ECE 3254 Laboratory **Name _____**
Psipce simulation – simple resistor circuit **Date: _____**
Laboratory #1 Fall 2009 **Bench: _____**
PRELAB (100 points total)

Prelab assignment (to be completed before coming to Lab 1):

1. Read through the entire Lab1 handout and Lab1 notes.
   Obtain Pspice Student 9.1 from the "Pspice 9.1 Student Ver" link on the ECE 3254 Lab web site.
   Install Pspice on a portable computer. **Bring this computer to class.** (10 points)
   To complete the Lab more quickly, you may wish to begin the procedure in Part I and Part III of the handout.

2. What is the equation for Ohms law in terms of voltage (V), current (I), and resistance (R)? (10 points)

3. Draw a circuit showing a voltage source, a resistor, and a voltmeter that measures the voltage across the resistor (hint: draw fig 1-1 with a Voltmeter added). (15 points)

4. Draw a circuit showing a voltage source, a resistor, and an ammeter to measure the current through the resistor (hint: draw fig 1-1 with an Ammeter added). (15 points)

5. For the circuit in Fig 1-1, calculate the voltage across R1 and the current through R1, with Vs = 9VDC and R1 = 1kΩ. Show equation(s) and values used in calculation(s). Copy your expected voltage and current to Part I-19, Part II-3 and Part II-4 of the Lab data sheet. (20 points)

6. For the circuit in Fig 1-2 with Vs = 9.0V and R1 = 1kΩ, and R2 = 560Ω, calculate the currents and voltages that you would expect to measure across and through the resistors. Show equation(s) and values used in calculation(s). Copy your expected voltages and current to Part III-9 and Part IV-2 of the Lab data sheet. (30 points)

This page left blank for calculations

ECE 3254
Simple Resistor Circuit Simulation and Measurement
Laboratory #1
Last Revised: 09-01-2009

Bring your portable computer with Pspice student 9.1 installed.

NOTE: You will want to save your work frequently as you go so that if something happens to Pspice, you do not have to repeat all of your work. You can do this by clicking File → Save, or by clicking the save button .

Record all measurements and calculations on the data sheet.
Always include the units for each measurement and calculation!
A manual for each piece of equipment is available if you need it. If you are unsure of how to make a measurement, it is your responsibility to ask for help or to consult the manual.

We will use the Agilent 34401A and Fluke Model 45 Digital Multimeter (DMM) to measure resistance, DC volts, and DC current. Measure the resistance of the resistor using the Agilent 34401A multimeter set on 2-wire ohms. NOTE: Individual resistors must always be removed/separated from a circuit when the resistor's resistance is measured.

Part I: Pspice Simulation of a Resistor and a DC Voltage Source

Enter the circuit of Fig 1-1 into the Pspice Schematic, with Vs = 9VDC and R1 = 1kΩ.

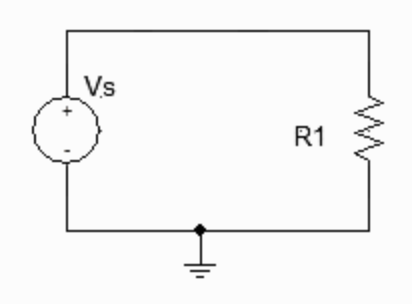


Figure 1-1

Draw the circuit in Pspice Schematic

1. Start Pspice by clicking Start → Programs → Pspice Student → Schematics
   When Pspice has started, you should have a window like this

2. Click "File" → "New", then "File" → "Save As" → Lab1-I,
   Save the file in a convenient location where you can easily find it

3. At the top of the window, click (see images below) "Draw" → "Get New Part" → scroll down to "r" → click "Place" → move the cursor to the schematic area and right click the mouse to place the resistor in the schematic. See the image with instruction 6 for a suggestion about how to arrange your parts. [Note: After a part has been placed, you may continue to place more of the same type part, select another part, or hit the esc key to end part insertion.]

