

Department of Electrical and Electronic Engineering  
Shahjalal University of Science and Technology

**EEE 126: Electrical Circuit Simulation Laboratory**

**Experiment # 03: Determination of equivalent resistance and Circuit analysis with source and resistance sweeping.**

**Objectives:** In this simulation lab, we shall learn

- How to determine equivalent resistance of a circuit.
- How to run independent source sweeps and plot the results.
- How to run resistance sweeps and plot the results.

**Introduction:** Various types of analysis such as dc sweep, ac sweep, transient, transfer function, sensitivity, parametric etc. can be done by using PSpice. Variation of voltages and currents in different nodes due to variation of any voltage, current, resistance or any model parameter can be observed graphically by using dc sweep analysis.

**Pre-lab:** For the circuit shown in fig.1

1. Determine the equivalent resistance between terminals 5 and 0.
2. Determine the current through  $R_5$ , voltages at node 3 and 5, if 10 ohm resistance is connected between node 5 and 0.

**Circuit diagram:**

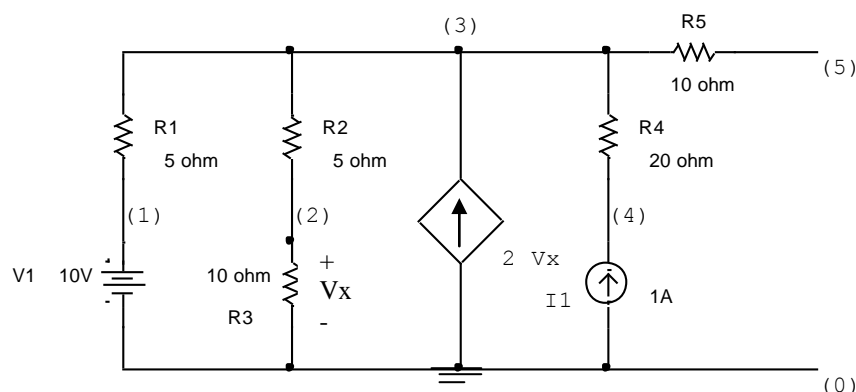


Fig.1

**A. Equivalent Resistance**

**Procedure:**

1. Place a 1A current source  $I_L$  between terminals 5 and 0 and inactivate all the independent sources (i.e. Set  $V1=0$ ,  $I_1=0$ ) in fig.1, and then write down the 'Schematic net list' in **Microsim Text Editor**. Save the file.
2. Open the file in **PSpiceAD**. If simulate successfully, select 'Examine output' option under the file menu of **PSpiceAD**. And find out the node voltage of node 5 from 'circuit description'. This voltage is the equivalent resistance between terminals 5 and 0.  
Is this equal to that you have calculated in Pre-lab.?
3. Place a 1A current source  $I_L$  between terminals 5 and 0 and inactivate all the independent sources (i.e. Set  $V1=0$ ,  $I_1=0$ ) in fig.1, and then draw the circuit diagram in **Microsim Schematic window**. Save the file.
4. Click Analysis Setup dialog box and select 'Bias point detail' button.
5. Simulate the schematic and Click the 'Enable Bias voltage display' icon from the **Microsim Schematic window**.
6. Find out voltage of node 5 from schematic, which is the equivalent resistance of the circuit.  
Is this equal to that you have calculated in Pre-lab.?

## B. Source sweeping

### Procedure:

1. Place a 10 ohm resistance  $R_L$  between terminals 5 and 0 in fig.1, and then write down the 'Schematic net list' in **Microsim Text Editor**.
2. Include the following in Microsim Text before .end

```
.DC LIN V1 0 20 1
.probe
```

which will sweep the source  $V_1$  from 0V to 20V in 1V increment. Save the file.
3. Open the file in **PSpiceAD**. If simulate successfully, select 'Run Probe' option under the file menu of **PSpiceAD**.
4. X-axis of the probe window is  $V_1$ . Click 'Add Trace' of the Probe window and select V (3), V (5) and I (R5). It will plot V (3) vs.  $V_1$ , V (5) vs.  $V_1$  and I (R5) vs.  $V_1$ .
5. From the plot determine V (3), V (5) and I (R5) for  $V_1=10$  volts.  
Are these values equal to that you have calculated in Pre-lab.?
6. Place a 10 ohm resistance  $R_L$  between terminals 5 and 0 in fig.1, and then draw the circuit diagram in **Microsim Schematic window**.
7. To give the node numbers in the schematic, double click in node 3 and Set attributes value Level= 3 .Similarly use Level=5 for node 5. Save the file.
8. In the Analysis Setup dialog box, click the 'DC Sweep' button and select 'Linear' type and 'Current source' as a sweep variable. Write  $I_1$  as a sweep variable 'Name' with Start value = 0A, End value = 5A and Increment = 1A.
9. Simulate the schematic. If simulate successfully, Probe window will appear. X-axis of the probe window is  $I_1$ . Click 'Add Trace' of Probe window and select V (3), V(5) and I(R5). It will plot V(3) vs  $I_1$ , V(5) vs  $I_1$  and I(R5) vs  $I_1$ .
10. From the plot determine V(3), V(5) and I(R5) for  $I_1=1$  amp.  
Are these values equal to that you have calculated in Pre-lab.?

