ECSE-2010   **Spring 2022`**   **Name_____Sec_____**

LABORATORY 1: LTSpice/Analog Discovery Introductory Circuits

PART A – Introduction to LTSpice
Materials need: Laptop

Choose a partner:
1) Pick a partner. You can choose up to two people for a maximum group of three. Remember, you will likely have to work with this person or persons for the rest of the semester
2) Only one laboratory report is needed for each group. Make sure you include both group members name on the front of the report. Also remember to add your partner(s) to Gradescope. (Only one person needs to submit it).
3) Please create a WebEx Teams space within the Spring 2021 Electric Circuits Team using your Last names, Alpha/Omega, and Section…for example, "Smith Jones Xu Alpha Lab Sec 1"
4) You will go through each lab section step by step. Then you will choose any of the circuits you went through (unless specified) to prove concepts listed at the end of each section in your Proof of Concepts document https://ecse.rpi.edu/~ssawyer/CircuitsSpring2022_all/Templates/AlphaOmega_ProofofConcepts.docx . You must answer any questions related to those concepts and provide mathematical calculation, simulation, and experimental data to support the proof of concept!
5) You are required to participate in six INDIVIDUAL check-ins by posting a short ***one-minute video in your Lab group's WebEx Space about your INDIVDUAL experience*** with the labs. An associated Gradescope online assignment will guide the content of your check-in video. An undergraduate student assistant is assigned to your lab group to continue the conversation, if needed, for mentorship.

Installation:
1) LTSpice
   a. Use the link http://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html
      i. Or find (LTSpice Download) on the Circuits webpage under Resources.
   b. Run the executable and follow the default installation.

LTSpice Starting a new projects:
1) Start "LTSpice XVII" (or earlier version)
2) Start a new Project under the File -> New Schematic
    a. Make sure your files are saved in a convenient directory. The root directory (C:\) or Desktop are probably not good choices. I would suggest creating a directory "C:\Circuits" and saving your work there.
    b. Pick an appropriate name (Laboratory_01 is a good choice)
3) To place components:
    a. The first time you start, click Edit in the task bar. Here you'll find symbols and their matching labels.
    b. You can choose to go down the menu and click Component or simply use the Toolbar above the schematic frame from this point on OR
        i. There are shortcut keys for adding components as well.
        ii. http://www.analog.com/media/en/simulation-models/spice-models/LTspice_ShortcutFlyerC.pdf?modelType=spice-models

A1: Bias Point Measurements for Resistive Circuits
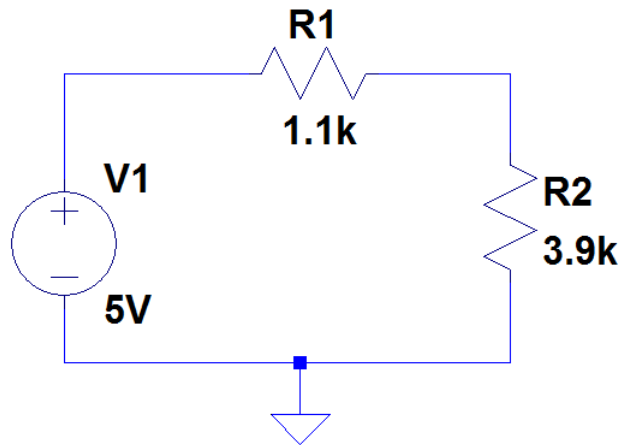
1) Create the circuit above in LTSpice and display the current, voltage and power values for the circuit similar to the diagram below. *The sample image below has different component values but gives an idea of what you need to show.*
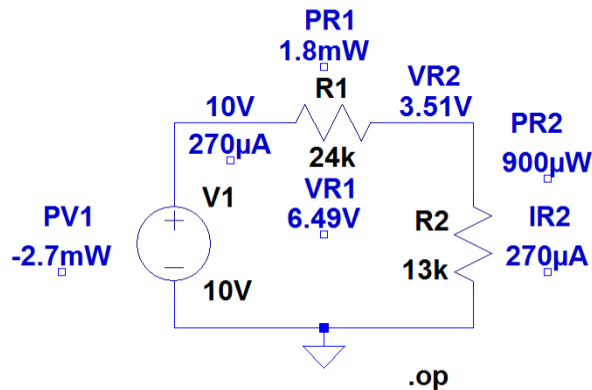
*Note: If you are confident that you can do this **without step-by-step instruction you can skip to A2.*** *Step-by-step instructions are below.*

