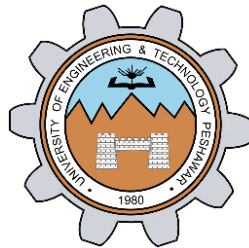**TITLE: VERIFICATION OF OHM'S**

LAW USING PSPICE

SIMULATION

LAB # 03

Spring 2023

CSE103L Circuits & Systems-I Lab

Submitted by: **Haris Khan**

Registration No. : **22PWCSE2161**

Class Section: **A**

"On my honor, as student of University of Engineering and Technology, I have neither given nor received unauthorized assistance on this academic work."

Student Signature: _____   _____

Submitted to:

Dr. Muniba Ashfaq

March 16, 2023

Department of Computer Systems Engineering

University of Engineering and Technology, Peshawar

OBJECTIVES OF LAB

Ohm's law:

Ohm's law is a fundamental law of electrical engineering that describes the relationship between voltage, current, and resistance in an electrical circuit. It states that the current passing through a conductor between two points is directly proportional to the voltage across the two points, provided that the temperature and other physical conditions remain constant.

The law was first formulated by Georg Simon Ohm, a German physicist, in 1827. Ohm conducted a series of experiments in which he measured the current flowing through a wire when different voltages were applied across it. He found that the current was directly proportional to the voltage, and that the ratio between the two was constant. This ratio is known as the resistance of the wire and is measured in ohms.

Mathematically, Ohm's law can be expressed as,

$$V = IR$$

Where V is the voltage (measured in volts), I is the current (measured in amperes), and R is the resistance (measured in ohms). This formula states that the voltage (V) applied across a conductor is equal to the product of the current (I) passing through it and the resistance (R) of the conductor.

In practice, Ohm's law is used to analyze the behavior of circuits and components in electrical engineering. It allows engineers to predict how changes in voltage, current, and resistance will affect the behavior of a circuit, and to design circuits that operate within safe limits.

It's important to note that Ohm's law only applies to conductors that have a constant resistance over a wide range of voltages and currents, such as resistors. For other components, such as diodes and transistors, the relationship between voltage and current is more complex and may not follow Ohm's law.

PSpice Simulation:

PSpice is a simulation software package used in electronic circuit design and analysis. It is widely used by electrical engineers to simulate and analyze complex circuits before building a physical prototype. The software provides a graphical user interface that allows engineers to build circuits using pre-built models and simulate the behavior of the circuits under different conditions.

The PSpice simulation software is developed by Cadence Design Systems and provides a wide range of simulation options, including transient, DC, AC, and frequency-domain analysis. It

supports both analog and digital circuits, as well as mixed-signal designs that combine both analog and digital components.

To use PSpice, engineers start by selecting electronic components from a library of pre-built models. These models include a wide range of electronic components such as resistors, capacitors, inductors, transistors, diodes, and op-amps. Engineers can then drag and drop these components onto the circuit schematic to build the circuit.

Once the circuit is built, engineers can set up the simulation parameters, such as the input signal, the simulation type, and the simulation time. The simulation can then be run, and the software will calculate the output signals and provide various simulation results, such as waveforms, frequency responses, and time-domain plots.

PSpice also provides a range of analysis tools to help engineers optimize the performance of their circuits. These tools include design rule checks, sensitivity analysis, and Monte Carlo simulations. The software also supports co-simulation with other software tools, such as MATLAB and Simulink.

In addition to simulating and analyzing circuits, PSpice can also be used to create models of electronic components for use in other software tools. These models can be exported in a variety of formats, including Verilog, VHDL, and IBIS.

Overall, PSpice is a powerful simulation tool that can help engineers design and analyze complex electronic circuits. It can save time and money by allowing engineers to simulate and optimize circuits before building a physical prototype, and can also be used to create models of electronic components for use in other software tools.

Procedure:

1) Identify the components: Identify the components we will need for our circuit, such as resistors, power supplies, and wires.
2) Determine the voltage: Determine the voltage that we will need to power the circuit.

3) Calculate the resistance: Calculate the resistance we will need for our circuit. We can use Ohm's law, which states that resistance (R) is equal to voltage (V) divided by current (I), or $R = V/I$.
4) Choose the resistors: Choose the resistors that will give us the desired resistance. Resistors come in different values, so choose the one that is closest to the resistance we calculated.
5) Draw the circuit diagram: Draw the circuit diagram using a schematic drawing tool or by hand. Use symbols to represent each component, such as a resistor symbol for the resistors and a power supply symbol for the voltage source.

