{ rad/sec}, \ f = 6.37 \text{ KHz}$$
$$\omega = 80000 \text{ rad/sec}, \ f = 12.7 \text{ KHz}$$
$$\omega = 100000 \text{ rad/sec}, \ f = 15.9 \text{ KHz}$$

(4) Using the values measured above, draw an approximate Bode plot.

(5) Using PSpice, draw the Bode plot for the above circuit and compare it with the experimental results drawn in (4).

C. Frequency Response Measurement of a Low-Pass RLC Circuit



(Figure) Frequency Response Measurement Circuit for a Low-Pass RLC Circuit

(1) Construct the circuit shown in the figure above, connecting a function generator to the input side and an oscilloscope to the output side. The values of each component are as follows:

$$R = 100\ \Omega,\ L = 100\ \text{mH},\ C = 0.1\ \mu\text{F}$$

(2) Set the function type of the function generator to sine wave, and set the voltage amplitude (peak-to-peak) to 2 volts. That is, set the difference between the low voltage and high voltage of the sine wave to 2 volts.

(3) Vary the frequency of the sine wave with the following values, and record the ratio of the output voltage magnitude to the input voltage magnitude.

$$\omega = 1000\ \text{rad/sec},\ f = 159\ \text{Hz}$$
$$\omega = 4000\ \text{rad/sec},\ f = 637\ \text{Hz}$$
$$\omega = 8000\ \text{rad/sec},\ f = 1.27\ \text{KHz}$$
$$\omega = 10000\ \text{rad/sec},\ f = 1.59\ \text{KHz}$$
$$\omega = 20000\ \text{rad/sec},\ f = 3.18\ \text{KHz}$$
$$\omega = 40000\ \text{rad/sec},\ f = 6.37\ \text{KHz}$$
$$\omega = 80000\ \text{rad/sec},\ f = 12.7\ \text{KHz}$$
$$\omega = 100000\ \text{rad/sec},\ f = 15.9\ \text{KHz}$$

(4) Using the values measured above, draw an approximate Bode plot.

(5) Using PSpice, draw the Bode plot for the above circuit and compare it with the experimental results drawn in (4).

(6) Repeat the above experiment by changing only the resistance value to $R = 1\ \text{K}\Omega$ in the circuit.

Note) Unlike an ideal inductor, actual inductors used in the experiments contain both inductance and coil resistance. In this experiment, this resistance value cannot be ignored. When drawing the Bode plot using PSpice, including the resistance component of the inductance reduces discrepancies with experimental results. Therefore, the resistance value of the inductor must be measured using a multimeter and added to the resistance value in the above circuit for calculation.

- **Review and Discussion of Experimental Results**

<Table> Frequency Response of a Low-Pass RC Circuit

ω (rad/sec)	f (Hz)	Amplitude of the input sinusoidal wave(V)	Amplitude of the output sinusoidal wave(V)	$\lvert G(j\omega)\rvert$	$20\log_{10}\lvert G(j\omega)\rvert$
1000	159				
4000	637				
8000	1.27K				
10000	1.59K				
20000	3.18K				
40000	6.37K				
80000	12.7K				
100000	15.9K				

<Table> Frequency Response of a High-Pass RC Circuit

ω (rad/sec)	f (Hz)	Amplitude of the input sinusoidal wave(V)	Amplitude of the output sinusoidal wave(V)	$\lvert G(j\omega)\rvert$	$20\log_{10}\lvert G(j\omega)\rvert$
1000	159				
4000	637				
8000	1.27K				
10000	1.59K				
20000	3.18K				
40000	6.37K				
80000	12.7K				
100000	15.9K				

<Table> Frequency Response of a Low-Pass RLC Circuit, $R = 100\Omega, L = 100\text{mH}, C = 0.1\mu\text{F}$

ω (rad/sec)	f (Hz)	Amplitude of the input sinusoidal wave(V)	Amplitude of the output sinusoidal wave(V)	$\lvert G(j\omega)\rvert$	$20\log_{10}\lvert G(j\omega)\rvert$
1000	159				
4000	637				
8000	1.27K				
10000	1.59K				
20000	3.18K				
40000	6.37K				
80000	12.7K				
100000	15.9K				

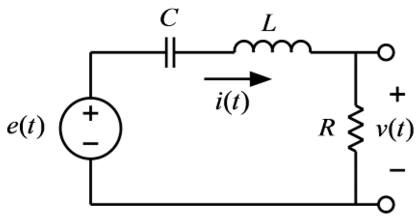
<Table> Frequency Response of a Low-Pass RLC Circuit, $R = 1\text{K}\Omega, L = 100\text{mH}, C = 0.1\mu\text{F}$

ω (rad/sec)	f (Hz)	Amplitude of the input sinusoidal wave(V)	Amplitude of the output sinusoidal wave(V)	$\lvert G(j\omega)\rvert$	$20\log_{10}\lvert G(j\omega)\rvert$
1000	159				
4000	637				
8000	1.27K				
10000	1.59K				
20000	3.18K				
40000	6.37K				
80000	12.7K				
100000	15.9K				

Lab 8. Frequency Response of an RLC Resonant Circuit

● Frequency Response of a Series RLC Resonant Circuit

To analyze the resonance phenomenon in a series RLC circuit, consider the circuit shown in the figure below. The input to this circuit is the voltage signal $e(t)$, and the output is the voltage $v(t)$ across the resistor.



(Figure) Series RLC Circuit

The differential equation for this circuit is as follows.

$$i(t) = C\frac{d}{dt}\left(e(t) - L\frac{di(t)}{dt} - Ri(t)\right)$$
$$v(t) = Ri(t)$$

Eliminating the current $i(t)$ from the above equation yields the following second-order differential equation:

$$LC\frac{d^2v(t)}{dt^2} + RC\frac{dv(t)}{dt} + v(t) = RC\frac{de(t)}{dt}$$

To obtain the transfer function, applying the Laplace transform to both sides of the above equation yields the following:

$$LCs^2V(s) + RCsV(s) + V(s) = RCsE(s)$$

Therefore, the transfer function of the above circuit is as follows.

$$G(s) = \frac{V(s)}{E(s)} = \frac{RCs}{LCs^2 + RCs + 1} = \frac{(R/L)s}{s^2 + (R/L)s + 1/(LC)}$$

On the other hand, the transfer function above can also be derived by applying the impedance function as follows. Since the impedance function of the entire series circuit is

$$Z(s) = \frac{1}{Cs} + sL + R\,,$$

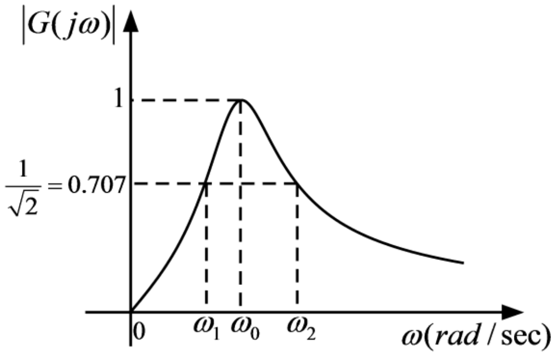
applying the voltage divider law yields the following transfer function, which matches the transfer function obtained above.

$$G(s)=\frac{V(s)}{E(s)}=\frac{R}{Z(s)}=\frac{R}{\frac{1}{Cs}+sL+R}=\frac{RCs}{LCs^2+RCs+1}=\frac{(R/L)s}{s^2+(R/L)s+1/(LC)}$$

From the transfer function above, the network function can be obtained as follows:

$$G(j\omega)=G(s)\big|_{s=j\omega}=\frac{(R/L)\,j\omega}{(j\omega)^2+(R/L)\,j\omega+1/(LC)}$$

The frequency response of the transfer function described above exhibits the characteristics of a band-pass filter, as shown in the following figure.



(Figure) Frequency response of a series RLC circuit

In the frequency response above, ω_0 is the resonant frequency and has the following relationship.

$$\omega_0=\frac{1}{\sqrt{LC}}$$

Additionally, ω_1 and ω_2 are the frequencies at which the magnitude of the frequency response reaches 0.707 times the maximum value. Using these frequencies, the frequency bandwidth (bandwidth) of the band-pass filter described above is defined as follows:

$$B=\omega_2-\omega_1$$

In the resonant circuit described above, it is convenient to express the transfer function or network function in the following standard form.

$$G(s)=\frac{(\omega_0/Q_0)s}{s^2+(\omega_0/Q_0)s+\omega_0^{\,2}}$$

$$G(j\omega)=\frac{(\omega_0/Q_0)\,j\omega}{(j\omega)^2+(\omega_0/Q_0)\,j\omega+\omega_0^{\,2}}$$

In the above equation, Q_0 is called the quality factor at the resonant frequency and is an important parameter in resonant circuits. In a series resonant circuit, Q_0 can be calculated as follows:

$$Q_0=\frac{L\omega_0}{R}=\frac{1}{R}\sqrt{\frac{L}{C}}$$

In particular, Q_0 has the following relationship with bandwidth and resonant frequency.

$$Q_0 = \frac{\omega_0}{B} = \frac{\omega_0}{\omega_2 - \omega_1}$$

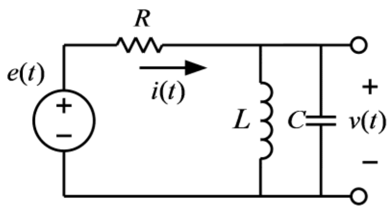
The greater the value of Q_0, the smaller the bandwidth, thereby increasing the selectivity of the band-pass filter. At the resonant frequency, the total impedance of the series RLC circuit is as follows:

$$Z(j\omega_0) = Z(s)\big|_{s=j\omega_0} = \frac{1}{jC\omega_0} + jL\omega_0 + R = -j\sqrt{\frac{L}{C}} + j\sqrt{\frac{L}{C}} + R = R$$

That is, at the resonant frequency, the impedance of the capacitor and the impedance of the inductor are equal in magnitude but opposite in direction, causing them to cancel each other out. Therefore, at the resonant frequency, it behaves as if the capacitor and inductor impedances disappear, leaving only resistance. At frequencies other than the resonant frequency, the total impedance is the sum of the resistance and the impedances of the capacitor and inductor. However, at the resonant frequency, the sum of the capacitive and inductive impedances becomes zero, leaving only the resistance. Consequently, the impedance value becomes the smallest. Therefore, the largest current flows at the resonant frequency, resulting in the phenomenon where the voltage across the resistor is the largest at the resonant frequency.

- **Frequency Response of a Parallel LC Resonant Circuit**

To analyze the resonance phenomenon in a parallel LC circuit, consider the circuit shown in the figure below. The input to this circuit is the voltage signal $e(t)$, and the output is the voltage $v(t)$ across the capacitor terminals. A parallel resonant circuit should be constructed by connecting a current source to a circuit where the three elements R, L, and C are all connected in parallel. However, doing so makes experimental measurement difficult. Therefore, we consider the circuit shown in the figure below. In fact, converting the circuit below into a Norton equivalent circuit reveals that it is equivalent to a circuit where the current source is connected in parallel with the three components R, L, and C.



