

Course: CSE209 Electrical Circuits

Expt No.: 4

Title: Bias Point Detail Analysis of DC Circuit With Dependent Sources Using PSpice Schematics

Objective:

1. To analyze Bias Point Detail of DC circuit with dependent source using PSpice Schematics.

Introduction:

A dependent source consists of two elements: the controlling element and the controlled element. The controlling element is either a voltage or a current and the controlled element is either a voltage or a current. There are four types of dependent sources that correspond to the four ways of choosing a controlling element and a controlled element. These four dependent sources are

- Voltage-controlled voltage source (VCVS)
- Voltage-controlled current source (VCCS)
- Current-controlled voltage source (CCVS)
- Current-controlled current source (CCCS)

In PSpice Schematics, the dependent sources can be found in the parts list. Click on the *get parts* list. VCVS is represented by the letter *E*, VCCS is represented by the letter *G*, CCVS is represented by the letter *H* and CCCS is represented by the letter *F* in PSpice. These parts have the shapes shown in Figure 1. The circular box represents the source and the other terminals are for the controlling parameter.

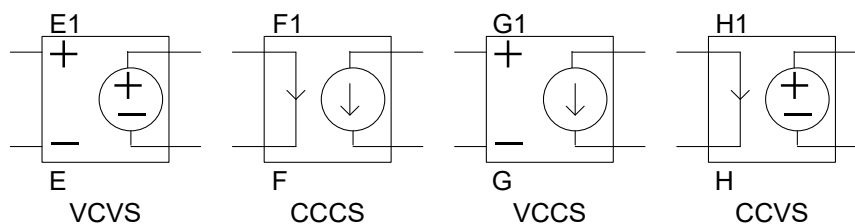


Figure 1. Shapes of dependent sources in PSpice Schematics.

The circuit of Figure 2 with VCVS can be drawn in PSpice Schematics as shown in Figure 3.

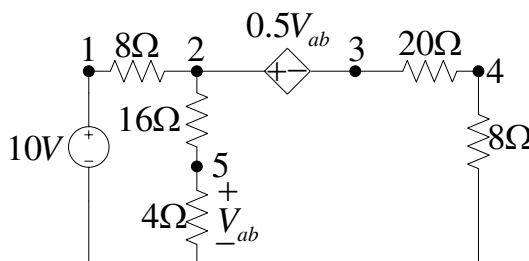


Figure 2. An example circuit with VCVS.

