 **East West University**

**Lab Report**

**Semester:** Summer-2024

**Course Title:** Electrical Circuits **Course Code:** CSE209

**Sec:** 01

**Expt No: 03**

**Expt Name:** Bias Point Detail Analysis of DC Circuit With Independent Sources Using PSpiceSchematics

**Group No: 07**

**Submitted by-**

Name: Sheikh Sarafat Hossain

Id: 2022-3-60-109

**Submitted to-**

Dr. Sarwar Jahan

Associate Professor

Department of Computer Science & Engineering

East West University

**Date of Performance: 11-July-2024**

**Date of Submission: 18-July-2024**

**Objectives:**

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

**Introduction to PSpice:**

PSpice is a powerful general purpose analog circuit simulator that is used to verify circuit designs and to predict the circuit behavior. PSpice can be used in two ways to simulate a circuit. In one method, the circuit is described by writing codes using the syntax of PSpice. The resulting file, which contains all the information of the circuit is called netlist. PSpice uses the netlist as its input and simulates the circuit. In the other method, the circuit is drawn graphically using a software tool called Schematics. Then PSpice uses the Schematic circuit as its input and simulates it. In this experiment, you will learn to use the PSpice circuit simulation using Schematics. We will use PSpice Student version available in VLSI Lab.

A diagram of a circuit

Description automatically generated

Figure 2: Circuit for lab practice

A diagram of electrical wiring

Description automatically generated

A screenshot of a computer

Description automatically generated

**Post-Lab Report Question:**

1 . Theoretically calculate all the currents and the voltages for the circuit shown in Figure 2.

**Solution:**

From the figure,

I4 = 3A

Applying KVL in mesh 1:

-10 + 20I1 + 10(I1-I2) = 0

30I1 – 10I2 = 10 ------------- (1)

Applying KVL at Super-mesh:

10(I2 - I1) + 30 (I2-I4) + 10I3 = 0

10I2 – 10I1 + 30I2 – 30I4 + 10I3 = 0 [∵ I4 = 3A]

40I2 – 10I1 - 30×3 + 10I3 = 0

∴ - 10I1 +40I2 + 10I3 = 90 ------------- (2)

And,

I2 – I3 = -10 ------------- (3)

By solving equation (1), (2), and (3), we get,

I1 = 0.28571A = 285.71mA,

I2 = -0.14286A = -142.86mA,

I3 = 9.85714A = 9857.14mA

So, I5 = I4 – I2 = 3 + 0.14286 = 3.14286A = 3142.86mA

And, I6 = I1 – I2 = 0.28571 + 0.14286 = 0.42857A = 428.57mA

Here,

V2 = + (I6 × R3)= 0.42857 × 10 = 4.2857V

And,

V3 = + (I3 × R4)= 9.85714× 10 = 98.5714V

2. Compare the theoretical solution of the circuit shown in Figure 2 with the solutions obtained from PSpice:

**Solution:**

|  |  |  |
| --- | --- | --- |
|  | PSpice | Theoretical value |
| Current | I1 = 285.71mA,  I4 = 3.143A,  I5 = 428.57mA,  I6 = 9.86A | I1 = 285.71mA,  I4 = 3.143A,  I5 = 428.57mA,  I6 = 9.86A |
| Voltage | V2 = 4.286,  V5 = 98.57 | V2 = 4.15V,  V5 = 98.77V |

