# ELECTRICAL – ELECTRONICS LABORATORY EXPERIMENTS

YILDIZ TECHNICAL UNIVERSITY

MECHATRONICS ENGINEERING

ERHAN AKDOĞAN   MEHMET HAKAN DEMİR

MEHMET EMİN AKTAN   AHMET TAHA KORU

İSTANBUL, 2017

ELECTRICAL – ELECTRONICS LABORATORY EXPERIMENTS

YILDIZ TECHNICAL UNIVERSITY
MECHATRONICS ENGINEERING

ERHAN AKDOĞAN   MEHMET HAKAN DEMİR

MEHMET EMİN AKTAN   AHMET TAHA KORU

İSTANBUL, 2017

T.C.
YILDIZ TEKNİK ÜNİVERSİTESİ
MAKİNE FAKÜLTESİ

ELECTRICAL - ELECTRONICS
LABORATORY EXPERIMENTS

Doç. Dr. Erhan AKDOĞAN
Arş. Gör. Mehmet Hakan DEMİR
Arş. Gör. Mehmet Emin AKTAN
Arş. Gör. Ahmet Taha KORU

TABLE OF CONTENTS

PREFACE

This book is prepared for the Electrical – Electronics Laboratory Course which is mandatory in 3rd semester of the Department of Mechatronic Engineering at Yıldız Technical University. The book involves 11 experiments which are applications of theoretical topics covered in Analog Electronics and Introduction to Electric Circuits. The experiments are prepared after 6 years' experience. All of the experiments are conducted in a laboratory before being included in the book. The book can be used in electrical – electronics laboratory courses of the engineering departments and advanced vocational studies.

Our experiences show that the experiments using breadboards support students to understand electrical circuits' behaviors in low voltage applications. Furthermore, the experiments help students to learn basics about components and to improve their electric circuit design skills. The book is prepared with this perspective.

We hope this book contributes to academicians' work and students' education. However, we would you like to thanks Research Assistant Abdurrahman Yılmaz for his valuable benefits to translating of this book.

Authors

2017

"To Commemorate the 15th of July Martyrs"

Laboratory Rules

1. Do not put anything on tables except experiment components and equipment.
2. Do not eat or drink or bring beverages.
3. Laboratory course is like class courses. Obey the class course rules in the laboratory.
4. Do not talk loud.
5. Turn off your cell phones.
6. Only concentrate on the experiments.
7. Use your time in the lab to finish the experiments to earn as much as experience and information possible. Do not hurry to save time.
8. During building the circuits, turn off all of the equipment. Do not turns on equipment before the corresponding assistant check the circuits.
9. While the circuits are active, do not plug in or out cables or touch the equipment.
10. Before the experiment, complete the preliminary works. Hence,
    a. You can use your time efficient,
    b. You can benefit more from the experiment,
    c. You can easily compare the analytic and experimental results.
    d. Unprepared students those didn't complete preliminary works, affect the other students' performance negatively.
11. Protect the equipment as they will be used in future laboratory courses as well.
12. Before leaving the laboratory, turn of all of the equipment, clean you table, put the equipment, the cables and the chairs to their places.

General Information and Laboratory Equipment

General Electrical – Electronics Terms

Voltage

The force generating the electron motion (current) is voltage. In other words, it is the potential difference between end terminals of a conductor. The unit is "Volt" which is symbolized by V. The voltage is measured by voltmeter. Voltmeter is connected to the end terminals in parallel (Figure 1).
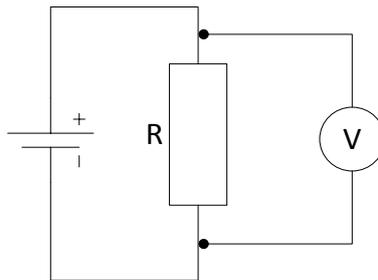
Figure 1 – Voltmeter is connected in parallel

Lower Power of Voltage		Upper Power of Voltage	
Pico Volt (pV)	10^{-12}	Kilo Volt (KV)	10^3
Nano Volt (nV)	10^{-9}	Mega Volt (MV)	10^6
Micro Volt (µV)	10^{-6}		
Milivolt (mV)	10^{-3}		

Current

Current is the flow of electric charge in unit time. The unit of the current is Ampere and symbolized with "A". It is measured by an ammeter. In order to measure the current, ammeter is connected in series.

1 ampere: 1 Coulomb of electric charge in 1 second.
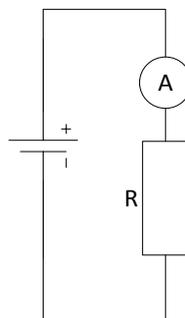
Figure 2 – Ammeter is connected in series

Lower Powers of Current		Upper Powers of Current	
Pico Ampere (pA)	10^{-12}	Kilo Ampere (KA)	10^3
Nano-Ampere(nA)	10^{-9}	Mega Ampere (MA)	10^6
MicroAmpere (µA)	10^{-6}		
Mili-Ampere (mA)	10^{-3}		

Amplitude

The half of the vertical distance between the maximum and minimum levels of a periodic signal (peak-peak) is called the amplitude. General equation of a periodic sinusoidal signal is defined as

$$y(t) = A \cdot sin(\omega t + \varphi)$$

where;

A: amplitude

ω: angular frequency

φ: phase angle

t: time

For electrical signals, the sinusoidal voltage can be rewritten as

$$v(t) = V \cdot sin(\omega t + \varphi)$$

Where, V is the amplitude or peak voltage.

The double of the peak voltage (2V) is named peak-to-peak voltage value.

Another important quantity in electric signals is the effective value (RMS). RMS value for a sinusoidal signal is

$$V_{RMS} = \frac{V}{\sqrt{2}}$$

A sinusoidal signal example can be seen in Figure 3.
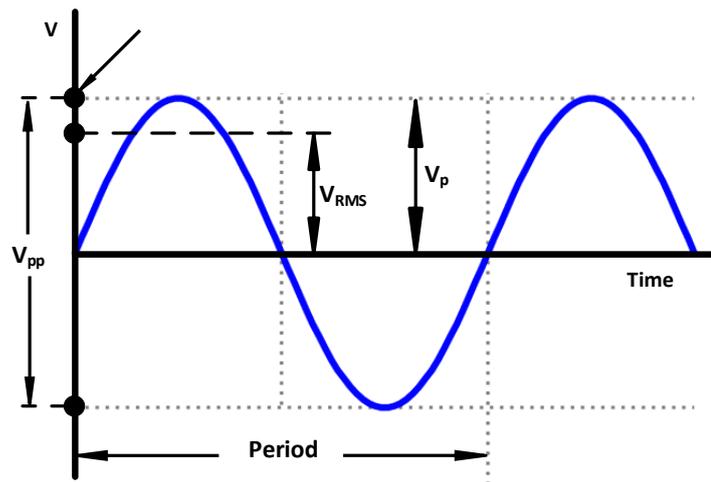
Figure 3 – Quantities of a sinusoidal signal

Let us consider the following equation

$$v(t) = 12 \cdot sin(\omega t) \quad [Volt]$$

According to the function,

V_p (peak value, amplitude): 12 V

V_{pp} (peak-to-peak voltage): 24 V

V_{RMS} (RMS voltage): 8,48 V

Breadboard Usage

Breadboard is used to build circuits without soldering components. Usually, it is used for prototyping or testing a new circuit design. Breadboards involve power rails and connected holes. The power rails are at the left and right of the breadboards and shown with blue and red lines. Usually, the red line is used for + and blue is used for − (GND) connections. Thus, the required power in different nodes of the circuit is connected to the power rails easily. The holes on the power rail connected each other from top to the bottom. If the coloured line is cut in half, the power line holes are connected in two groups, e.g., top to half and half to bottom (see Figure 4). In the middle part, there are connected holes. The **5** horizontal holes group are connected each other. They are used to place the components and build the circuit.
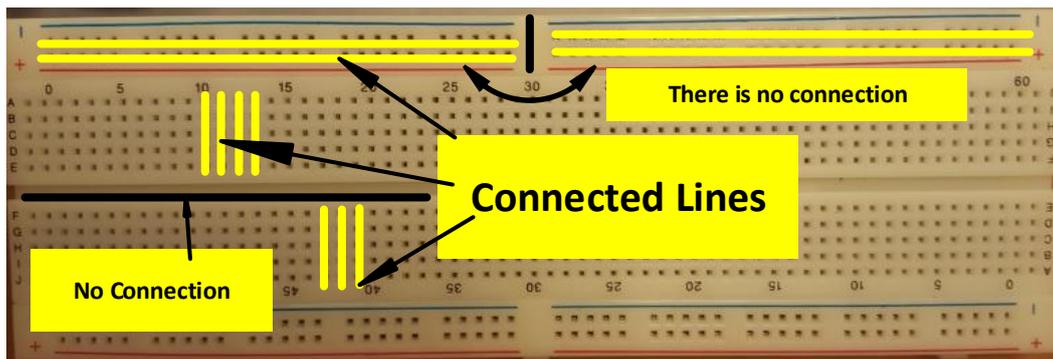
Figure 4 – Breadboard rails and connected holes

An example placement of a capacitor can be seen in Figure 5. On the right figure, the capacitor terminals are placed on the same connected hole group so that is short circuited (wrong connection). On the left figure, the terminals are place in different groups (correct connection).
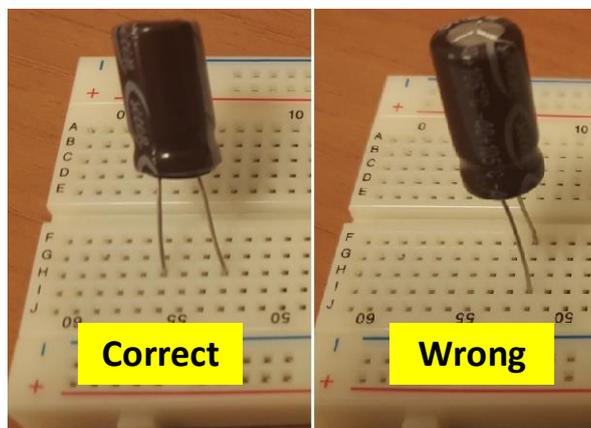
Figure 5 – Capacitor placement example

Integrated circuits are placed as seen in Figure 6.
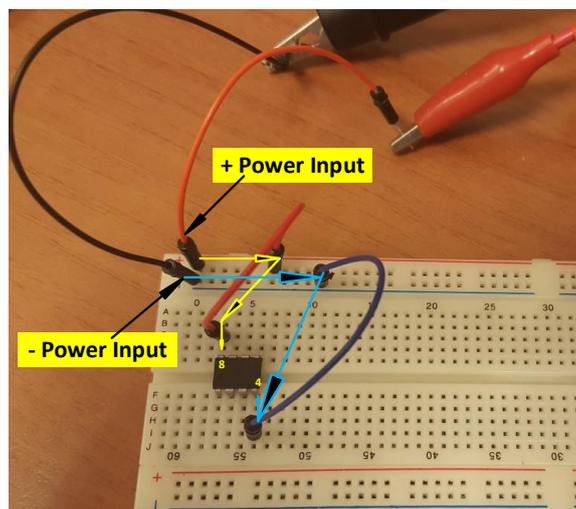
Figure 6 – Integrated circuit placement example

6

Laboratory Equipment

1. Multimeter

Multimeters are used to measure the current, voltage, resistance and capacitance. Furthermore, it can be used to short circuit and diode tests. A multimeter is seen in Figure 7.

Figure 7 – Multimeter

1. **DCV:** DC voltage measurement
2. **ACV:** AC voltage measurement
3. **DCA:** DC current measurement
4. **ACA:** AC current measurement
5. **Ω :** Resistance measurement
6. Short circuit and diode tests
7. Capacitance measurement
8. Automatic / Manual mode selection. In automatic mode, power of the unit (Kilo, Nano, etc.) is selected automatically by the multimeter. In manual mode, the power of the unit is selected by the user.
9. In order to select the power of the unit in manual mode.
10. **COM socket:** The black probe is connected here.

Red probe is connected in different ports with respect to measurement type. Explanations are below.

11. Red probe is connected here to measure voltage, resistance, capacitance or diode / short circuit tests.
12. Connected here to measure currents up to 2 A.
13. Connected here to measure currents up to 20 A.
14. Screen: Indicates the value of the measurement. Lower left of the screen indicates automatic/manual mode and DC – AC selection. Right of the screen indicates the unit.

2. DC Power Supply

It is used to generate DC. The user can set any voltage value between 0 – 24 V and set the current limits. If the circuit draws more current then the limit, the DC power supply turns itself off. A DC power supply can be seen in Figure 8.
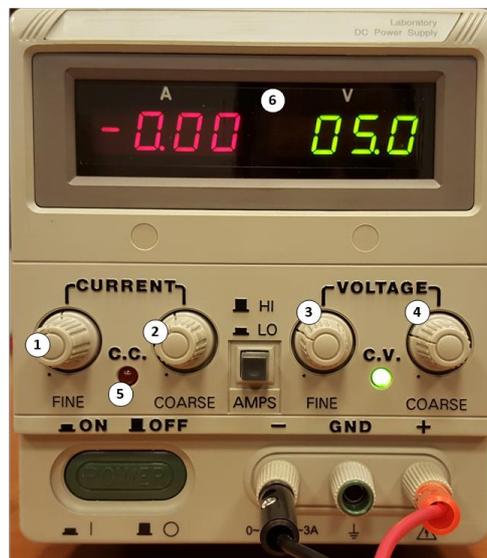
Figure 8 – DC power supply

1. Fine current limit setting
2. Coarse current limit setting
3. Fine voltage setting
4. Coarse voltage setting
5. **Short Circuit Indicator:** In case of a short circuit or an excessive current requirement more than the current limit, short circuit indicator flashes red.
6. **Screen:** Required current and the adjusted voltage are shown.

3. Function Generator

Those devices generate different types of AC signals. A function generator can be seen in Figure 9 and corresponding explanations of the adjustments can be seen below.

Figure 9 – Function generator

8

1. **Wave:** Selection of the signal type (sinusoidal, square, triangular)
2. Keypad to set frequency
3. Frequency unit selection (MHz, kHz, Hz)
4. Amplitude adjustment
5. Offset adjustment
6. **Display Screen:** The frequency value appears numerically on this screen. The lower part shows the frequency unit and the upper part shows the signal type.

4. Oscilloscope

Oscilloscope plots the signal on the input channels to a screen with respect to time. It can show the frequency, the amplitude and the period of the signal numerically. If oscilloscope includes multiple input channels, it can plot different signals at the same time. Oscilloscopes are connected to the circuit in parallel. An oscilloscope can be seen in Figure 10.
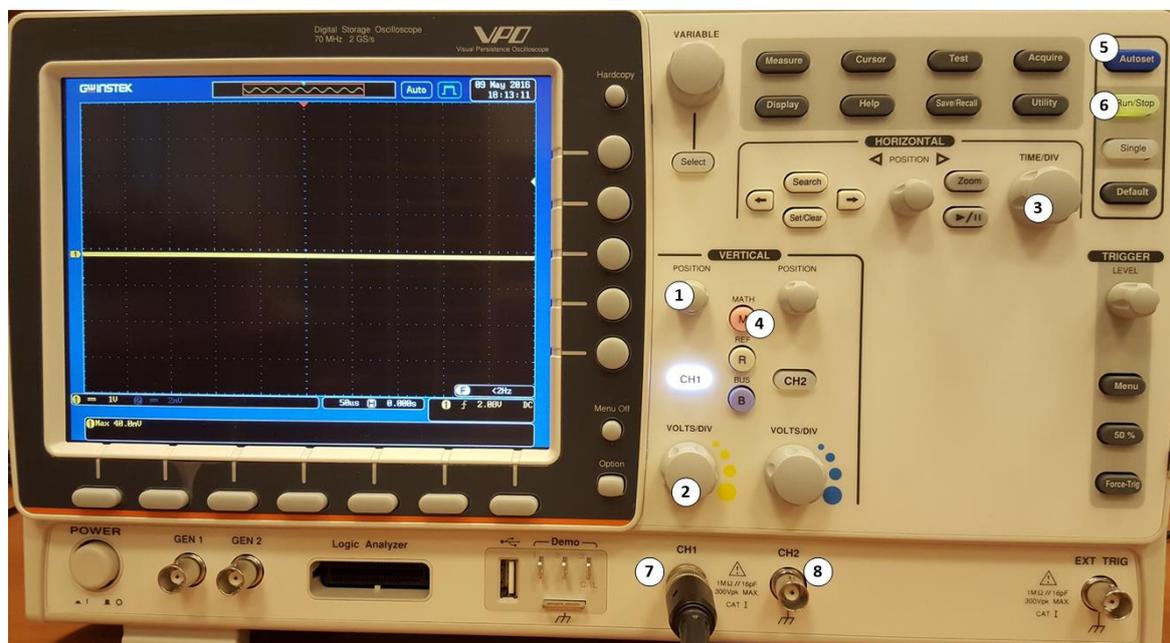
Figure 10 – Oscilloscope

1. **Position:** Adjusts the vertical placement of the signal
2. **VOLT/DIV:** Adjusts the voltage per square on the screen.
3. **TIME/DIV:** Adjusts the time per square on the screen.
4. **Math:** Used to set math operation on 2 signals on channels, e.g., addition, multiplication...
5. **Autoset:** Automatically sets VOLT/DIV and TIME/DIV to fit the signal on the screen.
6. **Run/Stop:** It is used to stop / resume signal flow.
7. **CH1:** Channel 1 Probe connection
8. **CH2:** Channel 2 Probe connection

List of Materials

General Equipment

- Breadboard
- Male – male jumper cables
- Crocodiles
- Pliers and Cutters

Experiment 1: Resistors and Capacitors

- 1 x 47Ω, 1 x 100Ω, 1 x 2.2KΩ and 1 x 10kΩ Resistor
- 1 x 100kΩ Potentiometer
- 1 x 220µF, 1 x 100µF, 1 x 47µF and 1 x 4.7µF Capacitor

Experiment 2: OHM's – Kirchhoff's Laws and AC Inductor – Resistor – Capacitor

- 1 x 47Ω, 1 x 100Ω, 1 x 2,2kΩ and 1 x 10kΩ Resistor
- 1 x 1.5nF Capacitor
- 1 x 100mH Inductor

Experiment 3: Series and Parallel RLC Circuits – Resonance Circuits

- 1 x 220 Ω and 1 x 10 Ω Resistor
- 1 x 100 nF Capacitor
- 1 x 100mH and 1 x 100 µH Inductor

Experiment 4: Investigation of Diode Characteristics

- 1 x 220 Ω / 1W Resistor
- 1 x 1 kΩ Potentiometer
- 1 x 1N4007 Diode

Experiment 5: Diode Rectifier Applications

- 1 x 3 kΩ Resistor
- 1 x 100 µF Capacitor
- 5 x 1N4001 Diode

Experiment 6: Zener Diode Characteristics and Applications

- 1 x 220 Ω / 1W and 1 x 56 Ω / 1W Resistor
- 1 x 1 kΩ Potentiometer
- 1 x 5.6 V Zener Diode
- 1 x BD139 Transistor

Experiment 7: BJT Transistor Characteristics

-   1 x 33 kΩ and 1 x 220Ω/1W Resistor
-   1 x 100kΩ Potentiometer
-   1 x BC237 Transistor

Experiment 8: BJT Amplifiers

-   1 x 100 Ω, 1 x 560 Ω, 1 x 2.2 kΩ, 1 x 12 kΩ Resistor
-   2 x 47uF Capacitor
-   1 x BC237 Transistor

Experiment 9: MOSFET Basics

-   1 x CD4007 MOSFET
-   1 x 100kΩ Potentiometer

Experiment 10: Operational Amplifier (OP-AMP) Circuits

-   1 x 1 kΩ,1 x 5 kΩ, 1 x 10 kΩ and 4 x 100 kΩ Resistor
-   2 x 10 kΩ Potentiometer
-   3 x LM 741 OP-AMP (+2 reserve)

Experiment 11: Integrator and Differentiator with OP-AMP

-   1 x 20 kΩ and 1 x 4.7 kΩ Resistor
-   2 x 470 nF Capacitor
-   2 x LM 741 OP-AMP (+2 reserve)

List of Materials

Experiment 1

Resistors and Capacitors

Required Components List:

- Resistors   : 47 Ω, 100 Ω, 2.2 kΩ and 10 kΩ
- Potentiometer   : 100 kΩ
- Capacitors   : 4.7 µF, 47 µF, 100 µF and 220 µF

  (PS: 1 piece per each component)

Experiment Instruments:

- Multimeter
- Breadboard
- DC Power Supply

Experiment Part 1 : Resistors

Purpose of the Experiment:	Equivalent resistance measurement by series, parallel and mixed connection of the resistors.
Teaching of the Experiment:	Resistor term and equivalent resistor value calculation depending on connection type will be covered.

Theoretical Information and Introduction to Part 1

Resistor can be defined as resistance to electrical current flow. Resistor, given name of an electrical variable, also expresses a circuit element used for current limiter in electronic circuits. Addition to current limiting purpose, the resistors in electronic circuits can be used for dividing the supply voltage of the circuit to adjust the voltage on the circuit elements and to obtain heat energy.

In equations or circuit schematics, the "R" letter is used to indicate resistors and the "Ω" symbol is used to explain the unit of resistors which is called as ohm (kΩ (kiloOhm) = $10^3\Omega$, MΩ (MegaOhm) = $10^6\Omega$).