2) Find and Add the Component (shortcut "F2"), VDC, by typing "voltage". Press "OK".
   a. A DC voltage source should follow your cursor.
   b. Left click to place. Press ESC to keep from duplicating. (If you make a mistake, you can press CTRL-V to cut the component).
   c. Move your cursor over the component. When a hand pointing left appears, right click, to edit the value of the DC voltage. Change the value to 5V.
3) Add resistors. (shortcut "R"). Press CTRL-R to rotate. Right click to place and ESC to keep from duplicating.
   a. They should be default labeled R1 and R2.
   c. Change values of R1 and R2 to 1.5k and 3.2k respectively by right clicking the component.
4) Connect the DC source and the two resistors in a closed loop by placing wire connections between each component. (shortcut "F3") Click on one of the square boxes connected to a circuit component and then move the mouse and click on another square box to place a wire connection.
5) Add a ground connection. (shortcut "G") You MUST use a ground in every circuit you simulate. Be sure to wire this as well.

Setting Operating Point DC Analysis

1) To add operating point analysis (.op) you have a few options
   a. Click ".op" in the tool bar on the far right and add text ".op" in the text box as shown below
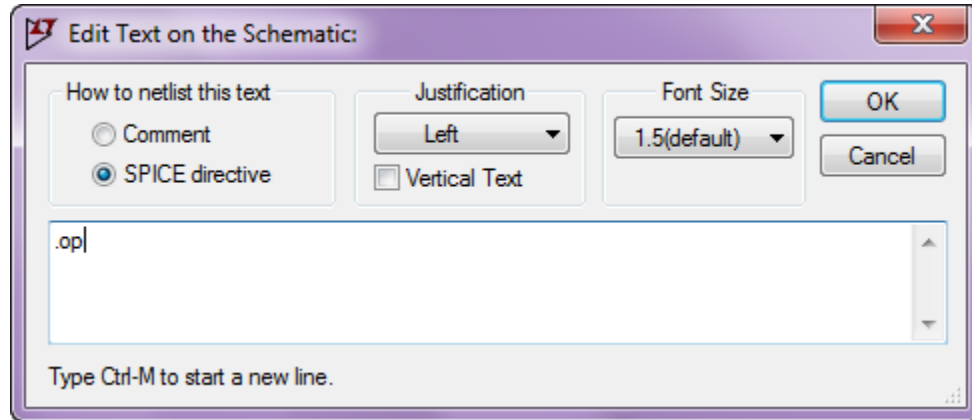
   b. Press shortcut "t" on your keyboard then change radio button to "SPICE directive" then add text ".op" in the text box below it
   c. Press "s" on your keyboard which automatically selects "SPICE directive" then add text ".op" in the text box below it
2) Place .op anywhere near the circuit
3) Run the simulation. (Click the "Running person" in the toolbox.)
4) Results will appear in a dialog box.
5) Label the circuit with voltage, current, and power by placing operating point data labels.
   a. FOR VR2:
      i. Right click on the wire to the right of R1 (this is the voltage above R2, Vn002 to ground which is the voltage across R2, VR2).
      ii. Right click to change the number of significant figures. For two decimal places use round(***data label****100)/100. For three significant figures multiply and divide by 1000. For four 10000 etc.
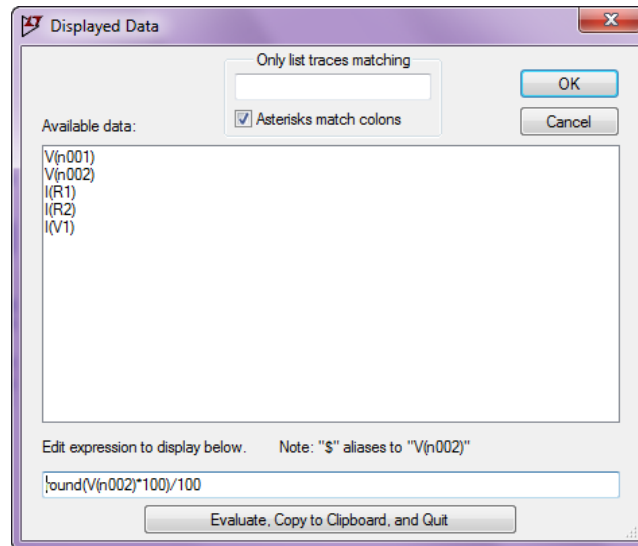
     iii. Press ok, and move the label anywhere convenient using the "Move" hand on the toolbar

  b. FOR IR2

     i. Right click on the wire to the right of R1 again. It will show voltage but you will change it by right clicking on it and choosing I(R2) which will show in the text box

     ii. Be sure to round this value to significant figures.

