VERIFICATION OF OHM'SLAW USING PSPICE SIMULATION

LAB # 03

Spring 2023

CSE103L Circuits & Systems-I Lab

Submitted by: **Naveed Ahmad**

Registration No: **22pwcse2165**

Section: **B**

"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 18, 2023)

Department of Computer Systems Engineering

University of Engineering and Technology, Peshawar

OHM'S LAW:

Statement: *"In electrical circuit current is directly proportional to applied voltage of resistance remain unchanged."*

Explanation:

Ohm's law is a fundamental principle in electrical engineering that describes the relationship between voltage, current, and resistance in an electrical circuit. The law states that the current flowing through a conductor between two points is directly proportional to the voltage across the two points, and inversely proportional to the resistance between them. In mathematical form, Ohm's law can be expressed as:

$$V = I \times R$$

where V is the voltage across the two points, I is the current flowing through the conductor, and R is the resistance of the conductor.

This means that if you increase the voltage across a conductor (with the resistance held constant), the current flowing through the conductor will increase proportionally. Conversely, if you increase the resistance of the conductor (with the voltage held constant), the current flowing through the conductor will decrease proportionally. Ohm's law is a simple but powerful tool for understanding and designing electrical circuits, and it is used extensively in electrical engineering and related fields.

MATHEMATICALLY:

$$V=IR$$

OBJECTIVE:

To familiarize with PSPICE and. using it to verify Ohm's law.

- How to install PSPICE.
- how working PSPICE.
- Verification of Ohm's law by PSPICE.

PSPISE:

"PSPICE is a circuit analysis tool that allows the user to simulate a circuit and extract key voltages and currents."

PSPICE is a circuit simulation program that allows us to create circuit schematics and simulate the behavior of electronic circuits. It uses mathematical models of electronic components such as resistors, capacitors, inductors, and transistors to predict how the circuit will behave under different conditions. This can be helpful for designing and testing circuits before they are physically built, as well as for troubleshooting existing circuits.

PSPICE is widely used in the field of electrical and electronics engineering, and it is available from various vendors, including Cadence Design Systems.

STEPS TO USE PSPICE:

- CREATE A CIRCUIT SCHEMATIC: we can use the graphical user interface of PSPICE to create a circuit schematic and can add components to the schematic, connect them together, and define the values of the components.

- DEFINE SIMULATION SETTINGS: we need to define the simulation settings, such as the type of analysis we want to perform (e.g., transient, AC, or DC analysis), the simulation time, and the simulation parameters.

- RUN SIMULATION: Once we have created the circuit schematic and defined the simulation settings, we can run the simulation. The simulation results will show we how the circuit behaves under different conditions.

- ANALYZE RESULTS: we can analyze the simulation results to determine the behavior of the circuit, and can look at the voltage and current waveforms, frequency response, and other characteristics of the circuit.

- OPTIMIZE CIRCUIT can be use the simulation results to optimize the circuit design. And can make changes to the circuit and run another simulation to see how the changes affect the circuit behavior.

- GENERATE REPORTS: Finally, we can generate reports that summarize the simulation results and the circuit design. This can be helpful for documentation and communication with others who may be working on the circuit design.