Resistors are discrete circuit components and the value of each resistor is either coded indirectly by colors or written directly by digits. A comprehensive example showing how to read the digit code on the resistor is given in Figure 1.1. The numerical equivalents of color codes on the resistors are listed in Table 1.1.
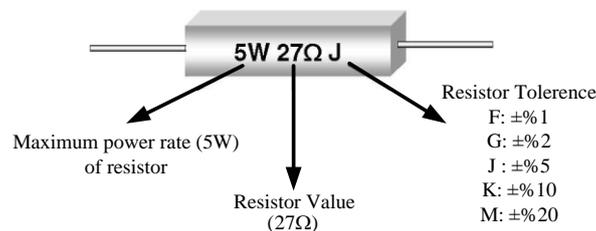
Figure 1.1 – Resistor value coding by digits

Table 1.1 – Resistors color coding

COLOR	Digit	Multiplier	Tolerance
Black	0	10^0	-
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
Purple	7	10^7	% 0.1
Gray	8	10^8	% 0.05
White	9	10^9	-
Gold	-	10^{-1}	% 5
Silver	-	10^{-2}	% 10

1st Band   2nd Band   Multiplier   Tolerance

($56 \times 10^4 = 560000 \; \Omega =$ **560 kΩ ± % 1**)
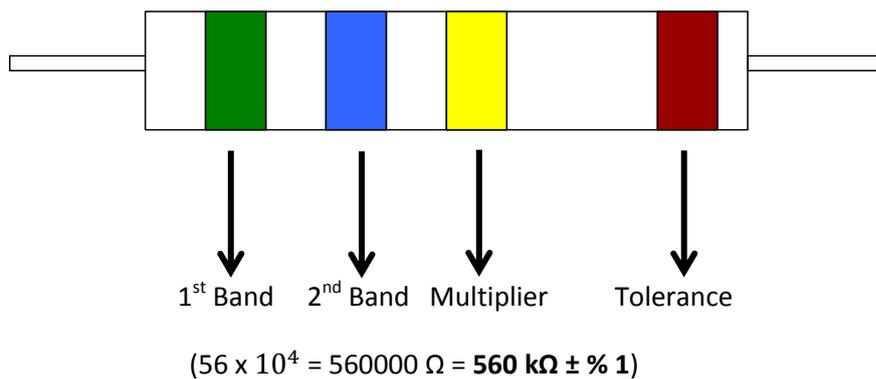
Figure 1.2 – Four band resistor example

1st Band 2nd Band 3nd Band Multiplier Tolerance

($237 \times 10^0 = 237 \; \Omega =$ **237 Ω ± % 10**)
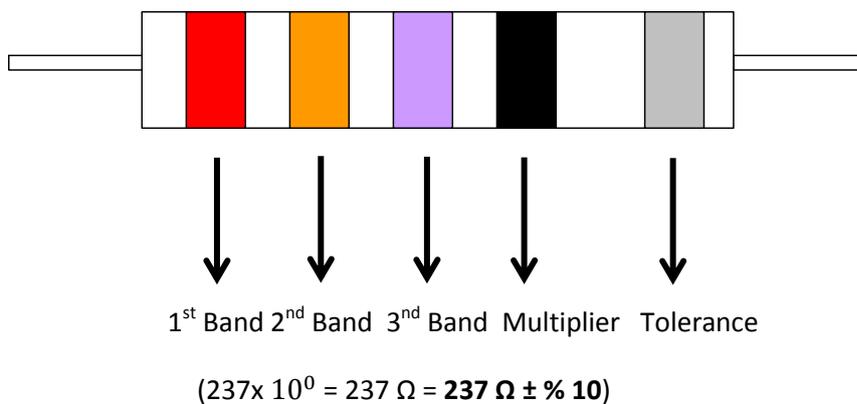
Figure 1.3 – Five band resistor example

($25 \times 10^3 = 25000 \; \Omega = 25$ **kΩ ± %5**)
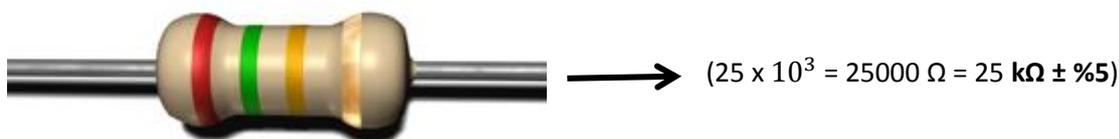
Figure 1.4 – Example resistor and its value

The resistors are connected in series or parallel to the electronic circuits with respect to user intended aim. The given equations below can be used to calculate equivalent resistor value.

Connected in Series	Connected in Parallel
$R_{eq} = R_1 + R_2 + \cdots$	$R_{eq} = \left(\dfrac{1}{R_1} + \dfrac{1}{R_2} + \cdots \right)^{-1}$
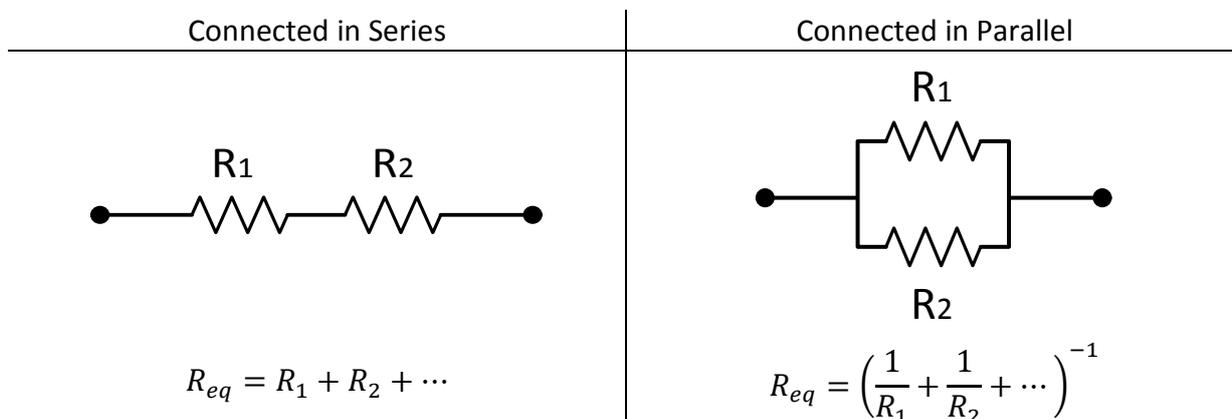
Preliminary Work – 1

✓ Read and comprehend "Things to do in Experiment Part 1".
✓ Calculate the equivalent resistor value in Figure 1.5 analytically and write the calculated value to suitable part of Table 1.2.
✓ Simulate and find the equivalent resistor value in Figure 1.5 by using Multisim Program and write the simulated value to suitable part of Table 1.2.
✓ Simulate the potentiometer circuit given in Figure 1.6 by using Multisim Program and fill the suitable parts of the Table 1.3 with observed values on simulation.

Experimental Procedures for Part 1

Part 1 Schematics
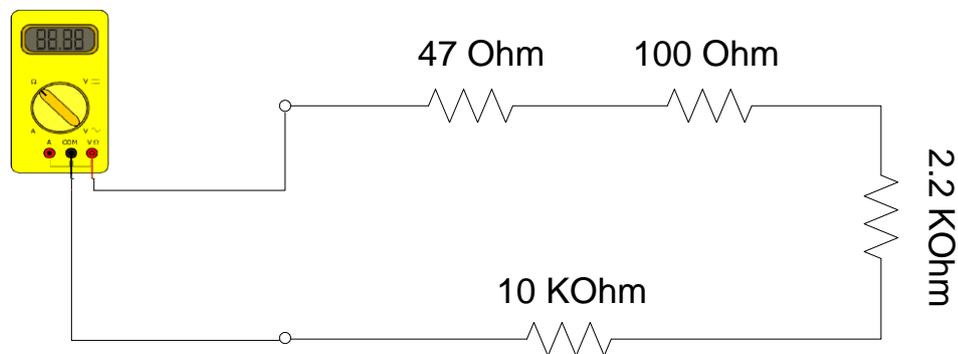
Figure 1.5 –Resistors connected in series experiment schematic

Figure 1.6 – Potentiometer experiment schematic

Experiment Steps

1. Implement the circuit given in Figure 1.5 by using breadboard.
2. As seen on the schematic, measure equivalent resistor value of the circuit by using a multimeter and fill the suitable gap on Table 1.2.

3. Compare the calculation and measurement results by considering the tolerance of used resistors in the circuit.
4. Implement the circuit given in Figure 1.6 for potentiometer measurement.
5. Rotate the potentiometer and measure the change in resistor value of the potentiometer when the multimeter is in resistance measurement mode and record to Table 1.3.

Part 1 Results

Table 1.2 – Resistor in series experiment observation table

	R_T $(k\Omega)$
Calculated	
Simulated	
Measured	

Table 1.3 – Potentiometer experiment observation table

		Potentiometer Shaft Position		
		Minimum	**Medium**	**Maximum**
Resistor Value	Simulated			
	Measured			

Part 1 Remarks

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

Experiment Part 2 : Capacitors

Purpose of the Experiment:	Equivalent capacitor measurement by series, parallel and mixed connection of the capacitors
Teaching of the Experiment:	Equivalent capacitor value calculation depending on connection types and structures will be covered.

Theoretical Information and Introduction to Part 2

A *capacitor* is a passive two-terminal component used for electrostatic energy storage in an electrical field. Two conductive plates located opposed to each other when separated by any dielectric material (paper, ceramic, air, plastic, etc.), capacitor is obtained. Each conductive plate becomes one of the terminals of capacitor. The capacitors are commonly used in electronic circuits for many purposes such as filtering, rectifying, isolation and energy storage depending on the features like electrical energy storage, quick discharge of stored energy in the event of a short circuit, blocking DC current, passing AC current and adding phase difference between input and output signals.

In equations or circuit schematics, the "C" is used to indicate capacitors and the "F" is used to explain the unit of capacitors which is called as farad.

$$1 \, farad = 1 \, \frac{coulomb}{volt}$$

The unit of capacitor (Farad) indicates the capacity of electrical load; therefore, this component is called as "capacitor". As a result, the load stored by capacitor (Q) is in direct proportion to voltage and storage capacity value of capacitor.

$$Q = C.V$$

The metric conversions for the commonly used capacitors types are added below.

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

Non Polarized	Polarized			
Notation Forms				Capacitor Types

The capacitors are connected in series or parallel to the electronic circuits with respect to user intended aim. The equivalent capacitor value for different connection types can be calculated as below.

Capacitors in Series	Capacitors in Parallel
The charge of each capacitor is equal when the capacitors connected in series. $$Q_1 = Q_2 = \cdots = Q_n$$ The equivalent capacitor value is calculated by $$C_{eş} = \left(\frac{1}{C_1} + \frac{1}{C_2} + \cdots + \frac{1}{C_n}\right)^{-1}$$	The voltage of each capacitor is equal when the capacitors connected in parallel. $$V_1 = V_2 = \cdots = V_n$$ The equivalent capacitor value is calculated by $$C_{eş} = C_1 + C_2 + \cdots + C_n$$
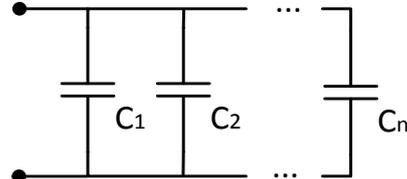
Preliminary Work – 2

✓ Read and comprehend "Things to do in Experiment Part 2".
✓ Calculate the equivalent capacitor value in Figure 1.7 analytically.
✓ Calculate each capacitor voltage value in Figure 1.8 analytically.
✓ Simulate the circuit given in Figure 1.8 by using Multisim Program and find the voltage level of the capacitors. (PS: In order to observe correct values of capacitor voltages, apply given settings: Simulate →Mixed Mode Simulation Settings → Use Real Pin Models. After observation, please set the system default settings.)
✓ Record achieved results to the suitable parts of the Table 1.4 and Table 1.5.

## Experimental Procedures for Part 2

Part 2 Schematics
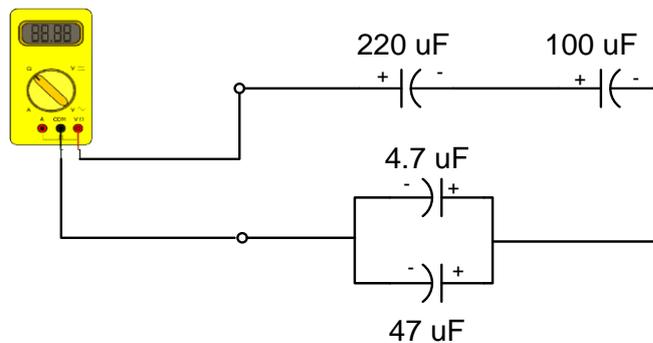
Figure 1.7 – Capacitor circuit connected in series and parallel, equivalent capacitor calculation
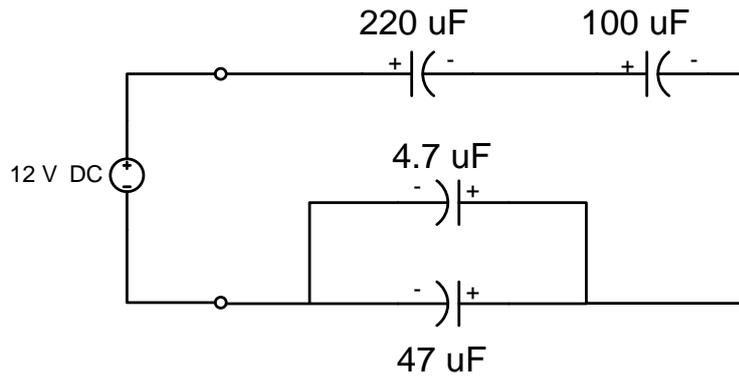
Figure 1.8 – Capacitors connected in series and parallel

Experiment Part 2 Steps

1. Calculate the equivalent capacitor value (C_T) in Figure 1.7 analytically and write the calculated value to suitable part of Table 1.4.
2. Implement the circuit given in Figure 1.7 by using breadboard. **Do not apply voltage to the circuit!!!**
3. As seen on the schematic, measure equivalent capacitor value of the circuit by Multimeter and fill the suitable gap on Table 1.4.
4. Apply 12V DC voltage to the terminals of the circuit as in Figure 1.8.
5. Measure voltage levels of each capacitor by Multimeter and write the measured values suitable parts of Table 1.5.
6. Compare the calculation, simulation and measurement results by considering the tolerance of used capacitors in the circuit.

Part 2 Results

Table 1.4 – Equivalent capacitor value observation part

	C_T (µF)
Calculated	
Measured	

Table 1.5 – Capacitors voltage levels observation part

	Capacitors Voltage Levels			
	$220\ \mu F$	$100\ \mu F$	$47\ \mu F$	$4.7\ \mu F$
Calculated				
Simulated				
Measured				

23

Part 2 Remarks

Experiment 2

Ohm's - Kirchhoff's Laws and AC Inductor-Resistor-Capacitor

Required Component List:

- Resistors   : 47 Ω, 100 Ω, 2.2 kΩ and 10 kΩ
- Inductors   : 100 mH
- Capacitors   : 1.5 nF

(PS: 1 piece per each component)

Experiment Instruments:

- Multimeter
- DC Power Supply
- Signal Generator
- Breadboard

Experiment Part 1: Ohm's and Kirchhoff's Laws

Purpose of Experiment:	Learning basic electrical circuit laws
Teaching of Experiment:	Basic circuit theories such as Ohm and Kirchhoff will be comprehended

Theoretical Information and Introduction to Part 1

Ohm's Law

The current flowing through closed loop electrical circuit is in direct proportion to voltage applied to circuit and inverse proportion to resistive load of the circuit.
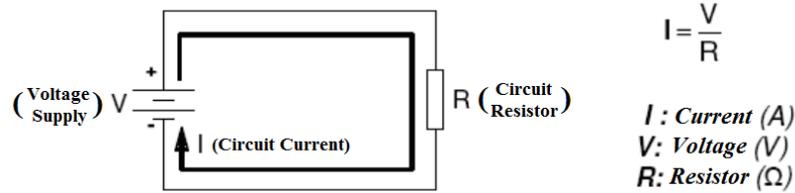

Figure 2.1 – Ohm's Law

On the other hand, power in an electrical circuit is equal to the multiplication of voltage and current as formulized below. The unit of power is Watt (W).

$$P = VxI$$

Ohm's Law says that the voltage and resistance are not affected by any change in the circuit, but the current changes with respect to the resistance and voltage. The relations between voltage, current, resistance and power are shown in the following table according to Ohm's law.

$V\ (Volt)$	IxR	$\sqrt{PxR}$	P/I
$R\ (Ohm)$	V/I	V^2/P	P/I^2
$I\ (Amper)$	V/R	P/V	$\sqrt{P/R}$
$P\ (Watt)$	VxI	V^2/R	I^2xR

Kirchhoff's Laws

In 1845, the German physicist Gustav Robert Kirchhoff developed two laws, namely current and voltage laws.

Current Law

The sum of the currents flowing through the parallel connected resistors equals to the main-branch current at the node of the parallel resistors.
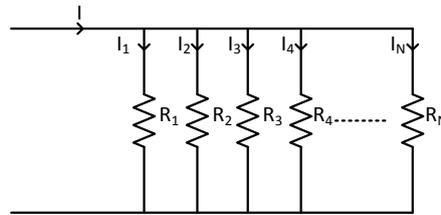
Figure 2.2 – Kirchhoff's Current Law

$$I = I_1 + I_2 + I_3 + \cdots + I_N$$

In a closed loop electrical circuit, the total charge flowing into a node must be the same as the total charge flowing out of the node.

$$I_{in} = I_{out}$$
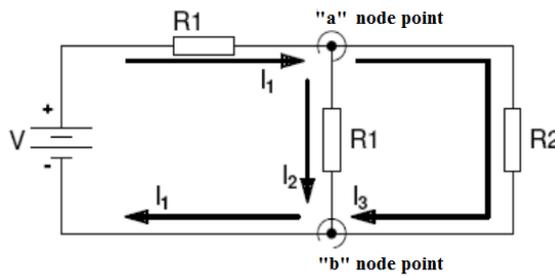
Figure 2.3 – Kirchhoff's Current Law

Voltage Law

In a closed loop electrical circuit, the total voltage spent on serial elements of the circuit must be equal to voltage applied to the circuit.
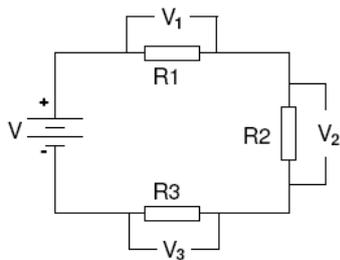
$$V = V_1 + V_2 + V_3$$

Figure 2.4 – Kirchhoff's Voltage Law

Preliminary Work – 1

Calculate the equivalent resistor value (R_T); shown current values I_1, I_2 and I_3; indicated voltage levels V_1, V_2, V_3 and V_4 of given circuit on Figure 2.5 analytically, and measure them by simulating the same circuit using Multisim. Fill the suitable parts of Table 2.1.

Experimental Procedures for Part 1
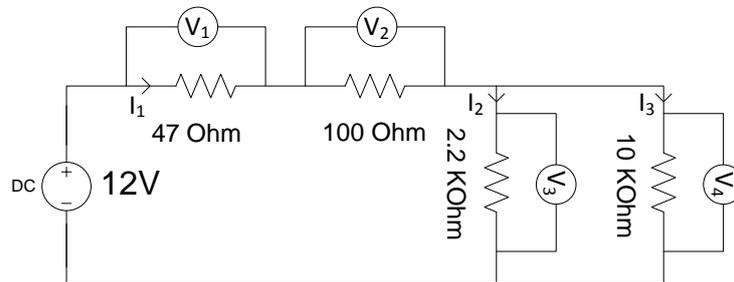
Part 1 Schematic

Experiment Part 1 Steps

1. Implement the circuit given in Figure 2.5 by using breadboard. <u>Do not connect the power supply.</u>
2. Using Multimeter, measure equivalent resistor value of the circuit.
3. Connect the power supply to the circuit as shown in Figure 2.5 and apply DC 12V.
4. Measure the currents I_1, I_2 and I_3 which are shown on schematic.
5. Measure the voltages V_1, V_2, V_3 and V_4 which are shown on schematic.
6. Write obtained values to the suitable parts of Table 2.1.

Part 1 Results

Table 2.1 - Observation table of Part 1

	R_T	I_1	I_2	I_3	V_1	V_2	V_3	V_4
Calculated								
Simulated								
Measured								

Part 1 Remarks

_____
_____
_____
_____
_____
_____
_____
_____
_____

Experiment Part 2: AC Resistor-Capacitor-Inductor

Purpose of the Experiment:	Learning behaviors of the basic circuit components such as inductor, capacitor and resistor under AC voltage.
Teaching of the Experiment:	Inductor structure and behaviors of the resistor, capacitor and inductor under AC voltage will be comprehended.

Theoretical Information and Introduction to Part 2

The resistive load of capacitors or inductors under AC signal is defined as **reactance**. Reactance occurs with the applied voltage in the capacitors, and in proportion to the current generated by the voltage applied to the coils. Reactance symbol is "*X*" and its unit is Ohm (Ω). For any signal with frequency f, ($V_{in} = V sin(wt)$),

Capacitive reactance, X_C (for capacitor C):

$$i = C \frac{dV_{in}}{dt}$$

$$X_C = \frac{1}{2\pi f C}$$

Inductive reactance, X_L (for inductor L):

$$V_{in} = L \frac{di}{dt}$$

$$X_L = 2\pi f L$$

relations can be used to calculate reactance. If the amplitude of AC signal is V_s and it is applied to an inductor or capacitor whose reactance is X_e, then the **amplitude of the current** will be calculated as:

$$I_s = V_s / X_e$$

Similarly, if the effective value of AC signal is V_{RMS} and it is applied to an inductor or capacitor whose reactance is X_e , then the **RMS value of the current** can be calculated as:

$$I_{RMS} = V_{RMS} / X_e$$

Preliminary Work – 2

For the circuits given on Figure 2.6, 2.7 and 2.8, apply the AC analysis and find the RMS (effective) values of AC currents theoretically. Simulate the same circuits by Multisim and measure the RMS current and voltage values of resistor, inductor and capacitor. Write the calculation results and the simulation measurements to the suitable parts of the tables given in "Part 2 Results".

