Brac University

Department of Electrical & Electronic Engineering

Semester Summer-24

Course Number: EEE101L

Course Title: Electrical Circuits I Laboratory

Section: 06

Lab Report

Experiment no. 02

Name of the experiment: Introduction to Series-Parallel Circuits (Software Simulation)

Prepared by:

Name: Tanzeel Ahmed ID: 24321367

Group Number: **02**

Other Group members:

Sl.	***ID***	***Name***
1.	24121083	Abontika Das
2.	24121046	Lamea Akter Shimla
3.	24121238	Tahosin Tasnia Nuha

Electrical Circuits I Laboratory

EEE 101L

Department of Electrical & Electronic Engineering (EEE)

Brac University

Experiment No. 2

Introduction to Series-Parallel Circuits

1. **Objective:** The experiment is to acquaint the students with series-parallel circuits and to give them the idea about how to connect different circuits in bread board.

2. **Theoretical Background:** An electrical circuit is a continuous path or array of paths through which an electrical current can flow. The two different ways in which components of a circuit can be connected are called "series" and "parallel". In a series connection, components are connected one after another; therefore, the same current flows through all of them. In a parallel connection, the circuit components are connected side by side. That is, the positive and negative sides of each component are respectively connected together; therefore, each has the same potential drop across. In this lab, we will explore measurements of current and potential difference in simple circuits.

3. **Equipment:**

    - DC power supplies
    - Resistors
    - Bread board/ Trainer board
    - One multimeter