Note: LTSpice assumes conventional current flows from positive to negative terminal of the power supply (pointing down). Therefore, all resistors current will be labeled as negative in LTSpice. To make positive simply add a negative in front of I(R1) and I(R2).

  c. For PR2

     i. An equation can be written P=VI with rounding to significant figures is included round(I(R2)*V(n002)*10000)/10000

  d. Add text above all the numerical labels by pressing "t". The radio button should automatically be on "Comment". Type VR2 in the text box for example and place near the value in the figure.

  e. Repeat this process for the voltage current and power for R1 and power for the voltage source in your figure. *Remember VR1 is the difference between points V(n001) and V(n002) in the diagram. Also remember power supplied must equal power absorbed by the resistors!*

A2: Making Changes

1) Change R1 to 22k and R2 to 42k. Rerun the simulation.
    a. Comment on the new voltage and current measurements. Are the results consistent with expectations? Why?
    b. Did the power levels go up, down or stay the same? Why?
2) Replace the voltage source with a 5mA current source. (Component "current")
    a. What can you say about the current as R1 and R2 change?
    b. What happens to the power if both resistors are reduced back to their 1.5k and 3.2k values (from the 22k and 42k values)?
    c. Include a screen shot of your results in your report.

A3:   Unlimited-All Access Homework Solutions and Practice Problem Solutions!

1) Use LTSpice to implement the circuit in HW1, problem 5 (shown below but values are in HW1).
2) Using LTSpice, find the current through R5. When you complete HW1, compare your calculated answer with your simulated answered. (The LTSpice simulation does not constitute a solution when submitting HW1. When you hand in HW1, you must include your work demonstrating that you can obtain the solution by hand.)

3) Discuss the polarity of IR5, your handwritten calculations and how LTSpice determines current direction (look this up online if needed!)
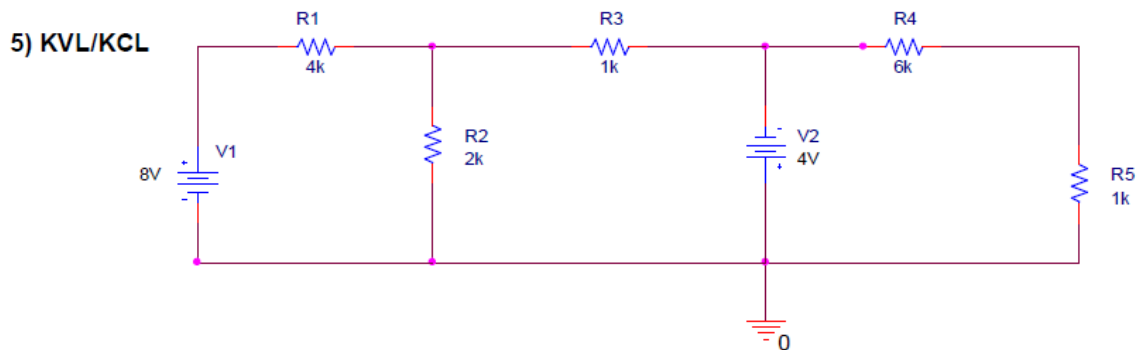
4) Change the location of your ground (move it to a different node) and verify that the current through R5 has not changed.

ECSE-2010   **Spring 2022`**   **Name_____Sec_____**

Part A: Proof of Concepts List

Prove Ohm's Law

Prove that power supplied must equal power absorbed (include polarity!)

Prove how the placement of ground affects the circuit (or doesn't affect it)

Prove the calculation of IR5 in HW1 using KCL/KVL

Note: The word prove means <u>demonstrate</u> using simulation, mathematical calculation, and experimental results to show that a concept is valid. It is not a mathematical definition of a proof. If you find limitations to that concept, include it!

Please include **screen shots** **of your results in your Proof of Concept Report.** Make sure they are easy to read and simulation results are labeled.