6) Connect the components: Connect the components together as per the circuit design. Use wires to connect the components to each other and to the power supply.
7) Verify the circuit: Once the circuit is drawn, verify that it meets the requirements of the design, such as the desired resistance and voltage.
8) Test the circuit: Test the circuit by connecting it to a power source and measuring the voltage and current using a multimeter. Verify that the measured values match the design specifications.
9) Make any necessary adjustments: If the measured values do not match the design specifications, make any necessary adjustments to the circuit design and re-test.
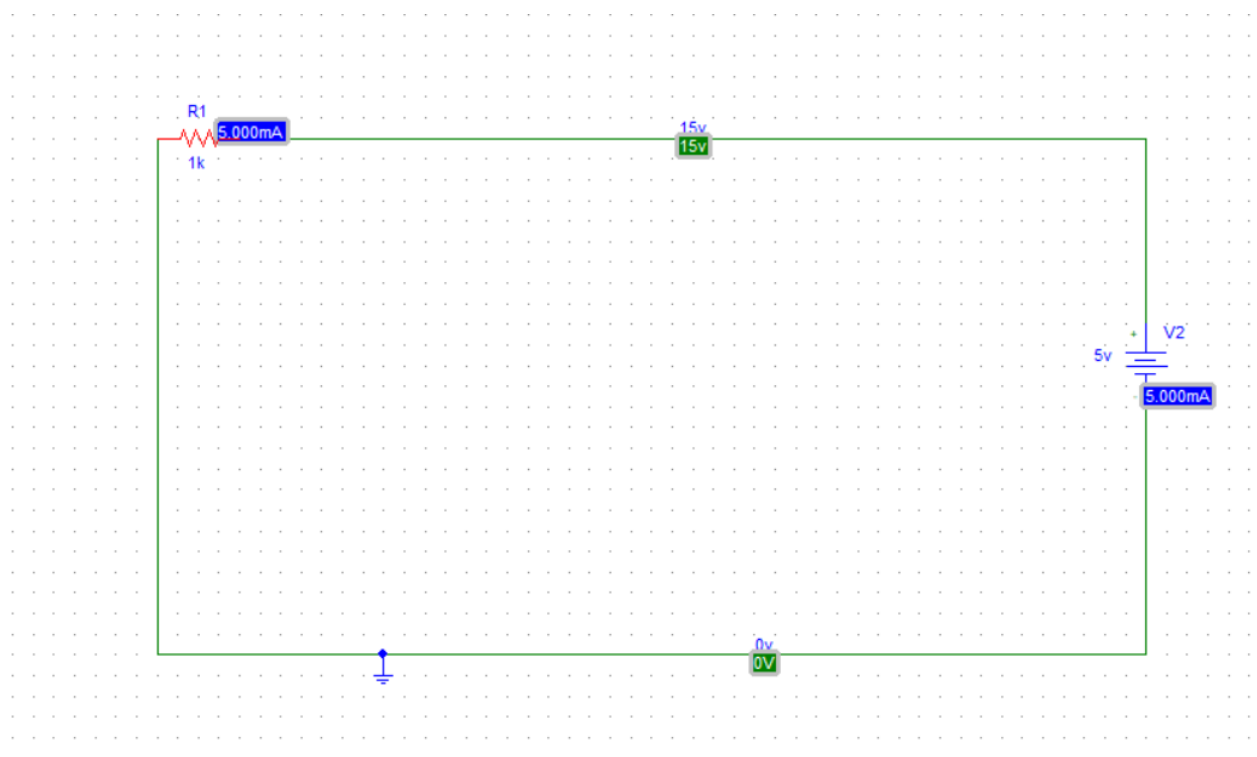
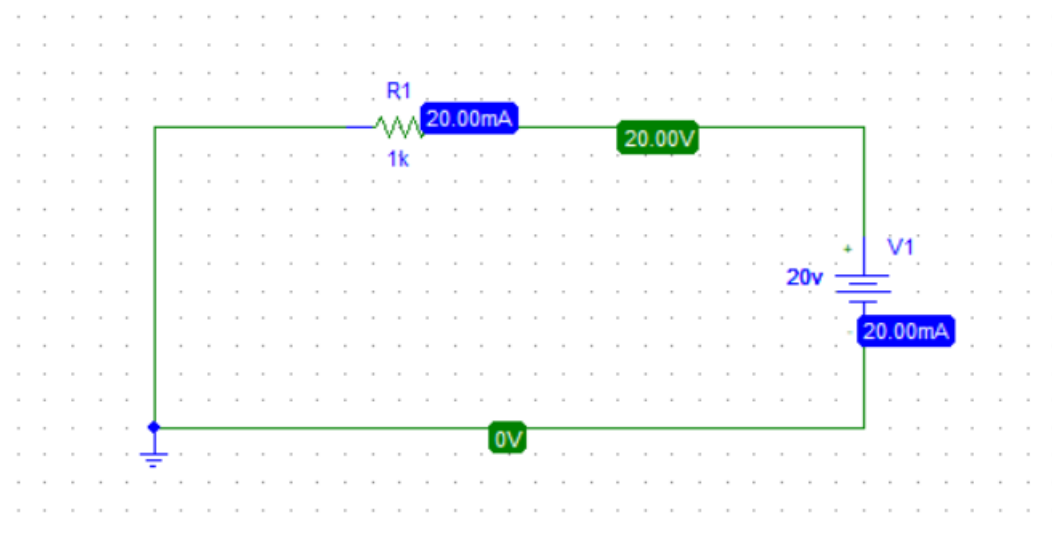# *Observations and calculations:*
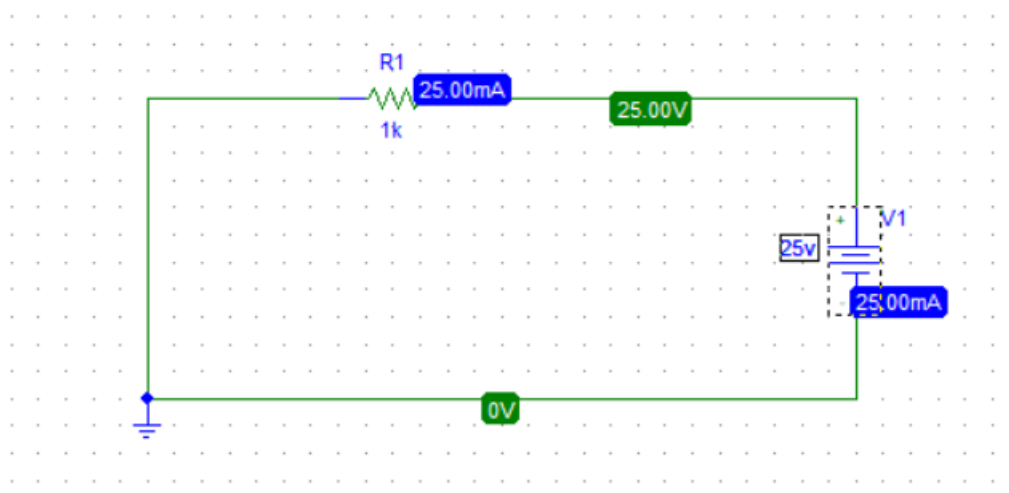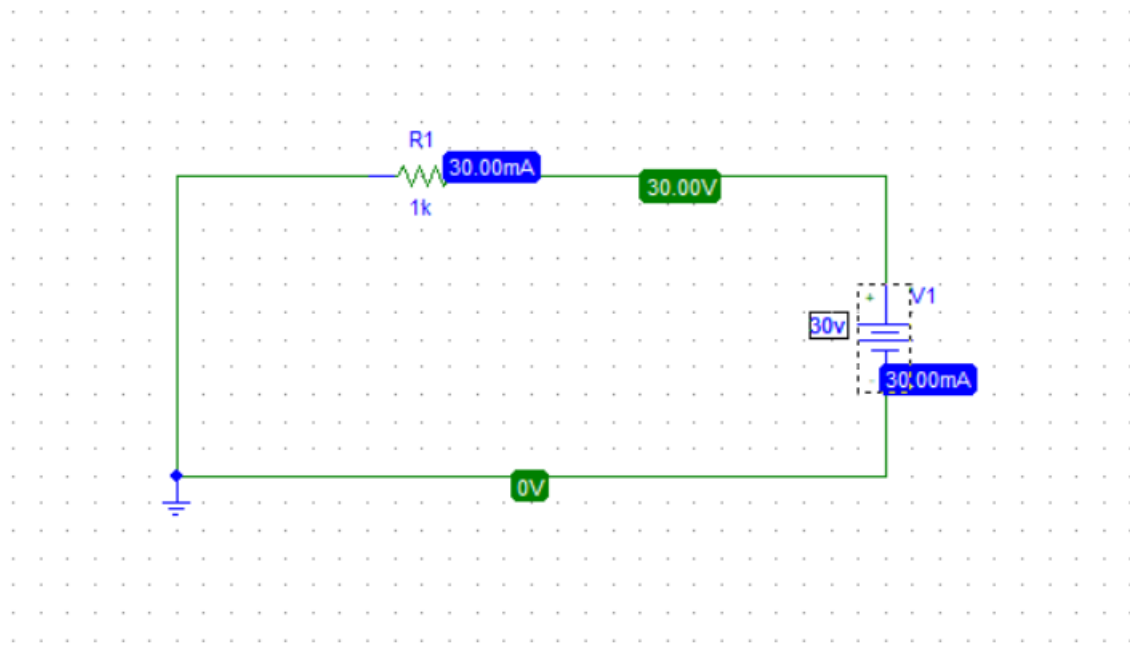
1) *Schematic Diagram:*
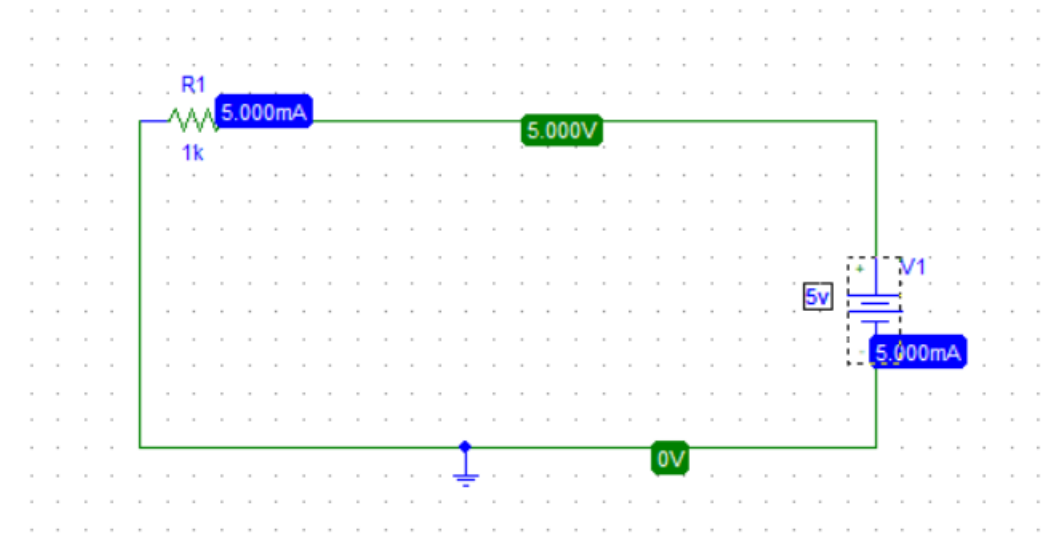
5V
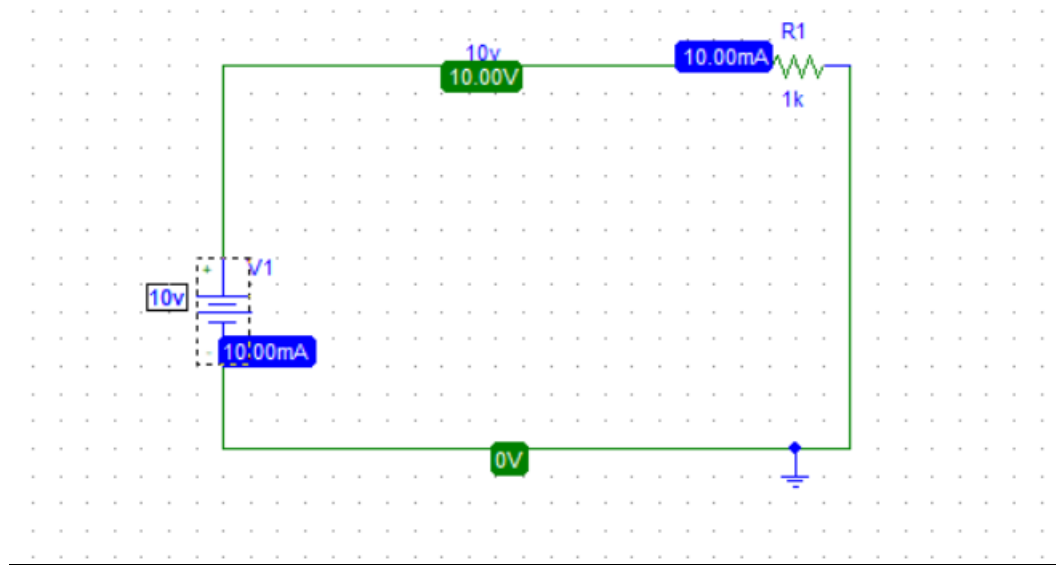
10V

