

Course: CSE209 Electrical Circuits

Expt No.: 3

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

Objectives:

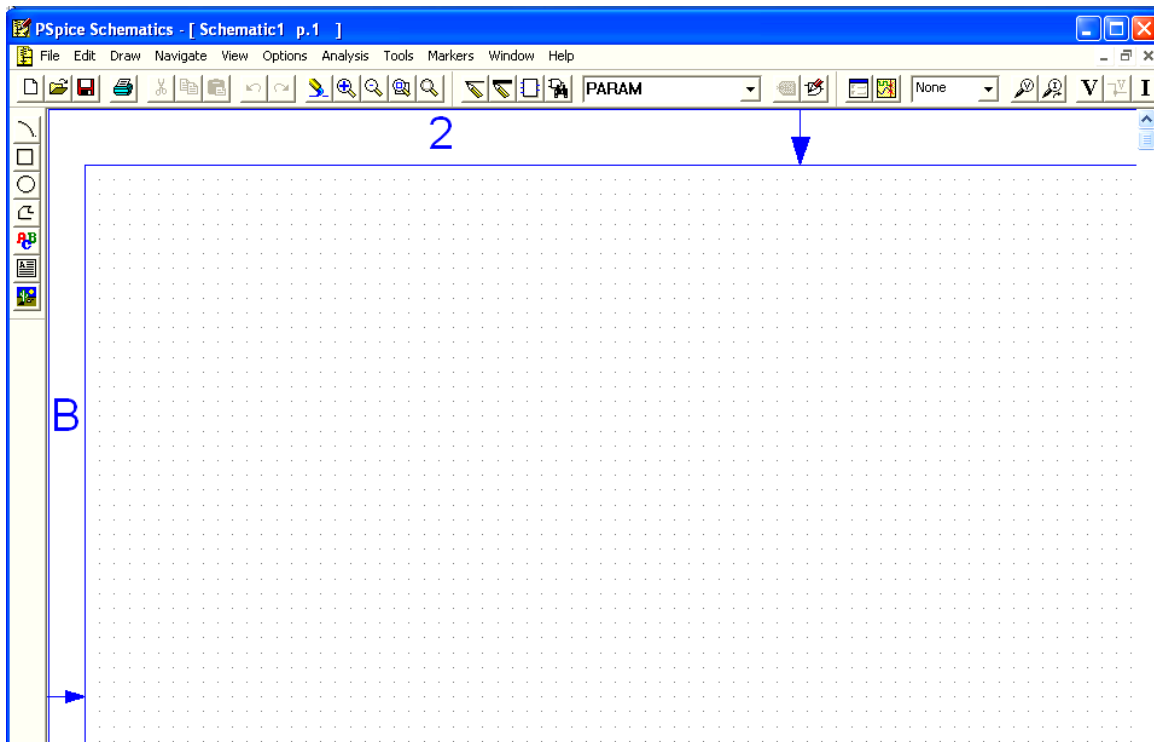
1. To learn fundamentals of PSpice.
2. To analyze Bias Point Detail of DC circuit using PSpice Schematics.


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.

Steps to Follow for Circuit Simulation using PSpice Schematics:

1. Select **Schematic** under PSpice to get the following schematic window.



2. Get the parts you need to simulate your circuit by clicking on the 'get new parts' button .

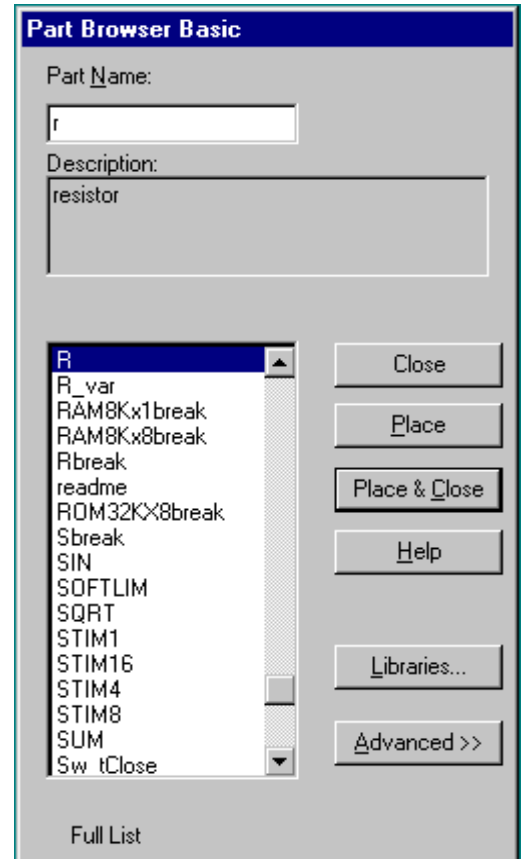
Once the “Part Browser Basic” window is open, select a part that you want in your circuit. This can be done by typing in the name or scrolling down the list until you find it.