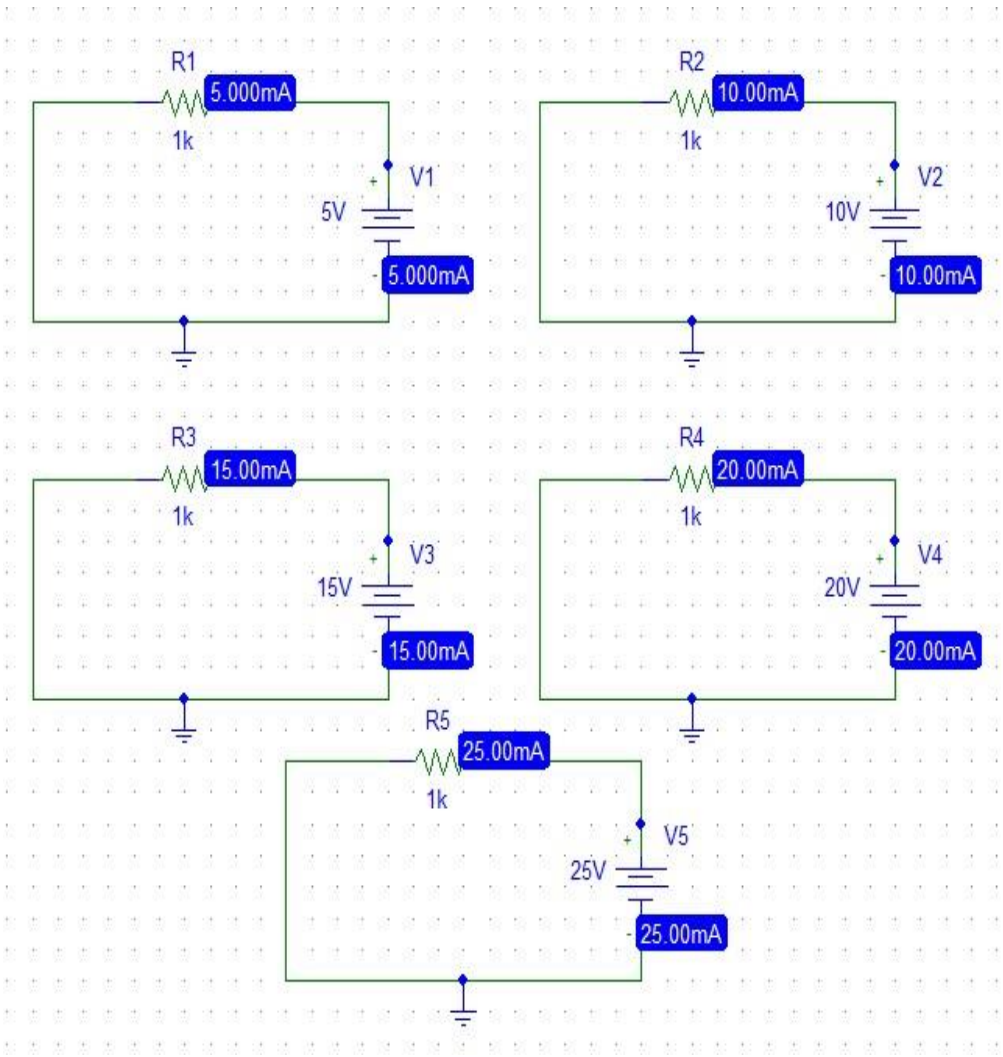


Fig # 1

Fig 1 show voltage apply from 5V to 25v and current from 5.00mA to 25.00mA. from diagram we can conclude that voltage and current have direct relation.

When applied voltage is

V =5v

I=5mA

V =25v

I=25mA and so on.

Here resistance in each circuit is constant (i.e. 1k ohm) in each case.

