

Course: CSE109 Electrical Circuits

Expt No.: 1

Title: Introduction to Circuit Elements and Variables

### Objectives:

1. To get familiar with circuit variables (voltage and current) and circuit elements (voltage source and resistance).
2. To learn how to measure dc voltage across a circuit element using a voltmeter.
3. To learn how to measure dc current through a circuit element using an ammeter.
4. To learn how to measure resistance of a resistor using a multimeter.
5. To verify Ohm's Law.

### Theory:

There are two types of elements in an electric circuit – active elements and passive elements. An active element supplies energy. A voltage source or a battery is an active element. The emf of a battery is measured using the unit volt (V). A passive element absorbs energy. A resistor is a passive element. The resistance of a resistor is measured using the unit Ohm ( $\Omega$ ).

There are two fundamental circuit variables – current through a circuit element and voltage across a circuit element. The current through a circuit element is measured using the unit Ampere (A) and the voltage across a circuit element is measured using the unit Volt (V).

A simple electric circuit is shown in Figure 1. The emf of the battery is  $E$  Volt and the resistance of the resistor is  $R \Omega$ . The current drawn from the battery and the current passing through the resistor are same and is  $I$  A. The voltage drop across the resistor is  $V$  Volt. The voltage drop across the resistor is exactly equal to the emf of the battery, that is,  $E = V$ .

The Ohm's Law states that  $V = IR$ . If we plot  $V$  vs.  $I$  (taking  $I$  as independent variable), we have a straight line passing through the origin and the slope of the line is  $R$ .

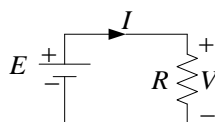


Figure 1: A simple electric circuit.

An ammeter is used to measure current and a voltmeter is used to measure voltage. As shown in Figure 2, an ammeter is connected in series with an element, current through which is to be measured. A voltmeter is connected in parallel with an element, voltage across which is to be measured. If you connect an ammeter in parallel with an element, the meter will be damaged. If you connect a voltmeter in series with an element, it will not give you correct result. So, **make sure that an ammeter is not connected in parallel and a voltmeter is not connected in series.**

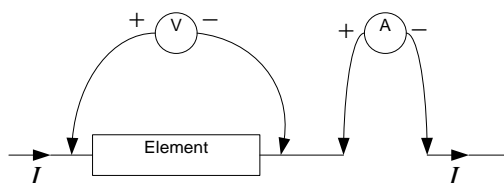


Figure 2. Connection of ammeter and voltmeter.

### Circuit Diagram:

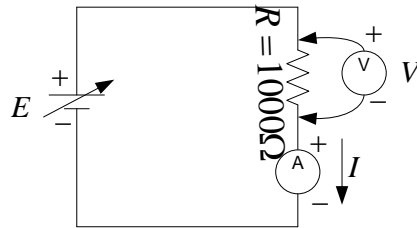


Figure 3. Circuit for experiment.

### Pre-Lab Report Question:

1. Theoretically calculate the values of  $I$  for the circuit of Figure 3 for  $E = 5, 6, 7, 8, 9, 10$  V and  $R = 1000\Omega$ .

### Equipments and Components Needed:

1. DC power supply
2. DC ammeter
3. DC voltmeter
4. Multimeter
5. Resistor  $1000\Omega$
6. Breadboard
7. Connecting wires

### Lab Procedure:

1. Measure the resistance of the resistor supplied using a multimeter and record it in Table 1.
2. Construct the circuit of Figure 3. Set the value of  $E$  at 5, 6, 7, 8, 9, and 10 volts and measure the corresponding  $V$  and  $I$  and record them in Table 1.
3. Have the datasheet signed by your instructor.

Table 1. Experimental Datasheet.

Observation number	Set Value of $E$ (V)	Measured Value of $V$ (V)	Measured Value of $I$ (mA)	Measured Value of $R$ ( $\Omega$ )
1	5			
2	6			
3	7			
4	8			
5	9			
6	10			

### Post-Lab Report Questions:

1. Theoretically calculate the values of  $I$  using measured values of  $V$  and  $R$ . Compare the theoretical values with the measured values and comment on any discrepancy.
2. Theoretically calculate the values of  $R$  from the measured values of  $V$  and  $I$  using Ohm's law. Compare the calculated and measure values of  $R$  and comment on any discrepancy.
3. Compare the set value of  $E$  and the measured value of  $V$  and comment on any discrepancy.
4. Plot  $V$  vs.  $I$  (taking  $I$  as independent variable) and fit a straight-line passing through the origin. From the plot determine the resistance of the supplied resistor using Ohm's law. Compare this value with the measured value and comment on any discrepancy.
5. Discuss how voltage or current is measured using a multi-range meter.

Course: CSE109 Electrical Circuits

Expt No.: 2

Title: Series-Parallel DC Circuit and Verification of Kirchhoff's Laws

### Objectives:

1. To learn analysis of dc series-parallel circuit.
2. To verify Kirchhoff's Voltage Law (KVL).
3. To verify Kirchhoff's Current Law (KCL).

### Theory:

Kirchhoff's Voltage Law (KVL) states that **the sum of the voltage rises around a closed path is equal to the sum of the voltage drops**. The KVL can be written in the following mathematical form:

$$\sum V_{\text{rises}} = \sum V_{\text{drops}} .$$

The sum of the voltage rises and the sum of the voltage drops are to be calculated in a given direction (normally in the clockwise direction). For example, in the simple series circuit of Figure 1, there are two voltage sources ( $E_1$  and  $E_2$ ) and two resistors ( $R_1$  and  $R_2$ ). The voltage drops across the two resistors are  $V_1$  and  $V_2$ , respectively. If we write KVL equation for the clockwise direction, then the KVL equation will be

$$E_1 - E_2 = V_1 + V_2 .$$

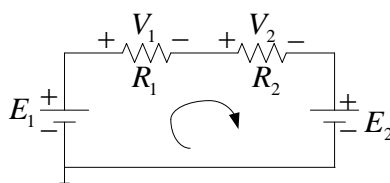


Figure 1. A simple series dc circuit.

Kirchhoff's Current Law (KCL) states that **the sum of the currents entering a node of a circuit is equal to the sum of the currents leaving the node**. The KCL can be written in the following mathematical form:

$$\sum I_i = \sum I_o .$$

For example, in the simple parallel circuit of Figure 2, there is a voltage source ( $E$ ) and two resistors ( $R_1$  and  $R_2$ ). The source current drawn from the voltage source is  $I_s$ . The currents through resistors  $R_1$  and  $R_2$  are  $I_1$  and  $I_2$ , respectively. If we consider the node  $a$  of the circuit, then  $I_s$  is entering the node and  $I_1$  and  $I_2$  are leaving the node. Then, the KCL equation for the node  $a$  is

$$I_s = I_1 + I_2 .$$

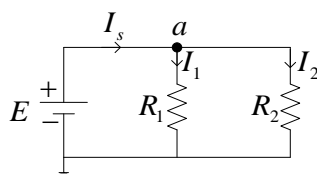


Figure 2. A simple parallel dc circuit.

A series-parallel circuit is one that is formed by a combination of series and parallel resistors. For solving series-parallel circuit, parallel combinations of resistors and series combination of resistors are clearly identified. Then series-parallel reduction method is used to determine the values of the circuit variables. For example, in the simple series-parallel circuit of Figure 3, the resistors  $R_2$  and  $R_3$  are in parallel and this parallel combination is in series with the resistor  $R_1$ . As the resistors  $R_2$  and  $R_3$  are in parallel,  $V_2 = V_3$ . Let  $R_p = R_2 \parallel R_3$ . Then, the equivalent resistance of the series-parallel combination is  $R_{eq} = R_1 + R_p$ . Now, the circuit variables can be calculated using the formulas

$$I_1 = \frac{E}{R_{eq}}$$

$$V_1 = I_1 R_1$$

$$V_2 = V_3 = I_1 R_p$$

$$I_2 = \frac{V_2}{R_2}$$

$$I_3 = \frac{V_3}{R_3}$$

The KVL equations for the circuit of Figure 3 can be written as

$$E = V_1 + V_2$$

$$E = V_1 + V_3$$

The KCL equation for the circuit of Figure 3 can be written as

$$I_1 = I_2 + I_3$$

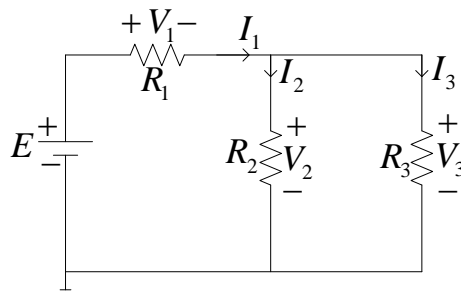


Figure 3. A simple series-parallel dc circuit.

### Circuit Diagram:

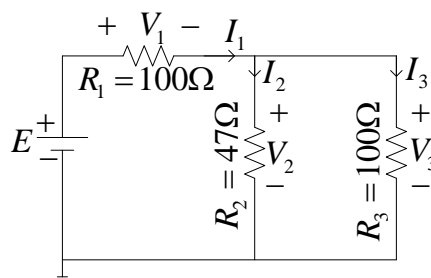


Figure 4. Circuit for experiment.

**Pre-Lab Report Questions:**

1. Theoretically calculate the values of  $V_1$ ,  $V_2$ ,  $V_3$ ,  $I_1$ ,  $I_2$ , and  $I_3$  of the circuit of Figure 4 with  $E = 3\text{V}$ .
2. From the calculated values, show that (i)  $V_2 = V_3$ , (ii) KVL holds, that is,  $E = V_1 + V_2$ , and (iii) KCL holds, that is,  $I_1 = I_2 + I_3$ .