(Figure) Parallel LC Circuit

The impedance function for the portion where L and C are connected in parallel in the above circuit is as follows:

$$Z(s) = \left(\frac{1}{Cs}\right) \| \left(Ls\right) = \frac{\frac{L}{C}}{\frac{1}{Cs} + Ls} = \frac{Ls}{1 + LCs^2}$$

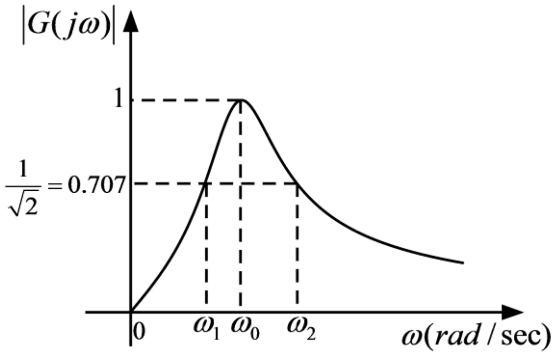
Therefore, using the voltage division law, the transfer function of the above circuit is as follows:.

$$G(s) = \frac{\frac{Ls}{1+LCs^2}}{R + \frac{Ls}{1+LCs^2}} = \frac{(L/R)s}{LCs^2 + (L/R)s + 1} = \frac{s/(RC)}{s^2 + s/(RC) + 1/(LC)}$$

From the transfer function above, the network function can be obtained as follows:

$$G(j\omega) = G(s)\big|_{s=j\omega} = \frac{j\omega/(RC)}{(j\omega)^2 + j\omega/(RC) + 1/(LC)}$$

The frequency response of the transfer function or network function described above exhibits the characteristics of a band-pass filter, as shown in the following figure.



(Figure) Frequency response of a parallel LC circuit

The transfer function or network function above can be expressed in the following standard form.

$$G(s) = \frac{(\omega_0/Q_0)s}{s^2 + (\omega_0/Q_0)s + \omega_0^2}$$

$$G(j\omega) = \frac{(\omega_0/Q_0)j\omega}{(j\omega)^2 + (\omega_0/Q_0)j\omega + \omega_0^2}$$

From the above standard form, the resonant frequency and the value of Q_0 can be calculated as follows:

$$\omega_0 = \frac{1}{\sqrt{LC}}, Q_0 = RC\omega_0 = R\sqrt{\frac{C}{L}}$$

Calculating the value of $Z(s)$ at the resonant frequency yields the following result.

$$Z(j\omega_0) = Z(s)\big|_{s=j\omega_0} = \frac{jL\omega_0}{1 - LC{\omega_0}^2} = \infty$$

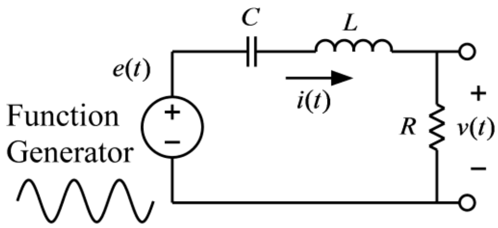
That is, at the resonant frequency, the impedance of the section where L and C are connected in parallel becomes infinite, preventing any current flow. Consequently, the magnitudes of the input voltage and output voltage become equal. At the resonant frequency, the currents through the parallel-connected capacitor and inductor have equal magnitude but opposite phase, causing them to cancel each other out and preventing current flow.

Instruments and Components Used

- Oscilloscope: 1
- Function generator: 1
- Capacitor: $0.1\ \mu\mathrm{F}$ x2
- Inductor: $100\ \mathrm{mH}$ x2
- Resistor: $1\ \mathrm{K\Omega}$ x1, $100\ \Omega$ x1

Experimental Methods and Procedures

A. Frequency Response Measurement of a Series RLC Resonant Circuit



(Figure) Frequency response measurement circuit for a series RLC resonant circuit

(1) Construct the circuit shown in the figure above, connecting a function generator to the input side and an oscilloscope to the output side. The values of each component are as follows.

$$R = 100\Omega, L = 100mH, C = 0.1\mu F$$

(2) Set the function type of the function generator to sine wave, and set the voltage amplitude (peak-to-peak) to 2 volts. That is, set the difference between the low voltage and high voltage of the sine wave to 2 volts.

(3) Measure the frequency and voltage magnitude at which the output voltage is maximized while varying the frequency of the sinusoidal wave. Multiplying this frequency (Hz) value by 2π yields the measured ω_0 value. Compare this measured value with the calculated value. However, since the resistance value of the inductor is not negligible, it must be included in the calculation to reduce the error compared to the experimental value. Furthermore, while theoretically the output voltage at the resonant frequency should equal the input voltage, actual measurements show it is significantly lower. This occurs because the inductor's resistance cannot be ignored; the voltage drop caused by this resistance is the reason.

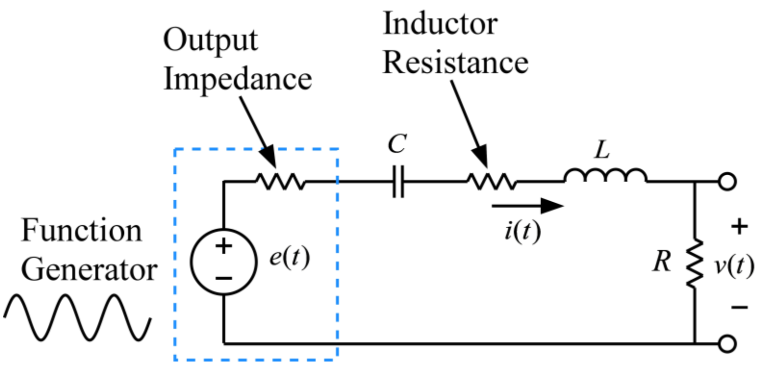
(4) Reduce the frequency of the sine wave to a value smaller than the resonant frequency, and measure the frequency at which the voltage becomes 0.707 times the voltage value at the resonant frequency. Multiplying this frequency (Hz) value by 2π yields the measured ω_1 value. Additionally, increase the sine wave frequency to a value higher than the resonant frequency and

measure the frequency at which the voltage becomes 0.707 times the voltage value at the resonant frequency. Multiplying this frequency (Hz) value by 2π yields the measured ω_2 value. Using the measured frequency values obtained in this manner, the quality factor can be measured using the following equation.

$$Q_0 = \frac{\omega_0}{\omega_2 - \omega_1}$$

Compare the measured Q_0 value with the calculated value.

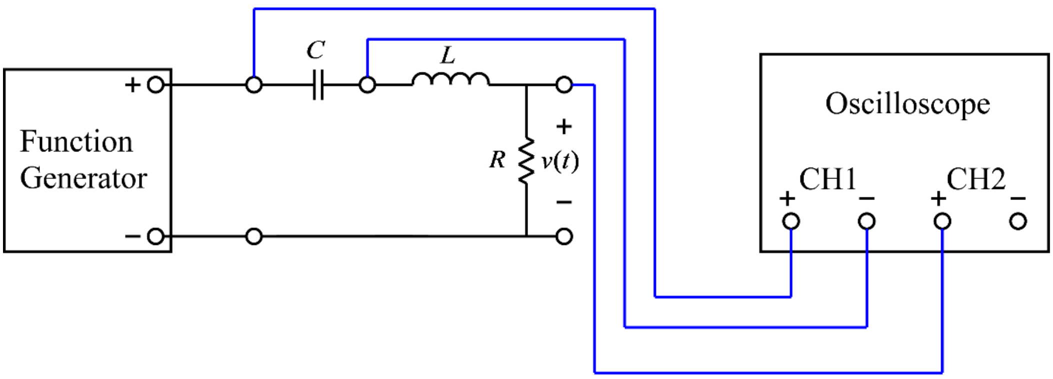
(5) Considerations during the experiment: When comparing calculated values with measured values using the circuit for this experiment, significant errors can be observed. There are two causes of error. One, as mentioned in the previous experiment, is the error due to the internal resistance of the inductor. If the internal resistance value of the inductor used in this experiment is not taken into account, it becomes a source of error. Another source of error is the output impedance of the function generator. The output impedance of the function generator is approximately 50 ohms, which is not insignificant compared to the 100 ohm output resistance value of this resonant circuit. After setting the sine wave amplitude of the function generator to 1 volt (peak-to-peak voltage is 2 volts), measuring the sine wave amplitude output from the function generator's output terminals with an oscilloscope reveals a measured value smaller than 1 volt due to the output impedance. For this reason, at the resonant frequency, the voltage across the output resistor is measured to be significantly smaller than the sine wave generated by the function generator. However, despite this reduced output voltage amplitude, the method for measuring the bandwidth frequency remains the same. Regardless of the voltage magnitude, the bandwidth is measured by determining the frequency corresponding to 0.707 times (-3dB) of the maximum voltage across the output resistor.



(Figure) Circuit considering the resistance of the inductor and the output impedance of the function generator

(6)(This experiment may not be possible depending on the type of function generator. Therefore, this experiment will not be performed.) Connect the oscilloscope as shown in the figure below.

Since the ground terminals of the oscilloscope's Channel 1 and Channel 2 are connected to each other, connect only the ground of one channel. After connecting as described, display both Channel 1 and Channel 2 simultaneously on the screen. Adjust the voltage scale and ground position for both channels to match. This allows observation of changes in the magnitude of the capacitor voltage and inductor voltage as the input frequency varies. Since the voltages across the capacitor and inductor are out of phase with each other, connecting the oscilloscope as shown below allows measurement of the two voltages with reversed polarities. This enables observation of the two voltages appearing in phase on the oscilloscope screen. At the resonant frequency, the magnitudes of the capacitor voltage and inductor voltage are equal. Therefore, by varying the function generator's frequency, you can find the resonant frequency where the magnitudes of the capacitor voltage and inductor voltage precisely match. Verify whether the resonant frequency measured in this manner corresponds to the frequency where the output voltage magnitude is greatest.

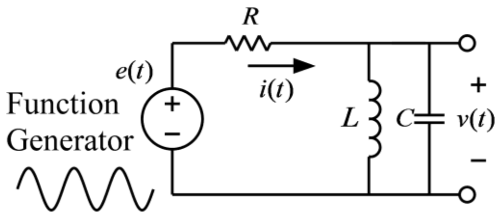


(Figure) Circuit for Simultaneously Measuring Capacitor and Inductor Voltages

(7) Repeat the above experiment using the component values below. When two capacitors are connected in series, their combined value is halved. When two inductors are connected in parallel, their combined value is halved. Therefore, apply the values below using the given components.

$$R = 1K\Omega, L = 100mH, C = 0.1\mu F$$
$$R = 100\Omega, L = 50mH, C = 0.05\mu F$$
$$R = 1K\Omega, L = 50mH, C = 0.05\mu F$$

B. Frequency Response Measurement of a Parallel LC Resonant Circuit



(Figure) Frequency response measurement circuit for a parallel LC resonant circuit

(1) Construct the circuit shown in the figure above, connecting a function generator to the input side and an oscilloscope to the output side. The values of each component are as follows.

$$R = 1K\Omega, L = 100mH, C = 0.1\mu F$$

(2) Set the function type of the function generator to sine wave, and set the voltage amplitude (peak-to-peak) to 2 volts. That is, set the difference between the low voltage and high voltage of the sine wave to 2 volts.

(3) Measure the frequency and voltage magnitude at which the output voltage is maximized while varying the frequency of the sinusoidal wave. Multiplying this frequency (Hz) value by 2π yields the measured ω_0 value. Compare this measured value with the calculated value. However, since the resistance value of the inductor is not negligible, it must be included in the calculation to reduce the error compared to the experimental value. Furthermore, while theoretically the output voltage at the resonant frequency should equal the input voltage, actual measurements show it is significantly lower. This occurs because the inductor's resistance cannot be ignored; the voltage drop caused by this resistance is the reason.

(4) Reduce the frequency of the sine wave to a value smaller than the resonant frequency, and measure the frequency at which the voltage becomes 0.707 times the voltage value at the resonant frequency. Multiplying this frequency (Hz) value by 2π yields the measured ω_1 value. Additionally, increase the sine wave frequency to a value higher than the resonant frequency and measure the frequency at which the voltage becomes 0.707 times the voltage value at the resonant frequency. Multiplying this frequency (Hz) value by 2π yields the measured ω_2 value. Using the measured frequency values obtained in this manner, the quality factor can be measured using the following equation.

$$Q_0 = \frac{\omega_0}{\omega_2 - \omega_1}$$

Compare the measured Q_0 value with the calculated value.

