



Ahsanullah University of Science and Technology

Department of Electrical and Electronic Engineering

**Laboratory Manual
For
Electrical and Electronic Sessional Course**

**Student Name:
Student ID. :**

Course No. : EEE 1102

Course Title : Electrical Circuits-I Lab

**For the students of the
Department of Electrical and Electronic Engineering
1st Year, 1st Semester**

Introduction to Basic Electrical Tools

Objective:

The objective of the experiment is to learn about the commonly used equipment used in the lab and how to properly use it.

Breadboard

Breadboard is a board used for electrical circuit prototyping. Before the circuit is permanently placed in a PCB (Printed Circuit Board) prototyping boards are used to verify the electrical circuit. Some other prototyping boards are: Perfboard, Stripboard, Veroboard etc.

Top two row (A and B) of the board are internally connected sideways and the holes in *group B* are connected vertically as show in the figure 1 and this sequence continue in the rest of the board.

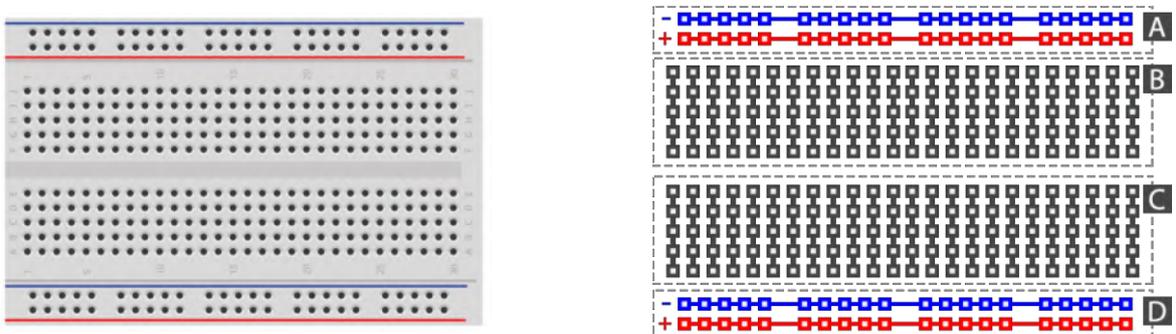


Figure 1.1: External and Internal construction of the breadboard

From circuit to Breadboard:

Closely observe the circuit diagram (electronic schematic) and the equivalent connection on breadboard.

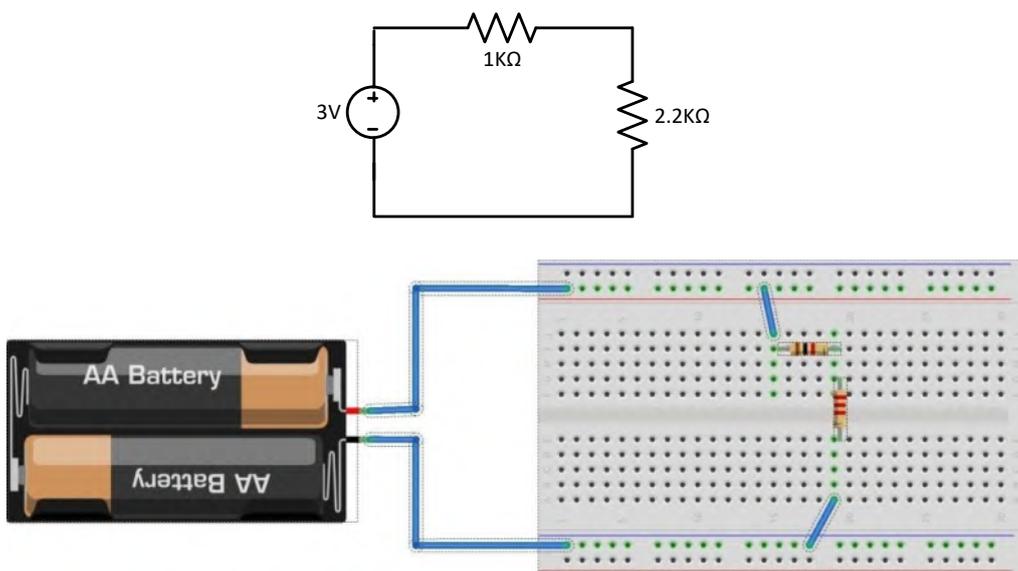
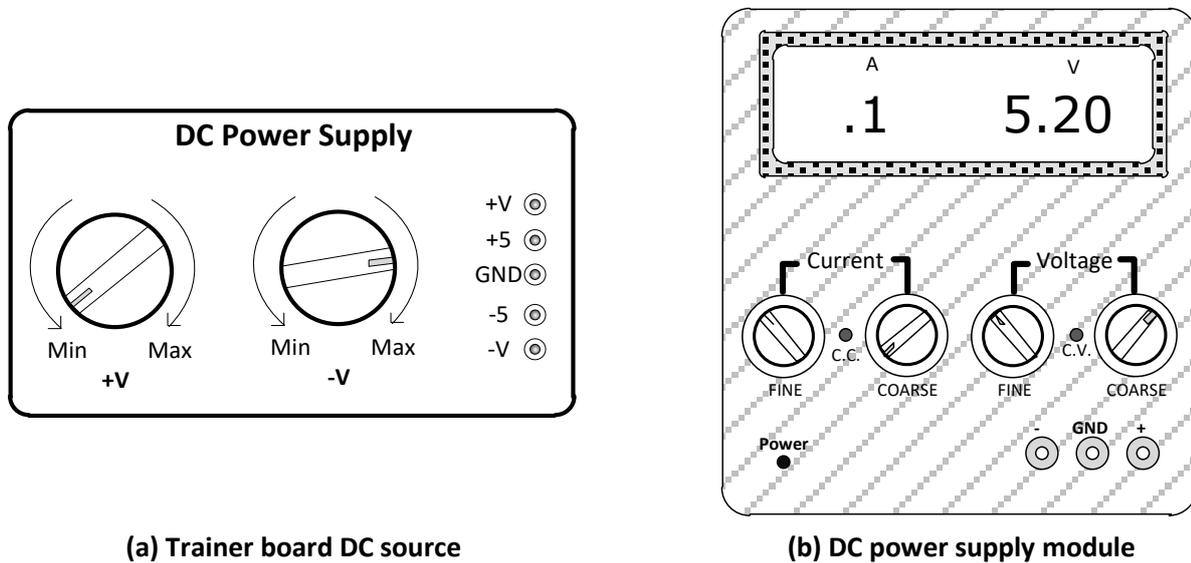


Figure 1.2: Circuit diagram and the equivalent breadboard connection

DC Power Supply:

In the Lab we have two DC source in the workbench. One can be found in the trainer board Fig 1.3(a) and another is a individual DC power supply module Fig 1.3(b).



(a) Trainer board DC source

(b) DC power supply module

Figure 1.3: DC power sources

Trainer board DC source:

Trainer board DC power supply can deliver two variable and two fixed DC voltage at the same time. The variable voltage can be adjusted using the two dial.

Voltage output at pin (reference to GND)	Value	Type
+V	1.2 V to 20 V	Variable
+5	5 V	Fixed
-5	-5 V	Fixed
-V	-1.2 V to -20V	Variable

DC power supply module:

The DC power supply module can deliver voltage ranging from 0V to 30V.

1. To set a particular voltage turn on the power supply.
2. To change the voltage in big step use the coarse dial and to precisely vary the voltage use the fine dial in the voltage group.
3. Observe the output voltage change.

The coarse and fine dial in current group is used to set the maximum current limit at the output.

Measuring Voltage:

1. To measure the voltage across the $1\text{K}\Omega$ resistor circuit in Fig 1.4 (a), construct the circuit as shown in Fig 1.4 (b).
2. Rotate the multimeter dial in the **V** position.
3. Connect the red and black multimeter lead as shown in the Fig 1.4(a) (**parallel to the resistor**)
4. Multimeter should display the Voltage

Warning: while measuring voltage multimeter dial SHOULD NOT be in mA (current) position. It might destroy/damage the meter

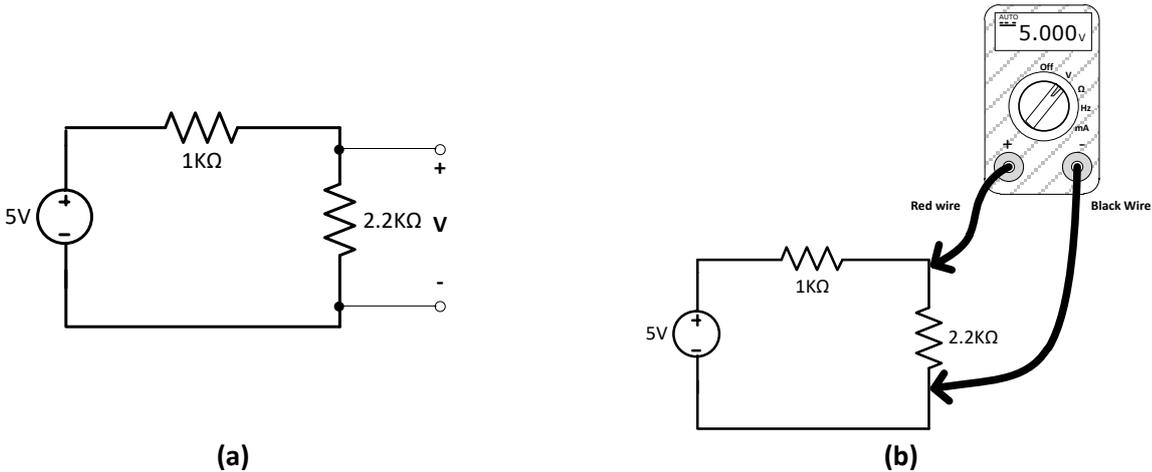


Figure 1.4: Circuit diagram and actual connection for measuring voltage

Measuring Current:

1. To measure the current in the series circuit in Fig 1.5(a) construct the circuit and then **create a break** the circuit as shown in Fig 1.5 (b)
2. Rotate the multimeter dial in the mA position.
3. Connect the red and black multimeter lead as shown in the Fig 1.5(b) (**in series with the circuit**)
4. Multimeter should display the current

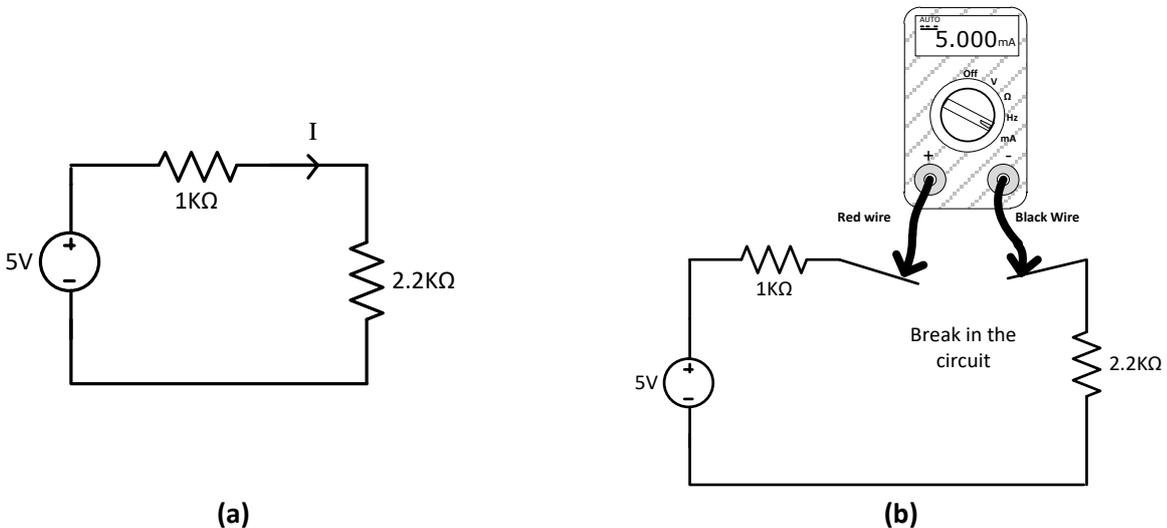


Figure 1.5: Circuit diagram and actual connection for measuring current

Experiment No.	: 01
Name of the Experiment	: Uses of Different Types of Switches.

OBJECTIVE:

Different types of switches are used in electrical circuits. Each type of switch has a particular feature and its uses obviously depend on its inherent property. Although various types of switches may be involved in a particular application, we, however, concentrate our interest into the following types of switches:

- a. Single pole single throw (SPST)
- b. Single pole double throw (SPDT)
- c. Double pole single throw (DPST)
- d. Double pole double throw (DPDT)

SCHEMATIC DIAGRAM:

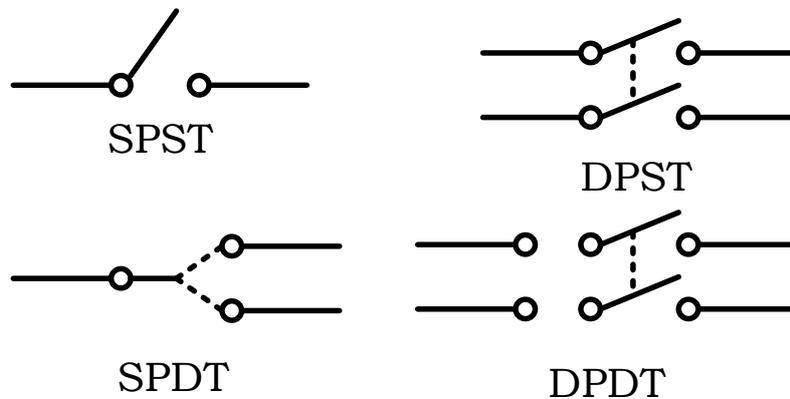


Figure 1.1: Different types of switches

EQUIPMENTS:

- Power cord-1 piece
- SPST switch-2 pieces
- SPDT switch-2 pieces
- Lamps: 60W-1piece
100W-1piece

PROCEDURES:

1. Connect a bulb so that it can be operated from the source by an SPST switch.
2. Connect a bulb so that it may be switched on by either of two SPST switches.
3. Connect two bulbs (one 60W and one 100W) so that either may be operated from a common source by its own switch.
4. Connect two bulbs (one 60W and one 100W) so that both may be operated simultaneously from a common source by one SPST switch.
5. Connect a bulb so that it may be operated independently by either of two SPDT switches from a source.

ASSIGNMENTS:

1. For each of the cases given below, mention whether the switches of procedure 2 are dependent on each other to turn the lamp ON:
 - a. When the switches are connected in series.
 - b. When the switches are connected in parallel.
2. Which method of procedure 4 is preferable and why?
3. Explain the variations of the brightness level of the two lamps in procedure 4.
4. What are the applications of the arrangement of procedure 5?
5. What is the drawback of the switch connection of the circuit in Figure 1.2?

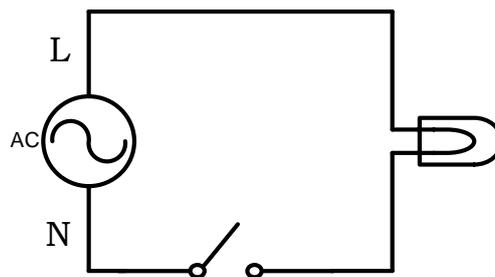


Figure 1.2: Circuit diagram for assignment no.-5

AHSANULLAH UNIVERSITY OF SCIENCE & TECHNOLOGY
DEPARTMENT OF ELECTRICAL & ELECTRONIC ENGINEERING

EEE -1 1 O2: ELECTRICAL CIRCUITS – I LAB.

Experiment No. : 02
Name of the Experiment : Verification of Ohm's Law.

OBJECTIVE:

To verify the following two equivalent forms of Ohm's Law:

- a. Express I as a function of V and R.
- b. Express V as a function of I and R.

THEORY:

Ohm's law describes mathematically how voltage 'V', current 'I' and resistance 'R' in a circuit are related. According to this law:

“The current in a circuit is directly proportional to the applied voltage and inversely proportional to the circuit resistance”.

Formula for voltage:

For a constant value of R, V is directly proportional to I
i.e. $V = IR$

Formula for current:

For a constant value of V, I is inversely proportional to R
i.e. $I = V/R$

EQUIPMENTS:

- Variable DC power supply -1 piece
- Digital multimeter (DMM)/ Analog multimeter-1 piece.
- Resistances: 1 K Ω , 2.2 K Ω , 3.3 K Ω , 4.7 K Ω , 5.6 K Ω , 10 K Ω -1 piece each.
- Trainer Board
- Connecting Wires.

CIRCUIT DIAGRAM:

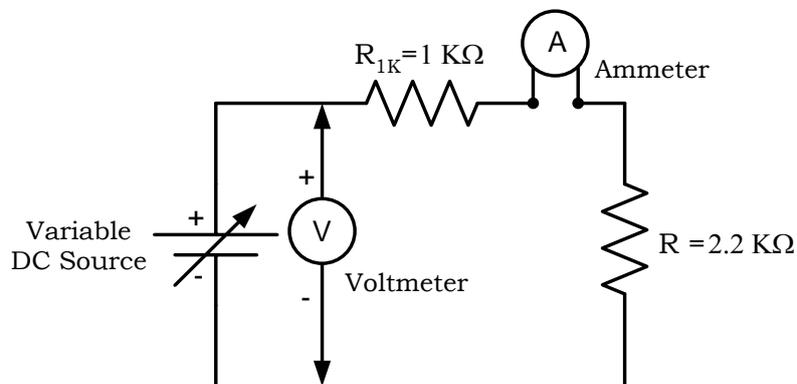


Figure 2.1: Verification of Ohm's Law

OHM'S LAW

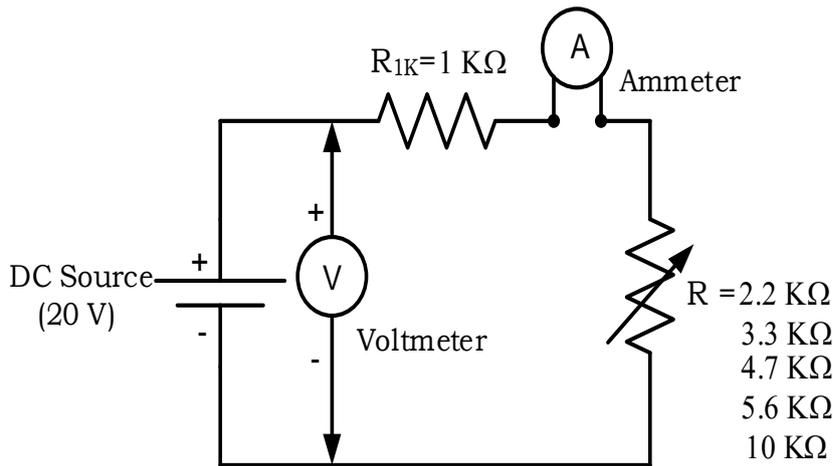


Figure 2.2: Verification of Ohm's Law

PROCEDURES:

Current versus voltage:

- Construct the circuit of Figure 2.1. Do not switch on the power supply.
- Turn on the power supply and adjust it to 5V by using Voltmeter. Measure the current I by ammeter and record it in the Table 2.2.
- Increase the values of voltage as shown in the Table 2.2. Measure the current I in turn and record the values in Table 2.2.
- Calculate the values of current I by using $I=V/R_T$. Use measured values of resistances.

Current versus resistance:

- Construct the circuit of Figure 2.2. Do not switch on the power supply.
- Turn on the power supply and adjust it to 20V by using Voltmeter. Measure the current I by ammeter for $R=2.2\text{ K}\Omega$ (Use measured values) and record it in the Table 2.3.
- Turn off the power supply and remove the resistance 2.2 KΩ. Replace it by resistor 3.3 KΩ.
- Now turn on the power supply. Measure and record the current I in turn, at each of the resistance settings shown in the Figure 2.2.
- Calculate the values of resistance R_T by using $R_T=V/I$. Use measured values of voltage and current.

DATA SHEET:

Table 2.1: Measuring Resistances by using Ohmmeter

Nominal values of R (kΩ)	Measured values of R by using Ohmmeter (kΩ)
1	
2.2	
3.3	
4.7	
5.6	
10	

Table 2.2: Current versus voltage

Supply Voltage (V)	Measured I by using Ammeter (mA)	$R_T = R_{1K} + R_{2.2K}$ [Use measured values of R] (kΩ)	Calculate I (mA) $I = V/R_T$
5			
10			
15			
20			
25			

Table 2.3: Current versus resistance

Supply Voltage (V)	Measured I by using Ammeter (mA)	R_T (kΩ) Use measured values of R	Calculate $R_T = V/I$ (kΩ)
20		$R_T = R_{1K} + R_{2.2K}$	
		$R_T =$	
20		$R_T = R_{1K} + R_{3.3K}$	
		$R_T =$	
20		$R_T = R_{1K} + R_{4.7K}$	
		$R_T =$	
20		$R_T = R_{1K} + R_{5.6K}$	
		$R_T =$	
20		$R_T = R_{1K} + R_{10K}$	
		$R_T =$	

Signature of the Teacher

ASSIGNMENTS:

1. What can you say about the relationship between voltage and current, provided that the resistance is fixed?
2. Plot a graph of I versus V keeping the value of resistance constant. Use measured values of I and V . Comment on the graph briefly.
3. Plot a graph of I versus R_T keeping the value of supply voltage constant. Use measured values of I and R_T . Comment on the graph briefly.

AHSANULLAH UNIVERSITY OF SCIENCE & TECHNOLOGY
DEPARTMENT OF ELECTRICAL & ELECTRONIC ENGINEERING

EEE -1 1 O2: ELECTRICAL CIRCUITS – I LAB.

Experiment No. : 03

Name of the Experiment: **To investigate the characteristics of a series DC circuit and to verify Kirchoff's Voltage Law (KVL).**

OBJECTIVE:

The objective of this experiment is to investigate the characteristics of a series DC circuit and to verify Kirchoff's Voltage Law (KVL).

THEORY:

In a series circuit (Figure 3.1) the current is same through all of the circuit elements.

The equivalent Resistance, $R_T = R_1 + R_2 + R_3$.

By Ohm's Law, the Current is

$$I = \frac{V_{Supply}}{R_T}$$

KVL states that the voltage rises must be equal to the voltage drops around a close circuit. Applying Kirchoff's Voltage Law around closed loop of Figure 3.1, we find,

$$V_{Supply} = V_1 + V_2 + V_3$$

Where, $V_1 = IR_1$, $V_2 = IR_2$, $V_3 = IR_3$

Current I is same throughout the circuit for figure 3.1.

The voltage divider rule states that the voltage across an element or across a series combination of elements in a series circuit is equal to the resistance of the element divided by total resistance of the series circuit and multiplied by the total impressed voltage. For the elements of Figure 3.1

$$V_1 = \frac{R_1 E}{R_T}, \quad V_2 = \frac{R_2 E}{R_T}, \quad V_3 = \frac{R_3 E}{R_T}$$

EQUIPMENTS:

- Variable DC power supply -1 piece
- Digital Multimeter (DMM)/ Analog multimeter-1 piece.
- Resistances: 1k Ω , 2.2k Ω , 4.7k Ω -1 piece each.
- Trainer Board-1 piece
- Connecting Wires.

SERIES DC CIRCUIT & KVL

CIRCUIT DIAGRAM:

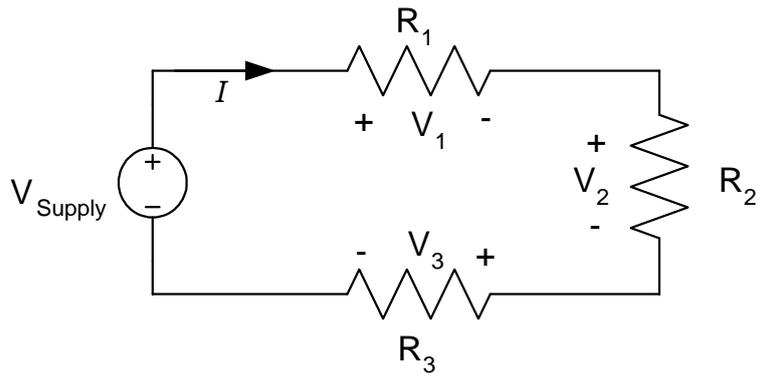


Figure 3.1

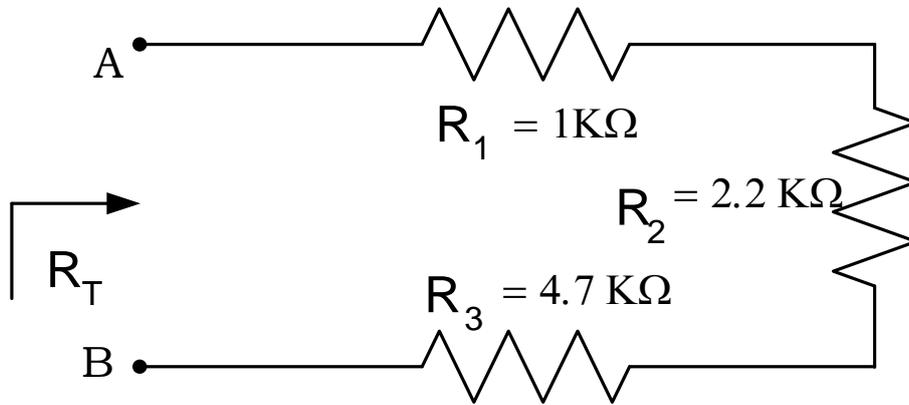


Figure 3.2

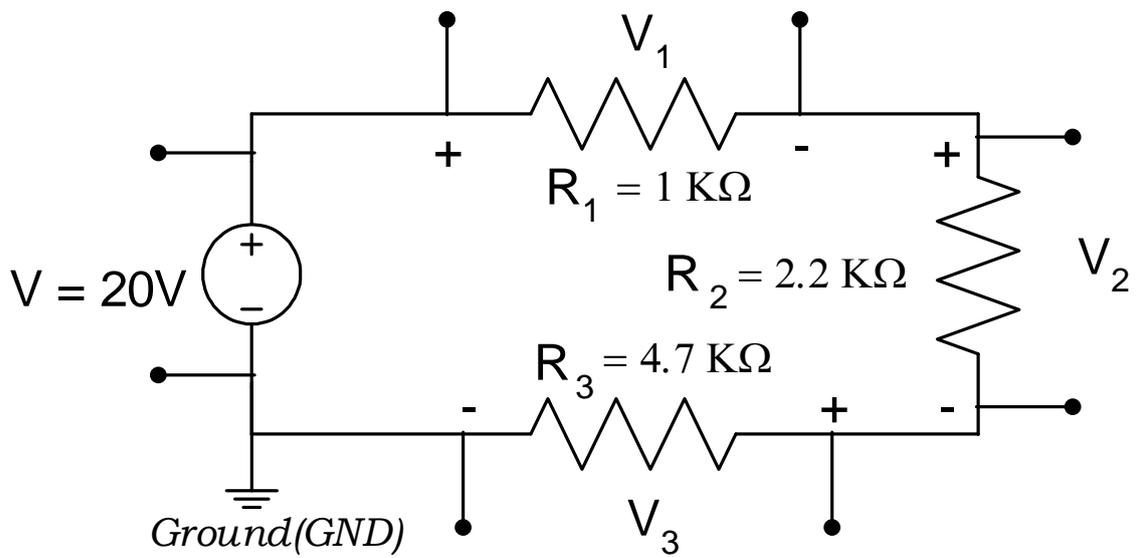


Figure 3.3

SERIES DC CIRCUIT & KVL

PROCEDURE:

1. Measure the resistances having values 100Ω , 220Ω & 470Ω by using Ohmmeter and record the values in Table 3.1.
2. Construct the circuit as shown in Fig 3.2.
3. Then measure input resistance R_T across points A-B using Ohmmeter and record that value in Table 3.1.
4. Now construct the circuit as shown in Fig 3.3. Turn on the DC power supply and set the DC supply to 20V by using Voltmeter.
5. Measure voltage across each resistor with Voltmeter and record in the Table 3.1
6. Calculate V_1 , V_2 and V_3 using Voltage Divider Rule (VDR). **[Use measured values of resistances for all calculations.]**

ASSIGNMENTS:

1. What can you deduce about the characteristics of a series circuit from observation Table 3.1?
2. From the data found in Table 3.1, mathematically prove that the current in the series network of figure 3.3 is equal for each resistance.
3. Verify KVL from the data obtained in Table 3.1.

DATA SHEET:

Table 3.1:

Nominal values of Resistance (k Ω)	Measured values of Resistance by Ohmmeter (k Ω)	Equivalent Resistance, R _T		Measured voltage across each resistor (V)	Calculated Voltage using VDR (V)
		Measured R _T by using Ohmmeter (k Ω)	Calculated R _T = R ₁ + R ₂ + R ₃ (k Ω)		
R ₁ =1.0				V ₁ =	
R ₂ =2.2				V ₂ =	
R ₃ =4.7				V ₃ =	

Calculation:

 Signature of the Teacher

AHSANULLAH UNIVERSITY OF SCIENCE & TECHNOLOGY
DEPARTMENT OF ELECTRICAL & ELECTRONIC ENGINEERING

EEE -1102: ELECTRICAL CIRCUITS – I LAB.

Experiment No. : 04

Name of the Experiment: **To investigate the characteristics of a Parallel DC circuit and to verify Kirchoff's Current Law (KCL).**

OBJECTIVE:

The objective of this experiment is to investigate the characteristics of a parallel DC circuit and to verify Kirchoff's Current Law (KCL).

THEORY:

In a parallel circuit (Figure 4.1) the voltage across parallel elements is the same.

The total or equivalent resistance (R_T) is given by,

$$\frac{1}{R_T} = \frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3} + \dots + \frac{1}{R_N}$$

If there are only two resistors in parallel, it is more convenient to use,

$$R_T = \frac{R_1 R_2}{R_1 + R_2}$$

In any case, the total resistance will always be less than the resistance of the smallest resistor of the parallel network.