**Equipments and Components Needed:**

1. DC power supply
2. DC voltmeter
3. DC ammeter
4. Multimeter
5. Resistor  $100\Omega$  (two) and  $47\Omega$  (one)
6. Breadboard
7. Connecting wires

**Lab Procedure:**

1. Measure the resistance values of the resistors supplied and record them in Table 1.
2. Construct the circuit of Figure 4. Set the value of  $E$  at 3 V. Measure the values of  $E$ ,  $V_1$ ,  $V_2$ ,  $V_3$ ,  $I_1$ ,  $I_2$ , and  $I_3$  and record them in Table 1.
3. From experimental data, (i) show that  $V_2 = V_3$ , (ii) verify KVL, that is,  $E = V_1 + V_2$ , and (iii) verify KCL, that is,  $I_1 = I_2 + I_3$ .

Table 1. Experimental Datasheet

Measured Value of $E$ (V)	Measured Value of $V_1$ (V)	Measured Value of $V_2$ (V)	Measured Value of $V_3$ (V)	Measured Value of $I_1$ (mA)	Measured Value of $I_2$ (mA)	Measured Value of $I_3$ (mA)	Measured Value of Resistances ( $\Omega$ )
							$R_1 =$ $R_2 =$ $R_3 =$

4. Have the datasheet signed by your instructor.

**Post-Lab Report Questions:**

1. Calculate the values of  $V_1$ ,  $V_2$ ,  $V_3$ ,  $I_1$ ,  $I_2$ , and  $I_3$  of the circuit of Figure 4 using measured values of  $E$ ,  $R_1$ ,  $R_2$ , and  $R_3$ . Compare the calculated values with the measured values and give reason if any discrepancy is found.
2. From the calculated values of  $V_1$ ,  $V_2$ ,  $V_3$ ,  $I_1$ ,  $I_2$ , and  $I_3$ , show that (i)  $V_2 = V_3$ , (ii)  $E = V_1 + V_2$  (KVL), and (iii)  $I_1 = I_2 + I_3$  (KCL).

**Course: CSE109 Electrical Circuits**

**Expt No.: 3**

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

**Objectives:**

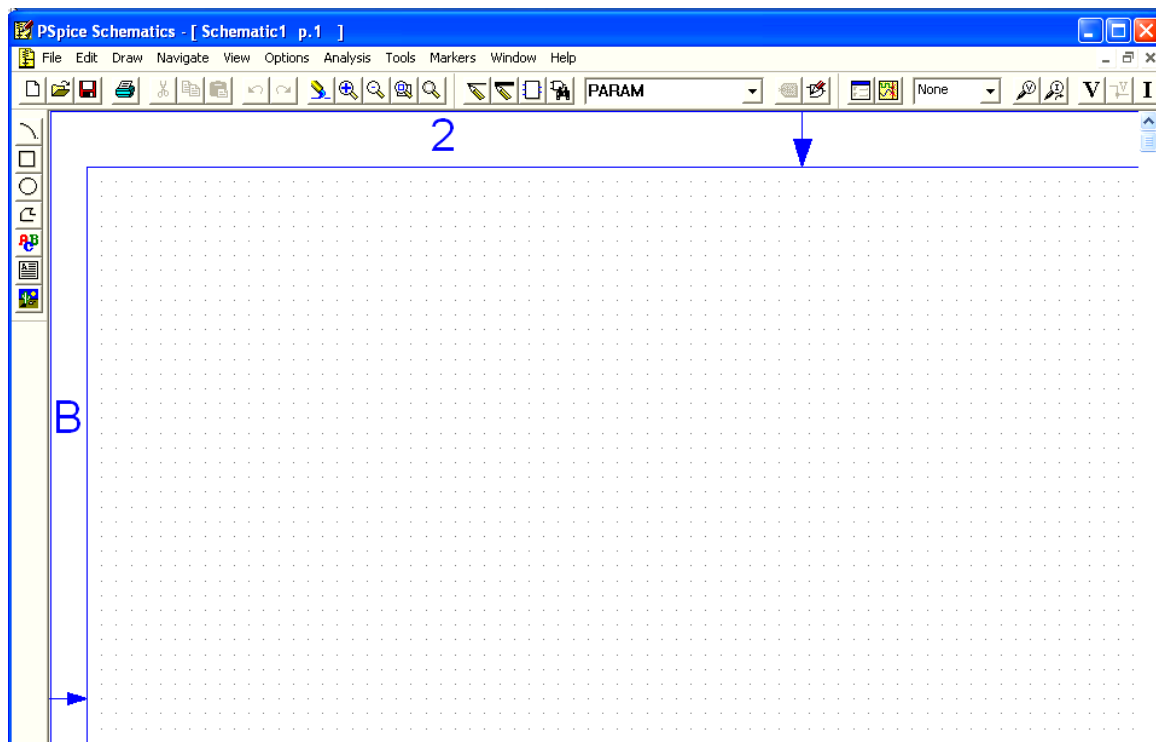
1. To learn fundamentals of PSpice.
2. To analyze Bias Point Detail of DC circuit using PSpice Schematics.


**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 .

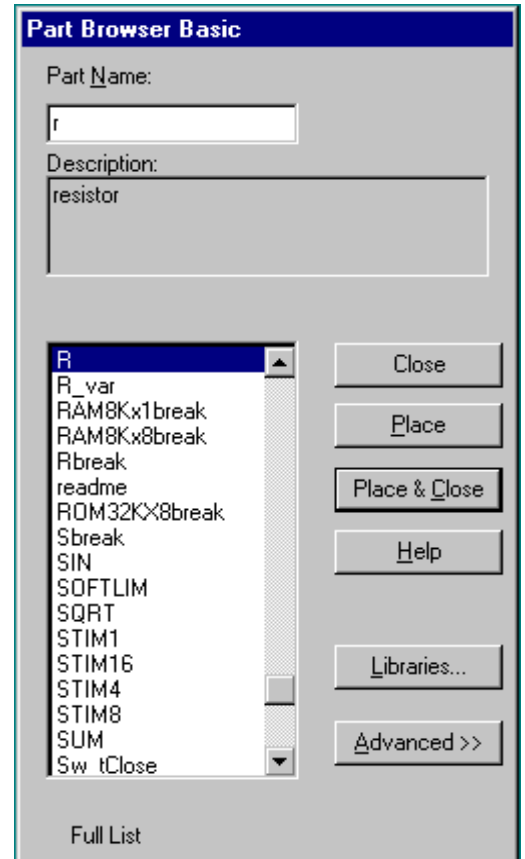
Once the “Part Browser Basic” window is open, select a part that you want in your circuit. This can be done by typing in the name or scrolling down the list until you find it.

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.

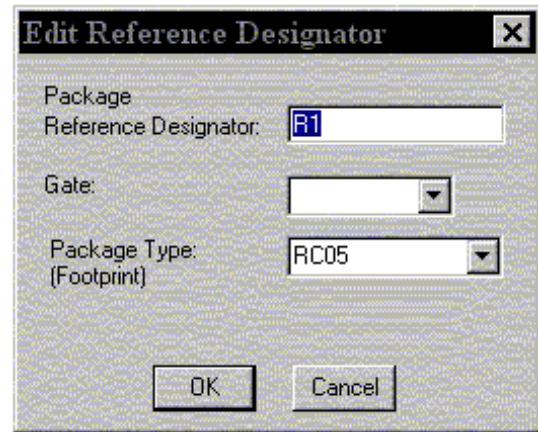
If you want to make a node (to make a wire go more than one place), click somewhere on the wire and then click to the part (or the other wire). Or you can go from the part to the wire.

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

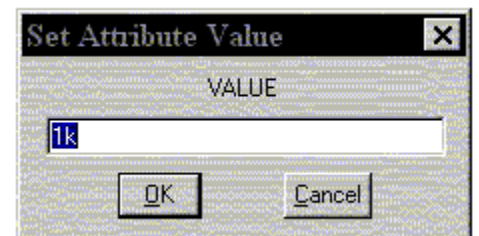
If you end up with extra dots near your parts, you probably have an extra wire, select this short wire (it will turn red), then press "Delete".

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.




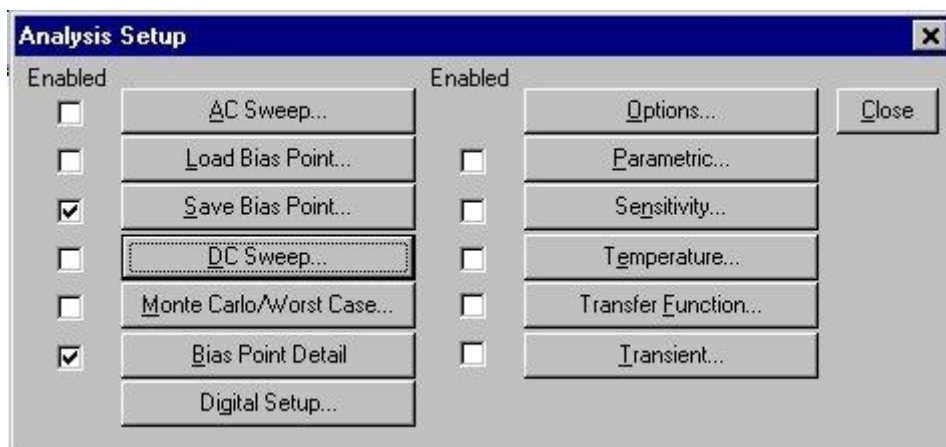
- Make sure you have a GND. This is very important. You cannot do any simulation on the circuit if you don't have a ground. If you aren't sure where to put it, place it near the negative side of your voltage source.
- Place Voltage and Current Bubbles. These are important if you want to measure the voltage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