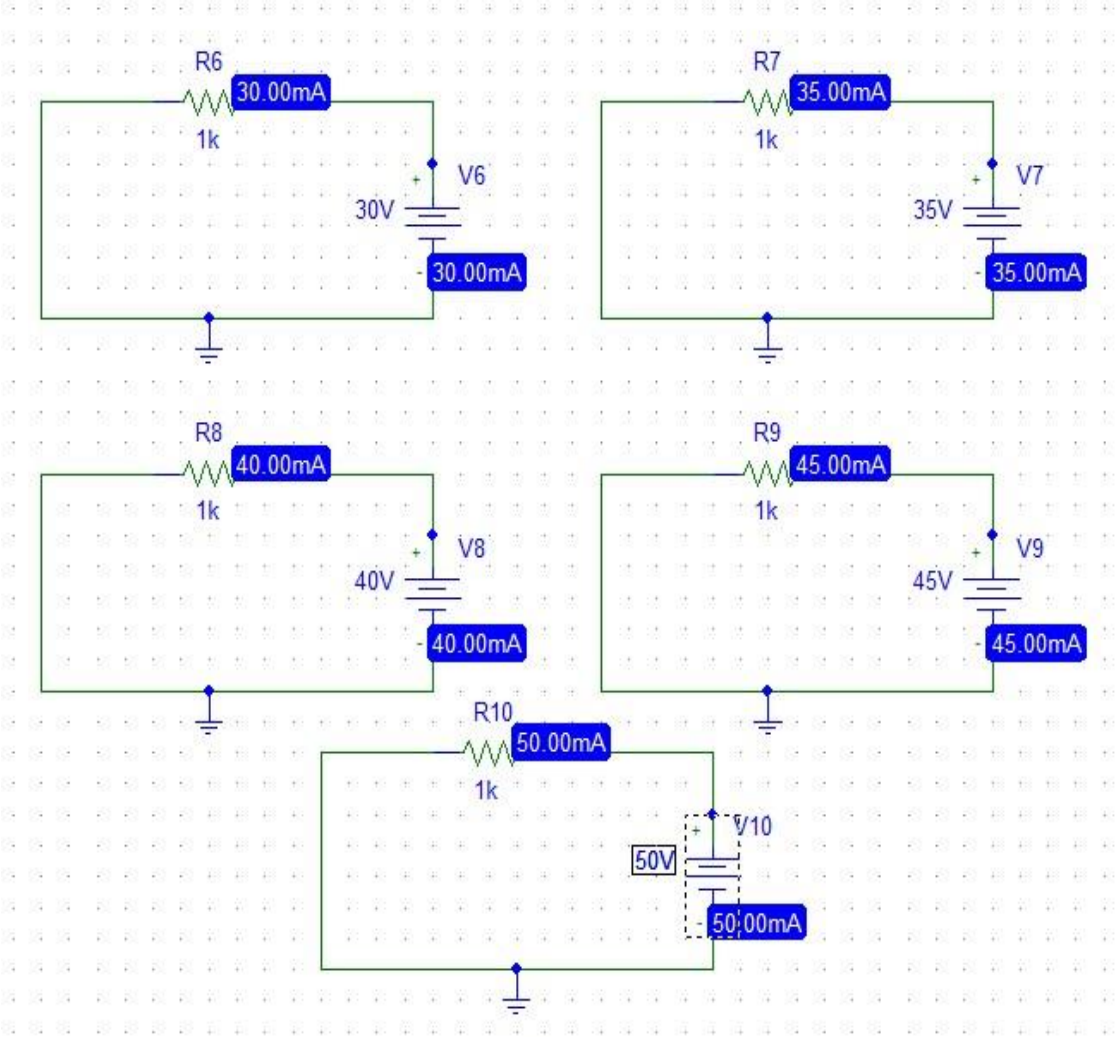


Fig #2

Fig 2 show voltage apply from 30V to 50v and current from 30.00mA to 50.00mA. from diagram we can conclude that voltage and current have direct relation.

When applied voltage is

V =30v

I=30mA

V =50v

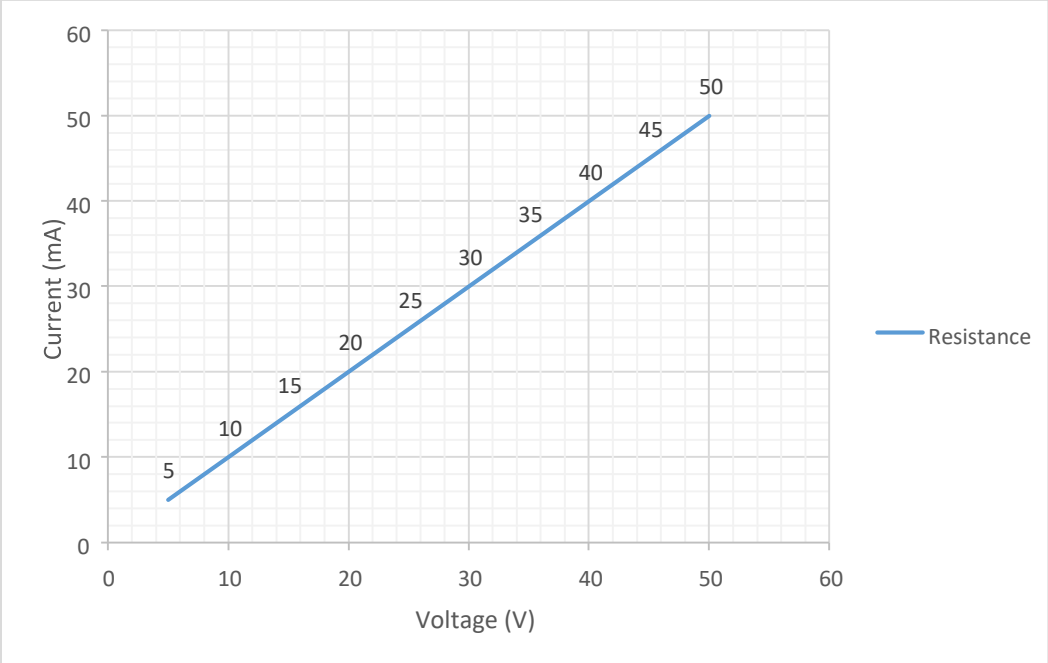
I=50mA

resistance in each circuit is constant (i.e. 1k ohm).

CALCULATION AND OBSERVATION

S No.	Voltage(V)	Resistance (R)	Current(I)
1	5v	1k Ω	5mA
2	10v	1k Ω	10mA
3	15v	1k Ω	15mA
4	20v	1k Ω	20mA
5	25v	1k Ω	25mA
6	30v	1k Ω	30mA
7	35v	1k Ω	35mA
8	40v	1k Ω	40mA
9	45v	1k Ω	45mA
10	50v	1k Ω	50mA

Graph:



Analysis:

From the graph as we table the applied voltage and current have direct relation if resistance unchanged. In other words, if you increase the voltage applied across a resistor, the current flowing through it will increase proportionally, as long as the resistance of the resistor remains constant. Similarly, if you decrease the voltage applied across the resistor, the current flowing through it will decrease proportionally.

This concept is particularly useful for designing and analyzing circuits with resistive elements, such as simple DC circuits, and for understanding the behavior of these circuits under different operating conditions. By applying Ohm's law, engineers can calculate the voltage, current, and resistance of different circuit elements and use this information to optimize the design and performance of the circuit.

Furthermore, the concept of voltage and current in relation to Ohm's law is fundamental to understanding other electrical concepts, such as power, energy, and efficiency, which are important in the design and operation of electrical systems.

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