KCL states that the currents entering a node must be equal to the currents leaving that node. For the network of Figure 4.1 the currents are related by the following expression:

$$I_T = I_1 + I_2 + I_3 + \dots + I_N$$

Applying current divider rule (CDR) for a circuit of only two resistors in parallel as shown in figure 4.2,

$$I_1 = \frac{R_2 I_T}{R_1 + R_2} \quad \text{and} \quad I_2 = \frac{R_1 I_T}{R_1 + R_2}$$

For equal parallel resistors, the current divides equally and the total resistance is the value of one divided by the 'N' number of equal parallel resistors, i.e.:

$$R_T = \frac{R}{N}$$

For a parallel combination of N resistors, the current I_1 through R_1 is:

$$I_1 = I_T \times \frac{\frac{1}{R_1}}{\frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3} + \dots + \frac{1}{R_N}} = (I_T * R_T) / R_1$$

EQUIPMENTS:

- Variable DC power supply -1 piece
- Digital Multimeter (DMM)/ Analog multimeter-1 piece.
- Resistances: 1K Ω , 2.2 K Ω , 4.7 K Ω -1 piece each.
- Trainer Board-1 piece
- Connecting Wires.

CIRCUIT DIAGRAM:

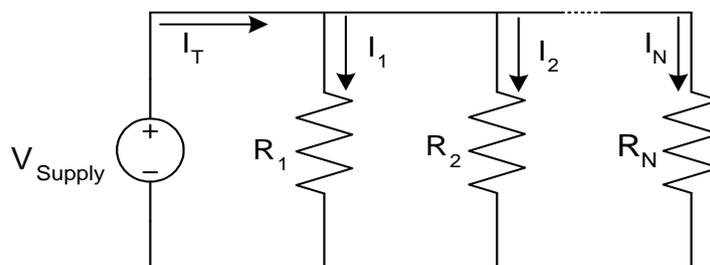


Figure 4.1

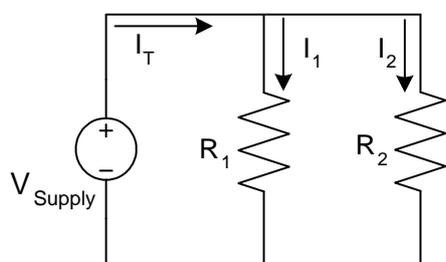


Figure 4.2

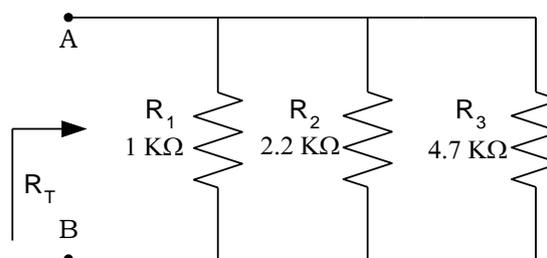


Figure 4.3

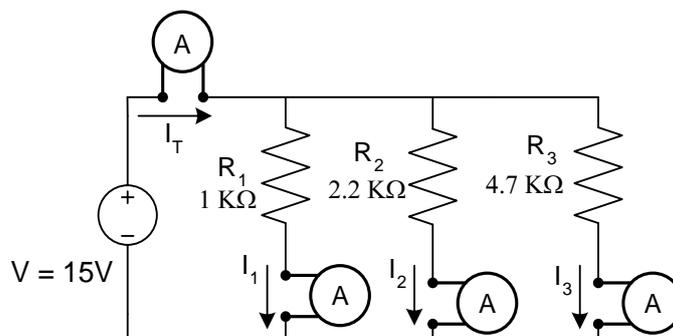


Figure 4.4

PROCEDURE:

1. Measure the resistances having values 1 K Ω , 2.2 K Ω & 4.7 K Ω by using Ohmmeter and record the values in Table 4.1.
2. Construct the circuit as shown in Fig 4.3.
3. Then measure input resistance R_T across points A-B using Ohmmeter and record that value in Table 4.1.
4. Now construct the circuit as shown in Fig 4.4. Turn on the DC power supply and set the DC supply to 15V by using Voltmeter.
5. Measure the currents I_T , I_1 , I_2 and I_3 by using Ammeter and record in the Table 4.1.
6. Calculate I_1 , I_2 and I_3 using Current Divider Rule (CDR). *[Use measured values of resistances for all calculations.]*

ASSIGNMENTS:

1. What can you deduce about the characteristics of a parallel circuit from observation Table 4.1?
2. From the data found in Table 4.1, Calculate I_1 , I_2 and I_3 using Ohm's Law.
3. Verify KCL from the data obtained in Table 4.1.

AHSANULLAH UNIVERSITY OF SCIENCE & TECHNOLOGY
DEPARTMENT OF ELECTRICAL & ELECTRONIC ENGINEERING

EEE -1 1 02: ELECTRICAL CIRCUITS – I LAB.

PARALLEL DC CIRCUIT & KCL

DATA SHEET:

Table 4.1:

Nominal values of Resistance (KΩ)	Measured values of Resistance by Ohmmeter (KΩ)	Equivalent Resistance, R _T		Measured current through each resistor (mA)	Calculated Current using CDR (mA)
		Measured R _T by using Ohmmeter (KΩ)	Calculated $\frac{1}{R_T} = \frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3}$ (KΩ)		
R ₁ =1				I ₁ =	
R ₂ =2.2				I ₂ =	
R ₃ =4.7				I ₃ =	

Calculation:

Signature of the Teacher

AHSANULLAH UNIVERSITY OF SCIENCE & TECHNOLOGY
DEPARTMENT OF ELECTRICAL & ELECTRONIC ENGINEERING

EEE -1 1 O2: ELECTRICAL CIRCUITS – I LAB.

Experiment No. : 05

Name of the Experiment : Use of Galvanometer as Ammeter and Voltmeter.

OBJECTIVE:

The objective of this experiment is to show how a galvanometer can be used as an ammeter and a voltmeter.

THEORY:

An ammeter is an instrument, while connected in series with a branch, measure the current of that branch. Ideally it should be of zero resistance, so that there is no voltage drop across it and hence it has no effect on the circuit where it is connected.

A voltmeter is an instrument, while connected in parallel with a branch, measure the voltage of that branch. Ideally it should be of infinite resistance, so that it draws no current and hence it has no effect on the circuit where it is connected.

A galvanometer is an instrument that can detect current. This meter can be used either as an ammeter or as a voltmeter. To use this meter as an ammeter, a very small resistance is connected in parallel with it so that a small voltage drop will occur across it. When the galvanometer is used as a voltmeter, a very high resistance is connected in series with it so that it will draw a little current.

EQUIPMENTS:

- Trainer Board - 1 piece.
- Digital Multi-meter - 1 piece.
- Galvanometer - 1 piece.
- Potentiometer - (10K Ω) 1 piece.
- Resistances: 10 Ω , 100 Ω , 1 K Ω , 10K Ω -1 piece each.

PROCEDURES:

PART A:

1. Construct an ammeter using a galvanometer as shown in Figure 5.1

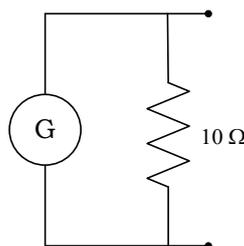


Figure 5.1: Constructed Ammeter circuit

- Now place the constructed ammeter in the circuit shown below in Figure 5.2.

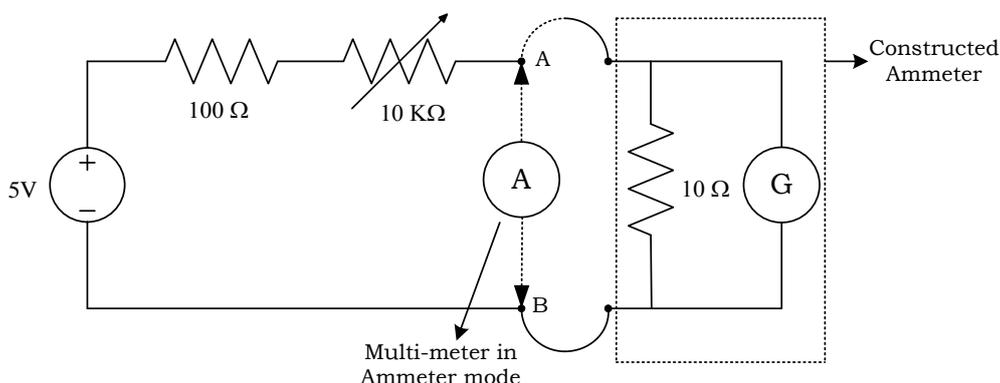


Figure 5.2: Use of galvanometer as an ammeter.

- Connect the constructed ammeter and vary the pot until full-scale deflection is obtained. Then disconnect the constructed ammeter and place actual ammeter. Measure the current from multi-meter in ammeter mode.

The value of each division of the galvanometer scale

$$= \frac{\text{Measured current}}{\text{No. of divisions deflected in full scale.}} =$$

- Connect the constructed ammeter and vary the pot. Note the number of divisions deflected by the galvanometer. Then disconnect the constructed ammeter and place the actual ammeter. Measure the current from multi-meter in ammeter mode. Record these readings in Table 5.1.
- Increase the pot several times. Repeat step 4 for each increment of pot and record the readings in Table 5.1.

Table 5.1: Data for Ammeter.

No. of divisions deflected by the galvanometer	Current = No. of divisions deflected X Value of each division. (mA)	Measured Current (Reading from multi-meter in ammeter mode) (mA)

PART B:

1. Construct a voltmeter using a galvanometer as shown in Figure 5.3.

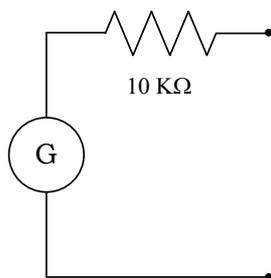


Figure 5.3: Constructed Voltmeter circuit

2. Now place the constructed voltmeter in the circuit shown below in Figure 5.4.

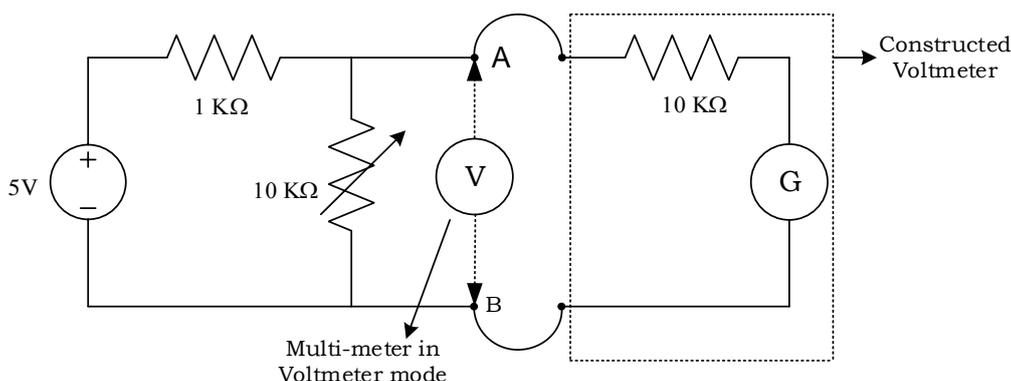


Figure 5.4: Use of galvanometer as voltmeter.

3. Connect the constructed voltmeter and vary the pot until full-scale deflection is obtained. Then place the actual voltmeter Measure the voltage from multi-meter in voltmeter mode.

The value of each division of the galvanometer scale

$$= \frac{\text{Measured Voltage}}{\text{No. of divisions deflected in full scale.}} =$$

4. Connect the constructed voltmeter and vary the pot. Note the number of divisions deflected by the galvanometer. Then place the actual voltmeter Measure the voltage from multi-meter in voltmeter mode. Record these readings in Table 5.2.
5. Increase the pot several times. Repeat step 4 for each increment of pot and record the readings in Table 5.2.

Table 5.2: Data for Voltmeter.

No. of divisions deflected by the galvanometer	Voltage = No. of divisions deflected X Value of each division. (v)	Measured Voltage (Reading from multi-meter in voltmeter mode) (v)

ASSIGNMENTS:

1. Comment on the results found in Part A and Part B.
2. Suppose we want to measure a current greater than the full-scale deflection of the constructed ammeter (Figure 5.1). In this case, should the resistance in parallel with the galvanometer be increased or decreased? Explain in brief.

Experiment no: 06

Experiment name: Introduction to LTspice and verification of superposition theorem using simulation.

In this experiment, we will learn how to analyze electric circuits in LTspice. The theory behind the studied circuits can be found in any standard circuit theory text book. It is a good idea to do some hand calculations for the circuits that are given and compare them with LTspice results.

6.1a Installation of LTspice

Installation of LTspice is quite easy. In order to install LTspice, search for “**ltspice download**” in Google (Fig. 6.1) and open the first result (Fig. 6.2).



Fig. 6.1: Searching for LTspice download page

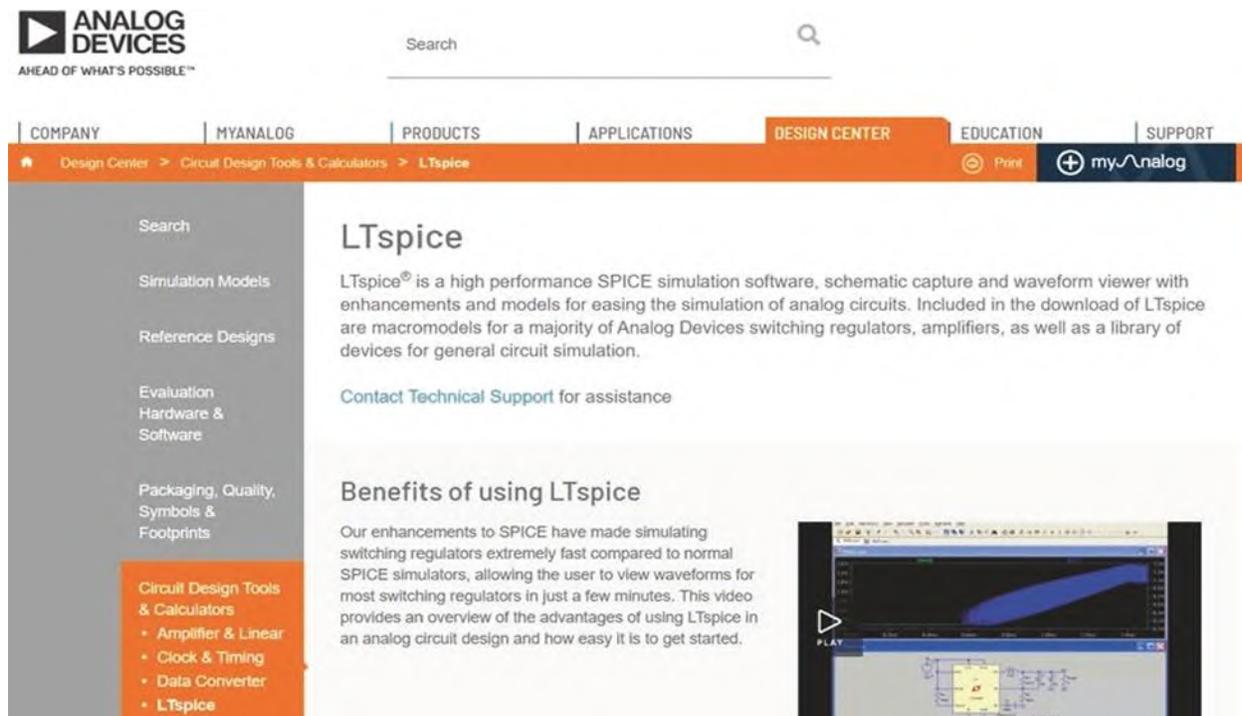


Fig. 6.2: LTspice webpage

Scroll down the screen until you see the **download** link (Fig. 6.3). **Download** the suitable version based on your **operating system**.

Download LTspice

Download our LTspice simulation software for the following operating systems:

- [Download for Windows 7, 8 and 10](#) Updated on Jun 3 2021 *
- [Download for Mac 10.9+](#) Updated on Jun 3 2021 *
- [Download for Windows XP](#) (End of Support)

*date displayed reflects the most recent upload date

LTspice Demo Circuits

Explore ready to run LTspice demonstration circuits with our Demo Circuits Collection. [View More](#)

Documentation

Additional support for LTspice can be found within our documentation, including keyboard shortcuts and a visual guide.

- [LTspice Information Flyer & Shortcuts \(PDF\)](#)
- [Mac OS X Shortcuts \(PDF\)](#)
- [Get Up and Running with LTspice](#)

Fig. 6.3: Download link

After **downloading** the file, **install** it. The LTspice environment is shown in Fig. 6.4.

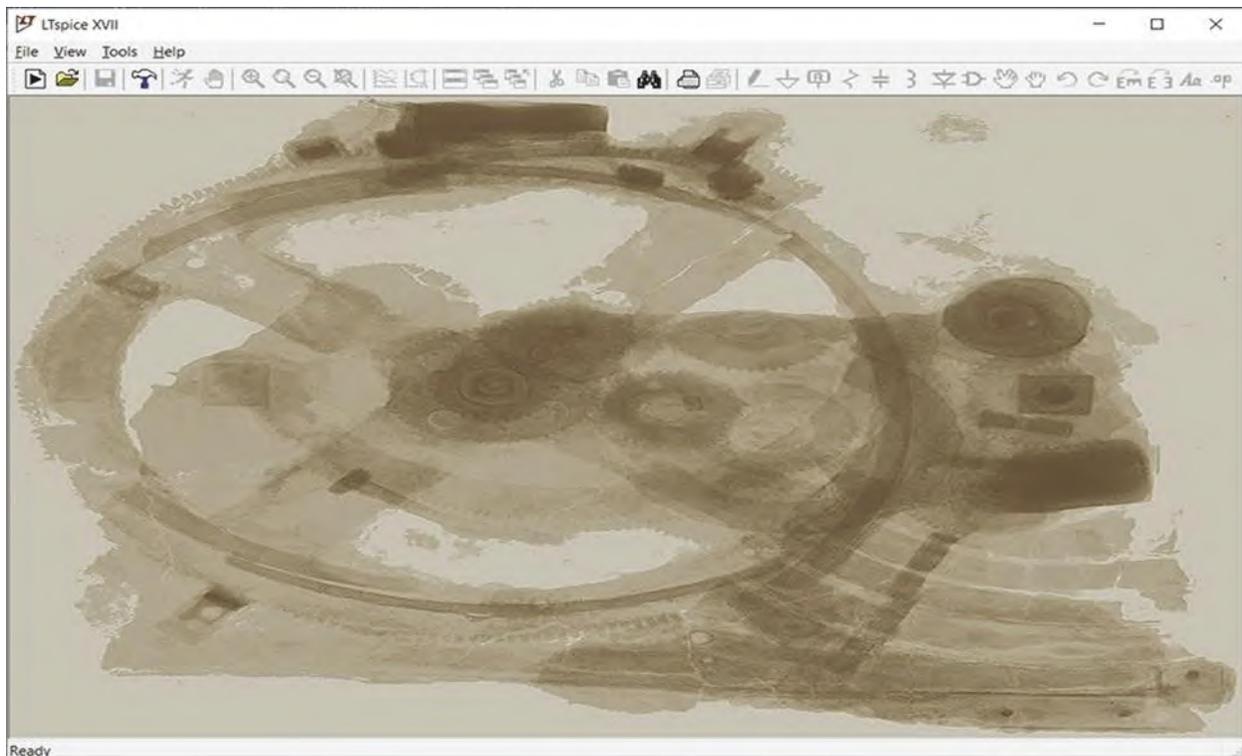


Fig. 6.4: LTspice environment

If you click the **Help**> **Help Topics** (Fig. 6.5), *LTspiceHelp* appears on the screen (Fig. 6.6).

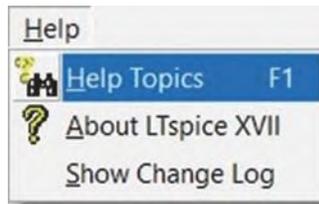


Fig. 6.5: Help menu

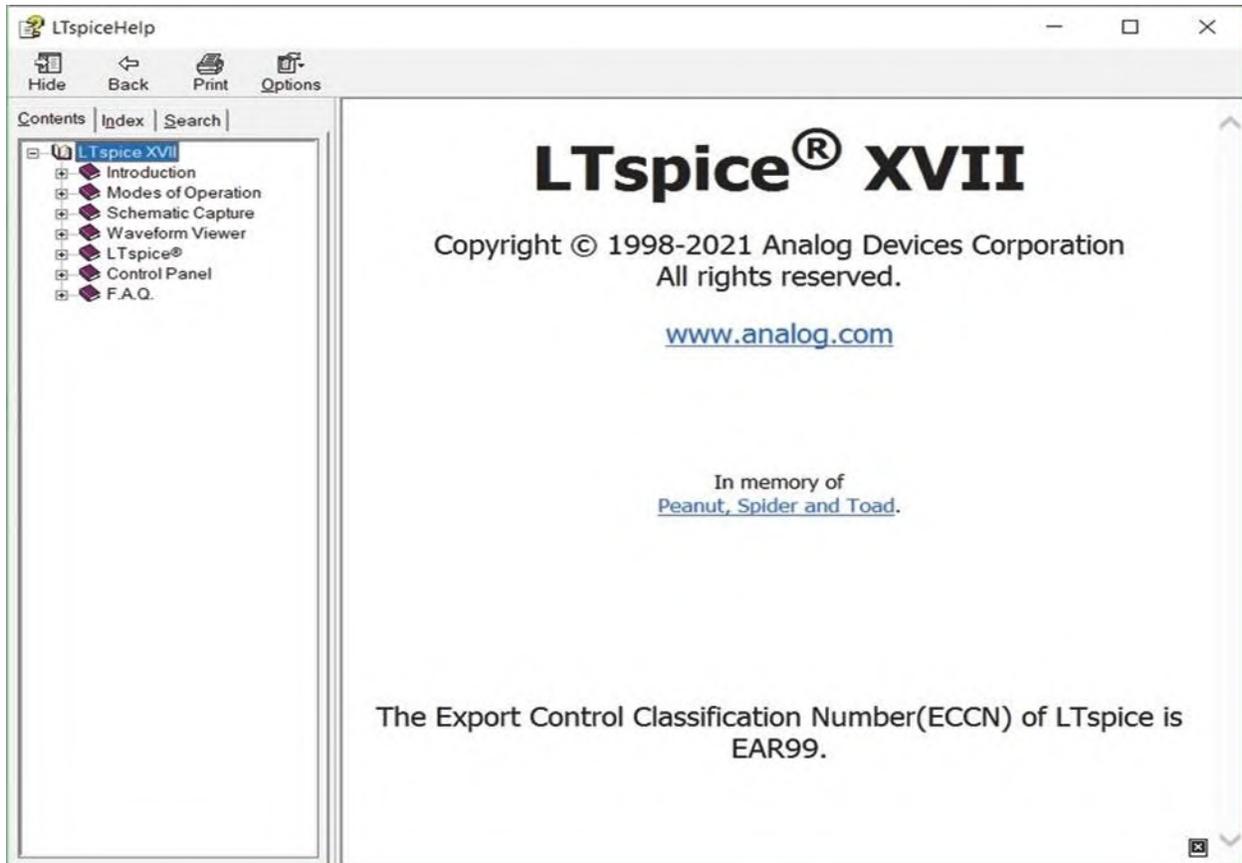


Fig. 6.6: LTspice Help window

The Dot Commands section of the help is a good reference to learn LTspice commands (Fig. 6.7).

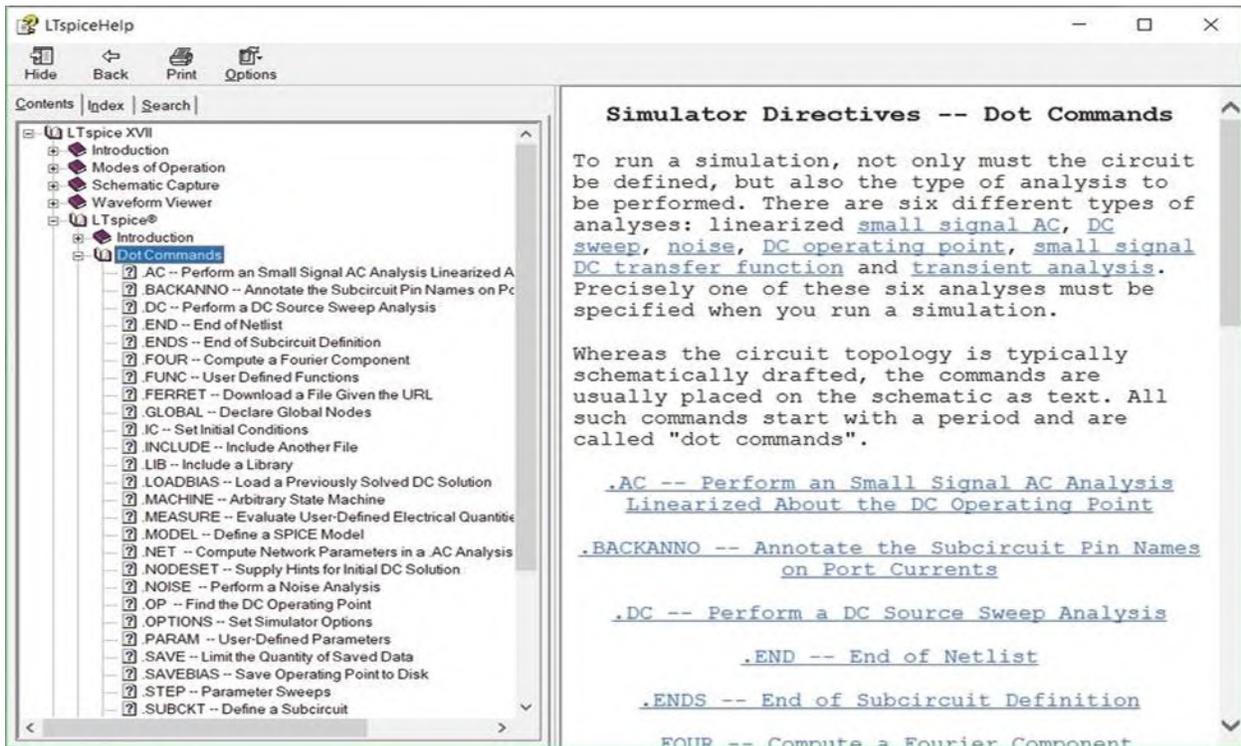


Fig. 6.7: Dot Commands section of LTspice Help

LTspice has an “examples” folder which contains many inspiring simulations. In order to open the “examples” folder, click the open icon (Fig. 6.8) and then go to the path that you installed LTspice (Fig. 6.9).

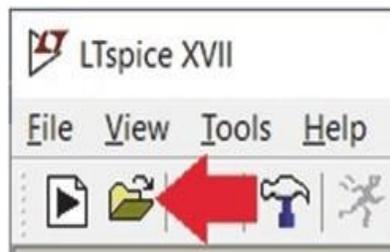


Fig. 6.8: Open icon

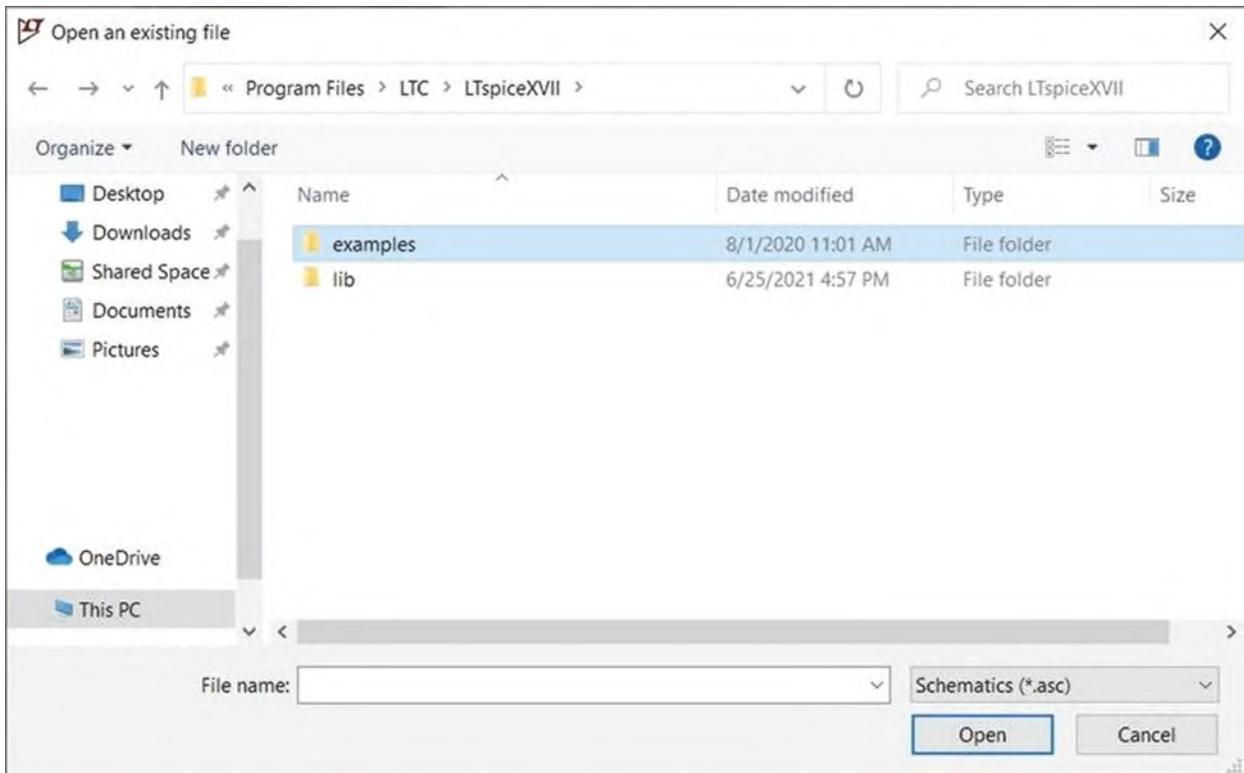


Fig. 6.9: Examples folder

Open the “examples” folder (Fig. 6.10).