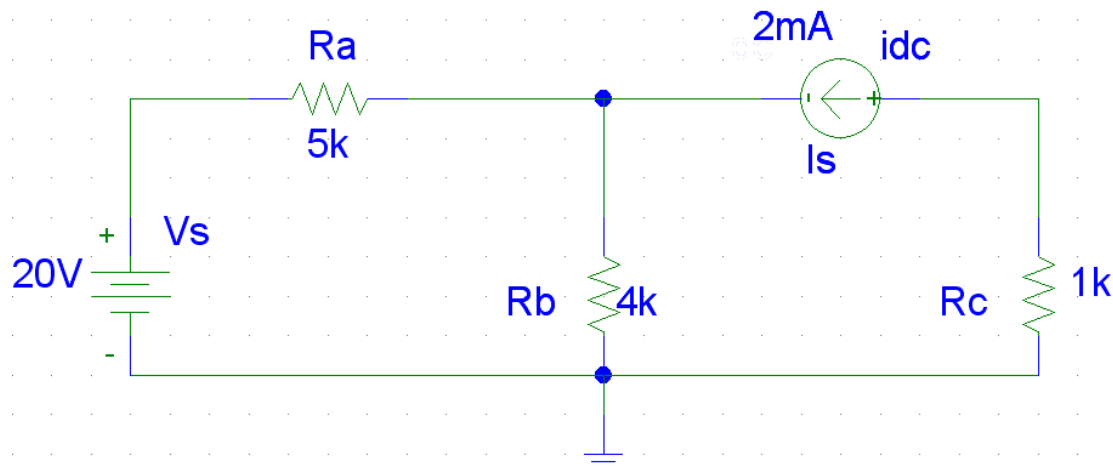
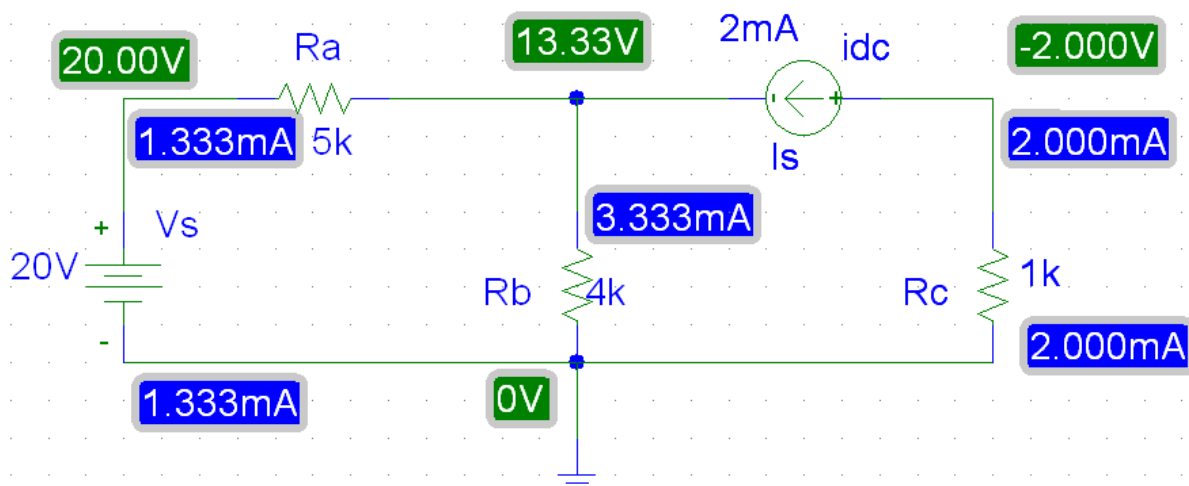


Figure 1: Example circuit.

(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

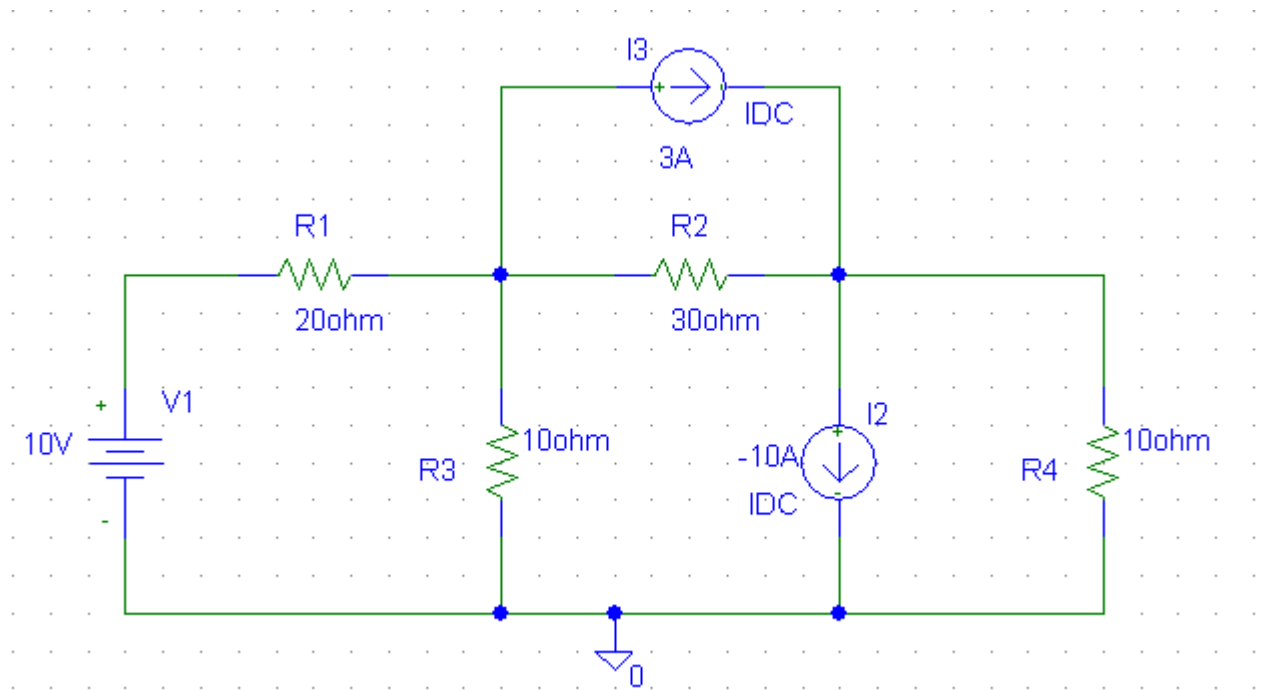


Figure 2. Circuit for lab practice

- (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.

**Course: CSE109 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.

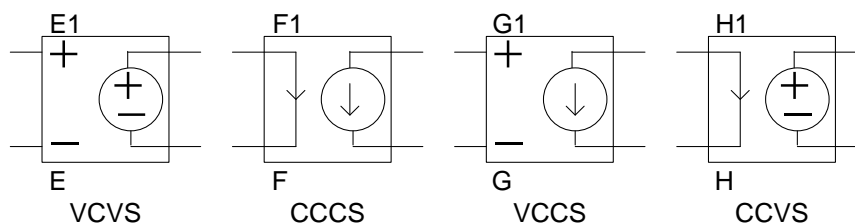


Figure 1. Shapes of dependent sources in PSpice Schematics.

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

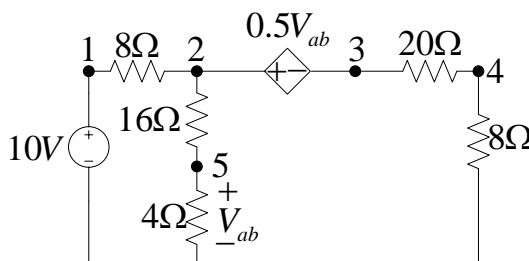


Figure 2. An example circuit with VCVS.