4. Circuit Diagram:

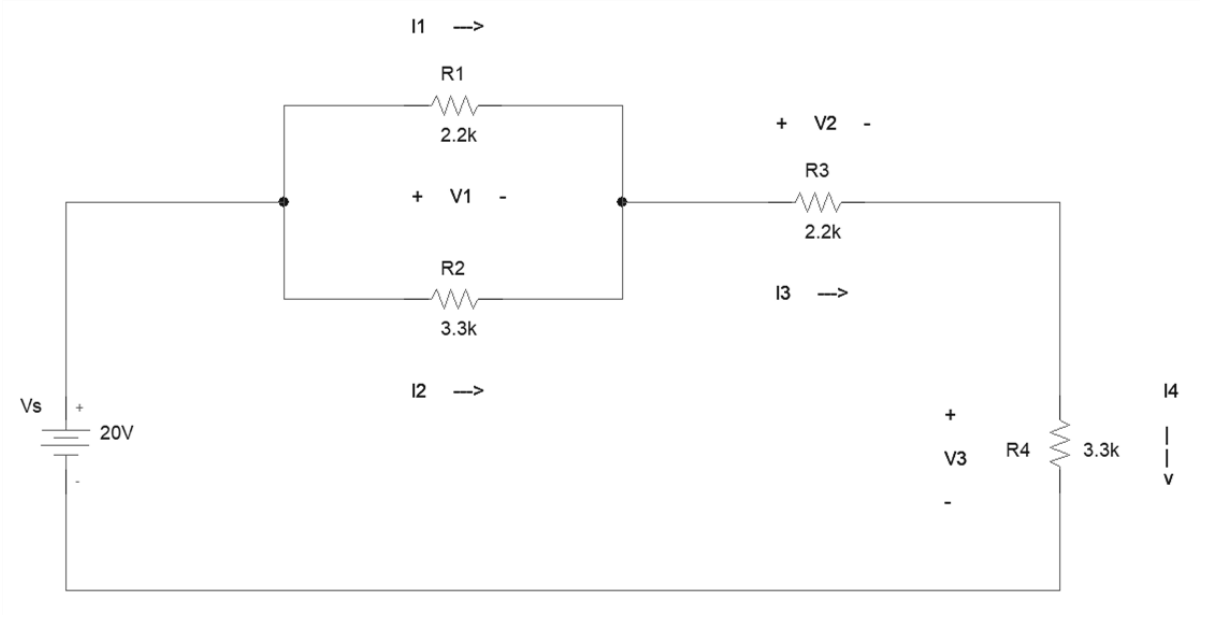


Fig. 1

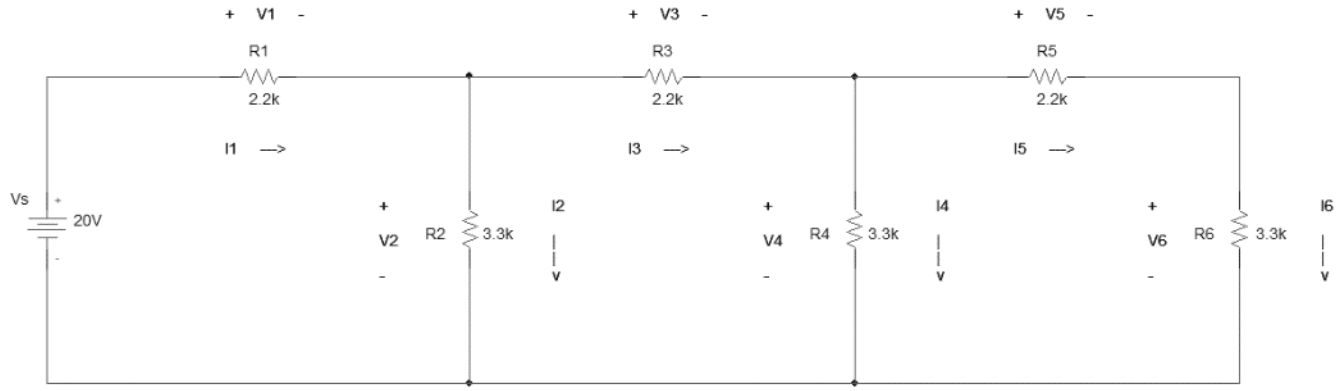
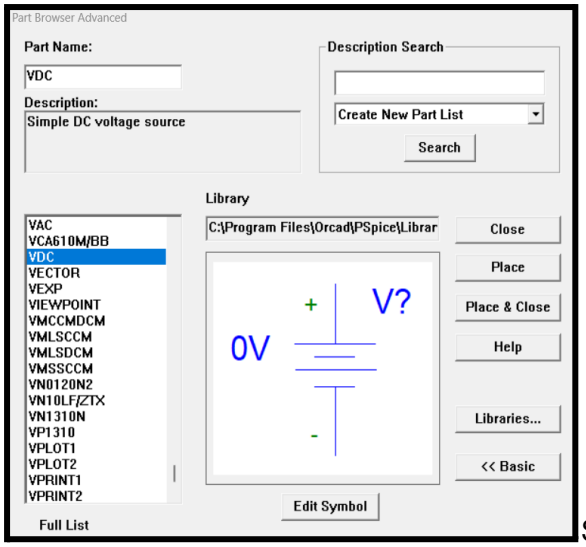


Fig. 2

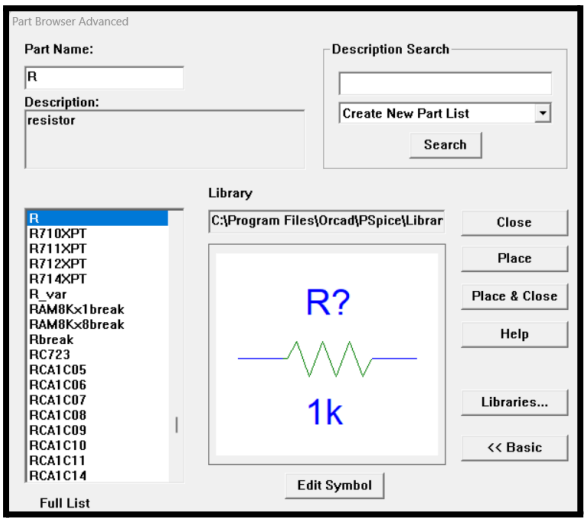
5. Procedure:

i. Set up the circuit as in the above figures.