Experimental Procedures for Part 2

Part 2 Schematics
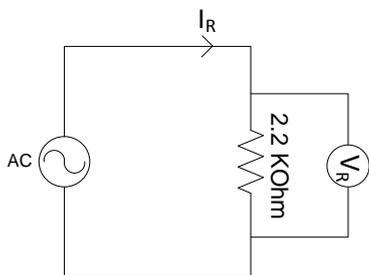

Figure 2.6 - Resistor in AC circuit
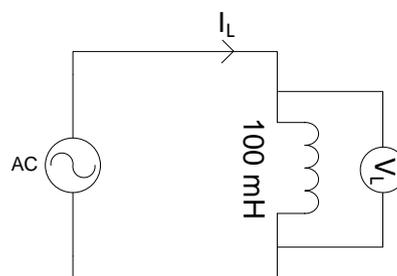

Figure 2.7 – Inductor in AC circuit
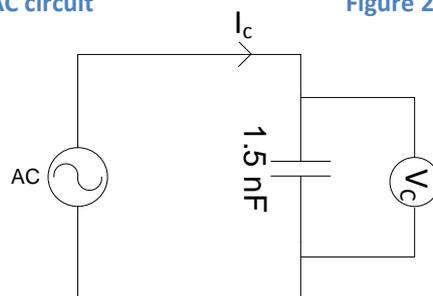
Figure 2.8 – Capacitor in AC circuit

Experiment Part 2 Steps

1- Implement the circuits given in Figure 2.6, 2.7 and 2.8 by using breadboard.
2- Apply 5V (RMS) AC sinusoidal signal to the circuit. (Set the 5V RMS value when the circuit is under load). The frequency of the applied AC signal should be set according to the desired frequencies in the tables below (5 kHz, 25 kHz and 50 kHz).
3- Measure the current and voltage values of the circuit for each frequency, and fill the suitable parts of the tables given in experiment results in terms of RMS values.

Part 2 Results

Table 2.2 – Resistor in AC circuit observation table

		5kHz	25kHz	50kHz
Calculated	V_R			
	I_R			
	$R = V_R/I_R$			
Simulated	V_R			
	I_R			
	$R = V_R/I_R$			
Measured	V_R			
	I_R			
	$R = V_R/I_R$			

Table 2.3 – Inductor in AC circuit observation table

		5kHz	25kHz	50kHz
Calculated	V_L			
	I_L			
	$R = V_L/I_L$			
Simulated	V_L			
	I_L			
	$R = V_L/I_L$			
Measured	V_L			
	I_L			
	$R = V_L/I_L$			

Table 2.4 – Capacitor in AC circuit observation table

		5kHz	25kHz	50kHz
Calculated	V_C			
	I_C			
	$R = V_C/I_C$			
Simulated	V_C			
	I_C			
	$R = V_C/I_C$			
Measured	V_C			
	I_C			
	$R = V_C/I_C$			

Part 2 Remarks

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

Experiment 3

Series and Parallel RLC Circuits
Resonance Circuits

Required Component List:

- Resistors   : 220 Ω and 10 Ω
- Capacitors   : 100 nF
- Inductors   : 100mH and 100 μH

(PS: 1 piece per each component)

Experiment Instruments:

- Multimeter
- Oscilloscope
- Signal Generator
- Breadboard

Experiment Part 1: Series and Parallel RLC Circuits

|	
Purpose of the Experiment:	Learning the working principles of RLC circuits
Teaching of the Experiment:	By comparing, parallel and serial RLC circuits will be comprehended.

Theoretical Information and Introduction

The resistive effect arising from RLC circuits due to the resistances and reactance is called as **impedance.** Impedance is symbolized by the letter "Z", and its unit is Ohm (Ω). Mathematically, the impedance can be represented in complex and polar forms.

The polar form shows both amplitude and phase characteristics as follows.

$$Z = |Z|e^{j\theta}$$
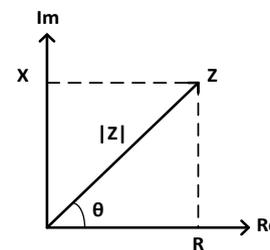
The amplitude $|Z|$ represents the ratio of the voltage amplitude difference to the current amplitude when the phase difference between voltage and current is θ. In complex form, the impedance is expressed as:

$$Z = R + jX$$

Given equation above, the letter "R" symbolizes resistor of impedance and "X" symbolizes reactance of impedance which is expressed in imaginary part. Ohm's Law using for resistor in DC circuits is also valid for impedance in AC circuits.

$$V = I.Z; \;\; |V| = |I||Z|$$

| Component	Impedance
Resistor (R)	$Z_R = R$
Inductor (L)	$Z_L = jX_L \, , X_L = 2\pi f L$
Capacitor (C)	$Z_c = -jX_c \, , X_c = 1/2\pi f C$

As Ohm's Law, Kirchhoff's Laws can also be used in AC circuit analysis.

Equivalent impedance for serial impedances;
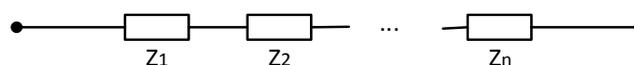
$$Z_{eq} = Z_1 + Z_2 + \cdots + Z_n$$

Equivalent impedance for parallel impedances:
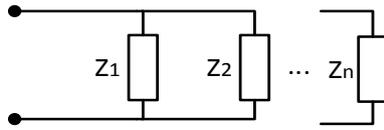
$$\left(\frac{1}{Z_{eq}}\right)^{-1} = \left(\frac{1}{Z_1} + \frac{1}{Z_2} + \cdots + \frac{1}{Z_n}\right)^{-1}$$

The equivalent impedance of series RLC circuit given below can be calculated using the equivalent impedance calculation formulas of series and parallel connected circuits.

$$Z_{eş} = \sqrt{R^2 + (X_L - X_c)^2}$$

$$X_L = 2\pi f L, X_C = \frac{1}{2\pi f C}$$
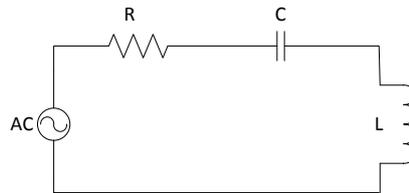

<div align="center">Figure 3.1 – RLC circuit</div>

The impedance of this circuit will be minimum when $X_L = X_C$ equality is satisfied.

$$X_L - X_C = 2\pi f L - \frac{1}{2\pi f C} = 0$$

In serial RLC circuits, the current in the circuit reaches its maximum value at resonance frequency since the minimum impedance is observed. When the frequency is solved in the last equation, "resonance frequency" is obtained.

$$\text{Resonance frequency: } f_0 = \frac{1}{2\pi\sqrt{LC}}$$

The AC circuits are classified with respect to the phase difference between the voltage and current. If phase difference is zero, then the circuit called as resistive. When the phase difference is equal to positive 90° or negative 90°, the circuit will be inductive and capacitive respectively. In more detail;

If we assume that the input voltage is $V_{in} = V_{max}\sin(wt)$, and calculate the voltage on each component of series RLC circuit in Figure 3.1;

$$V_R(t) = \frac{R}{Z_{eş}}V_{max}\sin(wt)$$

- The resistor does not generate any phase change in the voltage signal drop on it.

If the voltage on the resistor and the whole circuit are compared, it is seen that there is a phase difference. The mathematical expression of this phase shift is

$$\emptyset = \tan^{-1}(\frac{X_L - X_C}{R})$$

As one can see, if the capacitor is not connected to the circuit, this leads to a positive phase angle in RL circuit and the current lags the voltage. Conversely, if the inductor is not connected to the circuit, the phase angle is negative in the RC circuit and the current leads the voltage.

The expression of the voltage on capacitor is given below, and this voltage is delayed 90° according to the voltage on the resistor.

$$V_C(t) = \frac{X_C}{Z_{eş}} V_{max} \sin\left(wt - \frac{\pi}{2}\right)$$

The expression of the voltage on inductor added below can be found using similar method, and this voltage leads the voltage on the resistor by 90°.

$$V_L(t) = \frac{X_L}{Z_{eş}} V_{max} \sin\left(wt + \frac{\pi}{2}\right)$$

Figure 3.2 shows the voltage signals on the resistor, inductor and capacitor.
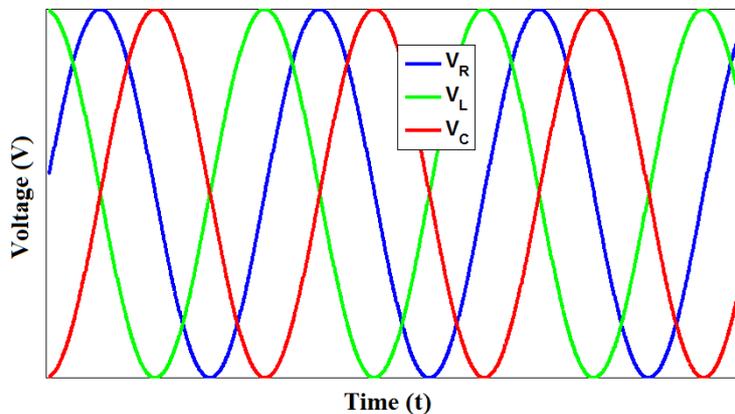
Figure 3.2 – The voltage generated on the resistor, inductor and capacitor by AC signal source

Preliminary Work – 1

Examine the circuit schematics given on Figure 3.3 and Figure 3.4. Calculate analytically the voltage values of inductor, resistor and capacitor for serial RLC circuit, and the absolute values of the currents for inductor, resistor and capacitor for parallel RLC circuit. Using computer program, find the same voltage and current values. Write obtained theoretical values and simulation results to the suitable parts of Table 3.1 and Table 3.2.

Experimental Procedures for Part 1
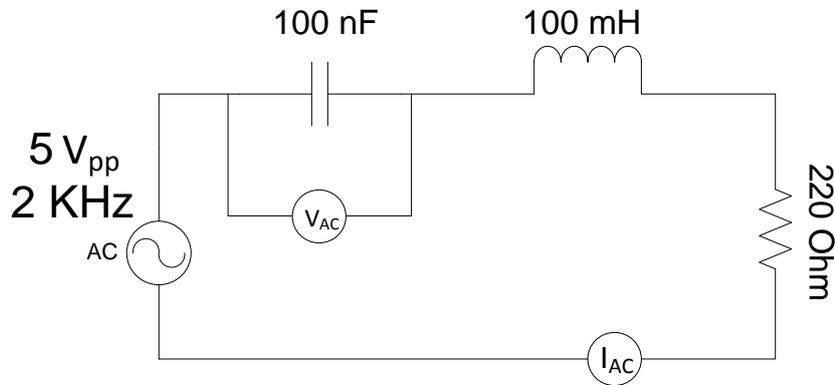
Part 1 Schematics

Figure 3.3 – Schematic for series RLC circuit
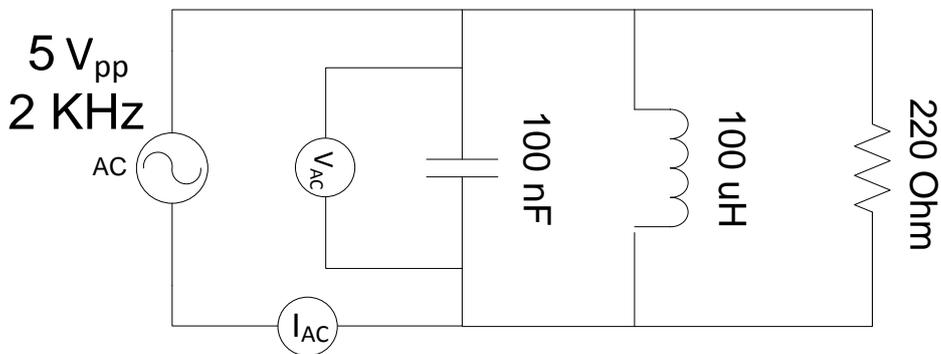
Figure 3.4 – Schematic for parallel RLC circuit

Experiment Part 1 Steps

For series RLC circuit,

1. Implement the circuit given in Figure 3.3 by using breadboard.
2. Connect the signal generator to the input of the circuit and apply 5Vpp / 2 kHz sinusoidal signal.
3. Measure desired values in Table 3.1 and write in terms of effective values.

For parallel RLC circuit,

1. Implement the circuit given in Figure 3.4 by using breadboard.
2. Connect the signal generator to the input of the circuit and apply 5Vpp / 2 kHz sinusoidal signal.
3. Measure desired values in Table 3.2 and write in terms of effective values.

Part 1 Results

Use effective values of the results to fill the tables.

Table 3.1 – Voltage observation table of series RLC circuit

	I	V_R	V_L	V_C
Calculated				
Simulated				
Measured				

Table 3.2 – Current observation table of parallel RLC circuit

	I	I_R	I_L	I_C
Calculated				
Simulated				
Measured				

Part 1 Remarks

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

Experiment Part 2: Examination of Resonance Circuits

Purpose of the Experiment:	Investigation of series and parallel resonance circuits working principles
Teaching of the Experiment:	Types and working principles of resonance circuits will be comprehended.

Theoretical Information and Introduction to Part 2

When capacitive and inductive reactance of RLC circuits driven by AC source is equal to each other ($X_C = X_L$) resonance effect occurs. The frequency seen resonance effect is called as *resonance frequency*.

The circuits to which the resistors, inductors and capacitors are connected in series are called series RLC circuits as in Figure 3.1. The total impedance of this circuit is calculated by the following equation.

$$Z_T = R + j(X_L - X_C)$$

If $X_L - X_C$ reactance value is equal to zero for any f_0 frequency value and the total impedance value of the circuit is the same as the resistance value found in the circuit. This is called series resonance and the frequency f_0 is **series resonance frequency** which can be calculated as:

$$X_L - X_C = 0 \longrightarrow X_L = X_C \longrightarrow 2\pi f_0 L = 1/2\pi f_0 C \longrightarrow f_0 = \frac{\omega_0}{2\pi} = \frac{1}{2\pi\sqrt{LC}}$$

At the resonance frequency (f_0), since the impedance of series RLC circuit and the resistor value are equal to each other, the impedance is at its minimum and the current in the circuit reaches correspondingly the maximum value. There is no phase difference between the current and voltage. In more detail, the total voltage on inductor and capacitor is equal to zero in series RLC circuits since the voltage on inductor and capacitor are the same size but opposite direction at the resonance frequency. Therefore, the maximum current is drawn from the circuit expressed as $I = V_{in}/R$. Frequency-impedance and frequency-current graphs of the series RLC circuits are indicated in Figure 3.5.
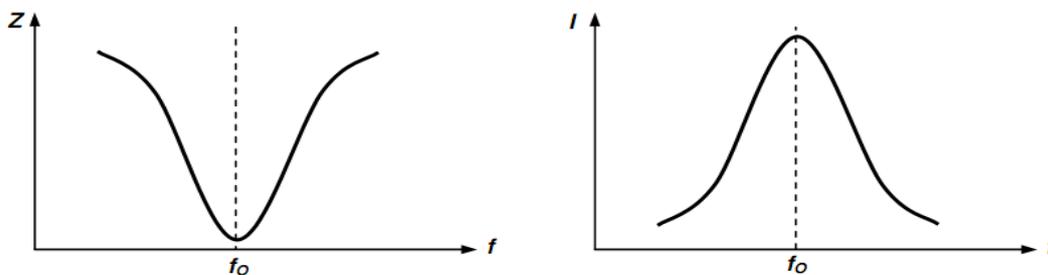

Figure 3.5 – Impedance-frequency and current-frequency relations of serial RLC circuits

Parallel resonance circuits are circuits to which the L and C elements are connected in parallel as shown in Figure 3.4. In these circuits, the impedances of the capacitor and inductor are also zero for an f_0 frequency value and the total impedance of the circuit depends entirely on the resistive impedance characteristic. This frequency at which the reactive (capacitor and inductor impedance) term is zero is called resonance frequency of the parallel resonance circuit, and it can be calculated as below.

$$f_0 = \frac{\omega_0}{2\pi} = \frac{1}{2\pi\sqrt{LC}}$$

At the resonance frequency f_0, it is observed that the currents of parallel connected capacitor and inductor are eliminate each other since they have the same value but opposite phase, and output voltage reaches its maximum level. In parallel RLC circuits, the current flowing on the circuit takes its minimum value since the total impedance of the circuit reaches its maximum level. Frequency-impedance and frequency-current graphs of the parallel RLC circuits are indicated in Figure 3.6.
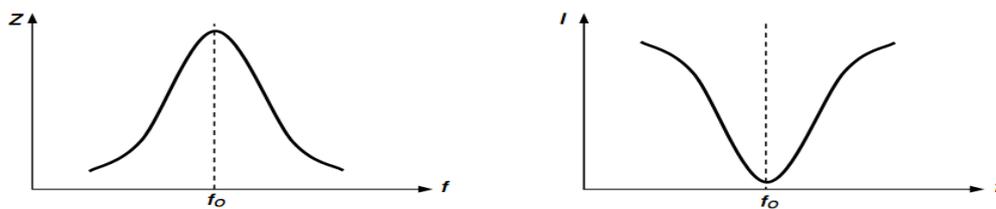

Figure 3.6 – Impedance-frequency and current-frequency relations of parallel RLC circuits

Preliminary Work – 2

Calculate analytically the resonance frequencies of given series and parallel RLC circuits on Figure 3.7 and 3.8. Measure the same frequencies by simulating the circuits in any simulation environment. In order to calculate by simulation apply different frequency inputs and find the maximum output voltage V_O for series and minimum output voltage for parallel whose frequency is resonance frequency.

Experimental Procedures for Part 2
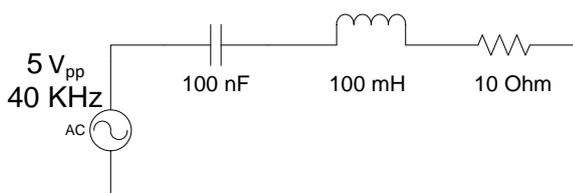
Part 2 Schematics


Figure 3.7 – Series RLC circuit schematic
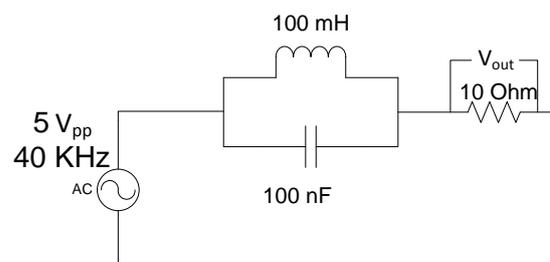
Figure 3.8 – Parallel RLC circuit schematic

43

Experiment Part 2 Steps

1. Implement the series RLC circuit given in Figure 3.7 by using breadboard.
2. Apply 5 Vpp / 40 kHz sinusoidal signal to the input of the circuit.
3. Find the maximum output voltage by tuning input voltage frequency and write the obtained result to suitable part of Table 3.3.
4. Implement the parallel RLC circuit given in Figure 3.8 by using breadboard.
5. Apply 5 Vpp / 40 kHz sinusoidal signal to the input of the circuit.
6. Find the minimum output voltage by tuning input voltage frequency and write the obtained result to suitable part of Table 3.3.

Part 2 Results

Table 3.3 – Resonance frequency observation table for series and parallel RLC circuits

	Resonance Frequency for Series RLC Circuit	Resonance Frequency for Parallel RLC Circuit
Calculated		
Simulated		
Measured		

Part 2 Remarks

Experiment 4

Investigation of Diode Characteristics

Required Component List:

- Resistor   : $1 \times 220\ \Omega$ / 1W
- Potentiometer   : $1 \times 1\ k\Omega$
- Diode   : $1 \times$ 1N4001

Experiment Instruments:

- Breadboard
- Multimeter
- DC power supply

Experiment: Investigation of Diode Characteristics

Purpose of the Experiment:	It is aimed to obtain the diode characteristics through experiments.
Teaching of the Experiment:	Understanding and observing the relationship between current and voltage of diodes

Theoretical Information and Introduction

Diode is an electrical component which allows an electric current to pass in one direction. The diodes are obtained by combining two semiconductors of type P and N. The pole P is called the "Anode" and the pole N is called the "Cathode". Diodes are one of the most frequently used elements in electronic circuits thanks to their many uses such as switching and rectification.
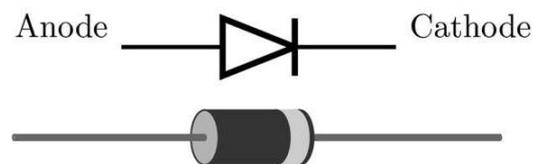

Figure 4.1 – Diode symbol on circuits and appearance

Diodes are generally fabricated from semiconductor materials such as silicon (Si) and germanium (Ge). These diodes have specific threshold voltages of 0.7 V for silicon and 0.3 V for germanium. As mentioned in Analog Electronics course, the internal resistance and threshold voltage values are necessary parameters for the correct modelling approach. These parameters are assumed to be zero for ideal diodes. A diode has low resistance in one direction (ideally zero), and high in the other (ideally infinite).

Current-voltage relationship of the diode can be written as:

$$I_D = I_S \left(e^{\frac{V_D}{nV_T}} - 1 \right)$$

where I_S is reverse bias saturation current (2.550 nA for 1N4007), V_D is diode voltage, I_D is diode current, n is quality factor and V_T is thermal voltage. Thermal voltage V_T is approximately 26 mV at room temperature.

The diodes have two connection types, either forward or reverse biasing, depending on the mode of connection to the circuit. The diode turns on and allows current to flow when anode (P pole) and cathode (N pole) are connected to the positive and negative terminals of the voltage source respectively. For forward biasing mode, the diode does not flow current until reaching threshold value (in reality a little current flows), and when the threshold value is exceeded, it begins to flow rapidly. In the case of reverse biasing, the anode of the diode is connected to the negative terminal of the voltage source, and the cathode of the diode is

connected to the positive terminal. The diode does not allow current to pass through and behaves like an isolator. However, when the reverse voltage is applied to the diode, still a current flows at very low level even though the diode is in insulation. It is called **leakage current** (reverse saturation current). This is an unwelcome situation. The amount of leakage current depends on the semiconductor material used to produce the diode.

