

## EHB211E Basics of Electrical Circuits

### HOMEWORK 1

You can download Pspice 9.1 Student Version via the following link. For the PSpice solutions, provide the schematic showing the voltage and current values.

<http://www.electronics-lab.com/downloads/circutedesignsimulation/?page=5>

- 1) Find  $V_{ab}$  and  $V_2$  in the Figure 1. Also, obtain and show these values using PSpice program.

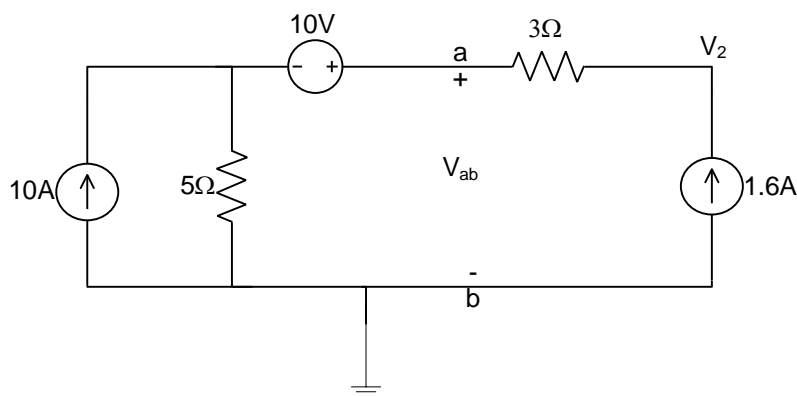


Figure 1

- 2) Find  $V_1$ ,  $V_2$  and  $V_3$  using nodal analysis in the Figure 2. Also, obtain and show the voltage values using PSpice program. You can take the bottom node as ground for Pspice simulation.

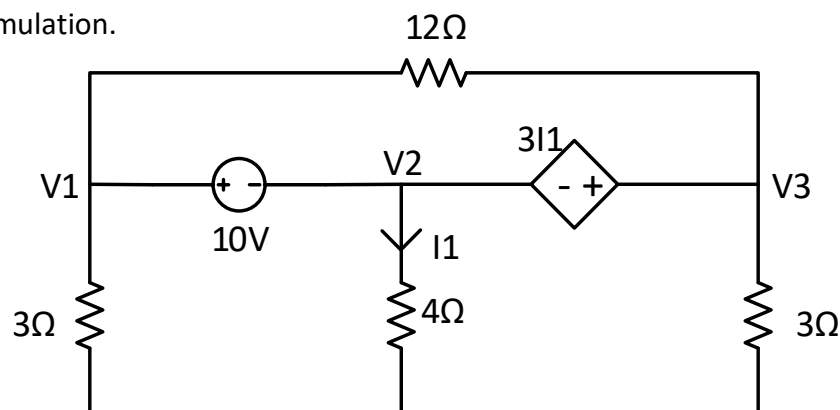


Figure 2

- 3) Find  $I_1$ ,  $I_2$  and  $I_3$  using mesh analysis in the Figure 3. Also, find and show these values using PSpice program. You can take the bottom node as ground for Pspice simulation.

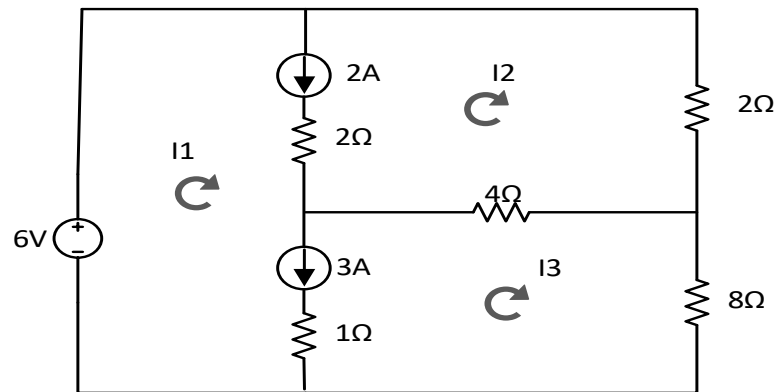


Figure 3

- 4) Figure 4 shows the current through and the voltage across an element. Find the total energy absorbed by the element for the period of  $0 < t < 4s$

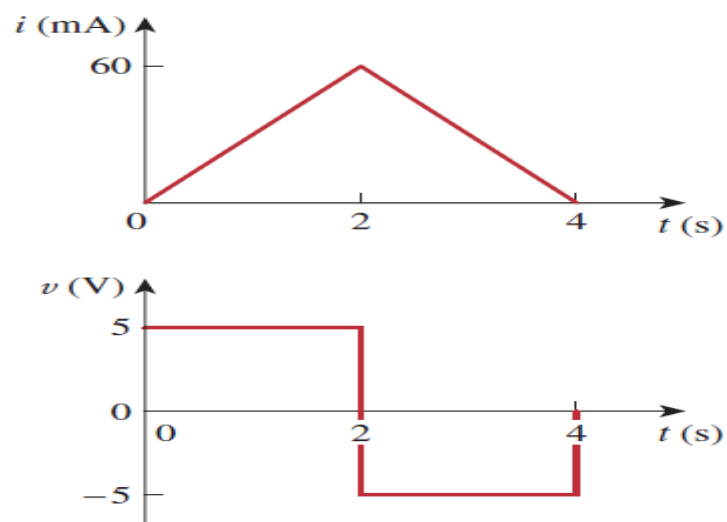


Figure 4

5) Calculate  $I_0$  in the Figure 5. Also, obtain and show  $I_0$  using PSpice program.

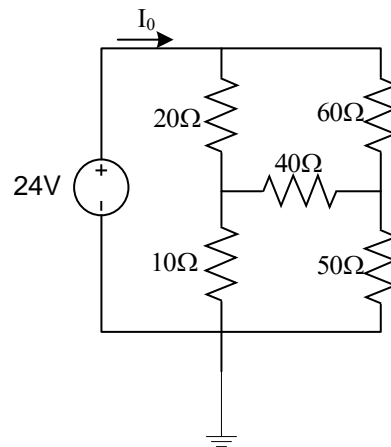


Figure 5