

PSpice Output Data – Voltages

Record the output data from PSpice in the second column. Complete the remaining columns.

<u>Item</u>	<u>PSpice Data</u>		
V_A			
V_B			
V_C			
<u>Item</u>	<u>Values Calculated from PSpice Data</u>	<u>Measured Data</u> *	<u>Percentage Difference</u>
$V_3 = V_A - V_B$			
$V_4 = V_C - V_B$			
$V_5 = V_B - 0$			

* From Exp. 3.

PSpice Output Data – Current

<u>Item</u>	<u>PSpice Data</u>	<u>Measured Data</u> *	<u>Percentage Difference</u>
I_{AB}			
I_{CB}			
I_{BG}			

* From Exp. 3.

PSpice Exercise

Use PSpice to analyse the circuit shown on page 83 of the textbook “Circuits” by Ulaby and Maharbiz. Refer to Figure 3-1 (b) for voltage and current references. The voltage source is 10 V and the current source is 3 A. The resistors are:

$$R_2 = 100 \Omega \quad R_3 = 100 \Omega \quad R_4 = 100 \Omega \quad R_5 = 100 \Omega \quad R_6 = 100 \Omega$$

$$R_7 = \text{last 3 digits of your VU ID } \Omega$$

On the attached sheets, include the following items in this section of the report:

- Drawing of the original circuit with node numbers labeled
- PSpice netlist
- List of output node voltages: V_0 , V_1 , V_2 , V_3 and
- List of currents I_1 and I_7 .