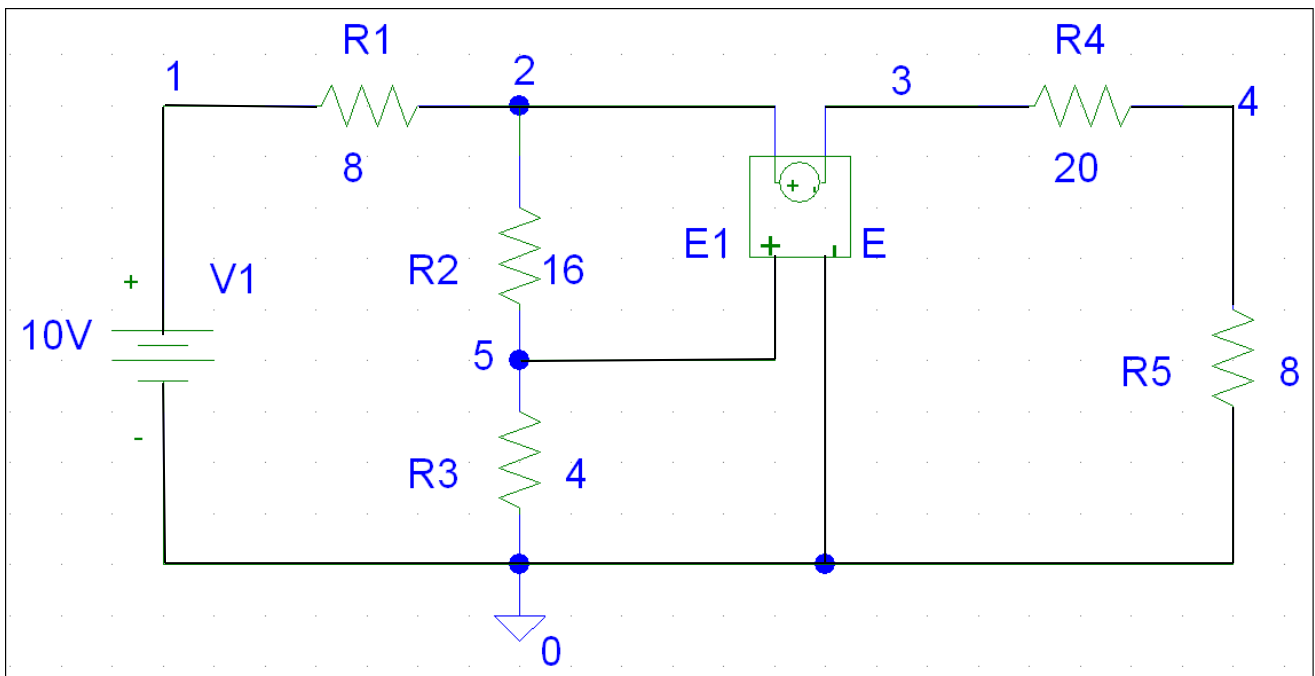
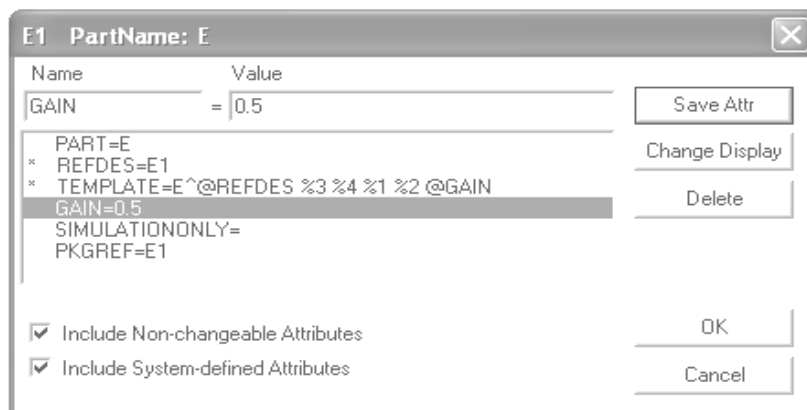


Figure 3. PSPice Schematic diagram for circuit of Figure 2 with VCVS.

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.

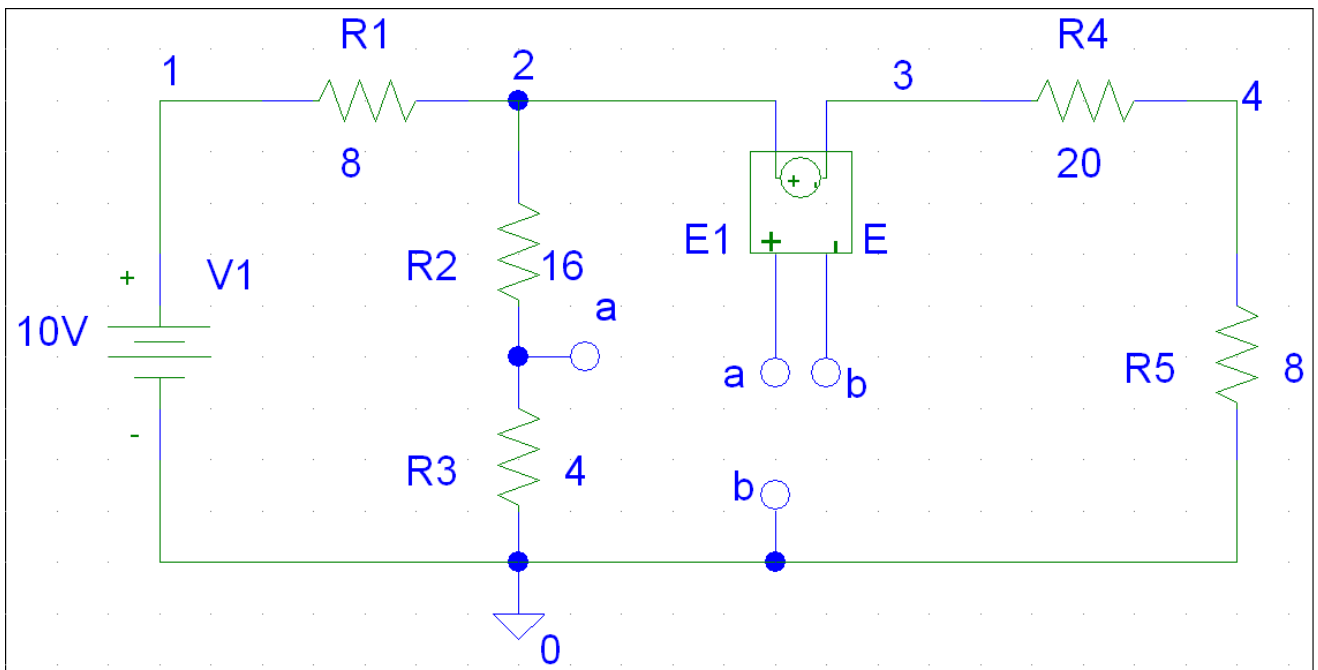


Figure 4. PSpice Schematic diagram with connection bubbles for circuit of Figure 2 with VCVS.

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

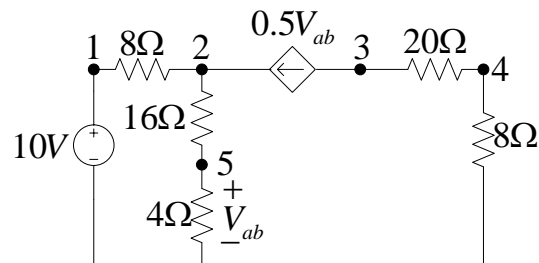


Figure 5. An example circuit with VCCS.

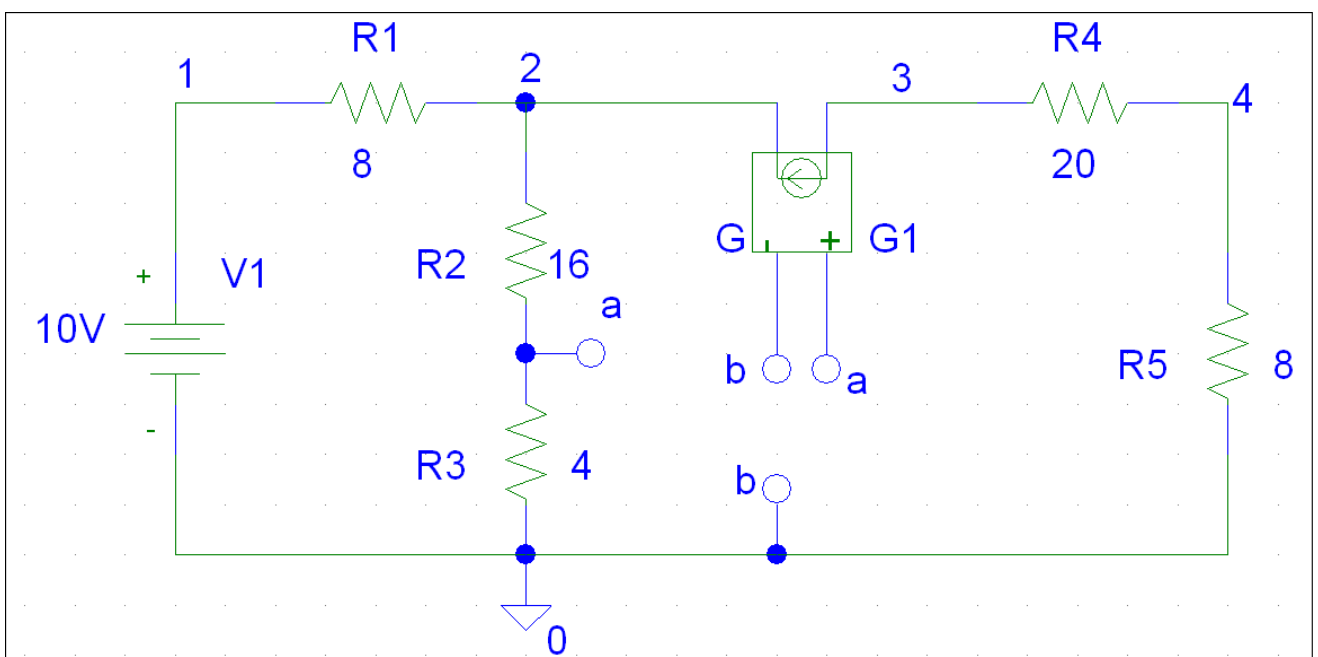


Figure 6. PSpice Schematic diagram with connection bubbles for circuit of Figure 5 with VCCS.

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

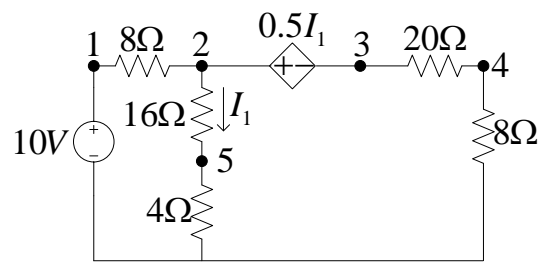


Figure 7. An example circuit with CCVS.