Some common parts are:

- r - resistor
- GND_ANALOG or GND_EARTH - this is very important, **you MUST have a ground in your circuit**
- VAC and VDC – voltage sources
- IAC and IDC – current sources

Upon selecting your parts, click on the place button and then click where you want it to be placed.


Once you have all the parts you need, close the window.



3. Place the Parts in the places that make the most sense. Just select the part and drag it where you want it.

To rotate parts so that they fit in your circuit nicely, click on the part and press "Ctrl+R" (or Edit>Rotate). To flip them, press "Ctrl+F" (or Edit>Flip).

If you have any parts left over, just select them and press "Delete".

4. Connect the parts with wires. Go to the tool bar and select "Draw Wire" . With the pencil looking pointer, click on one end of a part, when you move your mouse around, you should see dotted lines appear. Attach the other end of your wire to the next part in the circuit.

Repeat this until your circuit is completely wired.

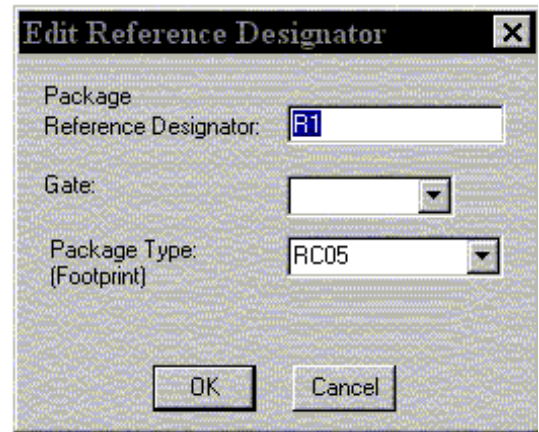
If you want to make a node (to make a wire go more then one place), click somewhere on the wire and then click to the part (or the other wire). Or you can go from the part to the wire.

To get rid of the pencil, right click or press “Esc”.

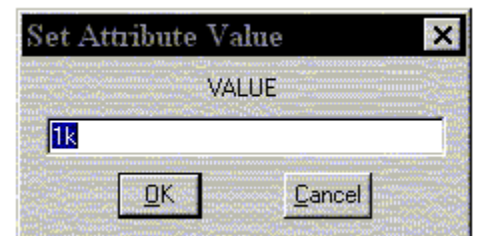
If you end up with extra dots near your parts, you probably have an extra wire, select this short wire (it will turn red), then press "Delete".

If the wire doesn't go the way you want (it doesn't look the way you want), you can make extra bends in it by clicking in different places on the way (each click will form a corner).



- To change the name of a part, double click on the present name (C1, or R1 or whatever your part is), then "Edit Reference Designator" window will pop up. In the "Package Reference Designator", you can type in the name you want the part to have.




- If you want to change the value of the part, you can double click on the present value and the "Set Attribute Value" window will appear. Type in the new value and press OK.




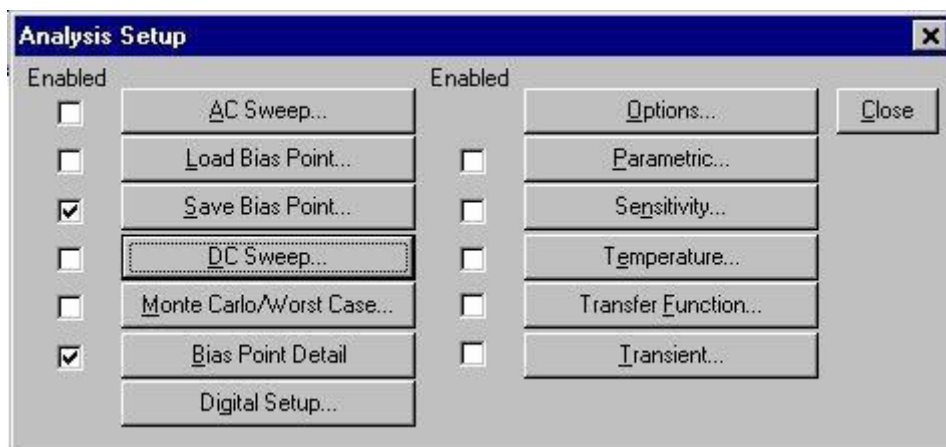
- Make sure you have a GND. This is very important. You cannot do any simulation on the circuit if you don't have a ground. If you aren't sure where to put it, place it near the negative side of your voltage source.
- Place Voltage and Current Bubbles. These are important if you want to measure the voltage at a point or the current going through that point.