(5) Repeat the above experiment using the component values below. When two capacitors are connected in series, their combined value is halved. When two inductors are connected in parallel, their combined value is halved. Therefore, apply the values below using the given components.

$$R = 1K\Omega, L = 50mH, C = 0.05\mu F$$

● Review and Discussion of Experimental Results

<Table> Frequency Response of a Series RLC Resonant Circuit

R	L	C	Calculated value		measured value				
			ω_0	Q_0	ω_0	Voltage magnitude at the resonant frequency	ω_1	ω_2	Q_0
100Ω	$100mH$	$0.1\mu F$							
$1K\Omega$	$100mH$	$0.1\mu F$							
100Ω	$50mH$	$0.05\mu F$							
$1K\Omega$	$50mH$	$0.05\mu F$							

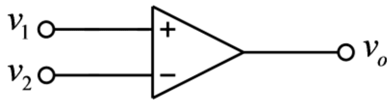
<Table> Frequency Response of Parallel RLC Resonant Circuit

R	L	C	Calculated value		measured value				
			ω_0	Q_0	ω_0	Voltage magnitude at the resonant frequency	ω_1	ω_2	Q_0
$1K\Omega$	$100mH$	$0.1\mu F$							
$1K\Omega$	$50mH$	$0.05\mu F$							

Lab 9. Basic Characteristics of an Operational Amplifier

Ideal Operational Amplifier

An OP amp, also known as an operational amplifier, is a differential linear amplifier composed of multiple transistors. OP amps can be used in circuits performing mathematical operations such as addition, integration, and differentiation, and are widely used in video and audio amplifiers, oscillators, and other applications. The notation for an OP amp is as shown in the following figure.



(Figure) Operational amplifier circuit symbol

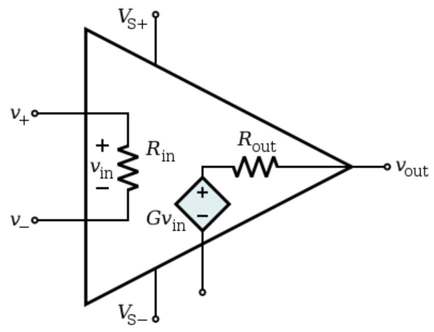
As shown in the figure above, the input terminals consist of a positive terminal and a negative terminal. At the output terminal, the voltage amplified from the difference between the two input terminals is output. That is, the output voltage is given by the following equation, where G is the amplification gain.

$$v_o = G(v_1 - v_2)$$

The gain value G of an ideal OP amp is infinite. Therefore, if the voltage at the positive terminal is even slightly greater than the voltage at the negative terminal, the output voltage is positive infinity ($+\infty$). Furthermore, if the voltage at the positive terminal is even slightly less than the voltage at the negative terminal, the output voltage is negative infinity ($-\infty$). Another characteristic of an ideal OP amp is that its input resistance is infinite. That is, even if another circuit is connected to the input terminals, the current flowing through the input terminals is always zero. Furthermore, the output resistance of an ideal OP amp is zero. Consequently, even if another circuit is connected to the output circuit and current flows through it, no voltage drop occurs due to this.

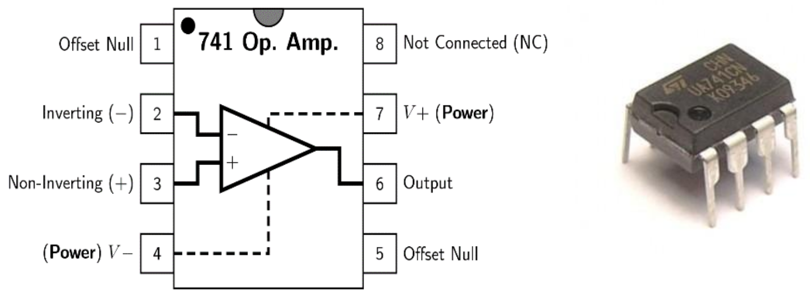
Real OP Amp

While the characteristics of actual OP amps used in circuits differ from those of ideal OP amps, actual OP amps exhibit characteristics similar to ideal OP amps. Therefore, when analyzing circuits composed of OP amps, it is common to assume the actual OP amp is an ideal OP amp for analysis. An ideal OP amp has infinite voltage gain, but the voltage gain of a real OP amp, while quite large, is finite, not infinite. Furthermore, its input resistance is not infinite but has a very large resistance value. Consequently, while current flows through the input terminals, it is often so small that it can be neglected. The output resistance value is also not zero but has a small resistance value, which can often be ignored. The diagram below shows the equivalent circuit of an OP amp. In an ideal OP amp, $G=\infty, R_{in}=\infty, R_{out}=0$, but in a real OP amp, G and R_{in} have very large values, R_{out} has a very small value.



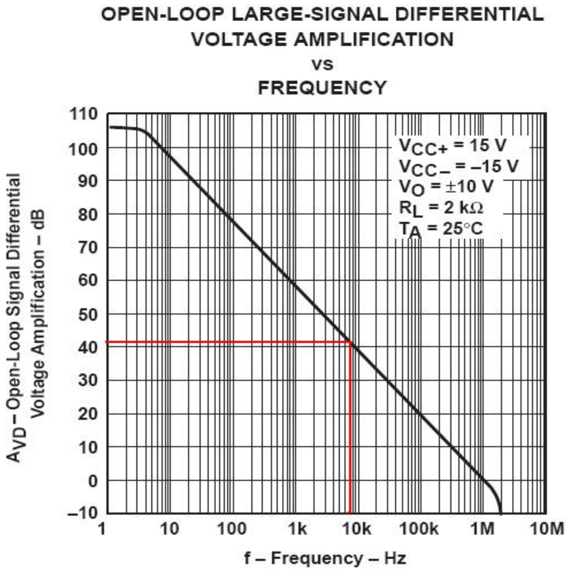
(Figure) OP amp equivalent circuit

To use an OP amp, you must connect a DC power supply to provide DC voltage. The voltage supplied to the OP amp is typically a positive voltage and a negative voltage, each around 10V to 15V in magnitude. When drawing OP amp circuits, the power supply connections are often omitted for simplicity, but in reality, the power supply must be connected for operation. The following diagram shows the 741 OP amp IC, the most commonly used OP amp.



(Figure) Pin configuration of an actual operational amplifier

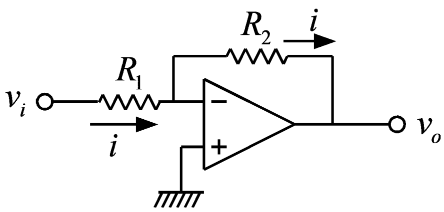
The voltage gain value in the equivalent circuit above is G While it was stated that the actual OP amp has a finite large value rather than infinity, this value is not simply a constant; it has a gain value that varies with frequency, as shown in the figure below. Such a graph is called the frequency response of the OP amp, and the figure below is the frequency response graph of the 741 OP amp.



(Figure) Frequency response of an operational amplifier

As seen in the frequency response above, the gain at low frequencies exceeds 100dB, which is approximately 10^5 This is a fairly large value exceeding [specific value]. However, as the frequency increases, the gain value decreases. For a frequency of 1 MHz, the gain value can be seen to decrease to approximately 0 dB (a gain of 0 dB corresponds to a gain magnitude of 1). Therefore, when designing an OP amp circuit, it is necessary to consider this frequency characteristic in the design.

Consider the following inverting amplifier circuit, the most basic circuit using an op-amp.



(Figure) Inverting amplifier

In the circuit above, the R_2 resistor performs the following feedback action. Specifically, when the output voltage increases, it increases the voltage at the input terminal to decrease the output voltage. Conversely, when the output voltage decreases, it decreases the voltage at the input terminal to increase the output voltage. This action ensures the output voltage maintains a finite voltage value. Since the voltage gain of an ideal OP amp is infinite, the voltage between the input terminals must remain at 0 to achieve a finite output voltage value. For this reason, the – terminal in the above circuit maintains the same ground voltage as the + terminal, i.e., 0V. Furthermore, since the current flowing into the input terminals is 0, the current flowing through the resistor is as follows.

$$i = \frac{v_i}{R_1}$$

Using this relationship, the following equation for the output voltage can be derived.

$$v_o = -R_2 i = -\frac{R_2}{R_1} v_i$$

The gain between the input and output of the above circuit is equal to (R_2 / R_1). However, in actual circuits, the gain value is not the same at all frequencies. At low frequencies, it has a gain of (R_2 / R_1), but above a certain frequency, the gain becomes less than (R_2 / R_1). That is, the frequency range over which the above circuit can be used as a voltage amplifier with a constant gain value is limited.

● Instruments and Components Used

- Oscilloscope: 1
- Function generator: 1
- DC Power Supply: 1
- OP amp: 741 x1
- Resistor: $1K\Omega$ x1, $10K\Omega$ x1, $100K\Omega$ x1

● Experimental Methods and Procedures

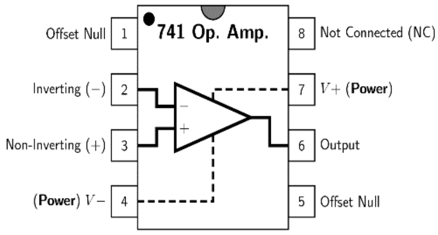
A. Power supply connection for the OP amp

(Figure) DC Power Supply Connection

(1) With nothing connected to the DC power supply shown in the figure above, set both CH1 and CH2 voltages to approximately 15V. An error of about 0.1 to 0.2V is acceptable at this time.

(2) Next, begin connecting the power supply while it is turned off. First, connect the – terminal (black terminal) of the power supply's CH1 channel and the + terminal (red terminal) of the CH2 channel with a black wire, as shown in the figure above. Doing this provides +15V at the red terminal of CH1 and -15V at the black terminal of CH2, while the terminals connected by the black wire become ground.

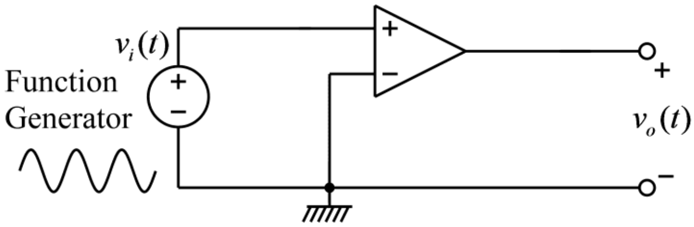
(3) Connect the red terminal of CH1 in the above photo to pin 7 (V+) of the 741 OP amp, and connect the black terminal of CH2 to pin 4 (V-) of the 741 OP amp. The terminal connected by the black wire in the above photo will be connected to the circuit's ground.



(Figure) Pin configuration of an operational amplifier

B. Open Circuit Characteristics

(1) In principle, an operational amplifier is used to form a feedback circuit. However, to examine the open-circuit characteristics of the op-amp, a circuit without feedback is configured as shown in the figure below. In this case, the ground of the circuit below is connected to the terminal connected by the black wire from the power supply above.



(Figure) OP amp open-circuit characteristic measurement circuit

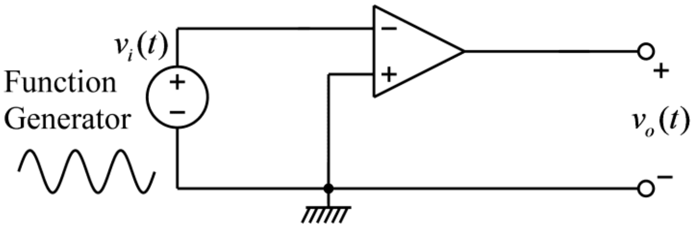
(2) Connect a function generator to the input side and an oscilloscope to the output side.