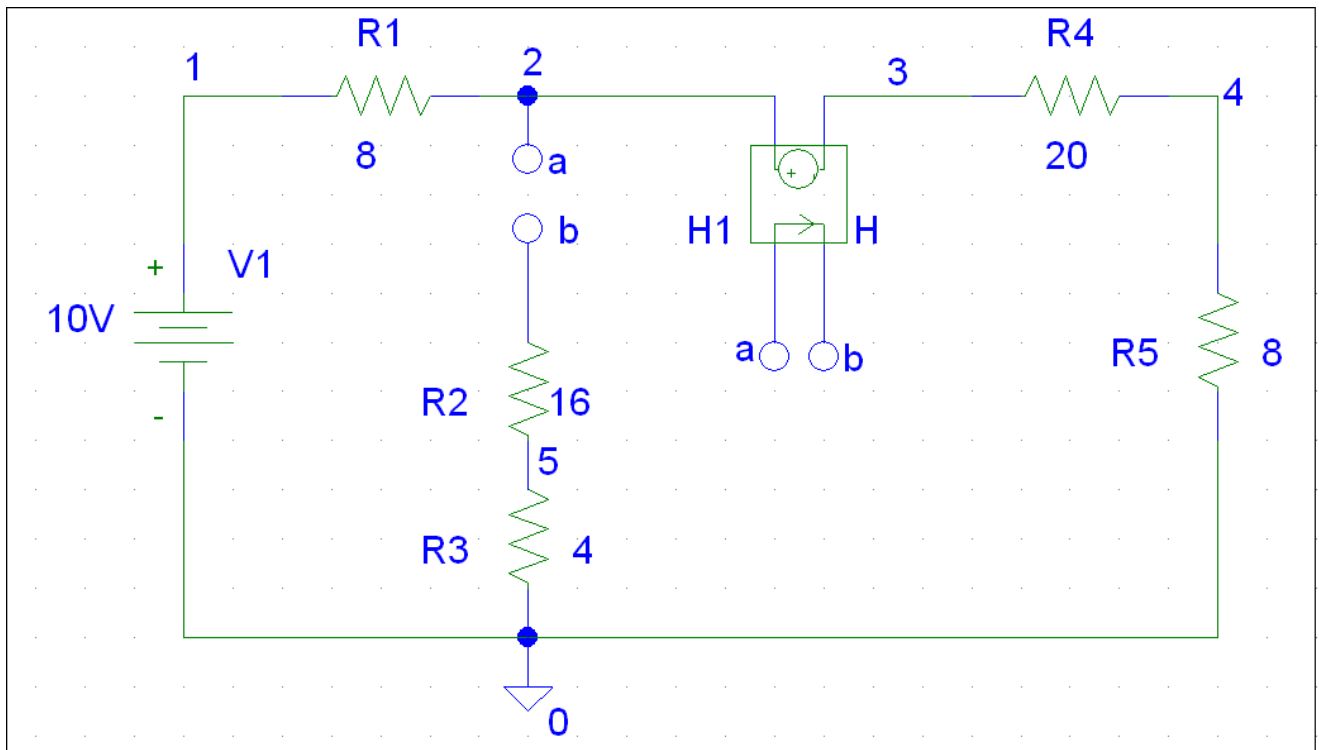


Figure 8. PSpice Schematic diagram with connection bubbles for circuit of Figure 7 with CCVS.

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

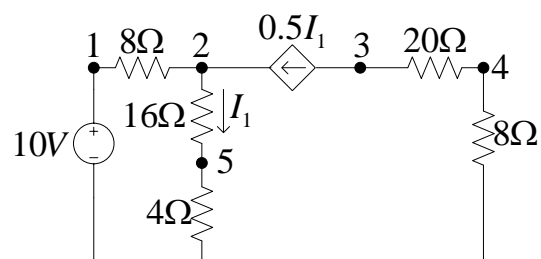


Figure 9. An example circuit with CCCS.

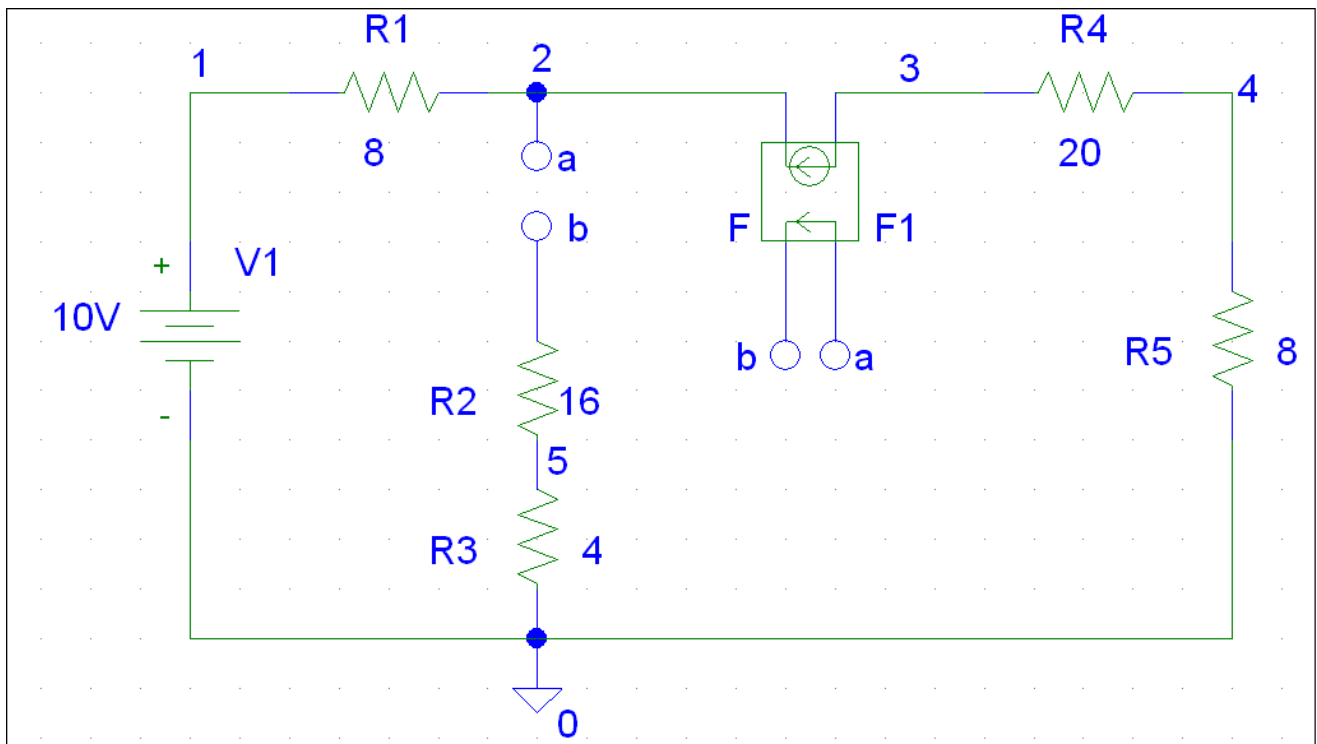


Figure 10. PSpice Schematic diagram with connection bubbles for circuit of Figure 9 with CCCS.

**Lab Practice problem:**

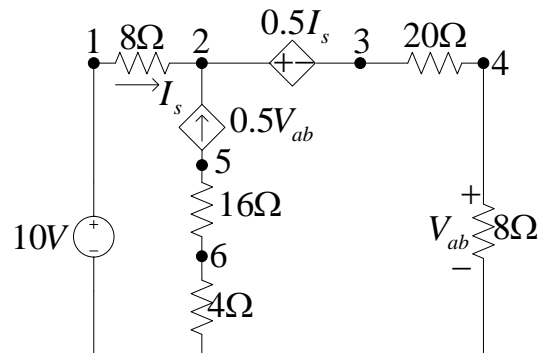


Figure 11. A circuit with VCCS and CCVS.

- 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.

**Course: CSE109 Electrical Circuits**

**Expt No.: 5**

**Title: Verification of Superposition Theorem**

**Objective:**

1. To verify the superposition theorem theoretically, experimentally, and using PSpice simulation.

**Theory:**

Superposition theorem works for linear circuits. The superposition theorem states that if a linear circuit contains more than one source, the voltage across or the current through any element may be determined by algebraically adding the contribution of each source acting alone with other sources remaining inactive. A voltage source is made inactive by setting its voltage value to zero (or by replacing it with a short circuit).

**Circuit Diagrams:**

$$E_1 = 10V \quad E_2 = 5V \quad E_3 = 5V$$

$$R_1 = 33\Omega \quad R_2 = 47\Omega \quad R_3 = 33\Omega \quad R_4 = 47\Omega \quad R_5 = 47\Omega \quad R_L = 68\Omega$$

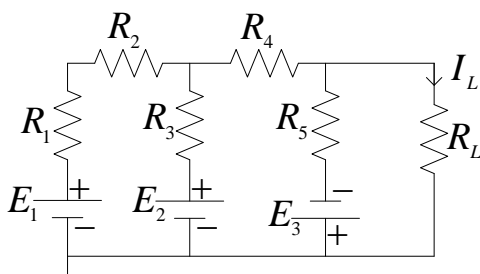


Figure 1. Circuit with all sources active.

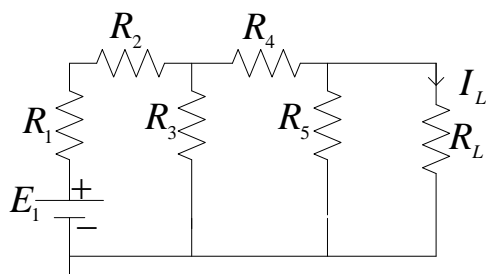


Figure 2. Circuit with  $E_1$  source active.

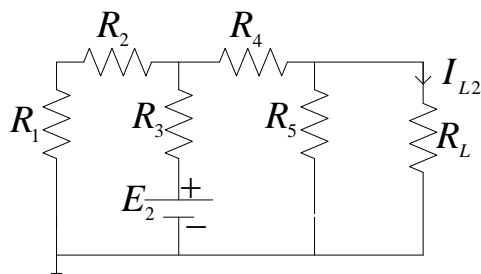


Figure 3. Circuit with  $E_2$  source active.