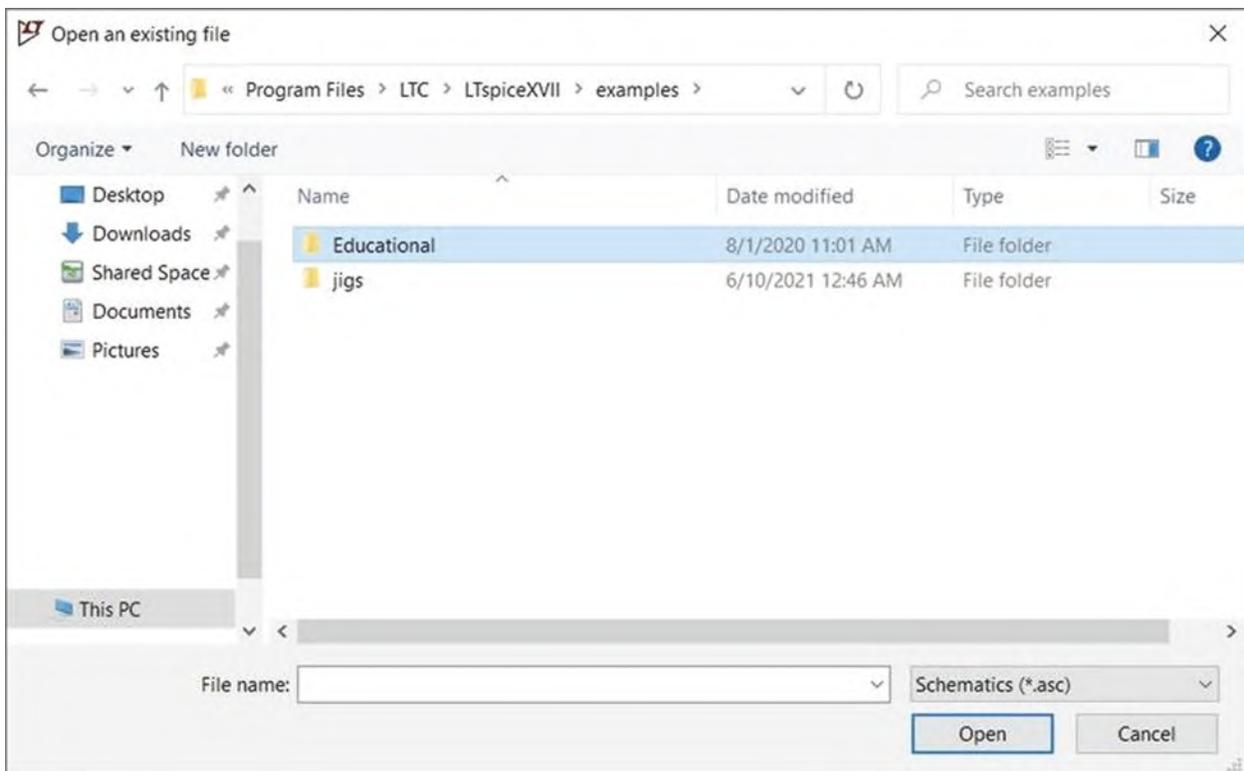


Fig. 6.10: Educational folder

Open the “**Educational**” folder (Fig. 6.11). Now you have access to the ready to use sample simulation files.

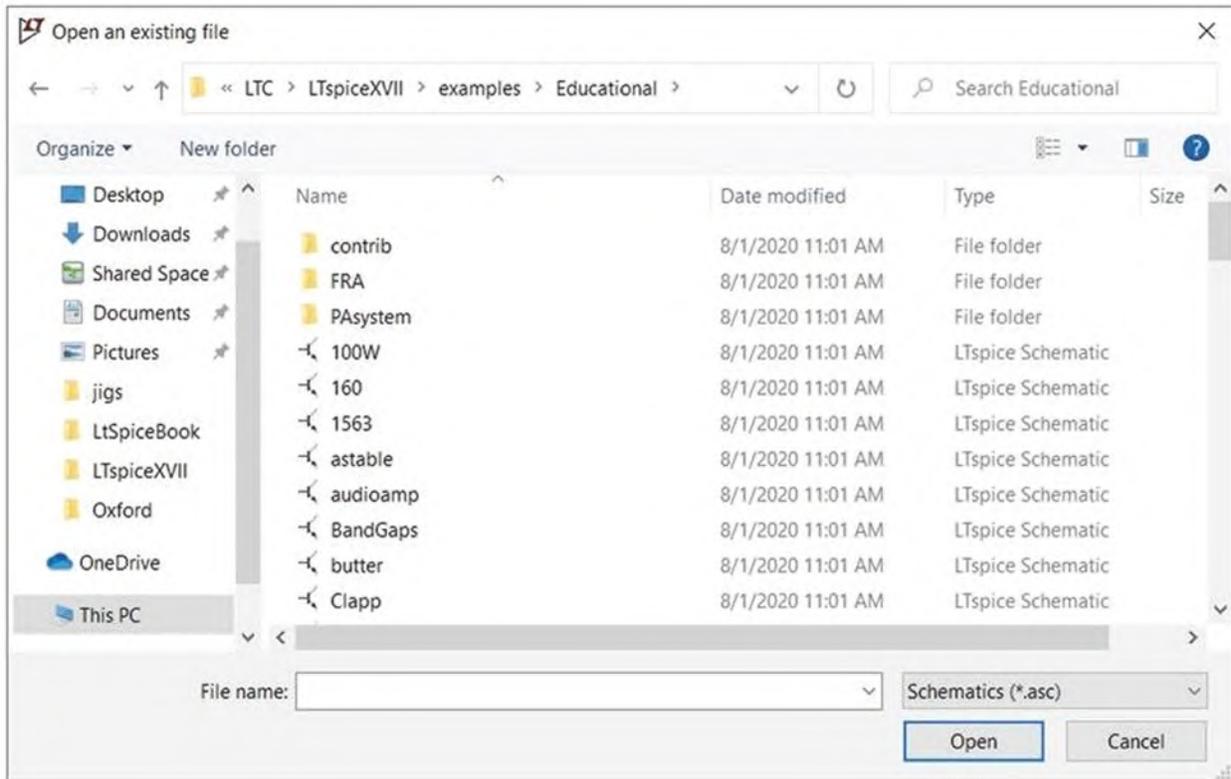


Fig. 6.11: Available sample simulation files

6.1b Example 1: Simple Resistive Voltage Divider

In this example, we want to simulate the behavior of following simple voltage divider circuit (**Fig. 6.12**) of experiment no 02.

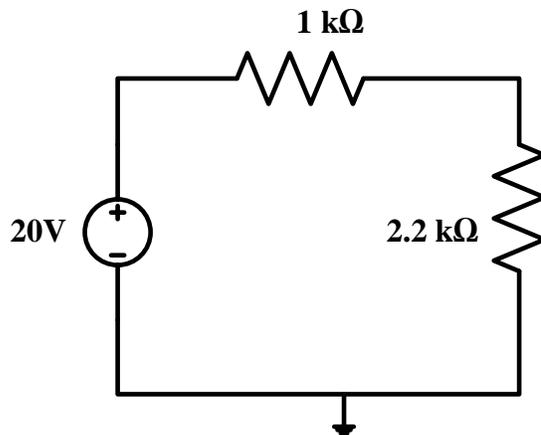


Fig. 6.12 Simple voltage divider circuit

Run the LTspice (Fig 6.13).

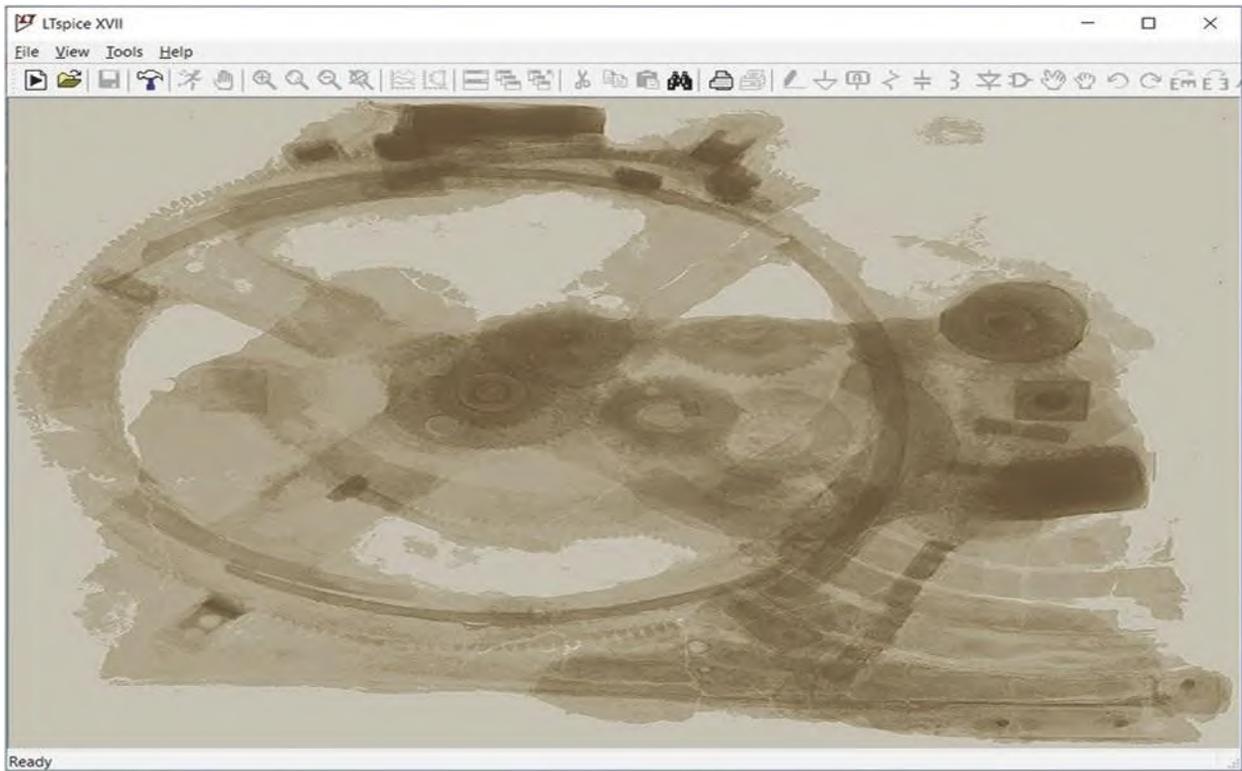


Fig 6.13 LTspice environment

Click the New Schematic icon (Fig 6.14). This opens a new schematic for you (Fig 6.15).

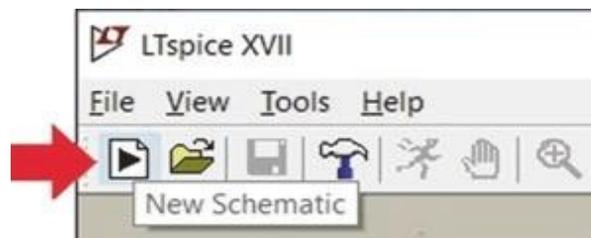


Fig 6.14 New Schematic icon

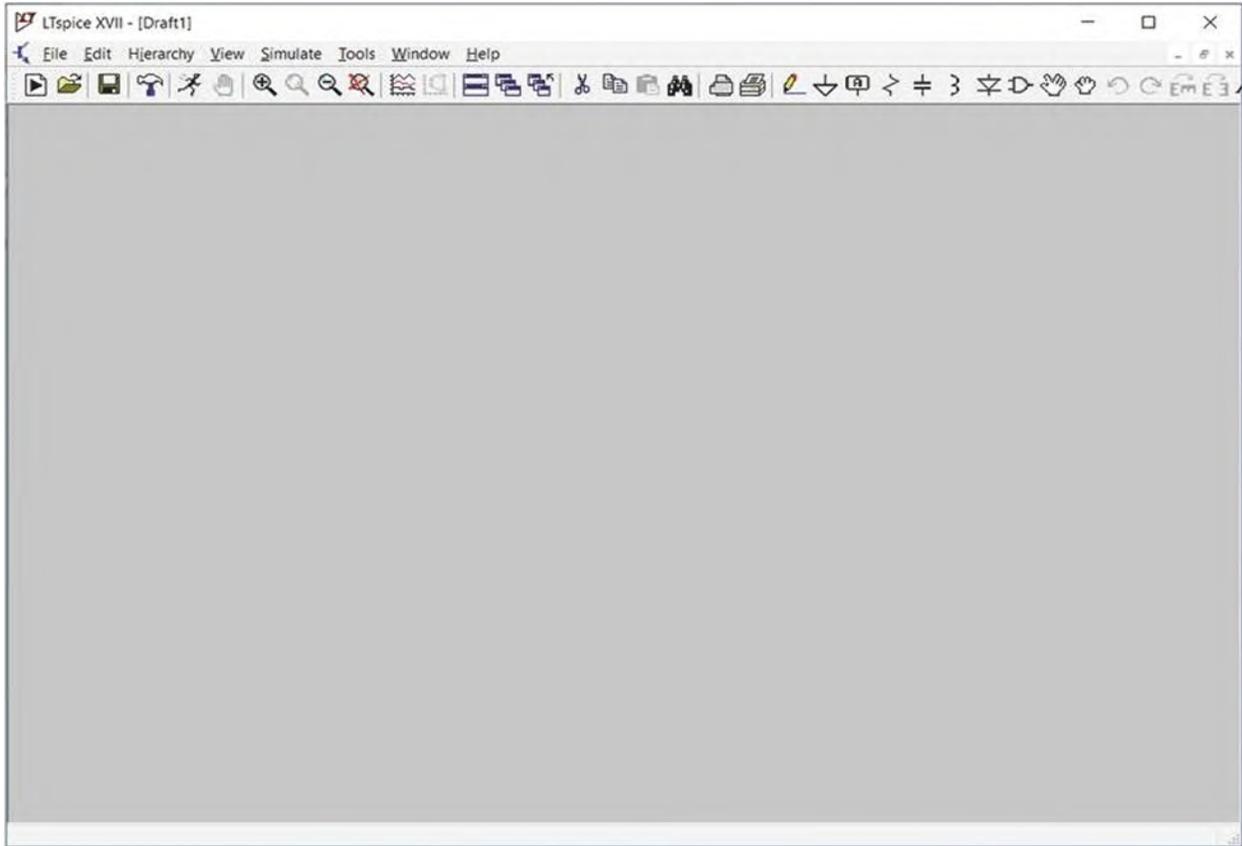


Fig 6.15 New schematic file is opened

Click the resistor icon (Fig 6.16). After clicking the resistor icon, the mouse pointer changes to a resistor. If you press the **Ctrl+R**, the resistor will be rotated. Press the **Ctrl+R** once and then click on the schematic to add a resistor to it (Fig 6.17).

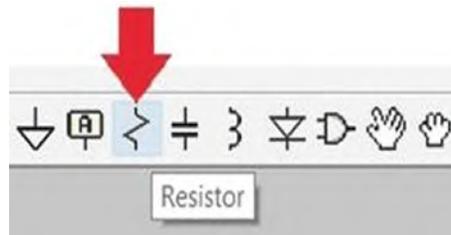


Fig 6.16 Resistor icon

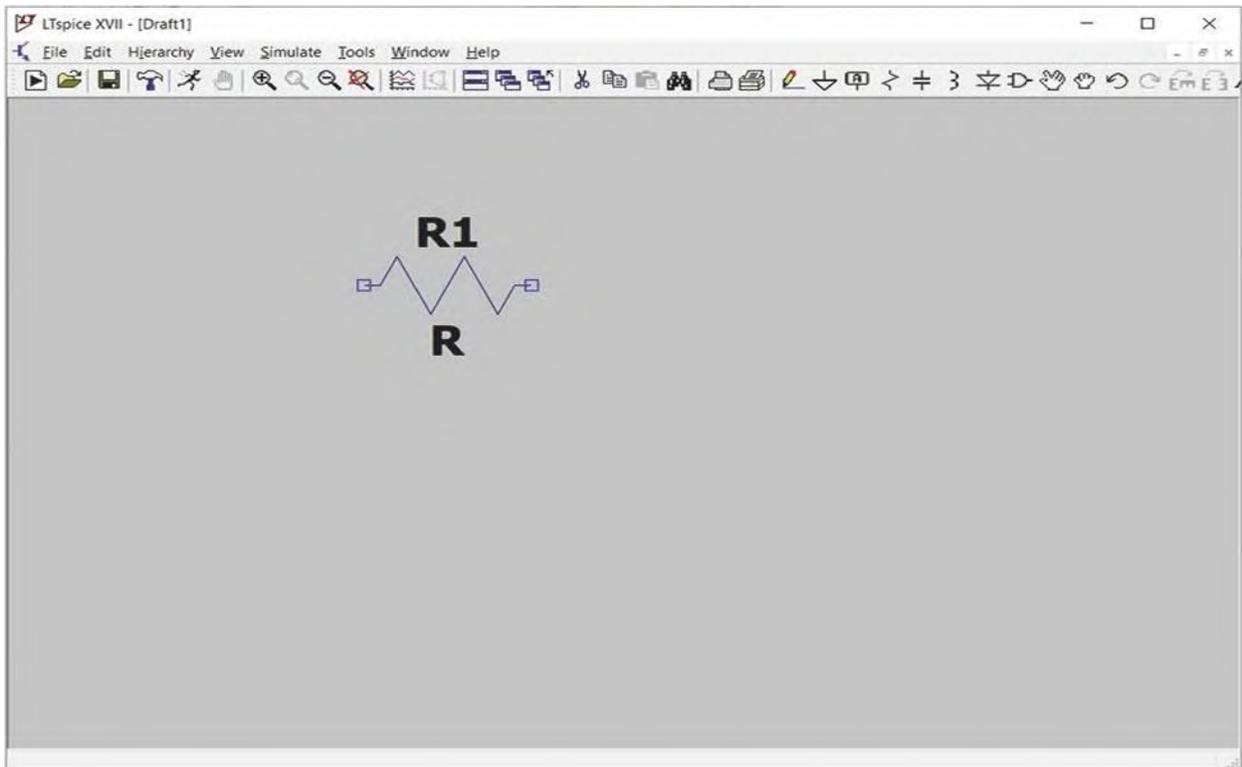


Fig 6.17 Addition of a resistor to the schematic

Add another resistor to the schematic (Fig 6.18), and after that, press the **Esc** key of your keyboard.

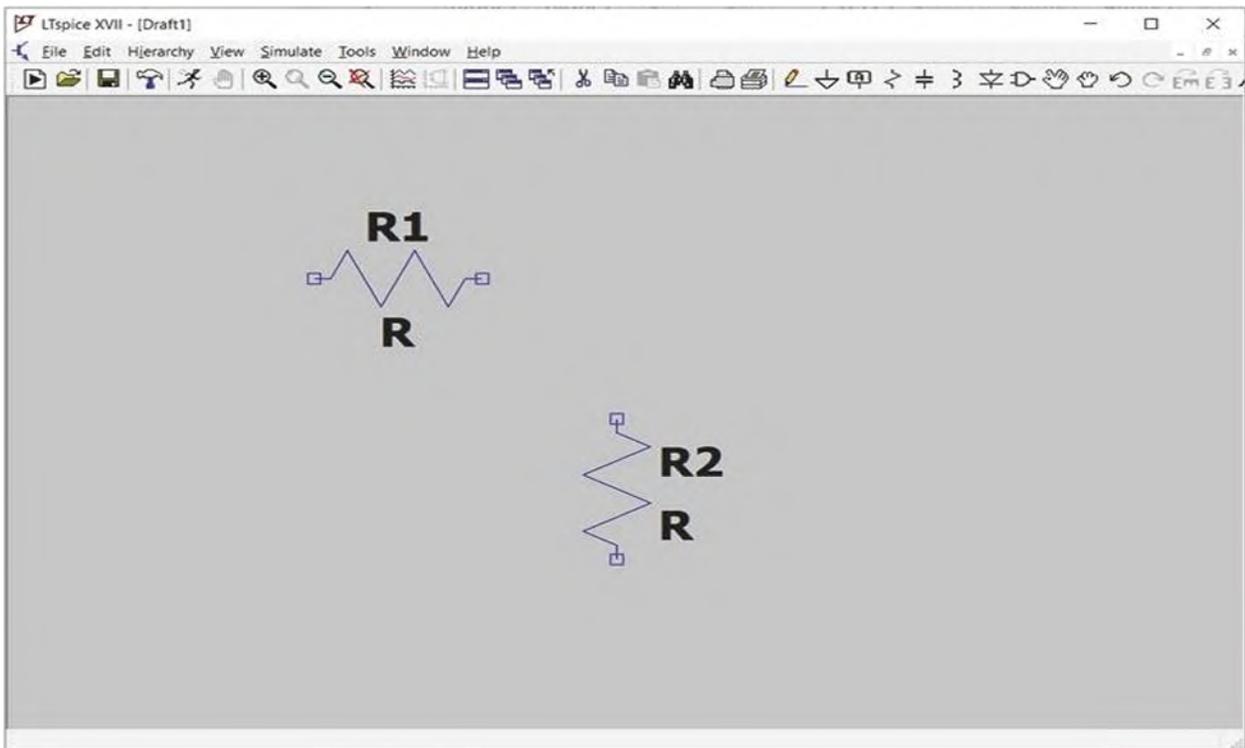


Fig 6.18 Addition of second resistor to the schematic

Clicking the Resistor icon (Fig 6.16) is not the only way to add resistors to the schematic. You can press the 'R' key of your keyboard as well. The shortcut keys of LTspice are shown in Fig 6.19. It is a good idea to memorize these shortcuts since they help you to draw the schematic easier and faster.

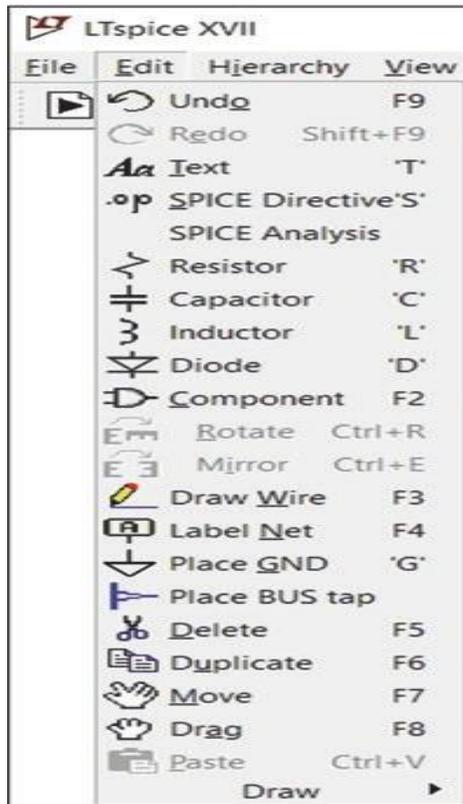


Fig 6.19 Edit menu

Click on the Component icon (Fig 6.20).

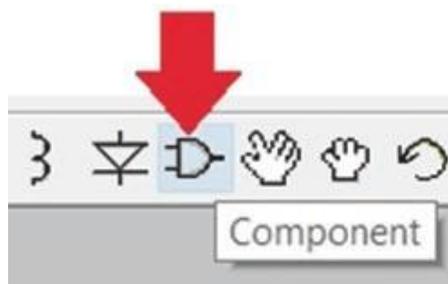


Fig 6.20 Component icon

After clicking the Component icon, the Select Component Symbol window (Fig 6.21) appears. Enter “voltage” in the component search box to find the voltage source block. The current source block can be found by searching for “current,” as well (Fig 6.22).

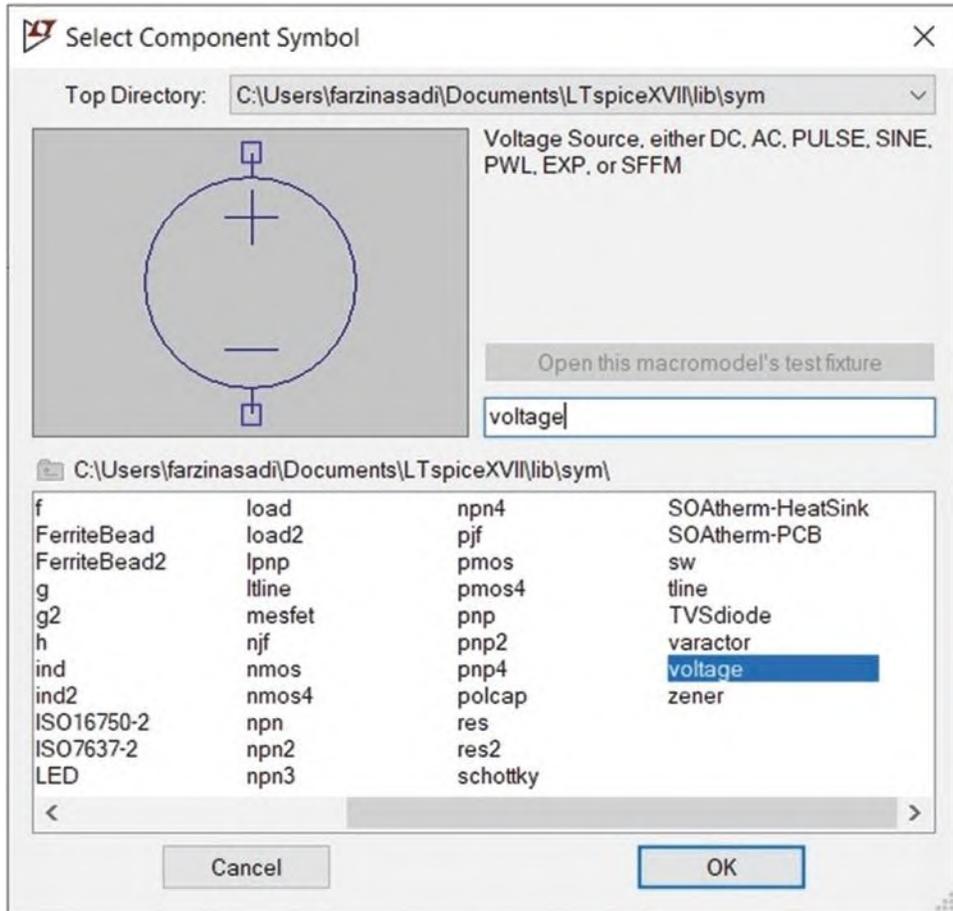


Fig 6.21 Voltage source block

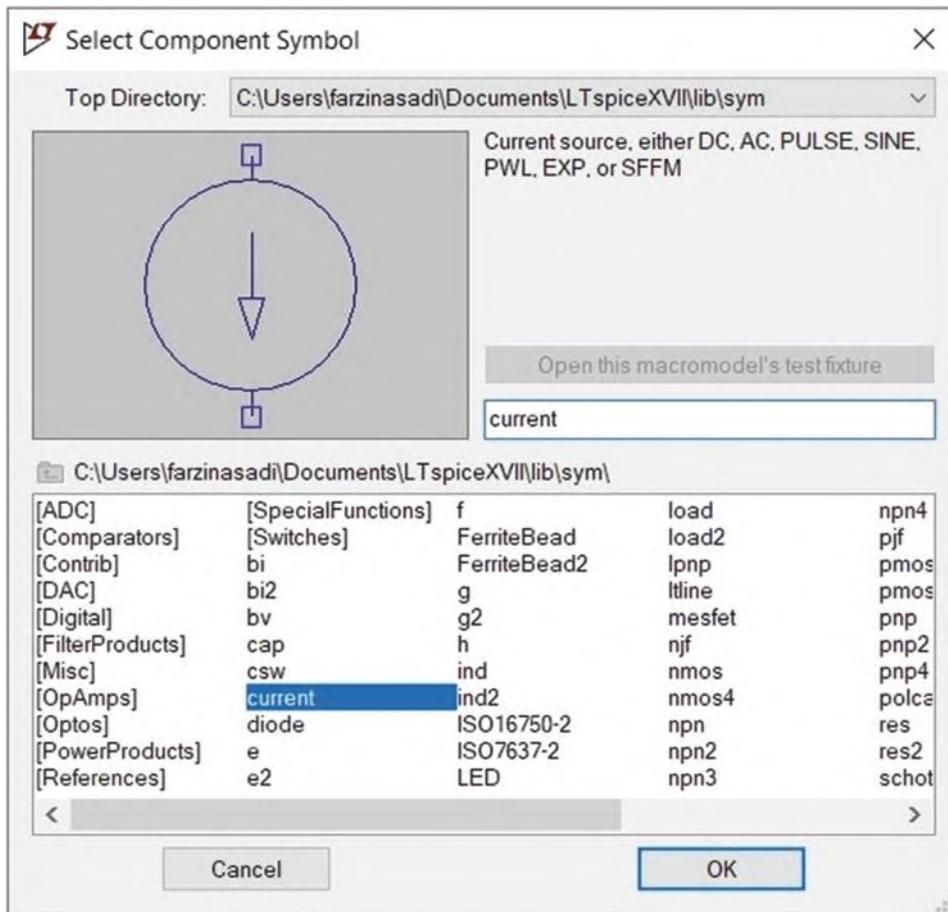


Fig 6.22 Current source block

Add a voltage source block to the schematic (Fig 6.27).

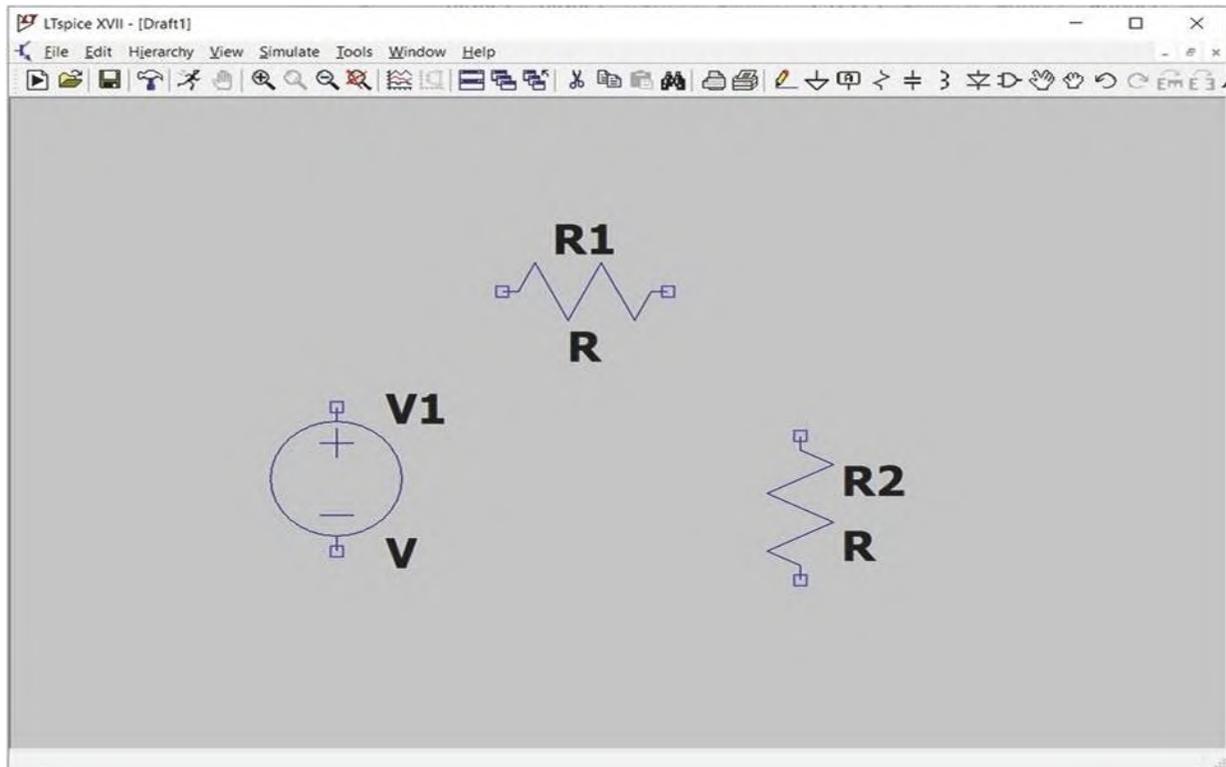


Fig 6.27 Addition of a voltage source to the schematic

You can delete a component with the aid of scissor icon (Fig 6.28). Click the scissor icon (or press F5) and then click on the component that needs to be removed.

