CENG 215 Circuits and Electronics

LAB #2 Feuille

Place: PC Lab

Aim

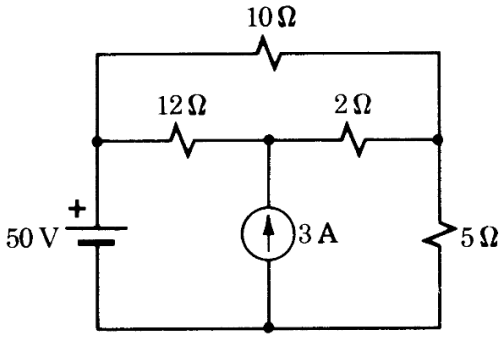
To build and analyze various resistive networks in PySpice and to compare the analysis results with the theoretical analysis results.

Materials/Devices:

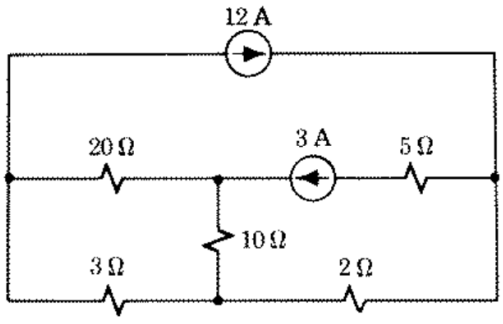
PySpice

Work to be done:

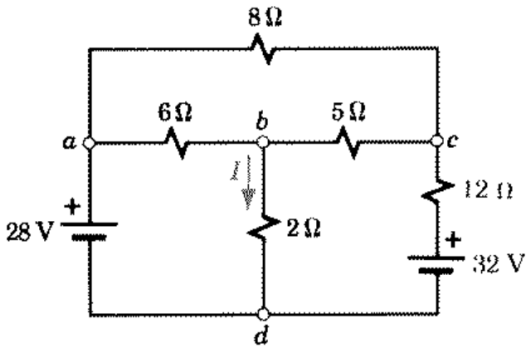
1. Build the following circuit in PySpice circuit simulator.
    a. Find the current in 5 Ω resistor by simulation.
    b. Find it analytically and compare the results.



2. Build the following circuit in PySpice circuit simulator.
    a. Find the voltage across the 20 Ω resistor by simulation.
    b. Find it analytically and compare the results.



3. Build the following circuit in PySpice circuit simulator.
    a. Find the current in 2 Ω resistor by simulation.
    b. Find it analytically and compare the results.
    c. Calculate the powers of 28v and 32v sources in your Python program.



Final Remarks

-