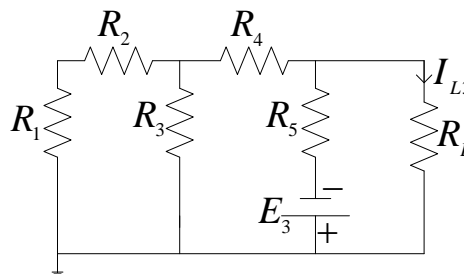


Figure 4. Circuit with  $E_3$  source active.

**Pre-Lab Report Question:**

1. Theoretically calculate the values of  $I_L$ ,  $I_{L1}$ ,  $I_{L2}$ , and  $I_{L3}$  of the circuits of Figures 1 through 4. From the calculated values, show that the superposition theorem holds, that is,  $I_L = I_{L1} + I_{L2} + I_{L3}$ .



**Equipments and Components Needed:**

1. DC power supply
2. Trainer board
3. DC ammeter
4. Multimeter
5. Resistors  $33\Omega$  (two),  $47\Omega$  (three),  $68\Omega$  (one)
6. Breadboard
7. Connecting wires

**Lab Procedure:**

1. Measure the resistance values of the given resistors and record them in Table 1.
2. Construct the circuit with all voltage sources active as shown in Figure. 1. For the  $E_1 = 10V$  source, use DC power supply. For the  $E_2 = 5V$  and  $E_3 = 5V$  sources, use the fixed voltage sources of the trainer board (be careful of the polarity of the voltage sources). Measure the values of the voltage sources and record them in Table 1. Measure  $I_L$  and record it in Table 1.
3. Construct the circuit with only voltage source  $E_1$  active as shown in Figure 2. This may be done by removing the voltage sources  $E_2$  and  $E_3$  from the circuit and replacing them with short circuits. **Caution: Do not try to replace any voltage source with a short circuit by directly connecting a wire across it. This will burn the trainer board.** Measure the value of  $I_{L1}$  and record it in Table 1. This is the current through the  $R_L = 68\Omega$  resistor when only the  $E_1 = 10V$  source is active.
4. Construct the circuit with only voltage source  $E_2$  active as shown in Figure 3. Measure the current  $I_{L2}$  and record it in Table 1. This is the current through the  $R_L = 68\Omega$  resistor when only the  $E_2 = 5V$  source is active.
5. Construct the circuit with only voltage source  $E_3$  active as shown in Figure 4. Measure the current  $I_{L3}$  and record it in Table 1. This is the current through the  $R_L = 68\Omega$  resistor when only the  $E_3 = 5V$  source is active (be careful of the polarity of this source).
6. From the experimental data, show that the superposition theorem holds, that is,  $I_L = I_{L1} + I_{L2} + I_{L3}$ .
7. Have the datasheet signed by your instructor.

Table 1. Experimental Datasheet.

Measured Value of $E_1$ (V)	Measured Value of $E_2$ (V)	Measured Value of $E_3$ (V)	Measured value of $I_L$ with all sources active (mA)	Measured value of $I_{L1}$ with only $E_1$ active (mA)	Measured value of $I_{L2}$ with only $E_2$ active (mA)	Measured value of $I_{L3}$ with only $E_3$ active (mA)	Measured values of resistors ( $\Omega$ )
							$R_1 =$ $R_2 =$ $R_3 =$ $R_4 =$ $R_5 =$ $R_L =$

**Post-Lab Report Questions:**

1. Calculate the values of  $I_L$ ,  $I_{L1}$ ,  $I_{L2}$ , and  $I_{L3}$  of the circuits of Figures 1 through 4 using the measured values of  $E_1$ ,  $E_2$ ,  $E_3$ ,  $R_1$ ,  $R_2$ ,  $R_3$ ,  $R_4$ ,  $R_5$ , and  $R_L$ . From the calculated values show that the superposition theorem holds. Compare these calculated values of currents with the experimental values and comment on any discrepancy observed.
2. Solve the circuits of Figures 1 through 4 using PSpice. Include the PSpice circuits with only currents shown. From the PSpice solution show that the superposition theorem holds. Compare the PSpice solutions with the theoretical solutions and comment on any discrepancy found.

Course: CSE109 Electrical Circuits

Expt No.: 6

Title: Verification of Thevenin's theorem

### Objective:

1. To verify the Thevenin's theorem theoretically, experimentally, and using PSpice simulation.

### Theory:

Thevenin's theorem states that a linear two-terminal network can be replaced by an equivalent circuit containing a voltage source  $E_{th}$  in series with a resistance  $R_{th}$ .  $E_{th}$  is equal to the open circuit voltage between the terminals and  $R_{th}$  is the ratio of the open circuit voltage to the short circuit current through the terminals. Experimentally,  $E_{th}$  may be measured by measuring the open circuit voltage and  $R_{th}$  can be calculated by measuring the open circuit voltage and the short circuit current.

### Circuit Diagrams:

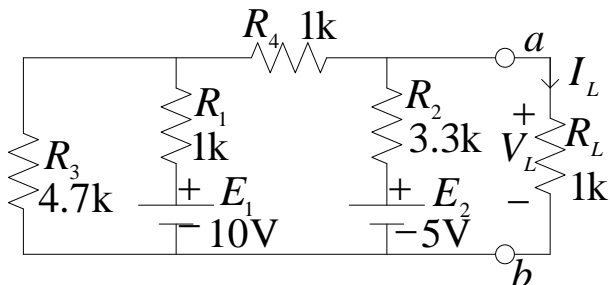


Figure 1. Circuit diagram whose Thevenin's equivalent to be determined.

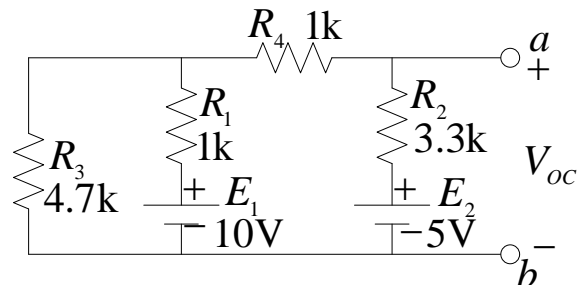


Figure 2. Circuit diagram to measure the open circuit voltage.

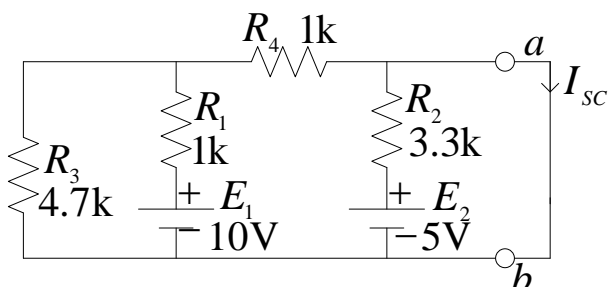


Figure 3. Circuit diagram to measure the short circuit current.

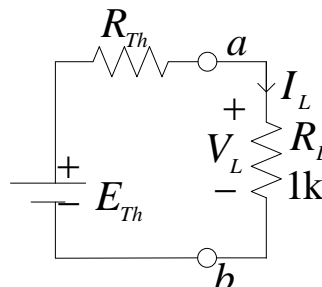


Figure 4. Circuit diagram to verify Thevenin's theorem.

### Pre-Lab Report Questions:

1. Theoretically calculate  $V_L$  and  $I_L$  in Figure 1. Then theoretically calculate  $V_{OC}$  in Figure 2 and  $I_{SC}$  in Figure 3. From the values of  $V_{OC}$  and  $I_{SC}$ , determine  $E_{th}$  and  $R_{th}$ . Theoretically calculate  $V_L$  and  $I_L$  in Figure 4. Verify the Thevenin's theorem from calculated data?

**Equipments and Components Needed:**

1. DC power supply
2. Trainer board
3. DC ammeter
4. Multimeter
5. Resistors (three  $1\text{k}\Omega$ , one  $4.7\text{k}\Omega$ , one  $3.3\text{k}\Omega$ )
6. Decade Resistance box
7. Breadboard
8. Connecting wires

**Lab Procedure:**

1. Measure the resistance values of the given resistors and record them in Table 1.
2. Construct the circuit of Figure 1. Use DC power supply for the 10V source and trainer board for the fixed 5V source. Measure  $E_1$ ,  $E_2$ ,  $V_L$ , and  $I_L$  and record them in Table 1.
3. Remove  $R_L$  as shown in Figure 2. Measure the open circuit voltage  $V_{OC}$  and record it in Table 1.
4. Connect nodes  $a$  and  $b$  with a wire as shown in Figure 3. Measure the short circuit current  $I_{SC}$  and record it in Table 1.
5. Determine  $E_{th} = V_{OC}$  and  $R_{th} = V_{OC}/I_{SC}$  and record them in Table 2.
6. Construct the circuit of Figure 4. Adjust the power supply voltage to make its value equal to  $E_{th}$ . Select  $R_{th}$  from the decade resistance box. Measure  $V_L$  and  $I_L$  and record them in Table 2.
7. Verify the Thevenin's theorem from data of Tables 1 and 2.
8. Have the datasheet signed by the instructor.