Diodes may fail due to the reasons such as over current, ambient temperature, connection errors and sudden voltage increase. To test the diodes using multimeter, first set multimeter to the part where the diode symbol is located and then touch to the diode poles with the multimeter probes by forward biasing. The voltage drop on the diode is measured between 0,2 V and 0,95 V in this direction. If the same procedure is followed for reverse biasing mode, the measurement result will be around 0 V. In this case, it is concluded that the diode is good. Otherwise, the tested diode is faulty (bad or shorted).

Preliminary Work

Set circuits in Figure 4.2 and 4.3 in a simulation program. By turning potentiometer knob from minimum to maximum with 10 steps, measure the diode current I_D and the diode voltage V_D. Calculate the quality factor n for different current and voltage values in corresponding tables (I_S is 2.550 nA). Record the achieved results to Table 4.1 and Table 4.2.

Experimental Procedures

Schematics
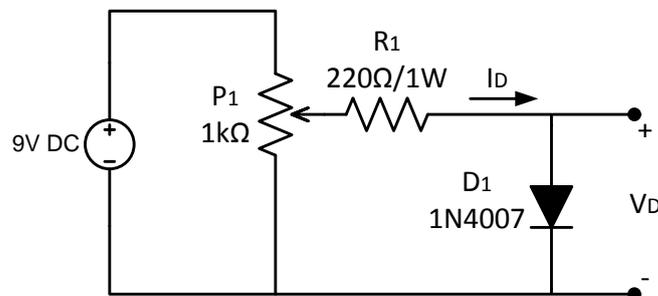
Figure 4.2 – Forward bias circuit
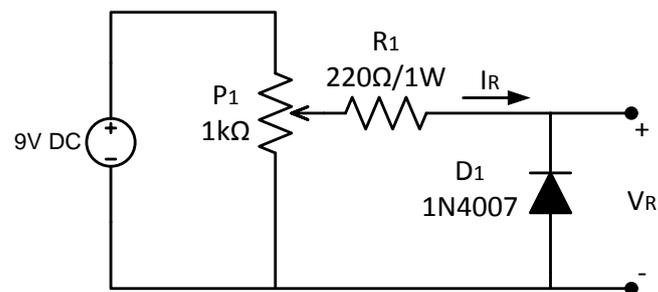
Figure 4.3 – Reverse bias circuit

Experiment Steps

1. Implement the circuit in Figure 4.2 on breadboard.
2. By turning the knob of potentiometer P_1 from min. to max. with 10 steps, measure the diode voltage V_F and current I_F. Record them in Table 4.1. Plot the diode forward bias characteristic on Figure 4.4 with respect your measurement values.
3. Implement the circuit in Figure 4.3 on breadboard.
4. By turning the knob of potentiometer P_1 from minimum to maximum with 10 steps, measure the diode voltage V_R and current I_R. Record them in Table 4.2. Plot the diode reverse bias characteristic on Figure 4.4 with respect your measurement values.
5. According to your measured voltage and current values (V_F, I_F, V_R, I_R), calculate the quality factor n and record them in Table 4.1 and Table 4.2.

Experiment Results

Table 4.1 – Current-Voltage relationship observation table for forward biased diode

		Forward Polarity									
Simulation	V_F										
	I_F										
	n										
Measurement	V_F										
	I_F										
	n										

Table 4.2 – Current-Voltage relationship observation table for reverse biased diode

		Reverse Polarity									
Simulation	V_R										
	I_R										
	n										
Measurement	V_R										
	I_R										
	n										
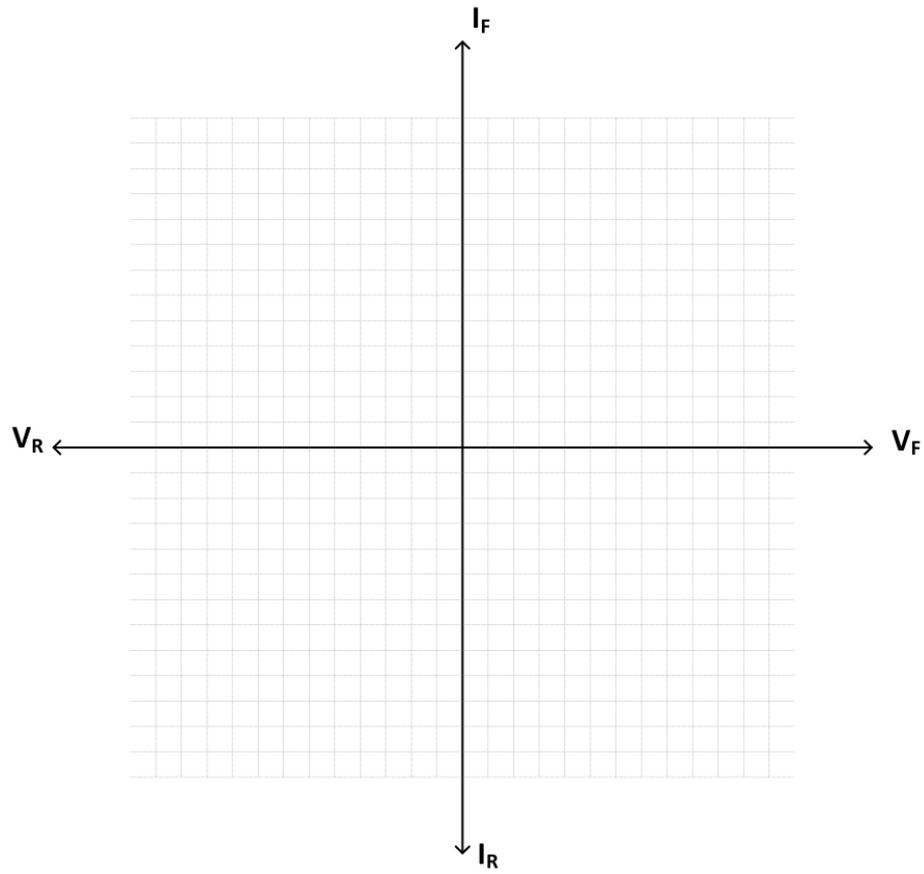
Figure 4.4 – Current-Voltage axes grid for diode characteristic

Remarks

Experiment 5

Diode Rectifier Applications

Required Component List:

- Resistor   : 1×3 kΩ
- Capacitor   : 1×100 µF
- Diode   : 5×1N4001

Equipment

- Multimeter
- Oscilloscope
- Signal Generator
- Breadboard

Experiment: Rectifier Applications

Purpose of the Experiment:	Learning rectifier applications using diodes
Teaching of the Experiment:	To be able to comprehend working principles of diodes

Theoretical Information and Introduction

In many electronic applications, the 220V 50 Hz AC signal received from the electric network needs to be converted to DC power. This process is generally accomplished by an electrical circuit including mainly transformers, diodes, voltage regulators, and such circuits are called "rectifier circuits". Diode rectifier circuits are divided into two parts as half-wave and full-wave rectifiers. DC power supplies and AC motor drives are examples of major uses for diode rectifier circuits.

The rectifier circuits where only the positive or negative alternation of the input AC signal is rectified are called *half-wave rectifier circuits*. The half-wave rectifier circuit allows only the positive (negative) portion of the sinusoidal signal at the input to pass, since the diode does not transmit the negative (positive) voltage on it depending on the its position. In other words, it only one alternance of the input signal is rectified. The peak value of the voltage achieved at the output of the rectifier is the same as the peak value of the input signal if the voltage drop on the diode is neglected. When a sinusoidal input signal is applied to the half-wave rectifier circuit, the average DC voltage at the output is calculated as follows:

$$V_{DC} = \frac{V_p}{\pi}$$

The circuits in which both the positive and negative alternance of the AC input are rectified are called *full-wave rectifier circuits*. Such rectifiers can be designed using 2 or 4 diodes. Thus, a signal with frequency 2 times the frequency of the input signal and located in the positive region will be obtained at the output. For a sinusoidal input signal, the average DC voltage at the output for the full-wave rectifier is calculated as follows,

$$V_{DC} = \frac{2V_p}{\pi}$$

The signals achieved at the outputs of both the half-wave rectifier and the full-wave rectifier for an input sinusoidal signal are clearly seen in Figure 5.1.
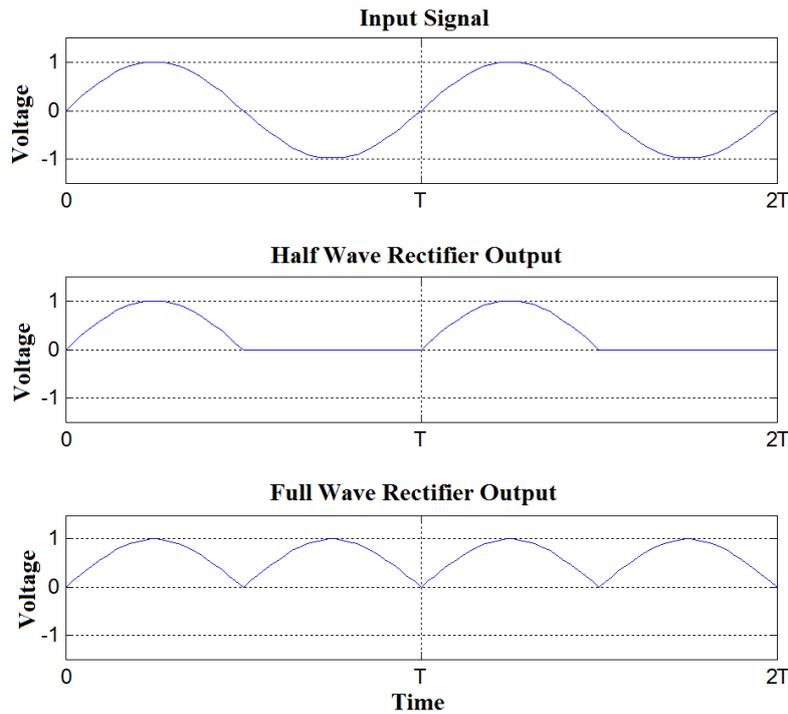
Figure 5.1 – Output signals of half wave and full wave rectifiers

Experimental Procedures

Preliminary Work

Calculate analytically the output DC average voltages of both half-wave and full-wave rectifier circuits given below. Simulate the same circuits given in Figure 5.2, 5.3, 5.4 and 5.5. Measure DC output voltage values via voltmeter. Record the results to Table 5.1.

Schematics
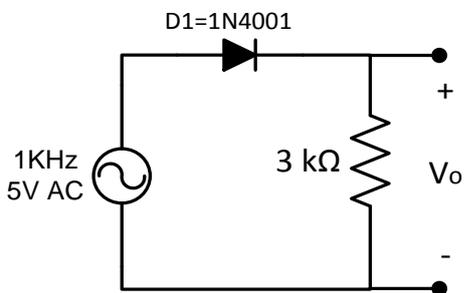
**Figure 5.2 – Unfiltered half wave rectifier**
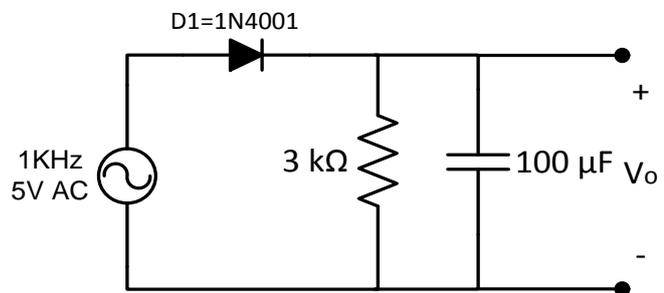
**Figure 5.3 – Filtered half wave rectifier**
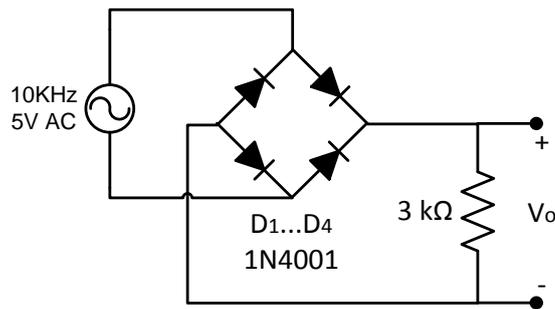
**Figure 5.4 – Unfiltered full wave rectifier**
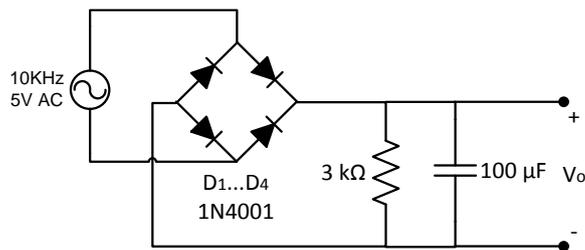
Figure 5.5 – Filtered full wave rectifier

Experiment Steps

Half Wave Rectifier Experiment

1. Implement the circuit given in Figure 5.2.
2. Apply 1 kHz 5V sinusoidal signal as the input of the circuit.
3. Plot the output signal of the circuit to Figure 5.6 using oscilloscope.
4. Measure the DC voltage at the output of the circuit with a voltmeter and hold the result in Table 5.1.
5. Install the capacitor as shown in Figure 5.3.
6. Plot the output signal of the circuit to Figure 5.7 using oscilloscope.
7. Measure the DC voltage at the output of the circuit with a voltmeter and hold the result in Table 5.1.

Full Wave Rectifier Experiment

1. Implement the circuit given in Figure 5.4.
2. Apply 10 kHz 5V sinusoidal signal as the input of the circuit.
3. Plot the output signal of the circuit to Figure 5.8 using oscilloscope.
4. Measure the DC voltage at the output of the circuit with a voltmeter and hold the result in Table 5.1.
5. Install the capacitor as shown in Figure 5.5.
6. Plot the output signal of the circuit to Figure 5.9 using oscilloscope.
7. Measure the DC voltage at the output of the circuit with a voltmeter and hold the result in Table 5.1.

Experiment Results

Table 5.1 – DC Output voltage levels observation table for different rectifier circuits

	DC Output Voltage Levels			
	Half Wave		Full Wave	
	Unfiltered	Filtered	Unfiltered	Filtered
Calculated				
Simulated				
Measured				✕
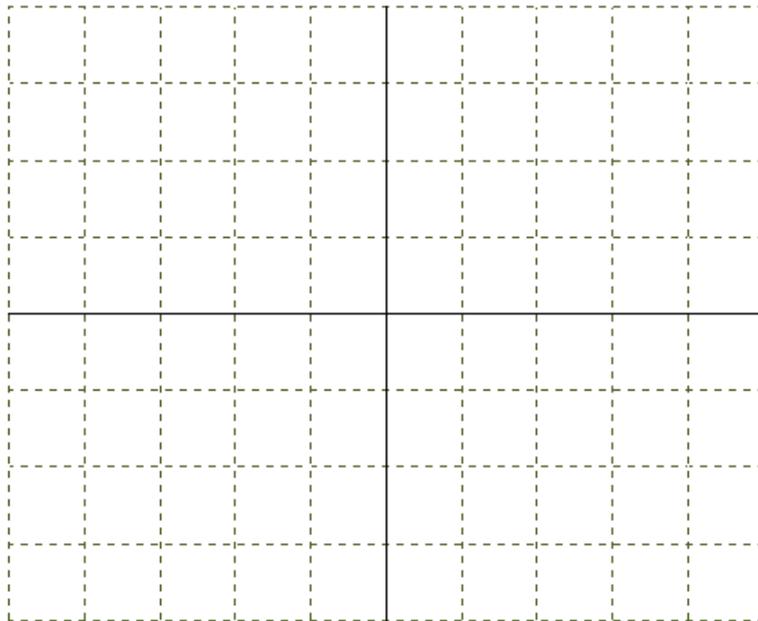
$$\frac{Volt}{Div} =$$

$$\frac{Time}{Div} =$$

Figure 5.6 – Output signal of unfiltered half wave rectifier circuit
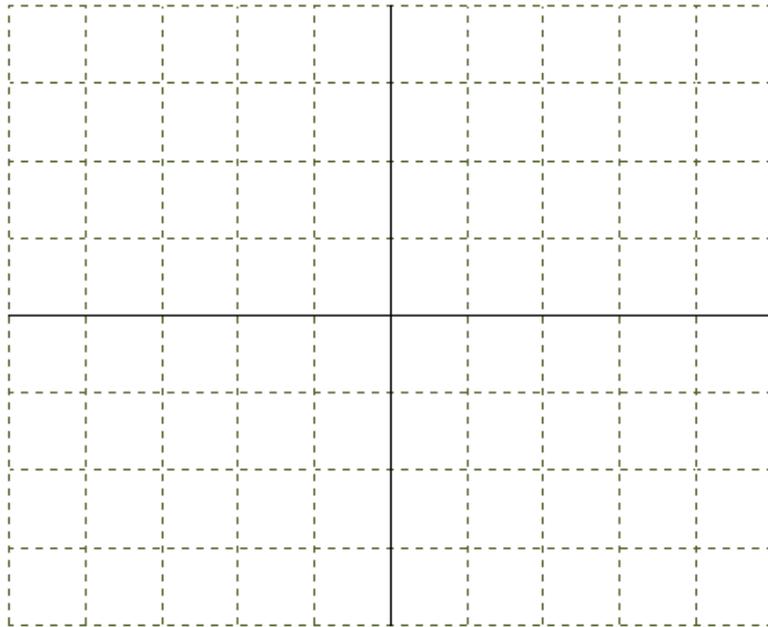
$$\frac{Volt}{Div} =$$

$$\frac{Time}{Div} =$$

Figure 5.7 – Output signal of filtered half wave rectifier circuit
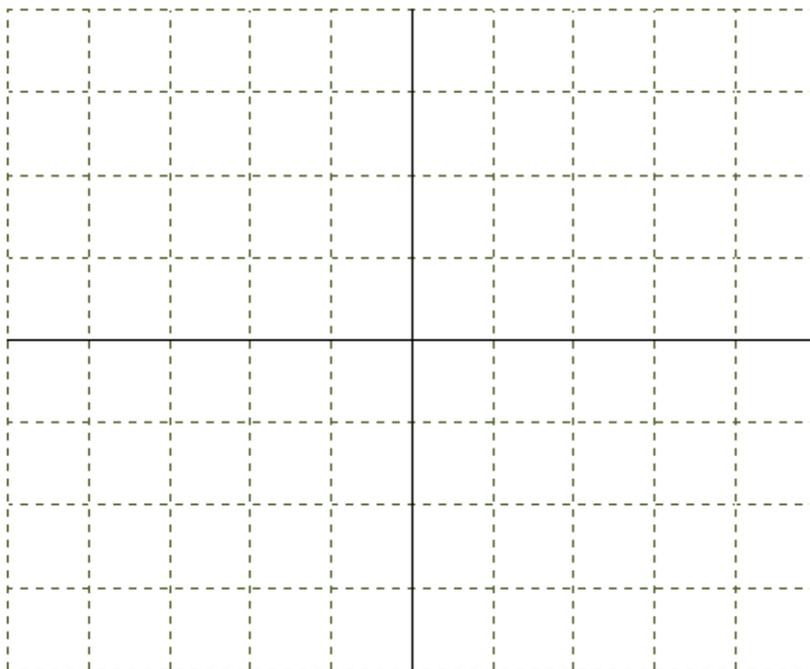
$$\frac{Volt}{Div} =$$

$$\frac{Time}{Div} =$$

Figure 5.8 – Output signal of unfiltered full wave rectifier circuit
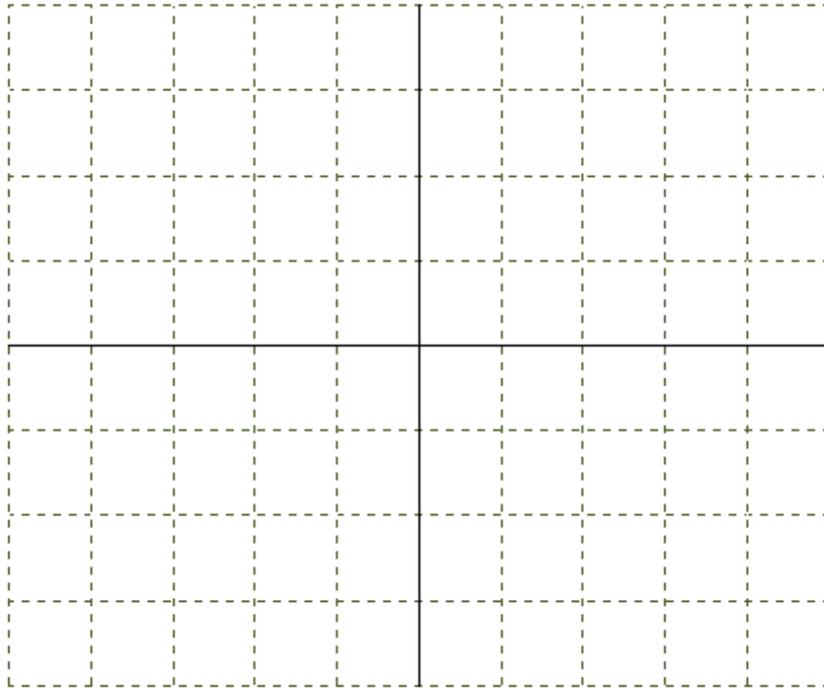
$$\frac{Volt}{Div} =$$

$$\frac{Time}{Div} =$$

Figure 5.9 – Output signal of filtered full wave rectifier circuit

Remarks

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

Remarks

Experiment 6

Zener Diode Characteristic and Applications

Required Component List:

- Resistors : 1 ×220 Ω / 1W, 1 × 56 Ω / 1W
- Potentiometer : 1 × 1 kΩ
- Zener Diode : 2 × 5.6 V
- Transistor : 1 × BD139

Equipment

- Multimeter
- Oscilloscope
- Signal Generator
- Breadboard

Experiment Part 1: Zener Diode Characteristic and Applications

Purpose of the Experiment:	Obtain Zener diode characteristics via experiments and build a regulator using Zener diode
Teaching of the Experiment:	Being able to comprehend the current-voltage relationship of the Zener diode and working principles of the Zener diode

Theoretical Information and Introduction to Part 1

Zener diodes are a special kind of semiconductor diodes, which, contrary to normal diodes, allow current to flow not only in one direction but also in other direction when the required voltage level is exceeded. Forward biased Zener diodes behave like a rectifier diode with 0.7 V threshold voltage level.