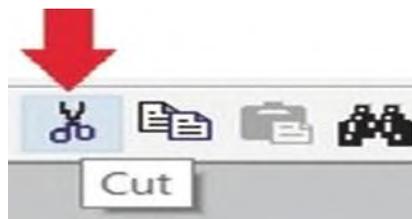


Fig 6.28 Cut icon

You can duplicate a component with the aid of Copy icon (Fig 6.29). Click the Copy icon (or press F6) and then click on the component that you want to make a copy of it. After clicking on the component, a copy of that component is attached to the mouse pointer. If you click on the schematic, the copied component will be attached to the schematic.

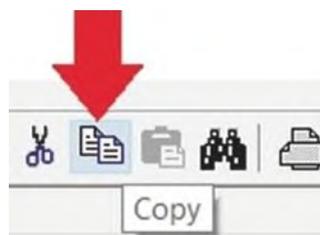


Fig 6.29 Copy icon

Use the Wire icon (Fig 6.30) to connect the components together (Fig 6.31).

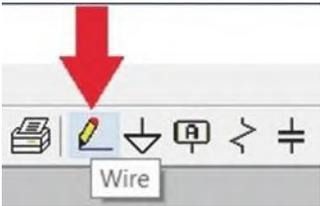


Fig 6.30 Wire icon

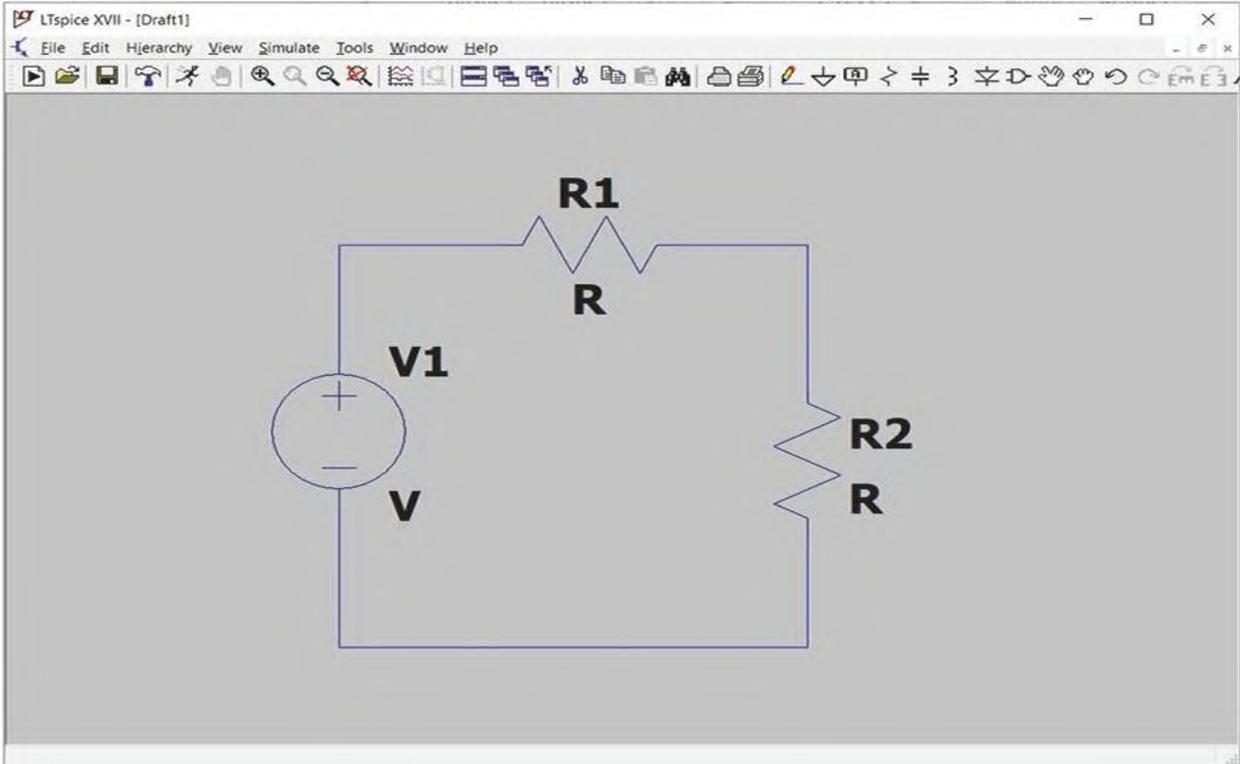


Fig 6.31 Connecting the components together

Use the ground icon (Fig 6.32) to add ground to your circuit (Fig 6.33). If you try to run a schematic without ground, the error message shown in Fig 6.34 appears.

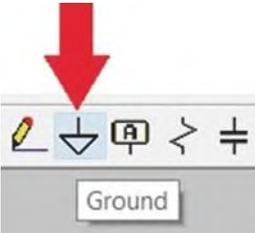


Fig 6.32 Ground icon

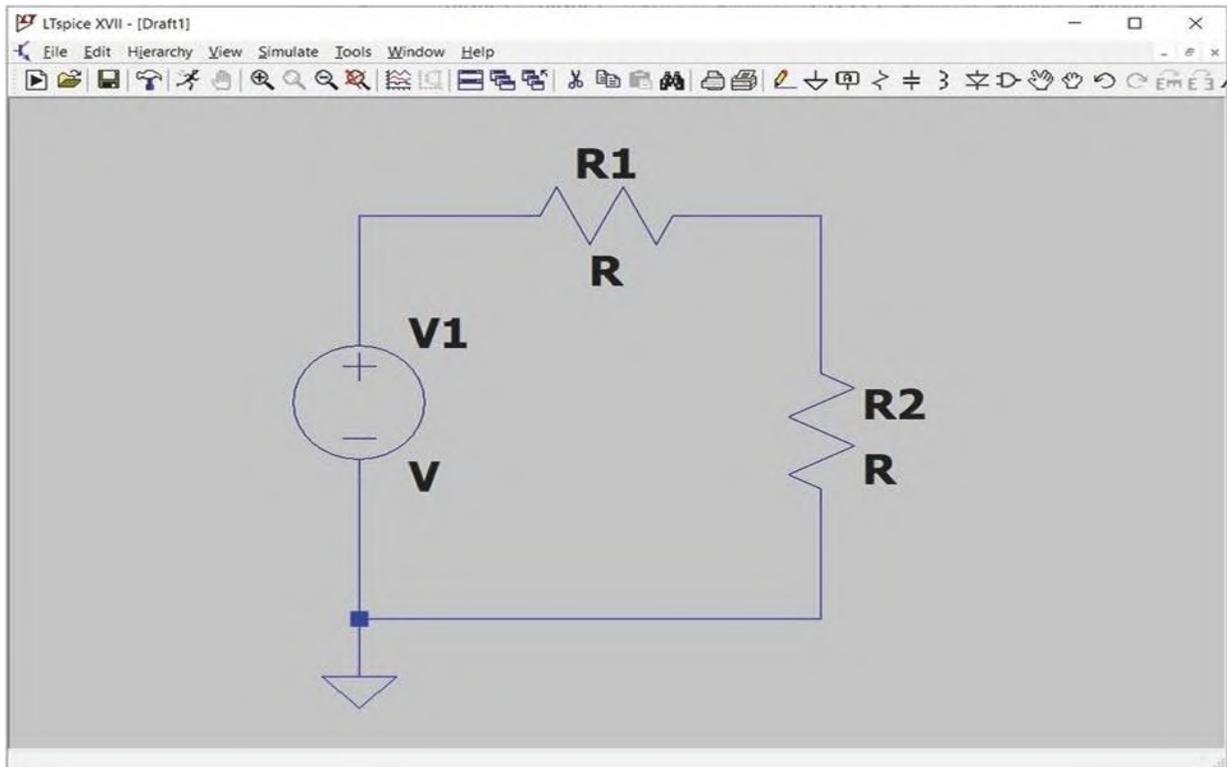
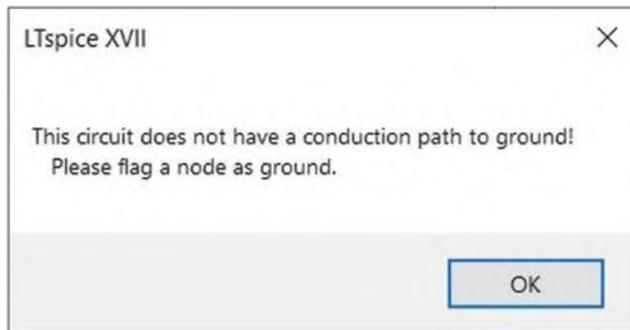


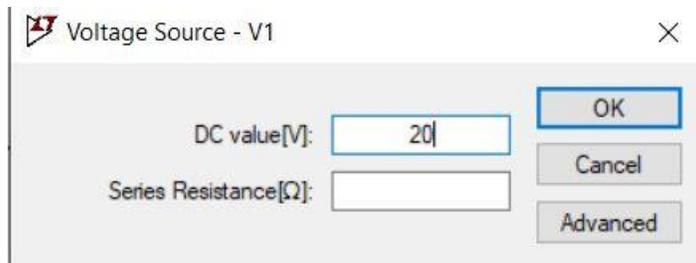
Fig 6.33 Addition of ground to the schematic

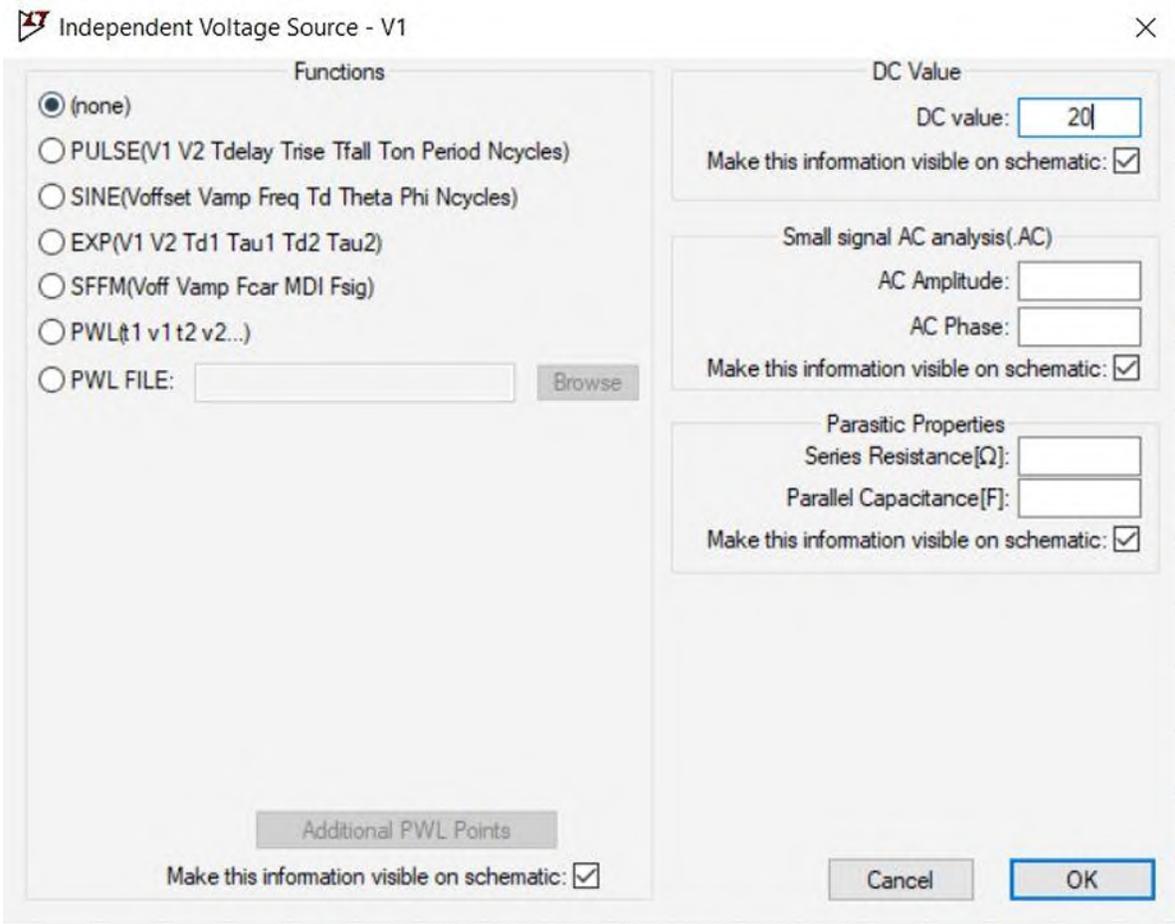
Fig 6.34 Error message for a circuit without ground



Right click on the voltage source **V1** and enter **20** to the DC value[V] box (Fig 6.35) and click the OK button. If you click the Advanced button in Fig 6.35, the window shown in Fig 6.36 appears and permits you to produce more complicated waveforms. In this example, we need a simple DC voltage source, so there is no need to change the settings of Fig 6.36.

Fig 6.35 Voltage source settings





6.36 Voltage source settings

After clicking the OK button of Fig 6.36, the schematic changes to what is shown in Fig 6.37. The value of DC voltage source is shown behind it.

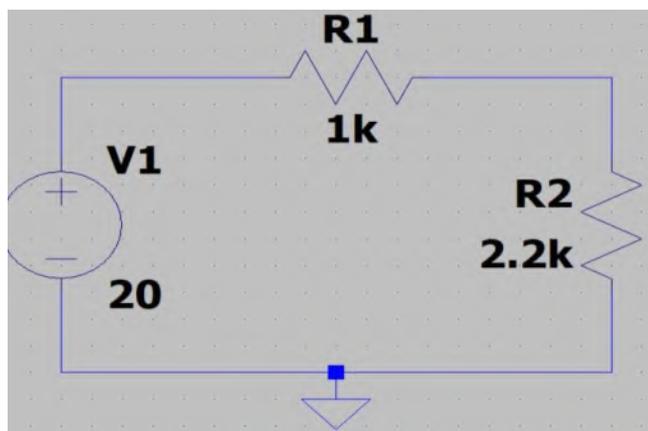


Fig 6.37 Value of voltage source is shown on the schematic.

Right click on the resistor **R1** and enter **1k** to the Resistance [Ω] box (Fig 6.38). Then click the OK button. After clicking the OK button, the schematic changes to what is shown in Fig 6.39. The value of resistor R1 is shown behind it. List of prefixes that can be used in LTspice is shown in Table A.

Fig 6.38 Entering the value of resistor R1

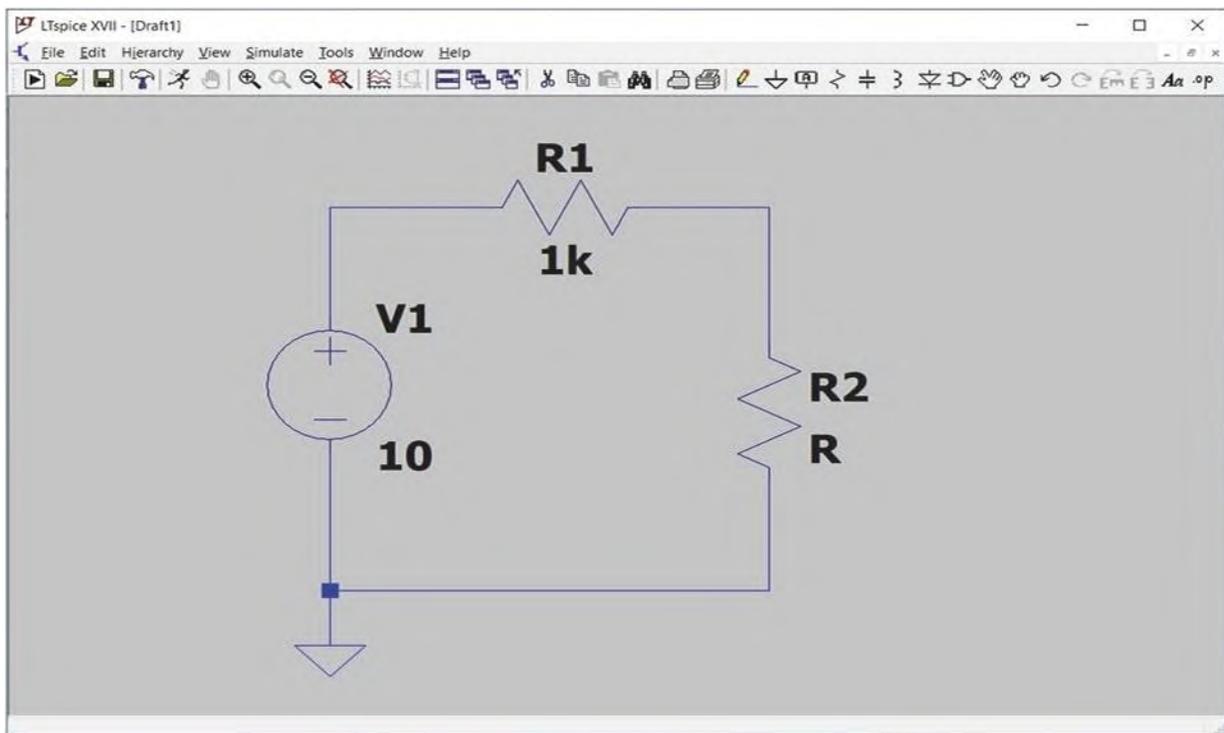
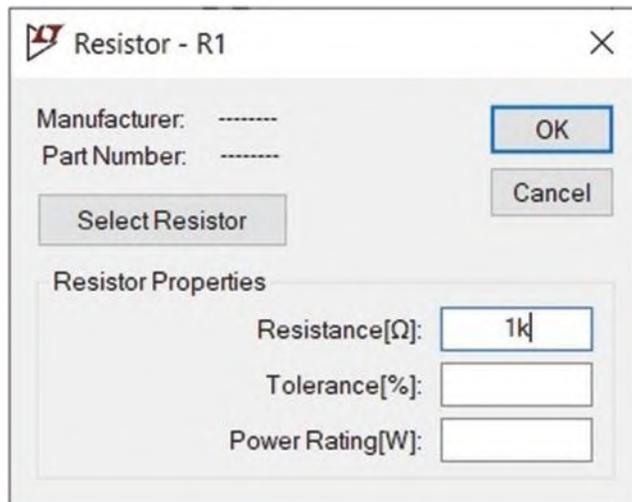


Fig 6.39 Value of resistor R1 is shown on the schematic

Table A Prefixes that can be used in LTspice

Unit (prefix)	Unit	Multiple
T	Tera	10^{12}
G	Giga	10^9
Meg	Mega	10^6
k	Kilo	10^3
m	Milli	10^{-3}
u	Micro	10^{-6}
n	Nano	10^{-9}
p	Pico	10^{-12}
f	Femto	10^{-15}

Right click on the resistor **R2** and enter **2.2k** to the Resistance [Ω] box (Fig 6.40). Then click the OK button. After clicking the OK button, the schematic changes to what is shown in Fig 6.41. The value of resistor R2 is shown behind it.

Fig 6.40 Entering the value of resistor R2

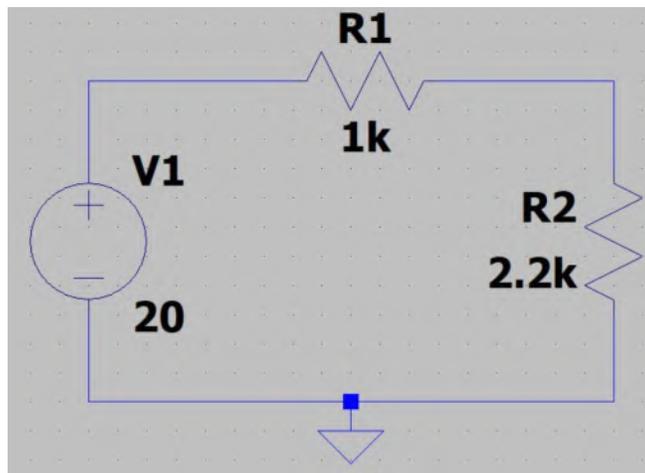
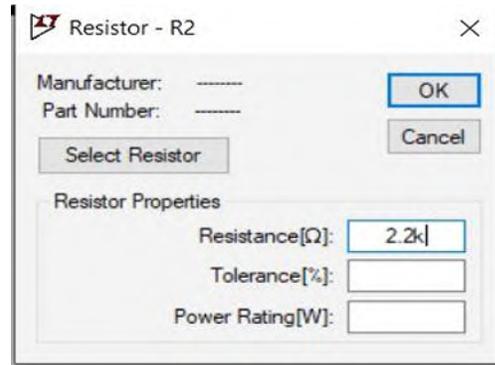


Fig 6.41 Value of resistor R2 is shown on the schematic

When you open a new schematic in LTspice, the default name Draftn (n shows a number) is assigned to it. If you want to save the file with your desired name, you need to use the File> Save As (Fig. 6.42). After Save As the file with desired name, you can click the Save icon (Fig. 6.43) to save the changes you apply to the schematic file.

Fig. 6.42: File > Save As

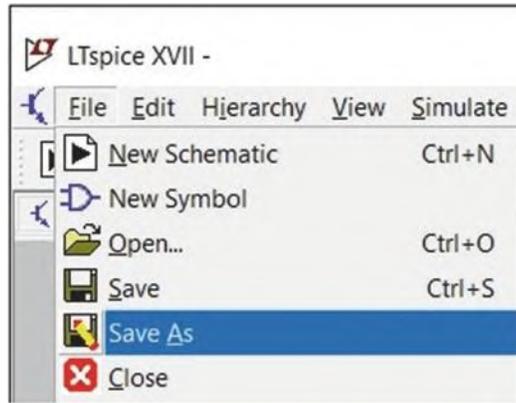
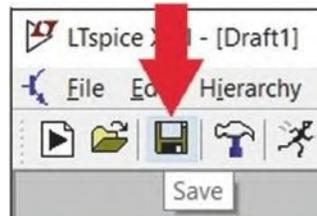


Fig. 6.43: Save icon



The Run icon (Fig. 6.44) can be used to determine the type of simulation and run the simulation. In this example, we want to find the steady-state DC voltages/currents. In order to do this, click the **Run** icon, (Fig. 6.44) and open the *DC op pnt* tab (Fig. 6.45).

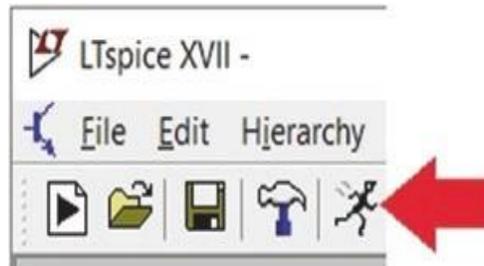


Fig. 6.44: Run icon

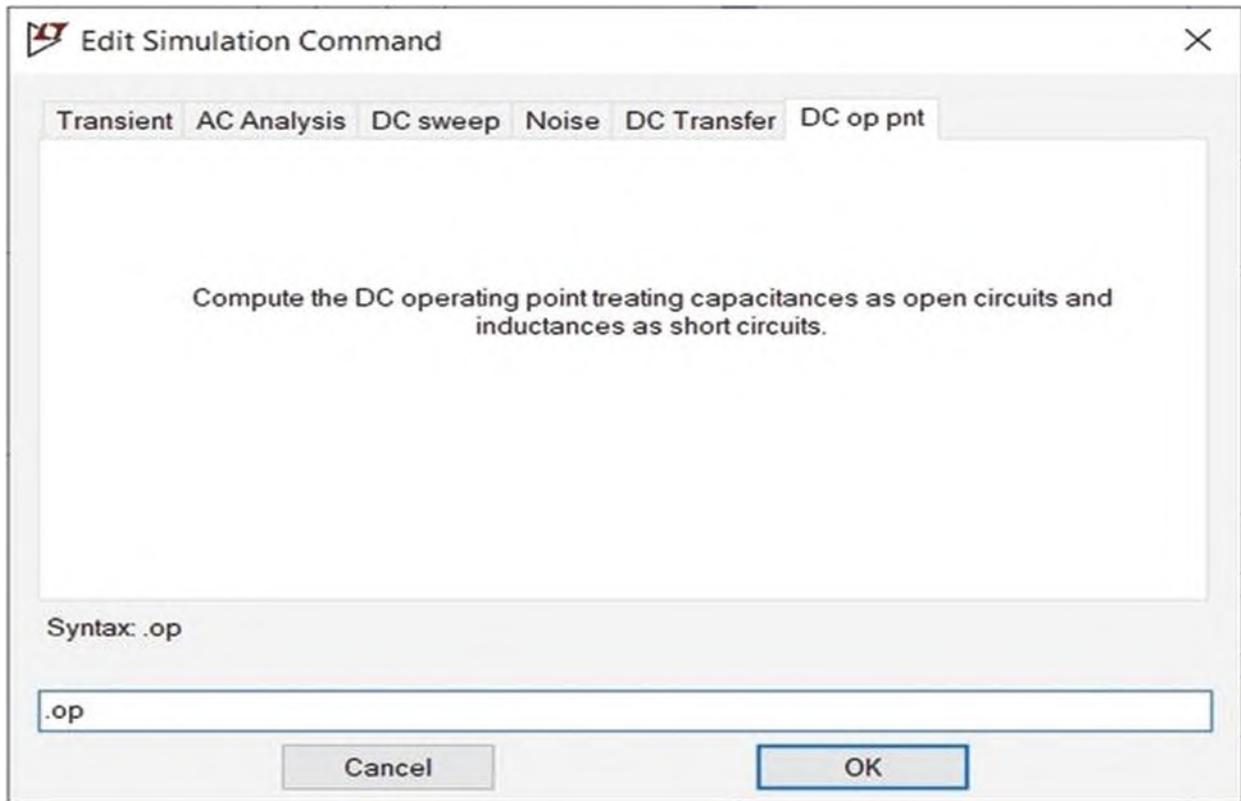


Fig. 6.45: DC op pnt tab

After clicking the **OK** button, the “.op” command is added to the schematic (Fig. 6.46) and the result shown in Fig. 6.47 appears.

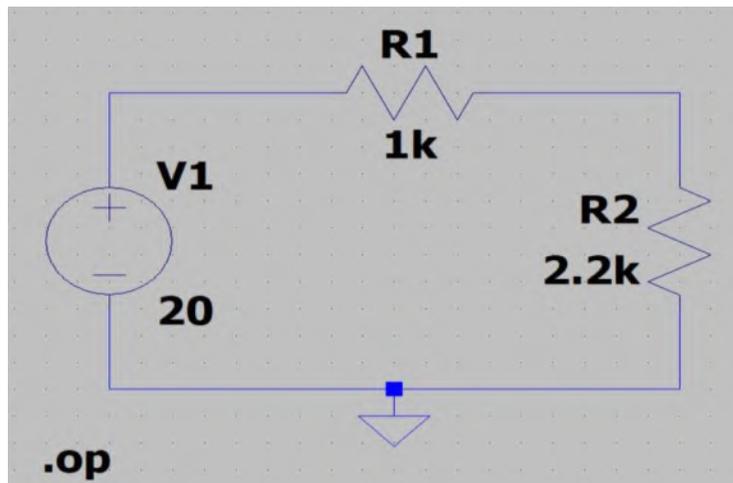


Fig. 6.46: “.op” command is added to the schematic

```

C:\Users\USER\Desktop\LTSPICE pic\Draft3.asc
--- Operating Point ---
V(n001):      20      voltage
V(n002):      13.75   voltage
I(R1):        -0.00625 device_current
I(R2):        -0.00625 device_current
I(V1):        -0.00625 device_current

```

Fig. 6.47: Simulation result for Example 1.

6.2a Verification of Superposition Theorem

Superposition theorem states that; “In any linear bilateral network having multiple sources, the response (voltage and current) in any element is equal to the summation of all responses caused by individual source acting alone.” In this experiment, we measure the current (or voltage) due to combination of sources of the original circuit and then measure the current for each individual source. Then verify the theorem by comparing the algebraic sum to of the currents due to individual sources with the current due to original setup. In this example, we want to verify superposition theorem for the following circuit (Fig. 6.48).

CASE 1: All the sources (E1 and E2) are active

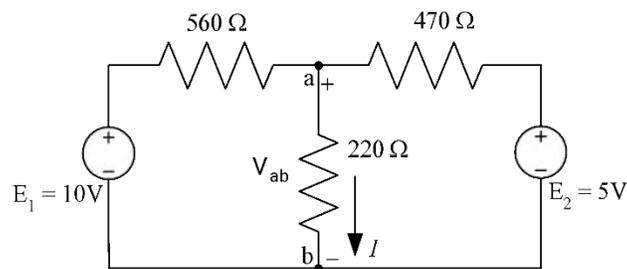


Fig 6.48: Experimental circuit (case 1: all the sources E_1 and E_2 active).

Run the LTspice (Fig 6.49).