To add voltage or current bubbles, go to the right side of the top tool bar and select "Voltage/Level Marker" (Ctrl+M)  or "Current Marker" .

- Fit the circuit to window by clicking .

- Analyze the circuit using the following steps.

Open the "Analysis Setup" window by clicking the  button. Enable the appropriate analysis options and then press close. In this experiment, we will use only the Bias Point Detail option. This option is used to calculate the voltages and currents in a DC circuit.



Click on the Simulate button on the tool bar .

Example of Circuit Solution

(i) Using the steps explained above draw and simulate the following circuit.

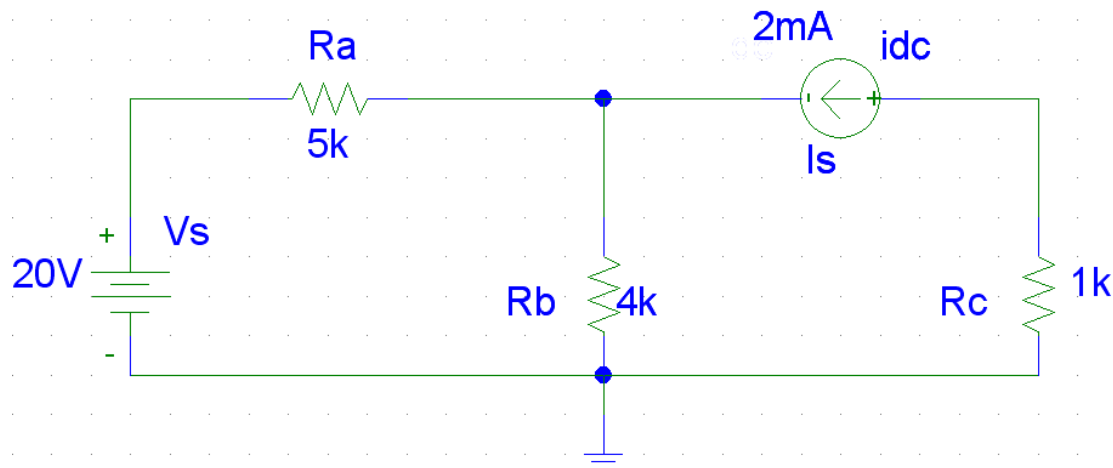
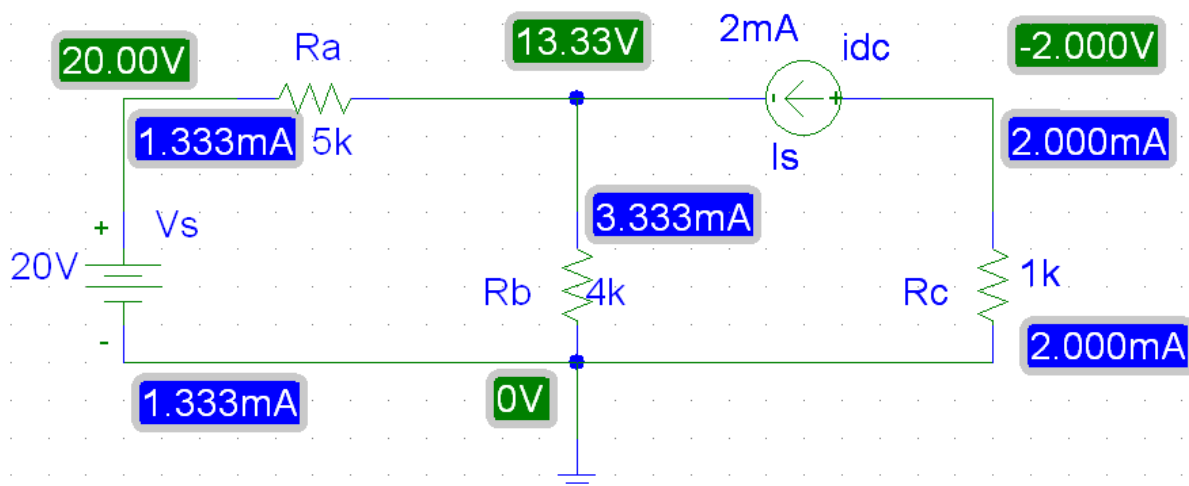


Figure 1: Example circuit.

