

ECE 0101 – Linear Circuits and Systems

Laboratory # 3 – Introduction to PSpICE

Objectives:

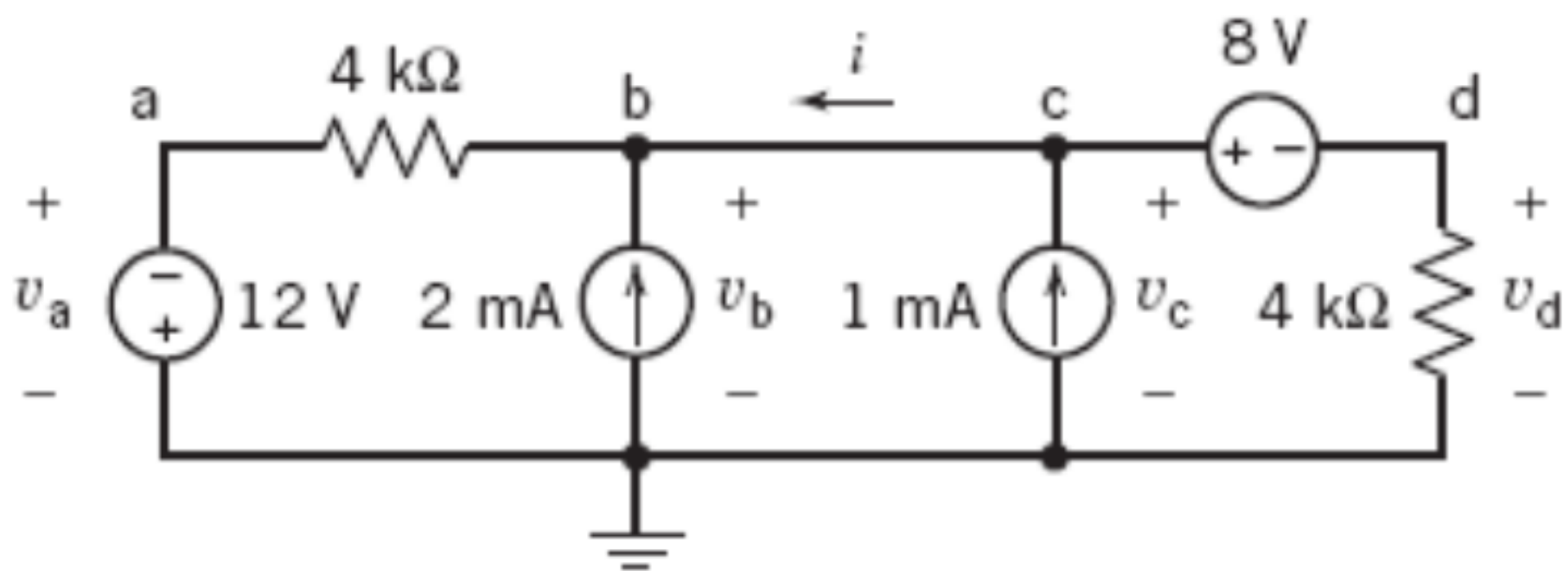
- Understand the graphical user interface for PSpice Capture and Simulation software.
- Be able to solve problems based on DC analysis and parameter sweeps in Pspice.

Introduction

This laboratory assignment will reinforce the concepts learned in class about the use and behavior of transformers. The circuit diagram shown in Figure 1 shows a circuit with a resistor connected to an ideal transformer. In the laboratory experiment the waveform generator will be used in place of the voltage source $v_s(t)$.

Homework

(30 points) The voltages v_a , v_b , v_c , and v_d in the circuit below are the node voltages corresponding to nodes a, b, c, and d. The current i is the current in a short circuit connected between nodes b and c. Determine the values of v_a , v_b , v_c , and v_d and of i . Is the direction of current i the same as initially assumed? Now, change the direction of current i and redo the circuit analysis. Did the voltage values change?