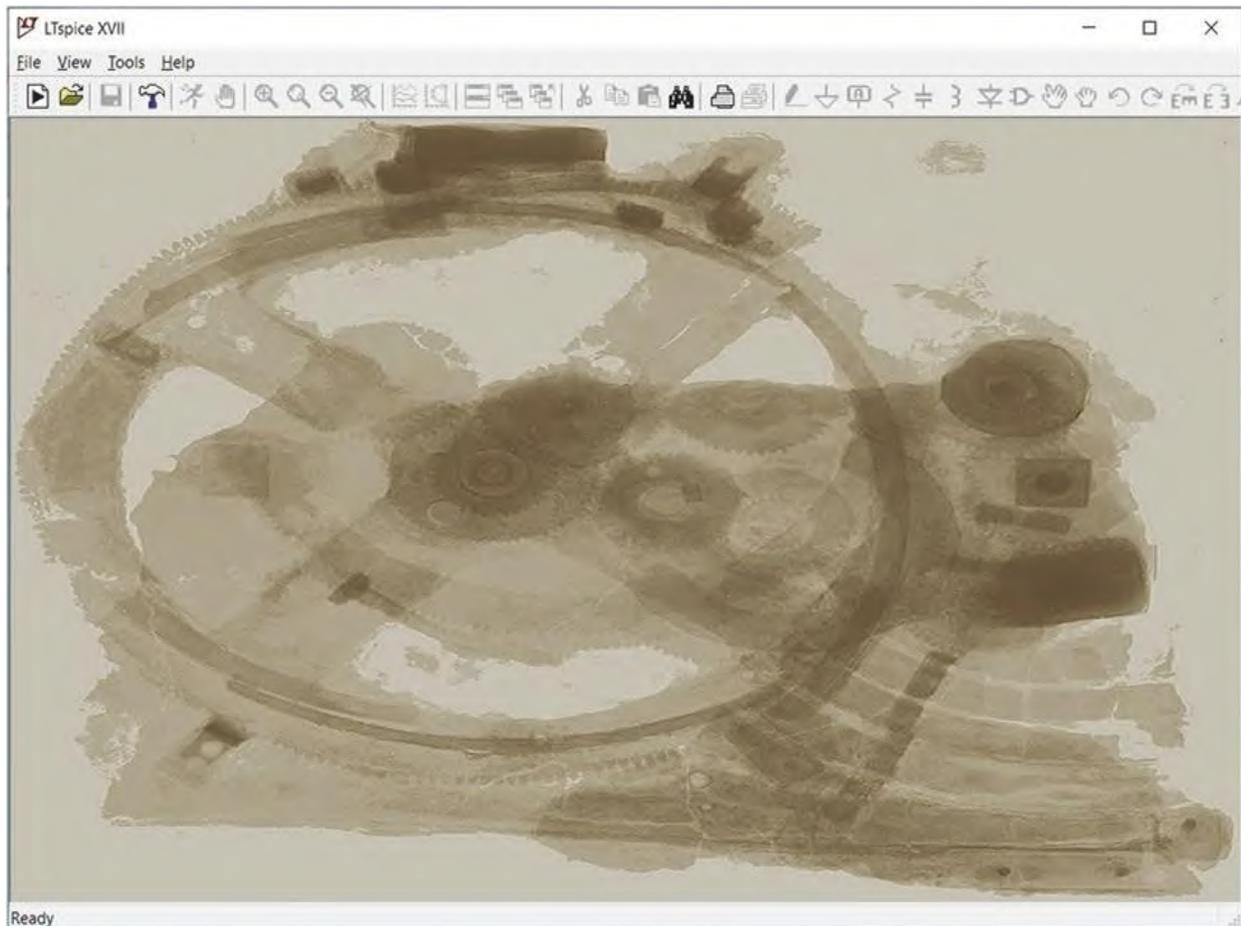


Fig 6.49: LTspice environment

Click the **New Schematic** icon (Fig 6.50). This opens a new schematic for you (Fig 6.51).

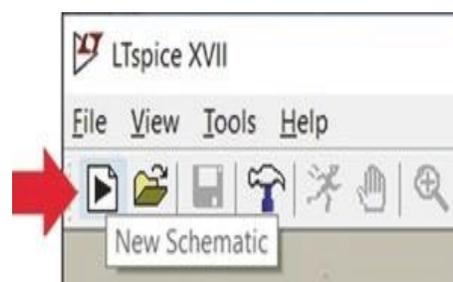


Fig 6.50: New Schematic icon

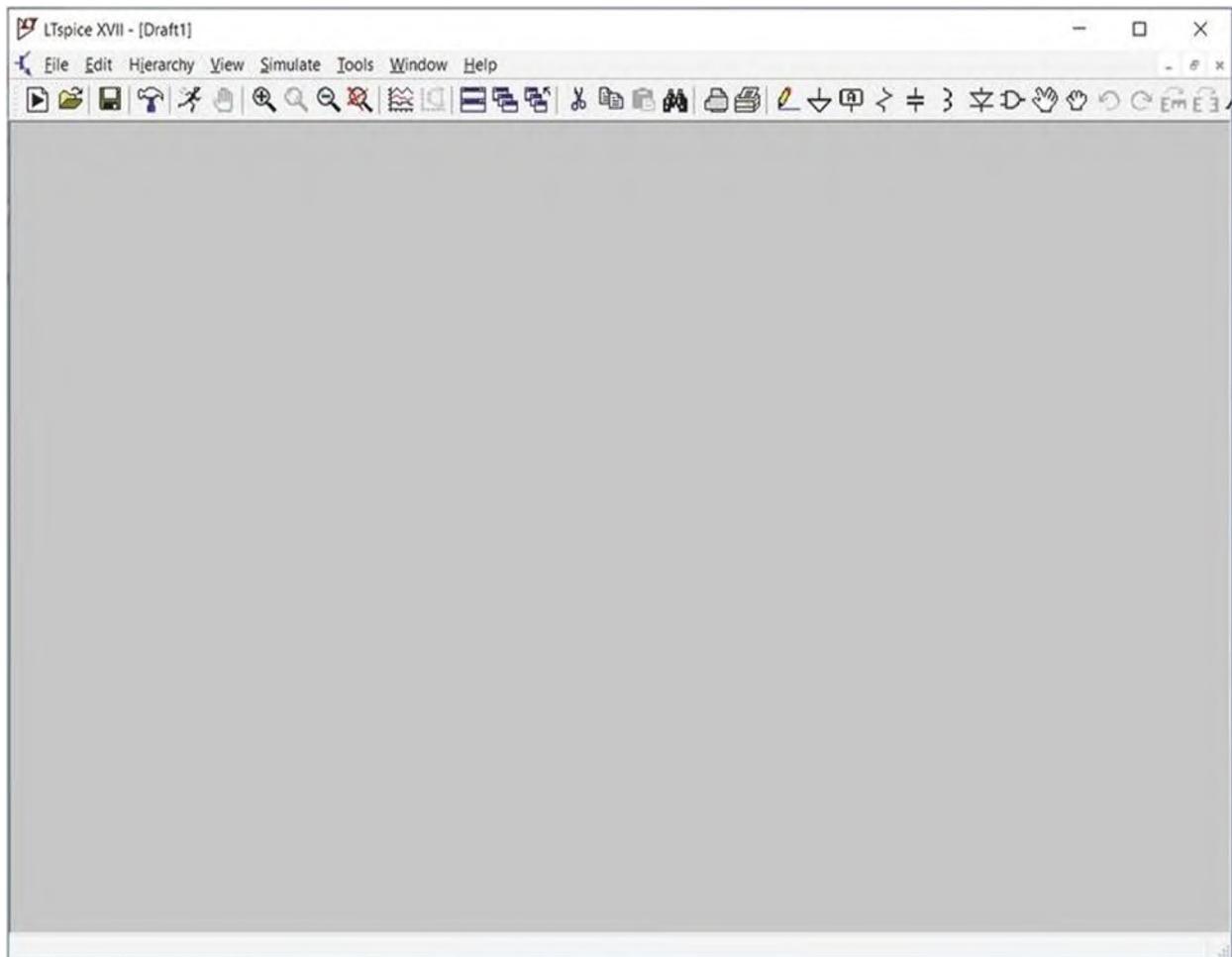


Fig 6.51: New schematic file is opened

Click the resistor icon (Fig 6.52). After clicking the resistor icon, the mouse pointer changes to a resistor. If you press the **Ctrl+R**, the resistor will be **rotated**. Press the **Ctrl+R** once and then click on the schematic to add a resistor to it (Fig 6.53).

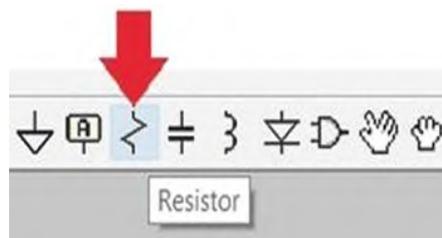


Fig. 6.52: Resistor icon

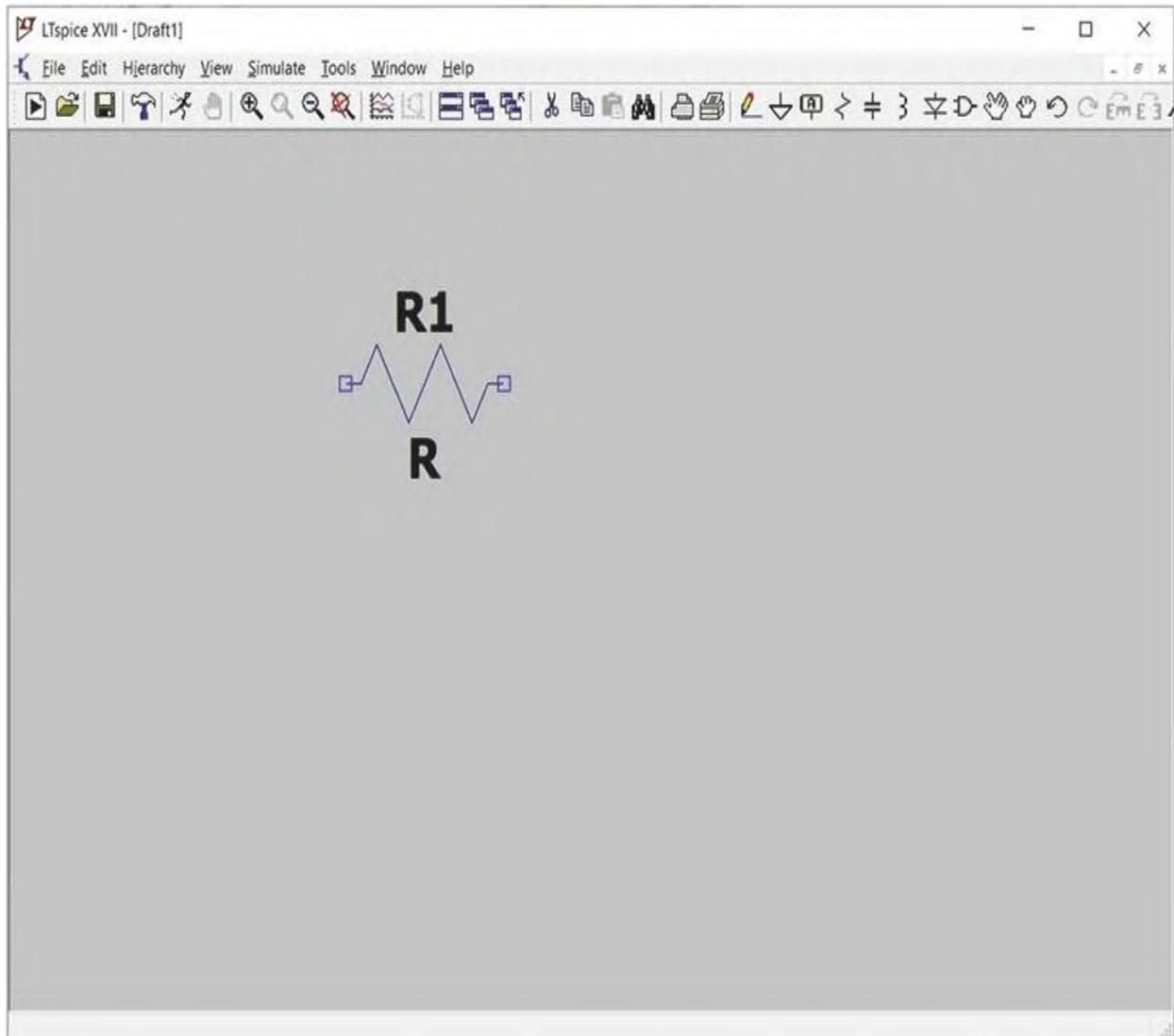


Fig 6.53: Addition of a resistor to the schematic

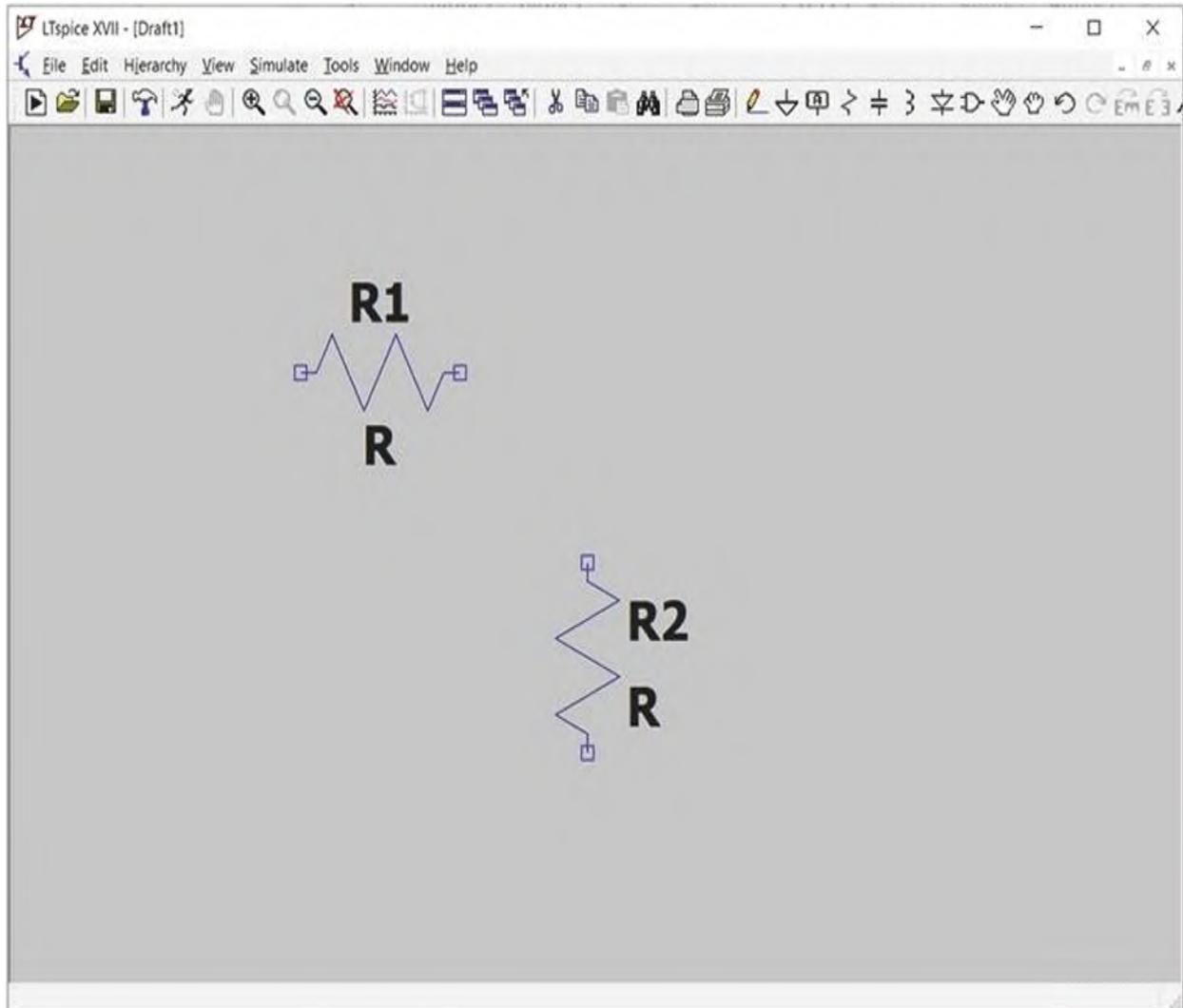


Fig 6.54: Addition of second resistor to the schematic

Add another two resistors to the schematic (Fig 6.54), and after that, press the “**Esc**” key of your keyboard.

Add a voltage source block to the schematic (Fig. 6.55).

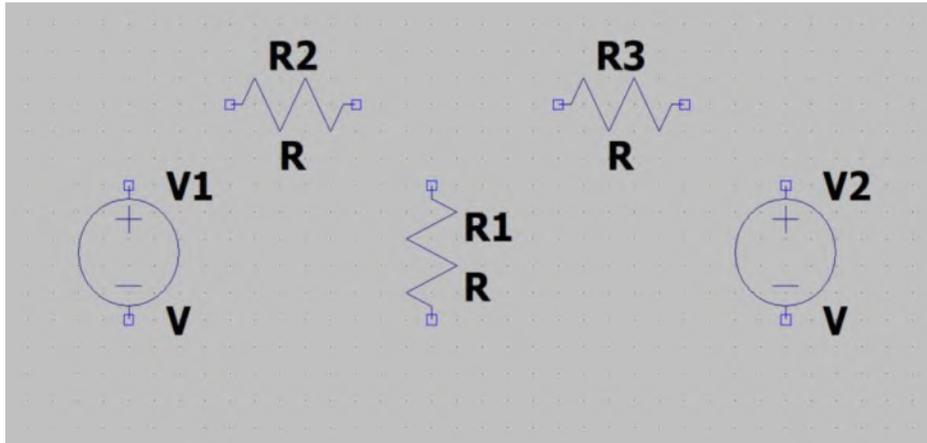


Fig. 6.55: Addition of a voltage source to the schematic

Use the Wire icon (Fig. 6.56) to connect the components together (Fig. 6.57).

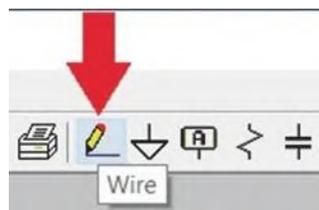


Fig. 6.56: Wire icon

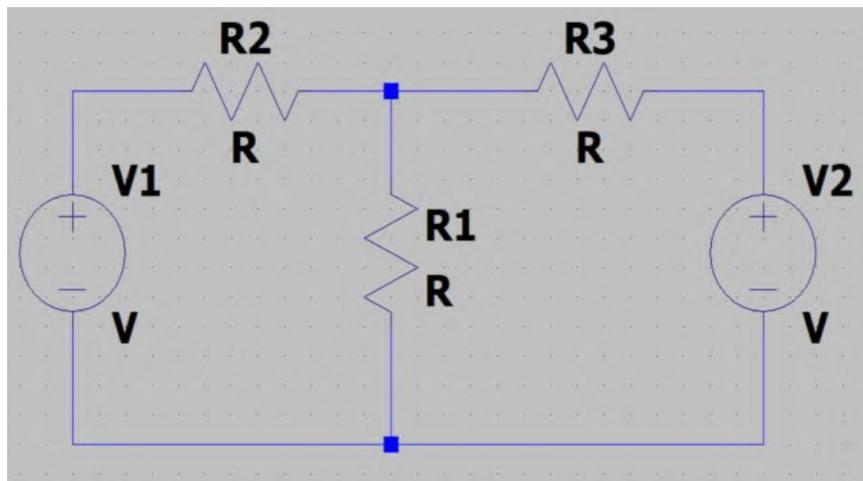


Fig. 6.57: Connecting the components together

Use the ground icon (Fig. 6.58) to add ground to your circuit (Fig. 6.59).

Fig. 6.58: Ground icon

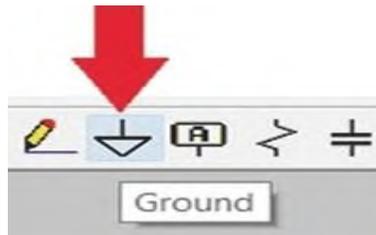
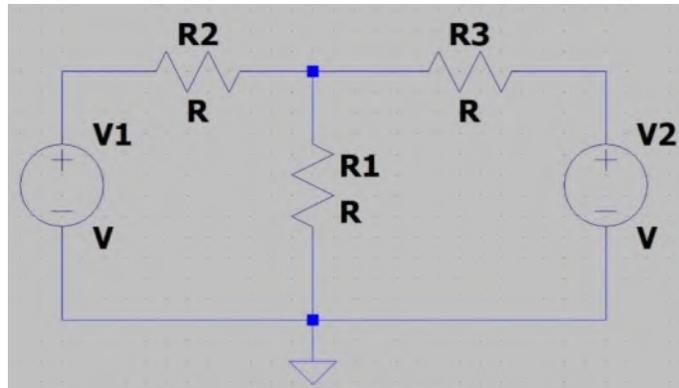
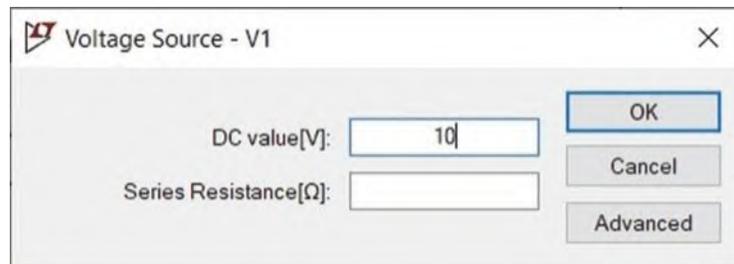


Fig. 6.59: Addition of ground to the schematic



Right click on the *voltage source V1* and enter **10** to the DC value[V] box (Fig. 6.60) and click the OK button. Do the same for *voltage source V2* and enter **5** to the DC value[V] box.

Fig. 6.60: Voltage source settings



After clicking the **OK** button, the schematic changes to what is shown in Fig. 6.61. **The value of DC voltage source is shown behind it.**

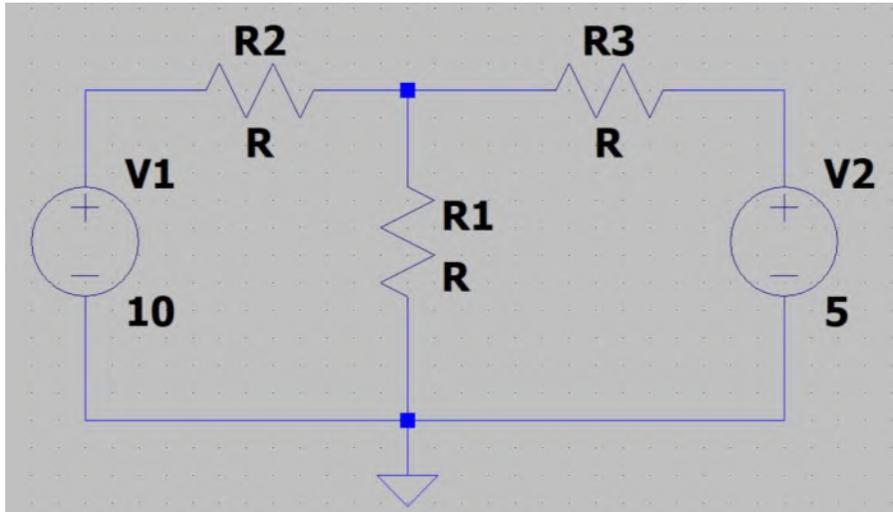


Fig. 6.61: Value of voltage source is shown on the schematic.

Right click on the resistor R1 and enter 220 to the Resistance [Ω] box (Fig. 6.62). Then click the OK button. After clicking the OK button, the schematic changes to what is shown in Fig. 6.63. The value of resistor R1 is shown behind it.

Fig. 6.62: Entering the value of resistor R1

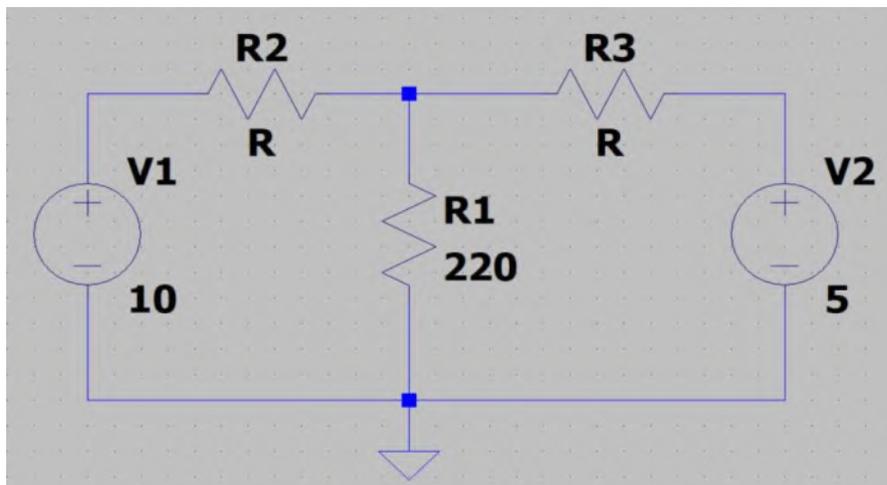
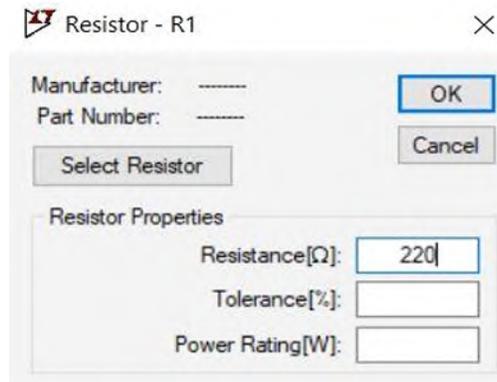


Fig. 6.63: Value of resistor R1 is shown on the schematic

Right click on the resistor **R2** and enter **560** to the Resistance [Ω] box (Fig. 6.63). Then click the OK button. The value of resistor R2 is shown behind it. Do the same for the resistor **R3** and enter **470** to the Resistance [Ω] box. After clicking the OK button, the schematic changes to what is shown in Fig. 6.64.

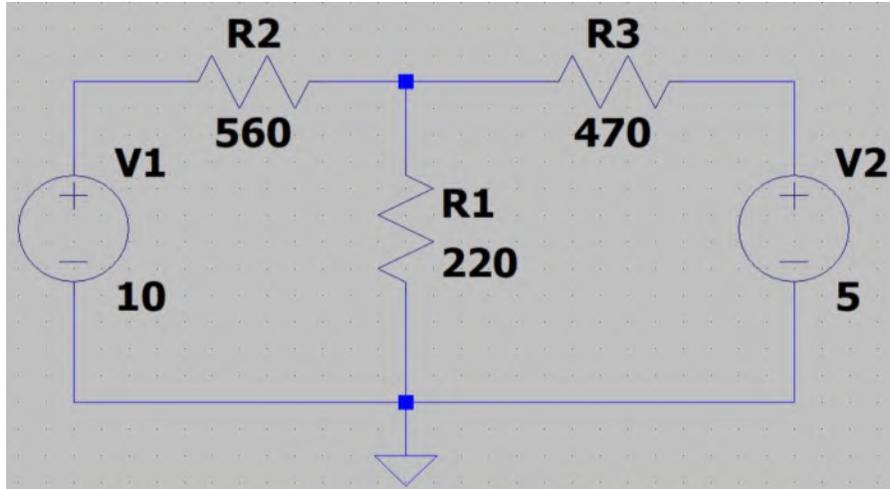


Fig. 6.64: Value of resistor R2 is shown on the schematic

You can change the components labels by right clicking on them and entering the new name. For instance, if you right click on the “**V1**” label in Fig. 6.64, the window shown in Fig. 6.65 appears and permits you to enter a *new name* to the voltage source. Enter “**E1**” for the 10V source and “**E2**” for the 5V source. The circuit with modified levels is shown in **Fig. 6.66**.

Fig. 6.65: Entering a new name

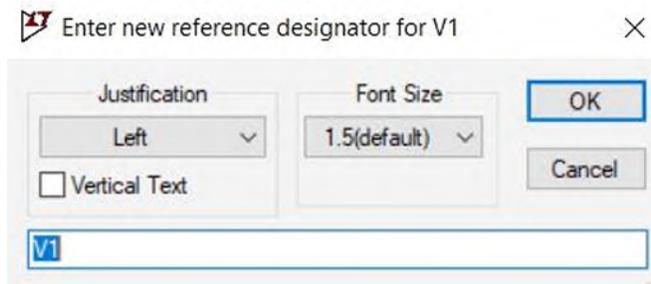
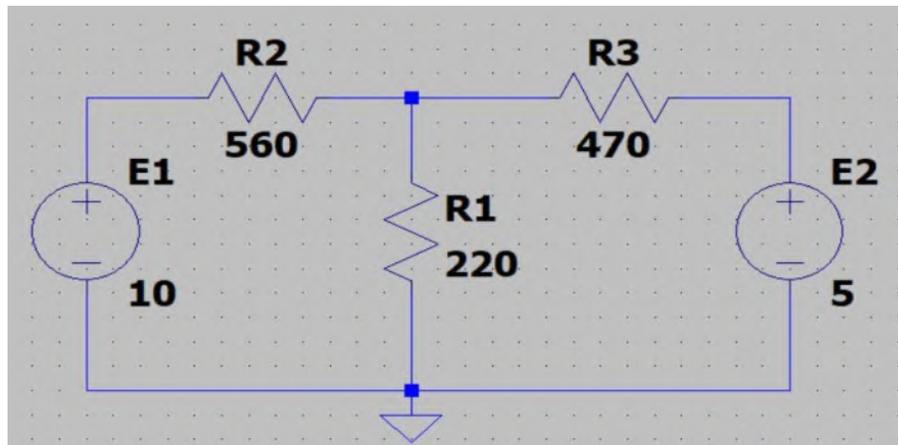


Fig. 6.66: Experimental circuit with labelled components



You can use the Label Net icon (Fig. 6.67) to give the desired names to the circuit nodes. After clicking the Label Net icon, the window shown in Fig. 6.68 appears and permits you to enter the desired name.

