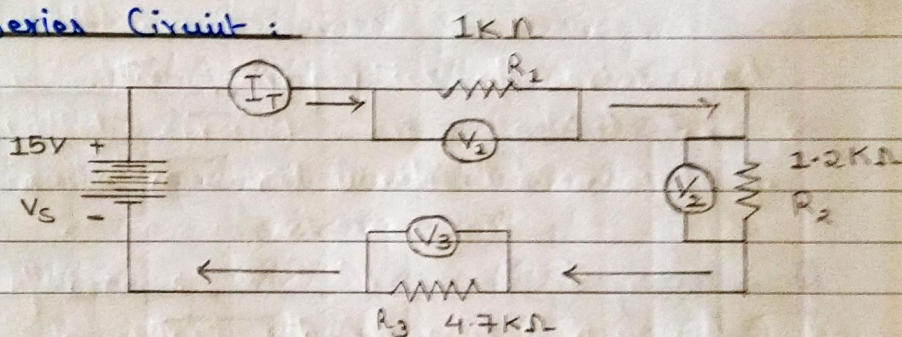| Exp no: 4	Title :
Data :	Series and Parallel
12-11-2021	Connection

Aim:

To study the properties of series and parallel connection.
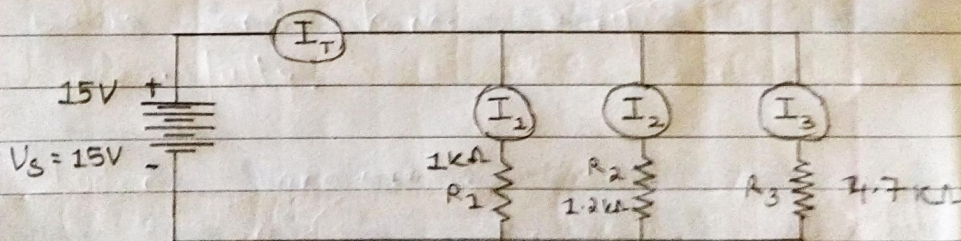
Apparatus :

LT spice software tool

Circuit Diagram:

Series Circuit :

Parallel Circuit :

Procedure:

1) Draw the series the and parallel circuits in the LT spice schematic.

2) Apply the voltage and resistance values

3) Label the nodes at appropriate places in the circuits.

4) Go to simulate tab and select edit Simulation command.

5) Select operating point analysis in the edit Simulation command.

6) Run the simulation.

7) Calculate the potential difference across each resistor and check for $V_s = V_1 + V_2 + V_3$.

8) Calculate the current through each resistor and check for $I_T = I_1 + I_2 + I_3$.

Theoretical Calculations:

Calculate the current through each resistor and check for $I_T = I_1 + I_2 + I_3$

$$I(R_1) = 0.015$$
$$I(R_2) = 0.0125$$
$$I(R_3) = 0.00319\#149$$
$$I_T = 0.03069\#15$$

for $V_s = V_1 + V_2 + V_3$

$$V(R_1) = 2.1739$$
$$V(R_2) \quad 2.6087$$
$$V(R_0) \quad 10.2174$$
$$V_s \approx 15 \quad 14.9961 \approx 15$$

Comparison of theoretical values to the simulated values:

	Theoretical values	Simulated values
V_s (Volt)	15	15
I_T (mA)	0.0306915	0.0306915

Result:
The properties of series and parallel circuits are studied through simulation and verified successfully.

Inference:
The theoretical value is the same as the same as the simulated values and hence verified successfully.

Student Signature:

S. P Ashwath

(Name: Ashvath Suresh Babu Piriya)