(3) Set the function type of the function generator to sine wave, and set the voltage amplitude (peak-to-peak) to 0.2V (200mV). That is, set the difference between the low voltage and high voltage of the sine wave to 0.2V. Set the frequency to 100Hz.

(3) Connect the function generator output to channel 1 of the oscilloscope and the output of the above OP amp circuit to channel 2, then compare the signals from both channels.

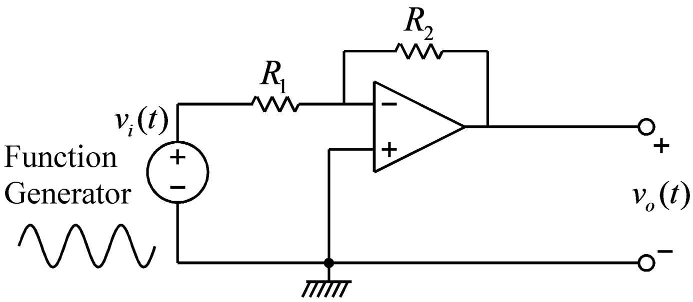
(4) Observe the output signal waveform and voltage magnitudes, and explain in the report why such outputs occur.

(5) Connect the function generator to the – terminal input instead of the + terminal, modify the circuit as shown below, and repeat the above experiment. Describe how the output signal differs compared to the previous experiment and explain the reason for this difference in the report.



(Figure) OP amp open-circuit characteristic measurement circuit

C. Measuring the frequency response of an inverting amplifier



(Figure) Inverting Amplifier Frequency Response Measurement Circuit

(1) Construct the circuit shown in the figure above, connecting a function generator to the input side and an oscilloscope to the output side. The values of each component are as follows.

$$R_1 = 1K\Omega, R_2 = 10K\Omega$$

(2) Set the function type of the function generator to sine wave, and set the voltage amplitude (peak-to-peak) to 0.2V (200mV). That is, set the difference between the low voltage and high voltage of the sine wave to 0.2V.

(3) Vary the frequency of the sine wave with the following values, and record the ratio of the output voltage magnitude to the input voltage magnitude.

$$f = 100Hz,\ \omega = 2\pi \times 100 \text{ rad/sec}$$
$$f = 1KHz,\ \omega = 2\pi \times 1000 \text{ rad/sec}$$
$$f = 10KHz,\ \omega = 2\pi \times 10,000 \text{ rad/sec}$$
$$f = 100KHz,\ \omega = 2\pi \times 100,000 \text{ rad/sec}$$
$$f = 1MHz,\ \omega = 2\pi \times 1000,000 \text{ rad/sec}$$

(4) Using the values measured above, draw an approximate Bode plot.

(5) At low frequencies, the output voltage magnitude remains constant, but beyond a certain frequency, a decrease in output voltage magnitude can be observed. As the frequency increases, the output value decreases. Find the frequency at which the output value decreases to approximately 70%. That is, find the frequency at which the output value decreases to approximately 70% of the value when a low-frequency input (e.g., 100Hz) is applied. This frequency is the approximate -3dB frequency, which is the frequency at which the DC gain

decreases by 3dB.

$$20\log\left(\frac{1}{\sqrt{2}}\right)=20\log(0.707)\approx-3\,\mathrm{dB}$$

(6) Change $R_2 = 1\ \mathrm{K\Omega}$ in the above inverting amplifier circuit and repeat the experiment.

(7) Change $R_2 = 100\ \mathrm{K\Omega}$ in the above inverting amplifier circuit and repeat the experiment.

(8) Describe the conclusions that can be drawn from the above experiments in the report.

- **Review and Discussion of Experimental Results**

<Table> Frequency Response of Inverting Amplifier $R_2 = 10\ \mathrm{K\Omega}$

ω (rad/sec)	f (Hz)	Amplitude of the input sinusoidal wave(V)	Amplitude of the output sinusoidal wave((V)	$\lvert G(j\omega)\rvert$	$20\log_{10}\lvert G(j\omega)\rvert$
	100				
	1K				
	10K				
	100K				
	1M				

-3dB Frequency: ____________Hz

<Table> Frequency Response of Inverting Amplifier $R_2 = 1\ \mathrm{K\Omega}$

ω (rad/sec)	f (Hz)	Amplitude of the input sinusoidal wave(V)	Amplitude of the output sinusoidal wave (V)	$\lvert G(j\omega)\rvert$	$20\log_{10}\lvert G(j\omega)\rvert$
	100				
	1K				
	10K				
	100K				
	1M				

-3dB Frequency: ____________Hz

<Table> Frequency Response of Inverting Amplifier $R_2 = 100\ \mathrm{K\Omega}$

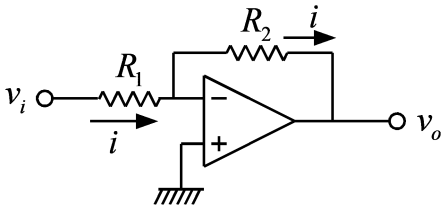
ω (rad/sec)	f (Hz)	Amplitude of the input sinusoidal wave(V)	Amplitude of the output sinusoidal wave (V)	$\lvert G(j\omega)\rvert$	$20\log_{10}\lvert G(j\omega)\rvert$
	100				
	1K				
	10K				
	100K				
	1M				

-3dB Frequency: ___________Hz

Lab 10. Basic OP Amp Circuits

● Inverting Amplifier

Consider the following inverting amplifier circuit using an operational amplifier..



(Figure) Inverting amplifier

In the circuit above, the - input terminal of the OP amp maintains the same ground voltage as the + input terminal, that is, 0V. Furthermore, since the current flowing into the OP amp input terminals is zero, the current flowing through the resistor is as follows:

$$i = \frac{v_i}{R_1}$$

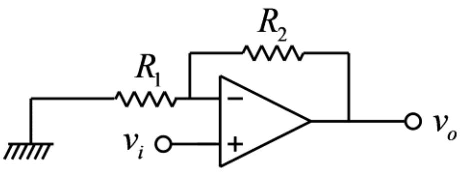
Using this relationship, the following equation for the output voltage can be obtained.

$$v_o = -R_2 i = -\frac{R_2}{R_1} v_i$$

Therefore, the voltage gain of the inverting amplifier is $-R_2 / R_1$.

● Non-inverting Amplifier

By swapping the input terminal and the ground terminal in an inverting amplifier, you can configure a non-inverting amplifier as shown in the figure below..



(Figure) Non-inverting amplifier

In the circuit above, the - input terminal of the OP amp maintains the same voltage as the + input terminal. Furthermore, since the current flowing into the OP amp's input terminals is zero, the following relationship holds.

$$v_i = \frac{R_1}{R_1 + R_2} v_o$$

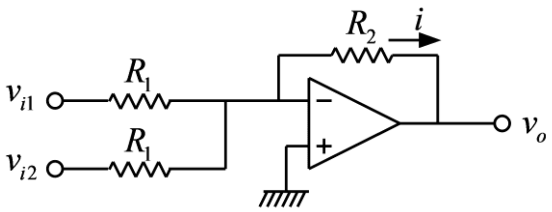
Using this relationship, the following equation for the output voltage can be obtained.

$$v_o = \left(1 + \frac{R_2}{R_1}\right) v_i$$

Therefore, the voltage gain of a non-inverting amplifier is $1 + (R_2 / R_1)$.

- **Summing Amplifier**

By applying an inverting amplifier, a summing amplifier can be constructed that combines voltages and outputs the result, as shown in the following figure.



(Figure) Summing amplifier

In the circuit above, the - input terminal of the OP amp maintains the same ground voltage as the + input terminal, i.e., 0V. Furthermore, since the current flowing into the OP amp input terminals is zero, the current flowing through the feedback resistor R_2 is as follows.

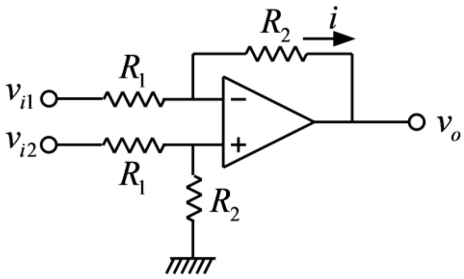
$$i = \frac{v_{i1}}{R_1} + \frac{v_{i2}}{R_1}$$

Using this relationship, the following equation for the output voltage can be obtained.

$$v_o = -R_2 i = -\frac{R_2}{R_1}\left(v_{i1} + v_{i2}\right)$$

- **Difference Amplifier**

When common noise or a DC voltage exists on two signal lines, a difference amplifier like the one below can be used to remove this common noise signal or common DC voltage and amplify only the differential voltage between the two signal lines.



(Figure) Difference amplifier

In the circuit above, no current flows into the input terminals of the OP amp. Therefore, the voltage value at the + input terminal of the OP amp is as follows.

$$\frac{R_2}{R_1+R_2}v_{i2}$$

Furthermore, since the negative input terminal and positive input terminal of the OP amp maintain the same voltage, and the current flowing into the OP amp input terminals is zero, the current flowing through the feedback resistor R_2, i, is as follows:

$$i=\frac{1}{R_1}\left(v_{i1}-\frac{R_2}{R_1+R_2}v_{i2}\right)$$

Using the above current value, the output voltage can be calculated as follows.

$$v_o=\frac{R_2}{R_1+R_2}v_{i2}-R_2 i=\frac{R_2}{R_1+R_2}v_{i2}-\frac{R_2}{R_1}\left(v_{i1}-\frac{R_2}{R_1+R_2}v_{i2}\right)=\frac{R_2}{R_1}\left(v_{i2}-v_{i1}\right)$$

Common noise or DC voltage present at both inputs of this circuit is removed, and a voltage output is generated that is equal to the difference voltage between the two inputs multiplied by the gain value (R_2/R_1).

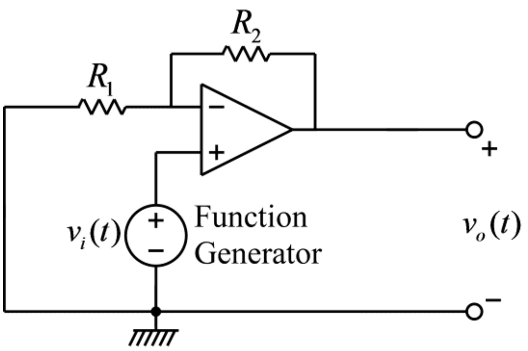
Instruments and Components Used

- Oscilloscope: 1
- Function generator: 1
- DC power supply(dual): 1
- OP amp: 741 x1
- Resistor: $1\ \mathrm{K\Omega}$ x3, $10\ \mathrm{K\Omega}$ x5, $100\ \mathrm{K\Omega}$ x3
- Potentiometer: $10\ \mathrm{K\Omega}$ x1

Experimental Methods and Procedures

Precautions when configuring OP amp circuits: In OP amp circuits, the output signal may oscillate. In such cases, connecting a capacitor (0.1 uF) between the power supply terminal (+15V connection terminal) and ground of the OP amp IC usually eliminates the oscillation.

A. Non-inverting amplifier



(Figure) Non-inverting amplifier circuit

(1) Construct the circuit shown in the figure above, connecting a function generator to the input side and an oscilloscope to the output side. The values of each component are as follows:

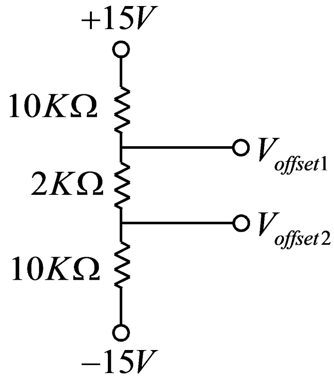
$$R_1 = 1\ \mathrm{K\Omega},\ R_2 = 2\ \mathrm{K\Omega}$$

(2) Set the function type of the function generator to sine wave, and set the voltage amplitude (peak-to-peak) to 0.2V (200mV). That is, set the difference between the low voltage and high voltage of the sine wave to 0.2V. Set the frequency to 1 kHz.