Zener diodes are used for purposes such as protection in circuits, generation of reference voltage and voltage regulation. Figure 6.1 shows a Zener diode and its symbol in a circuit.

Figure 6.1 – Zener diode

Such diodes are circuit elements with PN-junction and designed to operate in the breakdown region under reverse bias. Under reverse biasing, they create the constant voltage region and this voltage value is called the "breakdown voltage".
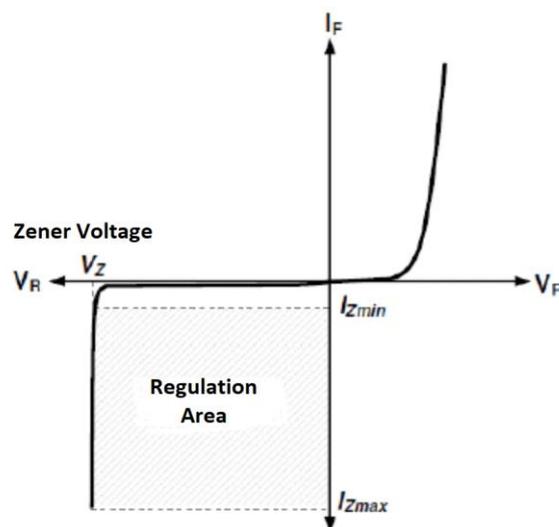

Figure 6.2 – Zener diode characteristic curve

There is a difference in the reverse bias region between Zener diode and an ordinary diode. An ordinary diode stays as open circuit until maximum reverse bias voltage. The zener diode, however, starts transmission at the Zener breakdown voltage (V_z) in this region. The voltage drop on Zener diode remains constant. The breakdown voltages of Zener diodes vary depending on admixture ratio of the PN junction, and Zener diodes with the breakdown voltages between 1.8 V and 200 V are generally used.

As can be easily observed from the characteristic curve, when the reverse bias voltage exceeds the Zener voltage, the diode starts to pass high current. For this reason, the Zener diodes in a circuit must be used with a resistor to limit the current flowing through and to prevent damage. The resistor to be used must allow at least 5mA of current (I_{zmin}) to pass through the Zener diode and at the same time must compensate the load current. Another point to consider is that the amount of power to be dissipated on the resistor and the Zener diode must not exceed the limit values.

Preliminary Work – 1

Simulate the circuits including the forward and reverse bias of the Zener diode shown in Figure 6.3 and Figure 6.4 and record the achieved results in Table 6.1 and Table 6.2.

## Experimental Procedures for Part 1

### Part 1 Schematics
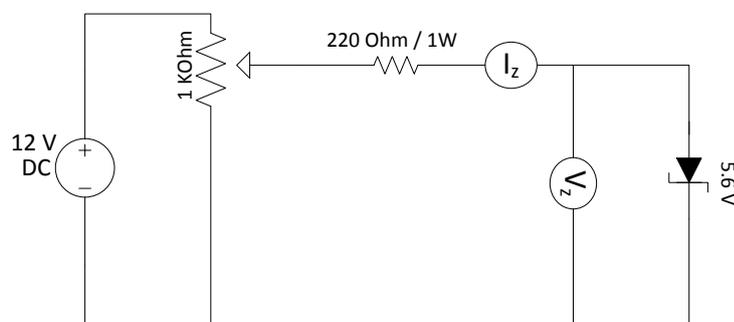
Figure 6.3 – Forward biased circuit
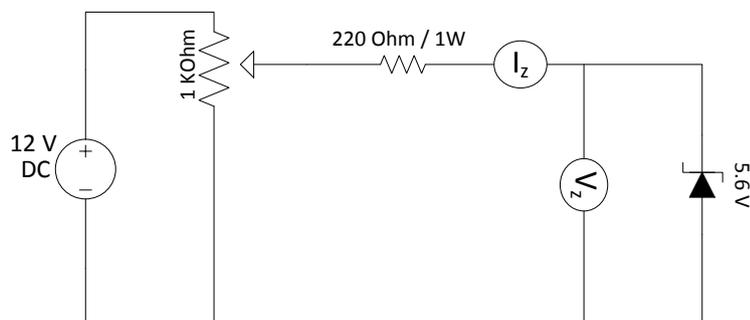
Figure 6.4– Reverse biased circuit

Experiment Part 1 Steps

1. Implement the circuit in Figure 6.3 using breadboard.
2. Set the potentiometer shaft from minimum to maximum at regular intervals, and record the forward biasing voltage and current values of Zener diode to Table 6.1.
3. Implement the circuit in Figure 6.4 using breadboard.
4. Set the potentiometer shaft from minimum to maximum at regular intervals, and record the reverse biasing voltage and current values of Zener diode to Table 6.2.
5. Plot the Zener diode characteristic to Figure 6.5 using data in Table 6.1 and Table 6.2.

Part 1 Results

Table 6.1 – Observation table for forward biased Zener diode

		Forward Bias							
Simulation	V_Z (V)	0	0.3	0.5	0.6	0.65	0.7	0.75	0.8
	I_Z								
Measurement	V_Z (V)	0	0.3	0.5	0.6	0.65	0.7	0.75	0.8
	I_Z								

Table 6.2 – Observation table for reverse biased Zener diode

		Reverse Bias								
Simulation	V_Z (V)	6	5.9	5.8	5.7	5.6	5.5	5	2.5	0
	I_Z									
Measurement	V_Z (V)	6	5.9	5.8	5.7	5.6	5.5	5	2.5	0
	I_Z									

Figure 6.5 – Zener diode characteristic

Part 1 Remarks

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

Experiment Part 2: Regulator Circuit with Zener Diode

Purpose of the Experiment:	Learning the working principles of Zener diode regulator circuit on a real application
Teaching of the Experiment:	Be able to comprehend working principles of Zener diode regulator circuits

Theoretical Information and Introduction to Part 2

As a result of fluctuations in network voltage and load current, the level of average DC voltage at the output of the rectifier circuits also varies. These changes in the voltage level result in some disadvantages in the operating conditions of the electronic circuit elements. It is possible to remove this problem by using regulator circuits. The purpose of the regulator circuits is to keep the output voltage constant despite changes in the input voltage or the load current (within certain limits). There are various integrated circuits produced to accomplish this function, but the Zener diode regulator circuits are one of the methods that are frequently used for this purpose. Figure 6.6 shows a simple regulator circuit with a Zener diode. V_i represents the unregulated DC input voltage and V_O represents the regulated DC output voltage. R_s and R_L are current limiting and load resistors respectively.
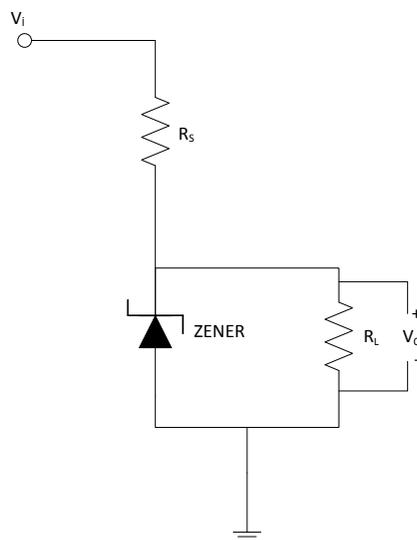

Figure 6.6 – Basic regulator circuit using Zener diode

The important point for Zener diode regulator circuit design is that the Zener voltage value of the Zener diode to be used in the circuit should be selected according to the operating voltage of the load. For example, if the required supply voltage is 5 V, then the zener diode should be 5 V. Furthermore, the unregulated DC input voltage (V_i) should not fall below the selected Zener voltage level. The reason is that if the input voltage is lower than the Zener voltage of the diode, the Zener diode will not operate and will have an open-circuit characteristic, and the output voltage will take a value determined by R_s and R_L.

Another important point is the selection of the current limiting resistor (R_s). This resistor should be selected to meet the minimum Zener current and load current even at the minimum level of the unregulated DC input voltage.

In addition, when the circuit is run with no load, it should be considered that the power consumed on the Zener diode will increase since the current to be passed through the load will also pass through the Zener diode. This may cause the Zener diode to break down if the Zener diode and the current limiting resistor are not selected suitably.

Preliminary Work – 2

Simulate the regulator circuit shown in Figure 6.7 on any simulation environment and record the desired results to Table 6.3.

Experimental Procedures for Part 2

Part 2 Schematics
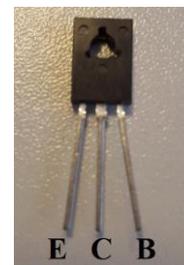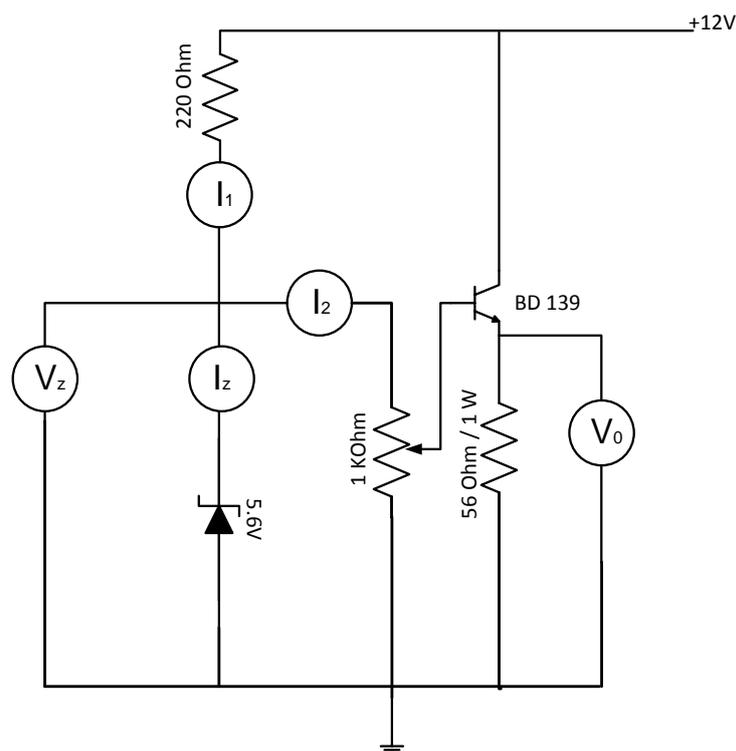
Figure 6.7 – Regulator circuit for the experiment

Experiment Part 2 Steps

1. Implement the circuit in Figure 6.7 using breadboard.
2. Set the potentiometer shaft to the minimum, middle and maximum levels and measure the desired parameters in Table 6.4 and record to the table.

Part 2 Results

Table 6.3 – Simulation results observation table for regulator circuit

	Simulation Results		
	Pot. at minimum	Pot. at the middle (0,5k)	Pot. at maximum
I_1			
I_2			
I_z			
V_z			
V_o			

Table 6.4 – Observation table for regulator circuit

	Experiment Results		
	Pot. at minimum	Pot. at the middle (0,5k)	Pot. at maximum
I_1			
I_2			
I_z			
V_z			
V_o			

Part 2 Remarks

Experiment 7

BJT Transistor Characteristics

Required Component List:

- Resistors   : 1 ×33 kΩ, 1 × 220 Ω / 1W
- Potentiometer   : 1 × 100 kΩ
- Transistor   : 1 × BC237

Equipment

- Multimeter
- Oscilloscope
- Breadboard

Experiment: BJT Transistor Characteristics

Purpose of the experiment:	Obtaining BJT transistor characteristics.
Teaching of the experiment:	To be able to understand working principles of the BJT Transistor characteristics.

Theoretical Information and Introduction

The transistor is a circuit component with three terminals formed by two PN junctions. These terminals are base (B), emitter (E) and collector (C). They are produced as PNP or NPN. The most common areas of use are the amplifier and the switch. They convert DC energy to AC energy in amplifier circuits. Therefore, the word "trans" is used. The transistor must be supplied by DC voltage for operation. This is called biasing. Resistors are also connected to the transistor in biasing. The transistor circuits can be named according to which end of the circuit ground is connected (common base, common emitter and common collector). The operating points of the transistors are determined by the DC power supply and resistor values connected in biasing. The NPN and PNP transistors can be seen in Figure 7.1.


NPN   PNP
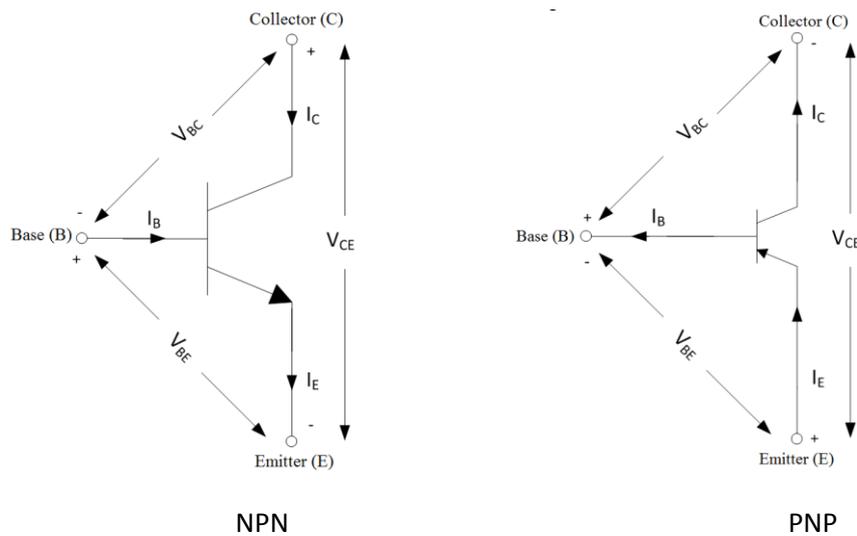
Figure 7.1 – NPN and PNP transistors

Transistors have many advantages, such as small size and low energy consumption, long working life and long lifetime, ready to work at any time, low operating voltage, possibility to work with battery and cheap price.

Preliminary Work

Set circuit in Figure 7.2 in a simulation program. Perform the simulations by following the steps given below. Fill in the relevant fields in the Table 7.1, Table 7.3, Table 7.5, Table 7.7 and Table 7.9 according to the simulation results. Draw the desired graphs in Figure 7.3, Figure 7.4, Figure 7.5 and Figure 7.6 through any computer program (MATLAB, Excel etc.) according to the obtained simulation data.

Experimental Procedures
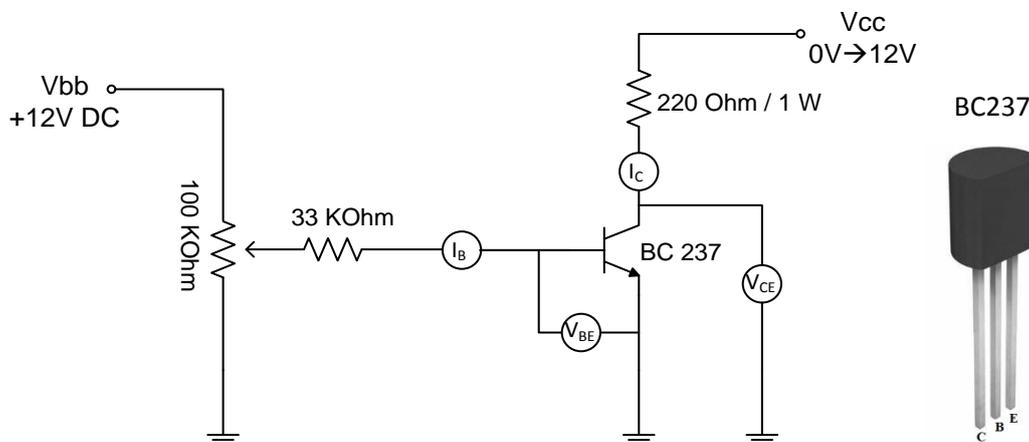
Schematic

Experiment Steps

1. Implement the circuit in Figure 7.2 on breadboard.
2. Set the V_{CC} and V_{BB} to 12 V. Change the I_B current to 25 μA intervals by adjusting the P_1 potentiometer settings. Write V_{BE}, I_C, β values to the Table 7.2 for different I_B values. Draw the characteristics between Figure 7.3 – 7.5 using the values in this table.
3. Set the I_B to 25 μA. Increase the V_{CC} value from 0 V to 12 V at regular intervals, and record the V_{CE} and I_C values in Table 7.4.
4. Apply Step 3 for 50 µA, 75 µA, 100 µA and record the results in the Tables 7.6, 7.8, and 7.10. Using the values in the tables, draw the characteristic $I_C - V_{CE}$ of the transistor in Figure 7.6.

Experiment Results

Table 7.1 – Observation table for constant V_{CC} (Simulation)

	$V_{CC} = 12\ V$											
V_{BE}												
I_B	0 μF	25 μF	50 μF	75 μF	100 μF	125 μF	150 μF	175 μF	200 μF	225 μF	250 μF	275 μF
I_C												
β												

Experiment 7: BJT Transistor Characteristics

Table 7.2 – Observation table for constant V_{CC} (Experimental)

V_{BE}												
I_B	0 μF	25 μF	50 μF	75 μF	100 μF	125 μF	150 μF	175 μF	200 μF	225 μF	250 μF	275 μF
I_C												
β												

(Table header spanning: $V_{CC} = 12\ V$)


Figure 7.3 – $I_B - V_{BE}$ graph for $V_{CC} = 12\ V$
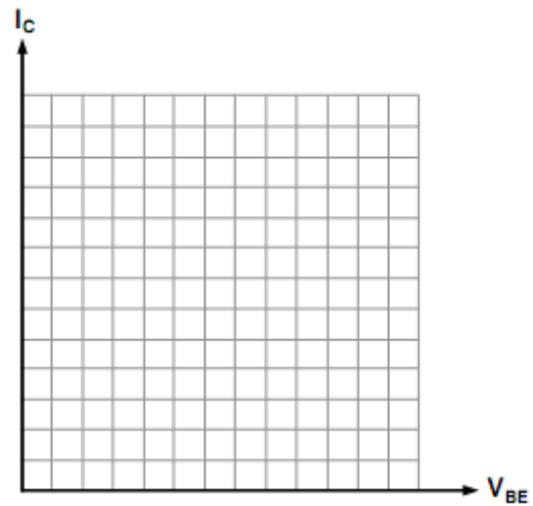

Figure 7.4 – $I_C - V_{BE}$ graph for $V_{CC} = 12\ V$
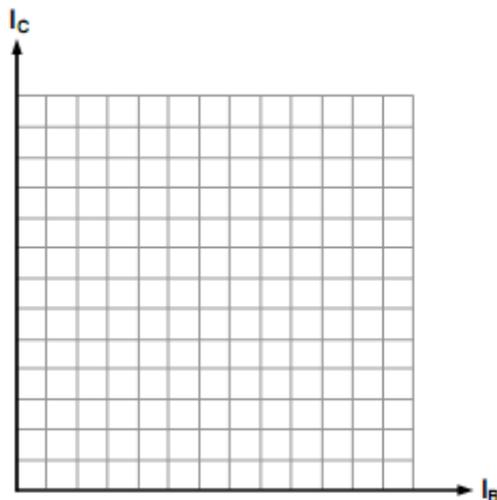
Figure 7.5 – $I_C - I_B$ graph for $V_{CC} = 12\ V$

Table 7.3 – Observation table for constant $I_B = 25\ \mu A$ (Simulation)

$I_B = 25\ \mu A$											
V_{CE}											
I_C											

Table 7.4 – Observation table for constant $I_B = 25\ \mu A$ (Experimental)

$I_B = 25\ \mu A$											
V_{CE}											
I_C											

Table 7.5 – Observation table for constant $I_B = 50\ \mu A$ (Simulation)

$I_B = 50\ \mu A$											
V_{CE}											
I_C											

Table 7.6 – Observation table for constant $I_B = 50\ \mu A$ (Experimental)

$I_B = 50\ \mu A$											
V_{CE}											
I_C											

Table 7.7 – Observation table for constant $I_B = 75\ \mu A$ (Simulation)

$I_B = 75\ \mu A$											
V_{CE}											
I_C											

Table 7.8 – Observation table for constant $I_B = 75\ \mu A$ (Experimental)

$I_B = 75\ \mu A$											
V_{CE}											
I_C											

Table 7.9 – Observation table for constant $I_B = 100 \ \mu A$ (Simulation)

$I_B = 100 \ \mu A$											
V_{CE}											
I_C											

Table 7.10 – Observation table for constant $I_B = 100 \ \mu A$ (Experimental)

$I_B = 100 \ \mu A$											
V_{CE}											
I_C											
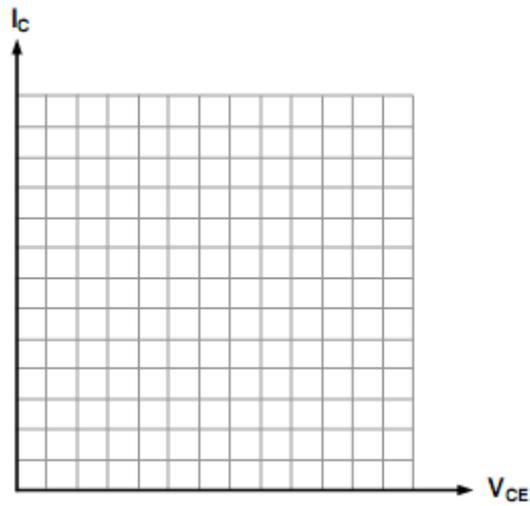
Figure 7.6 – $I_C - V_{CE}$ characteristics for different I_B

Remarks

Experiment 8

BJT Amplifiers

Required Component List:

- Resistors   : 1×100 Ω, 1×560 Ω, 1×2.2 kΩ, 1×12 kΩ
- Capacitors   : 2×47 uF
- Transistor   : $1 \times$ BC237

Equipment

- Multimeter
- Oscilloscope
- Signal Generator
- Breadboard

Experiment: BJT Amplifiers

|	
Purpose of Experiment:	Analysis of the behavior of Class A amplifiers.
Teaching of Experiment:	To be able to understand the working principle of class A amplifier designed by using transistor.