**Table 1.** Experimental Datasheet for determining Thevenin's equivalent circuit.

Measured Value of $E_1$	Measured Value of $E_2$	Measured Value of $V_L$	Measured Value of $I_L$	Measured value of $V_{OC}$	Measured value of $I_{SC}$	Measured values of resistors ( $\text{k}\Omega$ )
						$R_1 =$ $R_2 =$ $R_3 =$ $R_4 =$ $R_L =$

**Table 2.** Experimental Datasheet for Thevenin's equivalent circuit.

$E_{th} = V_{OC}$	$R_{th} = V_{OC}/I_{SC}$	Measured Value of $V_L$	Measured Value of $I_L$

**Post-Lab Report Questions:**

1. Theoretically calculate  $V_L$  and  $I_L$  in Figure 1 using measured values of  $E_1$ ,  $E_2$ ,  $R_1$ ,  $R_2$ ,  $R_3$ ,  $R_4$ , and  $R_L$ . Then theoretically calculate  $V_{OC}$  in Figure 2 and  $I_{SC}$  in Figure 3 using measured values of  $E_1$ ,  $E_2$ ,  $R_1$ ,  $R_2$ ,  $R_3$ ,  $R_4$ , and  $R_L$ . From the values of  $V_{OC}$  and  $I_{SC}$ , determine  $E_{Th}$  and  $R_{Th}$ . Theoretically calculate  $V_L$  and  $I_L$  in Figure 4 using calculated values of  $E_{Th}$  and  $R_{Th}$  and the measured value of  $R_L$ . Verify the Thevenin's theorem from calculated data?
2. Compare the measured values and the calculated values from step 1 and comment on any observed discrepancy.
3. Using PSpice, simulate the circuit of Figure 1 and determine  $V_L$  and  $I_L$ . Simulate the circuit of Figure 2 and determine  $V_{OC}$ . For this purpose, connect a 0A current source between nodes  $a$  and  $b$ . Simulate the circuit of Figure 3 and determine  $I_{SC}$ . For this purpose, connect a 0V voltage source between nodes  $a$  and  $b$ . Determine the values of  $E_{th}$  and  $R_{th}$ . Simulate the circuit of Figure 4 and determine  $V_L$  and  $I_L$ . Verify the Thevenin's theorem from simulated data.

Course: CSE109 Electrical Circuits

Expt No.: 7

Title: DC Circuit Analysis in PSpice using Source and Resistance Sweep

### Objectives:

1. To analyze DC circuit in PSpice by sweeping source and resistance.
2. To verify maximum power transfer theorem.

### Introduction:

In PSpice, DC analysis may be performed by varying the value of a DC voltage source or by varying a resistance. The results of such sweeps may be graphically viewed using the Probe tool of PSpice.

### Circuit Diagram:

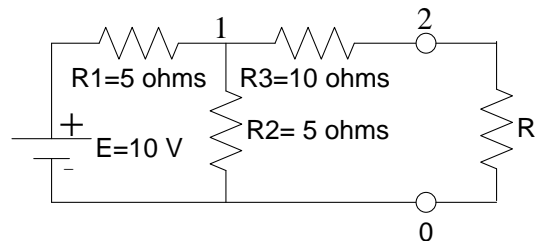


Figure 1. Example circuit.

### Pre-Lab Report Question:

1. Theoretically calculate the value of  $R_L$  for maximum power transfer in the circuit of Figure 1.
2. Theoretically calculate the node voltages  $V(1)$  and  $V(2)$  in the circuit of Figure 1. Also calculate the current  $I(R3)$  passing through the resistance  $R3$ .

### Lab Procedure:

1. Draw the circuit of Figure 1 (except the load resistance  $R_L$ ) in Schematics and simulate the circuit to determine the open circuit voltage  $V_{oc}$  between nodes 2 and 0. For this purpose, connect a 0A current source between nodes 2 and 0.
2. Remove the 0A current source from nodes 2 and 0. Connect a 0V voltage source between nodes 2 and 0. Simulate the circuit to determine the short circuit current  $I_{sc}$  flowing from node 2 to node 0.
3. Calculate  $E_{th} = V_{oc}$  and  $R_{th} = V_{oc} / I_{sc}$  from the simulations performed in steps 1 and 2.  $R_L = R_{th}$  for maximum power transfer
4. Remove the 0V voltage source from nodes 2 and 0. Connect a 10 Ohm resistance  $R_L$  between nodes 2 and 0. In the *Analysis Setup* dialog box, click the *DC Sweep* button. Select *Linear* type and *Voltage source* as a sweep variable. Write E as a sweep variable name with *Start value* = 0V, *End value* = 20V and *Increment* = 1V. Simulate the circuit. If simulation is successfully completed, a *Probe* window will appear with E being the x-axis. Click *Add Trace* in Probe window and select  $V(1)$  and  $V(2)$ . Add another plot to the window and add  $I(R3)$ . From the plots using the cursor, determine the values of  $V(1)$ ,  $V(2)$  and  $I(R3)$  at  $E = 10V$ .

5. Now, vary the resistance  $R_L$  and observe circuit variables as functions of  $R_L$ .
  - a. Double-click on the value label of the resistor  $R_L$ , which is to be varied. This will open a *Set Attribute Value* dialog box. Enter the name **{RVAR}** (including the curly braces) in place of the component value.
  - b. Choose *Get New Part* from the menu and select the part named *param*. Place the box anywhere on the schematic page. Double-click on the word *PARAMETERS* in the box title to bring up the parameter dialog box. Set the *NAME1*= **RVAR** (without the curly braces), which is the same name given to the resistor to be varied, and the *VALUE1*= 10 (or any other arbitrary value).
  - c. In the *Analysis Setup* dialog box, click the *DC Sweep* button and select *Linear* type and *Global Parameter* as a sweep variable. Type **RVAR** as a sweep variable *Name* with *Start value* = 1, *End value* = 20 and *Increment* = 0.1. Simulate the circuit and if simulation is successful, a Probe window will appear.
  - d. From the plots, determine  $V(1)$ ,  $V(2)$  and  $I(R3)$  for  $R = 10$  ohm.
  - e. Delete the existing plots and select  $I(R3)*I(R3)*RVAR$  for plot as a function of load resistance. Note that  $I(R3)*I(R3)*RVAR$  represents the load power. Determine the maximum value of the load power and the value of load resistance  $R_L$  for which the load power is the maximum. How does this value compare with  $R_{th}$  of the circuit?
6. Take printouts of the simulated circuits corresponding to steps 1 and 2. Also take printouts of the Probe plots corresponding to steps 4 and 5. Have the printouts signed by the instructor.

**Post-Lab Report Questions:**

1. Compare the values of  $V(1)$ ,  $V(2)$  and  $I(R3)$  obtained in steps 4 and 5(d).
2. Compare the load resistance  $R_L$  for maximum power transfer obtained in steps 2 and 5(e).
3. Compare the theoretical solutions with the solutions obtained from PSpice and comment on any observed discrepancy.

**Course: CSE109 Electrical Circuits**

**Expt No.: 8**

**Title: Experimental Study of Sinusoids and Their Characteristics**

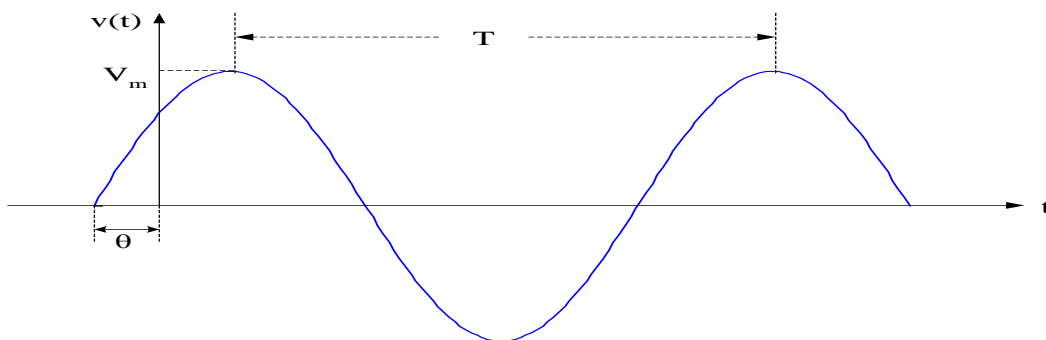
**Objectives:**

1. To observe the sinusoids in the oscilloscope using a simple RC circuit.
2. To read characteristics of the sinusoid from the oscilloscope and match the values with their corresponding measured values.

**Theory:**

Any sinusoid (voltage or current) is a periodic function of time and has positive value for half of the period and negative value for the rest half of the period. It is characterized by three parameters: (i) amplitude, (ii) frequency, and (iii) phase. A voltage sinusoid with amplitude  $V_m$ , period  $T$  ( $f = 1/T$ ), and phase  $\theta$  is shown in Figure 1. This can be mathematically expressed as  $v(t) = V_m \sin(\omega t + \theta)$ , where  $\omega = 2\pi f$ . The RMS value of the voltage sinusoid is

$V = \sqrt{\frac{1}{T} \int_0^T v^2 dt} = \frac{V_m}{\sqrt{2}}$ . Using AC voltmeter and ammeter, we can measure the RMS value of a sinusoid.

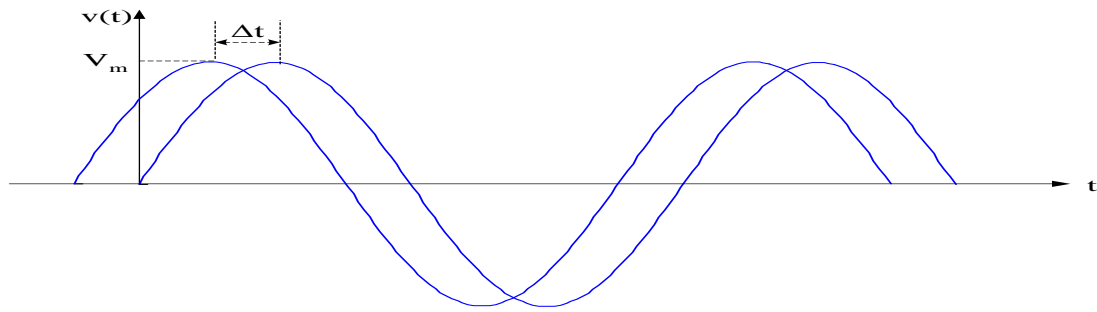


**Figure 1.** A sinusoidal voltage waveform.

**Measurement of Phase Difference in Oscilloscope**

The phase difference between two sinusoids (like voltage and current) can be measured in the oscilloscope by observing them in the dual mode. For this, observe one sinusoid in channel-1 and the other in channel-2. Turn on the cursor that measure  $\Delta T$  and place them between the adjacent peaks of the two sinusoids as shown in Figure 2. Measure  $\Delta t$  as shown in Figure 2. Now turn off either channel-1 or channel-2 and observe one sinusoid. Measure the time period  $T$  as shown in Figure 1. Calculate the phase difference in degree between the two sinusoids from  $\Delta\theta = \Delta t * 360^\circ / T$  or  $\Delta\theta = \Delta t * 360^\circ * f$ .



