Circuits And System 1

CSE103L Circuits & Systems-I Lab

LAB REPORT # 6

2020

Submitted to:

Engr. Faiz Ullah

Submitted by:

TAYYABA

Registration No:

19PWCSE1854

Semester: **2nd**

Class Section: **C**

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

Student Signature: _____

Thursday, March 12, 2020

Department of Computer Systems Engineering

University of Engineering and Technology, Peshawar

ASSESSMENT RUBRICS LAB # 6

Verification of KCL using PSPICE

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. Kirchhoff's Current Law (Statement, Mathematical Expression, Circuit Diagram)	Correct KCL 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	All experimental results are completely shown in form of table for both cases of using same resistors and for different resistors with varying applied source voltage [Marks 4]	Experimental results are partially shown and some of the observations are missing [Marks 2]	No experimental results are shown [Marks 0]	
6. Analysis	Analysis and discussion about all experimental results are shown [Marks 2]	Analysis and discussion about experimental results are partially shown [Marks 1]	Analysis is not shown [Marks 0]	
Total Marks Obtained:_____ Instructor Signature: _____				

Verification of Kirchhoff Current Law (KCL) using PSPICE

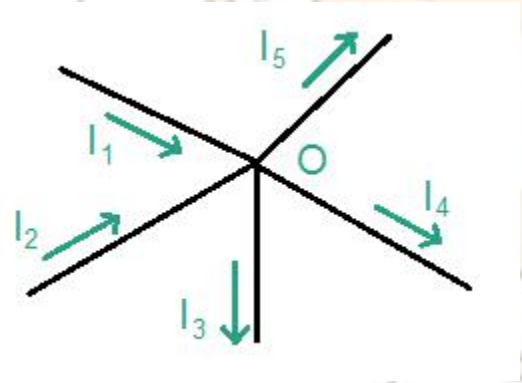
Objectives:-

1. To learn about Kirchhoff's Current Law (**KCL**) , its mathematical expression and its circuit diagram..
2. Learn to draw circuit for **KCL** in **PSPICE** software.
3. To verify **KCL** using **PSPICE** software.

Kirchhoff's Current Law:-

Kirchhoff's Current Law states that the sum of current into a junction is equal to the sum of current out of junction. The junction is a point where two or more then current paths joins together.

Circuit Diagram:-



Mathematical Expression:-

According to above diagram:

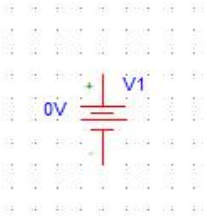
$$I_{in} = I_{out}$$

$$I_1+I_2 = I_3+I_4+I_5$$

$$I_1+I_2 - I_3 - I_4 - I_5 = 0$$

PSPICE Simulator:-

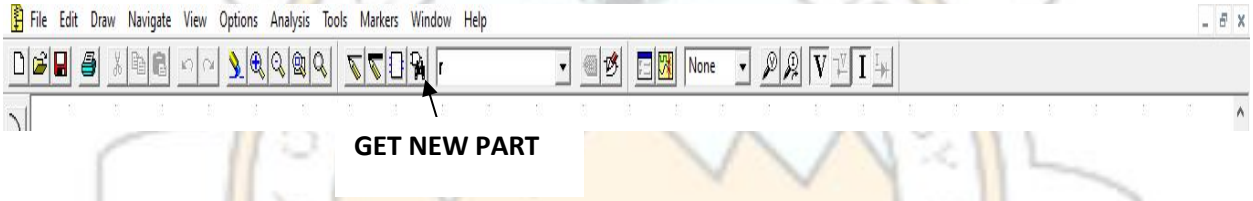
PSPICE is a computer-aided simulation program that enables you to design a circuit and then simulate the design on a computer. As this is one of its main purposes, it is used extensively by electronic design engineers for building a circuit and then testing out how that circuit will simulate. There are a lot of things we can do with **PSPICE**, but the most important things for you to learn are



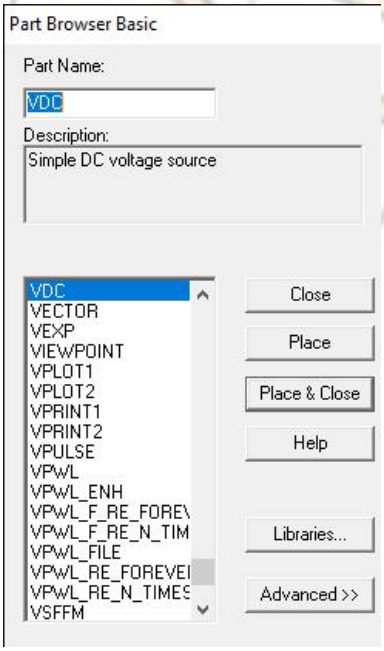
1. Design and draw circuits.
2. Simulate circuits.
3. Analyze simulation results.

