University of engineering & technology Peshawar

Circuit & system -1

Lab report # 7

Fall 2020

Submitted by: Ashfaq Ahmad

Section: B

Reg No: 19PWCSE1795

Semester: 2nd

Submitted to:

Eng: faizullah sir

Department Of Computer System Engineering

Rubrics of lab

LAB REPORT ASSESSMENT				
Criteria	**Excellent**	**Average**	**Nill**	**Marks Obtained**
1. Objectives of Lab	All objectives of lab are properly covered [Marks 0.5]	Objectives of lab are partially covered [Marks 0.25]	Objectives of lab are not shown [Marks 0]	
2. Kirchoff's Voltage Law, Kirchoff's Current Law, Ohm's Law. (Statement, Mathematical Expression, Circuit Diagram)	Correct KVL, KCL and Ohm's Law statement and mathematical expression is written. Circuit diagram shown is correct and properly labeled [Marks 1]	KCL statement or mathematical expression or circuit diagram is missing or circuit diagram is not properly labeled [Marks 0.5]		
3. PSPICE Simulator	Brief introduction of PSPICE simulator [Marks 1]	Brief introduction of PSPICE simulator Is not shown [Marks 0]		
4. Procedure	All experimental steps are shown in detail [Marks 1.5]	Some of the experimental steps are missing [Marks 1]	Experimental steps are missing [Marks 0]	
5. Observations & Calculations **a) Verification of KCL** **b) Verification of KVL**	All experimental results are completely shown in form of table for all given laws. [Marks 3]	Experimental results are partially shown and some of the observations are missing [Marks 1.5]	No experimental results are shown [Marks 0]	
6. Analysis **a) Analysis about KVL** **b) Analysis about KCL** **c) Analysis about Ohm's Law**	Analysis and discussion about all experimental results are shown [Marks 3]	Analysis and discussion about experimental results are partially shown [Marks 1.5]	Analysis is not shown [Marks 0]	
Total Marks Obtained:_____ Instructor Signature: _____				

Analyzing Complex Circuit through P Spice.

1. Objectives:

- To Verify the Basic Laws of Circuits using P Spice on Complex Circuits.
- We will be able to explain given laws after studding this lab.
  - Kirchhoff's Current Law
  - Kirchhoff's Voltage Law and
  - Ohm`s Law
- To introduce PSPICE software.
- To Know How to Design a Circuit in P Spice.

2. a) Kirchhoff Current Law (KCL):

Kirchhoff's Current Law (KCL) is Kirchhoff's first law that deals with the conservation of charge entering and leaving a junction.

Gustav Kirchhoff's Current Law is one of the fundamental laws used for circuit analysis.

Statement:

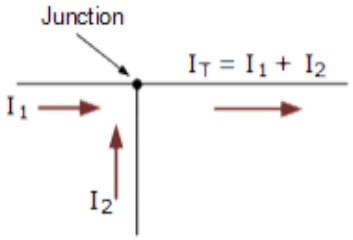
The total current entering a circuits junction is exactly equal to the total current leaving the same junction. This is because it has no other place to go as no charge is lost.

Mathematical Expression:

In other words the algebraic sum of ALL the currents entering and leaving a junction must be equal to zero as:

$$\Sigma I_{IN} = \Sigma I_{OUT}.$$

Circuit Diagram:



Circuit diagram for Kirchhoff current law

b) Kirchhoff Voltage Law (KVL):

Kirchhoff's Voltage Law (KVL) is Kirchhoff's second law that deals with the conservation of energy around a closed circuit path.

Gustav Kirchhoff's Voltage Law is the second of his fundamental laws we can use for circuit analysis.

Statement

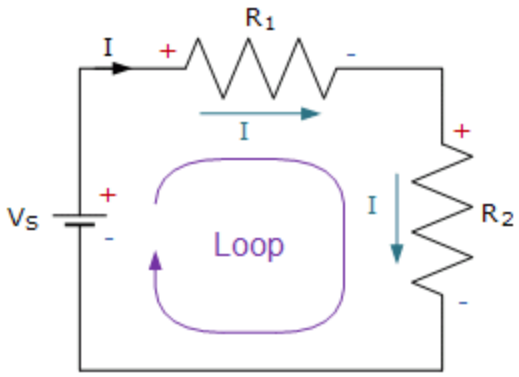
For a closed loop series path **the algebraic sum of all the voltages around any closed loop in a circuit is equal to zero**.

Mathematical Expression:

In other words the algebraic sum of ALL the potential differences around the loop must be equal to zero as:

$$\Sigma V = 0.$$

Circuit Diagram:



c) Ohm`s Law:

This relationship between current, voltage, and relationship was discovered by German scientist Georg Simon Ohm.

