East West University

LAB REPORT

Course Code and Name: CSE 209 ; ELECTRICAL CIRCUIT
Experiment no: 05 **Group no: Individual**
Experiment name: Bias Point Detail Analysis of DC Circuit With Dependent 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 analyze Bias Point Detail of DC circuit with dependent source using PSpice Schematics.

THEORY AND EXPERIMENTAL METHODS:

In electric circuit there are two types of sources:
1. Independent source
2. Dependent source.

Dependent source contains two elements such as the controlling element and the controlled element which either can be current or voltage.
There are also four types of dependent source. They are:
1. Voltage-controlled voltage source (VCVS)
2. Voltage-controlled current source (VCCS)
3. Current-controlled voltage source (CCVS)
4. Current-controlled current source (CCCS)

But in PSpice they are described with different symbols or alphabets. Which are:
1. Voltage-controlled voltage source (VCVS) as E1.
2. Voltage-controlled current source (VCCS) as G1.
3. Current-controlled voltage source (CCVS) as H1.
4. Current-controlled current source (CCCS) as F1.

CIRCUIT DIAGRAM:

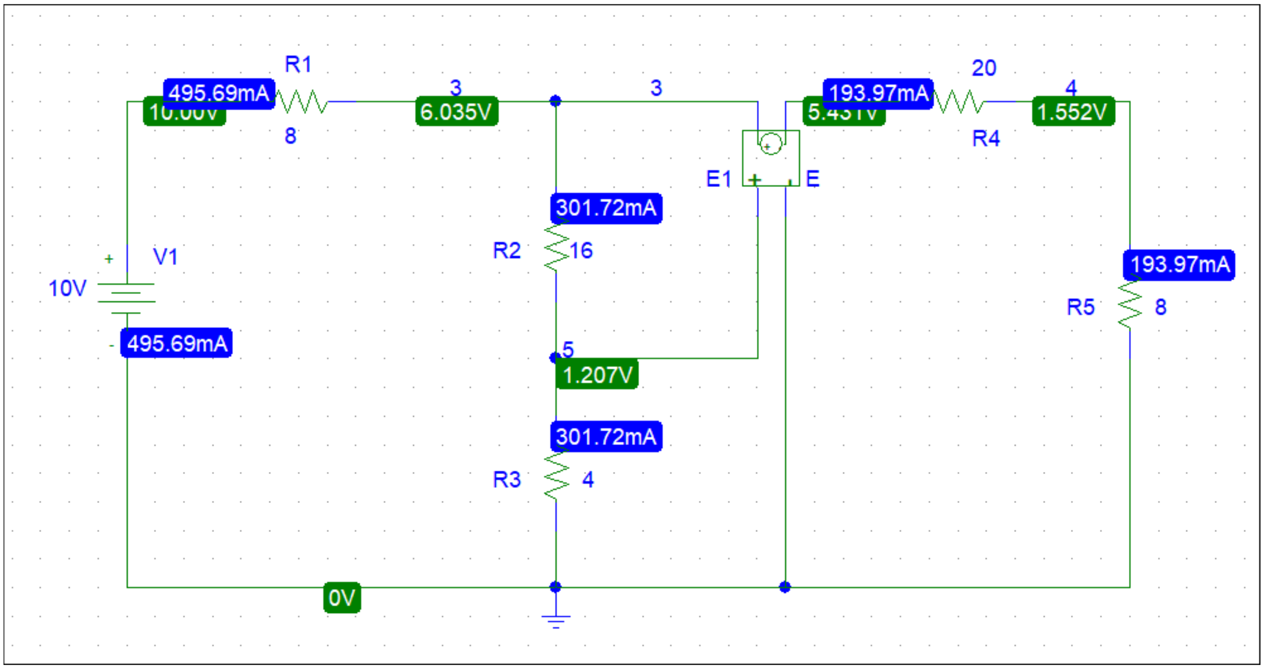


Figure 2

*** Schematics Netlist ***
V_V1 1 0 10V
R_R1 1 3 8
R_R2 5 3 16
R_R3 0 5 4
R_R4 4 $N_0001 20
R_R5 0 4 8
E_E1 3 $N_0001 5 0 0.5