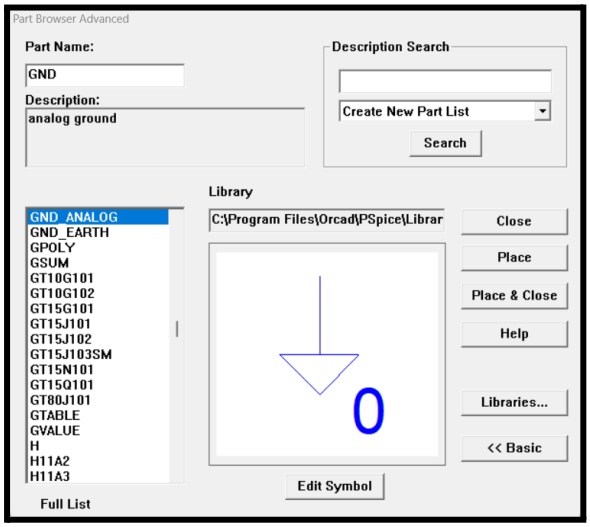
ii. Measure the voltage across the resistors and calculate the currents.



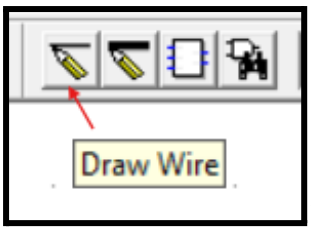
Selection of Voltage Source



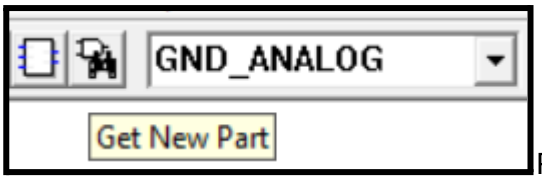
Selection of Resistor



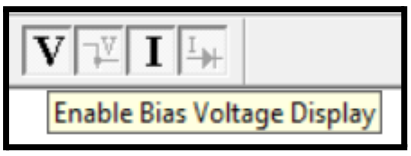
Selection of Ground



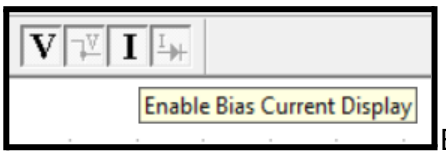
Wire tool



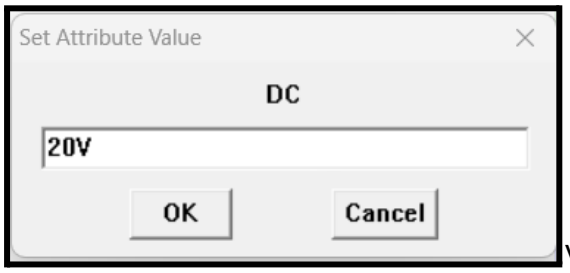
Parts menu



Bias Voltage



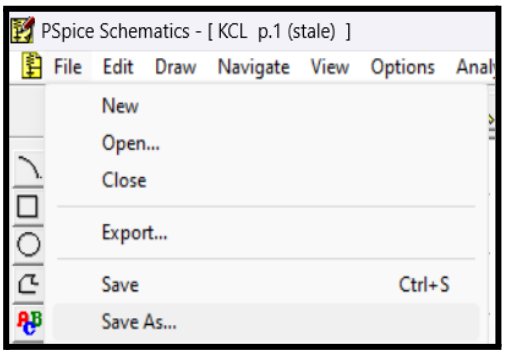
Bias Current



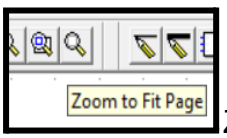
VDC Set (Voltage source value set)

Resistor set (R value set)