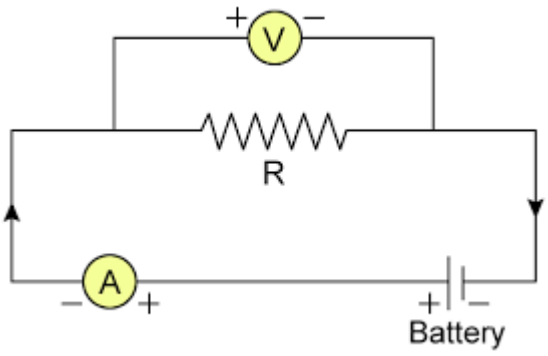
Statement

Ohm's law states that the voltage or potential difference between two points is directly proportional to the current or electricity by keeping resister and other physical condition constant.

Mathematical Expression:

V=IR.

Circuit Diagram:

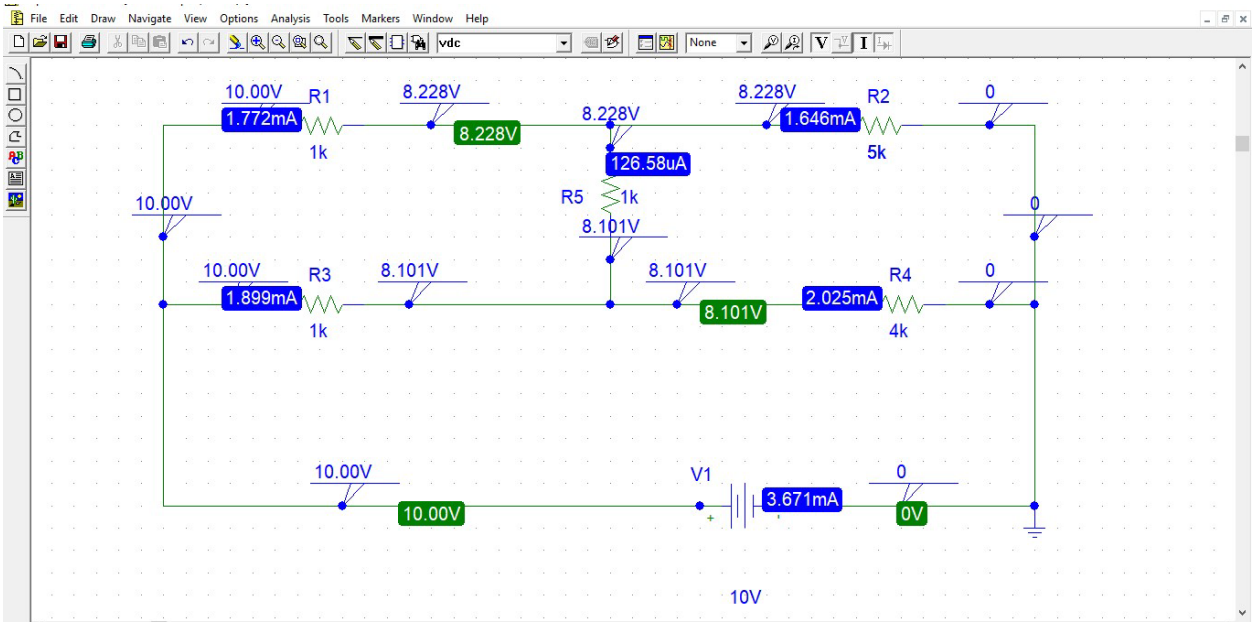


P Spice:

PSPICE is a circuit analysis tool that allows the user to simulate a circuit and extract key voltages and currents. Information is entered into PSPICE via one of two methods; they are a typed 'Net List' or by designing a visual a schematic which is transformed into a netlist.

Procedure:

- Open schematic program of Pspice.
- Click on the "Get New Part" button on the toolbar
- Type 'r' in the search bar and place three the resistors on the white sheet
- Type 'vdc' in the search bar and place it on the white sheet
- Type 'gnd-earth' and place on the white sheet
- Now arrange these components on the white sheet according to the circuit
- Click on the simulation button in the toolbar and also make sure



that the voltage and current biase buttons are pressed so that you can take readings of circuit.

OBSERVATIONS:

Verification of KCL:

Taking a Node:

$I_1 - I_2 - I_5 = 0$, $I_1 = I_2 + I_5$

$I_1 = 1.772mA$, $I_2 = 126.58\mu A$, $I_5 = 1.646mA$

Putting values:

$1.772mA = 126.58\mu A + 1.646mA$

$1.772mA = 1.772mA$

So, Current entering through a node is equal to current leaving a node.

Verification of KVL:

Taking the second loop:

Voltage drops at three resistors

Voltage drop (1) = 0-8.228 = -0.8228V

Voltage drop (2) = 8.101-0 = +8.101V

Voltage drop (3) = 8.228-8.101 = +0.127V

$V_2 + V_4 + V_5 = 0$

-8.228V + 8.101V + 0.127V = 0

0=0

Hence sum of voltages across loop is zero verified.

CONCLUSION:

CONCLUSION ABOUT KCL:

As seen from the Equation above in the Verification of KCL, The current entering is equal to the current leaving the node, Thus Proving the Kirchhoff`s Current Law.

CONCLUSION ABOUT KVL:

Similarly, The potential differences in the Loop Sum up to Zero and Hence proving the Second Law by Kirchhoff which is about Voltage.

The end