



East West University

LAB REPORT

Course Code and Name: CSE 209 ; ELECTRICAL CIRCUIT	
Experiment no: 04 Group no: Individual	
Experiment name: Bias Point Detail Analysis of DC Circuit With Independent Sources Using PSpice Schematics	
Name of student & Id:	
B M Sharhia Alam	ID: 2021-3-60-016
Course Instructor information: M Saddam Hossain Khan(SHK) Senior Lecturer Department of Computer Science and Engineering East West University	
Date of Report Submitted: 11 December ,2022	

OBJECTIVE:

1. To gain knowledge about fundamentals of PSpice.
2. To experiment Bias Point Detail of DC circuit using PSpice Schematics.

THEORY AND EXPERIMENTAL METHODS:

In PSpice the program is run in order to draw circuit schematics. The program will let us run simulations and see graphic results. In this method, the circuit is described by writing codes using the syntax of PSpice. The resulting file is called netlist. PSpice uses 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, we will learn to use the PSpice circuit simulation using Schematics. There are a lot of things can be done with PSpice such as design and draw circuits, simulate circuits, analyze simulation results.

CIRCUIT DIAGRAM:

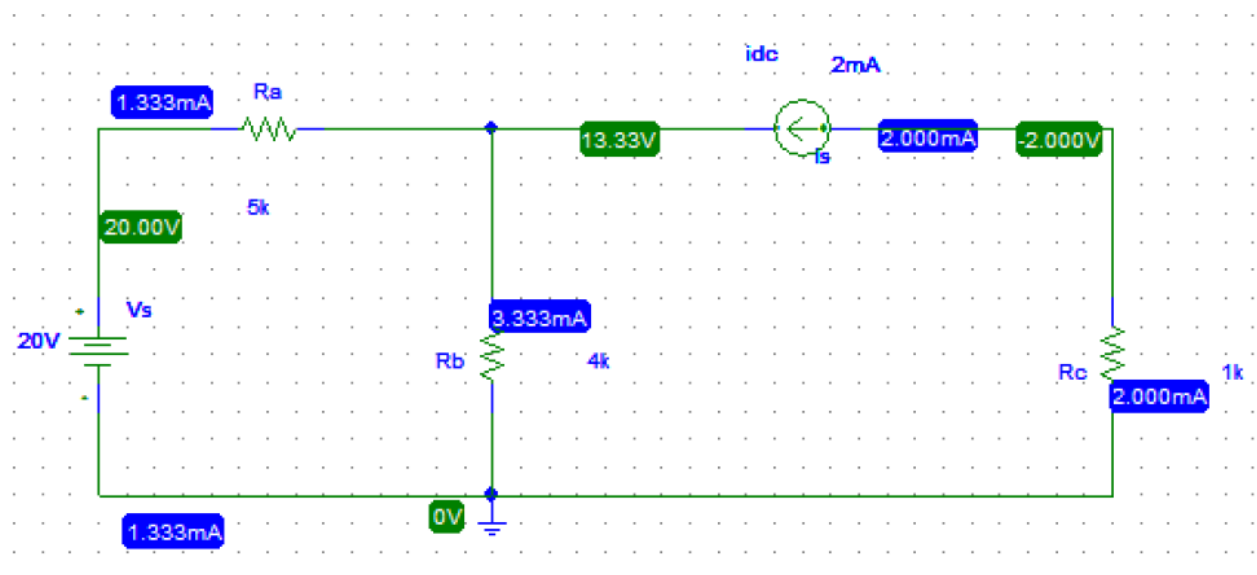


Figure 01: Simple Simulated Circuit

*** Schematics Netlist ***

R_Ra \$N_0002 \$N_0001 5k

R_Rb 0 \$N_0001 4k

R_Rc 0 \$N_0003 1k

V_Vs \$N_0002 0 20V

I_Is \$N_0003 \$N_0001 DC 2mA

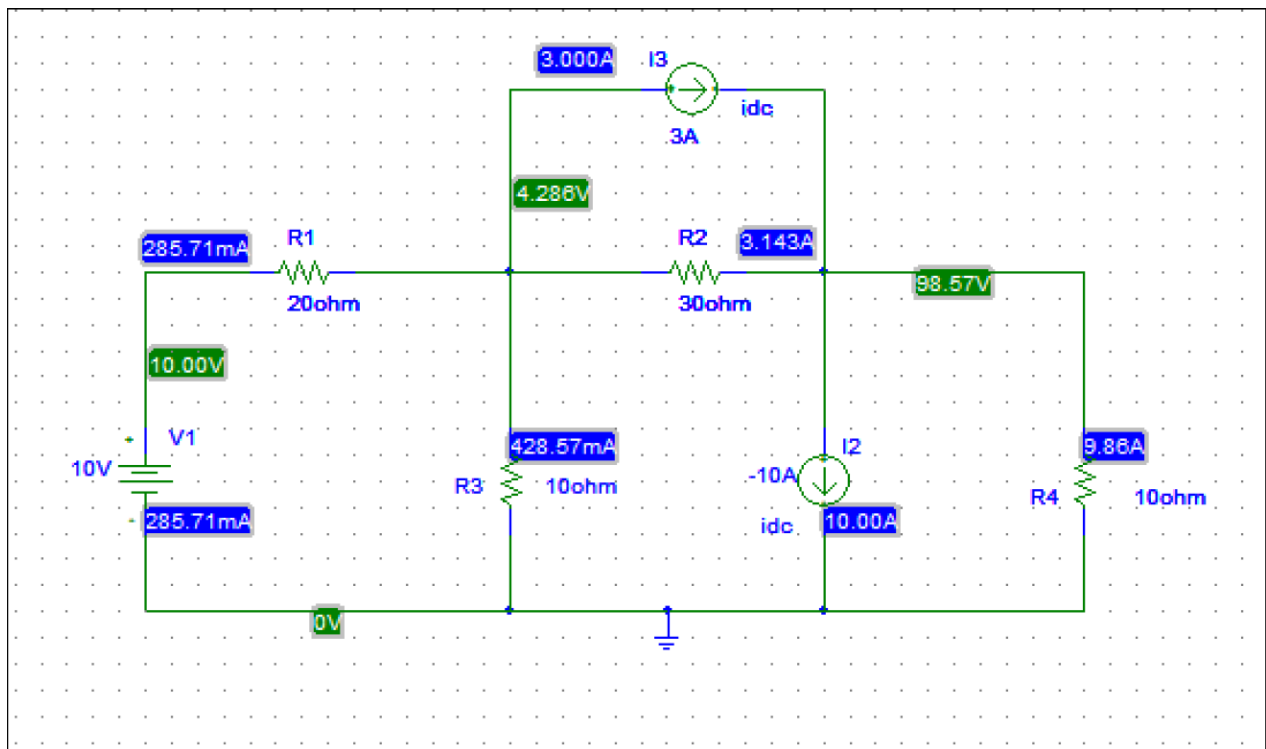


Figure 02: Simulated Circuit for Lab Practice.

