**East West University**

**Department of CSE**

**LAB REPORT**

|  |  |  |
| --- | --- | --- |
| **Course Code and Name:**  CSE209 Electrical Circuits | | |
| **Experiment no: 04** | | |
| **Experiment name:**  Bias Point Detail Analysis of DC Circuit with Dependent 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:**  22 December,2021 | **Pre-Lab Marks:** |  |
| **Post Lab Marks:** |  |
| **TOTAL Marks:** |  |
|  |  |  |

ABSTRACT

The experiment is designed for learning value of dependent source and be able to use that in PSpice. The objective of this lab is get used to with work of dependent sources and this is very significant as circuit voltage and current can be easily verified. The simulation values and 1 mathematical calculation values come a bit different when we calculate voltage rather than the value of current is same.

Objectives:

1. To learn fundamentals of PSpice.

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

3. Using correct measure of the values from the analysis

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

Circuit diagram using PSpice simulator

Diagram

Description automatically generated

Figure 1

Diagram, calendar

Description automatically generated

Figure 2

Schematics Netlist of Figure 1:

V\_V1 $N\_0001 0 10V

R\_R1 $N\_0001 $N\_0002 8

R\_R2 a $N\_0002 16

R\_R3 0 a 4

R\_R4 $N\_0004 $N\_0003 20

E\_E1 $N\_0002 $N\_0003 a 0 1

R\_R5 $N\_0004 0 8

Schematics Netlist of Figure 2:

V\_V1 $N\_0001 0 10V

R\_R1 $N\_0001 $N\_0002 8

R\_R2 a $N\_0002 16

R\_R3 0 a 4

R\_R5 0 $N\_0003 8

R\_R4 $N\_0004 $N\_0003 20

G\_G2 $N\_0004 $N\_0002 a 0 1

Calendar

Description automatically generated

Figure 3

A picture containing timeline

Description automatically generated

Figure 4

Schematics Netlist of Figure 3:

V\_V1 $N\_0001 0 10V

R\_R1 $N\_0001 $N\_0002 8

R\_R2 a $N\_0002 16

R\_R3 0 a 4

R\_R5 0 $N\_0003 8

R\_R4 $N\_0004 $N\_0003 20

F\_F1 $N\_0004 $N\_0002 VF\_F1 1

VF\_F1 a 0 0V

Schematics Netlist of Figure 4:

V\_V1 $N\_0001 0 10V

R\_R1 $N\_0001 $N\_0002 8

R\_R2 a $N\_0002 16

R\_R3 0 a 4

R\_R5 $N\_0003 0 8

R\_R4 $N\_0003 $N\_0004 20

H\_H1 $N\_0002 $N\_0004 VH\_H1 1

VH\_H1 a 0 0V

Practice problem:

A picture containing graphical user interface

Description automatically generated

Figure 5

Schematics Netlist of Figure 5:

V\_V1 $N\_0001 0 10V

H\_H1 y $N\_0002 VH\_H1 1

VH\_H1 x y 0V

R\_R1 $N\_0001 x 8

G\_G1 $N\_0003 y a 0 1

R\_R2 a $N\_0002 20

R\_R3 a 0 8

R\_R4 $N\_0004 $N\_0003 16

R\_R5 0 $N\_0004 4

Results And Discussion:

POST LAB QUESTION’S ANSWER

* 1. Theoretically calculate all the currents and the voltages for the circuit shown in the circuit. Solution:

A circuit with VCCS and CCVS

Here,

Applying KCL we get,

Applying KVL in super mesh,

Voltage in node 1, V1 = 10V

Voltage in node 2, V2 = (12 × 8) +10 = 106 V

Voltage in node 3, V3 = 106 + 0.5Is = 106 +6 = 112V

Voltage in node 4, V4 = 8 × 4 = 32 V

Voltage in node 6, V6 = - 16 ×4 = -64 V

Voltage in node 5, V5 = V6 – (16 × 16) = -64 – 256 = -320V

Current in R1 = 12A

Current in R2 = 16A

Current in R3 = 16A

Current in R4 = 4A

Current in R5 = 4A

* 1. Compare the theoretical solution of the circuit shown in Figure 11 with the solutions obtained from PSpice simulation.

|  |  |  |
| --- | --- | --- |
|  | PSpice | Calculated |
| Currents |  |  |
|  |  |
|  |  |
| Voltages |  |  |
|  |  |
|  |  |
|  |  |
|  |  |
|  |  |

Solution: Comparing the theoretical solution of the circuit obtained from above with the solution obtained from PSpice.

Conclusions:

This experiment has been able to simulate the circuits as PSpice and test the results. Previously we had tested our circuits practically, but this is more efficient. Even if the experiment is done through an online environment still the value of all diagrams had to be calculated manually.