(ii) In “Analysis Setup” window only enable “Bias Point Detail” option.

(iii) To examine the node voltages click the **V** button and to examine the current through each part click the **I** button.



(iv) You can also generate the netlist from the schematic by using the Analysis>Create Netlist menu. To see the created netlist use Analysis>Examine Netlist menu. Study the structure of the netlist and relate the entries in the netlist with your schematic circuit diagram.

* Schematics Netlist *

```
R_Ra  $N_0002 $N_0001  5k
V_Vs  $N_0002 0      20V
R_Rb  0    $N_0001  4k
I_Is   $N_0003 $N_0001 DC 2mA
R_Rc  0    $N_0003  1k
```

Lab Practice Problem

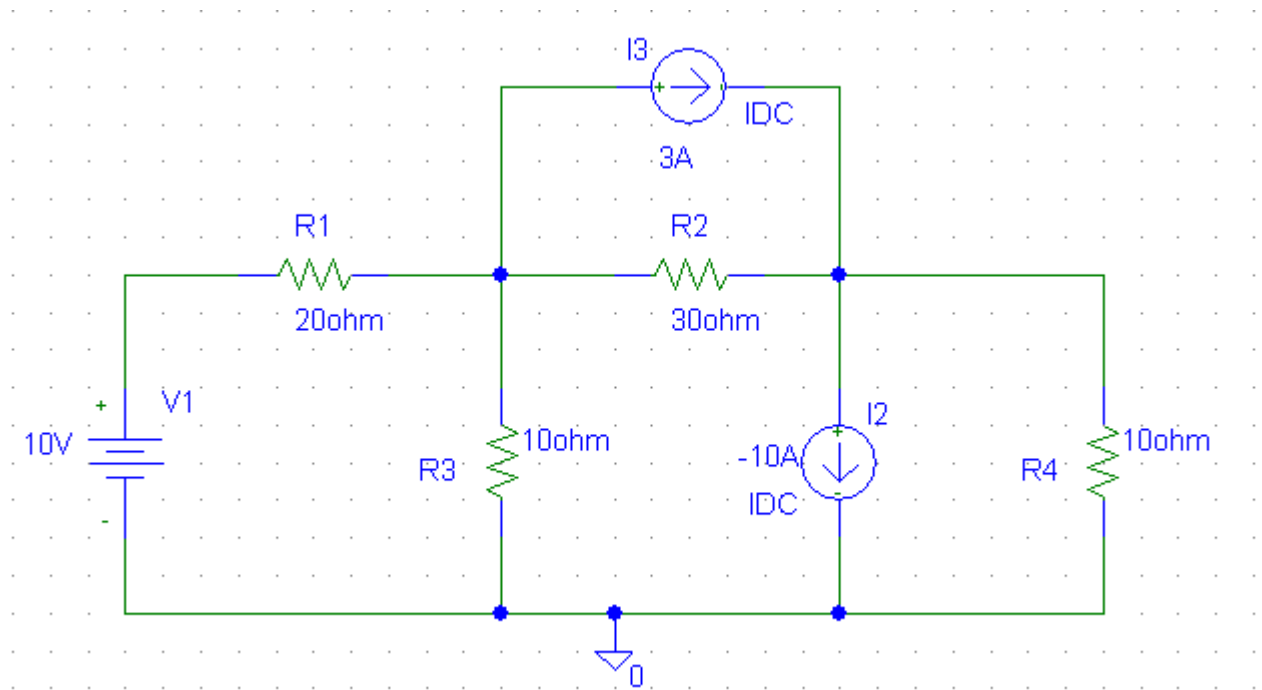


Figure 2. Circuit for lab practice

- (i) Draw the circuit as shown in Figure 2 using PSpice Schematic.
- (ii) Simulate the circuit and obtain the solution of all voltages and currents.
- (iii) Generate the netlist file and study how each circuit element is entered in the netlist.
- (iv) Take printouts of the schematic circuit diagram showing voltage and current results and the schematic netlist. Have the printouts signed by your instructor.

Post-Lab Report Question:

1. Theoretically calculate all the currents and the voltages for the circuit shown in Figure 2.
2. Compare the theoretical solution of the circuit shown in Figure 2 with the solutions obtained from PSpice.