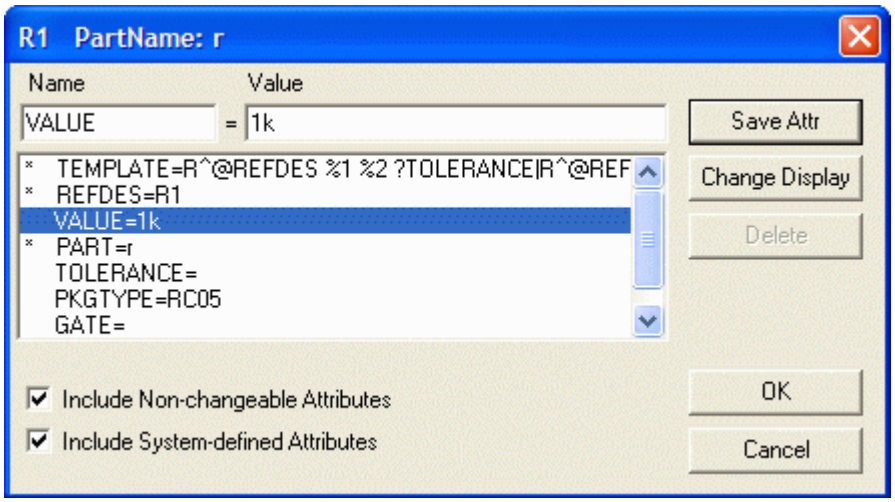
4. Now scroll down to "VDC" → click "Place" → move the cursor to the schematic area and right click mouse to place the voltage source in the schematic.

5. Now scroll to “GND_Earth” → click “Place” → move the cursor to the schematic area and right click mouse to place the ground in the schematic. Click the Esc key after you have placed the ground.

6. Click “Close” on the Part Browser Basic box. You should now have something that looks like this in your schematic area.

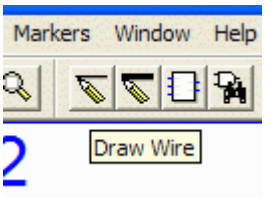
7. Right click the resistor to get the move position indication on the cursor (four way arrow) and the resistor will turn a different color. On toolbar, click “Edit” → “Rotate” (to rotate the resistor to a vertical position). Drag the resistor down to align it in parallel with the Voltage Source.

8. Double click the resistor R1 to open the Attributes box. If the value is not already 1k, click the “VALUE” line and type “1k” in the value space. Click OK to close the Attributes box.



9. Double click the name on V1 and set it to “Vs”. Double click the value (0V) and se it to “9V”.

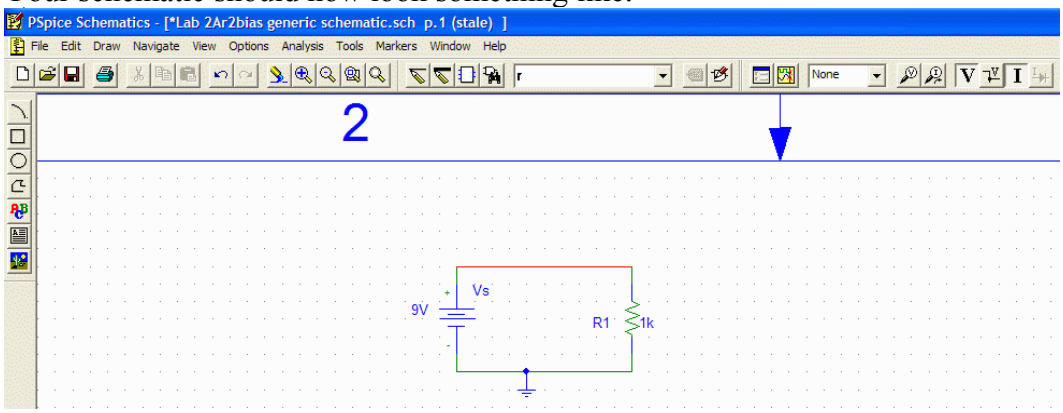
10. Place cursor over the draw wire pencil in the toolbar. “Draw Wire” should appear in a pop-up box. Click the pencil to activate wire drawing. Use the pencil to connect the top of the voltage source to the top of the resistor, the bottom of the voltage source to the ground, and the ground to the bottom of the resistor.



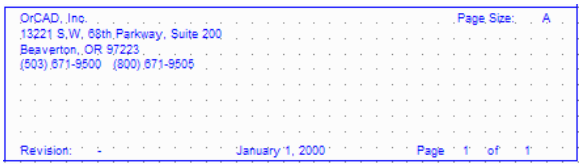
11. Click the V and I buttons on the toolbar  to turn “Enable Voltage Bias Display” and Enable Current Bias Display”. They will change to  when they are turned on.

12. If you have not already done this, save your file!

Your schematic should now look something like:



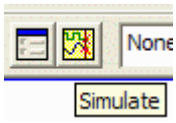
13. At the bottom right corner of the schematic, double click the first line of the drawing information and replace "OrCAD, Inc." with your name.