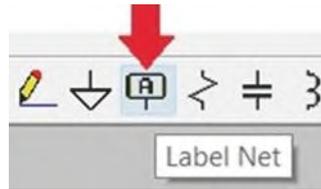


Fig. 6.67: Label Net icon

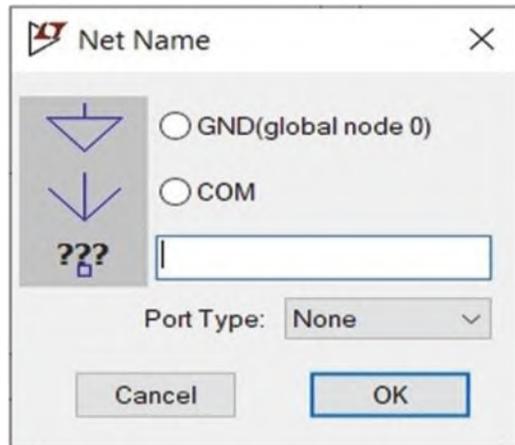


Fig. 6.68: Net Name window

Let's give the name "a" to the node which is connected to the upper terminal of resistor R1. To do this, click the Label Net icon and enter "a" to Net Name window and click the **OK** button (Fig. 6.69). After clicking the OK button, the entered name (a) is attached to the mouse pointer and a small square is under it. Place the square on the upper terminal of resistor R1 (Fig. 6.70) and click. After clicking, the upper terminal of R1 is renamed to "a" (Fig. 6.71).

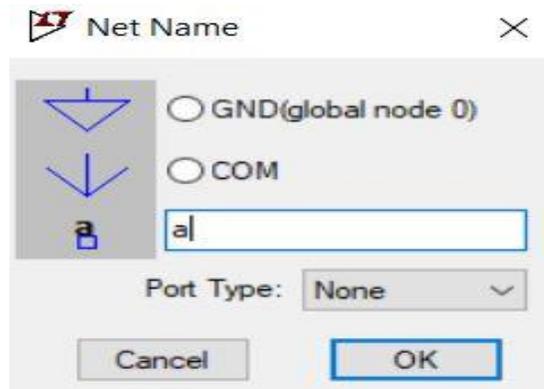


Fig. 6.69: Entering "a" to Net Name window

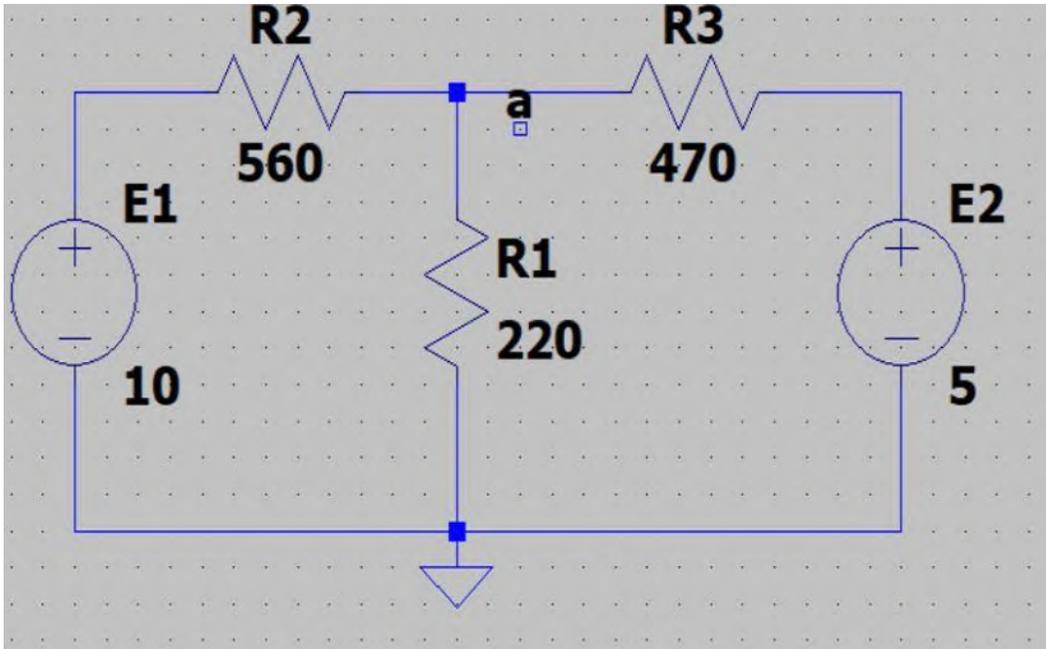


Fig. 6.70: Assigning the name “a” to upper terminal of R1 (=right terminal of R2)

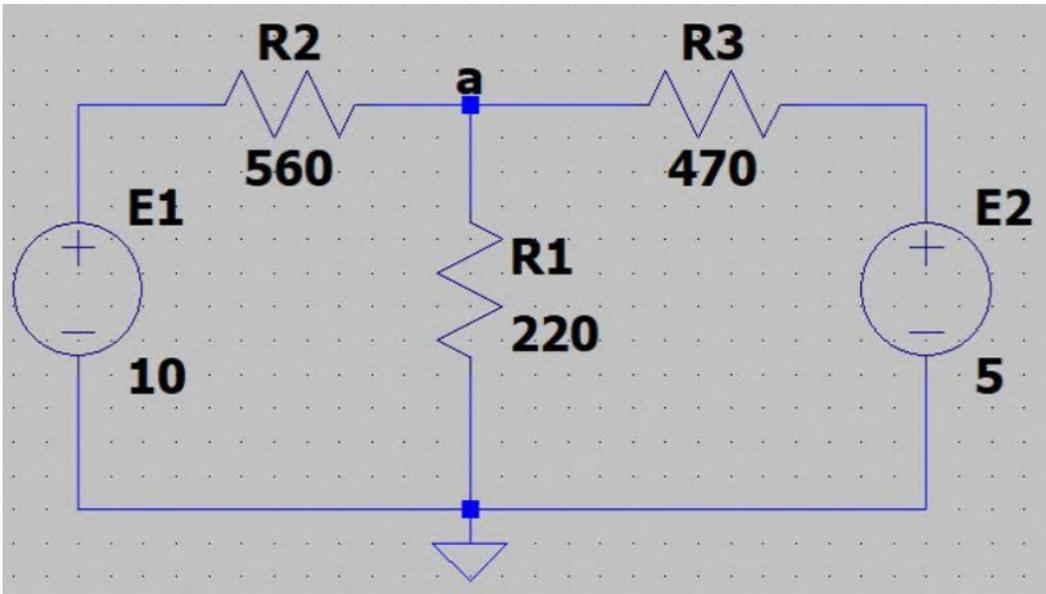


Fig. 6.71: Upper terminal of R1 (=right terminal of R2) is renamed to “a”

Now do the same to name the node connected to the lower terminal of **R1** to “**b**” (Fig. 6.72).

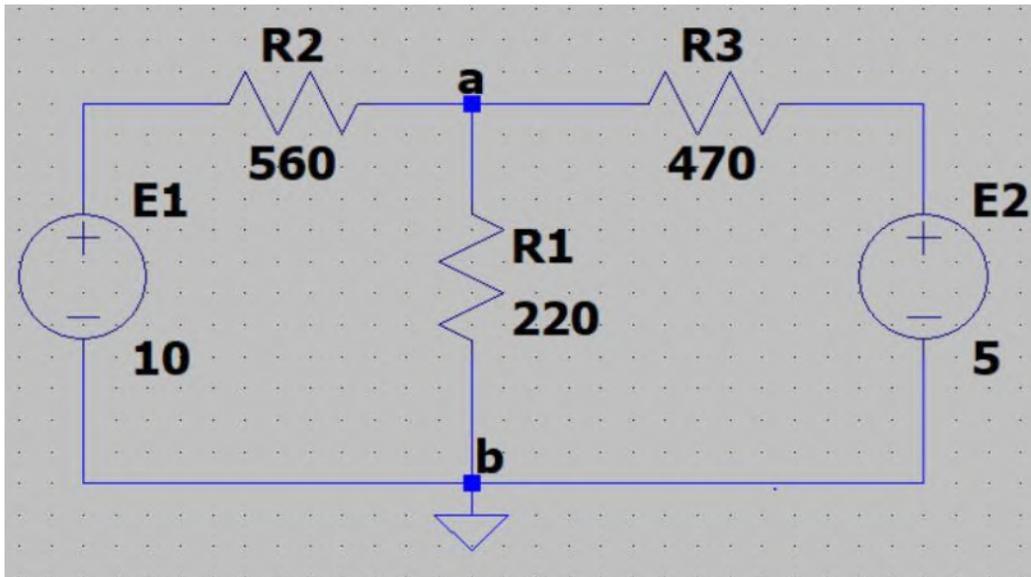


Fig. 6.72: Renaming the ground node to “**b**”

You can add text to the schematic by clicking the Text icon (Fig. 6.73). After clicking the Text icon, the window shown in Fig. 6.74 appears and permits you to enter the desired text. After entering the desired text, click the OK button and then click on the schematic to add the text to it (Fig. 6.75).

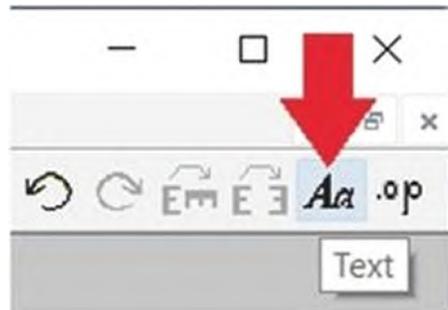


Fig. 6.73: Text icon

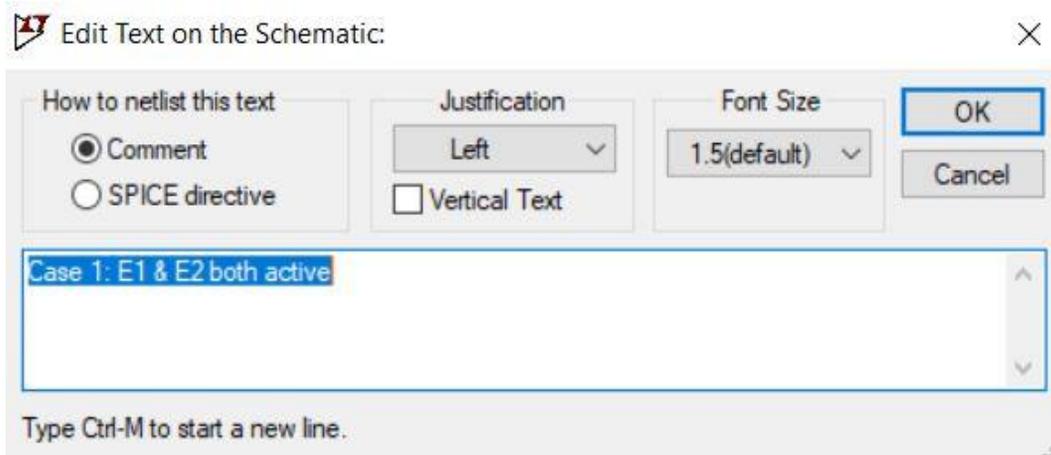


Fig. 6.74: Comment radio button

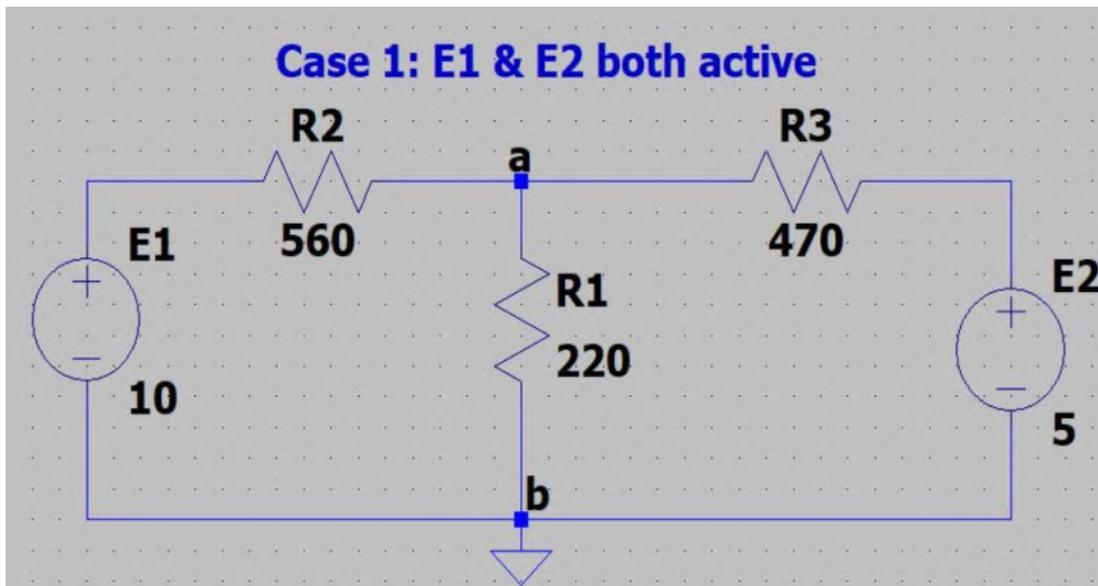


Fig. 6.75: Addition of a comment to the schematic

You can zoom in/out with the aid of mouse scroll button or the icons shown in Fig. 6.76. If you press the spacebar key of your keyboard, the best settings are selected to show the drawn schematic.

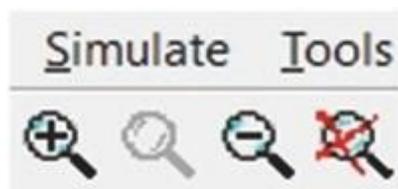


Fig. 6.76: Zoom icons

If you want to save the file with your desired name, you need to use the File> Save As (Fig. 6.77). After Save As the file with desired name, you can click the Save icon (Fig. 6.78) to save the changes you apply to the schematic file.

Fig. 6.77: File> Save As

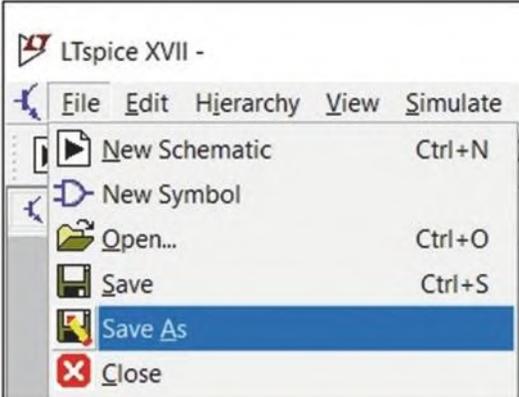
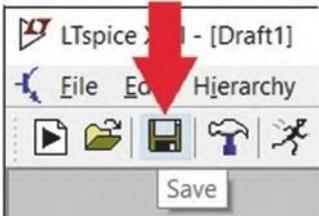


Fig. 6.78: Save icon



The Run icon (Fig. 6.79) can be used to determine the type of simulation and run the simulation. In this example, we want to find the steady-state DC voltages/currents. In order to do this, click the **Run** icon, (Fig. 6.79) and open the *DC op pnt* tab (Fig. 6.80).



Fig. 6.79: Run icon

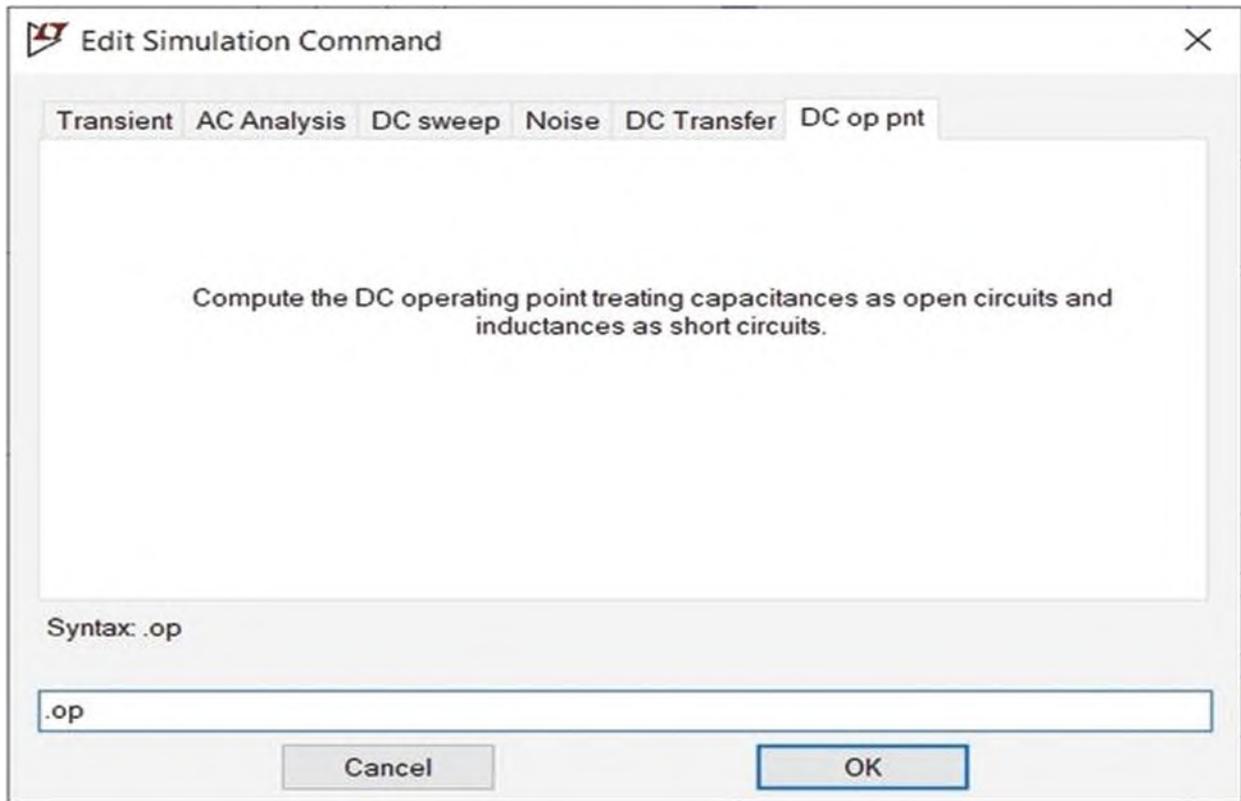


Fig. 6.80: DC op pnt tab

After clicking the **OK** button, the “.op” command is added to the schematic (Fig. 6.81) and the result shown in Fig. 6.82 appears. **Record the simulation data in Table 6.1.**

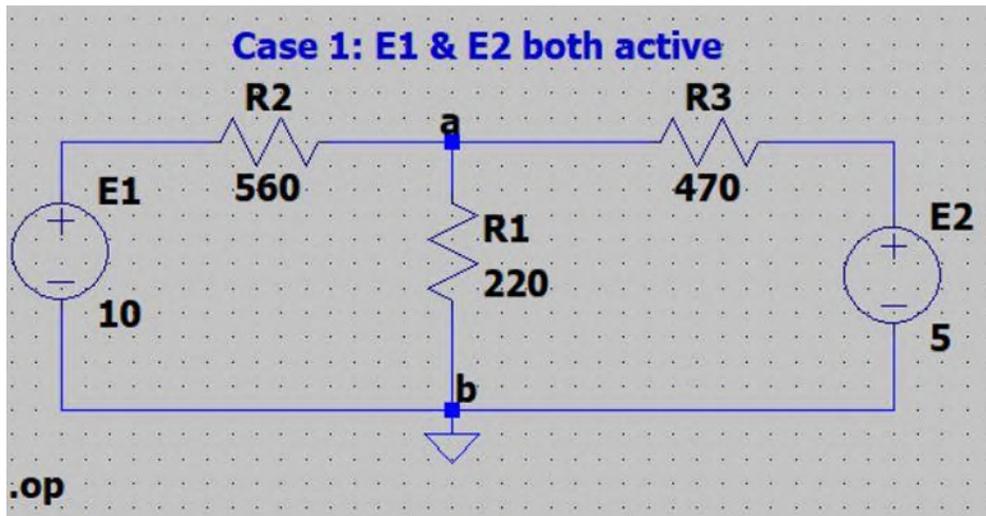


Fig. 6.81: “.op” command is added to the schematic

CASE 2: Make voltage source E₂ deactivate and measure the voltage V_{ab} and record the value

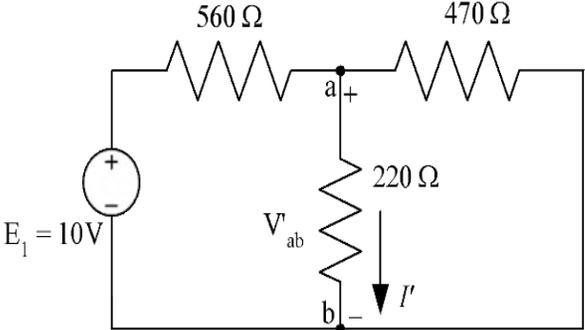


Fig. 6.83: Only E₁ activated

Draw the circuit in LTspice (Fig. 6.83).

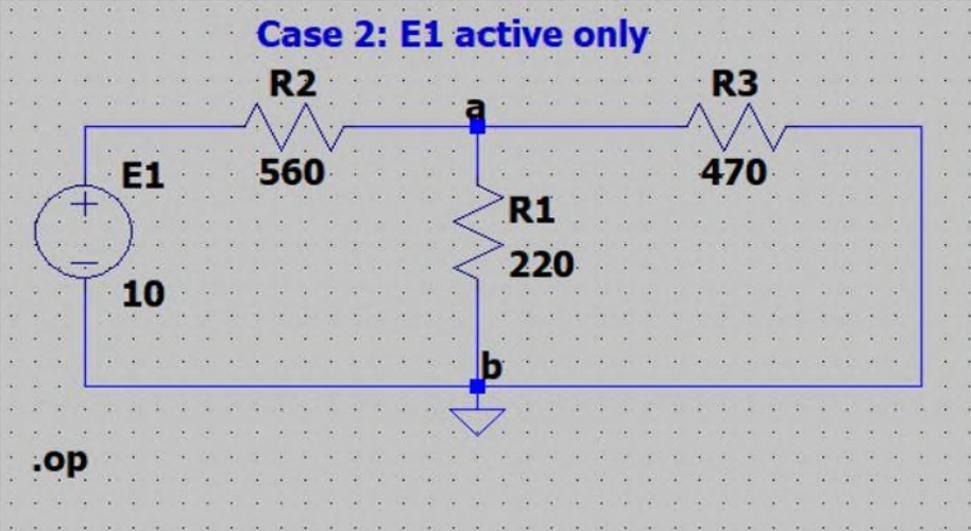


Fig. 6.83: LTspice execution of experimental circuit (case 2)

Click the **Run** icon, select *DC op pnt* tab, click **OK** and record the simulation data (Fig. 6.84) in **Table 6.1**.

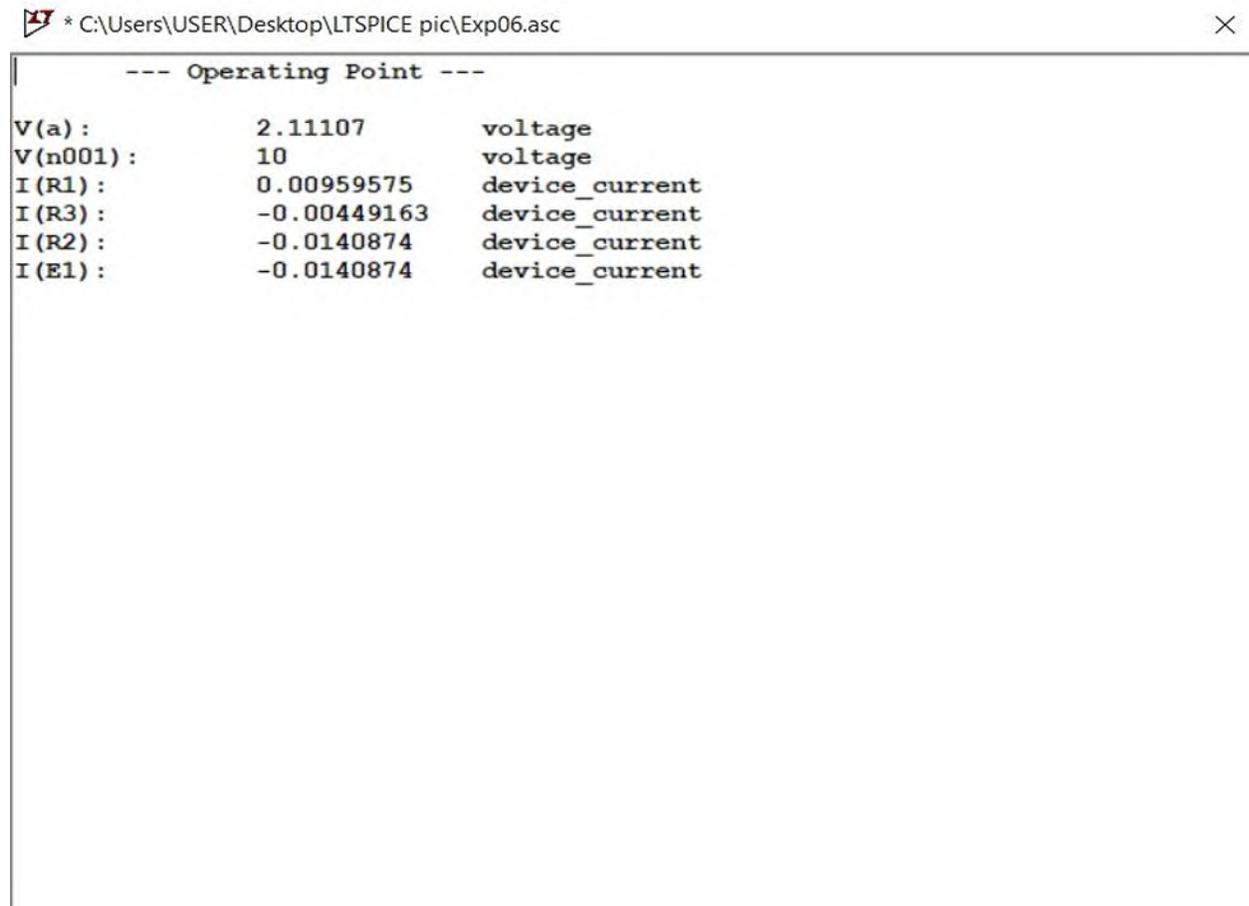


Fig. 6.84: Simulation result for case 2 (only E1 active)

CASE 3: Make voltage source E_1 deactivate and measure the voltage V''_{ab} and record the value

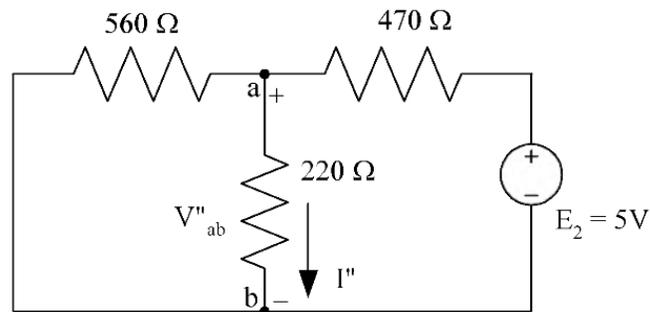


Fig. 6.85: Only E_2 activated

Draw the circuit in LTspice (Fig. 6.86).

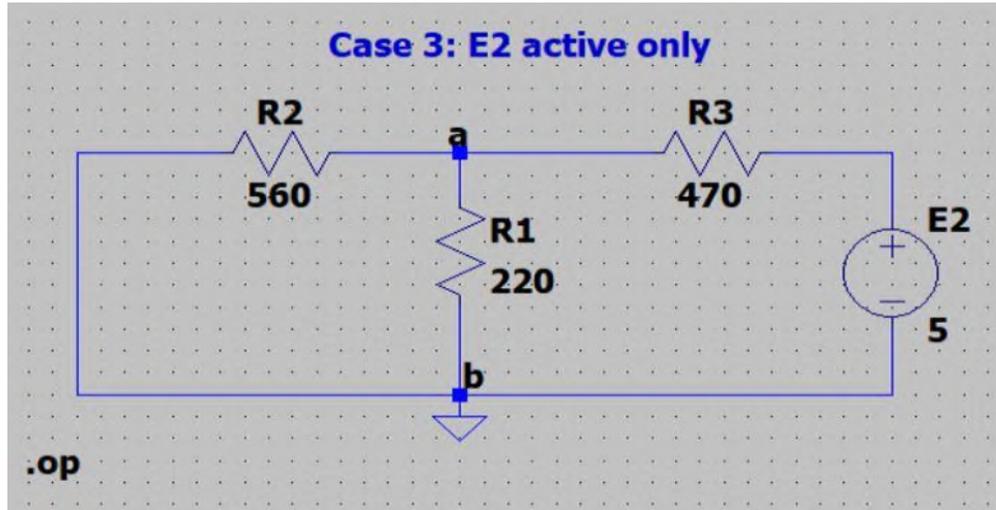


Fig. 6.86: LTspice execution of experimental circuit (case 2)

Click the **Run** icon, select *DC op pnt* tab, click 'OK' and record the simulation data (Fig. 6.87) in Table 6.1. Now verify superposition theorem.

