**East West University**

**Department of CSE**

**LAB REPORT**

|  |  |  |
| --- | --- | --- |
| **Course Code and Name:**  CSE209 Electrical Circuits | | |
| **Experiment no: 03** | | |
| **Experiment name:**  Experiment name: Bias Point Detail Analysis of DC Circuit with Independent Sources Using PSpice Schematics | | |
| **Semester and Year:**  Fall 2021 |  | |
| **Name of Student:**  D.M. Rafiun -Bin- Masud | **Course Instructor information:**  M Saddam Hossain Khan  Senior Lecturer, Department of Computer Science and Engineering | |
| **Student Id:**  2019-3-60-137 |
| **Date of Report Submitted:**  15 December,2021 | **Pre-Lab Marks:** |  |
| **Post Lab Marks:** |  |
| **TOTAL Marks:** |  |

PSpice is a simulation-based circuit analysis program. The objective of this lab is get used to with PSpice works and this is very significant as circuit voltage and current can be easily verified. The simulation values and mathematical calculation values come similar.

Objectives:

1.To learn fundamentals of PSpice.

2.To analyze Bias Point Detail of DC circuit using PSpice Schematics.

Circuit diagram using PSpice simulator

A picture containing application

Description automatically generated

Figure 1-Result of simulation

Schematics Netlist:

I\_I2 $N\_0001 0 DC -10A

R\_R1 $N\_0003 $N\_0002 20ohm

R\_R2 $N\_0001 $N\_0002 30ohm

R\_R4 $N\_0001 0 10ohm

R\_R3 0 $N\_0002 10ohm

V\_V1 $N\_0003 0 10V

I\_I3 $N\_0002 $N\_0001 DC 3A

# Experimental Data Representing Table:

Table 1: Experimental Datasheet.

|  |  |  |  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- | --- | --- | --- |
| Resistor  R Ω | Voltage V1 | Voltage  V2 | Voltage  V3 | Voltage V4 | Current of R1, i1 | Current of R2, i5 | Current of R3, i6 | Current of R4, i4 |
| R1= 20 Ω  R2 = 30Ω  R3 = 10Ω  R4 = 10Ω | 10V | 4.286V | 98.57V | 98.57V | 285.71mA | 3143mA | 428.57mA | 9860mA |

Results and Discussion

Post Lab Question/Answer: Theoretically calculate all the currents and the voltages for the circuit shown in the Figure