$$V_a = -12V$$

$$\frac{V_a - V_b}{4k\Omega} + i + 2mA = 0$$

$$V_d = V_b - 8$$

$$i_d = \frac{V_b - 8}{4k\Omega}$$

$$i = 1mA - \frac{V_b - 8}{4k\Omega}$$

$$- \frac{12 - V_b}{4k\Omega} + 1mA - \frac{V_b - 8}{4k\Omega} + 2mA = 0$$

$$\frac{-4 - 2V_b}{4k\Omega} + 3mA = 0$$

$$V_b = 4V$$

$$\therefore V_a = -12V$$

$$V_b = V_c = 4V$$

$$V_d = -4V$$

$$i = 1mA - \frac{4 - 8}{4}$$

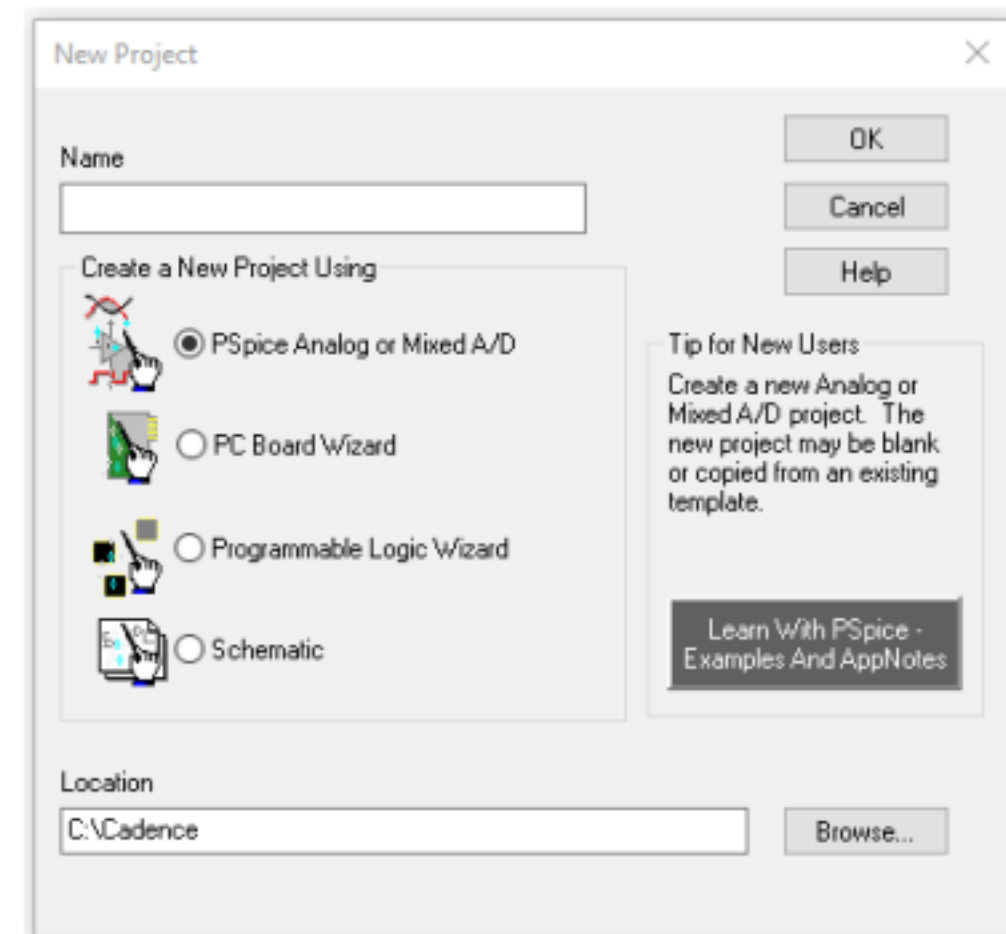
$$= 2mA$$

Yes, the direction is as same as initialized

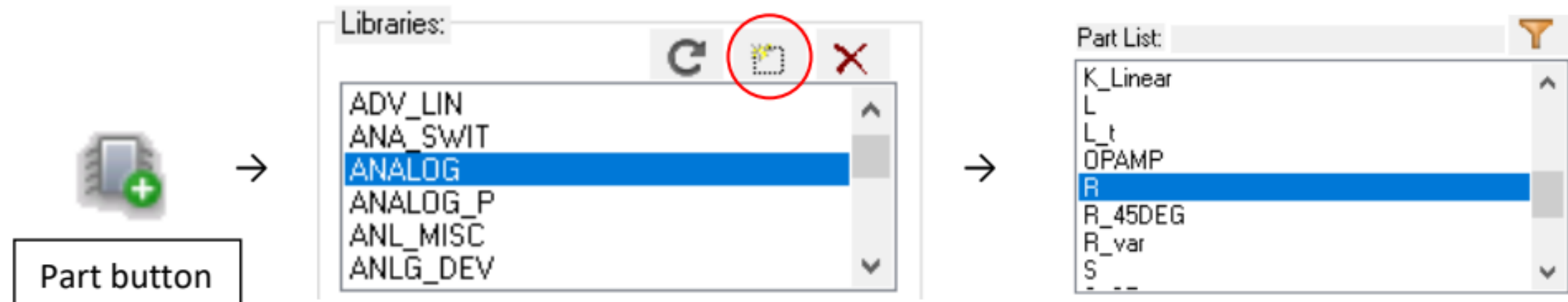
the voltage won't change

Laboratory Procedure

1. Start the PSpice schematic entry software, which is called OrCAD Capture. If you search for "Capture" you will find the program in Windows. Click on File → New → Project, then select "PSpice Analog or Mixed A/D" under the "Create a New Project using" area. Name your project whatever you prefer and click OK. In the next dialog box, select "Create a blank project". The other option is for creating a project based on a template that you use often, but we won't be using that here.



2. On the right side of the OrCAD window are buttons allowing you to place or manipulate components on your schematic. Click the "part" button, then click on the "ANALOG" library, then click on the letter "R" for a resistor. Place this resistor in your schematic (you can rotate components by typing R). If the ANALOG library is not in the list you will have to find it in the directory using the folder button circled in red below.



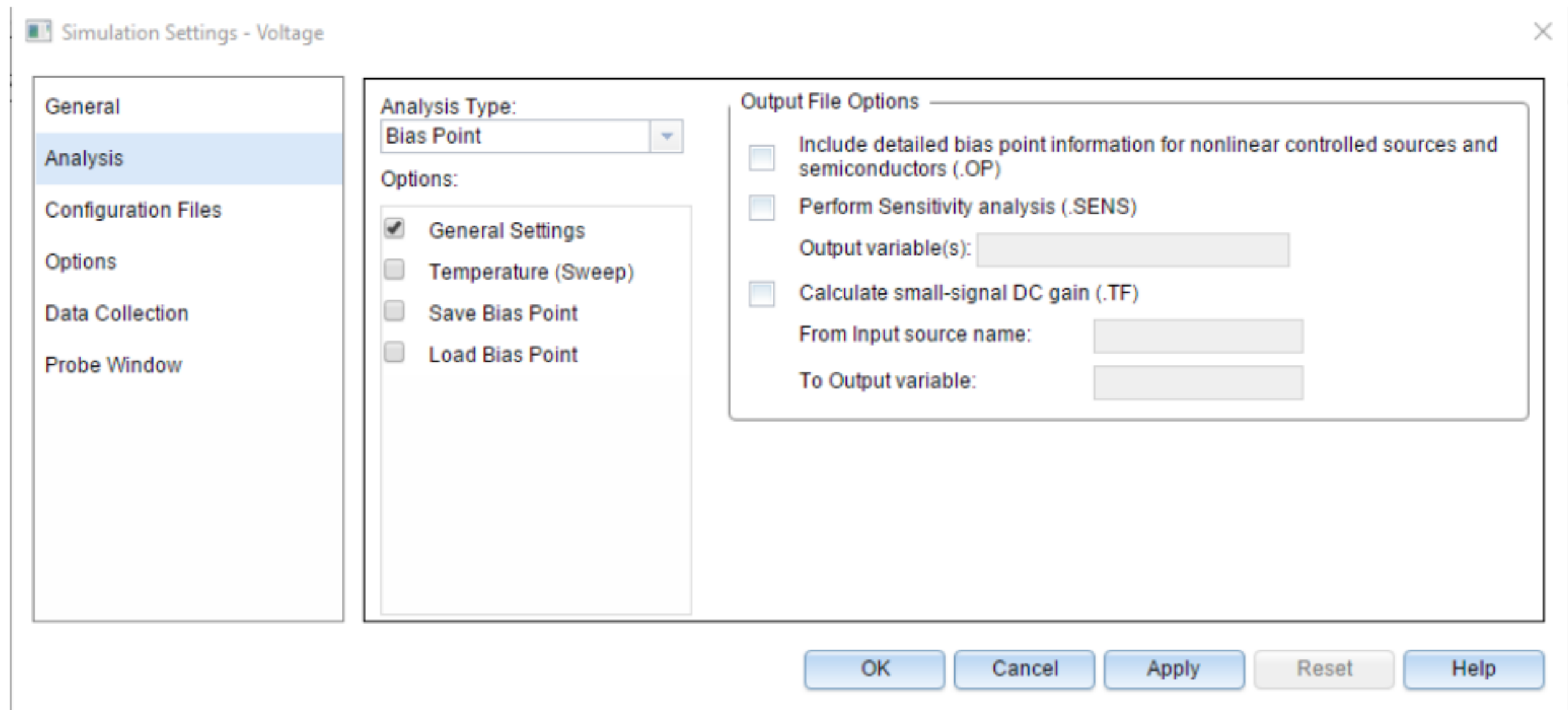
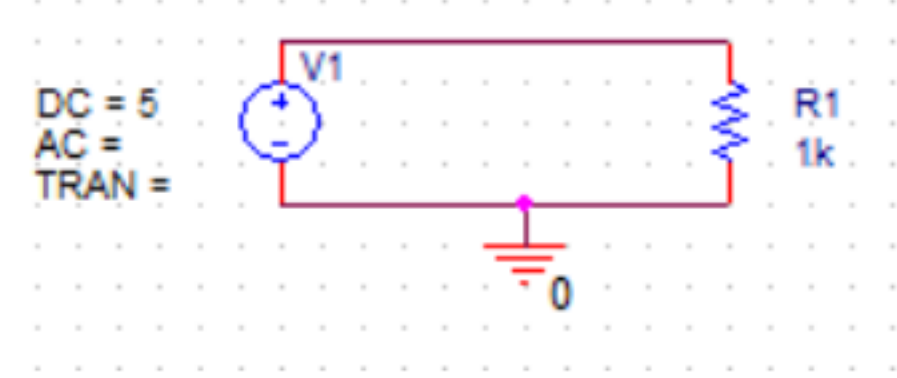
3. Next, find the "VSRG" element from the "SOURCE" library and place this on your schematic. Double-click on the voltage source, and enter "5" into the dialog box under "DC", for DC voltage. You can also enter parameters for AC and transient voltage, but we won't use those in this lab.

	AC	Color	DC	Designator	Graphic
AGE1		Default	5		VSRG.Normal

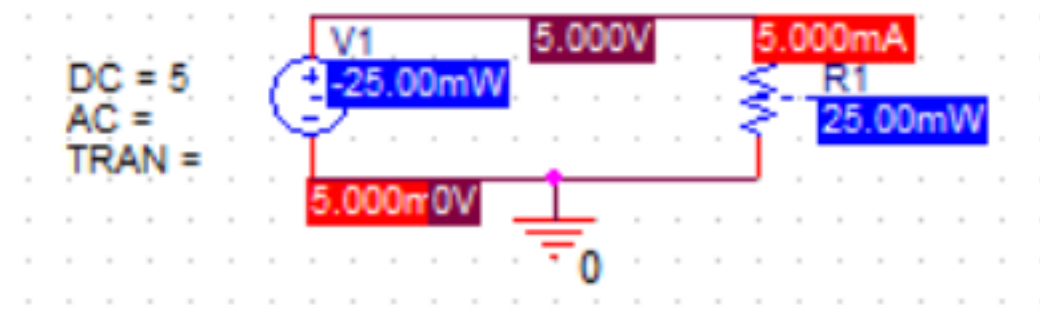
4. Using the wire tool, shown in the graphic to the right, connect the voltage source and the resistor together. Additionally, click on the ground tool to place a ground on the negative end of the voltage source.



5. Your schematic should look similar to the one shown in the graphic to the right. If it does, click on PSpice → New Simulation Profile and enter “DC” as the simulation name. For the Analysis Type, select “Bias Point” and press “OK”.



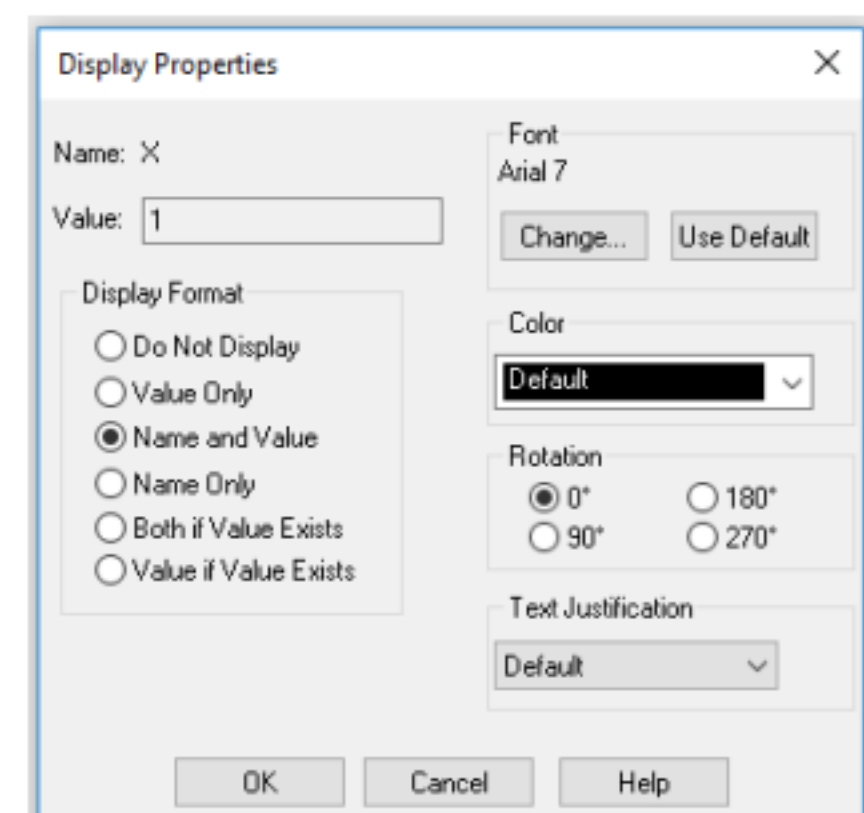
6. Select PSpice → Run. Other windows may appear because the simulation actually occurs in other software programs. Once it is complete return to the schematic window and click on the “V”, “I”, and “W” buttons in the upper frame to display voltage at nodes as well as current through and power in elements. You should see something similar to the graphic to the right.



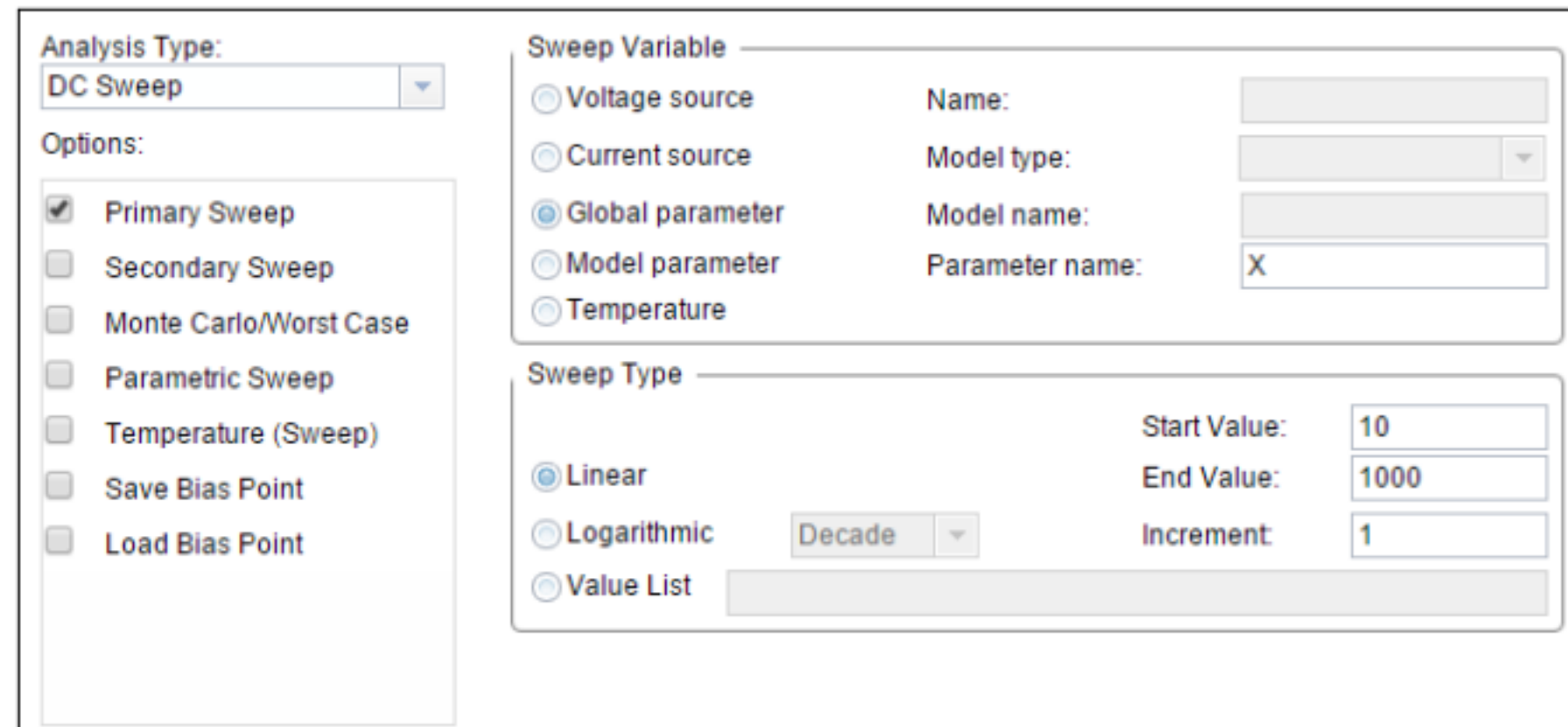
7. Double-click on the resistor and replace the value of “1k” with “{X}”, the variable X in curly brackets. From the SPECIAL library, add the pseudo-component “PARAM” to the schematic. Double-click on it, then click on “New Property...” and call this property X, the same as your variable name. Find this new column and enter any value you want.

Source Package	Source Part	Value	X
PARAM	PARAM.Normal	PARAM	1

Right-click on that value click “Display” and then select “Name and Value”.

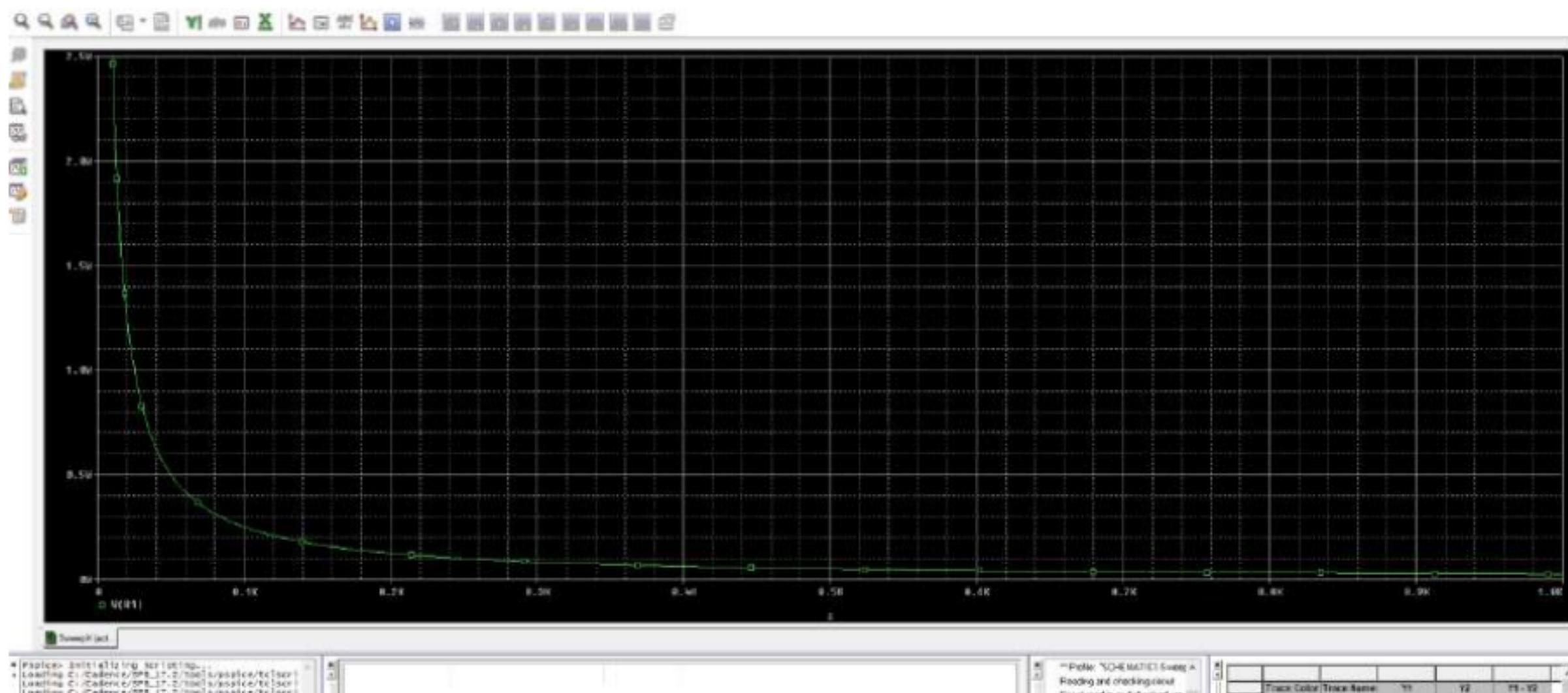


- Create a new simulation profile and call it "SweepX". Under analysis type, select DC Sweep, under "Sweep Variable" select "Global parameter" and enter X in the box for parameter name. Under sweep type, select "Linear" with a start value of 10, an end value of 1000, and an increment of 1 (ohms).



The screenshot shows the PSpice simulation profile configuration dialog box. The "Analysis Type" is set to "DC Sweep". Under "Options", "Primary Sweep" is checked. Under "Sweep Variable", "Global parameter" is selected, and the "Parameter name" is set to "X". Under "Sweep Type", "Linear" is selected, with a "Start Value" of 10, an "End Value" of 1000, and an "Increment" of 1. The "Decade" dropdown is also visible.

- Run the simulation, which will bring up a blank graph window. Click the "add trace" button, shown in the graphic to the right, and select W(R1) to plot the power through the resistor as you sweep through its resistance. Your graph should look like the graphic shown below.

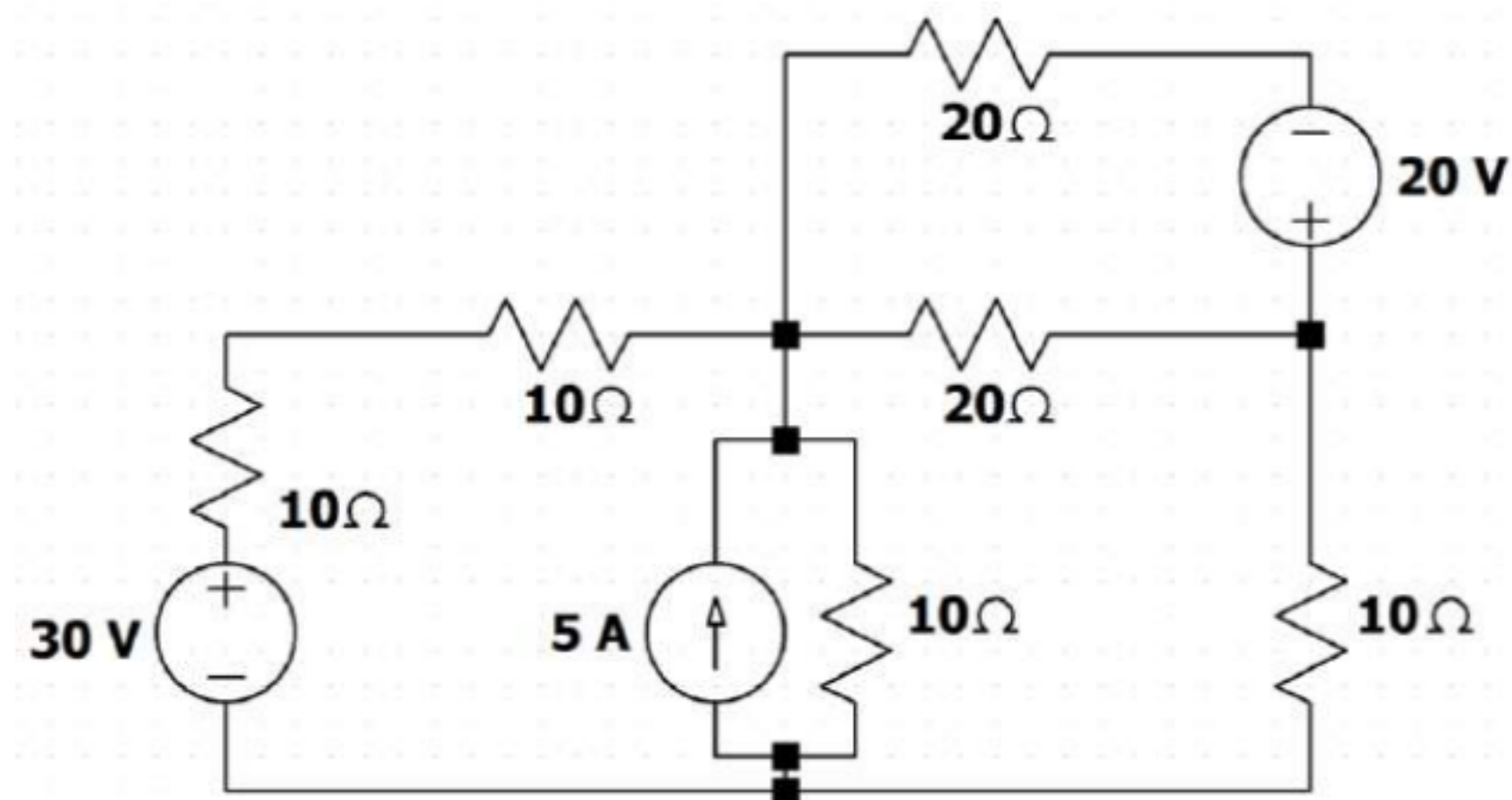


Use these tools to complete the following analyses.

Laboratory Assignment

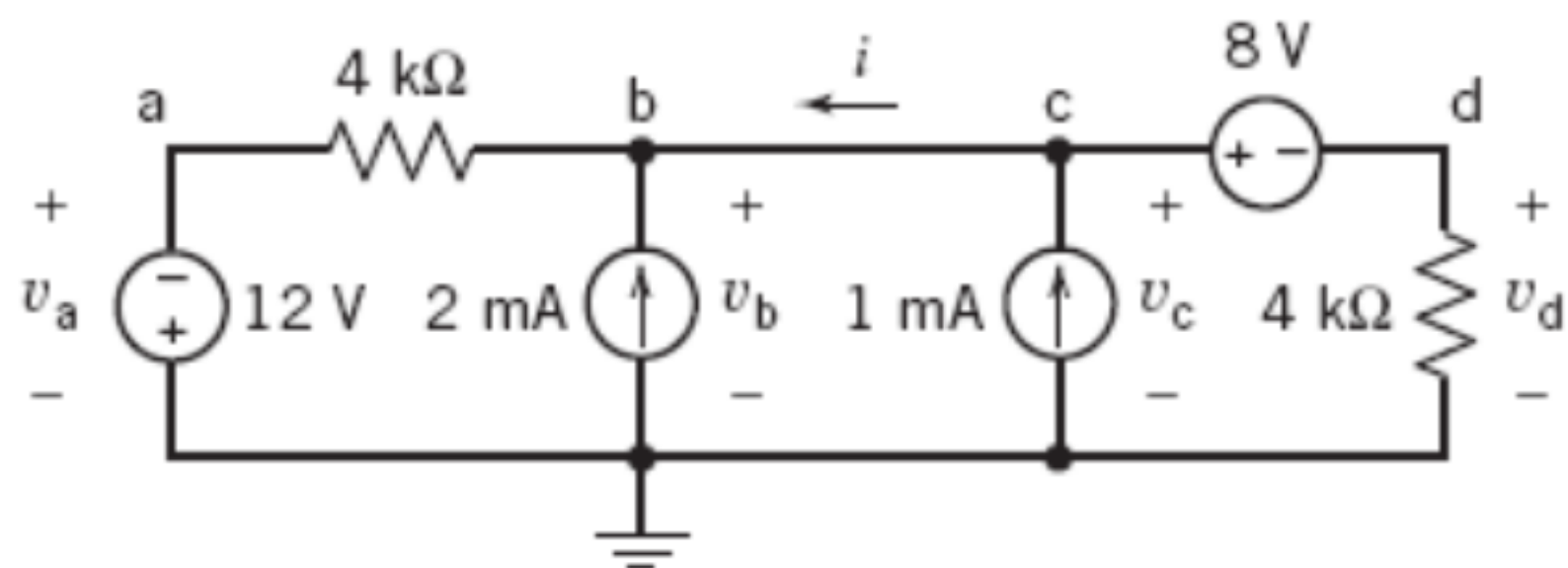
Problem #1

(40 points) Label the current through each resistor in the following circuit by simulating it in PSPICE. The current source is called ISRC in the SOURCE library. Please print out and attach a screenshot of your simulated circuit.

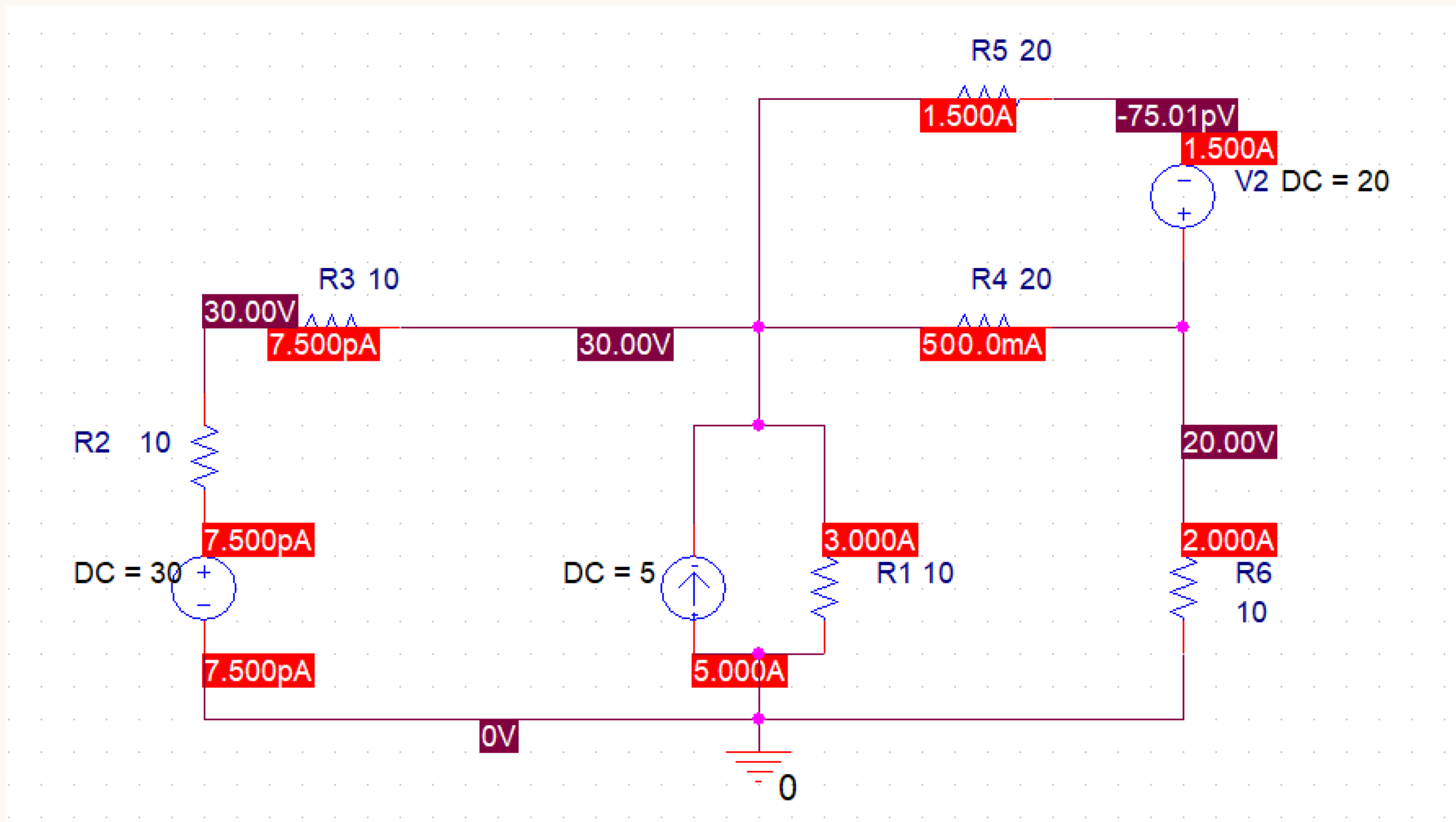


Problem #2

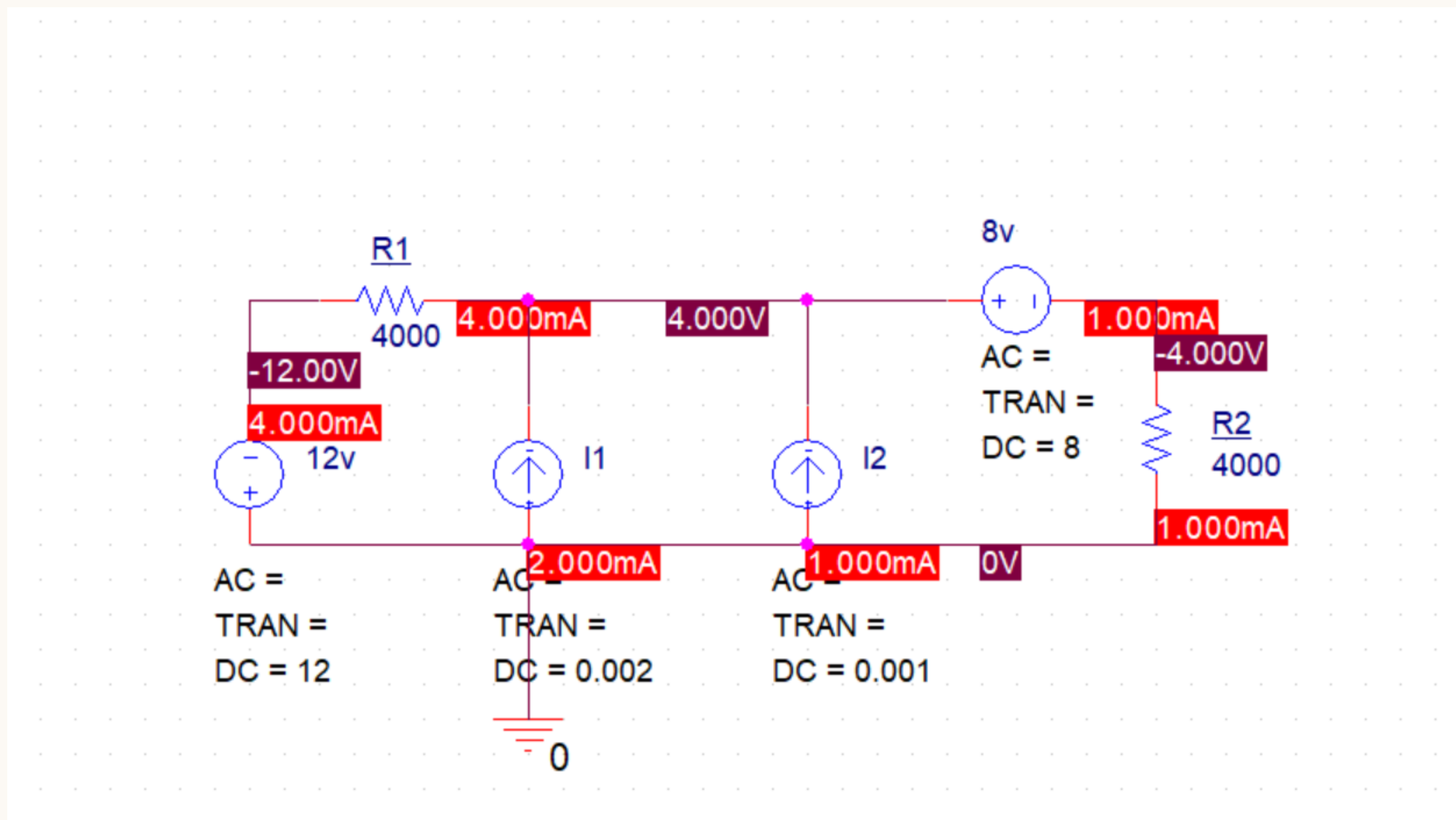
(30 points) Simulate the homework problem and determine the values of v_a , v_b , v_c , and v_d and of i . How does this compare to your answer to the homework problem? Please print out and attach a picture of your simulated circuit.



1.



2.



the results match my answers