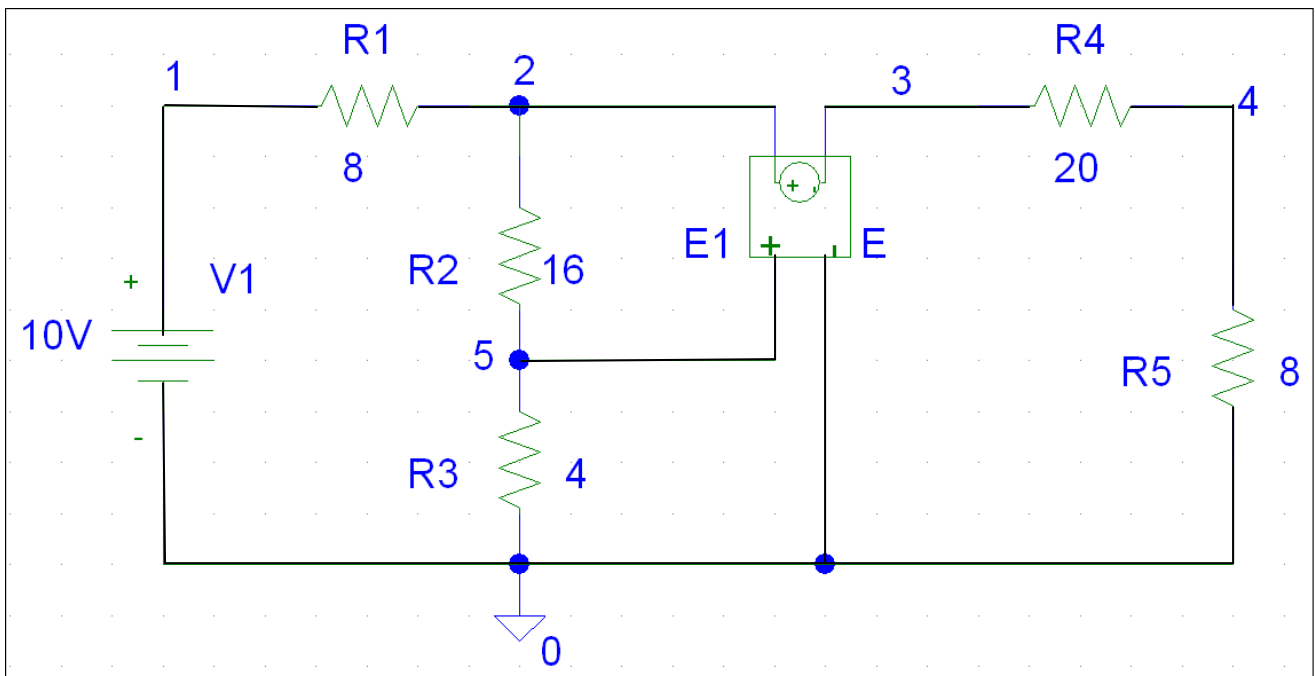
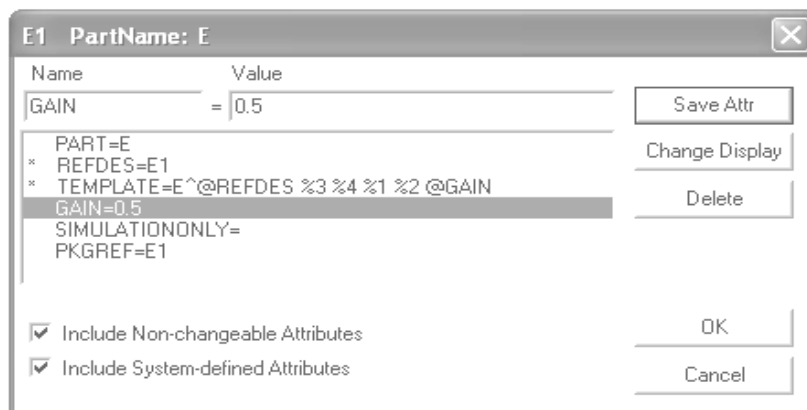


Figure 3. PSPice Schematic diagram for circuit of Figure 2 with VCVS.

To set the gain value, select VCVS and double click. The part attributes option will be activated. Click on the GAIN option and then write the GAIN value on the Value box. Click the Save Attr and then click OK.



Now simulate the circuit for voltages and currents.

For a circuit containing multiple numbers of dependent sources and meshes, the interconnection of the controlling nodes may become complicated. To make the interconnections easier, a connection bubble named **Bubble** can be used. To work with the bubble connector, click on the *add new parts* option and write b on the parts name box. Then select BUBBLE from full list column. Now, click the place and close option to work with the connection bubble. Place the bubble in one corner of the interconnection and double click it. In the Set Attribute Value option write a in the LABEL box and click OK. Copy the bubble in the other corner of the interconnection. The connection is then automatically made, you need not have to draw the wire to complete the connection. Similarly you can connect another bubble with different label for another interconnection. The circuit of Figure 2 with VCVS can be drawn with bubbles as shown in Figure 4.