Theoretical Information and Introduction

An amplifier can be implemented with FET, MOSFET or BJT. We can also divide amplifiers into small signal amplifiers and power amplifiers. In this section, power amplifiers will be examined in general. Power amplifiers are often used in the industry for many purposes, such as amplifying sound, generating power for a motor to operate, or powering a transmitter.

Depending on the transmission situation (biasing properties), the amplifiers are classified as follows.

- **Class A:** The amplifier is active at 100% of the input signal.
- **Class B:** The amplifier is active at 50% of the input signal. It is active in either positive or negative part of the signal.
- **Class AB:** The amplifier is active between more than 50% and less than 100% of the input signal.
- **Class C:** The amplifier is active in less than 50% of the input signal.
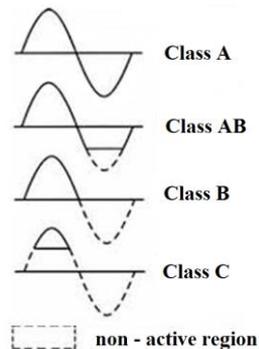
Figure 8.1 – Classification of amplifiers according to their transmission status

In this experiment, we will focus on Class A amplifiers, and you can find detailed information on both Class A and other class amplifiers from the course sources.

Class A amplifiers are the type of amplifier that is in transit at every moment of the input signal.

Advantages:

- Designing is easy compared to amplifiers in other classes. It can be designed using a single transistor.

- Since the amplifying element is permanently active, it has an almost linear AC resistance.
- There is no "back-opening time" since the amplifying element never shuts down. This means good performance and high stability at high frequencies.
- The crossover disruption does not occur because it is in transmission all the time. For example, in class B amplifiers, there are 50% transmissions, so two amplifiers are designed and 100% transmission is provided. This degradation takes place in the moments of transition from negative to positive, when switching from one transistor active to the other passive state.

Disadvantages:

- It works with low efficiency. Because they are in continuous conduction, they consume a lot of power and battery life is short if you design a battery-powered amplifier.

Preliminary Work

Calculate the desired values theoretically in Table 8.1 for the schematic of the experiment shown in Figure 8.2 and perform the simulation by following the experiment steps and record the simulation results in the appropriate places in Table 8.1.

Experimental Procedures
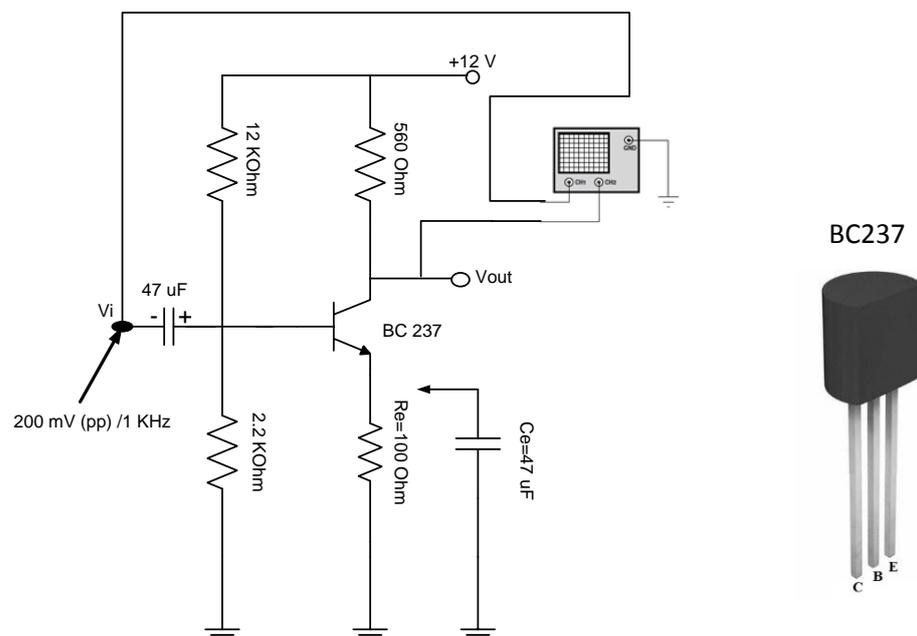
Schematic

Figure 8.2 – Schematic of the experiment

Experiment Steps

1. Implement the circuit in Figure 8.2 without bypass capacitor C_E on breadboard.
2. Measure the values of the DC working point specified in Table 8.2 using multimeter and record in the table.
3. Measure the values of the AC operation for the input signal given in Figure 8.3 and record the appropriate values in Table 8.1. The pick to pick voltage value of the input signal and the pick to pick voltage value of the output signal will give the gain value.
4. Draw output signal to the Figure 8.4.
5. Connect bypass capacitor C_E to the circuit.
6. Repeat steps 2 and 3 and record the results obtained when C_E is in operation in the appropriate sections in Table 8.1.
7. Draw output signal when C_E is in operation to the Figure 8.4.

Experiment Results

200 mV

V/d: 40 mV

T/d: 0.5ms
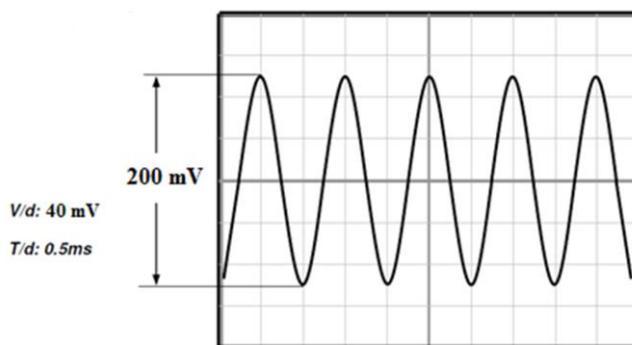
Figure 8.3 – V_i Input signal
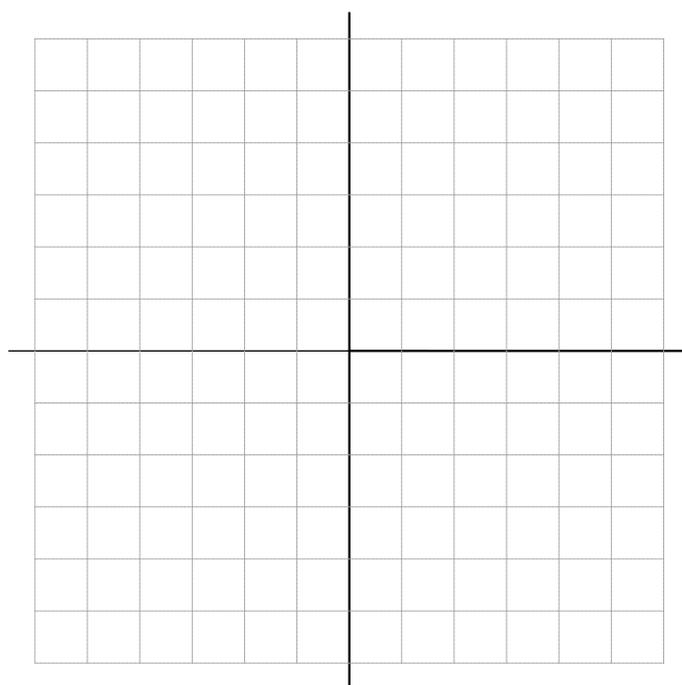
Volt/Div (Ch1) =

Volt/Div (Ch2)=

Time/Div=

Figure 8.4 – Output signal (C_E disabled)
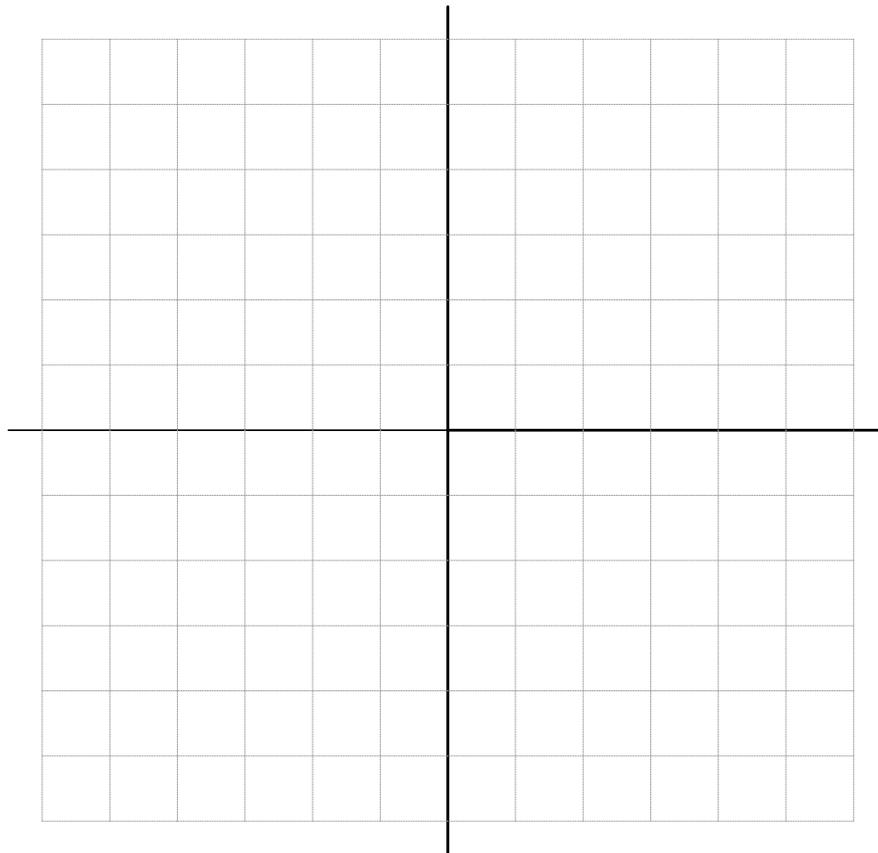
Volt/Div (Ch1) =

Volt/Div (Ch2)=

Time/Div=

Figure 8.5 – Output signal (C_E enabled)

Table 8.1 – Observation table for Class A amplifier

	C_E disabled			C_E on		
	Calculation	**Simulation**	**Measurement**	**Calculation**	**Simulation**	**Measurement**
V_B						
I_B						
I_C						
I_E						
V_E						
V_{CE}						
A_V						

Remarks

Experiment 9

MOSFET Basics

Required Component List:

- MOSFET : 1 × CD4007
- Potentiometer : 1 × 100 kΩ

Equipment

- Multimeter
- DC Power Supply
- Breadboard

Experiment: MOS-FET Basics*

* Başkent University EEM217 coded course note was used in preparing this experiment.

Purpose of Experiment:	Examination of working principles of MOSFETs.
Teaching of Experiment:	Understanding the operating principles of MOSFETs.

Theoretical Information and Introduction

MOSFET (Metal Oxide Semiconductor Field Effect Transistor) is type of a field effect transistor. There are basically three terminals, gate, source and drain. Four terminal MOSFETs have another terminal called Base. But this terminal is connected to the source end from the inside and cannot be reached from the outside. Figure 9.1 shows MOSFETs with depletion and enhancement type with 4 and 3 terminals.
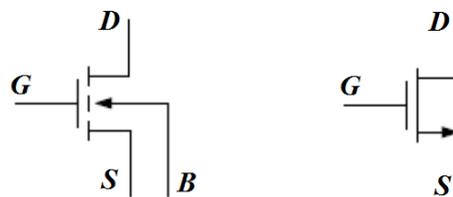

Figure 9.1– 4 and 3 terminal MOSFETs (Enhancement type, depletion type)

For operation of the MOSFETs, two voltage sources must be connected as shown in Figure 9.2. These sources form V_{GS} and V_{DS} voltages.
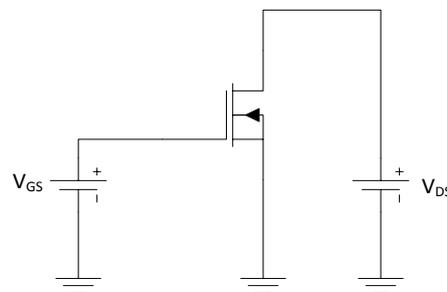

Figure 9.2 – V_{GS} and V_{DS} connections

Due to the effect of V_{DS} voltage, I_{DS} current is generated between drain and source. This I_{DS} current is controlled by the V_{GS} gate - source voltage. The V_{GS}, V_{DS} and I_{DS} values of the MOSFET can be changed and used for various purposes. There are three regions of MOSFET: cut-off, linear and saturation. In the cut-off region, the threshold voltage value is low and the MOSFET does not work. In this experiment, the operating regions of the MOSFET in saturation and linear regions will be examined. The I_{DS} / V_{DS} graph is shown in Figure 9.3 at constant V_{GS} values of a typical MOSFET. The linear region is the region where the V_{GS} voltage is low and the I_{DS} current linearly increases. As the V_{DS} voltage increases, the I_{DS} current is being constant. When the V_{DS} voltage equals the saturation voltage ($V_{D(SAT)}$),

MOSFET reaches saturation. The saturation voltage is related to the V_{GS} voltage and the threshold voltage (V_T), which is a different physical parameter for each MOSFET.
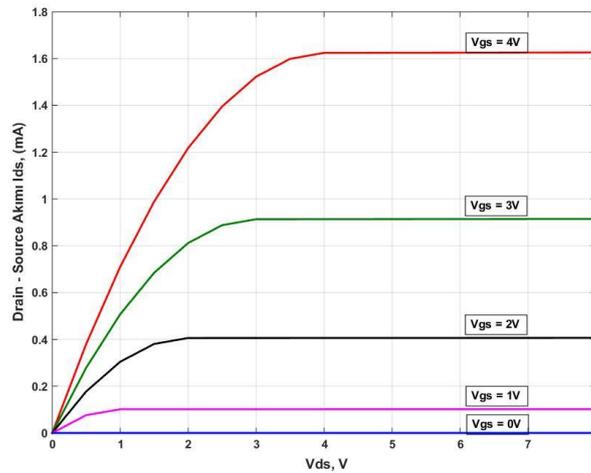

Figure 9.3 – I_{DS} / V_{DS} graph

When $V_{D(SAT)} = V_{GS} - V_T$;

$V_{D(SAT)} \geq V_{DS} \geq 0$: saturation region

$V_{DS} \geq V_{D(SAT)} \geq 0$: linear region

The I_{DS} current is calculated by the following equation when operating in the saturation region.

$$I_{DS} = \mu_0 C_{ox} \left(\frac{W}{L}\right) \left[(V_{GS} - V_T) - \frac{V_{DS}}{2}\right] V_{DS}$$

W: Width of Gate

L: Length of Gate

C_{ox} : Oxide layer thickness between gate and body

The MOSFET operates as a voltage-controlled resistor when in the saturation region. Drain - source resistance can be found by taking partial derivative of I_{DS} according to V_{DS}.

$$\frac{\partial I_{DS}}{\partial V_{DS}} \left[\mu_0 C_{0x} \left(\frac{W}{L}\right) \left[(V_{GS} - V_T) - \frac{V_{DS}}{2}\right] V_{DS}\right] = \mu_0 C_{ox} \left(\frac{W}{L}\right) (V_{GS} - V_T - V_{DS}) = \frac{1}{r_{ohmic}}$$

The active region is usually used with the purpose of amplifying the input signal. When operating in this region, the I_{DS} current is calculated by the following equation.

$$I_{DS} = \frac{1}{2} \mu_0 C_{ox} \left(\frac{W}{L}\right) (V_{GS} - V_T)^2$$

The linear region is usually used with the purpose of amplifying the input signal. The MOSFET operates as a voltage controlled current source in the linear region. But this current source is not ideal. The R_{ds} is under the influence of the small signal equivalent resistance.

The MOSFET can be connected in a linear region to act as a diode. With this connection, the voltage on the MOSFET can be adjusted to be used as the reference voltage. In addition, this connection is also used in applications such as current mirror. The connection diagram is shown in Figure 9.4. The voltage on the MOSFET is found by solving the previous I_{DS} equation with $V_{GS} = V_{DS}$ equation as follows.

$$V_{MOSDIODE} = V_T + \sqrt{\frac{I_{DS}}{\frac{1}{2}\mu_0 C_{ox} \left(\frac{W}{L}\right)}}$$
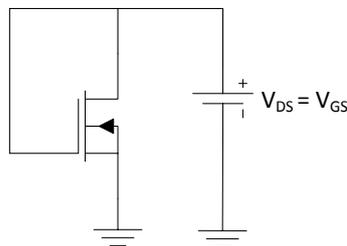
Figure 9.4 – MOSFET structure

Preliminary Work

CD4007 MOSFET catalogue values to be used in the experiment: (Courtesy of Ass. Prof. Dr. David M. Binkley, Clark Hopper M.S, and Harold Hearne M.S., The Uni. of North Carolina);

$\mu_0 C_{ox} = 166.67 \ \mu A/V^2$

W/L = 3

$V_T = 1.45V$

$V_A = 1/\lambda = 100V$

The pin connections of the CD4007 enhancement type MOSFET can be seen in Figure 9.5.
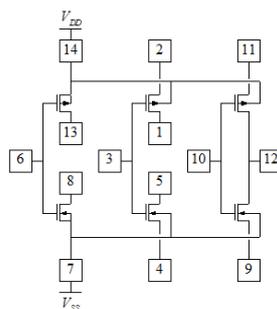
Figure 9.5 – CD4007 pin connections

For the MOSFET shown in Figure 9.2, calculate the I_{DS} currents at the voltage values given below.

a) $V_{GS} = 3V$; $V_{DS} = 0.5V$
b) $V_{GS} = 5V$; $V_{DS} = 0.5V$
c) $V_{GS} = 2V$; $V_{DS} = 3V$

In Figure 9.6, Figure 9.7 and Figure 9.8, set up the schematic diagrams in a simulation program and perform the necessary simulations by following the steps given below. Fill in the blanks in Table 9.1, Table 9.3 and Table 9.5 according to the results of the simulations.

Experimental Procedures

The CD4007 MOSFET will be used in this experiment. According to the internal structure of the MSOFET shown in Figure 9.5, there are three n-channel and three p-channel structures. The supply of n-channel MOS transistors is connected to V_{SS} (pin 7), and the supply of p-channel MOS transistors is connected to V_{DD} (pin 14). In this case, pin 7 should be negative voltage, pin 14 positive voltage.

Note that there is no voltage on the circuit when connecting the CD4007. Otherwise it may be damaged. It can also quickly break down due to the electrostatic effect. For this reason you should not touch its terminals.

Before implementing the circuit,

- Measure resistance values by multimeter. Make sure each resistor has 2% error margin. This will make your current measurements sensitive.
- Before connecting the circuit, make sure that the V_{GS} power supply is set to + 4V.

Note: First connect the DC voltage, then the AC voltage. Disconnect the AC voltage before DC voltage.

Schematics
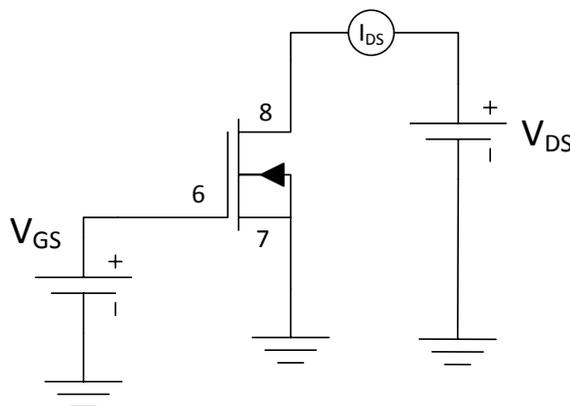
<p style="text-align:center">**Figure 9.6 – Schematic of the experiment**</p>
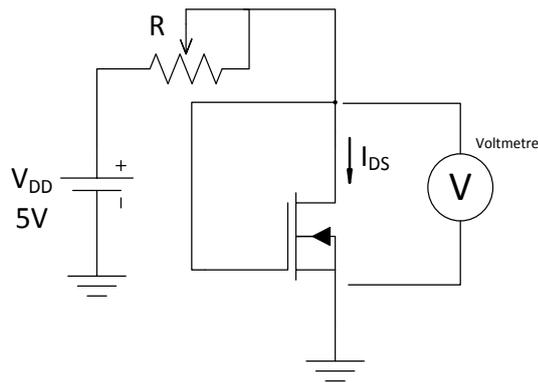
**Figure 9.7 – Schematic of the experiment**
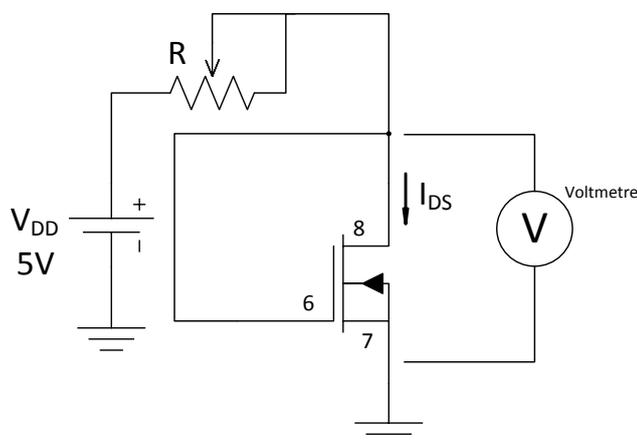
Figure 9.8 – Schematic of the experiment

Experiment Steps

1. Implement the circuit in Figure 9.6 on breadboard. Adjust V_{GS} to +4V. When changing the V_{DS} voltage between 0 and + 5V, measure the I_{DS} current and record in Table 9.2. Increase the voltage with 0.4V intervals.