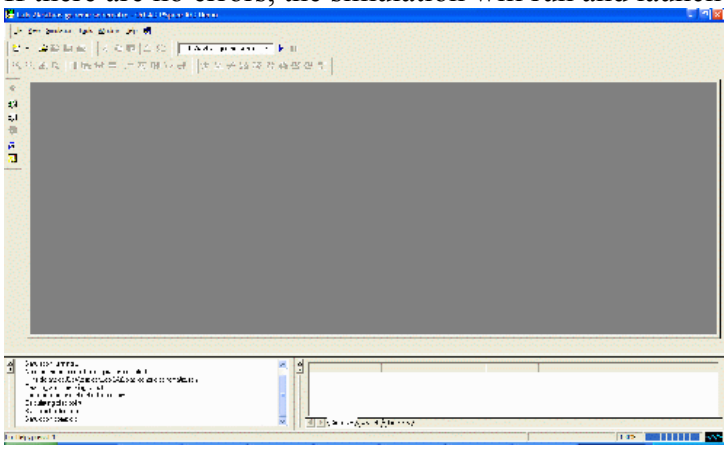
Double click "13221 S.W. 68th Parkway, Suite 200" and replace the text with ECE3254.
Double click "Beaverton, OR 97223" and replace it with 3254 Lab 1-I
Double click "(503) 671-9500 (800) 671-9505", and replace it with a blank or anything else you wish to enter.
Double click "January 1, 2000" and replace it with today's date.

14. Save your file!

Run the Resistor and DC Source Circuit Simulation

15. Click "Analysis" → "Simulate" or click the Simulate button  (if you hold the cursor over the simulate button, the word Simulate will appear below the toolbar) 

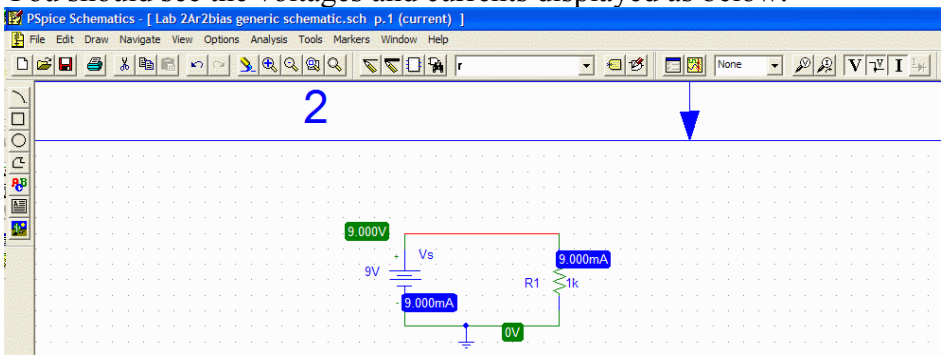
16. If there are no errors, the simulation will run and launch a new full page window that looks like



In the lower left corner, you should see "simulation complete".

17. Close the simulation window. Go to the Schematic window.

18. You should see the voltages and currents displayed as below.



19. Click the voltage and current bias displays and drag them slightly clear of the circuit components. Show your instructor this simulation and record your results on the Data Sheet.

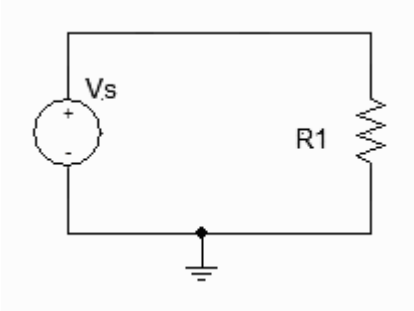
Part II: Measure the Voltage, Current, Resistance, and Verify the Pspice Simulation

1. Turn on the lab bench power. Turn on any instruments that do not start up. Turn on the computer power and contact your instructor if there are errors reported by Labview.

2. Obtain a 1000 Ω 1/4W resistor (color code Brown / Black / Red / Gold) resistor from the parts drawer.

   Measure the resistance with the Fluke 45 meter. Repeat this measurement with the Agilent 33401A meter.

3. Insert the 1000Ω resistor into the protoboard and connect as shown in Fig 1-1. For V_S, set the Agilent W3631A power supply to 9V and connect the power supply to the resistor.