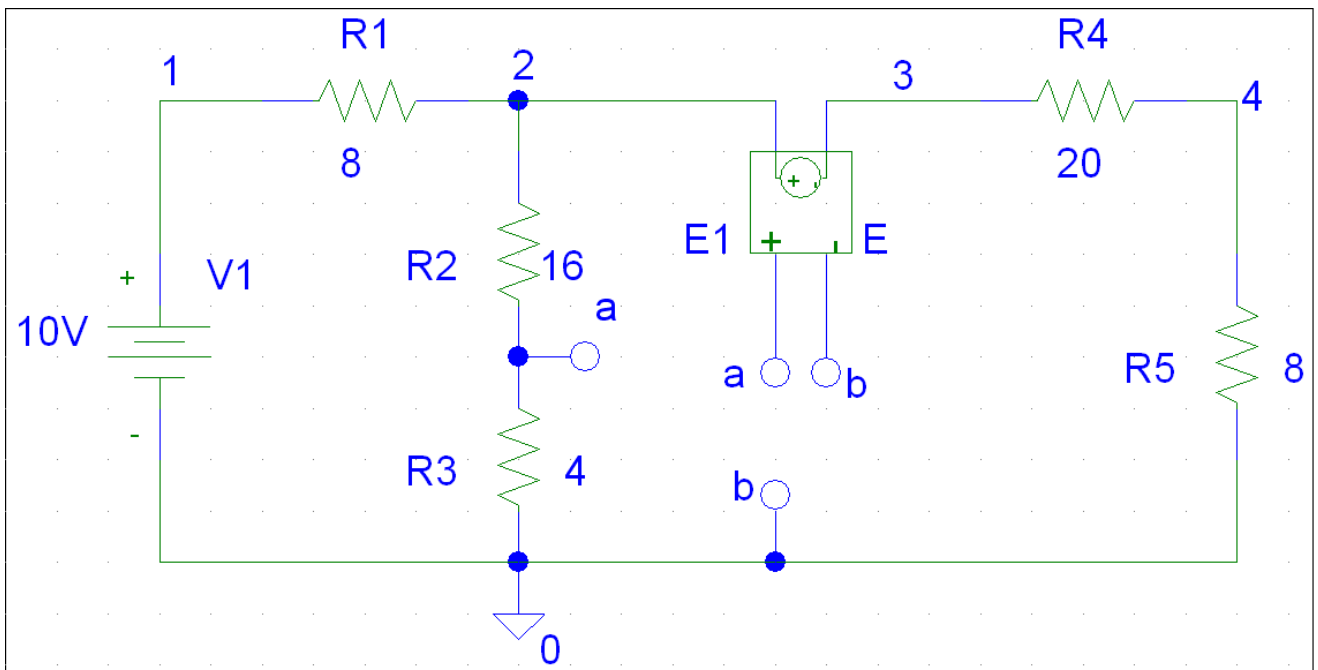


Figure 4. PSpice Schematic diagram with connection bubbles for circuit of Figure 2 with VCVS.

The circuit in Figure 5 with VCCS can be drawn in PSpice Schematics as shown in Figure 6.

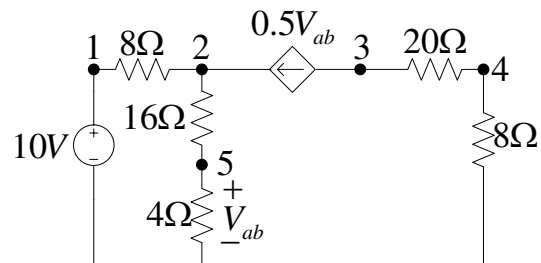


Figure 5. An example circuit with VCCS.

