

7. PSpice Simulation II

**Objectives:** This experiment emphasizes the software simulation for circuit behaviors. Specifically, this lab is to solve three circuit problems by using PSpice.

**Pspice:** Introduction to the Pspice was presented, and is available on the class note web page.

---

**NOTE 1:** The problem #1 must be solved, and its result in hardcopy must be submitted within the lab hour. The PCs in room #3121 can also be used. If the PCs are not accessible because of the use by another class, the deadline of the problem#1 can be extended until 6pm. This must be done individually.

**NOTE 2:** The problems #2 and #3 can be worked and may be submitted **within a week**. This must be done individually.

**PRE-LAB -7**

There is no pre-lab for this experiment.

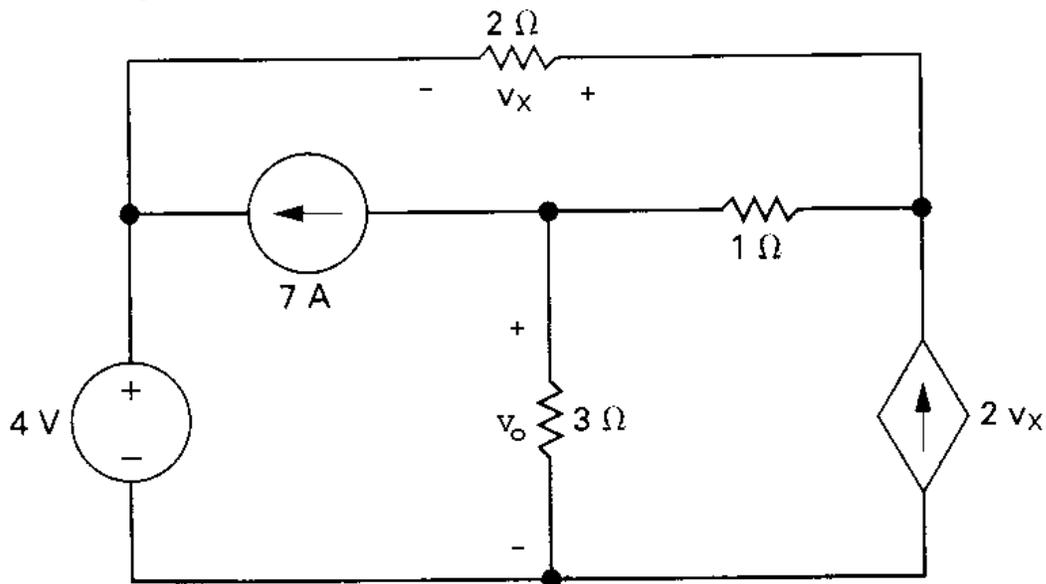
---

LAB -7:                      Name: \_\_\_\_\_                      ID#: \_\_\_\_\_

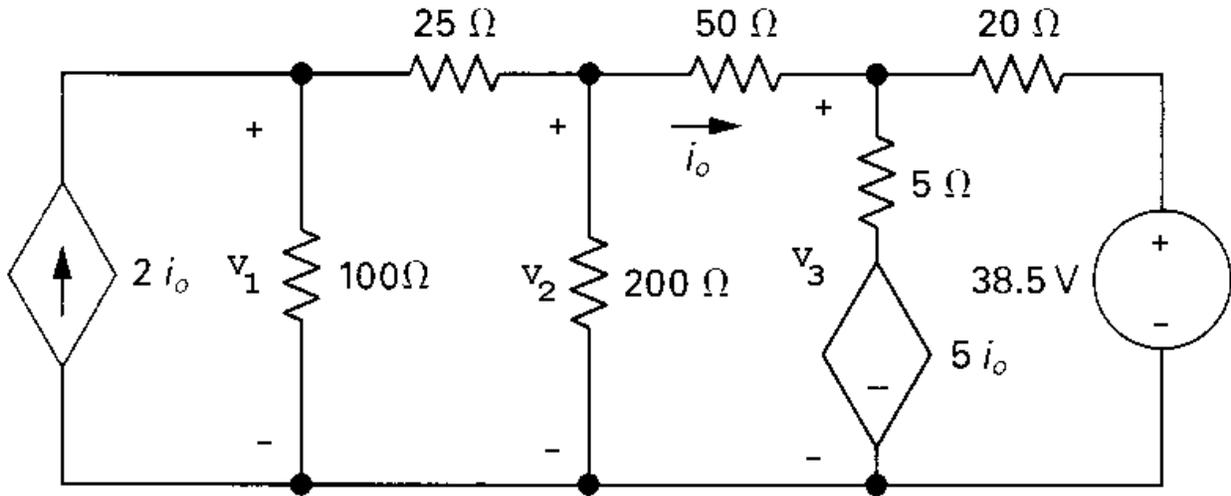
**\*Solve the following 3 problems using PSpice.**

1. This problem should be solved during the lab hour

- (i) Find the voltage  $V_o$  from the circuit below.
- (ii) Find the power delivered/absorbed by the 7A current source.



2. (i) Find the voltages  $V_1$ ,  $V_2$ , and  $V_3$ .  
(ii) Find the power consumed by the  $5\Omega$  resistor.



3. (i) Find the power absorbed by the 2A current source.  
(ii) Find the power absorbed/delivered by the 5A current source.  
(iii) Find the voltage  $V_\Delta$ .