2. Draw the I_{DS} - V_{DS} graph in Figure 9.9 with the values in Table 9.2.

3. Determine the V_{GS} threshold voltage for V_{DS} = 5V. For this, the V_{GS} value, which is the negligible level of the IDS current, must be selected. Assume the negligible level is 5 µA.

4. Implement the circuit in Figure 4.2. Observe the relationship between the V_{GS} change and the resistance value via ohmmeter. Record these values in Table 9.4.

5. Implement the 100 µA DC current source circuit with the MOSFET shown in Figure 9.8. Set the I_{DS} current to 100 µA by adjusting the potentiometer and record the R resistance value in Table 9.6.

6. Measure the voltage across the MOSFET as shown in Figure 9.8 via the multimeter and record in Table 9.6.

7. Record in Table 9.6 which resistance values of I_{DS} current reaches 100 µA, 75 µA and 50 µA by turning the potentiometer.

97

Experiment Results

Table 9.1 – V$_{DS}$, V$_{GS}$, I$_{DS}$ values (Simulation)

V_{DS}	I_{DS}		
	$V_{GS} = 2V$	$V_{GS} = 3V$	$V_{GS} = 4V$
0.4 V			
0.8 V			
1.2 V			
1.6 V			
1.8 V			
2.0 V			
2.4 V			
2.8 V			
3.2 V			
3.6 V			
4.0 V			
4.4 V			
4.8 V			
5.0 V			

Table 9.2 – V$_{DS}$, V$_{GS}$, I$_{DS}$ values (Experimental)

V_{DS}	I_{DS}		
	$V_{GS} = 2V$	$V_{GS} = 3V$	$V_{GS} = 4V$
0.4 V			
0.8 V			
1.2 V			
1.6 V			
1.8 V			
2.0 V			
2.4 V			
2.8 V			
3.2 V			
3.6 V			
4.0 V			
4.4 V			
4.8 V			
5.0 V			

Table 9.3 – V$_{GS}$ – R (Simulation)

V_{GS}	1.5 V	2.0 V	3.0 V	4.0 V	5.0 V
R					

Table 9.4 – V$_{GS}$ – R (Experimental)

V_{GS}	1.5 V	2.0 V	3.0 V	4.0 V	5.0 V
R					

Table 9.5 –I_{DS} current related to the potentiometer values (Simulation)

R_{pot}	I_{DS}	V_{DS}
	100 μA	
	75 μA	
	50 μA	

Table 9.6 – I_{DS} current related to the potentiometer values (Experimental)

R_{pot}	I_{DS}	V_{DS}
	100 μA	
	75 μA	
	50 μA	

I_{DS}

V_{DS}
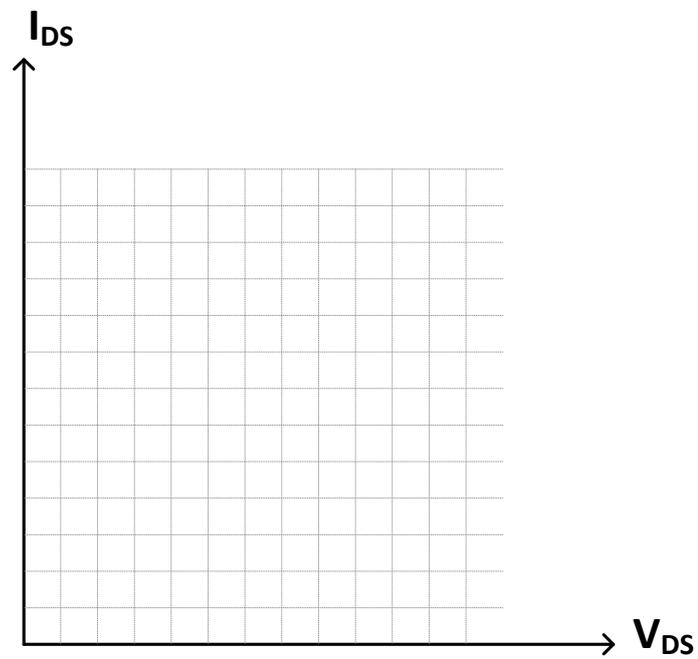
Figure 9.9: I_{DS} - V_{DS} graph

Remarks

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

Experiment 10

Operational Amplifier (OP-AMP) Circuits

Required Component List:

- Resistors : 1 × 1 kΩ, 1 × 5 kΩ, 1 × 10 kΩ, 4 × 100 kΩ
- Potentiometer : 2 × 10 kΩ
- OP-AMP : 3 × LM741

Equipment

- Multimeter
- Oscilloscope
- Signal Generator
- Breadboard

Experiment: Operational Amplifier (OP-AMP) Circuits

Purpose of Experiment:	Investigation of the operation of inverting, non-inverting, summing and difference amplifier circuits using OP-AMP.
Teaching of Experiment:	To be able to comprehend the purpose of using OP-AMP and amplifier circuits using OP-AMP.

Theoretical Information and Introduction

OP-AMPs that have been used towards the end of the 1960s can be described as very high gain differential amplifiers. OP-AMPs are voltage and current gain circuits. They can also be called "Analog Computers" because of their mathematical operation capacities. Figure 10.1 shows an OP-AMP symbol, input, output and supply terminals.
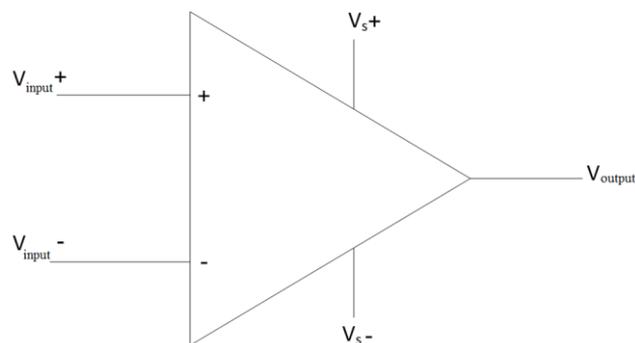
Figure 10.1– OP-AMP symbol and input/output terminals
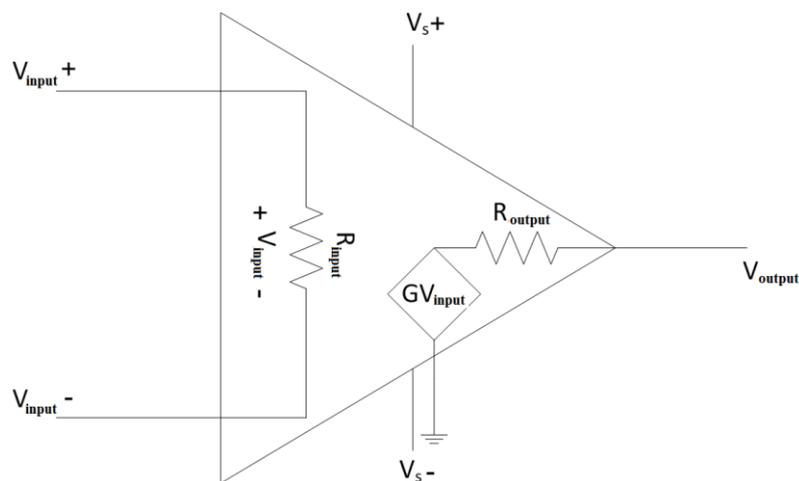
Figure 10.2– Ideal equivalent circuit of OP-AMP

Figure 10.2 shows the equivalent circuit of OP-AMP. Ideal OP-AMP generally has the following characteristics.

- Infinitely open loop gain $G = V_{output}/V_{input}$
- Infinite input resistance, R_{input}. Due to this infinite input resistance, no current flows between the two terminals of the OP-AMP.

103

- It has zero input offset voltage. Thus, the voltage values at the two inputs of the OP-AMP are equal to each other.
- Zero phase shift and infinite bandwidth.
- Zero output resistance, R_{output}.
- Infinite common mode rejection ratio.

For typical OP-AMPs, the input impedances are very high and close to 5 MΩ and the output impedances are about zero. The high input impedance prevents a negative effect by loading the previous circuit or the power supply to which it is connected. The gain is around 200.000 in typical OP-AMPs. The bandwidths of OP-AMPs are around 1 MHz.

OPAMPs are supplied symmetrically. In the OP-AMP symbol, the $V_s +$ and $V_s -$ are the terminals to which the supply source is connected. One of the positive or negative supply values can be 0 (zero). It should not be forgotten that positive and negative supplies are relative in terms of electricity. Figure 10.3 shows the basic display of the OP-AMP supply circuit.
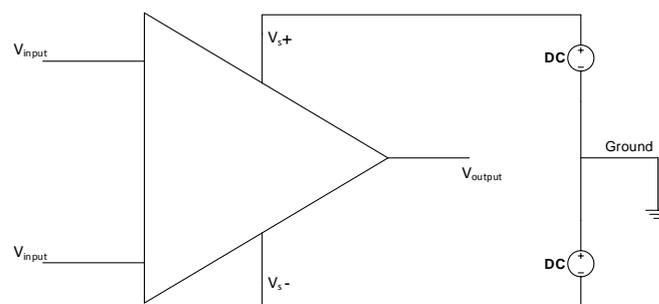

Figure 10.3– OP-AMP supply circuit

For an OP-AMP, two types of gains can be mentioned, open loop and closed loop. The open-loop gain is determined based on its own characteristic of OP-AMP, while the closed-loop gain is determined by a feedback resistor connected externally to a circuit with OP-AMP. The level of voltage amplification applied to the input of an OP-AMP amplifier circuit is limited by the supply voltage of the OP-AMP. In other words, in a circuit with OP-AMP it is not possible to achieve a higher amplification than the supply voltage ± V$_s$ of OP-AMP.

For the ideal OP-AMP, the 0 volt value must be read at the output when no signal is applied to the input. However, in practice, it is observed that a very small voltage is generated between the input terminals of the OP-AMP. For this reason, the value of this offset voltage is multiplied by the gain of OP-AMP and transferred to the output voltage. To avoid this, some of the OP-AMPs have terminals to compensate for this constant voltage at the input.

OP-AMPs can be used for mathematical operations such as addition-subtraction, multiplication-division, integral and derivative. It is also used in voltage follower and comparator circuits. In the comparator circuits, the signals applied to the positive and negative inputs of the OP-AMP are compared and when the voltage applied to the positive

terminal is greater than the voltage applied to the negative terminal, $+V_s$ is observed from the output. The voltage gain of the voltage follower circuit is 1 and the input and output signals are at the same phase.

OP-AMPs are used in two separate circuits that are related to each other, where only the voltage needs to be transferred without current exchange. It can be used to isolate two circuits from each other. In addition to all these applications, OP-AMPs are often used in rectifier, filtering and instrumentation applications.

The most commonly used OP-AMPs are the LM 741 and LM 747. These include a single OP-AMP in the LM 741 and two OP-AMPs in the LM 747. The LM 741 to be used in the experiments to be carried out in Figure 10.4 and the internal structure of this OP-AMP show a detailed connection of the terminals. For more detailed information, please refer to related catalogues (datasheets).
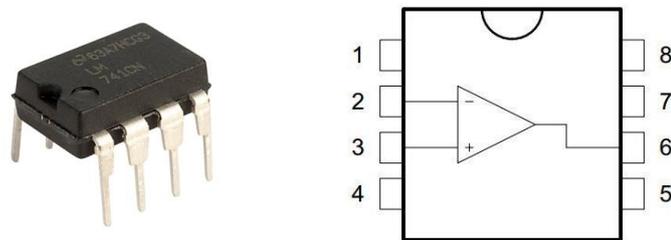

Figure 10.4– LM 741 and internal structure

Pin Number	Function
1	Offset reset
2	$V_{input} -$
3	$V_{input} +$
4	$V_s -$ (negative supply pin)
5	Offset reset
6	V_{output}
7	$V_s +$ (positive supply pin)
8	NC

Preliminary Work

Calculate the parts that theoretically must be calculated by following the steps to be applied to the inverting, non-inverting, summing and difference amplifier circuits shown in Figure 10.5, Figure 10.6, Figure 10.7 and Figure 10.8 respectively and record them in the appropriate sections in the tables below. Simulate these circuits with a simulation program and fill in the tables according to the results you have obtained. Randomly select the values to be applied to the input of the OP-AMP for the summing and difference amplifier circuits shown in Figure 10.7 and Figure 10.8 so that they are not larger than 12 V and record these selected values in Table 10.3 and Table 10.4. Use these values during the experiment.

Experimental Procedures

Schematics
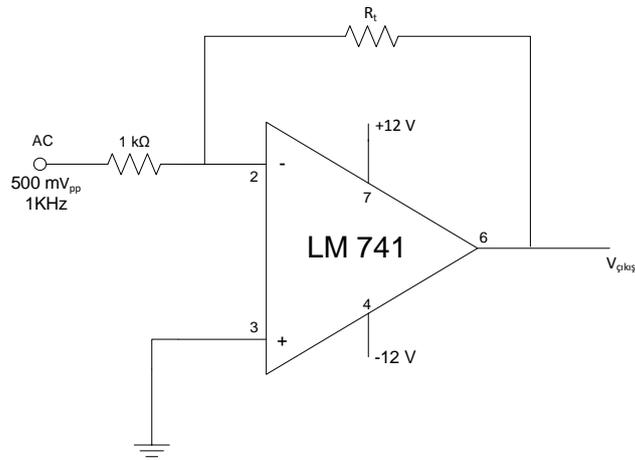
Figure 10.5– Inverting amplifier
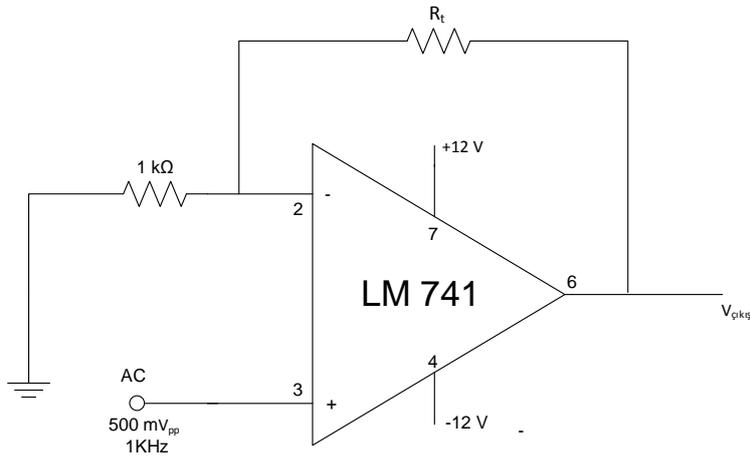

Figure 10.6– Non-Inverting amplifier
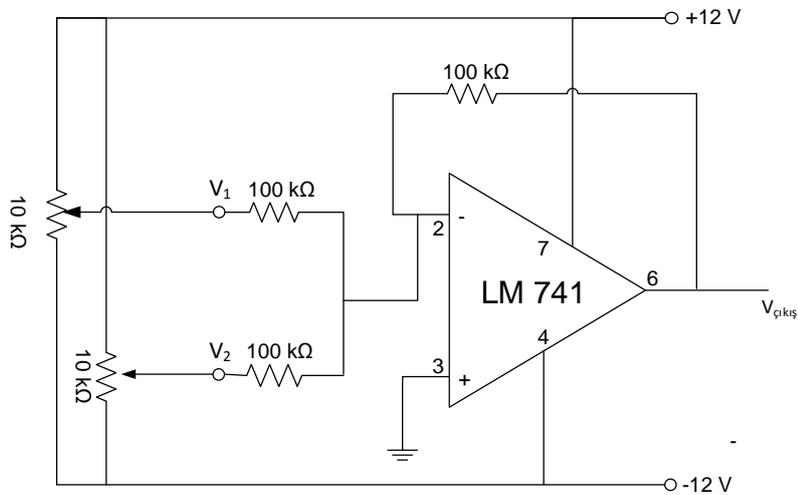

Figure 10.7– Summing amplifier
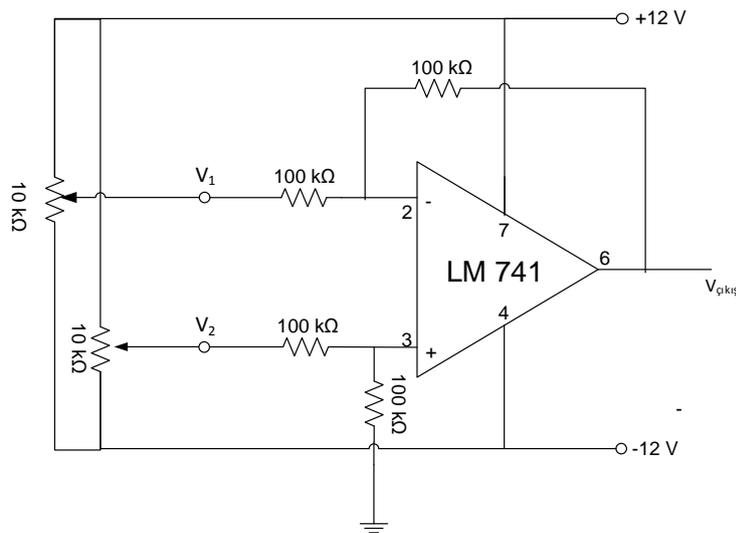
Figure 10.8– Difference amplifier circuit

Experiment Steps

1. Implement the inverting amplifier circuit shown in Figure 10.5 on the breadboard.
2. Apply a sinusoidal signal with a frequency of 1 kHz and a magnitude of 500 mV V_{pp} as the input of the circuit (V_{in}).
3. Observe the signal at the circuit output by selecting the resistor R_t to be 5 kΩ and 10 kΩ, and draw in Figure 10.9 and Figure 10.10. (Record the Volt / div and Time / div values shown on the oscilloscope.)
4. According to the outputs obtained, calculate the gain of the amplifier circuit and record in Table 10.1. Compare with the theoretical and simulation results you have already found.
5. Implement the non-inverting amplifier shown in Figure 10.6 on the breadboard.
6. Apply a sinusoidal signal with a frequency of 1 kHz and a magnitude of 500 mV V_{pp} as the input of the circuit (V_{in}).
7. Observe the signal at the circuit output by selecting the resistor R_t to be 5 kΩ and 10 kΩ, and draw in Figure 10.11 and Figure 10.12. (Record the Volt / div and Time / div values shown on the oscilloscope.)
8. According to the outputs obtained, calculate the gain of the amplifier and record in Table 10.2. Compare with theoretical and simulation results you have already found.
9. Implement the summing amplifier circuit shown in Figure 10.7 on the breadboard.
10. By adjusting the input voltages, which are chosen with the potentiometers, measure the output for each combination with the multimeter. Please note that the output voltage may approach the supply voltage by 1 ~ 2 Volts.
11. Record the experimental results you have obtained to the Table 10.3. Compare with the theoretical and simulation results you have already found.
12. Implement the difference amplifier circuit shown in Figure 10.8 on the breadboard.

13. By adjusting the input voltages, you choose during the simulations with the potentiometers, measure the output for each combination with the multimeter. Please note that the output voltage may approach the supply voltage by 1 ~ 2 Volts.

14. Record the experimental results you have obtained to the Table 10.4. Compare with the theoretical and simulation results you have already found.

