Verification of Kirchhoff's Current and Voltage law

Date:	23/9/2021	**Name:**	Dhruv Karmokar
Experiment No:	1	**Reg. No:**	21BAI1604

Aim:

To verify Kirchhoff's current law and voltage law for a simple resistive circuit.

Software required: LTSpice software

Theory:

Kirchhoff's current law: Kirchhoff's Current Law states that "The algebraic sum of all currents entering and exiting a node must be equal to zero."

Kirchhoff's voltage law: Kirchhoff's Voltage Law states that "The algebraic sum of all voltages in a loop must equal zero"

Procedure:

1 Install LTspice and open the software (Click No if requested update).

2 Create a new schematic, or open an existing schematic file.

3 Add/Modify/Delete electrical components into the schematic.

4 Add components from the library, and place them at proper positions.

5 Add wires to connect all components; fulfill the circuit/schematic.

6 Right click on the components, and change their values.

7 After this, click on the simulate icon and open the Edit Simulation Command. In that go for 'DC op pnt' and click OK. Click on run to see voltage and current values. If all are positive values then our circuit is correct otherwise you need to change the direction of resistors until you get positive values.

8 After this, click on the simulate icon and open the Edit Simulation Command. In that go for the transient icon, enter stop time as 1 and click OK. After that click on run option and you can see a new draft along with the circuit. You can see intervals of time. Here we are going to plot the voltage and current waveforms.

9 To find out the voltage values you need to click on the circuit by using a pointer. You will see a waveform plot on the graph, that is the voltage at node 1. To plot other voltage at different graphs, right click on the graph and click on Add plot pane.

10 To plot voltage across a resistor, place the pointer on the wire and drag to the other side of the resistor. Using this, find the voltage at all nodes.

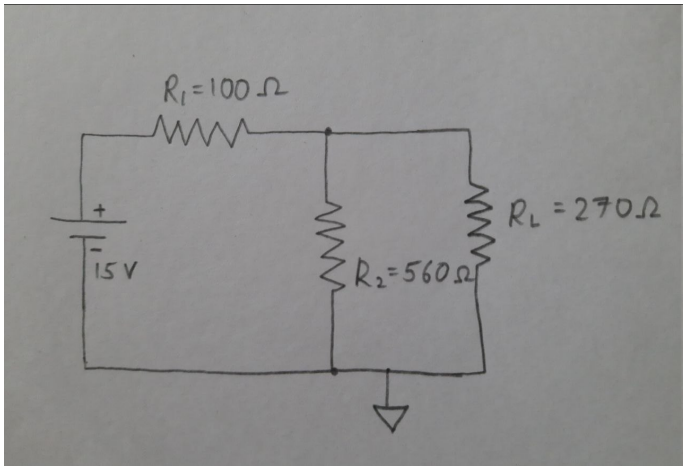
11 To plot current supplied by the battery, add plot pane and click on the resistor to get current passing through each resistor.

12 Color of the trace panel can be adjusted in "Tools", "Color Prefer", "WaveForm" tab in the pulldown menu.

13 To label the circuit/schematic, use the text icon and add the label.

THIS IS THE PROCEDURE TO FIND OUT CURRENT AND VOLTAGE VALUES OF A CIRCUIT BY USING LTSPICE SOFTWARE.

Circuit diagram:



Theoretical calculations:

Resistances R_2 & R_3 are in parallel

Equivalent resistance $= \frac{R_2 \times R_3}{R_2 + R_3} = \frac{560 \times 270}{560 + 270} = 182.16\,\Omega$

$R_{eq.1} = 182.16\,\Omega$

R_{eq} & R_1 are in series

Total Equivalent Resistance $= R_{eq.} = R_{eq.1} + R_1$

$= 182.16 + 100$

$= 282.16\,\Omega$

Total Current $= i = \frac{V}{R_{eq}} \Rightarrow i = \frac{15}{282.16} = 0.0531\ A$