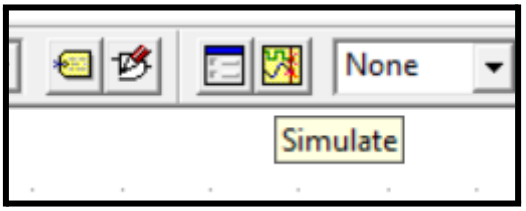
Text tools



File saving



Zoom



Begin Simulation

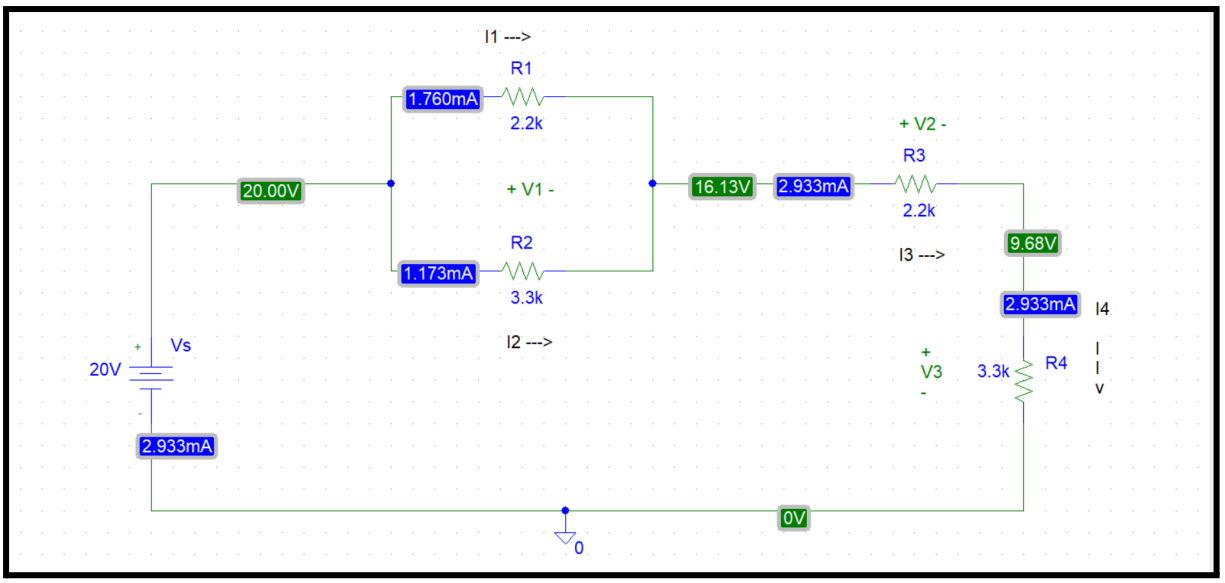


Fig.1 Series-parallel circuit simulated in PSpice Schematics

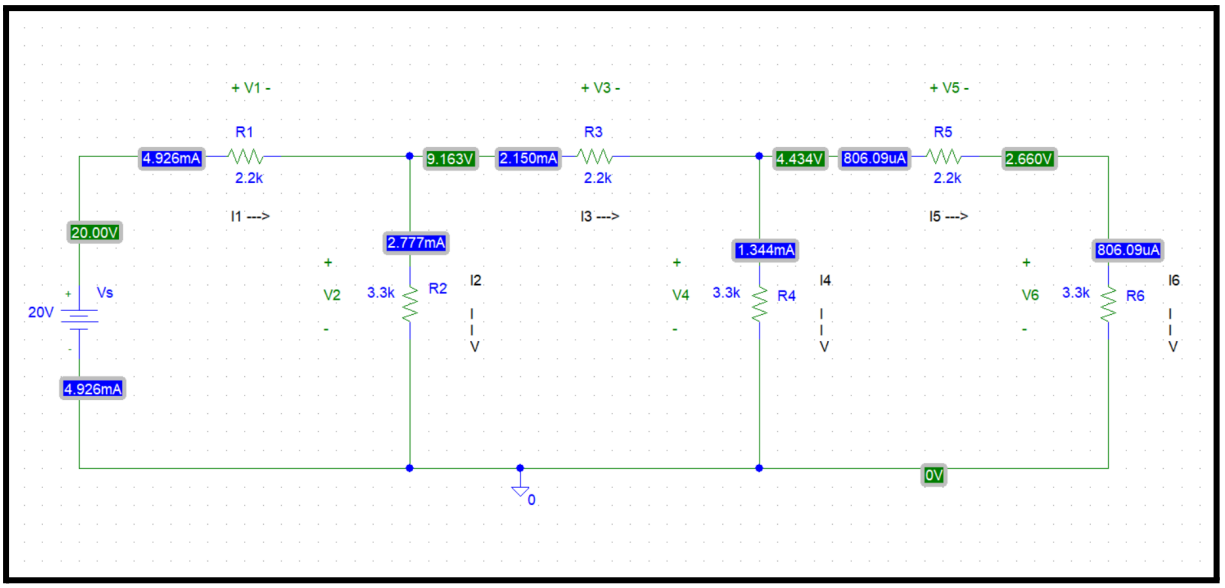


Fig.2 Series-parallel circuit simulated in PSpice Schematics

6. Data Table:

Fig. 1:

R_1 (kΩ)	R_2 (kΩ)	R_3 (kΩ)	R_4 (kΩ)
2.2	3.3	2.2	3.3

V_1 (V)	V_2 (V)	V_3 (V)
3.87	6.45	9.68

I_1 (mA)	I_2 (mA)	I_3 (mA)	I_4 (mA)
1.760	1.173	2.933	2.933

Fig. 2:

R_1 (kΩ)	R_2 (kΩ)	R_3 (kΩ)	R_4 (kΩ)	R_5 (kΩ)	R_6 (kΩ)
2.2	3.3	2.2	3.3	2.2	3.3

V_1 (V)	V_2 (V)	V_3 (V)	V_4 (V)	V_5 (V)	V_6 (V)
10.873	9.163	4.729	4.434	1.774	2.660

I_1 (mA)	I_2 (mA)	I_3 (mA)	I_4 (mA)	I_5 (mA)	I_6 (mA)
4.926	2.777	2.150	1.344	0.80609	0.80609

Faculty Signature and Date

7. Caution:

1. Don’t switch on the supply until the circuit has been checked by your teacher.
2. Take care of the reading of the apparatus.
3. Take care of any bare circuit element in energized condition.

Discussion of the software simulation

Equipments required:

1. PSpice Schematics software
2. Suitable device (PC or Laptop)

Simulation procedure:

Circuit figure.1

1. Open PSpice Schematics software.
2. Open the parts menu (click on the icon).
3. Search the necessary parts for making a series-parallel circuit (VDC, GND, R).
4. Place the parts on designated places following the provided diagram and close the parts menu.
5. Using the wire tool connect all the parts in the circuit.
6. Set the values of all the parts.
7. Rename all the parts for easier identification (VDC=Vs)
8. Use the Draw Text and Text Box tool to mark necessary information.
9. Enable Bias Voltage and Current display.
10. Use the zoom to fit page tool.
11. Save the file with a suitable name.
12. Begin circuit simulation.
13. Attach a screenshot of the circuit in the report document.
14. Fill out the data tables with necessary information (Voltage and current through each component).

Part-B: Circuit figure.2

1. Open PSpice Schematics software.
2. Open the parts menu (click on the icon).
3. Search the necessary parts for making a series-parallel circuit (VDC, GND, R).
4. Place the parts on designated places following the provided diagram and close the parts menu.
5. Using the wire tool connect all the parts in the circuit.
6. Set the values of all the parts.
7. Rename all the parts for easier identification (VDC=Vs)
8. Use the Draw Text and Text Box tool to mark necessary information.
9. Enable Bias Voltage and Current display.
10. Use the zoom to fit page tool.
11. Save the file with a suitable name.
12. Begin circuit simulation.
13. Attach a screenshot of the circuit in the report document.
14. Fill out the data tables with necessary information (Voltage and current through each component).