Template:
https://ecse.rpi.edu/~ssawyer/CircuitsFall2019_all/Labs/Circuits_OmegaLabDocs/ 04_Deliverables/03_ProofofConcepts.docx

> **Note for Part A, there isn't a built circuit to provide experimental results.** You should still provide analytical hand calculations and discuss your results while answering any questions in the lab.
>
> But for part B and C and future labs, you should compare analytical hand calculations, simulations, and experimental built circuits as often as possible. You will then discuss these results and answer any questions. **<u>The ultimate goal is to eliminate the question, "Is this right?" from you engineering vocabulary….forever. You can figure out just how right you are on your own!</u>** Soon you will ALSO DEFINE what is right….but not yet….

Part B: Analog Discovery Introductory Circuits

Materials need: Analog Discovery Board and Circuits Kit, Resistors

You may also use the ADALM1K Board but TAs are not trained on it yet. Also, instructions below are not for M1K boards. Feel free to explore using it and comment about any complications or approved changes made. Seek TA, SA for approval.

Installations:
1) Analog Discovery Board 1 (or Analog Discovery Board 2)
    a. In a web browser, open up the page
       https://store.digilentinc.com/waveforms-previously-waveforms-2015/
       Click on the link, "DOWNLOAD HERE"
    b. Follow the default install. If it asks about any Microsoft runtime libraries, let it install them.
    c. If you are using Windows 7, then open the Start Menu and find the Digilent tab and start the Waveform application. If you are using Windows 8 or Windows 10, find the Waveform application using the search tool and start it.
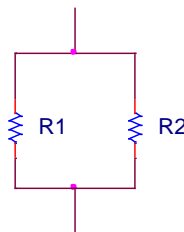
Background Circuit Concepts:
1) Series Resistors

   Resistors in series 'add' to an equivalent total resistance. For the two resistors shown above, the equivalent resistance is $R_{EQ} = R1 + R2$. For N resistors in series, this expression extends to $R_{EQ} = R1 + R2 + \cdots + RN$

2) Parallel Resistors

Resistors in parallel have an inverse relationship. For the two resistors shown above, the equivalent resistance is $\dfrac{1}{R_{EQ}} = \dfrac{1}{R1} + \dfrac{1}{R2}$. More generally, this expression is written as $R_{EQ} = \left(\dfrac{1}{R1} + \dfrac{1}{R2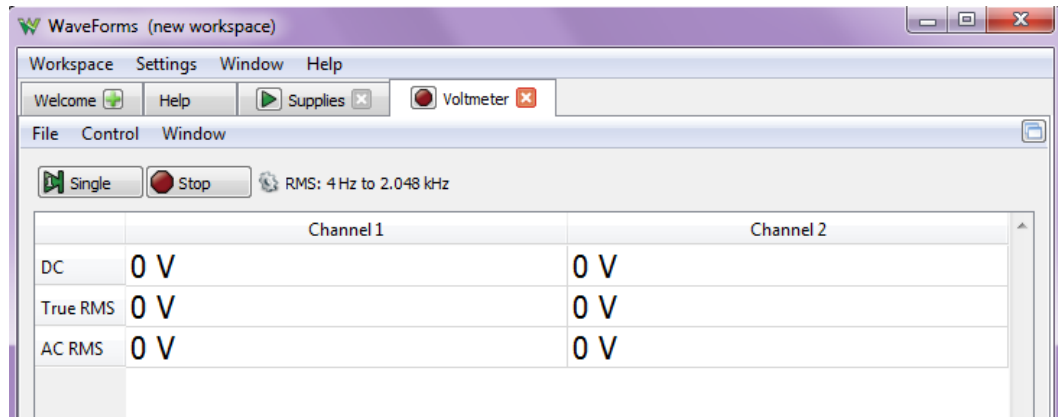} \right)^{-1}$. For N resistors in parallel, this expression extends to $R_{EQ} = \left(\dfrac{1}{R1} + \dfrac{1}{R2} + \cdots + \dfrac{1}{RN} \right)^{-1}$

Discovery Board basics:

1) Plug in the Discovery board using the USB connector
2) Start the Waveform applications (the Welcome tab should open).
3) In the window that opens, click on the Supplies icon (the third icon). You should see a DC power control window open.
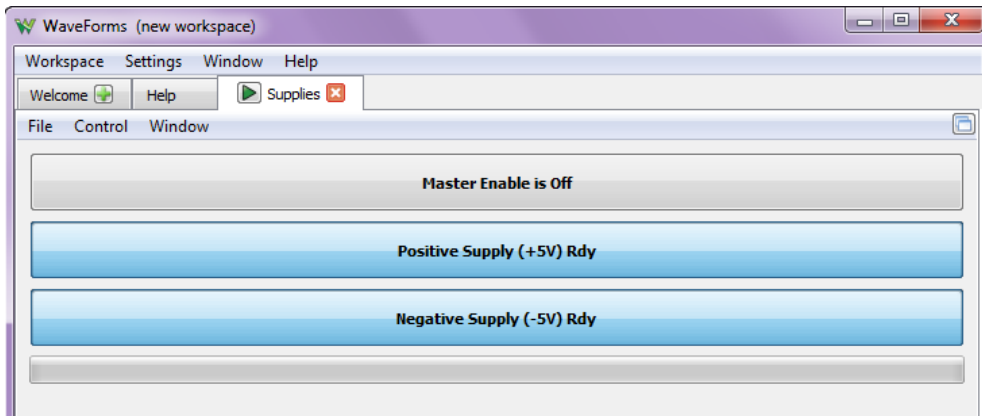
     a. Click the Master Enable is Off button to turn DC voltage on. It should now say Master Enable is On and be highlighted in blue.

     b. This provides access to a 5V/-5V supply using the V+/V- connections on the Discovery board.

     c. You find the connections and see that they are the solid red wire and the solid white wire, respectively.

4) To find Voltmeter, go back to the Welcome Tab

     a. Choose the forth icon, Voltmeter, and a new window should open.

     b. In the upper left corner of the Voltmeter window, click the Run box so that they you can use the channel inputs for digital measurements.

     c. For now, we only be using the DC measurements.

Written by J. Braunstein   *Modified by S. Sawyer Spring 2021: 1/19/2022*
Rensselaer Polytechnic Institute   *Troy, New York, USA*

9

d. There are two input channels, Channel 1 and Channel. They are represented as 1+/1- and 2+/2- on the discovery board.
e. Channel 1 is the solid orange wire (1+) and the orange wire with a white stripe (1-).
f. Channel 2 is the solid blue wire (2+) and the blue wire with a white stripe (2-).
g. Each channel needs two connections, to measure across a circuit component (equivalent to using a voltmeter)

B1: Simple Check - Power Supply Measurement

As an initial experiment to make sure everything is working correctly, we will measure the DC voltage using the channel 1 inputs.

Hardware
1) Connect six pin header to your proto-board, making sure that each pin is connected to a different row.
2) Connect the V+ (red wire) to one of the header pins and a ground (any of the black wires) to another pin.
3) Connect another six pin header to a different area on the proto-board. (This is for convenience, if you really want you can use only one six pin header).
4) Connect the 1+ (orange wire) to a header pin and the 1- (orange white stripe wite) to another header pins.
5) On your proto-board, connect the V+ row to the 1+ row and connect the ground row to the 1- row.

On the Voltmeter (computer)
1) Check the DC measured voltage. If everything is connected correctly and your software is setup, you will see a measurement of approximately 5 [V]

B2: Voltage Divider/Series Resistors

First, we will look at a simple voltage divider. Build the following circuit
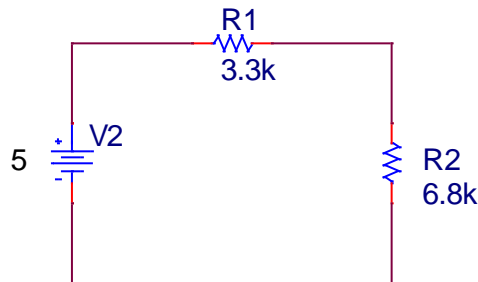
In this circuit, the positive side of the voltage source is the V+ connection (red wire) and the negative side of the voltage source is any of the grounds (black wires).

We will use the Discovery Board to make voltage measurements across each resistor.

Connect 1+ (orange wire) and 1- (orange white stripe) across R1.

Voltage across R1: _____

Connect 2+ (blue wire) and 2- (blue white stripe) across R2.

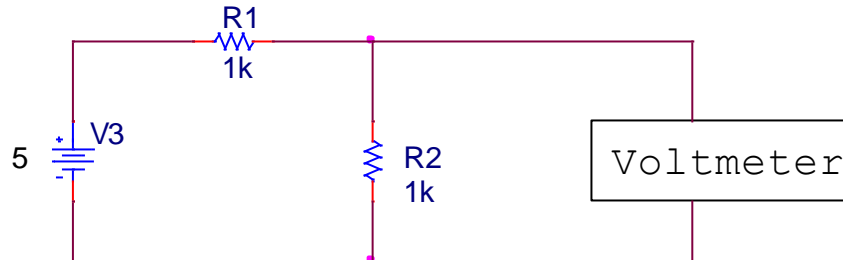Voltage across R2: _____

      a. Are these voltages consistent with expectations from KVL? In other words, does the voltage around the closed loop sum to zero? Note, the polarity of your connection matters. 1+ and 1- correspond to your 'guessed' polarity for Channel 1. Similarly, for Channel 2.

Replace R2 with a much larger resistor (100kΩ or more). Measure the voltage across R1 and R2 again. In this case we can <u>symbolically</u> say, VR2 ≈ _____

    b. With that in mind, circuit reduction in this simple circuit would result in an equivalent resistance when R2 >> R1, then REQ = R1+R2 ≈_____ (symbolically, not numerically).

B3: Effects of Measurement Equipment

1) Build the simple voltage divider circuit shown above. In a more complete model, the Voltmeter is part of the circuit. Ideally, it has no effect on the circuit.
2) Measure the voltage across R2.
    a. Does the measurement agree with expectations for the voltage divider circuit?
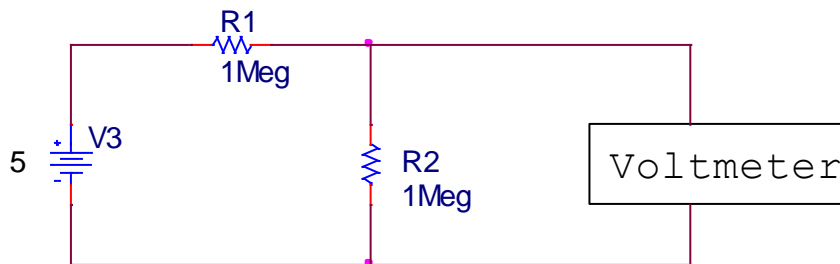
3) Replace the 1k resistors with 1Meg resistors and measure the voltage across R2 again.
    a. Does the measurement agree with expectations? What affect did the Channel 1 probes (the Voltmeter) have on the circuit? Why?

Part B: Proof of Concepts List

Prove circuit reduction (equivalent resistance)

Prove the effect of the measurement system on a circuit depending on load resistance for a voltage divider output

ECSE-2010   **Spring 2022`**   **Name_____Sec_____**

Part C: Alpha Laboratories Applications

C1: Voltage Divider as a Component (**not just a calculation**)

1) Name three ways a voltage divider can be used in a circuit. (You can Google it!)
2) Demonstrate one of those via simulation and DESCRIBE its purpose in a larger circuit. (Note: you CANNOT use the voltage divider for reducing a large total voltage in this example…too easy!)
3) Build the circuit and compare.
4) Have a conversation with an Omega Lab person or Circuits Mentors or Student Assistant on WebEx Teams. Ask them how they used or plan to use a voltage divider. Write their names down and what their application is.

Part C: Proof of Concepts List

Prove that voltage dividers are useful tools and not just a mathematical concept by demonstrating an interesting application

EXTRA CREDIT: Write in your metacognition journal (instructions and template in the link below, feel free to continue to edit a Google doc throughout the course to add entries).

https://ecse.rpi.edu/~ssawyer/CircuitsFall2019_all/Labs/Circuits_OmegaLabDocs/04_Deliverables/05_Circuits_Metacognition%20and%20Reflections.docx

SUMMARY of Concepts
Concept List that must be accounted for in your Proof of Concepts

PART A:
1. Prove Ohm's Law
2. Prove that power supplied must equal power absorbed (include polarity!)
3. Prove how the placement of ground affects the circuit (or doesn't affect it)
4. Prove the calculation of IR5 in HW1 using KCL/KVL

PART B:
1. Prove circuit reduction (equivalent resistance)
2. Prove the effect of the measurement system on a circuit depending on load resistance for a voltage divider output

PART C:
1. Prove (demonstrate) an interesting application for a voltage divider circuit