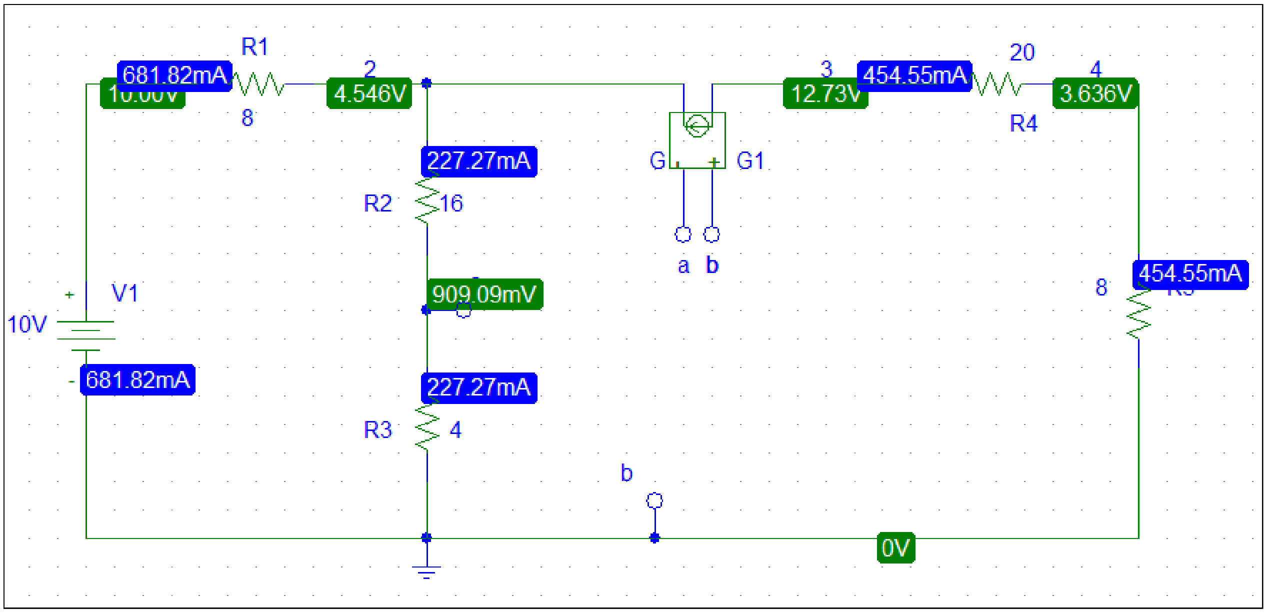


Figure 5

*** Schematics Netlist ***
R_R4 4 3 20
R_R1 1 2 8
R_R2 a 2 16
R_R3 0 a 4
R_R5 4 0 8
V_V1 1 0 10V
G_G1 3 2 0 a 0.5

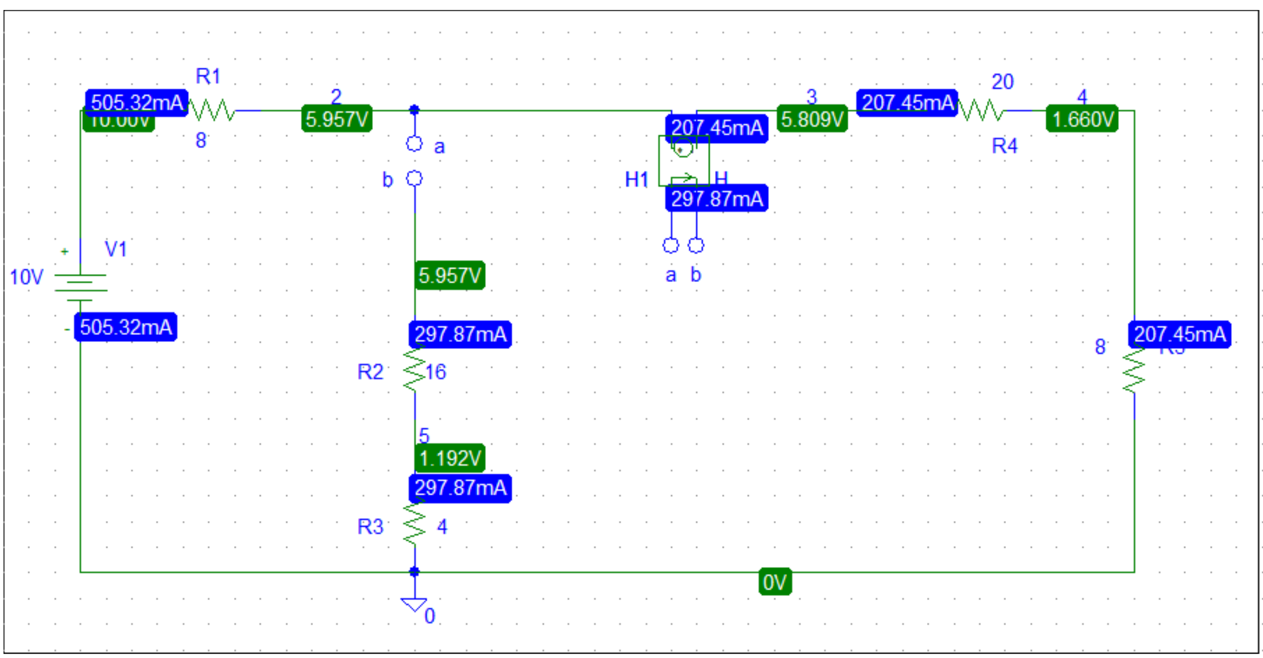


Figure 7

*** Schematics Netlist ***
V_V1 1 0 10V
R_R1 1 a 8