(3) Calculate the voltage gain of the non-inverting amplifier by measuring the actual resistance value, then measure the voltage gain of the actual circuit for comparison.

B. Summing Amplifier

(1) First, construct the following voltage divider circuit and measure $V_{offset2}$ and $V_{offset1}$.

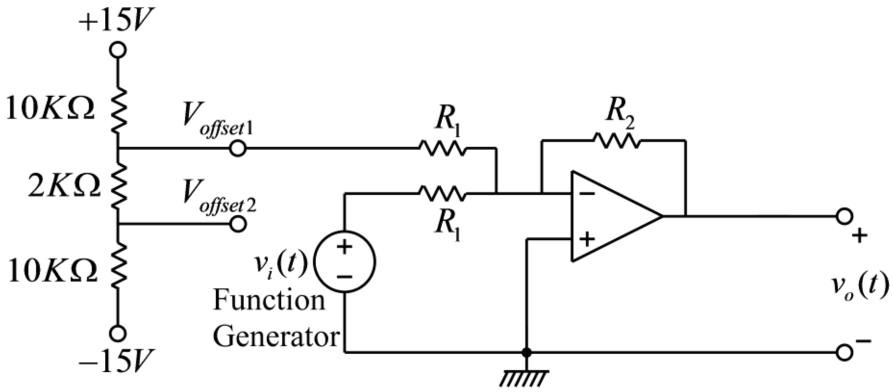


(Figure) Offset Voltage Generation Circuit

(2) Construct the circuit shown in the figure below, connecting a function generator to the input side and an oscilloscope to the output side. The values of each component are as follows.

$$R_1 = 10\ \text{K}\Omega,\ R_2 = 10\ \text{K}\Omega$$

Measure $V_{offset1}$ in the circuit below and describe in the report why this measured value differs from the $V_{offset1}$ voltage measured in (1).



(Figure) Summing Amplifier Experiment Circuit

(3) Set the function type of the function generator to sine wave and configure the voltage amplitude (peak-to-peak) to 2V. That is, set the difference between the low voltage and high voltage of the sine wave to 2V. Set the frequency to 1 kHz.

(4) Measure the maximum and minimum values of the output sine wave voltage to determine the amplitude of the output sine wave and the offset voltage. The amplitude of the sine wave is half the difference between the maximum and minimum values, and the offset voltage is the midpoint

voltage between the maximum and minimum values.

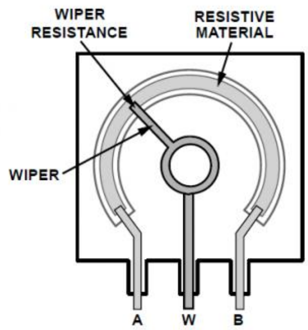
(5) In the circuit above, replace the wire connected to $V_{offset1}$ with the wire connected to $V_{offset2}$, then repeat the experiment in (4).

(6) After changing the resistance value as follows, repeat steps (2) to (5) of the experiment.

$$R_1 = 100\ \mathrm{K\Omega},\ R_2 = 100\ \mathrm{K\Omega}$$

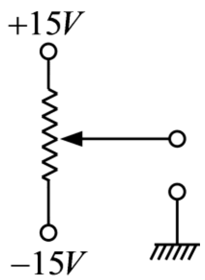
C. Difference Amplifier

(1) A potentiometer, also called a variable resistor, changes the resistance between the center terminal and either the right terminal or the left terminal as its shaft is rotated. The figure below shows the shape and internal structure of a potentiometer.



(Figure) Shape and Structure of a Potentiometer

Configure the circuit shown in the figure below using a potentiometer. Connect +15V and -15V to the two terminals of the potentiometer, respectively. Then, using a multimeter or oscilloscope, measure the voltage between the center terminal of the potentiometer and the ground terminal of the power supply. Observe the voltage change as you rotate the shaft by hand, noting the maximum and minimum voltage values.

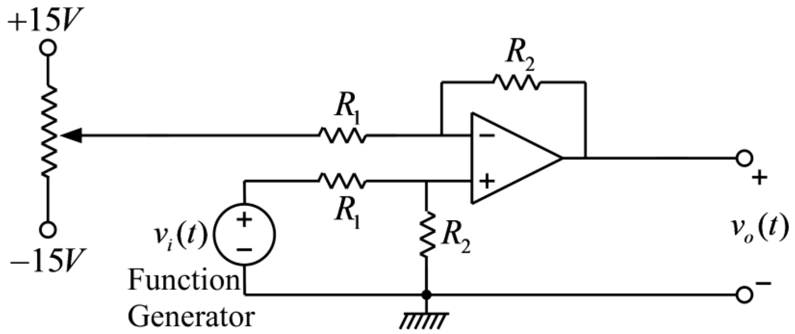


(Figure) Pententiometer Circuit

(2) Construct the circuit shown in the figure below using a potentiometer, connecting a function generator to the input side and an oscilloscope to the output side. The values of each component

are as follows.

$$R_1 = 10\ \text{K}\Omega,\ R_2 = 10\ \text{K}\Omega$$



(Figure) Difference Amplifier Experiment Circuit

(3) Set the function type of the function generator to sine wave and set the voltage amplitude (peak-to-peak) to 2V. That is, ensure the difference between the low voltage and high voltage of the sine wave is 2V. Additionally, adjust the function generator's offset so that the offset voltage of the sine wave is 1V. With these settings, the sine wave's minimum value should be 0V and the maximum value 2V; verify this using an oscilloscope. Set the frequency to 1 kHz.

(4) Measure the output voltage of the constructed OP amp circuit using an oscilloscope. Observe how the offset voltage of the output voltage changes as you rotate the potentiometer shaft. Adjust the potentiometer until the output voltage offset reaches 0V, and then measure the potentiometer voltage at this point using a multimeter. The potentiometer voltage is the voltage between the potentiometer's center terminal and the power supply ground.

Review and Discussion of Experimental Results

A. Non-inverting Amplifier

Measured value of R_1	Measured value of R_2	Calculated value of voltage gain $1+(R_2/R_1)$	Measured value of voltage gain

B. Summing Amplifier

Experiment Step	$V_{offset1}$	$V_{offset2}$
(1) Before connecting the summing amplifier		
(2) After connecting the summing amplifier		

Step Number	Maximum voltage	Minimum voltage	Amplitude of sinusoidal wave	Offset of sinusoidal wave
(4)				
(5)				

Experiment step (6)

Experiment Step	$V_{offset1}$	$V_{offset2}$
(1) Before connecting the summing amplifier		
(2) After connecting the summing amplifier		

Experiment step (6)

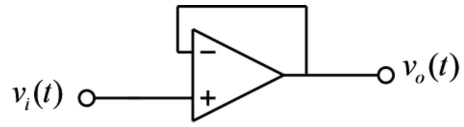
Step number	Maximum voltage	Minimum voltage	Amplitude of sinusoidal wave	Offset of sinusoidal wave
(4)				
(5)				

C. Difference Amplifier

Potentiometer voltage to set the output voltage offset to zero

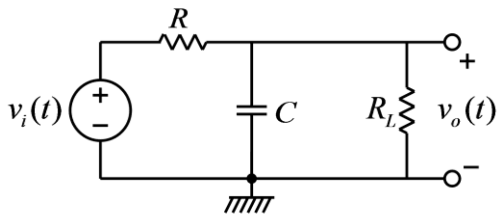
Lab 11. Basic Application Circuits for Operational Amplifiers

- **Voltage Follower/Impedance Buffer**



(Figure) Voltage follower

In the above OP amp circuit, the output voltage signal is always equal to the input voltage signal, so it is called a voltage follower. This circuit can be used to minimize the effect of a load resistor when it is connected to a circuit with a specific function. For example, consider the following circuit where a load resistor is connected to a low-pass filter circuit:

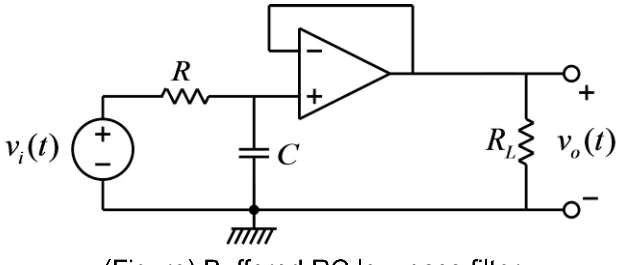


(Figure) RC low-pass filter

The transfer function when no load resistance R_L is connected to this filter circuit is as follows.

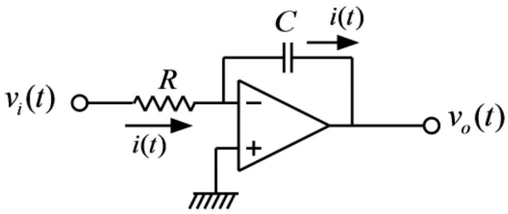
$$G(s) = \frac{1}{RCs+1}$$

However, as shown in the figure above, connecting a load resistor changes the transfer function, preventing the filter from performing its intended function. To solve this problem, a voltage follower circuit using an OP amp can be employed. Specifically, inserting a voltage follower between the low-pass filter and the load resistor, as shown in the figure below, ensures that the output voltage of the low-pass filter and the voltage across the load resistor are always equal. Since almost no current flows into the input terminals of the OP amp, the load resistor has negligible influence on the low-pass filter. When used for this purpose, the above circuit is also referred to as an impedance buffer.



(Figure) Buffered RC low-pass filter

- **Integrator**



(Figure) Integrator

The circuit above is an integrator circuit using an operational amplifier. In this circuit, the current $i(t)$ is as follows.

$$i(t) = \frac{v_i(t)}{R}$$

The negative (-) input terminal of the op-amp is a virtual ground at 0 volts. Therefore, the output voltage is the voltage obtained by multiplying the integral of the input voltage by a constant, as shown in the following equation.

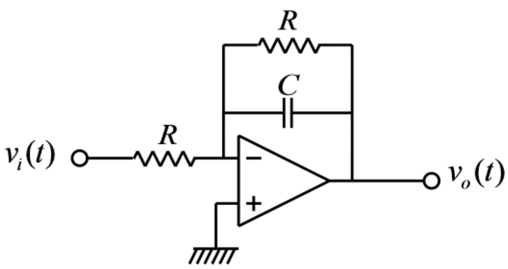
$$v_o(t) = -\frac{1}{C}\int_0^t i(\tau)d\tau + v_o(0) = -\frac{1}{RC}\int_0^t v_i(\tau)d\tau + v_o(0)$$

Furthermore, the transfer function of this circuit is as follows.

$$\frac{V_o(s)}{V_i(s)} = -\frac{1}{RCs}$$

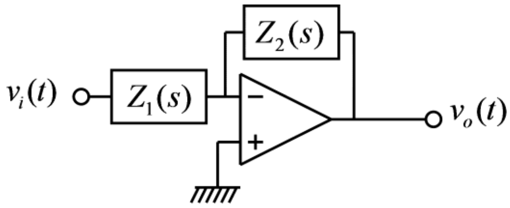
The integrator circuit described above continuously outputs the value obtained by integrating the input signal. Consequently, even with a small input value, if the input voltage is continuously applied, the output voltage exhibits a characteristic of either continuously increasing or continuously decreasing. Therefore, such integrator circuits are difficult to test. While integrator circuits are sometimes used independently, they are more commonly used as part of other circuits.