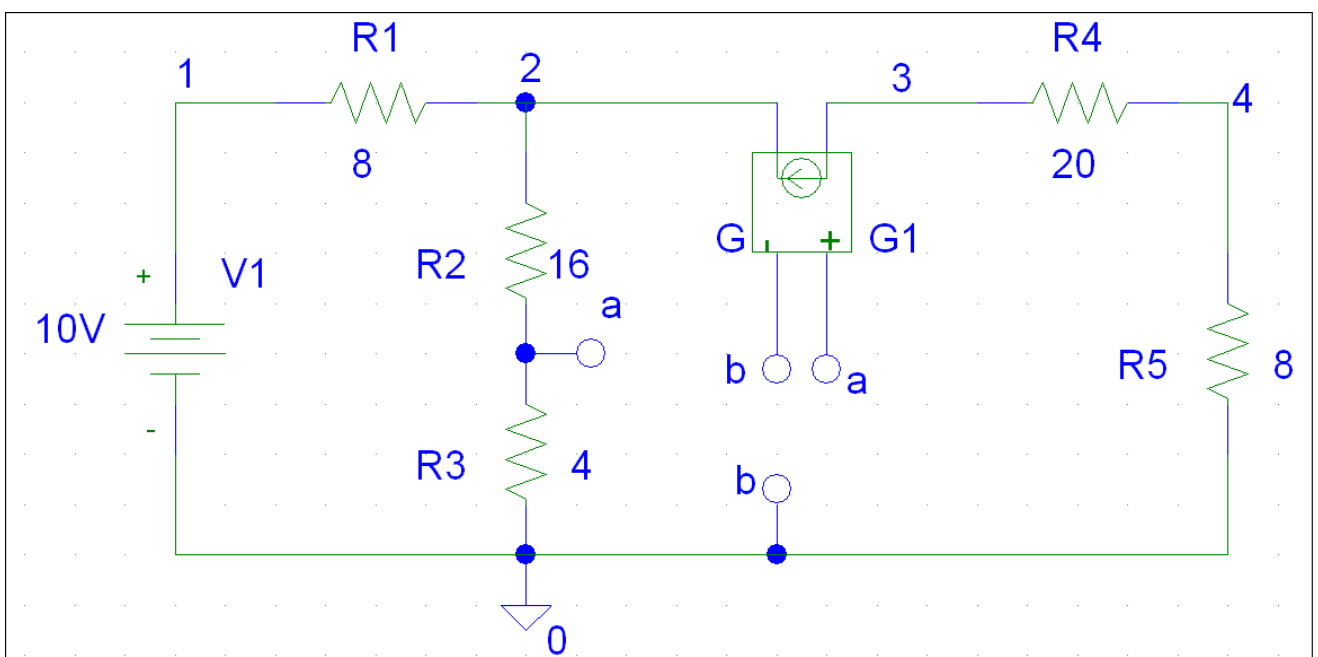


Figure 6. PSpice Schematic diagram with connection bubbles for circuit of Figure 5 with VCCS.

The circuit in Figure 7 with CCVS can be drawn in PSpice Schematics as shown in Figure 8.

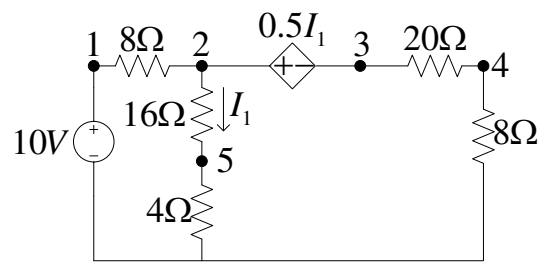


Figure 7. An example circuit with CCVS.