R_R2 5 b 16
R_R4 4 3 20
R_R5 4 0 8
R_R3 0 5 4
H_H1 a 3 VH_H1 0.5
VH_H1 a b 0V

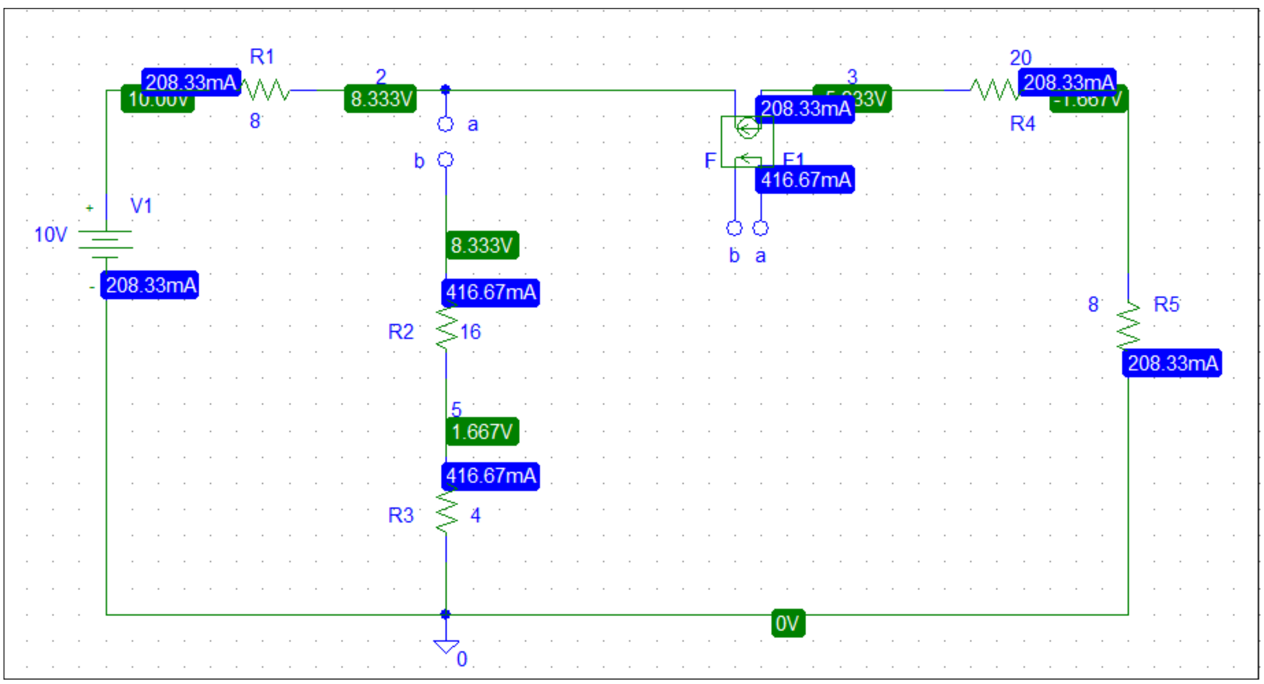


Figure 9

*** Schematics Netlist ***
V_V1 1 0 10V
R_R1 1 a 8
R_R2 5 b 16
R_R3 0 5 4
R_R4 4 3 20
R_R5 4 0 8
F_F1 3 a VF_F1 0.5
VF_F1 a b 0V