(Fig 1-1)

Connect the Fluke 45 multimeter in parallel with the resistor to measure the voltage across resistor. Repeat the measurement with the Agilent 33401A

4. Reconnect Fluke 45 multimeter in series with the resistor and measure the current through the resistor (You will have to lift one leg of the resistor and insert the current meter in series, and move the probe connection to the current shunt position on the meter panel as shown below). Repeat the measurement with the Agilent 33401A

Part III: Pspice Simulation of a Series Resistor Voltage Divider

Enter the circuit of Fig 1-2 into the Pspice Schematic, with Vs = 9VDC, R1 = 1kΩ, and R2 = 560Ω.

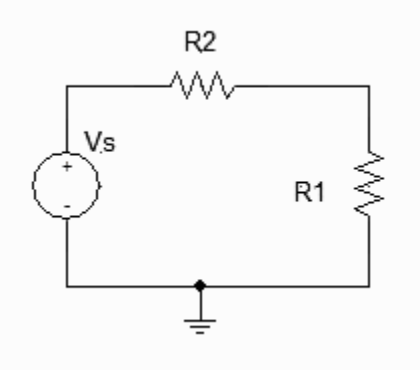


Figure 1-2

The easy way: modify the 1-1 schematic.

1. If it is not still open, start pspice and open your 1-1 schematic.
   Save it as Lab1-III (File → Save As → Lab1-III).

2. Right click the top wire near the voltage source – it will change color. Hit the delete key to remove the wire.

3. Insert a resistor as we did for part A or by using the "remembered part" drop down on the tool bar  (if the resistor is still there) in the space where the deleted wire was.
   Double click the resistor and set "REFDES" = "R2" and "VALUE" = "560".

4. Reconnect the wires to the resistor to complete the circuit.

5. At the bottom right corner of the schematic double click Lab 1-I and change it to Lab 1-III (and also change the date if needed).

6. Save your file!

Run the circuit simulation

7. Make sure that the Bias Display buttons are active. 

8. Click the simulation button to run the simulation as in part A. If there are no errors and the simulation completes, close the simulation window and go to the Schematic window. You should see the voltages and currents displayed. If you have errors, check your wiring, resistor attributes, and VDC attributes.

9. Click the voltage and current bias displays and drag them slightly clear of the circuit components. Show your instructor this simulation and record your results on the Data Sheet. Note: V_{R2} will be the difference between the voltages measured at each end of R2 (Vs - V_{R1})

Run a circuit transient simulation

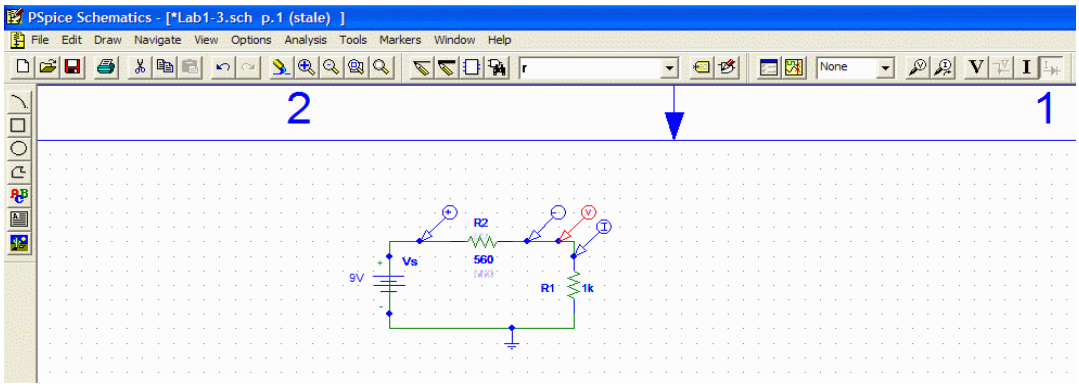
10. Click the Bias Display buttons to turn them off. 

11. Click the Voltage Probe (labeled V)  on the toolbar and place a voltage probe somewhere between R1 and R2 to measure V_{R1}.

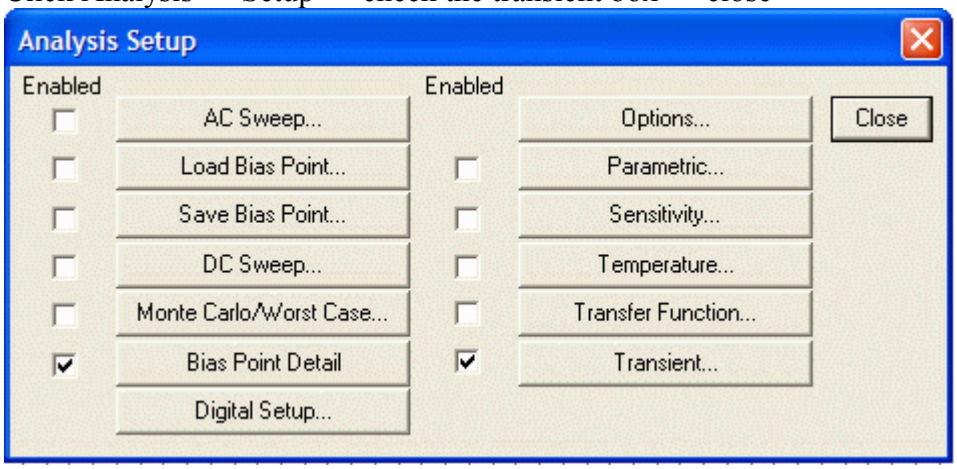
    Click Markers → Mark Voltage Differential
    place the first (+) marker between R2 and Vs to measure V_{R2}
    place the second (-) marker between R2 and R1 to measure V_{R2}

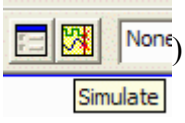
12. Click the Current probe (labeled I) and place a current probe at the top of R1 (NOTE: you will have to place the current probe at the connection point at the end of the resistor, it can be tricky to get the probe in the correct place). Hit the Esc key when you are done.

    Your schematic should now look something like this:



13. Click Analysis → Setup → check the transient box → close



14. Click Analysis → Simulate or click the Simulate button  (if you hold the cursor over the simulate button, the word Simulate will appear below the toolbar 

)

15. If there are no errors, the simulation will run and the result will show a line at about 5.8V, a line at about 3.7V, and another line near 0V. (If you have errors, make sure that your wires are all connected correctly, the resistor value has been set, and the power supply settings are correct.)

16. Click the "Toggle Cursor" button on the toolbar  The probe cursor value box should appear somewhere on the simulation near the lower right side. Move the mouse to 0.2s and click the left mouse button. A1 should show 200.000n and 5.762, 3.2308 or 5.769m, depending on which probe trace is active.[text moved]

17. Click the small square, diamond, or triangle to make a trace active. Note, your labels may be different.

18. Click the "Mark Label" button on the toolbar  . The values 200000n and some other number (for nS and V) should appear.

19. Click each of the other two traces (square, diamond, or triangle) and click the "Mark Label" for each trace selected.

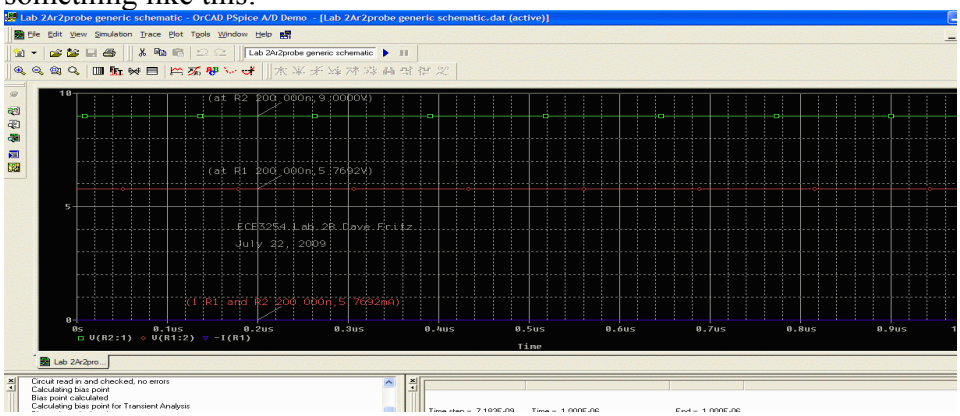
20. Click the "Toggle Cursor" button on the tool bar to turn off the cursor – the Probe Cursor box in instruction #7 should disappear.

21. Double click the top value label to bring up the label editor. Insert " VR1 = " at the beginning of the second number, and "V" at the end of the text string (should say: 200000n, VR1 = 5.7692V).
Double click the middle value label and insert " VR2 = " at the beginning of text label string, and "V" at the end of the text string (should say: 200000n, VR2 = 3.2308V).
Double click the bottom value label and insert " IR = " at the beginning of the current value, and "A" at the end of the text string (should say: 200000n, IR = 5.6792mA).

22. Click the "Text Label" button  on the tool bar and type ECE3254 Lab 1-III xxxx, where xxxx is your name. Click OK. Drag the text to somewhere near the center of the simulation and right click the mouse button to anchor the text.

23. Click the Text Label button on the tool bar and type the date. Anchor the date somewhere below the text that says ECE3254 Lab 1-III.

24. Show your instructor this simulation and record your results on the Data Sheet. It should look something like this:



Part IV: Measure the Series Resistor Voltage Divider

1. Obtain a 560Ω resistor (green, blue, brown, gold) from the parts drawer and measure its resistance with the two DMMs.

2. Using the protoboard, build the voltage divider shown in Fig 2-2. Keep V_S at 9V, let R_1 = 1000Ω (1kΩ) and let R_2. = 560Ω.

   Calculate the total resistance seen by Vs (and show equation used).

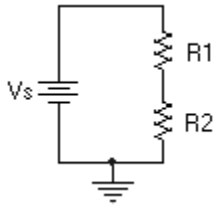


Fig 2-2

   Use either DMM to measure the voltages across R_1 and R_2 and the current through the resistors.

3. **Before you leave:** Shut down Windows, return cables to racks, return parts to correct drawer bins, return adapters to container, turn off bench power, clear bench, and place seat under bench. **(Counts as 5 points of the Lab grade)**

ECE 3254 Lab 1 Data Sheet **Date**_____ **Bench**_____
(100 points total)
Names: _____ _____ _____

Part I: Pspice Simulation of a Resistor and a DC Voltage Source

19. *Prelab V_{R1} expected* = _____*V*

Prelab I_{R1} expected = _____*mA*

Simulated Pspice V_{R1} = _____V (3)

Simulated Pspice I_{R1} = _____mA (3)

TA Verified Pspice simulation complete (names): (3)

_____ _____ _____

Compare the Pspice simulation with the prelab calculation. (5)

Part II: Measure the Voltage, Current, Resistance, and Verify the Pspice Simulation

2. With the Fluke 45 meter, the 1000Ω resistor measured resistance = _____Ω (2)

With the Agilent 33401A meter, 1000Ω resistor measured resistance = _____Ω (2)

3. *Prelab V_{R1} expected* = _____*V* Pspice V_{R1} expected = _____V (3)

Fluke 45 meas. V_{R1} =_____V (2) Agilent 33401A meas. V_{R1} =_____V (2)

4. *Prelab I_R expected* = _____*mA* Pspice I_R expected = _____mA (3)

Fluke 45 measured I_{R1} =_____mA (3) Agilent 33401A measured I_{R1} =_____mA (3)

Use the measured voltage and current to **calculate R** (show formula and calculation) (4)

Compare the measured resistances from the ohmmeters, the calculated resistance from the voltage and current measurements, and the resistor's marked value.(5)

Compare the resistor current and voltage measurements to the expected results from the prelab calculations and the pspice simulation. (5)

Part III: Pspice Simulation of a Series Resistor Voltage Divider

9. *Prelab V_{R1} expected* = _____*V* *Prelab V_{R2} expected* = _____*V*

Prelab I_R expected = _____*mA*

Simulated Pspice V_{R1} = _____V (3) Simulated Pspice V_{R2} = _____V (3)

Simulated Pspice I_R = _____mA (3) [Note: V_{R2} = V at R2 – V at R1]

TA Verified Pspice simulation complete (names): (5)

_____ _____ _____

24. Simulated Pspice V_{R1} = _____V (3) Simulated Pspice V_{R2} = _____V (3)

Simulated Pspice I_R = _____mA (3) [V_{R2} is directly measured by the differential probe]

TA Verified Pspice simulation complete (names): (5)

_____ _____ _____

Compare the Pspice simulations with the prelab calculations.(5)

Part IV: Measure the Series Resistor Voltage Divider

1. With the Fluke 45 meter, the 560Ω resistor measured resistance = _____Ω (2)

With the Agilent 33401A meter, 560Ω resistor measured resistance = _____Ω (2)

2. *Prelab V_{R1} expected* = _____*V* *Prelab V_{R2} expected* = _____*V*

Prelab I_R expected = _____*mA*

Expected R_{total} = _____Ω (2) Measured R total (R1+R2) = _____Ω (2)

Measured V_{R1} =_____V (2) Measured V_{R2} =_____V (2)

Measured I_{R1} =_____mA (3) Meter Used: _____ (1)

Compare the resistor current and voltage measurements to the expected results from the prelab calculations, and the pspice simulation from Part III. What may account for the differences?(5)

3. Shut down Windows, return cables to racks, return parts to correct drawer bins, return adapters to container, turn off bench power, clear bench, and place seat under bench. (5)