- **Active Low-Pass Filter**



(Figure) Active low-pass filter

By connecting a resistor in parallel across the capacitor terminals in the integrator circuit, an active low-pass filter circuit can be configured. The transfer function of an OP amp circuit containing a capacitor or inductor can be easily determined using the following circuit relationship:



(Figure) OP-amp circuit including impedance

The transfer function in the above circuit is as follows:

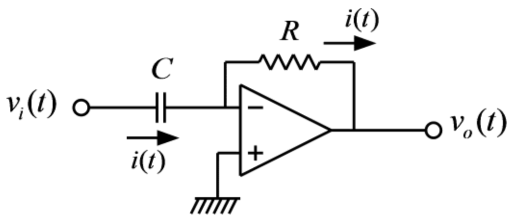
$$\frac{V_o(s)}{V_i(s)} = -\frac{Z_2(s)}{Z_1(s)}$$

Using this relationship, the transfer function of the above active low-pass filter can be easily obtained as follows.

$$\frac{V_o(s)}{V_i(s)} = -\frac{\left(R \,||\, \frac{1}{Cs}\right)}{R} = -\frac{1}{R}\left(\frac{\frac{R}{Cs}}{R+\frac{1}{Cs}}\right) = -\frac{1}{RCs+1}$$

The advantage of the above active low-pass filter is that, unlike passive low-pass filters composed solely of capacitors and resistors, it is unaffected by the value of the load resistance..

- **Differentiator**



(Figure) Differentiator

In the circuit of an integrator, swapping the positions of the resistor and capacitor results in a differentiator circuit as shown in the figure above. In this circuit, the current $i(t)$ is as follows as follows.

$$i(t) = C\frac{dv_i(t)}{dt}$$

The negative (-) input terminal of the op-amp is a virtual ground at 0 volts. Therefore, the output voltage is the voltage obtained by multiplying the derivative of the input voltage by a constant, as shown in the following equation.

$$v_o(t) = -Ri(t) = -RC\frac{dv_i(t)}{dt}$$

Furthermore, the transfer function of this circuit is as follows.

$$\frac{V_o(s)}{V_i(s)} = -RCs$$

If the following sine wave is input to this circuit,

$$v_i(t) = A\cos\omega t\ ,$$

the output is as follows:

$$v_o(t) = -A\omega\sin\omega t$$

That is, from the above equation, we can see that the magnitude of the output signal is proportional to the frequency of the input signal. This characteristic of the differential circuit results in sensitivity to noise. Generally, noise signals contain high-frequency components, and thus the differentiator circuit amplifies the magnitude of the noise signal's high-frequency components. For this reason, such differentiator circuits are rarely used as-is in actual circuits and are often employed in conjunction with a low-pass filter.

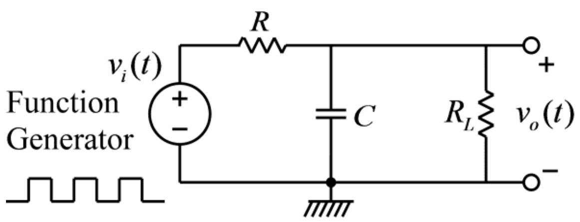
Instruments and Components Used

- Oscilloscope: 1
- Function generator: 1
- DC power supply(dual): 1
- Capacitor: $0.1\ \mu\text{F}$ x1
- OP amp: 741 x1
- Resistor: $1\ \text{K}\Omega$ x1, $10\ \text{K}\Omega$ x2, $100\ \Omega$ x2

실험 방법 및 절차

Precautions when configuring OP amp circuits: In OP amp circuits, the output signal may oscillate. In such cases, connecting a capacitor (0.1 uF) between the power supply terminal (+15V connection terminal) and ground of the OP amp IC usually eliminates the oscillation.

A. Voltage Follower



(Figure) RC low-pass filter

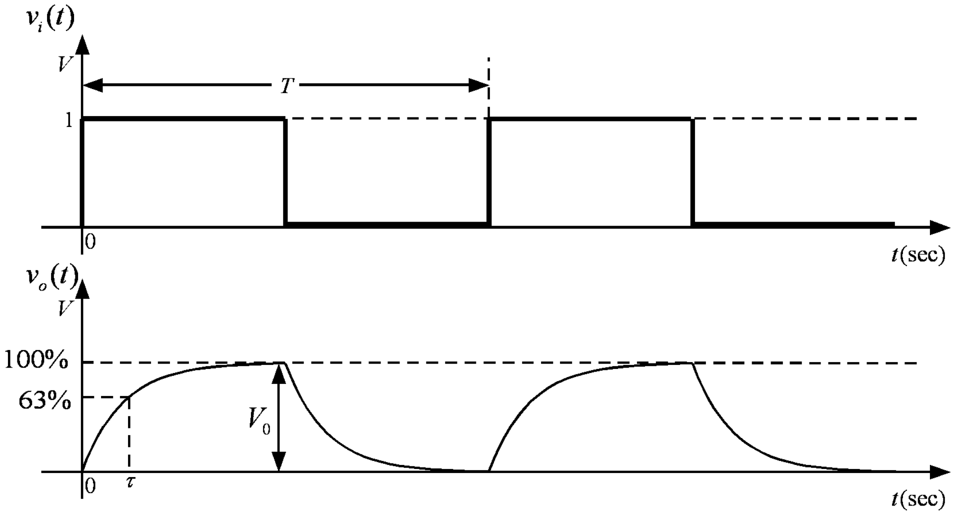
(1) Construct the circuit shown in the figure above, connecting a function generator to the input side and an oscilloscope to the output side. The values of the resistor and capacitor are as follows.

$$C = 0.1\ \mu\text{F},\ R = 10\ \text{K}\Omega,\ R_L = \infty$$

(That is, the R_L resistor is not connected.)

(2) Set the function type of the function generator to square wave, with a frequency of approximately 100Hz (period 10msec) and a voltage amplitude of 1 volt. That is, set the difference between the low voltage and high voltage of the square wave to 1 volt; the low voltage does not necessarily need to be 0 volts.

(3) Measure the time constant τ and the output value V_0 as shown in the following figure.

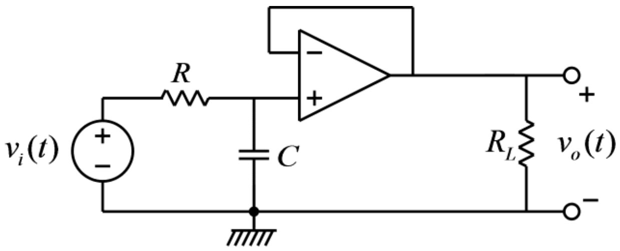


(Figure) Input waveform and output waveform

(4) Change the value of the load resistance as follows and repeat the measurement in (3).

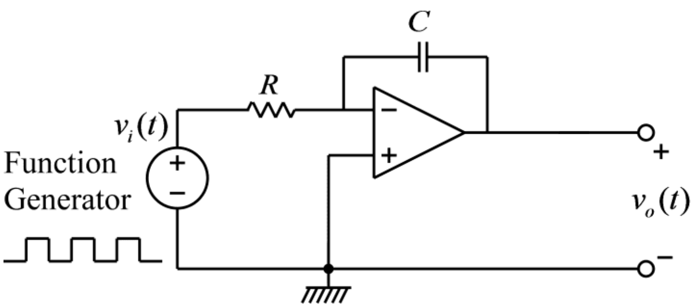
$$R_L = 10\ \text{K}\Omega$$
$$R_L = 1\ \text{K}\Omega$$

(5) Insert a voltage follower circuit between the filter circuit and the load resistor as shown in the figure below, and repeat the measurement in (4).



(Figure) Buffered RC low-pass filter

B. Integrator

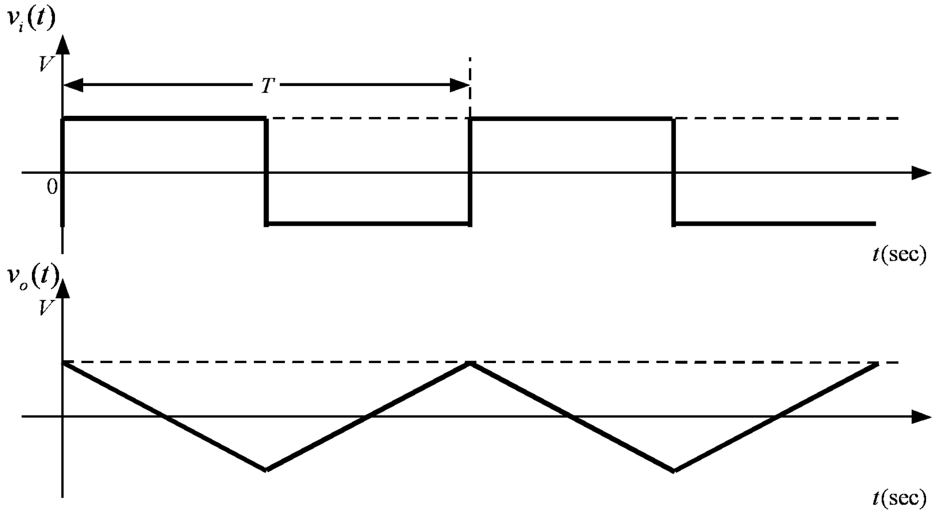


(Figure) Integrator

(1) Construct the integrator circuit shown in the figure above, connecting a function generator to the input side and an oscilloscope to the output side. The values of the resistor and capacitor are as follows.

$$C = 0.1\ \mu\text{F},\ R = 10\ \text{K}\Omega$$

(2) Set the function type of the function generator to square wave, with a frequency of approximately 100Hz (period 10msec) and a voltage amplitude of 1 volt. That is, set the difference between the low voltage and high voltage of the square wave to 1 volt. Integrating the square wave produces a triangular wave as shown in the figure below. At this point, if the average value (DC component) of the input square wave is not exactly zero, the output triangular wave will gradually increase or decrease in value. In actual experiments, the average value of the input square wave is often not exactly zero, making it difficult to continuously observe the output.

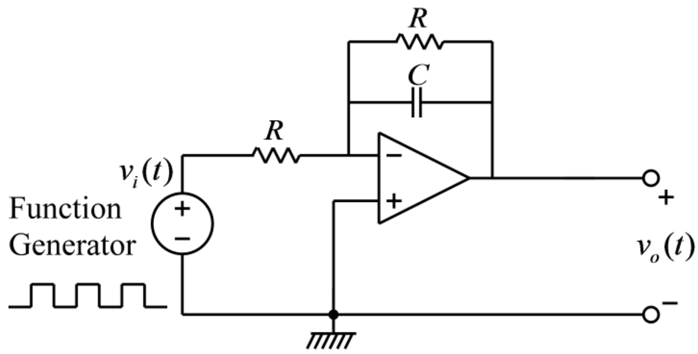


(Figure) Input waveform and output waveform

First, turn off the power supply connected to the OP amp. Set the voltage value for one division on the oscilloscope channel observing the output to 5 volts, ensuring the measurable voltage range is at least -15 to +15 volts. Next, when you turn on the power supply connected to the OP amp, you can observe the output voltage gradually increasing or decreasing until it reaches +15 volts or -15 volts. Theoretically, it should increase to an infinite magnitude, but in actual circuits, it cannot exceed the voltage applied to the OP amp. Therefore, the magnitude increases until the OP amp reaches saturation. After turning on the power, measure the time it takes to reach saturation, which should occur within a few seconds. At this point, the output waveform takes the form of a triangle wave. However, the triangle wave's shape is too small relative to the oscilloscope's measurement range, making it difficult to observe the triangle waveform clearly.
Next, turn off the power and wait briefly until the capacitor discharges, then turn the power back on and observe. However, this time, change the oscilloscope's voltage measurement mode from

DC to AC and reduce the voltage per division on the oscilloscope's scale for observation. This allows you to observe only the AC component, excluding the DC component, from the output waveform, enabling you to see the shape of the triangular wave. However, this normal measurement is only possible before the voltage saturates. Once the voltage saturates, the shape of the triangle wave becomes somewhat distorted. When the voltage saturates, turn off the power again to discharge the capacitor and repeat the measurement.
The slope value of the triangular wave can be calculated from the values of the capacitor and resistor and the magnitude of the input waveform. Measure the slope of the triangular wave and compare it with the calculated value. Additionally, after doubling the magnitude of the input voltage, verify that the slope of the triangular wave also doubles.