15V

20V

25V

30V
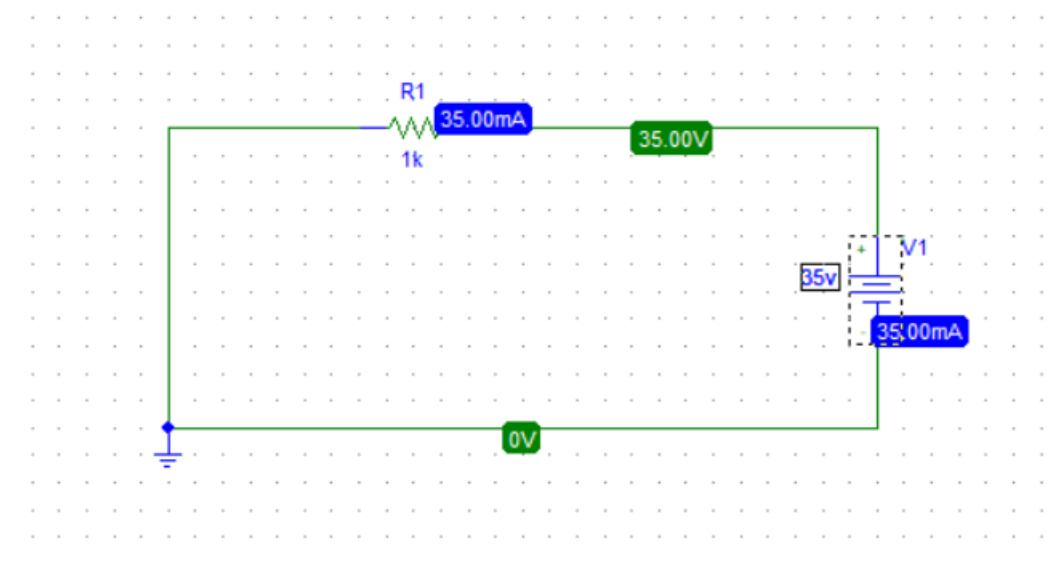
35V
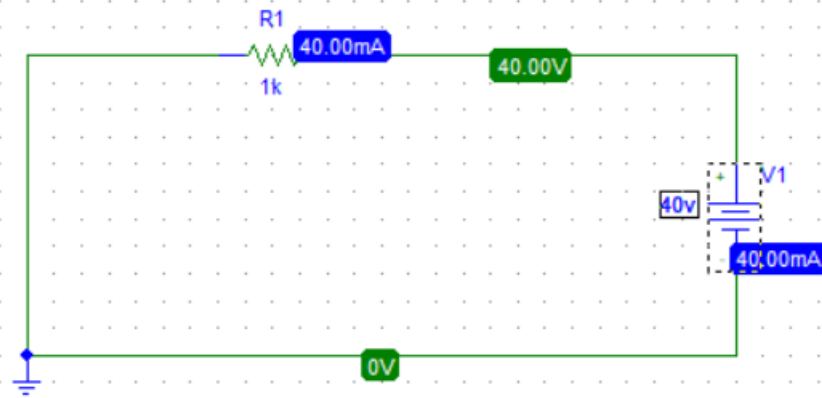