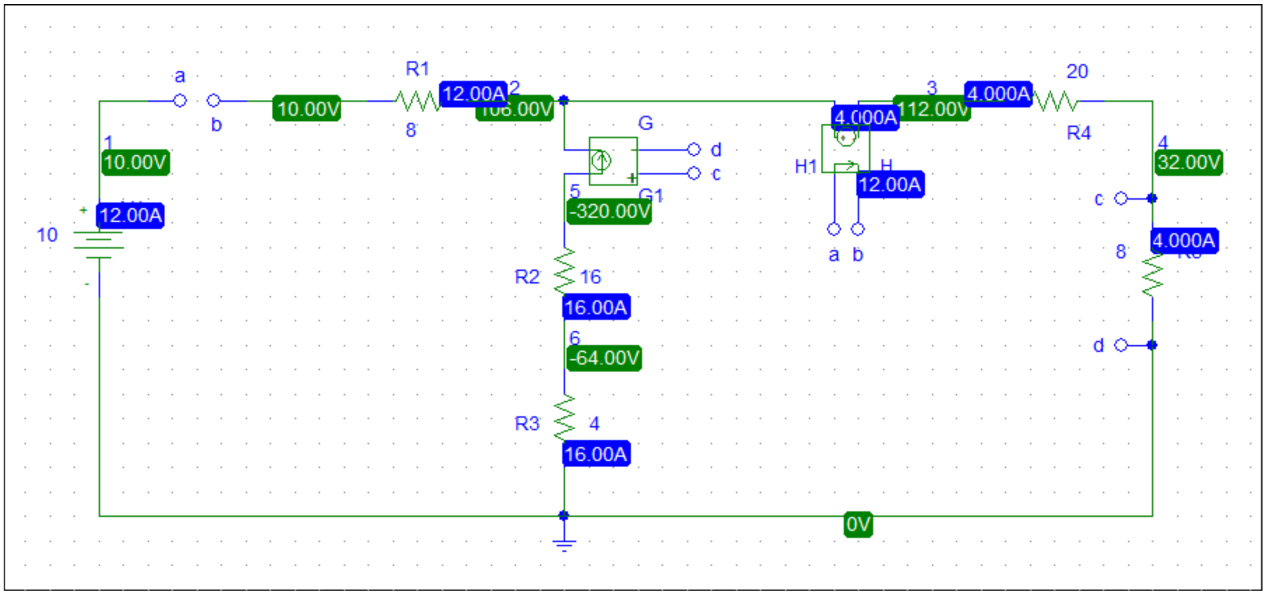
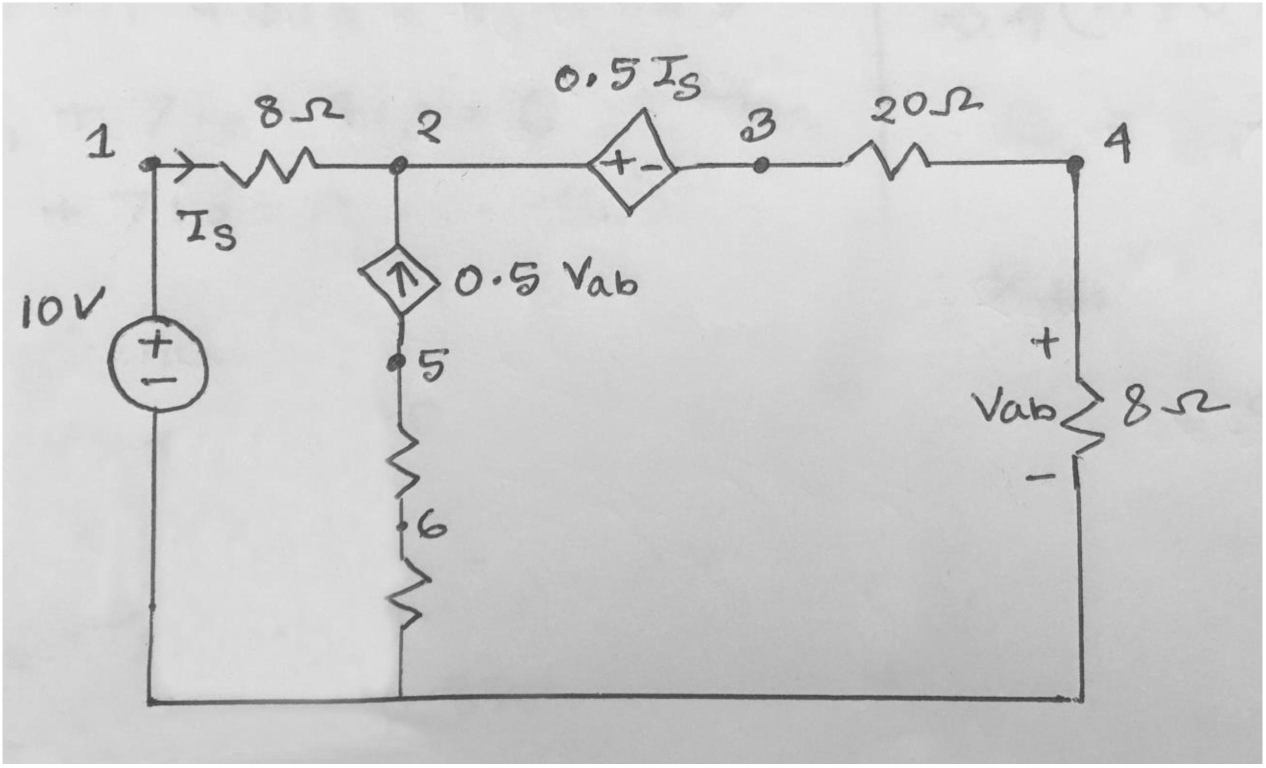