$I = 53.1\ mA$

Potential through $100\,\Omega = V_{100\Omega} = 53.1\ mA \times 100$

$= 5.3\ V$

Assume current through R_2 be I_2,

$I_2 = \frac{I \times R_1}{R_1 + R_2}$

$= \frac{53.1\ mA \times 270}{560 + 270} = \frac{14,337}{830}$

$I_2 = 17.27\ mA$

$I_3 = I - I_2$

$= 53.1 - 17.27$

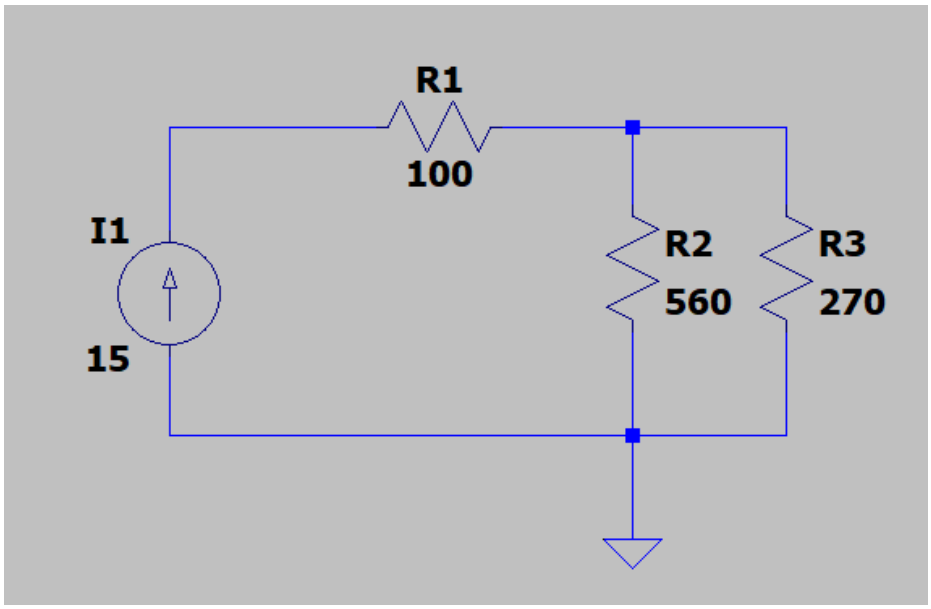
$I_3 = 35.83\ mA$

$V_2 = V_{560\,\Omega} = V_{270} = 15 - V_{100\Omega}$

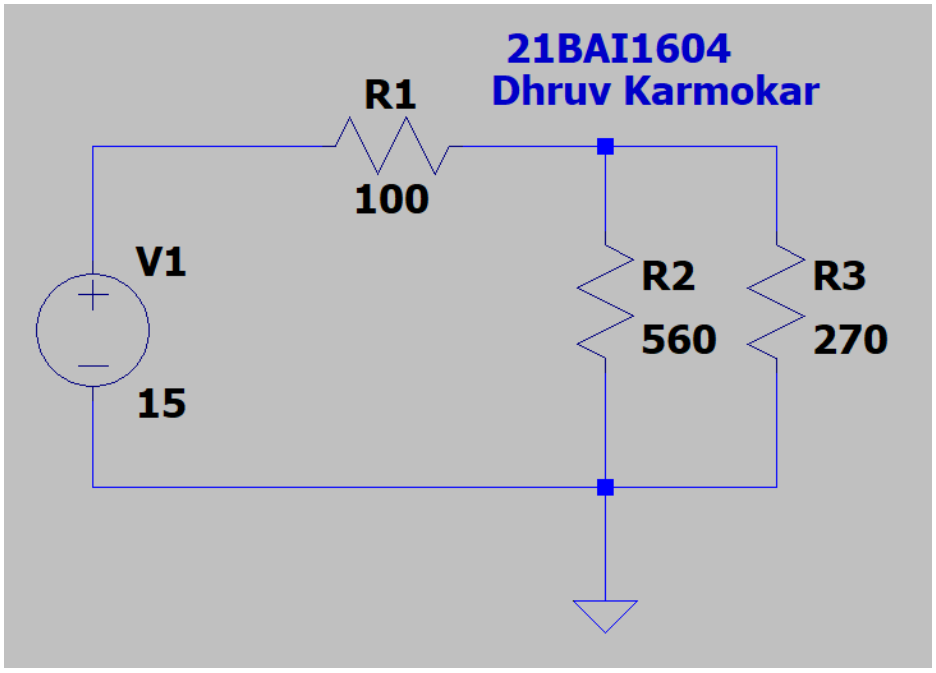
$= 15 - 5.3$

Voltage drop across R_3 & R_2 $= 9.7\ V$

LTSpice schematic diagram for KCL:

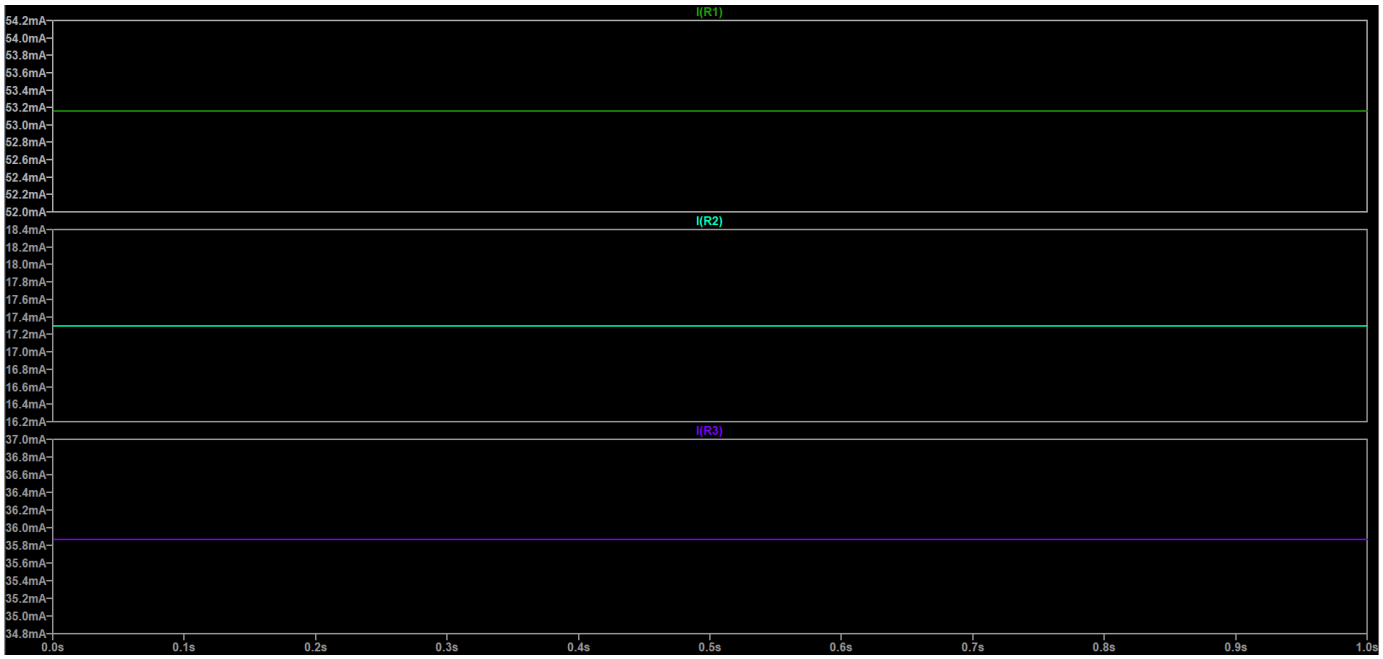


LTSpice schematic diagram for KVL:

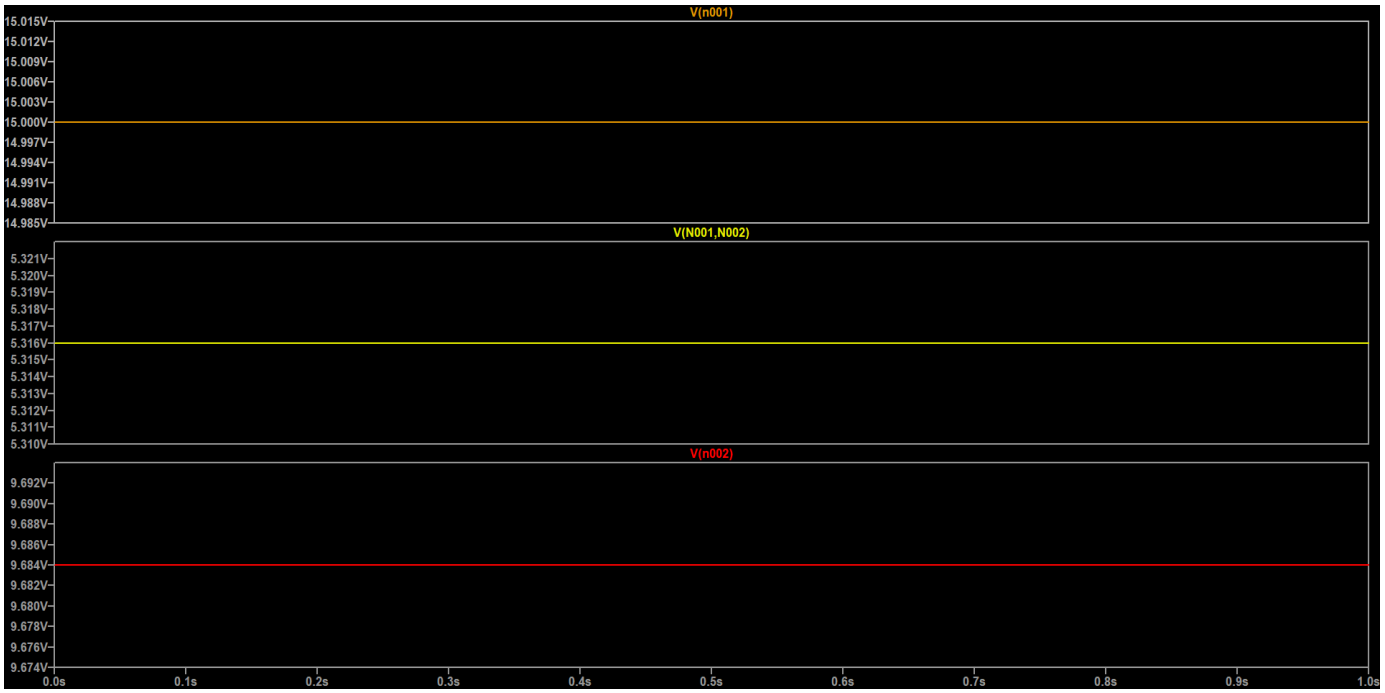


Results:

KCL-

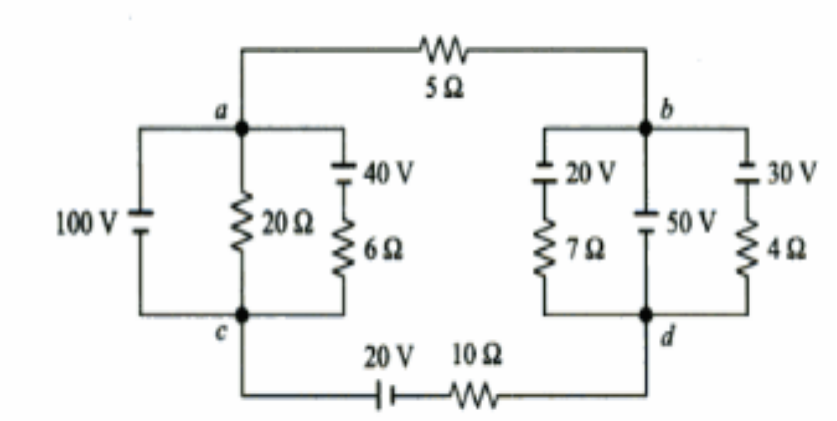


KVL-



Viva Question:

Find the current in all branches for the circuit given below with the help of LTSpice simulation.



Answer: