

Name \_\_\_\_\_ Lab Section \_\_\_\_\_ Date \_\_\_\_\_

**Overview:**

This laboratory experiment introduces the PSpice circuit simulation program with the schematic capture front end, Schematics.

**Introduction:**

Basic features of the PSpice circuit simulation program with the schematic capture front end (Schematics) will be explored using simple resistive circuit examples.

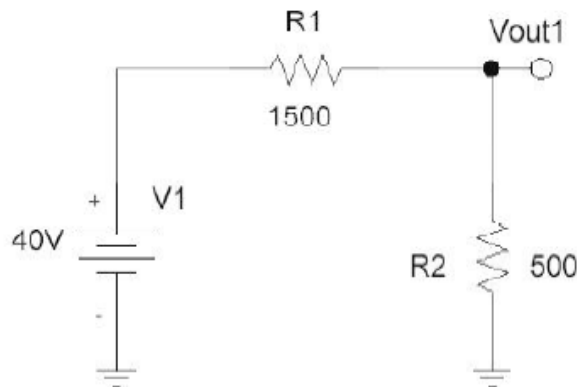
**Procedure:**

1. Turn on the computer and allow it to boot into the Windows operating system. Log in using your KSU student account.
2. Start PSpice/Schematics by *left-clicking* **Start** → **All Programs** → **PSpice Student** → **Schematics**. A blank schematic page should appear.

Schematics is a front-end program with a direct interface to other PSpice programs and options. Using Schematics it is possible to design, draw and simulate circuits. Simulation results can be analyzed using the PSpice waveform viewer (Probe).

Create the *DC Series Circuit* shown below on your schematic workspace by performing steps **3** through **13**.

**Note:** for the remainder of the steps in this lab exercise, the term “click” will be used to specify *left-clicking* the mouse and the term “right-click” will be used to specify *right-clicking* the mouse.



**Figure 1. A DC series circuit.**

3. Create the DC voltage source, V1, by doing the following:

Click **Draw** from the *Schematics* menu bar, to open the *Draw* drop-down menu, click **Get New Part...** to open the *Part Browser Basic* dialog box. In the *Part Browser Basic* dialog box, scroll the *Full List* box until the **VDC** option appears. Click on the **VDC** option. Click the **Place&Close** command button.

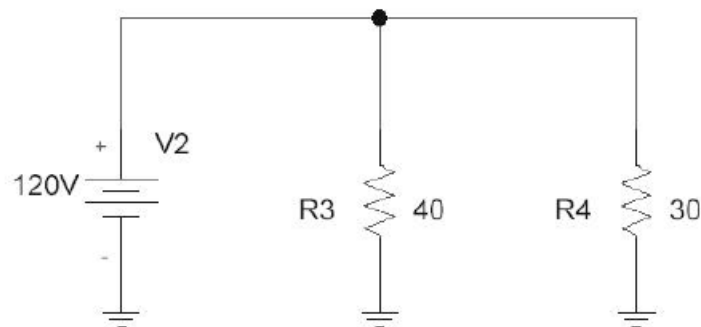
For the remainder of the steps in this lab, this sequence of procedures will be abbreviated as: Click **Draw**→**Get New Part**→**VDC**→**Place&Close**.

Position the DC voltage source in the left center of the schematic workspace by moving the mouse. Click to place the DC voltage source. Right-click to stop placing DC voltage sources. Note that the two step procedure **Draw**→**Get New Part** can also be executed by simply clicking the “Get New Part” icon which is located on toolbar above the schematic workspace.

4. In order to appropriately define the DC voltage source, V1, double-click on its schematic symbol. In the dialog box that opens, double-click on the **DC=** option of the list box, type "40V" (and remove “0V”) inside the *Value* text box, click the **Save Attr** command button. Next, click the **OK** command button.
5. Create the resistor, R1, by doing the following: Click **Draw**→**Get New Part**→**r**→**Place&Close**. Appropriately position the resistor by moving the mouse. Click to place the resistor. Right-click to stop placing resistors.
6. In order to appropriately define the resistor, R1, double-click on its schematic symbol. In the dialog box that opens, double-click on the **VALUE=** option of the list box, type "1500" (and remove “1k”) inside the *Value* text box, click the **Save Attr** command button. Next, click the **OK** command button.
7. Create the resistor, R2, by doing the following: Click **Draw**→**Get New Part**→**r**→**Place&Close**. Hold down the “Ctrl” key of the keyboard and press the “R” key of the keyboard simultaneously (this control key sequence will be specified as: **Ctrl-R** in future procedures). Note that this procedure causes the resistor, R2, to rotate counter-clockwise by 90 degrees. Appropriately position the resistor by moving the mouse. Click to place the resistor. Right-click to stop placing resistors.
8. In order to appropriately define the resistor, R2, double-click on its schematic symbol. In the dialog box that opens, double-click on the **VALUE=** option of the list box, type "500" (and remove “1k”) inside the *Value* text box, click the **Save Attr** command button. Next, click the **OK** command button.
9. Create the two ground symbols by doing the following: Click **Draw**→**Get New Part**→**EGND**→**Place&Close**. Appropriately position the first ground symbol by moving the mouse. Click to place the first ground symbol. Appropriately position the

second ground symbol by moving the mouse. Click to place the second ground symbol. Right-click to stop placing ground symbols.

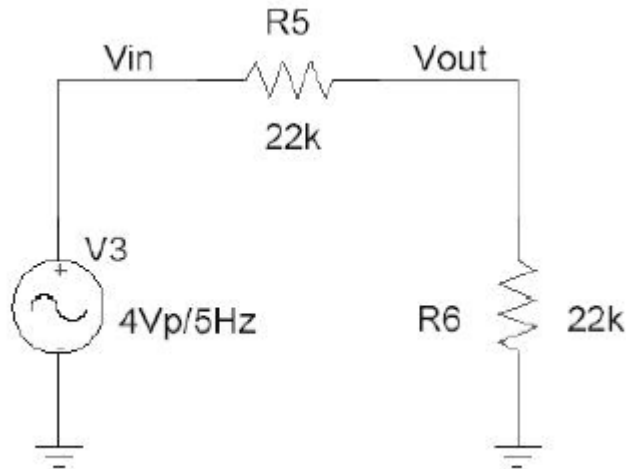
10. Connect (wire) components V1, R1 and R2 by doing the following: Click **Draw**→**Wire**. A pencil symbol will appear where the mouse pointer should be. Move the pencil so that it points at the top of the DC voltage source, V1. To start drawing a wire, click on the top of the DC voltage source and then move the pencil to point at the left terminal of the resistor, R1, and then click. Next, move the pencil so that it points at the right terminal of the resistor, R1. To start drawing a second wire, click on the right terminal of the resistor, R1, and then move the pencil to point at the top terminal of the resistor, R2, and then click. Right-click to stop drawing wires.
11. Create the output connection bubble by doing the following: Click **Draw**→**Get New Part**→**BUBBLE**→**Place&Close**. Press **Ctrl-R** three times to appropriately rotate the connection bubble. Properly position the connection bubble by moving the mouse. Click to place the bubble. Right-click to stop placing bubbles.
12. Label the connection bubble by doing the following: Double-click on the schematic symbol of the connection bubble. In the text box of the *Set Attribute Value* dialog box which is now open, type “Vout1” and then click the **OK** command button.
13. Create the text "*DC Series Circuit*" underneath the circuit by doing the following: Click **Draw** to open the *Draw* drop-down menu. From the open menu that appears, click **Text...** In the *Add Text* dialog box that appears, type "*DC Series Circuit*" inside the *Text* box. Click **OK**. The outline of a box should now appear at the cursor. Move the cursor to the desired location below the circuit and click. The text is now placed. Right-click to stop placing text. Your circuit should now look similar to the *DC Series Circuit* shown on the first page of this lab exercise. Note that parts can be deleted from the circuit by clicking on the schematic symbol of the part (the symbol will be highlighted when the part is selected) and then pressing the “Delete” key on your keyboard. This technique is useful when you are trying to modify or correct a circuit.
14. To the right of the *DC Series Circuit* that was just completed, create the *DC Parallel Circuit* shown below.



**Figure 2. A DC parallel circuit.**

Use the applicable techniques that were performed in steps 3 through 13 as a guide to create and define the *DC Parallel Circuit*. Note that the value of the DC voltage source, V2, is 120V. The value of the resistor, R3, is 40, and the value of the resistor, R4, is 30. Be sure to create the appropriate text below the circuit.

- To the right of the *DC Parallel Circuit* that you just completed, create the *AC Series Circuit* shown below.

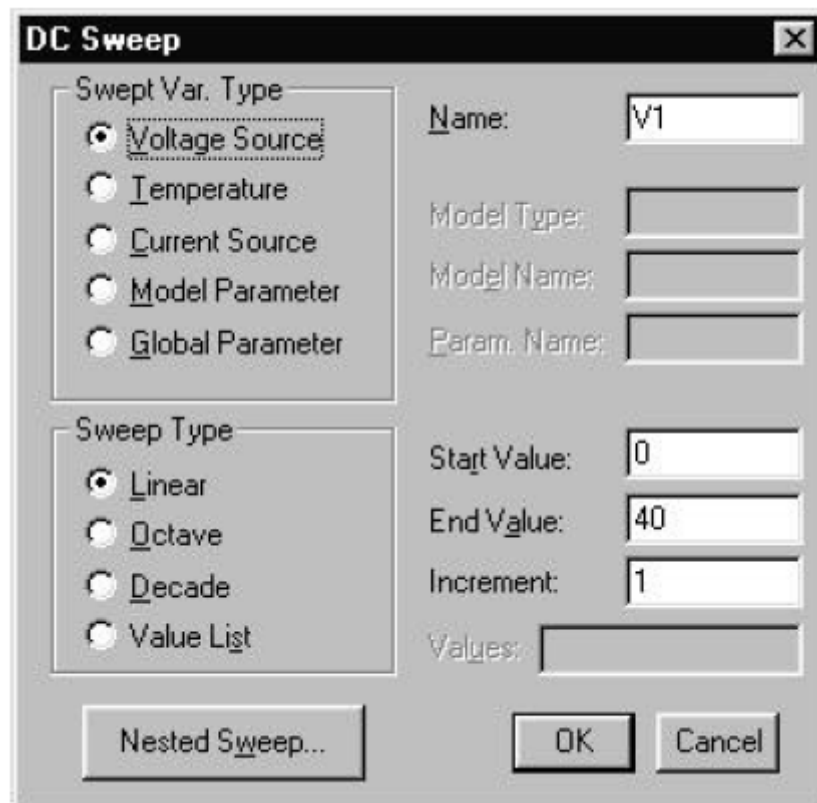


**Figure 3. An AC series circuit.**

- Create the sinusoidal AC voltage source, V3, by doing the following: Click **Draw**→**Get New Part**→**VSIN**→**Place&Close**. Appropriately position the sinusoidal AC voltage source. Click to place the sinusoidal AC voltage source. Right-click to stop placing sinusoidal AC voltage sources.
- In order to appropriately define the sinusoidal AC voltage source, V3, double-click on its schematic symbol. In the dialog box that opens, double-click on the **DC=** option of the list box, type "0" inside the *Value* text box, click the **Save Attr** command button. Next, double-click on the **AC=** option of the list box, type "2.828" inside the *Value* text box, click the **Save Attr** command button. Next, double-click on the **VOFF=** option of the list box, type "0" inside the *Value* text box, click the **Save Attr** command button. Next, double-click on the **VAMPL=** option of the list box, type "4" inside the *Value* text box, click the **Save Attr** command button. Next, double-click on the **FREQ=** option of the list box, type "5" inside the *Value* text box, click the **Save Attr** command button. Finally, click the **OK** command button. This procedure defines the AC voltage source, V3, as a 4 V peak (2.828 V rms), 5 Hz sinusoidal waveform.
- Use the applicable techniques that were performed in steps 3 through 13 as a guide to create and define the remaining portion of the *AC Series Circuit*. Note that the wire between V3 and R5 must be labeled as *Vin*. Double-click on the wire between V3 and R5. In the text box of the *Set Attribute Value* dialog box which is now open, type "Vin"

and then click the **OK** command button. Use a similar approach to label the wire between R5 and R6 as  $V_{out}$ . Note that the value of the resistor, R5, is  $22k$ , and the value of the resistor, R6, is also  $22k$ . Be sure to create the appropriate text underneath the circuit. Also create the text “4Vp/5Hz” and locate it adjacent to the sinusoidal AC voltage source, V3.

19. When the three circuits are completed, save your overall schematic as “ECET1010\_Lab2”.
20. You are now ready to set up a DC analysis for your schematic. Set up the DC analysis as follows. Click **Analysis** from the *Schematics* menu bar. From the open menu that appears, click the **Setup...** command. In the *Analysis Setup* dialog box that appears, click the **DC Sweep...** button. A *DC Sweep* dialog box should appear. Complete the *DC Sweep* dialog box as shown below:



**Figure 3. DC sweep dialog box.**

When you are finished click **OK**. Then click the **Close** button in the *Analysis Setup* dialog box.

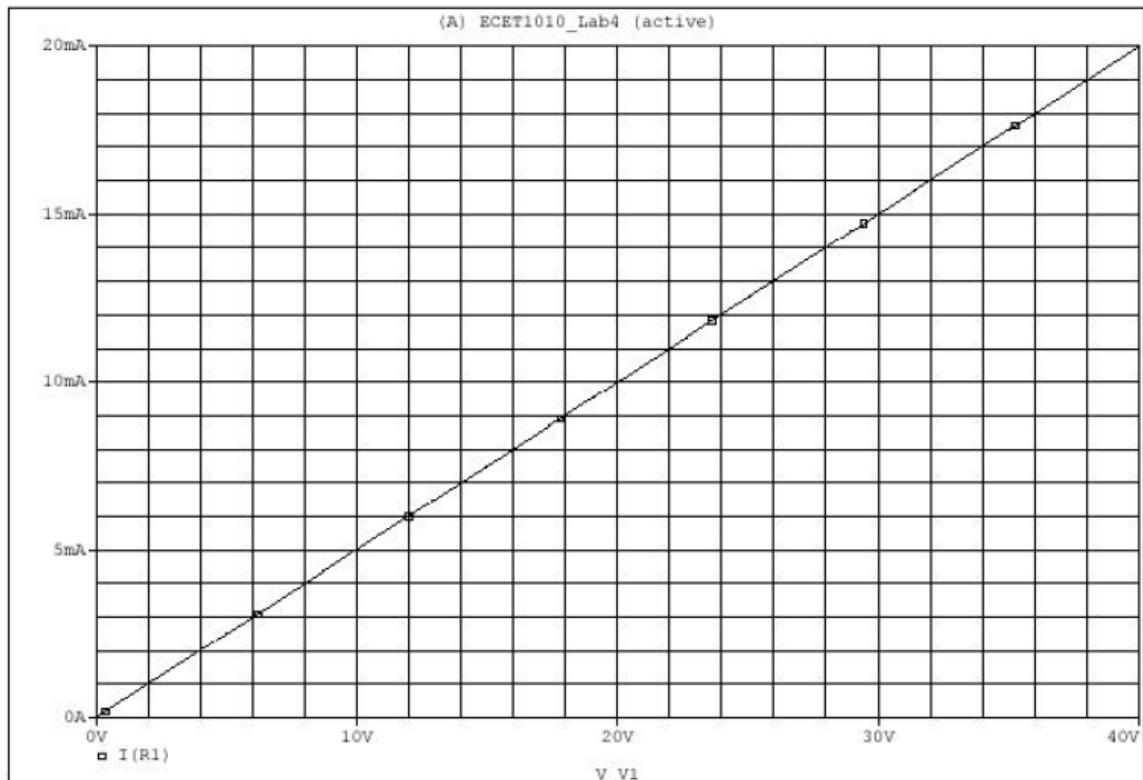
21. Click **Markers**→**Mark Current into Pin**. Position the tip of the marker on the left terminal of R1 by moving the mouse. Click to place the marker. Right-click to stop placing markers. Procedures **20** and **21** will allow a DC analysis to be performed. The

DC voltages and currents for each component in all three circuits will be evaluated. In addition a graph of  $I(R1)$  versus  $V1$  will be plotted (as  $V1$  is varied from 0 V to 40 V in 1 V increments) in the PSpice waveform viewer (Probe).

22. To start the simulation, click **Analysis** from the *Schematics* menu bar. From the open menu that appears, click the **Simulate** command. When a successful simulation is finished the PSpice waveform viewer (Probe) should display a graph similar to the *Graph of  $I(R1)$  [in milliamperes] versus  $V1$  [in volts]* shown on the next page.

**Show the display of your graph to your instructor.**

- Note that as  $V1$  increases,  $I(R1)$  increases.
- Observe that the graph is linear.
- Since Ohm's law states that  $V=I \cdot R$ , then the slope of the graph is equal to  $1/R$  (where  $R$  is the total resistance of the circuit).
- 



**Figure 4. Graph of  $I(R1)$  in milliAmperes versus  $V1$  in Volts.**

When you are finished analyzing the graph, close the PSpice waveform viewer by clicking the *Close* button in the uppermost right-hand corner of the screen. This should return you to the *Schematic* screen.

23. Click the “Enable Bias Voltage Display” icon which is located on the toolbar above the schematic workspace (it is the icon button with a large “V” label). Note that this causes

the DC (or average) voltage of each node in all three circuits to be displayed. If you click inside the box that is displaying the voltage, you can see the node to which it is associated.

- Observe that the *voltage divider rule* is verified in the *DC Series Circuit*.
- Observe that each resistor has the same voltage in the *DC Parallel Circuit*.
- Note that the DC (or average) voltage of each node in the *AC Series Circuit* is zero.

When you are finished analyzing the DC voltage display, disable the DC voltage display by clicking the “Enable Bias Voltage Display” icon again.

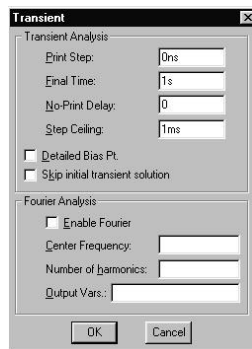
24. Click the “Enable Bias Current Display” icon that is located on the toolbar above the schematic workspace (it is the icon button with a large “**I**” label). Note that this causes the DC (or average) current through each component in all three circuits to be displayed. If you click inside the box that is displaying the current, you can see the component (and direction) to which it is associated.

- Observe that the current through each component in the *DC Series Circuit* is the same.
- Observe that *Kirchhoff’s current law* is verified in the *DC Parallel Circuit*.
- Note that the DC (or average) current through each component in the *AC Series Circuit* is zero.

When you are finished analyzing the DC current display, disable the DC current display by clicking the “Enable Bias Current Display” icon again.

25. Click the current marker that is located on the left terminal of R1. Note that the current marker is now highlighted. Press the “Delete” key on the keyboard to remove current marker.

26. You are now ready to set up a Transient analysis for your schematic. Set up the Transient analysis as follows. Click **Analysis** from the *Schematics* menu bar. From the open menu that appears, click the **Setup...** command. In the *Analysis Setup* dialog box that appears, click the **Transient...** button. A *Transient* dialog box should appear. Complete the *Transient* dialog box as shown below:



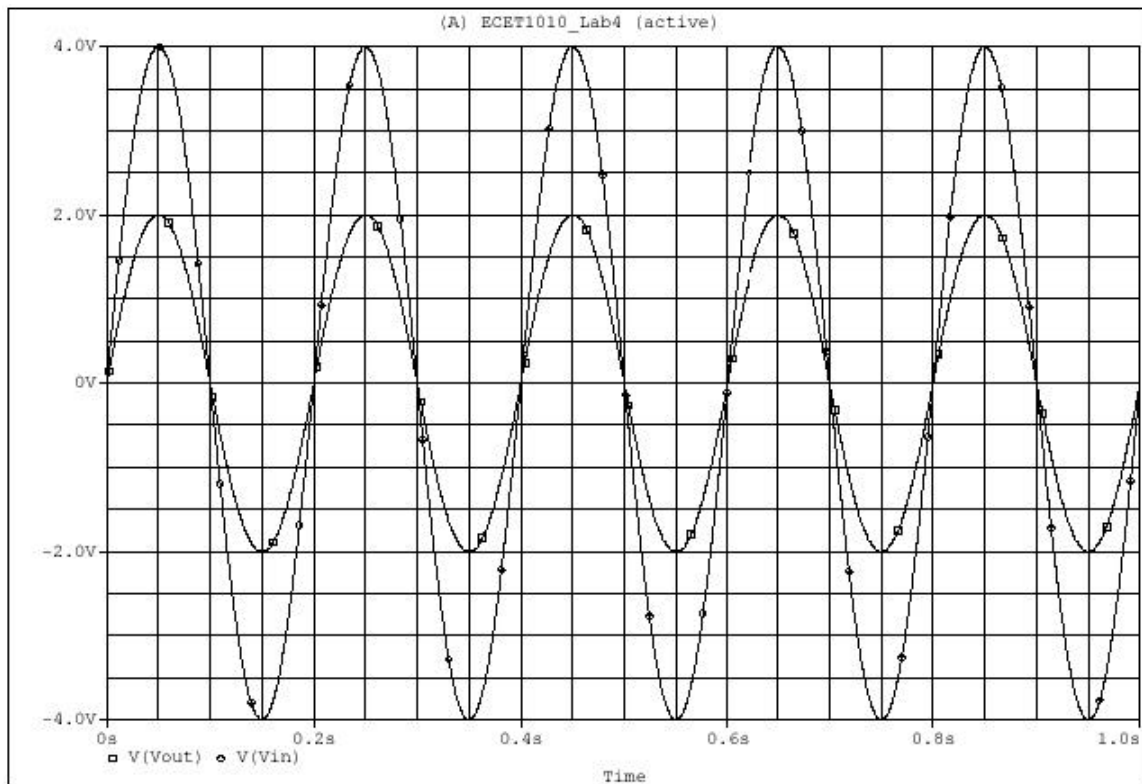
**Figure 5. Transient analysis dialog box.**

When you are finished click **OK**. In the *Analysis Setup* dialog box, uncheck the *Enable* box located to the left of the **DC Sweep...** button. Then click the **Close** button in the *Analysis Setup* dialog box.

27. Click **Markers**→**Mark Voltage/Level**. Position the tip of the marker to touch the wire labeled  $V_{in}$  by moving the mouse. Click to place the marker. Position the tip of a second marker to touch the wire labeled  $V_{out}$  by moving the mouse. Click to place the second marker. Right-click to stop placing markers.

Procedures **26** and **27** will allow a transient analysis to be performed. An accurate plot of voltage versus time (in seconds) will result in the PSpice waveform viewer (Probe). In this specific transient analysis,  $V_{in}$  and  $V_{out}$  will be plotted versus time (for a duration of 1 second).

28. To start the simulation, click **Analysis** from the *Schematics* menu bar. From the open menu that appears, click the **Simulate** command. When the simulation is finished the PSpice waveform viewer (Probe) should display a plot similar to the following:



**Figure 6. Plot of  $V_{in}$  and  $V_{out}$  in volts versus time in seconds.**

- Note that  $V_{in}$  is a 4 V peak, 5 Hz sinusoidal AC waveform as expected.
- Observe that 5 cycles (repetitions) of  $V_{in}$  occur during 1 second.



- Note that due to the *voltage divider rule*,  $V_{out}$  is half the peak amplitude of  $V_{in}$ .

29. **Show the displays of your waveforms and circuit drawings to your instructor.**

30. Have your instructor review your work. Make any required corrections. Once done and demonstrated to your instructor, turn in these worksheets for grading.

31. Make sure **NO WORK IS SAVED** on your computer. You may save your file to a USB drive or email it to yourself. Log off of your computer, return all equipment and furniture to its proper location, and make sure your lab bench is cleaned up before you leave.