C. Active Low-Pass Filter Experiment

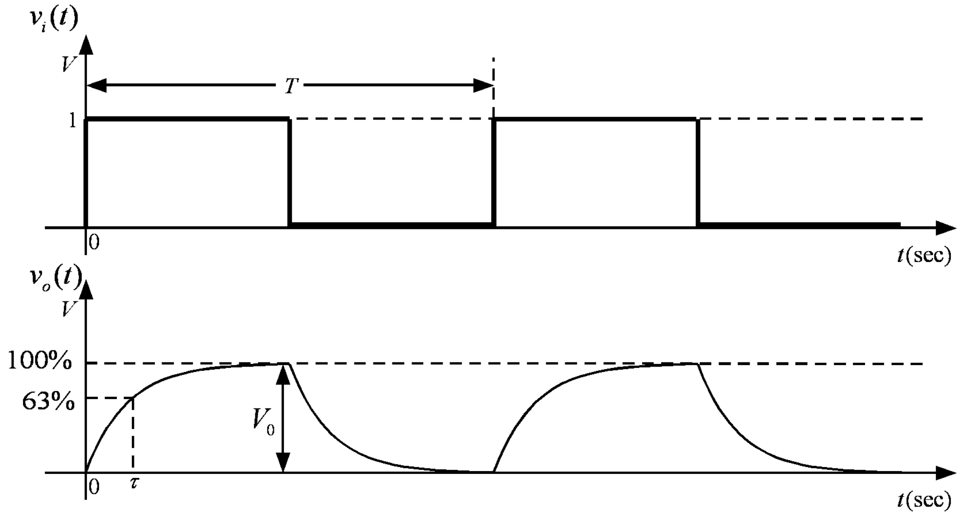


(Figure) Active low-pass filter

(1) Construct the circuit shown in the figure above, connecting a function generator to the input side and an oscilloscope to the output side. The values of the resistor and capacitor are as follows.

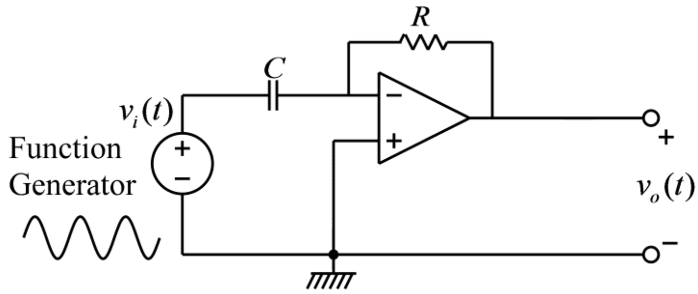
$$C = 0.1\ \mu\text{F},\ R = 10\ \text{K}\Omega$$

(2) Set the function type of the function generator to square wave, with a frequency of approximately 100Hz (period 10msec) and a voltage amplitude of 1 volt. That is, set the difference between the low voltage and high voltage of the square wave to 1 volt; the low voltage does not necessarily need to be 0 volts.
(3) Measure the steady-state time constant τ and the steady-state output value V_0 as shown in the following figure.



(Figure) Input waveform and output waveform

D. Differentiator

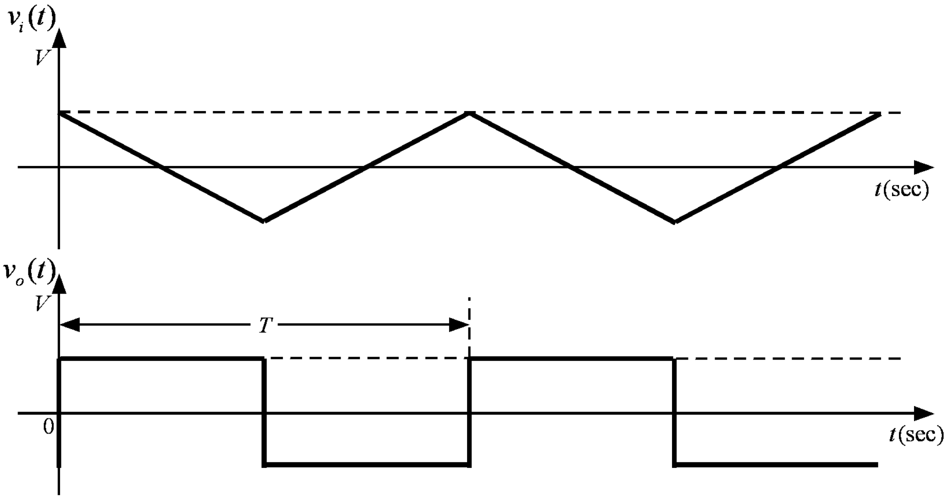


(Figure) Differentiator

(1) Swapping the positions of the capacitor and resistor in the integrator circuit creates a differentiator circuit shown above. Construct the circuit shown in the figure, connecting a function generator to the input side and an oscilloscope to the output side. The values of the resistor and capacitor are as follows.

$$C = 0.1\ \mu\text{F},\ R = 10\ \text{K}\Omega$$

(2) Set the function type of the function generator to a triangular wave, with a frequency of approximately 100Hz (period 10msec) and a voltage amplitude of 1 volt. That is, set the difference between the low voltage and high voltage of the square wave to 1 volt. Differentiating the triangular wave produces a square wave, as shown in the figure below. The amplitude of the square wave can be calculated from the values of the capacitor and resistor and the input waveform amplitude. Measure the square wave amplitude and compare it to the calculated value. Additionally, after doubling the input frequency, verify that the square wave amplitude also doubles. Furthermore, observe that a momentary peak occurs in the output waveform at the point where the input triangle wave is clipped, and consider the cause of such peaks.



(Figure) Input waveform and output waveform

(3) Set the function type of the function generator to sine wave, with a frequency of approximately 100Hz (period 10msec) and a voltage amplitude of 1 volt. That is, set the difference between the low voltage and high voltage of the square wave to 1 volt. Differentiating a sine wave produces another sine wave, but the phase changes. Furthermore, the amplitude of the output waveform is proportional to the frequency of the input sine wave. Increase the frequency from 100 Hz to 1 kHz and measure the amplitude of the output waveform. As with the previous case of the triangle wave input, the amplitude of the output sine wave can be calculated. Compare the calculated values with the measured values for each frequency.

Review and Discussion of Experimental Results

A. Voltage Follower

R_L	Without a Voltage Follower		With a Voltage Follower	
	Time constant τ	Output voltage V_0 in steady state	Time constant τ	Output voltage V_0 in steady state
∞				
10KΩ				
1KΩ				

B. Integrator

Amplitude of the input square wave	Calculated value	Measured value
	Slope of the triangular wave	Slope of the triangular wave
1 Volt		
2 Volt		

C. Active Low-Pass Filter

Calculated value		Measured value	
Time cosntant τ	Output voltage V_0 in steady state	Time constant τ	Output voltage V_0 in steady state

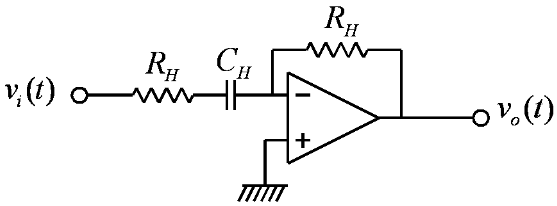
D. Differentiator

Frequency of the input triangular wave	Caluculated value	Measure value
	Amplitude of the square wave	Maplitude of the square wave
100 Hz		
200 Hz		

Frequency of the input sinusoidal wave	Calculated value	Measured value
	Amplitude of the sinusoiadal wave	Amplitude of the sinusoidal wave
100 Hz		
200 Hz		
400 Hz		
600 Hz		
800 Hz		
1000 Hz		

Lab 12. Active Filter

Active High-Pass Filter



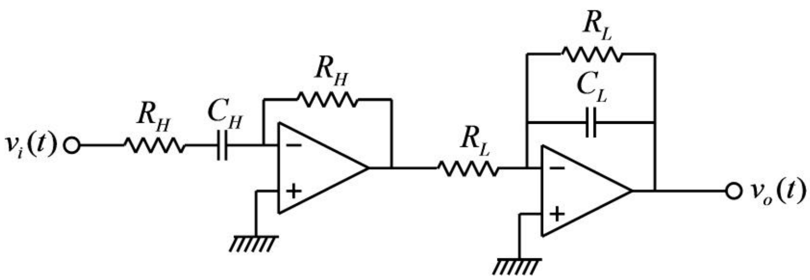
(Figure) Active high-pass filter

The circuit above is a high-pass filter using an OP amp, and its transfer function is as follows:

$$\frac{V_o(s)}{V_i(s)} = -\frac{R_H}{R_H + \dfrac{1}{C_H s}} = -\frac{R_H C_H s}{R_H C_H s + 1} = -\frac{s}{s+\omega_H}, \omega_H = \frac{1}{R_H C_H}$$

As seen in the transfer function above, the magnitude of the transfer function decreases as the frequency decreases. Conversely, as the frequency increases, the magnitude of the transfer function approaches 1. Therefore, this circuit does not pass low-frequency signals and only passes high-frequency signals, thus exhibiting the characteristics of a high-pass filter.

Active Band-Pass Filter



(Figure) Active band-pass filter

As shown in the figure above, connecting a low-pass filter and a high-pass filter in series forms a band-pass filter that allows only a specific band of frequency signals to pass through. The transfer function of this circuit is as follows.

$$H(s) = \frac{V_o(s)}{V_i(s)} = \left(-\frac{1}{R_L C_L s + 1}\right)\left(-\frac{R_H C_H s}{R_H C_H s + 1}\right) = \left(-\frac{\omega_L}{s+\omega_L}\right)\left(-\frac{s}{s+\omega_H}\right)$$

$$= \frac{\omega_L s}{s^2 + \left(\omega_L + \omega_H\right)s + \omega_L \omega_H}, \omega_L = \frac{1}{R_L C_L}, \omega_H = \frac{1}{R_H C_H}$$

For example, we determine the transfer function for the following resistance and capacitance values, plot the Bode plot, and examine the frequency response characteristics.