## C. Resistance sweeping

### Procedure:

1. Write down the 'Schematic net list' in **Microsim Text Editor** of the circuit shown in fig.1.
2. To include a variable resistance between terminals 5 and 0, include the following text in Microsim Text before .end

```
RL 5 0 {R}; Make R variable
.param R=10; R has an arbitrary value
.dc param R 1 20 1; R will sweep from 1 ohm to
; 20 ohm in 1 ohm increments
.probe
```
3. Save and Open the file in **PSpiceAD**. If simulate successfully, select 'Run Probe' option under the file menu of **PSpiceAD**.
4. X-axis of the probe window is R that means  $R_L$ . Click 'Add Trace' of Probe window and select V (3), V (5), I (R5). It will plot V (3) vs. R, V (5) vs. R and I (R5) vs. R.
5. From the plot determine V (3), V (5) and I (R5) for R=10 ohm.  
Are these values equal to that you have determined before?
6. Delete the previous plot and select V (5)\*I (R5). It will plot Load power, V (5)\* I (R5) vs. R. Determine the value of R at which V (5)\* I (R5) is maximum.  
Is this equal to Equivalent resistance that you have determined before?  
Mention the name of theorem that is verified by this plot?
7. Place a 10 ohm resistance  $R_L$  in fig.1, and then draw the circuit diagram in **Microsim Schematic window**. Save the file.
8. First, double-click the value label of the resistor,  $R_L$  that 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. Choose "**Get New Part**" from the menu and select the part named **param**.
9. Place the box anywhere on the schematic page. Now double-click on the word **PARAMETERS** in the box title to bring up the parameter dialog box. Set the NAME1= **RVAR** (no curly braces) and the VALUE1= 10 (any value) to the nominal resistance value.
10. 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 = 1.

11. Simulate the schematic and if simulate successfully Probe window will appear
12. X-axis of the probe window is R that means  $R_L$ . Click 'Add Trace' of Probe window and select V (3), V (5), I (R5). It will plot V (3) vs. R, V (5) vs. R and I (R5) vs. R.
13. From the plot determine V (3), V (5) and I (R5) for R=10 ohm.  
Are these values equal to that you have determined before?
14. Delete the previous plot and select V (5)\*I (R5). It will plot Load power, V (5)\* I (R5) vs. R. Determine the value of R at which V (5)\* I (R5) is maximum.  
Is this equals to Equivalent resistance that you have determined before?

**Exercise :** (1) Plot I vs  $V_1$  by sweeping  $V_1$  from 0 to 20 volts in 2 volts increment for the circuit shown in Fig.2 by using **Microsim Schematic window**.  
Mention the name of law that is verified by this plot?

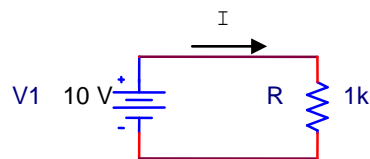


Fig.2

- (2) For the circuit shown in fig.3
  - (a) Assuming  $V_s = 0$  volts, sweep the value of I form 0A to 2A in 0.2 A increment and plot  $V_o$  vs I by using **Microsim Text editor**.
  - (b) Suppose, current source I is open from the terminals a and b. Determine the equivalent resistance between two terminals from the plot of (a)
  - (c) Suppose, current source I is replaced by a resistance. By using resistance sweeping, determine the value of resistance at which it will receive maximum power from the circuit. Also determine the maximum power.

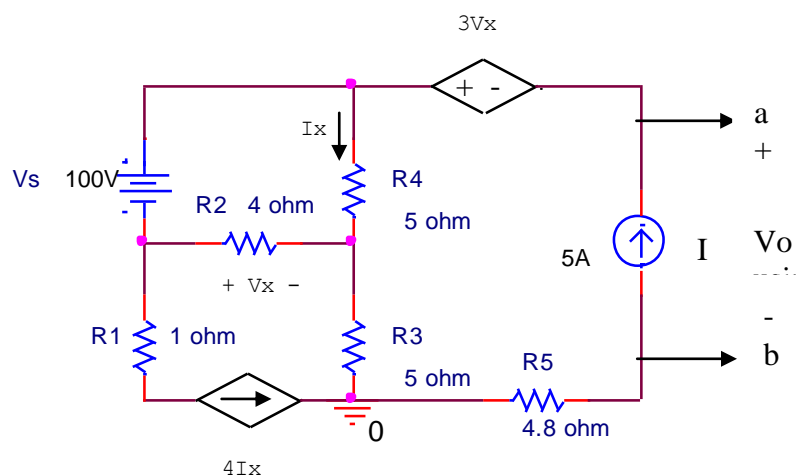


Fig.3