Experiment Results

Volt/div =   Time/div =   Volt/div =   Time/div =
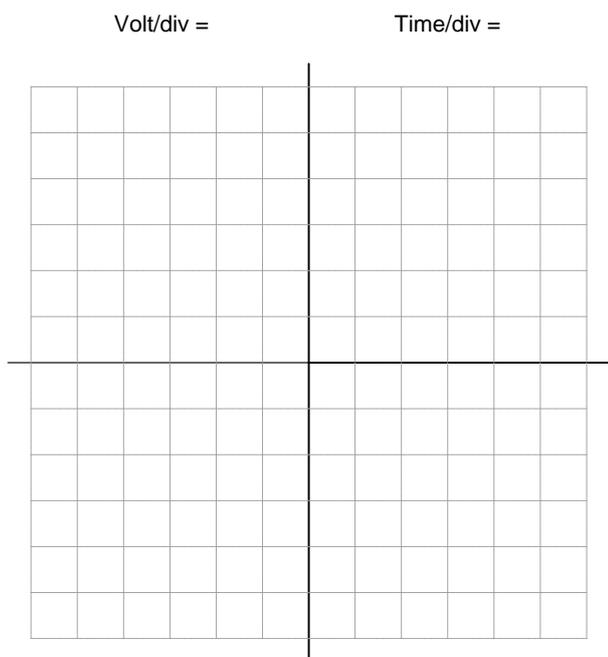
Figure 10.9 – The output of the circuit ($R_t = 5$ KΩ)   Figure 10.10 – The output of the circuit($R_t = 10$ KΩ)
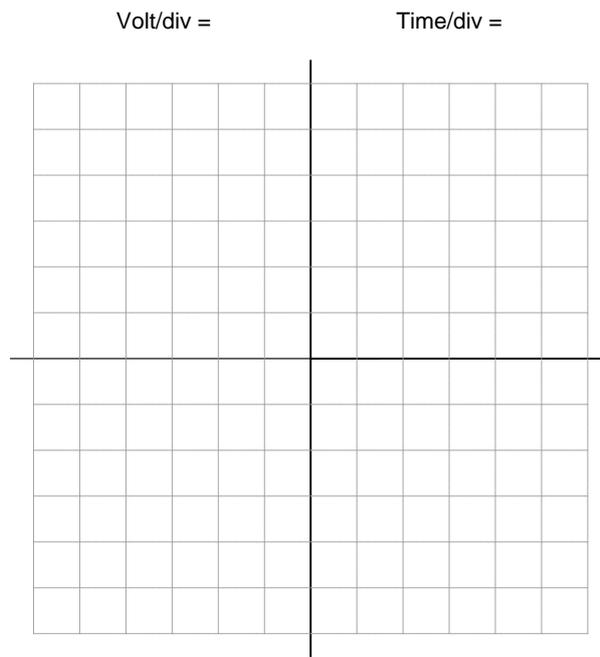
Volt/div =   Time/div =   Volt/div =   Time/div =
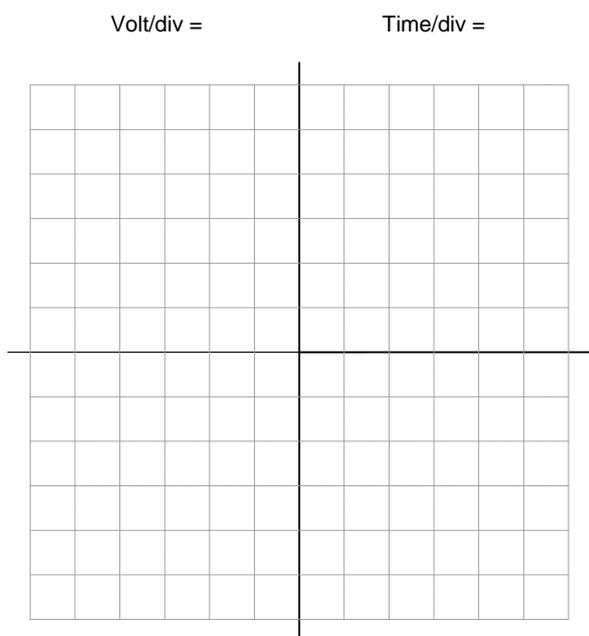
Figure 10.11 – The output of the circuit ($R_t = 5$ KΩ)   Figure 10.12 – The output of the circuit($R_t = 10$ KΩ)
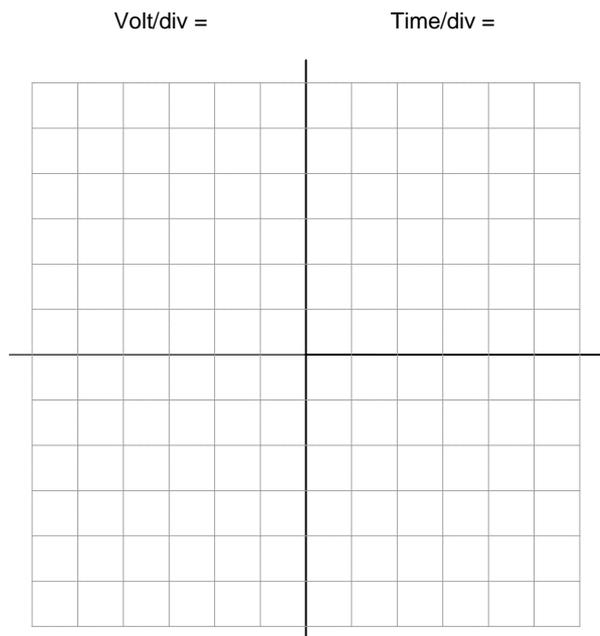
Table 10.1 – The voltage gains for inverting amplifier

	$A_V = V_{output}/V_{input}$	
	$R_t = 5\ k\Omega$	$R_t = 10\ k\Omega$
Theoretical		
Simulation		
Experimental		

Table 10.2 – The voltage gains for non-inverting amplifier

	$A_V = V_{output}/V_{input}$	
	$R_t = 5\ k\Omega$	$R_t = 10\ k\Omega$
Theoretical		
Simulation		
Experimental		

Table 10.3 – The output voltage values for summing amplifier

Inputs	V_1(V)						
	V_2(V)						
V_{output}	Theoretical						
	Simulation						
	Experimental						

Table 10.4 – The output voltage values for difference amplifier

Inputs	V_1(V)						
	V_2(V)						
V_{output}	Theoretical						
	Simulation						
	Experimental						

Remarks

Experiment 11

Integrator and Differentiator with OP-AMP

Required Component List:

- Resistors   : 1 ×20 kΩ, 1 × 4.7 kΩ
- Capacitors   : 1 × 470 nF
- OP-AMP   : 2 × LM741

Equipment

- Multimeter
- Oscilloscope
- Signal Generator
- Breadboard

Experiment: Integrator and Differentiator with OP-AMP

Purpose of Experiment:	Investigation of the differentiator and integrator circuits with OPAMP
Teaching of Experiment:	To be able to understand the working principles of differentiator and integrator circuits

Theoretical Information and Introduction

Differentiator Circuit
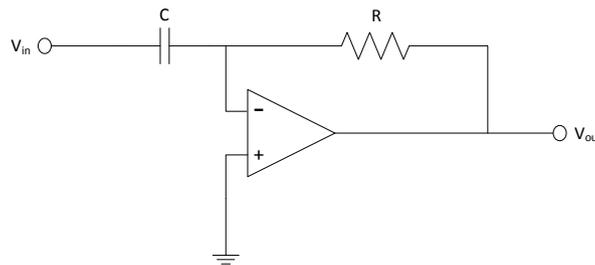
Figure 11.1– Differentiator circuit

The differentiator circuits, which are showed in Figure 11.1, generate an output depending on the rate of change of the input signal. Generally, the working logic in the differentiator circuits is that the output signal has higher amplitude when the input signal changes rapidly but vice versa if the input signal changes slowly then circuits generate the output signal with lower amplitude. If the amplitude of the input signal does not change then any signal cannot be seen at the output of the circuits because the derivative of the constant sign is zero. In other words, the differentiator circuits are needed to supply with the time varying signals (AC) as the input signals. The output voltage of the differentiator circuit in Figure 11.1 is determined depending on the input signal with V_{in} amplitude as follows

$$V_{out} = -RC \frac{dV_{in}}{dt}$$

where the derivation of the input signal with respect to time denotes the rate of change of the input signal. (-) sign in the equation indicates 180 degrees phase difference between the input and output signals.

The gain of the differentiator circuit is calculated as follows

$$K = R.C$$

One of the encountered problems in the differentiator circuits is that the gain of the circuit increases when worked at the high frequencies and the circuit becomes sensitive to the noise at these frequencies due to this increase in the gain. It is necessary to set a limit on high voltage gain to avoid this undesirable situation and hence, a resistor is connected in series with the capacitor in the circuit.

The differentiator circuits are generally used in applications in which the output signal is produced depending on the rapid changes in the signal level and the rate of change in the signal must be measured.

Integrator Circuit
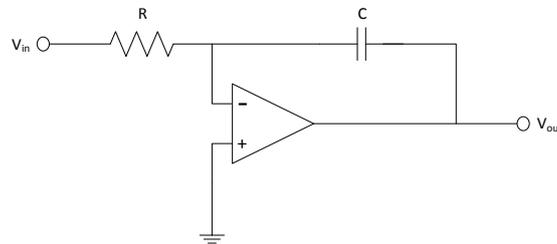
Figure 11.2 – Integrator circuit

The output amplitude of the integral circuit in Figure 11.2 is the sum of the area under the input signal when the time increases. As it can be understood from the description, if the area under the input signal increases with time, the output amplitude increases or vice versa, if it decreases then the amplitude of the output signal decreases. When input signal with V_{in} amplitude is applied, the output of the integrator circuit is determined as follows

$$V_{out} = -\frac{1}{RC} \int_0^t V_{in} dt$$

The gain of the circuit in Figure 11.2 is calculated as follows

$$K = \frac{1}{RC}$$

(-) sign in the output voltage equation indicates 180 degrees phase difference between the input and output signals. Similar to differentiator circuit, it is necessary to connect a resistor in parallel with the capacitor in the circuit in order to limit the high frequency gain in the integrator circuit.

Preliminary Work

Perform the simulations of the circuits given in Figure 11.3 and Figure 11.4 by following the application steps and using a simulation program. Draw the outputs to the Figure 11.5 and Figure 11.6.

Experimental Procedures

Schematics
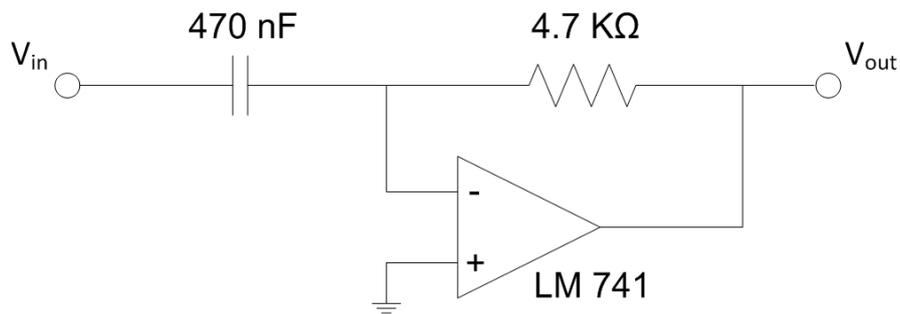

Figure 11.3– Differentiator circuit
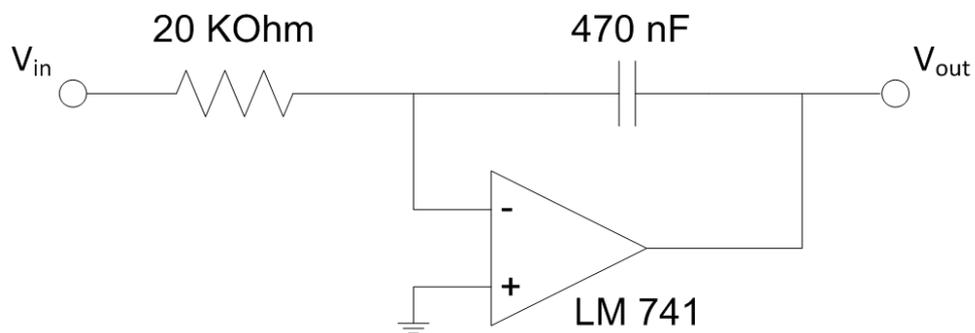
Figure 11.4 – Integrator circuit

Experiment Steps

1. Implement the circuit in Figure 11.3 on breadboard.
2. Apply a 200 mV **square wave** signal with a frequency of 50 Hz as the V_{in} input signal to the circuit.
3. Draw the signal which you observed at the output of the circuit to the Figure 11.7.
4. Implement the circuit in Figure 11.4 on breadboard.
5. Apply a 200 mV **triangle wave** signal with a frequency of 50 Hz as the V_{in} input signal to the circuit.
6. Draw the signal which you observed at the output of the circuit to the Figure 11.8.

Experiment Results

Numerical Results
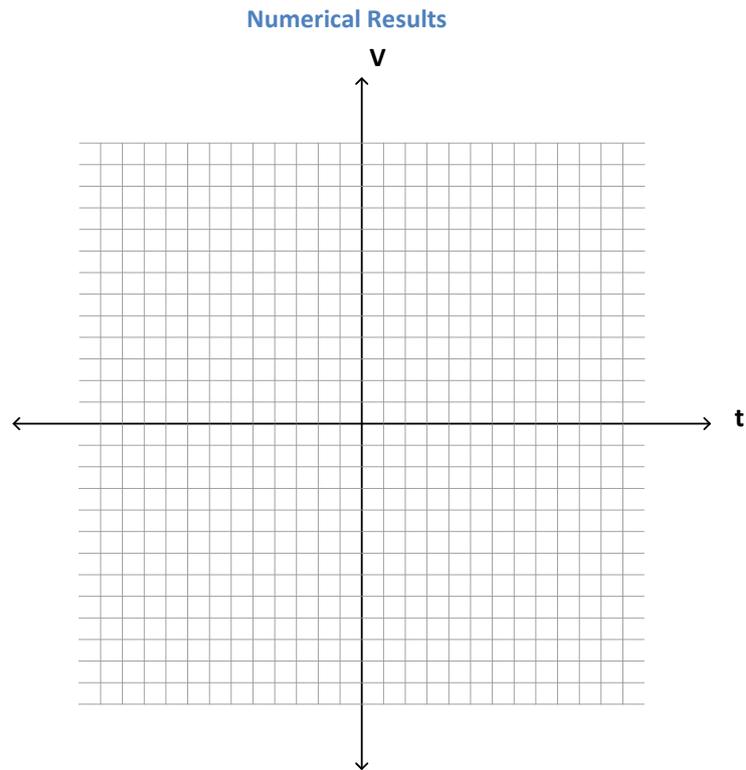
Figure 11.5 – The Output of the differentiator circuit (Simulation)
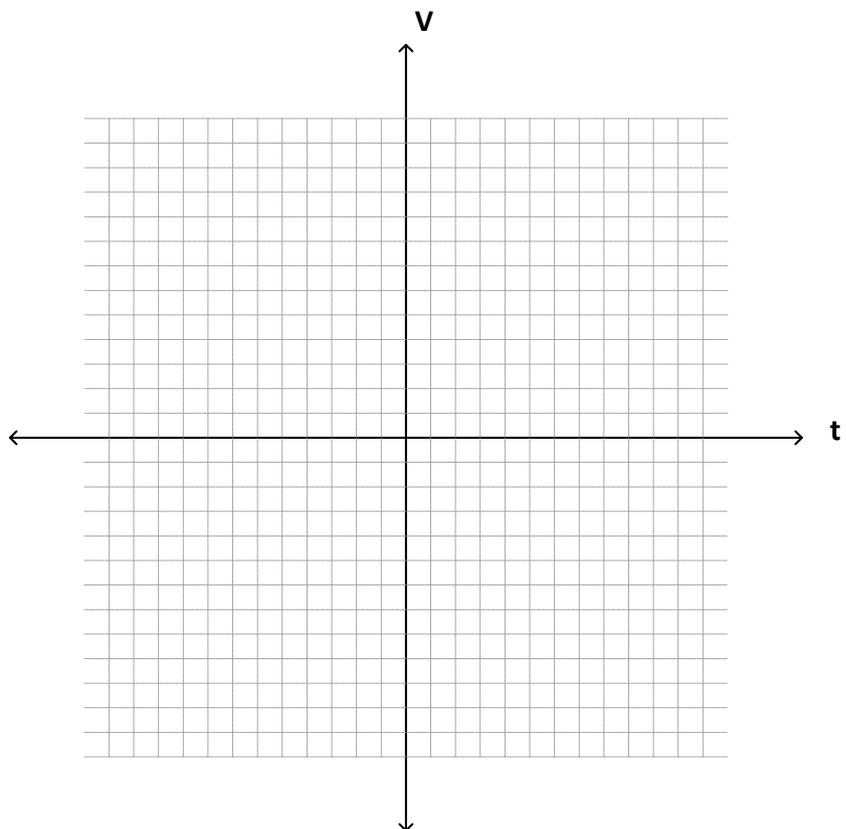

Figure 11.6 – The output of the integrator circuit (Simulation)

116

Experimental Results
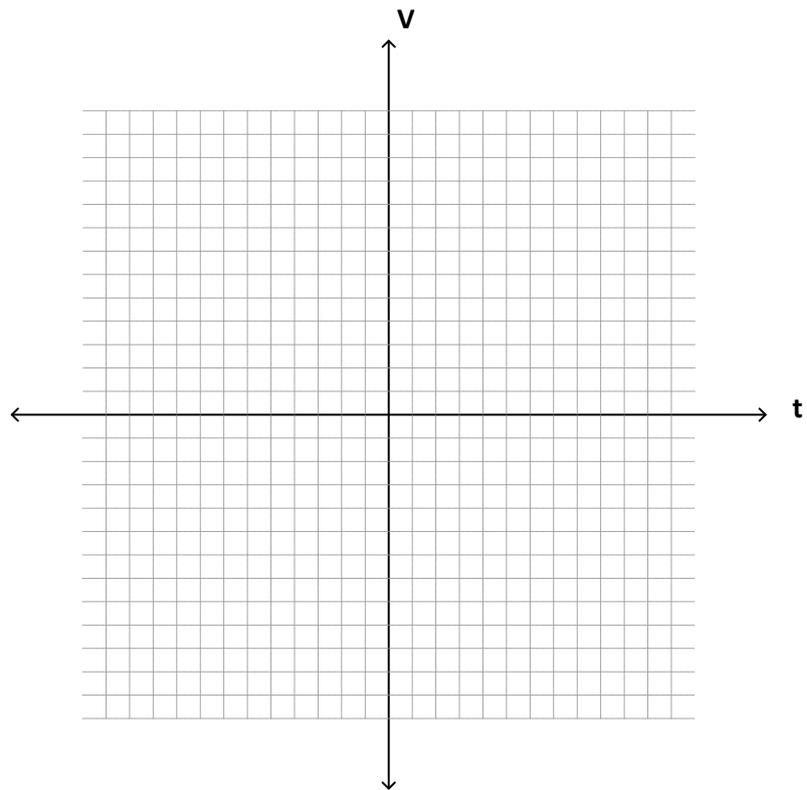
Figure 11.7 – The output of the differentiator circuit (Experimental)
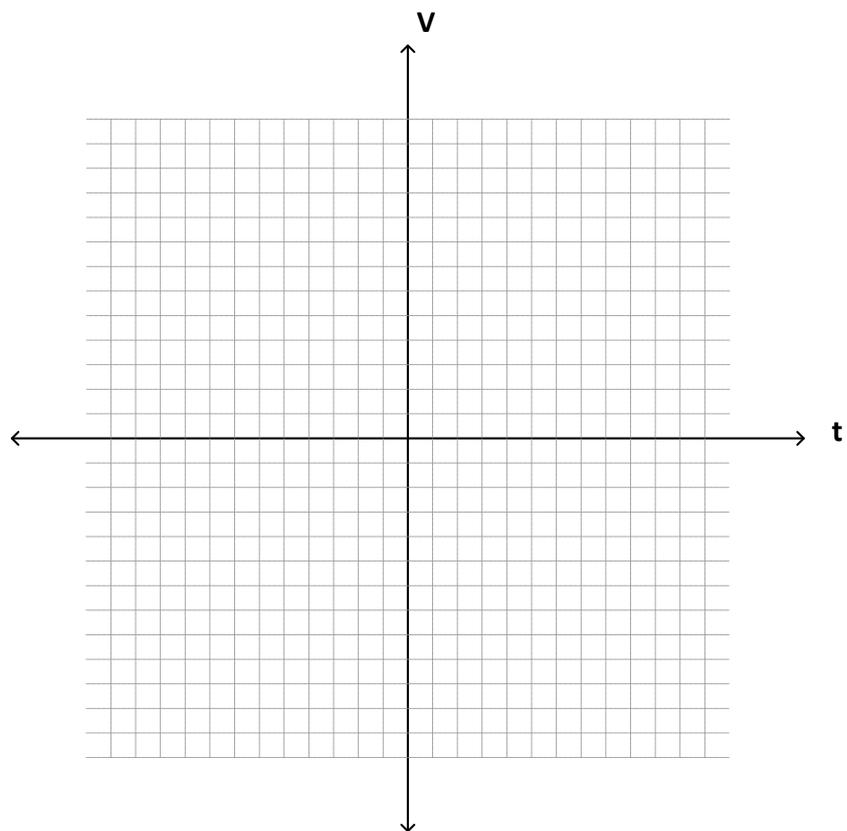

Figure 11.8 – The output of the integrator circuit (Experimental)

Remarks

REFERENCES

- Electronic Devices and Circuit Theory, Pearson Education, Boylestad, R. and Nashelsky, L. 10th ed. 2008
- Elektronik Cihazlar ve Devre Teorisi, Boylestad, R. and Nashelsky, L. (Türkçe Çeviri), Palme Yayıncılık, 2012.
- TEKO elektrik-elektronik eğitim setleri deney föyleri.
- http://www.baskent.edu.tr/~kcevik/eem214/
- http://eem.mf.duzce.edu.tr/Dokumanlar/eem_mf/
- http://mimoza.marmara.edu.tr/~kenan.savas/categories/ders_notlari/deney_foyleri/
- http://eng.harran.edu.tr/~nbesli/ETK/
- www.diyot.net
- www.robotiksistem.com
- www.maker.robotistan.com
- www.devreyapimi.com
- https://rdl.train.army.mil
- www.rapidonline.com/national-semiconductor-lm741-single-op-amp-dil8-82-0458
- https://www.conceptofeverything.com
- http://www.elektrotekno.net/rss.xml
- http://www.diyot.net/
- http://320volt.com/opamp/

Erhan AKDOĞAN

Dr. Akdoğan graduated from Yıldız Technical University, Electrical and Electronics Faculty, Electronics and Communication Engineering Department in 1999. He completed his MSc and Ph.D. in Marmara University, where he worked as Research Assistant. Between 2008 and 2009, he was in postdoctoral research at Hiroshima University, Japan. In September 2010, he started to work as a Ass. Prof. Dr. in Yıldız Technical University, Department of Mechatronics Engineering. In April 2015, he received the title of Associate Professor. He still works in the same department.

Mehmet Hakan DEMİR

Dr. Demir graduated from Kocaeli University, Engineering Faculty, Department of Mechatronics Engineering in 2009. After completing his MSc at the Department of Mechatronics Engineering at Istanbul Technical University, he received his Ph.D. in Department of Mechanical Engineering at Yıldız Technical University. Since April 2017, he has been working as Assistant Prof. Dr. in İskenderun Technical University, Deparament of Mechatronics Engineering.

Mehmet Emin AKTAN

Mehmet Emin Aktan graduated from Süleyman Demirel University Department of Mechatronics Education in 2010. He completed his MSc in Marmara University, Mechatronics Department in 2012. In 2013, he started his Ph.D. in Yıldız Technical University, Department of Mechatronics Engineering. He worked as a research assistant at Bartın University between 2010-2011 and Marmara University between 2011-2012. Since 2013, he has been working as a Research Assistant in the Department of Mechatronics Engineering at Yıldız Technical University.

Ahmet Taha KORU

Ahmet Taha Koru graduated from Bilkent University, Department of Electrical and Electronics Engineering in 2009. He received MSc degree from same university in 2011 and started to work as a research assistant in Yıldız Technical University, Department of Mechatronics Engineering. In 2017, he received Ph.D. degree in Yıldız Technical University, Department of Control and Automation Engineering. He still working in the same department.