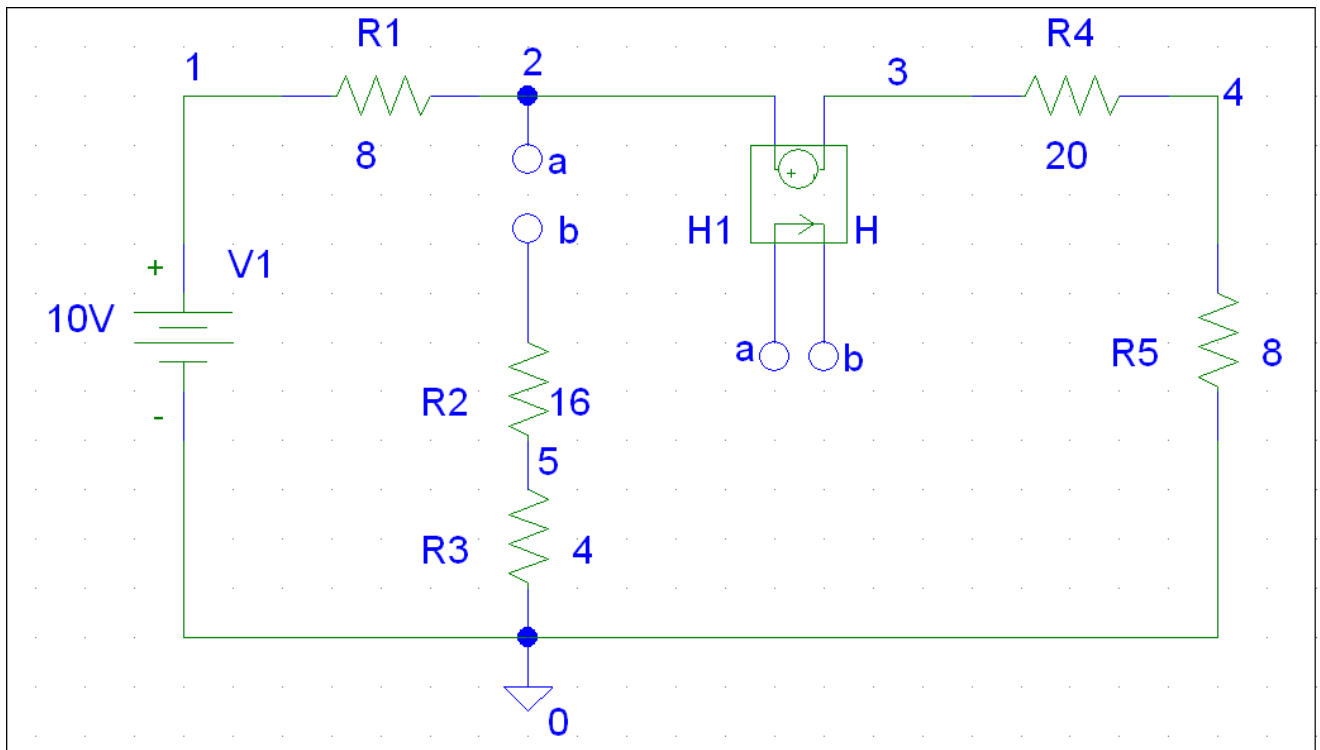


Figure 8. PSpice Schematic diagram with connection bubbles for circuit of Figure 7 with CCVS.

The circuit in Figure 9 with CCCS can be drawn in PSpice Schematics as shown in Figure 10.

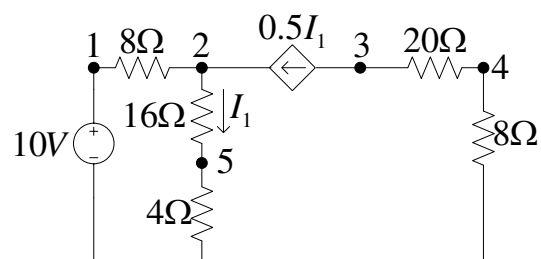


Figure 9. An example circuit with CCCS.

