

ENGR 252 LAB #4 - Thevenin Equivalent and Sensitivity Analysis

"This lab & associated report should be completed individually"

Objective

Utilizing computer Aided Design and Analysis tools to find Thevenin equivalent of a circuit and perform sensitivity analysis.

Related Principles

- *Electrical Circuits* textbook by Nilsson
- *OrCAD Capture and PSpice Integrated Tutorial*

Equipments

- Windows-based PC
- Cadence orCAD 16.6 lite
http://www.engrCS.com/tools_programs/16.6_OrCAD_Lite_Capture&PSpice_Products.zip
- USB hard disk or other removable drives

Supplies

- None

Preparation

- Completion of Lab 3 preparations and experiments.
- Review of Thevenin and Norton equivalents
Thevenin and Norton equivalent requires finding short circuit current and open circuit voltage which will be used to find Thevenin Resistance $R_{th} = V_{th}/I_{sc}$.

In Simulation, short and open may cause undefined results, therefore:

- $R = 1\mu$ between the terminals is used to simulate short circuit in order to find I_N or I_{sc} .
- $C=1p$ between the terminals is used to simulate open circuit in order to find V_{th} or V_{oc} . As operating frequency increases, value of the capacitor may have to be decreased in order to ensure large enough impedance to simulate open circuit.

- Sensitivity Analysis

Identify the node to be analyzed by placing  on the node using "Place Off-Page Connector" from the tools menu. Change the connector name to Out.

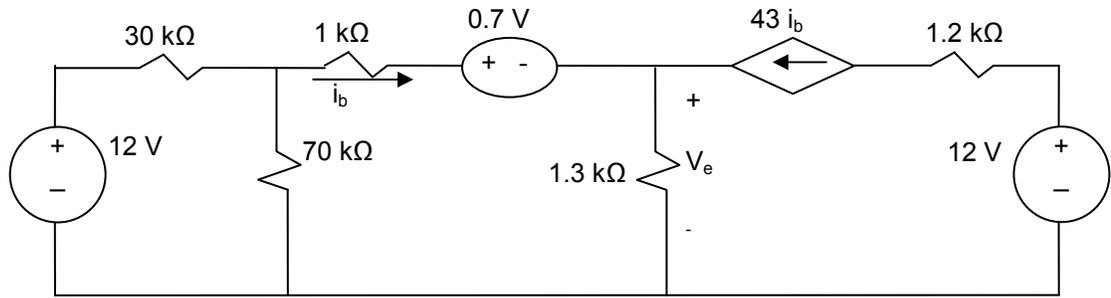
Configure the simulation to "Bias Point" Analysis and check the detailed bias point and sensitivity analysis check box with output variable set to V(Out).

Upon completion of simulation, use the View Output in PSpice window to view the sensitivity results under the heading "DC sensitivities of output V(Out)".

Experiment #1

When a voltmeter is used to measure the voltage V_e in the following circuit, it reads 5.5V.

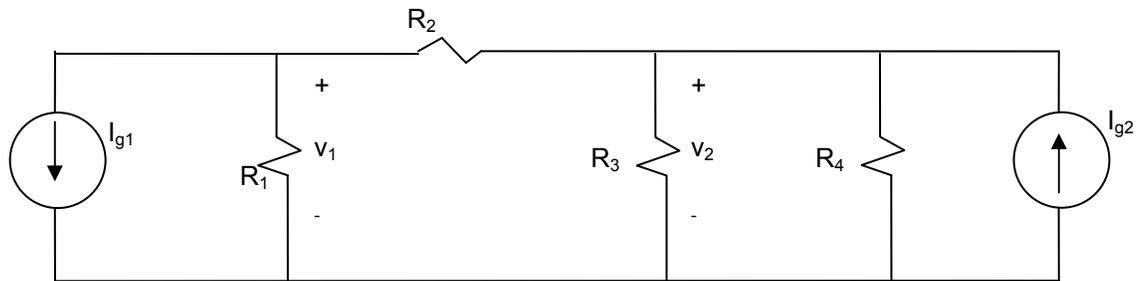
- What is the resistance of the voltmeter?
- What is the percentage of error in the voltmeter measurement?



Experiment #2

Assume the nominal values for the components in the circuit in the following circuit are:

$R_1 = 25 \Omega$; $R_2 = 5 \Omega$; $R_3 = 50 \Omega$; $R_4 = 75 \Omega$;
 $I_{g1} = 12 \text{ A}$; $I_{g2} = 16 \text{ A}$;



Use PSpice to predict values of v_1 and v_2 if I_{g2} decreases to 11 A.

Experiment #3

For the circuit described in Experiment #2, determine the sensitivity level of V_2 to the variation of each of the Resistors (R_1, R_2, R_3, R_4). Determine which resistor V_2 is most sensitive to.

Note: For this exercise, sensitivity is measured as % voltage change divided by the % resistance change.

Report Requirements

This lab and report must be completed individually. All reports must be computer printed (Formulas and Diagrams may be hand drawn) and at minimum include:

For each Experiment

- a) Clear problem statement; specify items given and to be found.
- b) Identify the theory or process used.
- c) Documents resulting circuit schematics, Net list, simulation results, and other relevant results.

For the report as a whole

- a) Cover sheet with your name, class, lab, completion date and team members' names.
- b) Lessons Learned from the experiments.
- c) A new experiment and expected results which provide additional opportunity to practice the concepts in this lab.