*** Schematics Netlist ***

R_R4 0 \$N_0001 10ohm

I_I2 \$N_0001 0 DC -10A

R_R3 0 \$N_0002 10ohm

V_V1 \$N_0003 0 10V

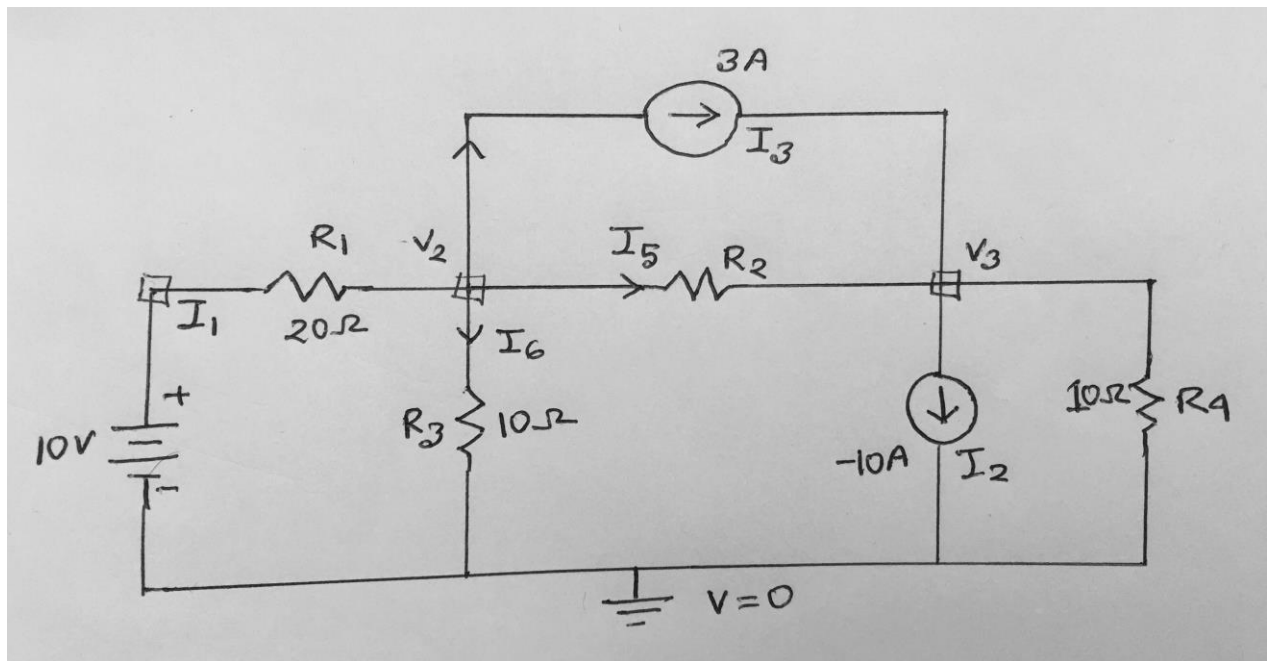
I_I3 \$N_0002 \$N_0001 DC 3A
R_R2 \$N_0002 \$N_0001 30ohm
R_R1 \$N_0003 \$N_0002

Result and discussion

By doing this experiment we have been able to simulate our circuits via PSpice and test the results. Previously we had tested our circuits practically, but this is more efficient.

Post-Lab Report Answers:

Answers to the questions no: 01



Applying KCL at node 2,

$$\Rightarrow -10 - V_{220} + 3 + V_2 - V_{330} + V_{210} = 0$$

$$\Rightarrow 11V_3 - 2V_2 = -150$$

$$\Rightarrow 2V_3 - 11V_2 = 150 \dots\dots\dots(1)$$

Applying KCL at node 3,

$$\Rightarrow -V_2 - V_{330} - 3 - 10 + V_{310} = 0$$

$$\Rightarrow 4V_3 - V_2 = 390$$

$$\Rightarrow V_2 = 4V_3 - 390 \dots\dots\dots(2)$$

From (1)

$$2V_3 - 11(4V_3 - 390) = 150$$

$$\Rightarrow -42V_3 = -4140$$

$$\Rightarrow V_3 = 98.57V$$

From (2)

$$V_2 = 4 \times 98.57 - 390$$

$$\Rightarrow V_2 = 4.286V$$

So,

$$V_1 = 10V$$

$$V_2 = 4.286V \quad V_3$$

$$= 98.57V$$

$$I_1 = 10 - V_{220}$$

$$= 285.71mA$$

$$I_5 = -314.28mA$$

$$I_4 = 428.57mA$$

$$I_6 = 985.7mA$$

$$I_2 = -10A$$

$$I_3 = 3A$$

Answers to the questions no: 02

		Theoretical solution	PSpice solution
Voltages	V1	10V	10V
	V2	4.286V	4.286V
	V3	98.57V	98.57V
Current	I1	285.71mA	285.71mA
	I2	-10A	-10A
	I3	3A	3A
	I4	428.57 mA	428.57 mA
	I5	-314.28mA	314.28mA
	I6	985.7 mA	985.7 mA

Result:

By doing this experiment we are able to simulate our circuits using PSpice and test the results. Previously we had tested our circuits practically, but this is more efficient.

Conclusion:

While doing this experiments, the readings were taken very carefully. Though there is some difference between calculated value and PSpice value, at the end of the experiment we finally gained practical knowledge that how to work with PSpice Schematic and independent source.