Figure 11

*** Schematics Netlist ***
G_G1 5 2 c 0 0.5
H_H1 2 3 VH_H1 0.5
VH_H1 a b 0V
V_V1 a 0 10
R_R3 0 6 4
R_R5 c 0 8
R_R4 c 3 20
R_R2 6 5 16
R_R1 b 2 8

Post-Lab Report Answers:



Answers to the questions no: 01

From the figure,

$Is = i_1$

$V_{ab} = 8i_2$

KVL at Super-mesh,

$-10 + 8i_1 + 0.5Is + 20i_2 + 8i_2 = 0$

$\Rightarrow -10 + 8i_1 + 0.5i_1 + 20i_2 + 8i_2 = 0$ [$\because Is = i_1$]

$\Rightarrow 8.5i_1 + 28i_2 = 10$………………….(i)

Then,

$i_2 - i_1 - 0.5Vab = 0$

$\Rightarrow i_2 - i_1 - 0.5 \times 8i_2 = 0$ [$\because V_{ab} = 8i_2$]

$\Rightarrow i_1 + 3i_2 = 0$………………………….(ii)

From equation (i) and (ii) we get,

$i1 = -12A$

$i2 = 4A$

So,

$i_o = i2 – i1$

$= 4 – (-12)$

$= 16A$

Now, Voltage at Node 1,

$V1 = 10V$

Voltage at Node 2,

Applying KCL at Node 2,

$V2 −108 − V2 −0.5Is28=16$

$\Rightarrow V2\ 8−108+V2\ 28−0.5\times(−12)28 =16$

$\Rightarrow 9V2\ 56=47728$

$\Rightarrow V2 =106V$

Voltage at Node 3,

$i2 = 4A$

$R= 20+8$

$=28\Omega$

$V3 = (4 \times 28)$

$= 112V$

Voltage at Node 4,

$i2 = 4A$

$R = 8\Omega$

$V4 = (4 \times 8)$

$= 32V$

Voltage at Node 5,

$i_o = 16A$

$R = 16+4$

$=20\Omega$

$V5 = - (16 \times 20)$

$= -320V$

Voltage at Node 6,

$i_o= 16A$

$R= 4\Omega$

$V6 = - (16 \times 4)$

$= -64V$

So,
$V1 = 10V$
$V_2 = 106V$
$V_3 = 112V$
$V_4 = 32V$
$V_5 = -320V$
$V_6 = -64V$
$i_1 = -12A$
$i_2 = 4A$
$i_o = 16A$

Answers to the questions no: 02

Compare the theoretical solution of the circuit with the solutions obtained from PSpice simulation:

		Theoretical solution	**PSpice solution**
Current	$i1$	-12A	12A
	$i2$	4A	4A
	i_o	16A	16A
Voltages	$V1$	10V	10V
	$V2$	106V	106V
	$V3$	112V	112V
	$V4$	32V	32V
	$V5$	-320V	-320V
	$V6$	-64V	-64V

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.