Procedure:-

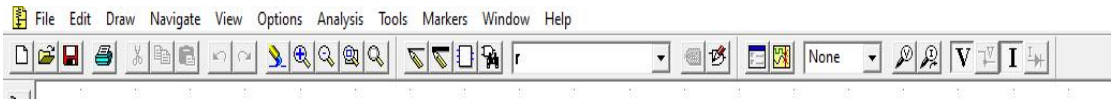
1. Run the **PSPICE** software. A blank window will be opened.
2. Click on "**Get New Part**" from toolbar.



3. Type on **part name** and name part we want. Suppose we want **DC Voltage** so we will type **VDC**



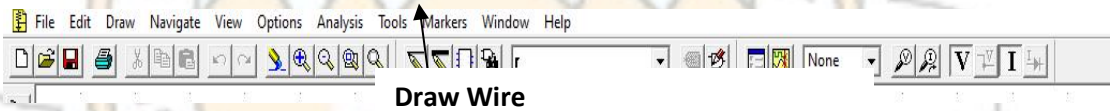
4. Place it and assign its name and set the voltage.



5. Now

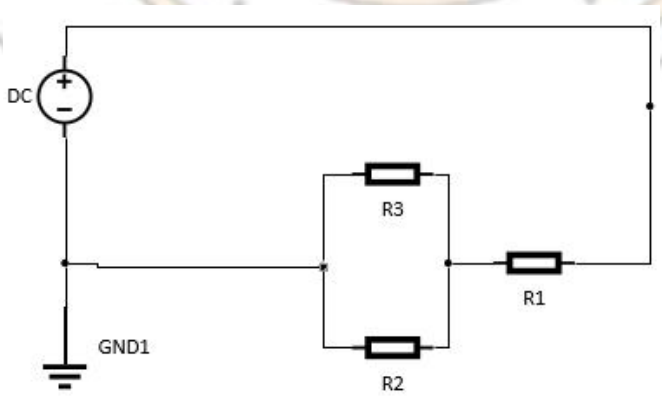
again click on "**Get New Part**" and type **r**. place one such that the resistor is connected in **series** and place two resistor such that they are connected in **parallel**. Assign the resistance to each resistors.

6. Connect the whole circuit using "**Draw wire**" from toolbar.



Draw Wire

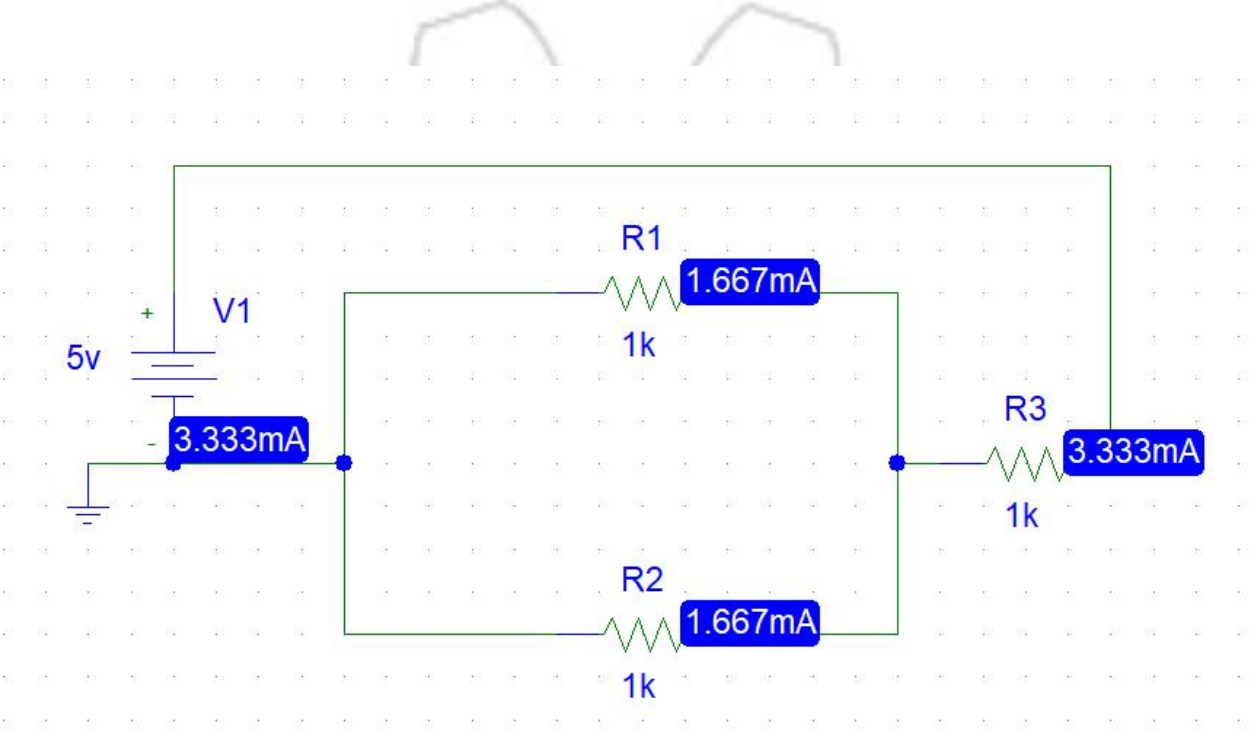
7. Then again click on "**Get New Part**" and type **GND** and place ground at the end of circuit.
8. Arrange the component according to this diagram:



9. Now click on "Simulate".

Observation & Calculation:-

Case 1 (Same Resistors):-



Table(Case 1):-

Sr. No.	Resistance	Current
1	1k Ω	166.67 uA
2	1k Ω	166.67 uA
3	1k Ω	3.333 uA

Analysis:-

As from experimental data we can easily observe that the current going inside the circuit is equal to current going outside the circuit. As in above diagram, the flowed to **R3** and **R1** and **R2** . Their sum is equal to total current supplied to that point.

I3 = I1 + I2

3.333 uA = 166.67 uA + 166.67 uA

Case 2 (Different Resistors):-

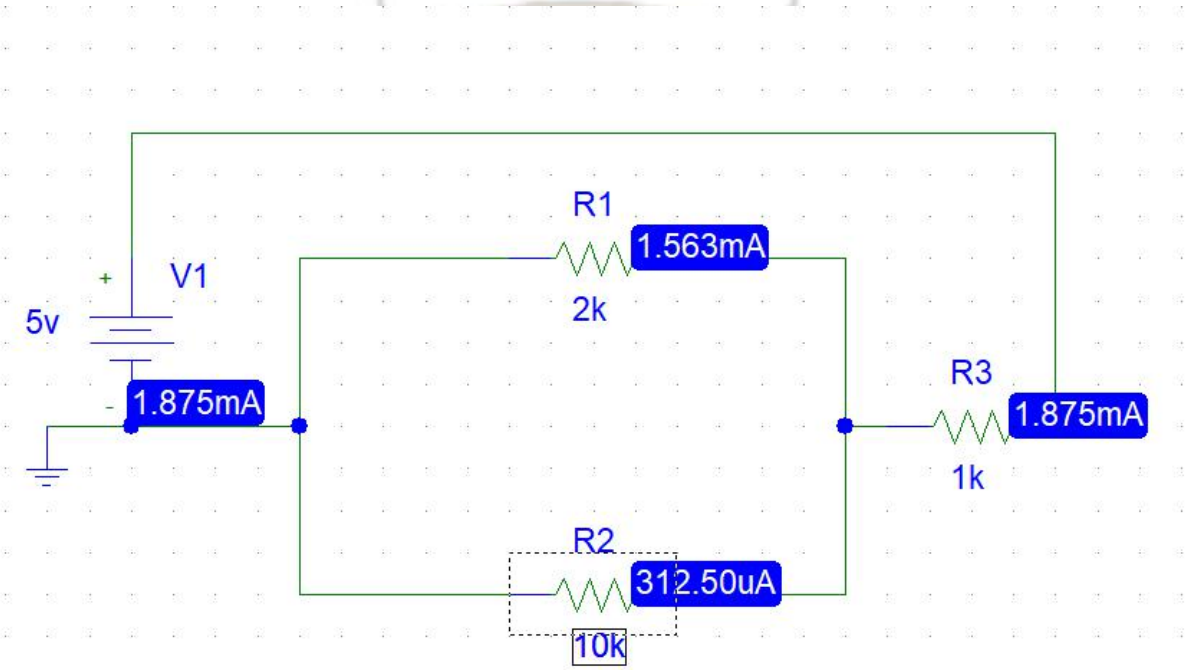


Table (Case 2):-

Sr. No.	Resistance	Current
1	2k Ω	1.563 mA
2	10k Ω	312.5 uA
3	1k Ω	1.875 mA

Analysis:-

As from experimental data we can easily observe that the current going inside the circuit is equal to current going outside the circuit. As in above diagram,current flowes from **R3** to **R1** and **R2** . Their sum is equal to total current supplied to that point.

I3 = I1 + I2

1.875mA = 312.50uA + 1.563mA

1.875mA = 1.875mA