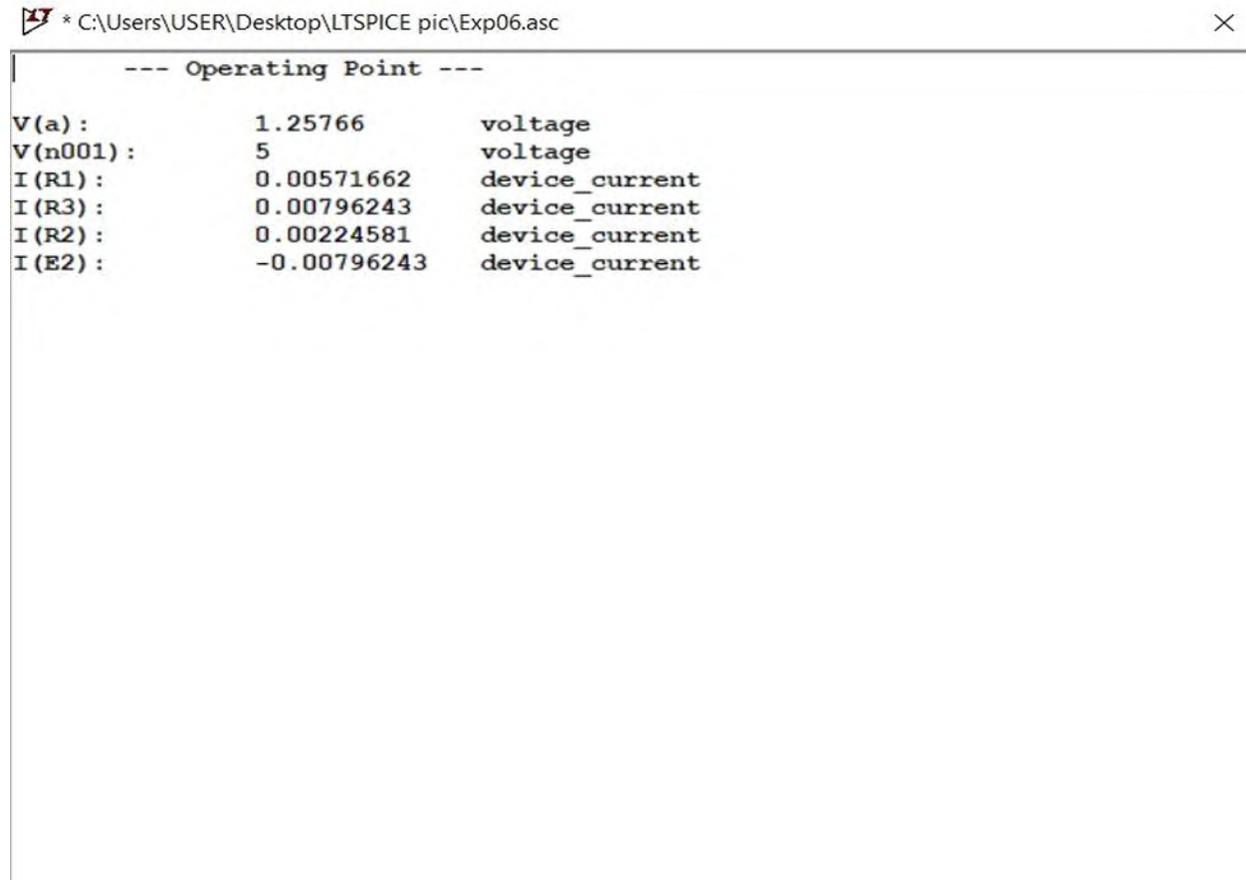


Fig. 6.87: Simulation result for case 3 (only E2 active)

Experiment 7:

Experiment name: Finding Equivalent resistance using simulation in LTSpice by Potentiometer method, Voc/Isc method and Vs/Is method.

Theory

In this experiment equivalent resistance with respect to two terminals is calculated through simulation. In potentiometer method a variable resistance is placed in between the two terminals and its resistance is varied until the voltage across the resistor is half the open circuit voltage between the terminals. In the Voc/Isc method the open circuit voltage and short circuit current are measured to find equivalent resistance. In the Vs/Is method a test voltage source is placed between the terminals by removing all other sources in the circuit and then the test voltage V_s and current through the source I_s is used to calculate equivalent resistance.

Potentiometer Method

The following process can be used to find equivalent resistance by the potentiometer method through simulation in LTSpice.

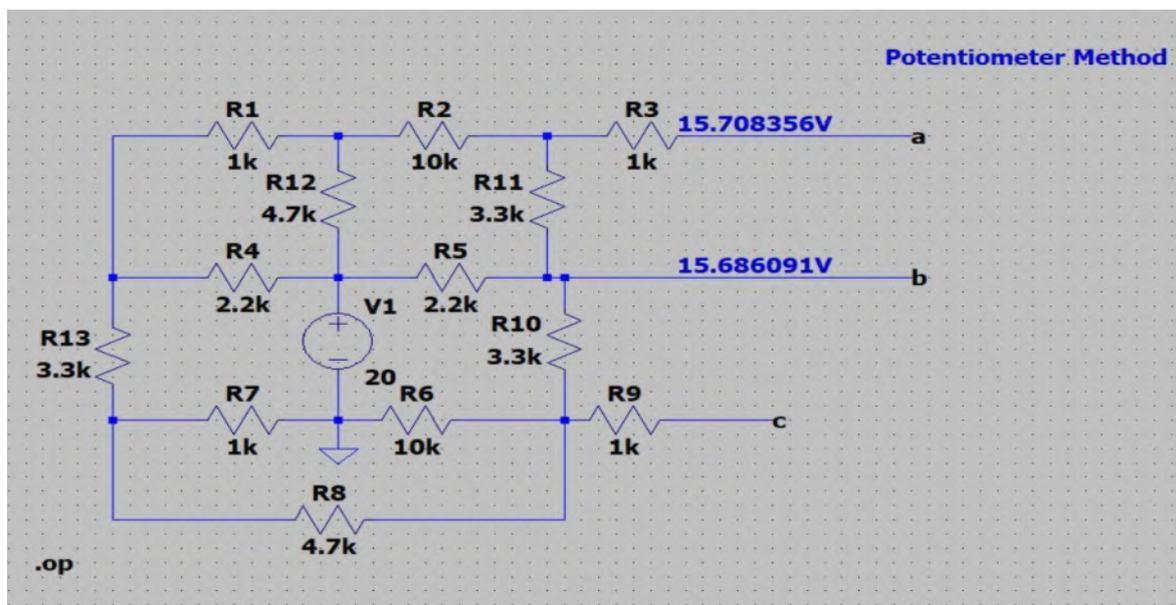


Figure : 1

- 1) Construct the circuit as shown in figure 1 in a new schematic.
- 2) Label the nodes across which the equivalent resistance is to be found as **a** and **b** using label net and perform DC operating point analysis .op to find the voltages of node a and b. Find the voltage difference between the nodes and divide it by 2.
- 3) Place a resistor (R14 in figure 2) between the nodes a and b as shown by figure 2.

- Right click on the resistor between a and b, and set its resistance parameter as {R}. (The curly brackets are needed but any name can be set)

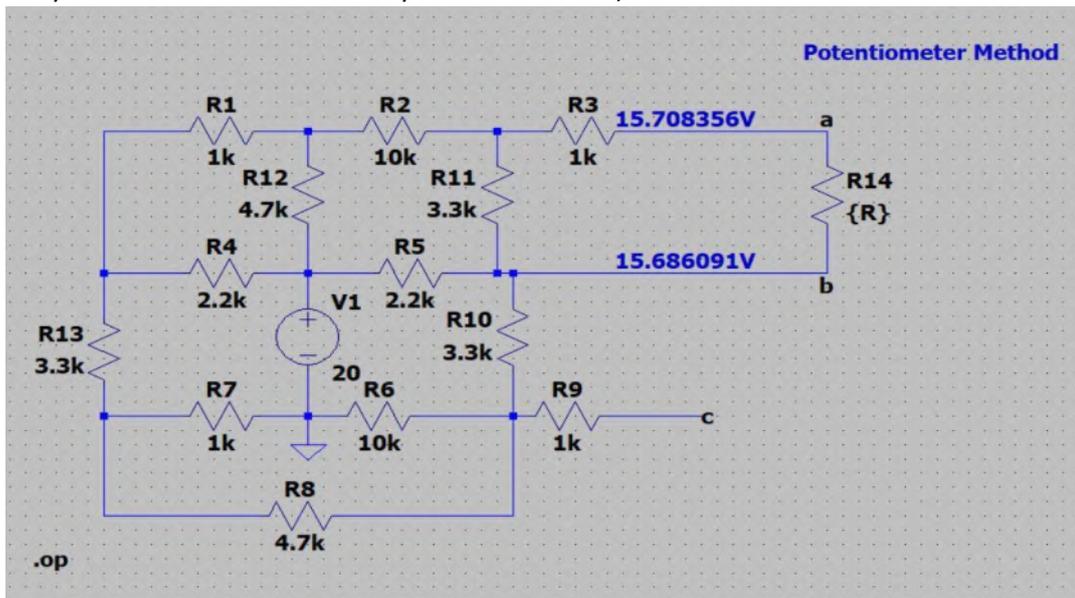


Figure: 2

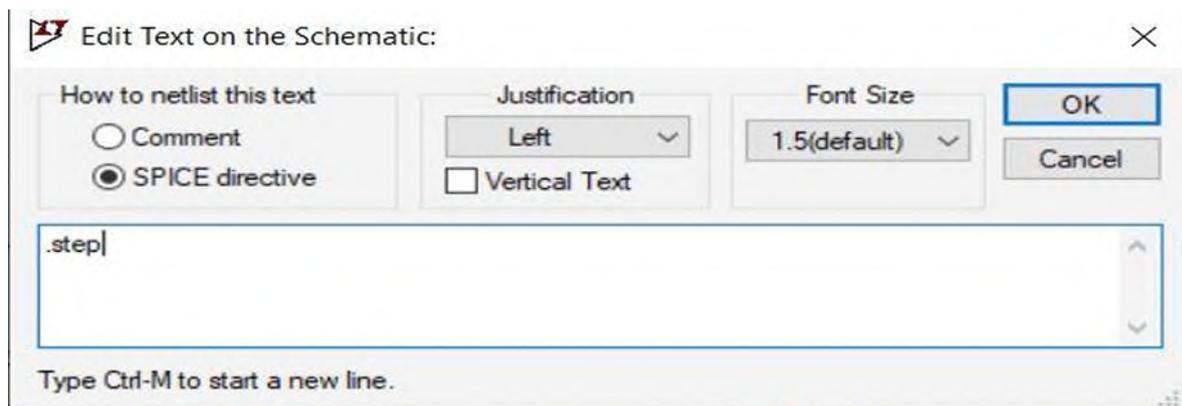


Figure: 3

- To set multiple values of {R} in the simulation click the SPICE Directive on the top right corner to open the Edit Text on the Schematic box as shown in figure 3.
- Write .step as shown in figure 3.
- Right Click the .step in the schematic shown in figure 4.

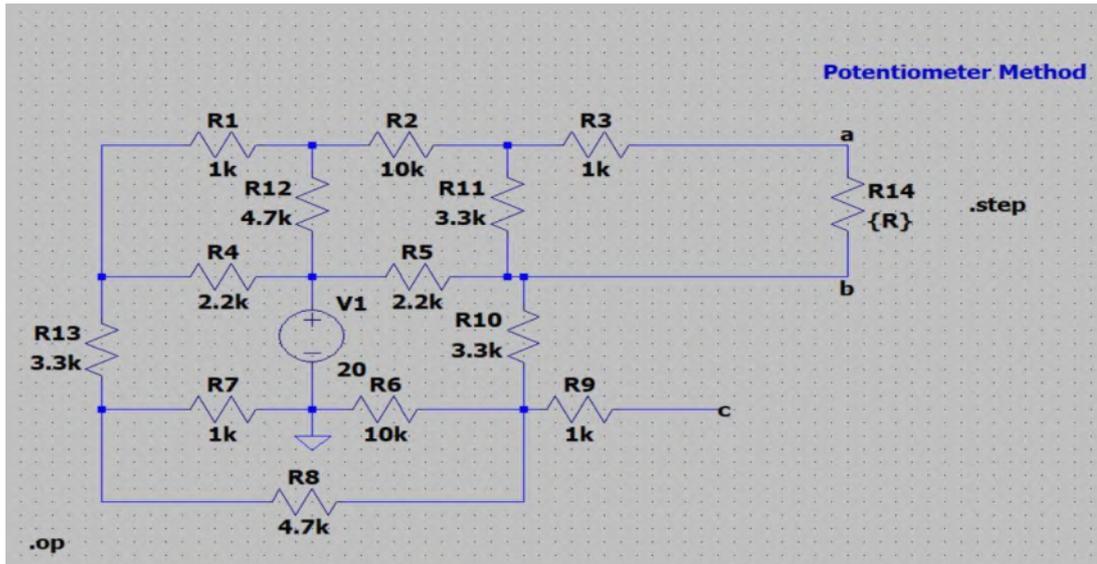


Figure: 4

- 8) The .step Statement Editor in figure 5 appears and set the parameters as shown in figure 5 to change the value of R from 1 to 10K with increment of 10 ohm.

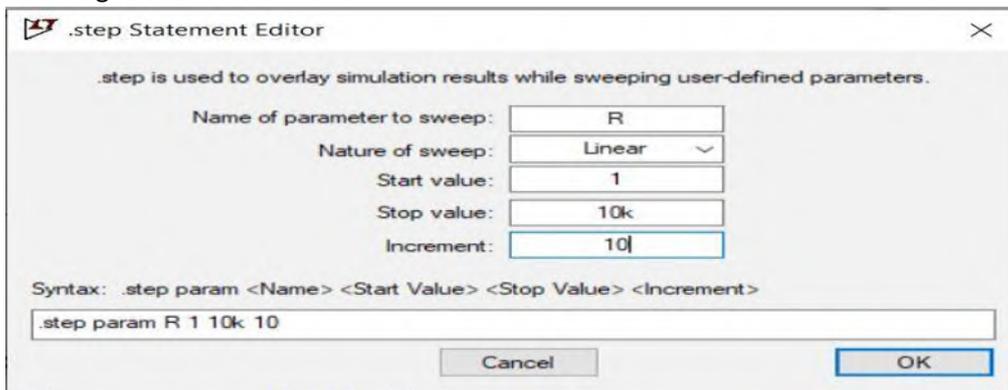


Figure: 5

- 9) Click Ok and run the simulation with the RUN button.
- 10) From the window shown in figure 6 place voltage markers at node a and b.
- 11) Then the trace on figure 7 appears

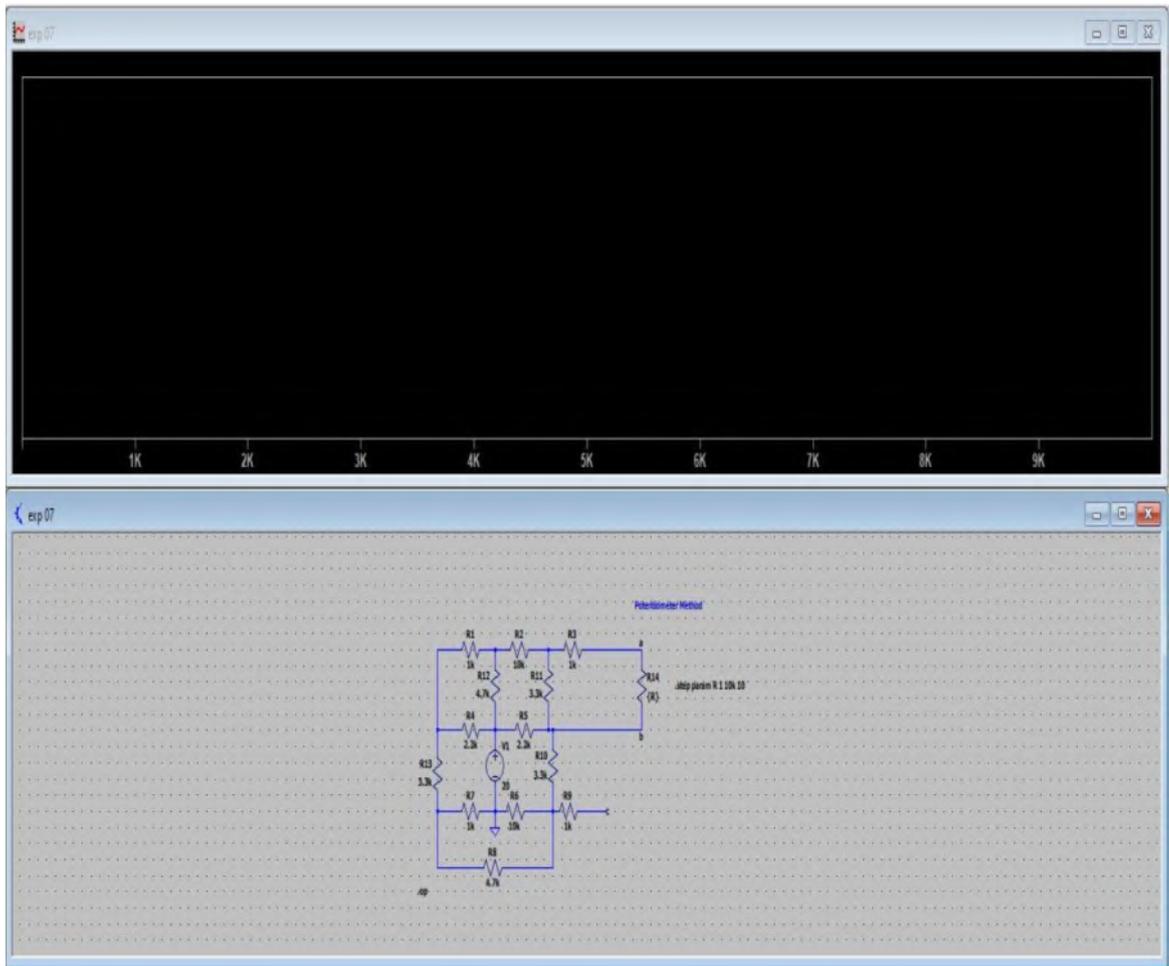


Figure: 6

- 12) Move the cursor on the trace (as shown in figure 7) to point 11.13 mV which is half of the open circuit voltage of node a and b found initially.
- 13) Then the corresponding resistance shown in figure 8 is the equivalent resistance of the circuit with respect to node a and b.

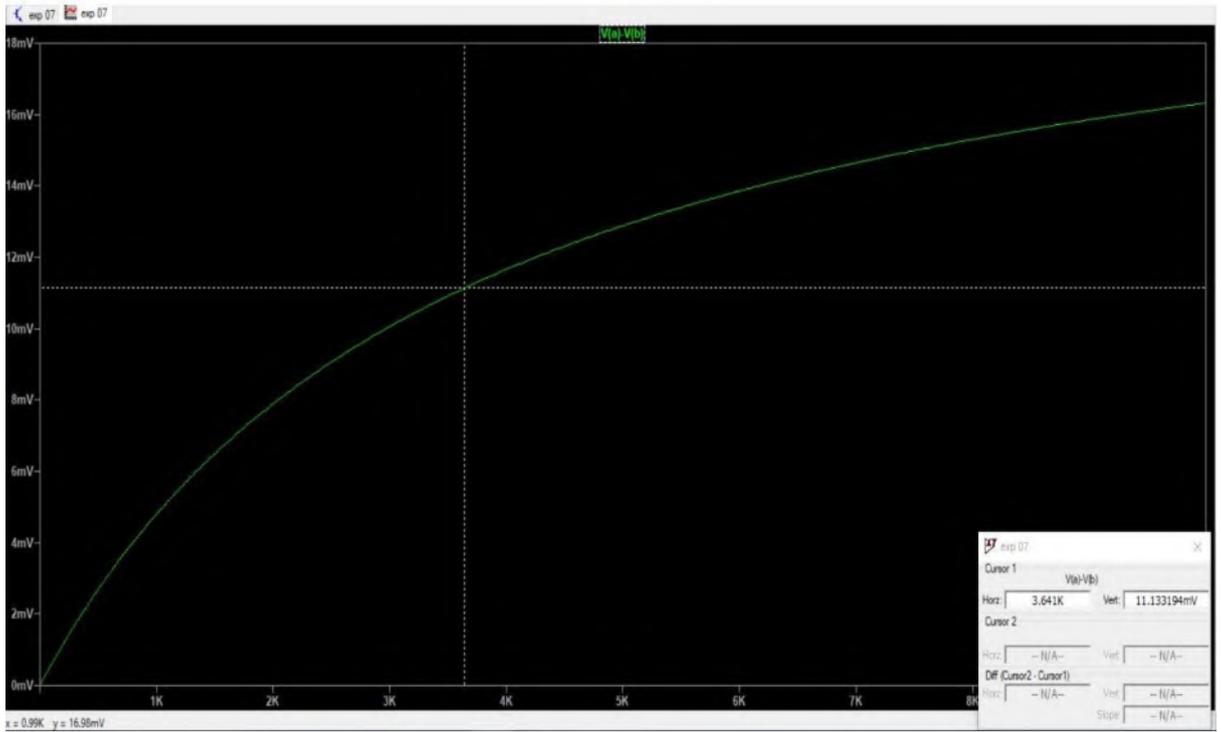


Figure: 7

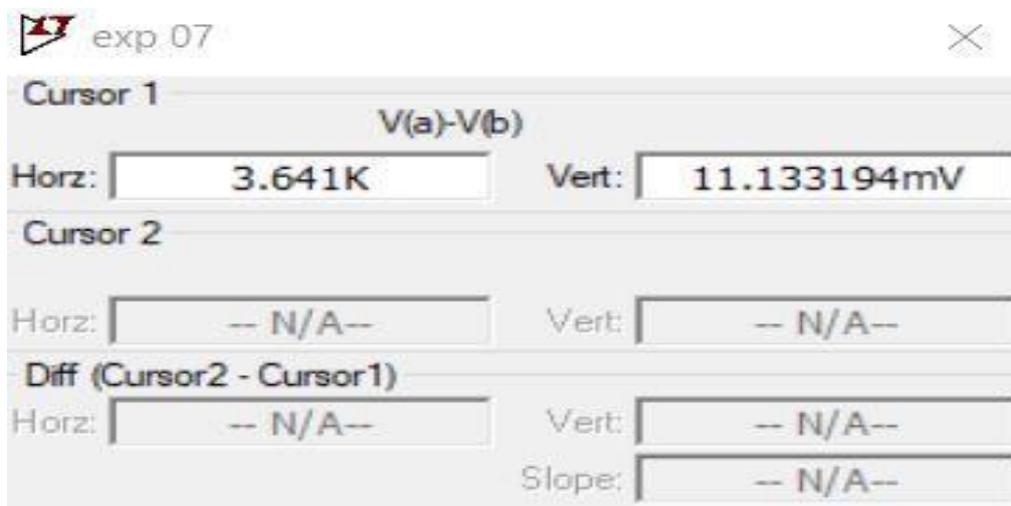


Figure: 8

Voc by Isc method

The following process can be used to find equivalent resistance by the Voc by Isc method through simulation in LTSpice.

- 1) Construct the circuit as shown in figure 9 in a new schematic.
- 2) Label the nodes across which the equivalent resistance is to be found as **a** and **b** using label net and perform DC operating point analysis .op to find the voltages of node a and b as shown in figure 10. The voltage difference between node a and b is the open circuit voltage.

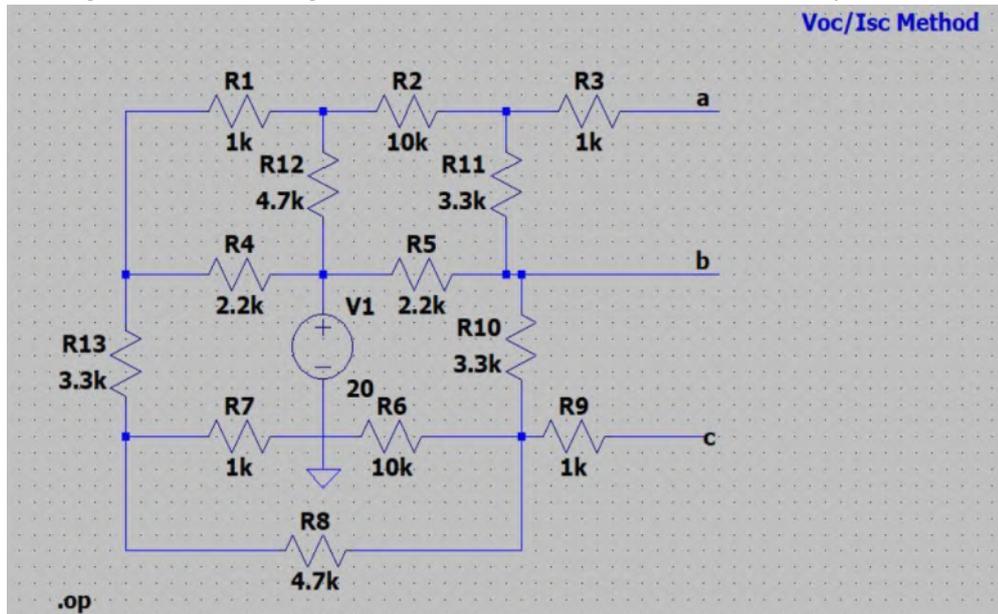


Figure: 9

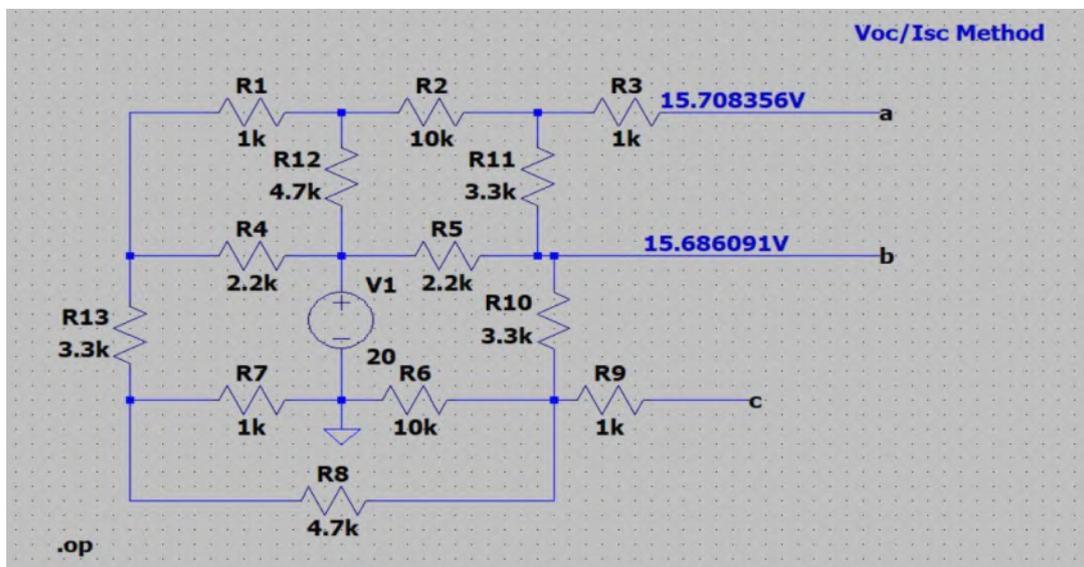


Figure: 10

- 3) Short the circuit between the nodes a and b as shown in figure 11 and perform DC Operating point analysis.
- 4) From the output window shown in figure 12 note the short circuit current through the resistor R3.
- 5) Then divide open circuit voltage by short circuit current to get the equivalent resistance.

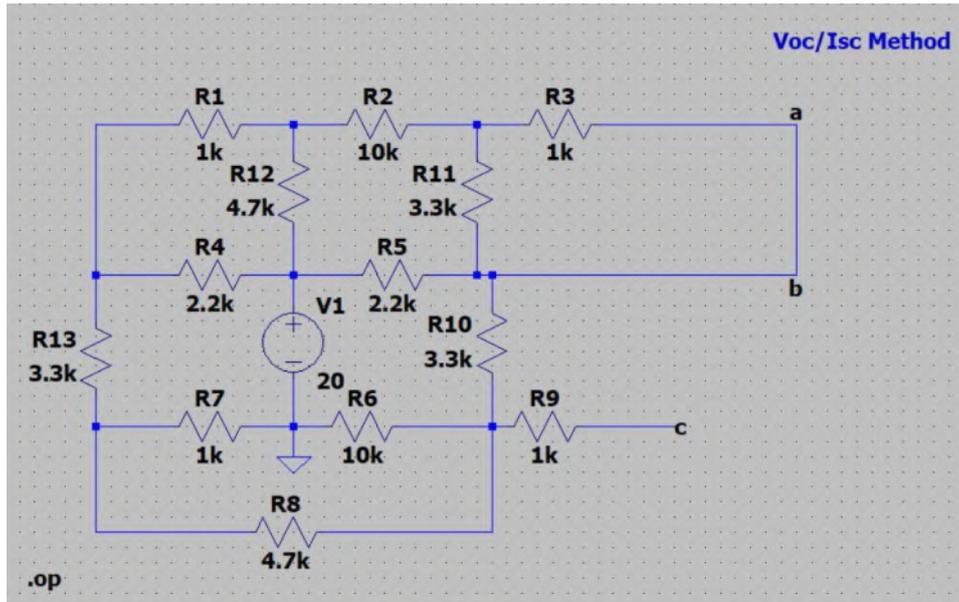


Figure: 11

* C:\Users\USER\Desktop\LTSPICE pic\exp 07.asc

--- Operating Point ---

V(n002) :	15.7739	voltage
V(n001) :	14.8827	voltage
V(n003) :	15.6942	voltage
V(b) :	15.6881	voltage
V(n004) :	20	voltage
V(n006) :	9.19392	voltage
V(n005) :	4.26579	voltage
V(c) :	9.19392	voltage
I(R1) :	0.000891201	device_current
I(R2) :	-7.96967e-06	device_current
I(R3) :	-6.11626e-06	device_current
I(R4) :	0.00232605	device_current
I(R5) :	-0.00195996	device_current
I(R6) :	0.000919392	device_current
I(R7) :	-0.00426579	device_current
I(R8) :	0.00104854	device_current
I(R9) :	0	device_current
I(R10) :	-0.00196793	device_current
I(R11) :	-1.85341e-06	device_current
I(R12) :	0.00089917	device_current
I(R13) :	-0.00321725	device_current
I(V1) :	-0.00518518	device_current

Figure: 12

Vs by Is method

The following process can be used to find equivalent resistance by the Vs by Is method through simulation in LTSpice.

- 1) Construct the circuit as shown in figure 13 in a new schematic.
- 2) Set the name of the node a and b and perform the DC operating point analysis .op.
- 3) From the output window shown in figure 14 and figure 15 find the Vs and Is in the circuit.
- 4) Then Vs divided by Is gives the equivalent resistance with respect to node a and b.

