East West University
Department of Computer Science and Engineering

Course: CSE209 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:

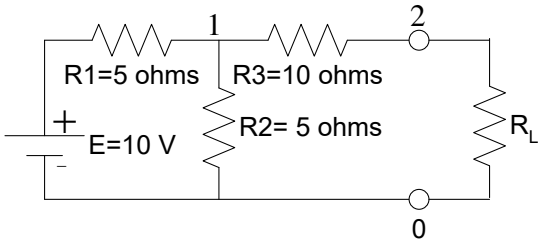

Figure 1. Example circuit.

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.