40V
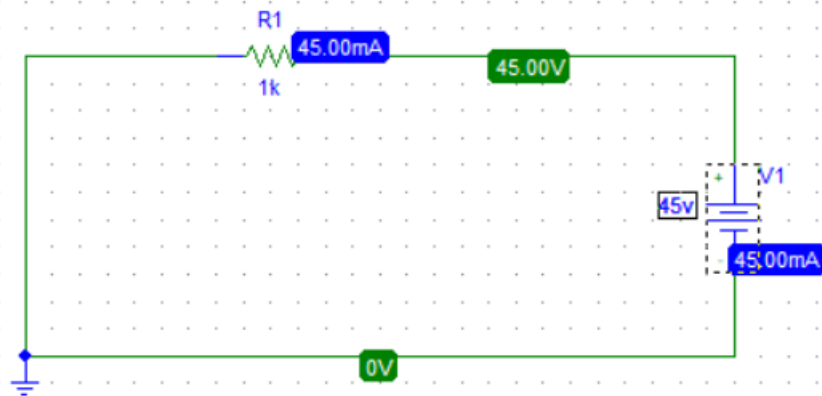
45V
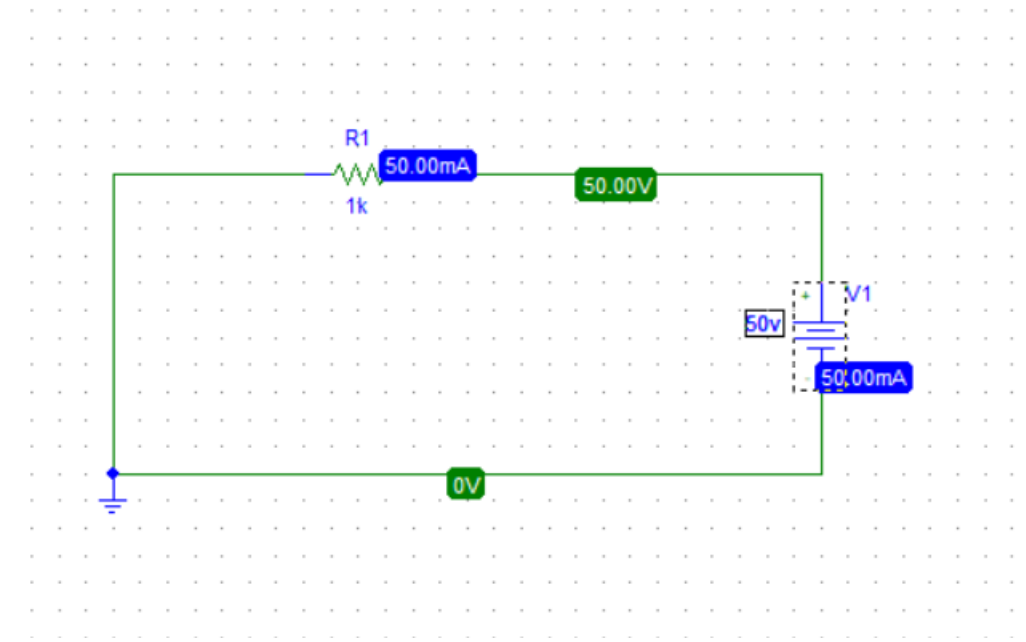
50V