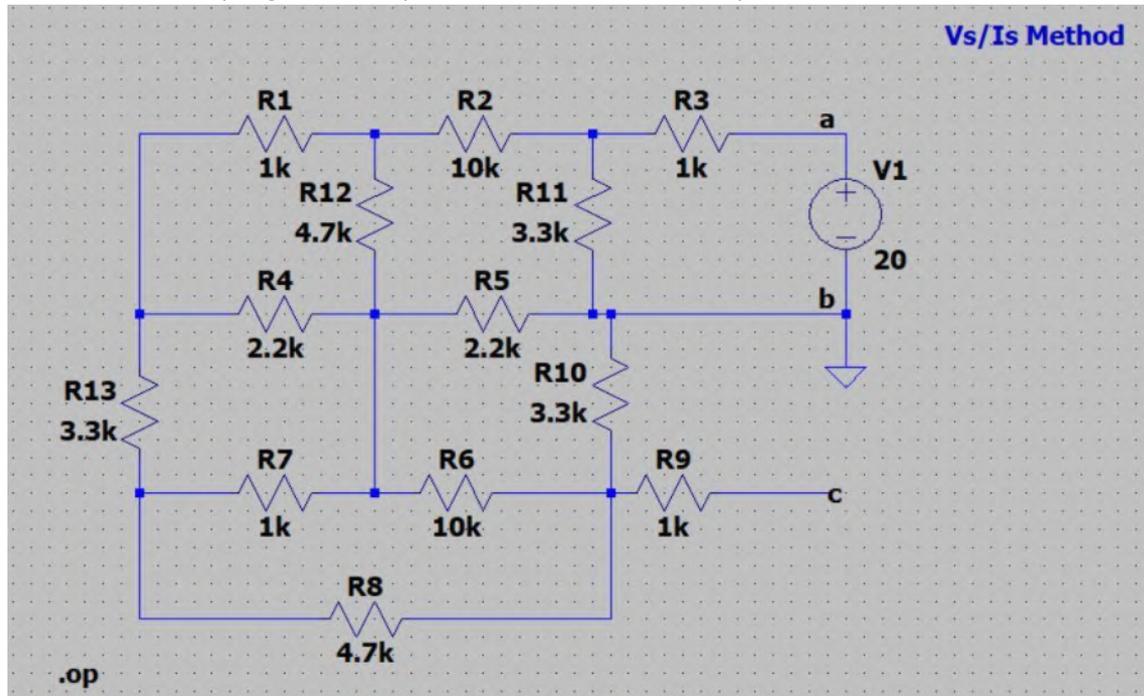


Figure: 13

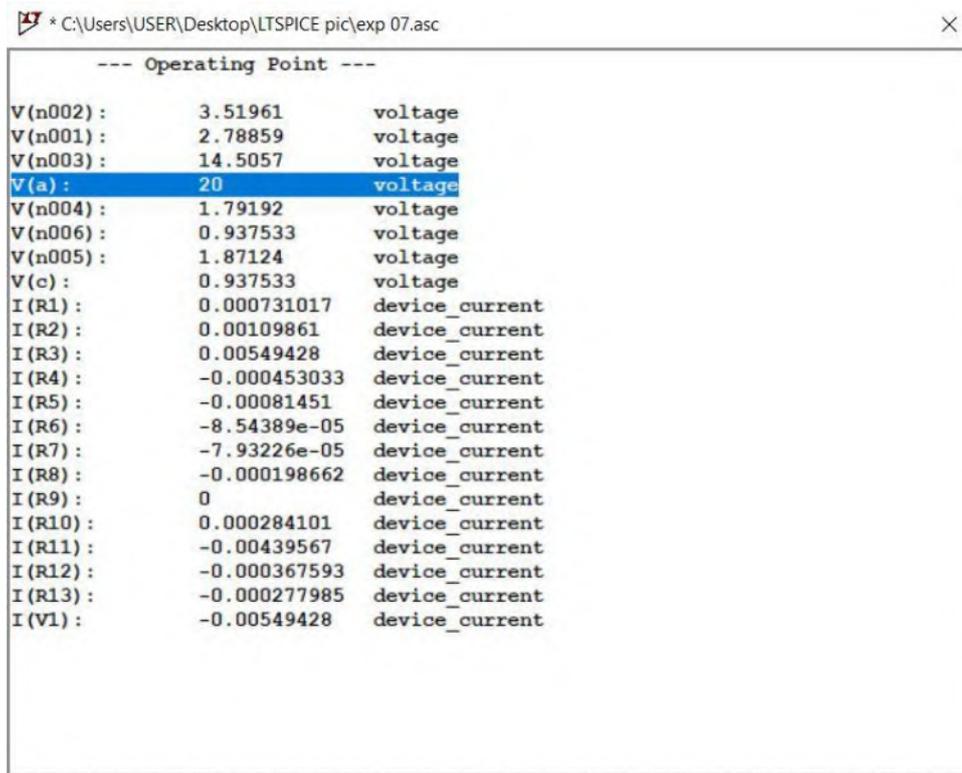


Figure: 14

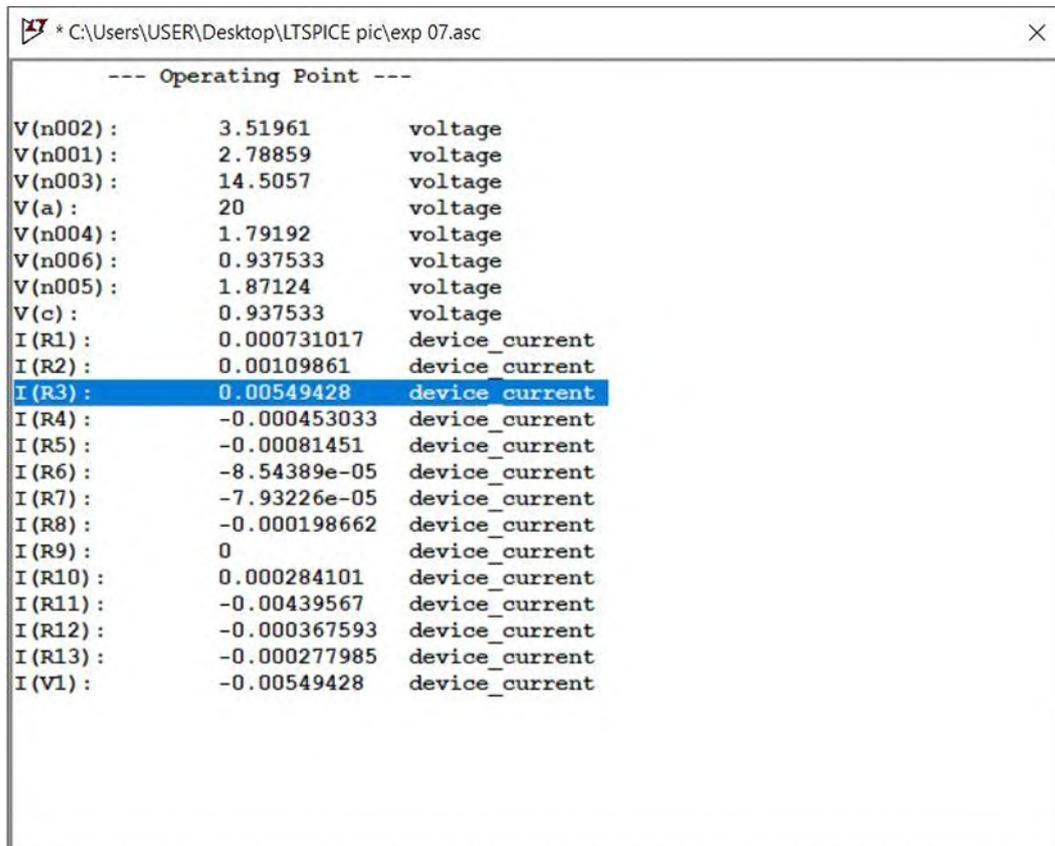


Figure: 15

AHSANULLAH UNIVERSITY OF SCIENCE & TECHNOLOGY
DEPARTMENT OF ELECTRICAL & ELECTRONIC ENGINEERING

EEE -1 1 O2: ELECTRICAL CIRCUITS – I LAB.

Experiment No. : 08
Name of the Experiment : Verification of Thevenin's Theorem and
Maximum Power Transfer Theorem.

OBJECTIVE:

To verify and interpret two most important theorems:

- (1) Thevenin's Theorem
- (2) Maximum Power Transfer Theorem.

THEORY:

Thevenin's Theorem states that the voltage across or current through element of a network can be calculated by constructing a Thevenin's equivalent circuit. This circuit is represented by a voltage source, called Thevenin's voltage, a resistance, called Thevenin's resistance and the element itself – all connected in series. After removing the element Thevenin's voltage and Thevenin's resistance are calculated, where the open circuit voltage (V_{oc}) measures Thevenin's voltage (V_{th}) and the resistance looking through the open circuited terminals represents Thevenin's resistance (R_{th}).

The maximum power transfer theorem states that maximum power can be transferred to the load when the load resistance is equal to equivalent resistance looking through the terminals where the load is connected.

EQUIPMENTS:

- Variable DC Power Supply (20 V)-1 piece.
- Trainer Board-1 piece.
- Digital Multimeter-1 piece.
- Resistances: 220 Ω -4 piece.
- Rheostat (1k) - 2 piece.
- Connecting Wires.

THEVENIN'S THEOREM

CIRCUIT DIAGRAM:

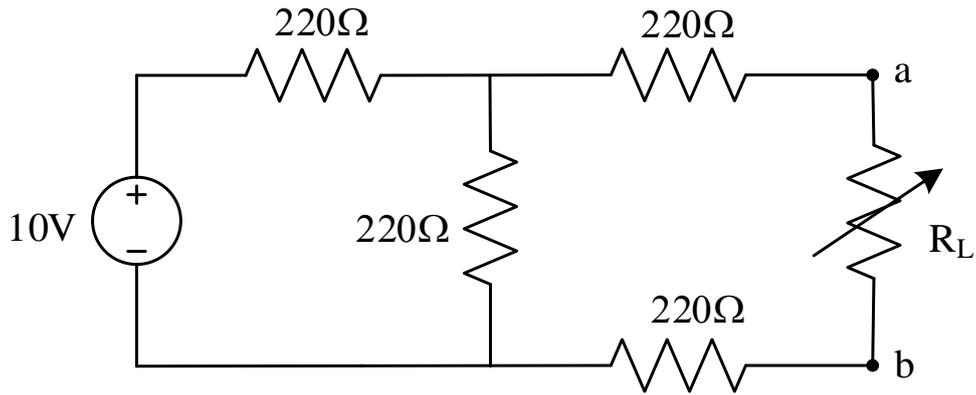


Figure 8.1: Circuit 01

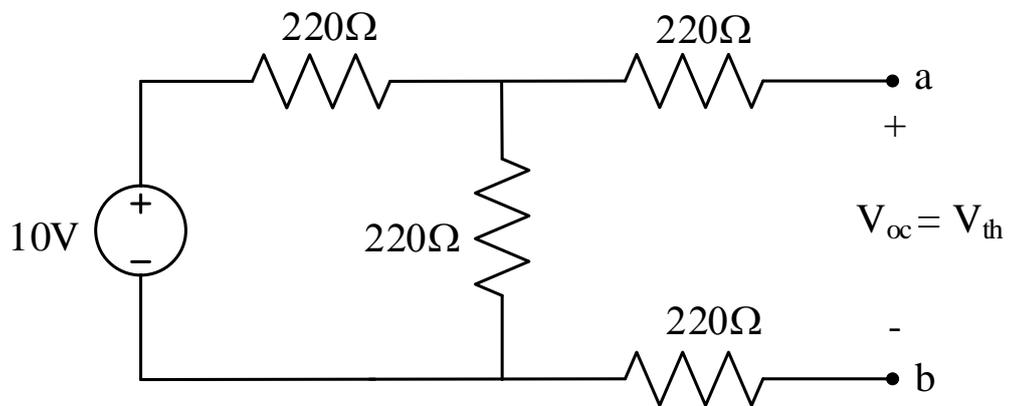


Figure 8.2: Circuit setup for finding V_{oc} or V_{th}

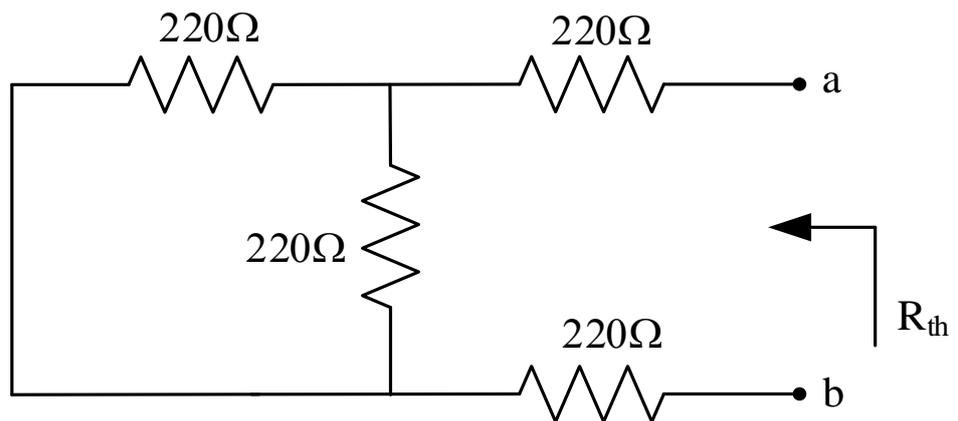


Figure 8.3: Circuit setup for finding R_{th}

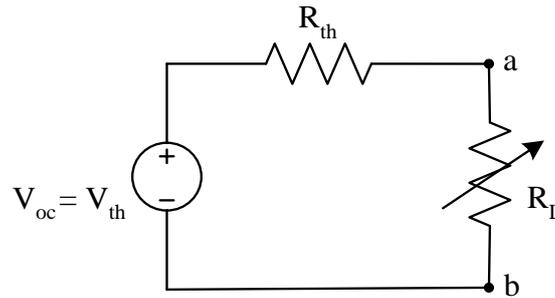


Figure 8.4: Thevenin's equivalent circuit

THEVENIN'S THEOREM

PROCEDURES:

1. Connect the circuit as shown in figure-8.1. Remove the load resistance (rheostat) as shown in figure-8.2. Measure the open circuited voltage, (V_{oc}). This represents Thevenin's voltage (V_{th}).
2. Now construct the circuit as shown in figure-8.3. Make voltage source deactivate (Remove the source and short the terminals). To measure the resistance R_{th} (Thevenin's equivalent resistance looking through terminals a-b), hold the ohmmeter at terminals a and b.
3. Finally construct the Thevenin's equivalent circuit as shown in figure-8.4.
4. Connect the circuit as shown in figure-8.1. Vary the load resistance (rheostat) from $100\ \Omega$ to $1K\ \Omega$ with a $100\ \Omega$ stepping. For each step measure voltage across the load resistance (rheostat) and calculate the current (I_L) through the load. Fill the table-8.1 using these values.
5. Now connect the same load resistance (rheostat) at the Thevenin's equivalent circuit as shown in figure-8.4 and vary the load resistance (rheostat) from $100\ \Omega$ to $1K\ \Omega$ with a $100\ \Omega$ stepping. For each step measure voltage across the load resistance (rheostat) and calculate the current I_L . Fill the table-8.2 using these values.

ASSIGNMENTS:

1. Plot V_L vs. R_L curves for both original and equivalent circuits on the same graph.
2. Plot I_L vs. R_L curves for both original and equivalent circuits on the same graph.
3. Plot P_L vs. R_L curves for both original and equivalent circuits on the same graph.
4. Comparing the graphs verify Thevenin's Theorem.
5. From P_L vs. R_L graphs verify maximum power transfer theorem.

THEVENIN'S THEOREM

DATA SHEET:

****[Use measured values of resistances for all calculations.]**

Table 8.1: For original circuit

R_L (Ω)	V_L (Volt)	$I_L = \frac{V_L}{R_L}$ (A)	$P_L = \frac{(V_L)^2}{R_L}$ (W)	$P_L = (I_L)^2 \times R_L$ (W)
100				
200				
300				
400				
500				
$R_{th} = R_L =$				
600				
700				
800				
900				
1000				

SAMPLE CALCULATION:

Signature of the Teacher
THEVENIN'S THEOREM

DATA SHEET:

****[Use measured values of resistances for all calculations.]**

Table 8.1: For Thevenin's equivalent circuit:

R_L (Ω)	V_L (Volt)	$I_L = \frac{V_L}{R_L}$ (A)	$P_L = \frac{(V_L)^2}{R_L}$ (W)	$P_L = (I_L)^2 \times R_L$ (W)
100				
200				
300				
400				
500				
$R_{th} = R_L =$				
600				
700				
800				
900				
1000				

SAMPLE CALCULATION:

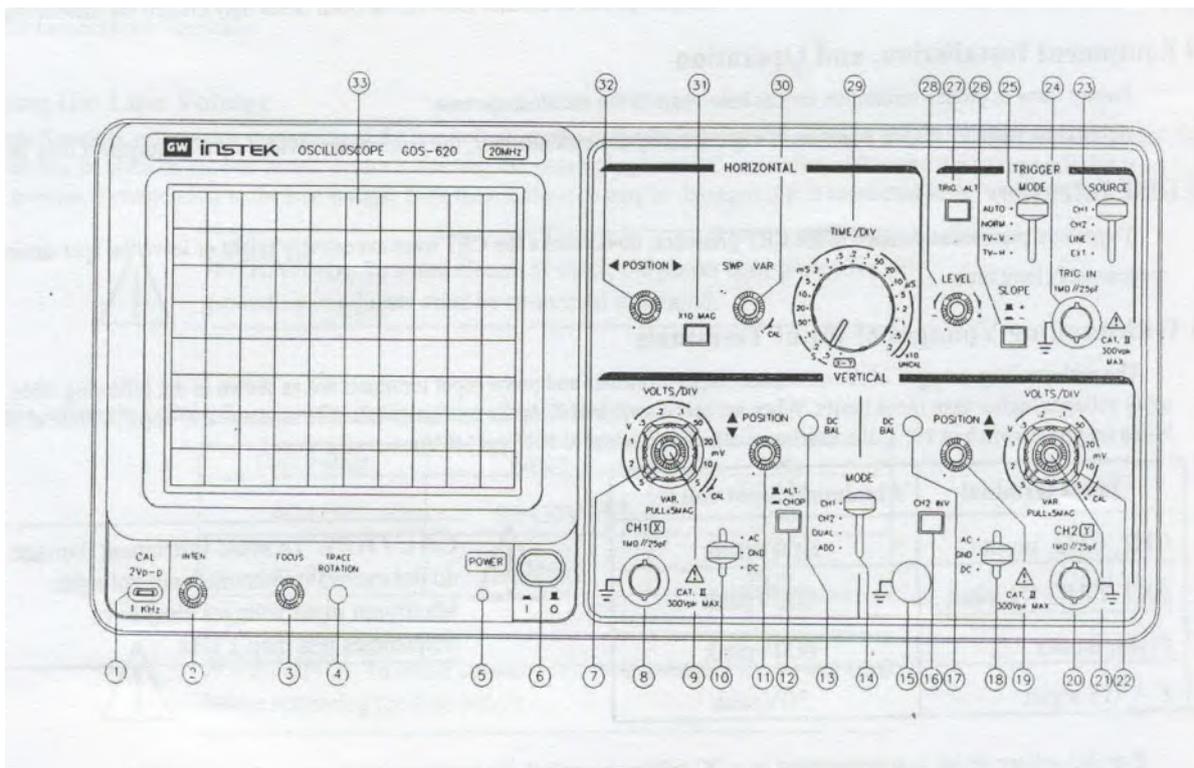
 Signature of the Teacher

Experiment No. : 09
Name of the Experiment : Introduction to Oscilloscope Operation.

OBJECTIVE:

This experiment is designed for the under-graduate students to introduce themselves and to be familiar with the oscilloscope and its operation. This experiment will help the students to have basic ideas about key functions of different knobs of oscilloscope and also to know how to measure voltage/current of a circuit-using oscilloscope as a measuring instrument.

FRONT VIEW OF THE LABORATORY OSCILLOSCOPE:



Introduction to front Panel:

The front panel consists of the following parts:

- CRT
- Vertical axis
- Triggering
- Time
- Others

BRIEF DESCRIPTION:

1. CRT:

- a) Power (6)
The main power switch.
- b) Inten (2)
Controls the brightness of the spot.
- c) Focus (3)
For focusing the spot for sharp image.
- d) Trace rotation (4)
For aligning the horizontal trace in parallel with graticule lines.
- e) Filter (33)

2. Vertical axis:

- a) CH1 (X) input (8)
Vertical input of CH1. When in X-Y mode this acts as an X-axis input
- b) CH2 (Y) input (20)
Vertical input of CH2. When in X-Y mode this acts as a Y-axis input
- c) AC-GND-DC (10,18)
Switch for selecting connection mode between input signal and vertical amplifier.
- d) Volt/Div (7,22)
Selection of vertical axis sensitivity, from 5mV/Div to 5V/Div in 10 ranges.
- e) Variable (9,21)
- f) CH1 & CH2 DC BAL (13,17)
- g) Position (11,19)
Control the position of the vertical trace or spot.
- h) Vert mode (14)
There are four positions to switch the operation of CH1 and CH2. When position in either CH1 or CH2; then oscilloscope operates as single channel instrument with CH1 or CH2 respectively. When position in DUAL then the oscilloscope operates as dual-channel of both CH1 and CH2. When position in ADD, then oscilloscope displays the algebraic sum (CH1+CH2) or difference (CH1-CH2). During difference operation, CH2 INV must be pushed.
- i) ALT/CHOP (12)
When this switch is released then CH1 and CH2 are alternately displayed. When this switch is engaged then CH1 and CH2 are chopped and displayed simultaneously.
- j) CH2 INV (16)
This inverts the CH2 input signal when this knob is pushed in.

3. Triggering:

- a) EXT TRIG IN input terminal (24)
- b) SOURCE (23)
 - 1) **CH1:** When Vert mode switch is at DUAL/ADD position select CH1 for internal triggering.
 - 2) **CH2:** When Vert mode switch is at DUAL/ADD position select CH2 for internal triggering.
 - 3) **TRIG.ALT:** It will alternately select CH1 and CH2 for internal triggering.
 - 4) **Line**
 - 5) **EXT**
- c) SLOPE (26)
 - 1) **'+':** Triggering occurs when triggering signal crosses triggering level in +ve going direction.
 - 2) **'-':** Triggering occurs when triggering signal crosses triggering level in -ve going direction.
- d) LEVEL (28)

To display synchronized stationary waveform and set a start point of it.
- e) TRIGGER MODE (25)

4. Time Base:

- a) TIME/DIV (29)

Ranges are available from 0.2 μ sec/div to 0.5 sec/div in 20 steps.

X-Y mode: This position is used when oscilloscope functions as an X-Y oscilloscope.
- b) SWP.VAR (30)
- c) Position (32)

Control the position of the horizontal trace or spot.
- d) x 10 MAG (31)

When this button is pushed, magnification of 10 occurs.

5. Others:

- a) CAL (1)

This terminal gives the calibration voltage of 2 Vp-p, 1 kHz, and positive square wave.
- b) GND (15)

The ground terminal of the oscilloscope mainframe.

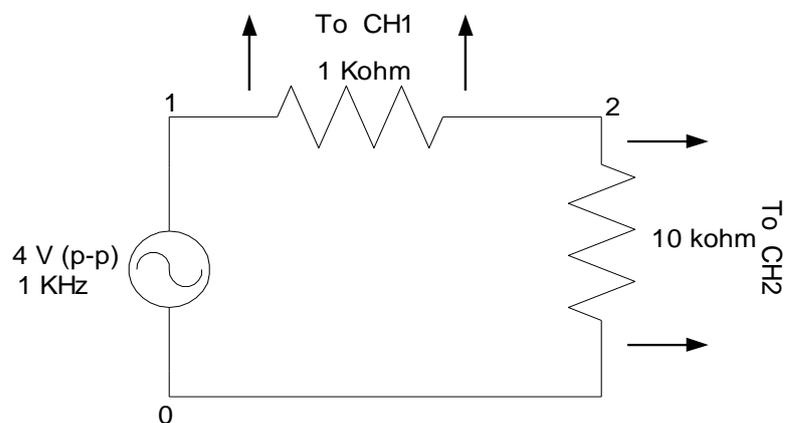
BASIC OPERATION WITH OSCILLOSCOPE:

1. Single channel operation.
2. Dual-channel operation.
3. ADD operation.
4. Frequency measurement
5. Sweep Magnification
6. X-Y Operation.
7. To display two input signals still on oscilloscope

EQUIPMENTS:

1. Oscilloscope - 1 unit
2. Oscilloscope probe (10 x)- 2 pieces
3. Signal Generator - 1 unit
4. Signal Generator probe - 1 piece
5. Resistor 1k Ω , 10k Ω
6. Bread Board - 1 piece.
7. Multi-meter

CIRCUIT DIAGRAM:



PROCEDURES:

1. Connect the circuit according to the above circuit diagram.
2. Set the AC-GND-DC of CH1 in the GND position and align the trace with horizontal central line and then set to AC position.
3. Now apply sine wave of 1 kHz from signal generator to CH1 and adjust its magnitude to 4V (p-p) by varying the attenuator knob of the signal generator.
4. If the signal is not still just slowly vary the 'level' knob to make it still.
5. Now disconnect the signal from CH1 and apply to terminals between 1 and 0.

6. Don't change the attenuator knob throughout the experiment.
7. Now connect the oscilloscope probes across 1 k Ω resistor to CH1 and across 10 k Ω resistors to CH2 according to circuit diagram.
8. Push CH2 INV button.

1. Single channel operation:

- a) Set the AC-GND-DC of both channels in the GND position and align the trace with horizontal central line and then set to AC position.
- b) Adjust the FOCUS control so that the trace image appears sharply.
- c) Set the VOLTS/DIV switch at 1 V and TIME/DIV switch at 0.5 ms position so that signal waveform is displayed clearly.
- d) Adjust vertical POSITION and horizontal POSITION controls in appropriate position so that the displayed waveform is aligned with the graticule and voltage (p-p) and period (T) can be read conveniently.
- e) Set the Vert mode to CH1 and measure the p-p voltage across 1 k Ω resistor. Find the rms value of the signal from the following diagram:
$$V_{rms} = V_{measured} (p-p) / (2\sqrt{2}) \text{ volt.}$$
- f) Measure the voltage across 1 k Ω resistor by multimeter and compare with the measured value.
- g) Set the Vert mode to CH2 and repeat procedure (e).
- h) Change the signal frequency to 100Hz, 10 kHz and observe the waveform.

2. Dual-channel operation:

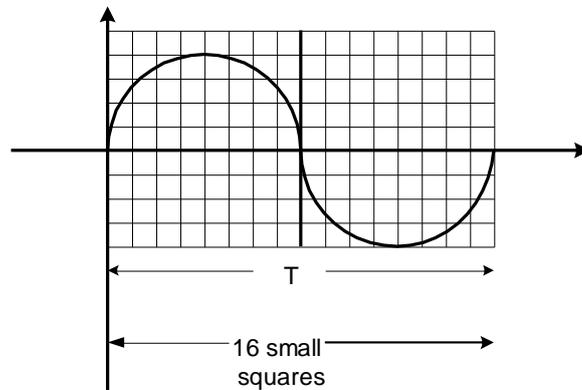
- a) Set the Vert mode switch to DUAL state so that both channels are displayed simultaneously. To display each channel separately change the vertical POSITION control of both channel to convenient position.
- b) When ALT/CHOP switch is released (ALT mode) signals respectively to CH1 and CH2 appear on screen alternately.
- c) When ALT/CHOP switch is pushed (CHOP mode) signals respectively to CH1 and CH2 are switched at 250 kHz.

3. ADD operation:

- a) When Vert mode switch is at ADD position then the displayed signal is the algebraic sum of CH1 and CH2. If the CH2 INV switch is pushed then displayed signal is the difference of CH1 and CH2.
- b) Observe the waveform for both cases and draw.

4. Frequency measurement:

- a) The frequency of any waveform can be measured by adjusting the TIME/DIV control knob of oscilloscope. Adjust the TIME/DIV control knob to position 0.5 ms to observe the waveform.
- b) Now measure the frequency of the wave using the following formula:
 1 large square or 5 small squares = t sec, here t = 0.5 ms
 # of small squares required to represent a full cycle of wave = n sec
 Where, n may have fraction value.
 Time period, $T = (n/5) \times t$ s
 Frequency, $f = 1/T$ Hz



Now compare this value with the main signal frequency.

- c) Now vary the TIME/DIV control knob to different position and repeat (b).

5. Sweep Magnification:

- a) Set the TIME/DIV switch at 0.5 ms and VOLTS/DIV at 1 V. Set AC-GND-DC position at GND position and align the trace with horizontal central line.
- b) Set the Vert mode at CH1 and AC-GND-DC at AC and then push x 10 MAG button.
- c) The displayed waveform will be expanded 10 times to the right and left with the centre of the screen as the centre of expansion.

6. X-Y operation:

- a) Set the TIME/DIV switch at X-Y position. Now CH1 acts as X-axis input and CH2 as Y-axis input.
- b) X-Y positions are adjusted by horizontal position and CH2 vertical position control respectively.
- c) Adjusted the amount of vertical Y-axis with CH2 VOLTS/DIV controls.

- d) Adjust the amount of horizontal X-axis with CH1 VOLTS/DIV controls.
- e) Observe the waveforms and draw.

7. To display two input signals still on oscilloscope:

- a) Apply 2v (p-p), 1 kHz ac signal (sine wave) from signal generator-1 to CH1 of oscilloscope.
- b) Set VOLT/DIV of CH1 to 1 V and TIME/DIV to convenient position to observe the wave shape clearly.
- c) Repeat (1) from signal generator-2 to CH2 of oscilloscope.
- d) Set VOLT/DIV of CH2 to 0.5 v.
- e) Now set the Vert mode to DUAL position and observe the wave shapes.
- f) Note two waves that were displayed individually still are not still now.
- g) Set the SOURCE to CH1 and then CH2 position and observe what happens.
- h) Push the TRIG. ALT button and observe the wave shapes.

Note:

- 1) Don't put any sort of electrical equipments (such as signal generator, dc supply etc.) on the top of the oscilloscope.
- 2) Place the oscilloscope away from any magnetic field (as far as possible)
- 3) If the GND horizontal line deviates significant amount then adjust it by rotating the position of oscilloscope.
- 4) Always check the 10x switch of probe according to your measurement.
- 5) Always avoid common grounding resulting from improper connection of knobs in the circuit.

COMMON MISTAKES USING OSCILLOSCOPE:

- 1) Changing the calibration knob during experiment.
- 2) Changing the SWP.VAR knob during experiment.
- 3) Improper 10x max for probe.
- 4) Improper 10x max for frequency.
- 5) Ground level adjustment during measurement.