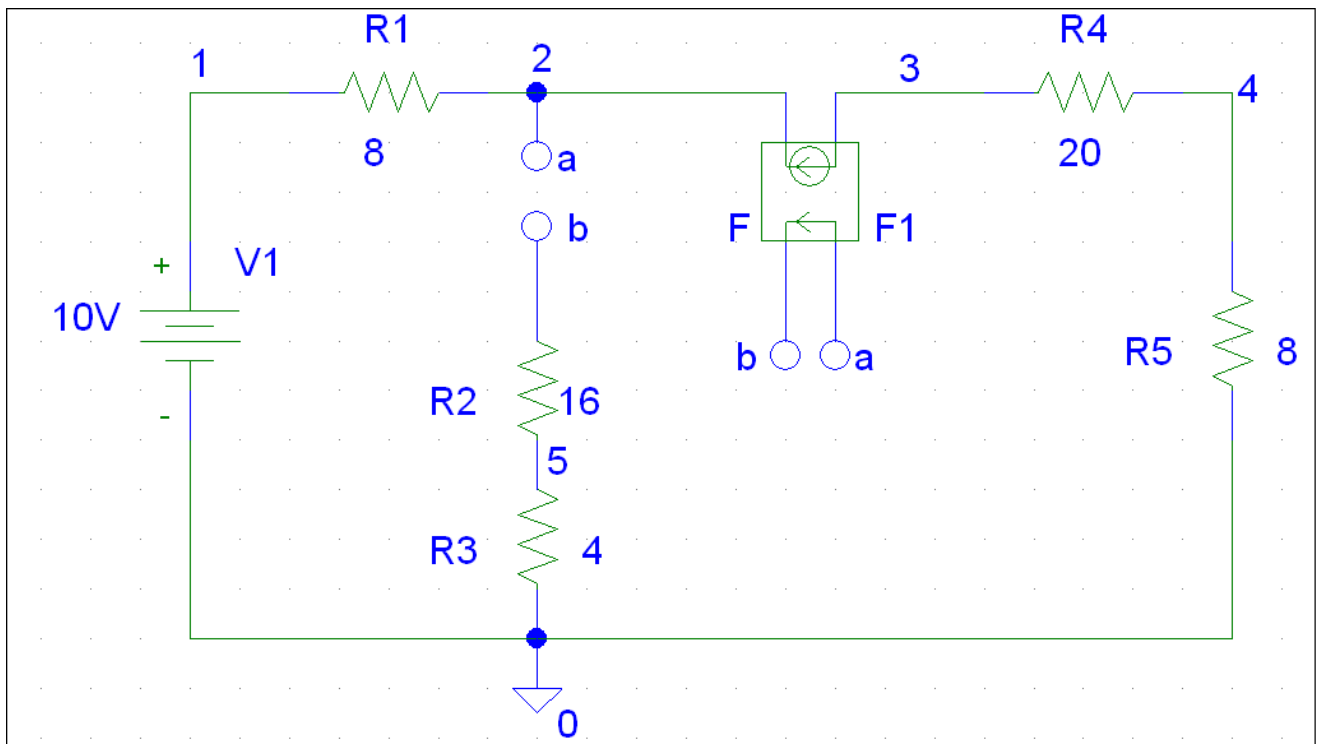


Figure 10. PSpice Schematic diagram with connection bubbles for circuit of Figure 9 with CCCS.

Lab Practice problem:

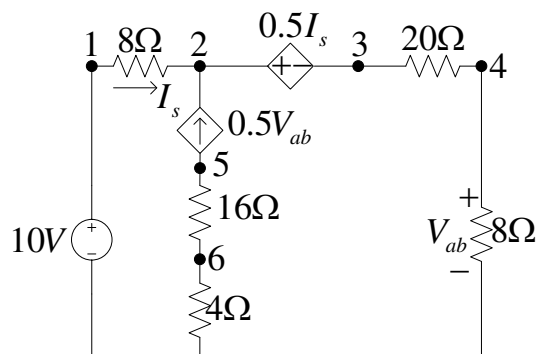


Figure 11. A circuit with VCCS and CCVS.

- Draw the circuit of Figure 11 using PSpice Schematic.
- Simulate the circuit and obtain the solution of all voltages and currents.
- Take printout of the schematic circuit diagram showing voltage and current results. Have the printouts signed by your instructor.

Post-Lab Report Question:

- Theoretically calculate all the currents and the voltages for the circuit shown in Figure 11.
- Compare the theoretical solution of the circuit shown in Figure 11 with the solutions obtained from PSpice simulation.