2) *Table:*

S.No	Voltage	Current	Resistor
1.	5V	5mA	1K
2.	10V	10mA	1K
3.	15V	15mA	1K
4.	20V	20mA	1K
5.	25V	25mA	1K
6.	30V	30mA	1K
7.	35V	35mA	1K
8.	40V	40mA	1K
9.	45V	45mA	1K
10.	50V	50mA	1K

3) *Graph:*
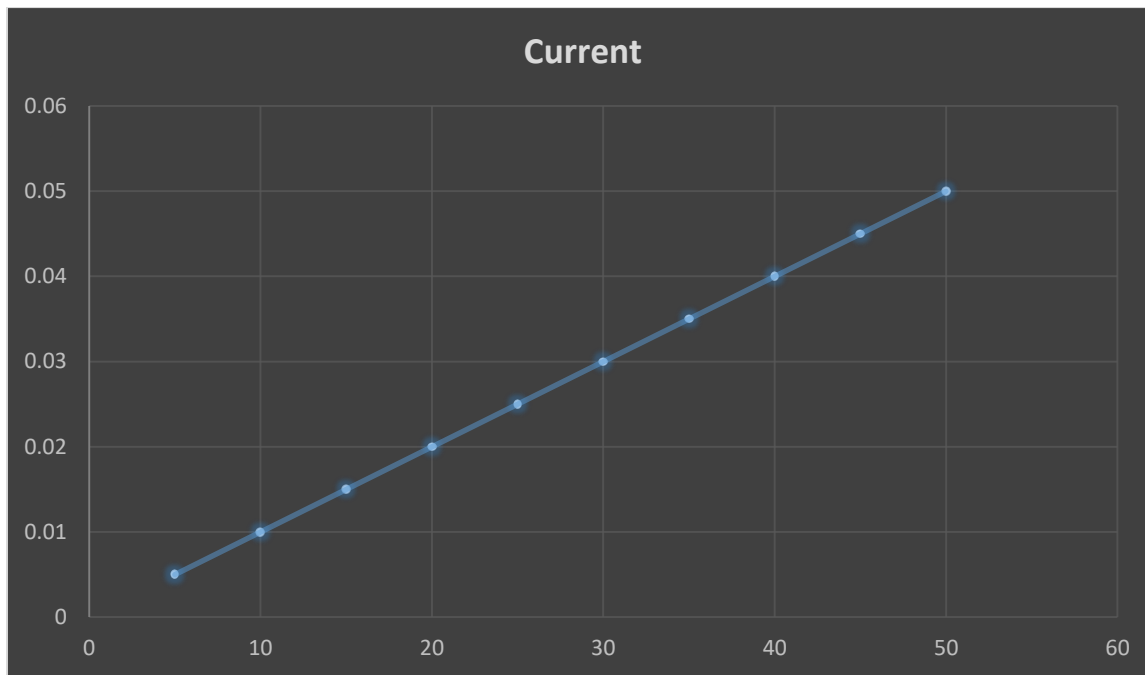
Voltage= X-Axis

Current= Y-Axis

4) *Analysis:*

Ohm's law is a fundamental concept in electrical engineering that relates the current flowing through a conductor to the voltage applied across it and the resistance of the conductor. It states that the current flowing through a conductor is directly proportional to the voltage applied across it and inversely proportional to its resistance. Mathematically, Ohm's law can be expressed as:

$$V = I*R$$

Where V is the voltage applied across the conductor, I is the current flowing through it, and R is the resistance of the conductor.

Verification of Ohm's law can be done through experimental measurements using a circuit consisting of a power source, a resistor, and a voltmeter and an ammeter. The voltage across the resistor is measured using the voltmeter, and the current flowing through the resistor is

measured using the ammeter. By varying the resistance of the resistor and measuring the voltage and current, the relationship between the two can be verified to be in accordance with Ohm's law.

PSpice simulation is a software tool used for electronic circuit analysis and design. It can be used to simulate the behavior of electronic circuits and verify their performance. PSpice simulation can be used to verify Ohm's law by simulating a circuit consisting of a voltage source, a resistor, and a current sensor. The voltage source is set to a known value, and the resistance of the resistor is varied while the current is measured using the current sensor. The simulation results can be compared to the expected values based on Ohm's law, and any discrepancies can be analyzed to identify potential issues with the circuit design or simulation setup.

Overall, both experimental verification and simulation using PSpice can be used to verify Ohm's law and ensure that electronic circuits are designed and functioning correctly.

LAB RUBRICS: (Circuits & Systems-I Lab)

Criteria & Point Assigned	Outstanding 4	Acceptable 3	Considerable 2	Below Expectations 1
Attendance and Attentiveness in Lab PLO10	Attended in proper Time and attentive in Lab	Attended in proper Time but not attentive in Lab	Attended late but attentive in Lab	Attended late not attentive in Lab
Equipment / Instruments Selection and Operation PLO1, PLO2, PLO3, PLO5,	Right selection and operation of appropriate equipment and instruments to perform experiment.	Right selection of appropriate equipment and instruments to perform experiment but with minor issues in operation	Needs guidance for right selection of appropriate equipment and instruments to perform experiment and to overcome errors in operation	Cannot appropriately select and operate equipment and instruments to perform experiment.
Result or Output/ Completion of target in Lab PLO9,	100% target has been completed and well formatted.	75% target has been completed and well formatted.	50% target has been completed but not well formatted.	None of the outputs are correct
Overall, Knowledge PLO10,	Demonstrates excellent knowledge of lab	Demonstrates good knowledge of lab	Has partial idea about the Lab and procedure followed	Has poor idea about the Lab and procedure followed
Attention to Lab Report PLO4,	Submission of Lab Report in Proper Time i.e. in next day of lab., with proper documentation.	Submission of Lab Report in proper time but not with proper documentation.	Late Submission with proper documentation.	Late Submission Very poor documentation