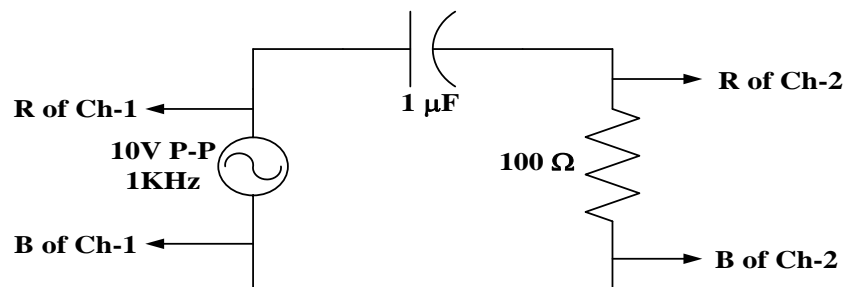


**Figure 2.** Illustration of how to measure the phase difference between two sinusoids.

### Impedance

Once you measure the amplitudes of the voltage and the current sinusoids and the phase difference between them, you can calculate the impedance of the circuit from  $Z = \frac{V_m}{I_m} \angle \theta$ .

### Circuit Diagram:



**Figure 3.** Circuit diagram for experiment.

### Pre-Lab Report Question:

1. Theoretically calculate the amplitude of the current flowing through the circuit shown in Figure 3 and the phase difference between the current and the input voltage.

### Equipments and Components Needed:

1. Resistance (100 Ω)
2. Capacitance (1 μF)
3. Multimeter
4. Connecting Wires
5. AC Voltmeter (0-3V range)
6. Signal generator
7. Oscilloscope

### Lab Procedure:

1. Measure the resistance value using multimeter and write it down in Table 1.
2. Turn on the oscilloscope. Turn on channel-1 and turn off channel-2. Push the GND button and set up the channel to reference ground by tuning the horizontal and vertical knobs.
3. Turn on the signal generator, change the signal to sine wave, and connect its output to the channel-1 of the oscilloscope.
4. Change the GND of channel-1 of the oscilloscope to DC mode and observe the signal.
5. Turn on the ΔV cursor of the oscilloscope and set the peak-to-peak value of the sinusoid to 10V by changing the AMPL knob of the signal generator.

6. Turn on the  $\Delta T$  cursor of the oscilloscope and set the frequency of the sinusoid to 1KHz or the time period T to 1mS by changing the FREQUENCY knob of the signal generator.
7. Connect the circuit shown in Figure 3 on the trainer board. The source of the circuit is the sine wave from the signal generator that you have set up in steps 4, 5, and 6.
8. Turn off channel-1 of the oscilloscope and turn on channel-2. Push the GND button and set up the channel to reference ground by tuning the horizontal and vertical knobs.
9. Connect channel-2 of the oscilloscope across the resistance as shown in Figure 3.
10. Change the GND of channel-2 of the oscilloscope to DC mode and observe the signal. This is the voltage sinusoid across the resistance. Measure the peak value and divide it by the measured value of the resistance. This is the amplitude of current. Write it down in Table 1.
11. Connect channel-1 of the oscilloscope as shown in Figure 3 and observe both the channels in dual mode. Measure the phase difference and write it down in Table 1. Also write which signal leads.
12. Using the AC voltmeter, measure the voltage across the resistance, across the capacitor, and across the source and write them down in Table 2.
13. Divide the voltage across the resistance by the measured value of the resistance. This is current through the resistance. Write it down in Table 2.

Table 1. Experimental Data from Oscilloscope.

Measured value of resistance ( $\Omega$ )	Set peak-to-peak value of source voltage (V)	Set source frequency (KHz)	Measured peak value of current through resistance (mA)	Measured phase difference between voltage and current (deg)	Which signal is leading?

Table 2. Experimental Data from Meter Reading.

Measured RMS value of source voltage (V)	Measured RMS value of voltage across capacitor (V)	Measured RMS value of voltage across resistance (V)	RMS value of current through resistance (mA)

### Post-Lab Report Questions:

1. Divide the amplitude of the signal generator voltage measured by the oscilloscope by  $\sqrt{2}$  and compare it with the measured RMS value by voltmeter.
2. Divide the amplitude of the current measured by the oscilloscope by  $\sqrt{2}$  and compare it with the measured RMS value.
3. Calculate the impedance by the measured values of voltage and current from the oscilloscope. Also calculate the impedance from  $Z = R - jX_C$ .
4. Calculate the impedance angle from the expression  $\tan^{-1}(X_C/R)$  and compare it with the phase difference measured from the oscilloscope.

**Course: CSE109 Electrical Circuits**

**Expt No.: 9**

**Title: AC Circuit Analysis using PSpice Schematics**

**Objective:**

1. To analyze simple AC circuit using PSpice Schematics.

**Circuit Diagram:**

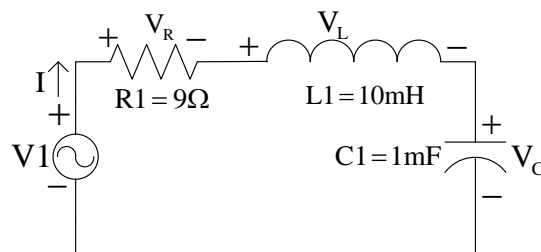


Figure 1. Example circuit.

**Pre-Lab Report Question:**

1. In Figure 1,  $V1 = 10\angle 0^\circ \text{V}$  and  $\omega = 200\pi \text{ rad/sec}$ . Calculate the current  $I$ , voltage across the resistance  $V_R$ , voltage across the inductance  $V_L$ , and voltage across the capacitance  $V_C$ . Determine the phase difference between the current  $I$  and the input voltage  $V1$  and between the current  $I$  and the voltages  $V_R$ ,  $V_L$ , and  $V_C$ .

**Lab Procedure:**

1. Draw the circuit as shown in the Figure 1 in PSpice Schematics window. The voltage source  $V1$  in the circuit is  $VSIN$ .
2. Double click on the voltage source and set  $VOFF = 0$ ,  $VAMPL = 10$ , and  $FREQ = 100$ . Save the settings.
3. Click on Analysis → Setup. Click on Transient. Set Print Step = 1ns and Final Time = 30ms. Click OK then close. Save the circuit setup.
4. Simulate the circuit and observe the input voltage signal ( $V1$ ) shape. To plot the input signal, click on add trace and select  $V(V1:+) - V(V1:-)$ .
5. Keep the input voltage signal in the plot. Add the current signal multiplied by 10 ( $I(R1)*10$ ) in the plot. Multiplication of the current signal by 10 is for better view with the voltage signal. Determine the phase difference between voltage and current. Write it down. Also write which signal is leading. To calculate the phase difference, determine the time difference  $\Delta t$  between the two signal and then obtain the phase difference from  $\Delta\theta = 360*\Delta t*f$ .
6. Delete both the voltage and current signals from the plot window. Now add the current signal  $I(R1)$  in the plot and measure its amplitude. Write it down.
7. Multiply the current signal by 10 ( $I(R1)*10$ ) and add the voltage signal across the resistance ( $V(R1:1) - V(R1:2)$ ). Find the phase difference between the two signals and write it down.
8. Remove the voltage signal across the resistance from plot and keep the current signal. Add the voltage signal across the inductance ( $V(L1:1) - V(L1:2)$ ). Find the phase difference between the two signals and write it down. Also write which signal is leading.

9. Remove both the signals from plot. Now add the current signal  $I(R1)$  and the voltage signal across the capacitance ( $V(C1:1) - V(C1:2)$ ). Find the phase difference between the two signals and write it down. Also write which signal is leading.
10. Remove all the signals from the probe window and add the RMS value of input voltage signal. To plot the RMS value of the input signal, click add trace and select  $\text{RMS}(V(V1:+) - V(V1:-))$ . Find the RMS value at around 20 ms and write it down.
11. Similarly, determine the RMS values of the voltage across the resistance, voltage across the inductance, voltage across the capacitance, and the current.

### Post-Lab Report Questions:

1. Calculate the impedance, both magnitude and phase angle, of the circuit from your readings of steps 5 and 6. Theoretically calculate the impedance and compare the two results.
2. Theoretically calculate the RMS values of current and the voltages across the resistance, across the inductance, and across the capacitance. Compare your calculated values with the readings taken in step 11.
3. Calculate the complex power consumed in the circuit from  $S = \frac{V_m I_m}{2} \cos(\Delta\theta) + j \frac{V_m I_m}{2} \sin(\Delta\theta)$  using your data from steps 5 and 6. Here  $V_m$  and  $I_m$  are the magnitudes of the input voltage and current, respectively and  $\Delta\theta$  is the phase difference between the current and the input voltage.
4. Calculate the power dissipated by the resistance from  $P = \frac{I_m^2 R}{2}$  using the data of step 6 and compare it with the real part of  $S$  calculated in question 3.