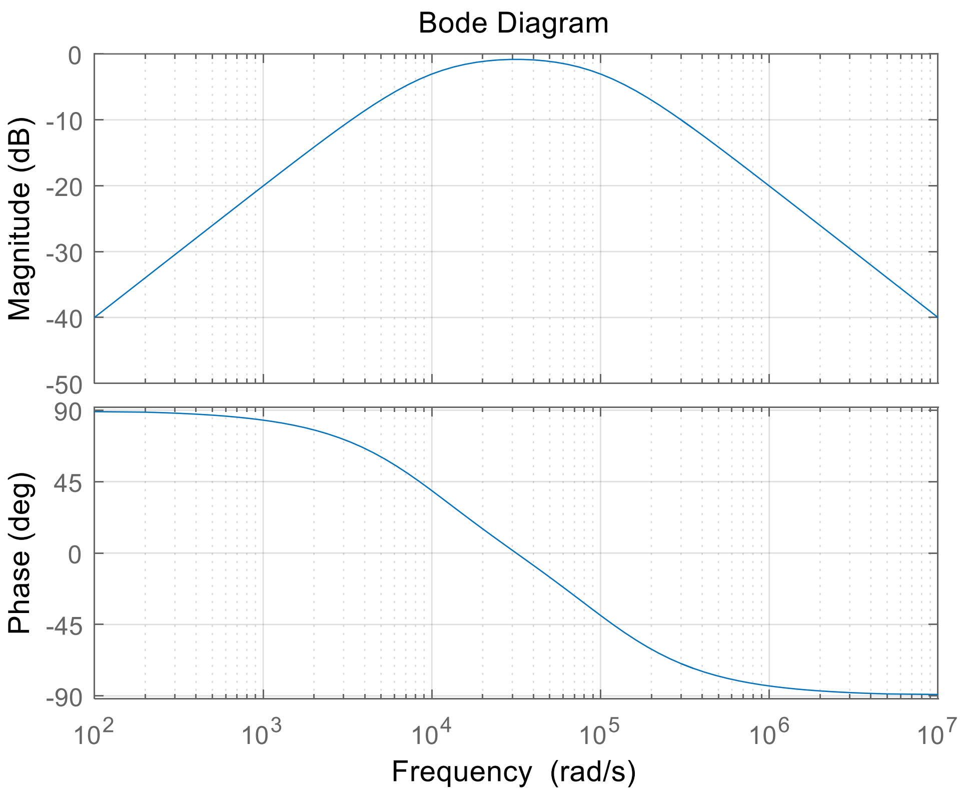
$$R_L = 100\ \Omega,\ C_L = 0.1\ \mu\text{F}$$
$$R_H = 1\ \text{K}\Omega,\ C_H = 0.1\ \mu\text{F}$$

$$H(s) = \frac{10^5 s}{s^2 + 1.1\times 10^5 s + 10^9}$$

From the above values, the center frequency is

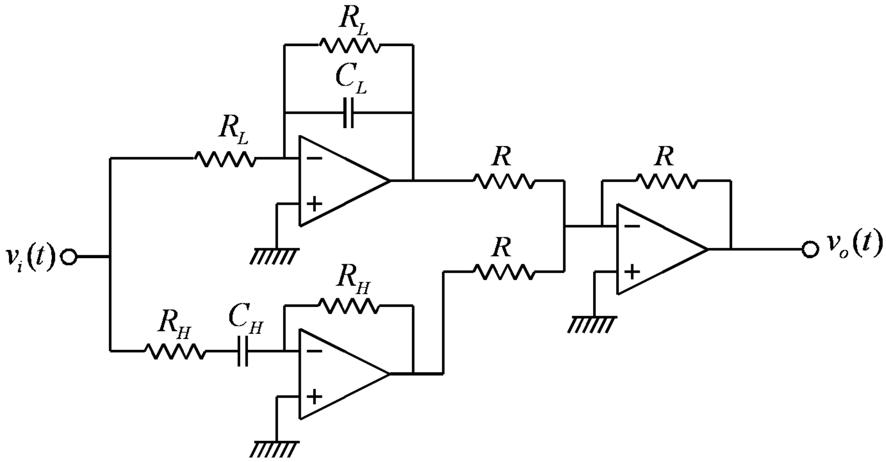
$$\omega_0 = \sqrt{\omega_L \omega_H} = \sqrt{10^9} = 3.16\times 10^4 (\text{rad/sec}) = 5.03\times 10^3 (\text{Hz})\,.$$

The Bode plot is as shown in the following figure.



(Figure) Band-pass Filter Bode Plot

- **Active Band-Reject Filter**



(Figure) Active Band-Reject Filter

As shown in the figure above, connecting a low-pass filter and a high-pass filter in parallel can form a band-reject filter that does not pass frequency signals within a specific band. The transfer function of the above circuit is as follows.

$$H(s) = \frac{V_o(s)}{V_i(s)} = \frac{1}{R_L C_L s + 1} + \frac{R_H C_H s}{R_H C_H s + 1} = \frac{\omega_L}{s + \omega_L} + \frac{s}{s + \omega_H}$$

$$= \frac{s^2 + 2\omega_L s + \omega_L \omega_H}{s^2 + (\omega_L + \omega_H) s + \omega_L \omega_H}, \omega_L = \frac{1}{R_L C_L}, \omega_H = \frac{1}{R_H C_H}$$

For example, we determine the transfer function for the following resistance and capacitance values, plot the Bode plot, and examine the frequency response characteristics.

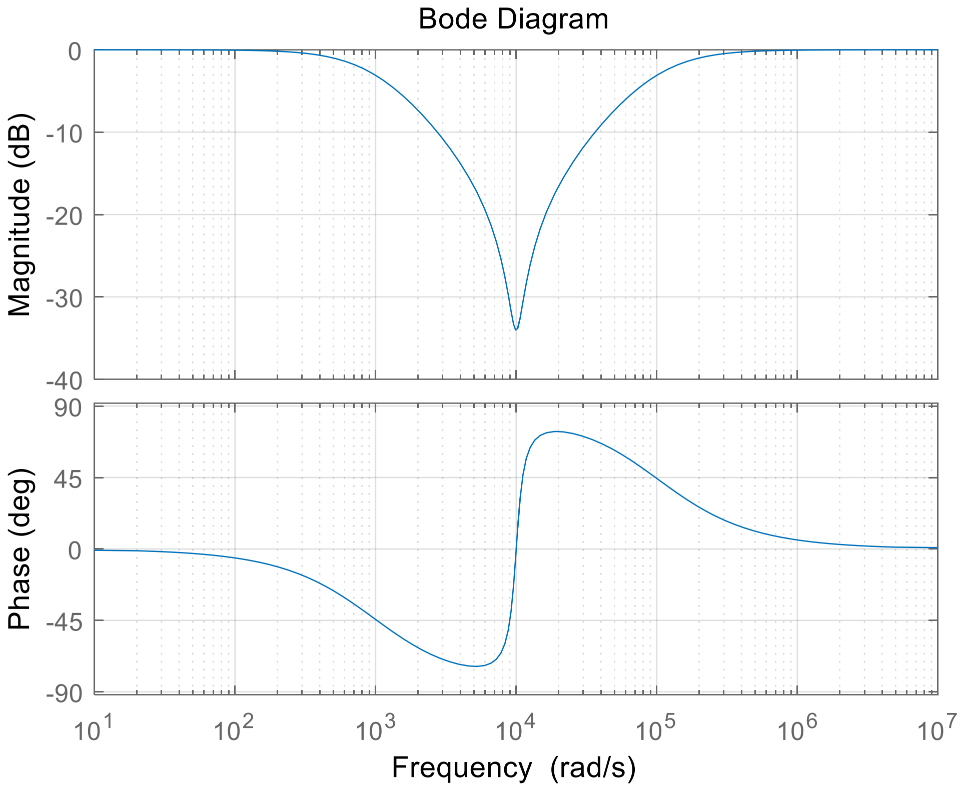
$$R_L = 10 \text{ K}\Omega,\ C_L = 0.1\ \mu\text{F}$$
$$R_H = 100\ \Omega,\ C_H = 0.1\ \mu\text{F}$$

$$H(s) = \frac{s^2 + 2000s + 10^8}{s^2 + 1.01 \times 10^5 s + 10^8}$$

From the above values, the center frequency is

$$\omega_0 = \sqrt{\omega_L \omega_H} = \sqrt{10^8} = 10^4 (\text{rad/sec}) = 1.59 \times 10^3 (\text{Hz}).$$

The Bode plot is as shown in the following figure.



(Figure) Band-reject filter Bode plot

Instruments and Components Used

- Oscilloscope: 1
- Function generator: 1
- DC power supply(dual): 1
- Capacitor: $0.1\ \mu\text{F}$ x2
- OP amp: 741 x3
- Resistor: $100\ \Omega$ x2, $1\ \text{K}\Omega$ x2, $10\ \text{K}\Omega$ x4

Experimental Methods and Procedures

Precautions when configuring OP amp circuits: In OP amp circuits, the output signal may oscillate. In such cases, connecting a capacitor (0.1 uF) between the power supply terminal (+15V connection terminal) and ground of the OP amp IC usually eliminates the oscillation..

A. Active Band-Pass Filter

(1) Construct an active band-pass filter using components with the following values, connecting a function generator to the input side and an oscilloscope to the output side.

$$R_L = 100\Omega, C_L = 0.1\mu\text{F}$$
$$R_H = 1\text{K}\Omega, C_H = 0.1\mu\text{F}$$

(2) Set the function type of the function generator to sine wave, and set the voltage amplitude (peak-to-peak) to 2 volts. That is, set the difference between the low voltage and high voltage of the sine wave to 2 volts.

(3) Vary the frequency of the sine wave with the following values, and record the ratio of the output voltage magnitude to the input voltage magnitude.

$$\omega = 10^3\ \text{rad/sec},\ f = 159\ \text{Hz}$$
$$\omega = 3\times 10^3\ \text{rad/sec},\ f = 477.7\ \text{Hz}$$
$$\omega = 10^4\ \text{rad/sec},\ f = 1.59\ \text{KHz}$$
$$\omega = 3.16\times 10^4\ \text{rad/sec},\ f = 5.03\ \text{KHz}$$
$$\omega = 10^5\ \text{rad/sec},\ f = 15.9\ \text{KHz}$$
$$\omega = 3\times 10^5\ \text{rad/sec},\ f = 47.8\ \text{KHz}$$
$$\omega = 10^6\ \text{rad/sec},\ f = 159.2\ \text{KHz}$$

(4) Using the values measured above, draw an approximate Bode plot.

B. Active Band-Reject Filter

(1) Construct an active band-reject filter using components with the following values, connecting a function generator to the input side and an oscilloscope to the output side.

$$R_L = 10\mathrm{K}\Omega, C_L = 0.1\mu\mathrm{F}$$
$$R_H = 100\Omega, C_H = 0.1\mu\mathrm{F}$$
$$R = 10\mathrm{K}\Omega$$

(2) Set the function type of the function generator to sine wave, and set the voltage amplitude (peak-to-peak) to 2 volts. That is, set the difference between the low voltage and high voltage of the sine wave to 2 volts.

(3) Vary the frequency of the sine wave with the following values, and record the ratio of the output voltage magnitude to the input voltage magnitude.

$$\omega = 10^2 \text{ rad/sec},\ f = 15.9 \text{ Hz}$$
$$\omega = 10^3 \text{ rad/sec},\ f = 159 \text{ Hz}$$
$$\omega = 3\times 10^3 \text{ rad/sec},\ f = 478 \text{ Hz}$$
$$\omega = 10^4 \text{ rad/sec},\ f = 1.59 \text{ KHz}$$
$$\omega = 3\times 10^4 \text{ rad/sec},\ f = 4.78 \text{ KHz}$$
$$\omega = 10^5 \text{ rad/sec},\ f = 15.9 \text{ KHz}$$
$$\omega = 10^6 \text{ rad/sec},\ f = 159.2 \text{ KHz}$$

(4) Using the values measured above, draw an approximate Bode plot.

- **Review and Discussion of Experimental Results**

A. Active Band-Pass Filter

ω (rad/sec)	f (Hz)	Amplitude of the input sinusoidal wave(V)	Amplitude of the output sinusoidal wave(V)	$\lvert H(j\omega)\rvert$	$20\log_{10}\lvert H(j\omega)\rvert$
10^3	159				
3×10^3	477.7				
10^4	1.59 K				
3.16×10^4	5.03 K				
10^5	15.9 K				
3×10^5	47.8 K				
10^6	159.2 K				

B. Active Band-Reject Filter

ω (rad/sec)	f (Hz)	Amplitude of the input sinusoidal wave(V)	Amplitude of the output sinusoidal wave(V)	$\lvert H(j\omega)\rvert$	$20\log_{10}\lvert H(j\omega)\rvert$
10^2	15.9				
10^3	159				
3×10^3	478				
10^4	1.59 K				
3×10^4	4.78 K				
10^5	15.9 K				
10^6	159.2 K				