LABORATORY 1: Basic Analysis and Engineering Practices

Contents

Written by S. Sawyer and A. Patterson
Rensselaer Polytechnic Institute

Fall 2024: 9/20/2024
Troy, New York, USA

Getting Started!

Lab Submission Instructions and Partners:

1) You can choose **up to two** people for a maximum group of three. Groups of 2 are acceptable. Working alone is discouraged but if necessary, it is possible. 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 all group members' names 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 Intro to ECSE Team** using your Last names, Alpha/Omega, and Section…for example, "Smith | Jones | Xu | Alpha | Sec 1"

4) You will work through each core lab section to learn some basic skills. **Then you will prove specified concepts listed at the end of each section.** This simply means that you will demonstrate how a specific concept works through clear and concise comparisons of mathematical analysis, simulation, and experimental measurements.

5) The template for the **Proof of Concepts** document that you will submit can be found here: https://sites.ecse.rpi.edu/courses/F24/ECSE-1010/resources/labs/templates/Proof_of_Concepts_Template_F24.docx

   Please answer any questions related to those concepts and provide mathematical calculation, simulation, and experimental data to support the proof of concept!

Purpose:

The objectives of this experiment are to gain some experience with the tools we use (i.e. the electronic test and measuring equipment and the analysis software) and to gain some fundamental understanding of voltage dividers as both a mathematical concept and a component you can use. ***In ECSE Math is REAL and can reveal useful information OR be part of an entire system you can implement to solve problems! We start with simple division.***

Student Preparation BEFORE doing this experiment: (Students should be able to)

- Apply Ohm's Law to determine the current through a resistor
- Determine the values of series and parallel combinations of resistors
- Download and install software on a Windows machine

Learning Outcomes AFTER doing this experiment: (Students will be able to)

- Simulate a simple resistive voltage divider using LTSpice
- Build and test a simple resistive voltage divider
- Apply circuit reduction concepts to resistive circuits to simplify the analysis to a simple equivalent circuit
- Be able to build simple resistive circuits driven by constant voltages using a small breadboard
- Articulate a series of questions posed about simple circuits and answer the questions using data obtained from physical experiments.
- Demonstrate to themselves and others the difference between tinkering and engineering by answering the question "Is this right?" through documenting the comparison of mathematical analysis, simulation, and experiment to prove a concept or an idea!

Software and Equipment Required:

- LTSpice
- MATLAB
- ADALM2000 (M2K) board with Scopy or Analog Discovery with Digilent Waveforms
- Breadboard
- Several different resistors and wires from the parts kit (ADALP2000)
- Alpha Experiment: potentiometer
- Omega Exploration Optional!: resistive sensors, Arduino, push buttons or switches

Learning from Proof of Skills applied to this lab:

Professional Accountability:

- I **can clearly document and compare a calculated, simulated, and experimental result** to answer the question "Is this right?" for myself

Circuit Simulation (LTSpice or equivalent):

- I can use operating point DC **analysis** to find voltages across a resistive circuit.
- I can step through parameters with **parametric analysis** to repeatedly measure voltages as I vary my resistance over a range of values.

Experimental Measurement and Personal Instrumentation:

- I can use my instrumentation board's function generator to **create a DC, sinusoid, and pulsed signal** and measure with its oscilloscope directly.
- I can **build a resistive circuit** and **measure DC voltage across ONE resistor using a dc input source** and vary dc voltage at least 3 times (-5, +5 and any voltage in between)

PART A [Core] – Math as a Tool to Help You Understand…

Material covered: Resistor Combinations and Voltage Dividers

Background: Series and Parallel Circuits and Voltage Divider

Series and Parallel Circuits: A fundamental concept we need to understand in order to analyze the circuits we will build is how to mathematically combine resistances.

If any number of resistances are connected in series, you simply add them to find the total resistance, $R_{Tot} = \sum_1^N R_i = R_1 + R_2 + \cdots + R_N$. This is summarized in **Figure 1** with two resistors in series. If we measure the voltage at nodes A and B (use the two leads of Channel 1 and Channel 2 of your instrumentation board) and apply Ohm's Law to get the current, the current will only be the same in the two circuits if they have the same equivalent resistance. In the figure, this is shown by the relationship R3 = R1+R2 where R1 and R2 are in series and can be added to give a single equivalent value.

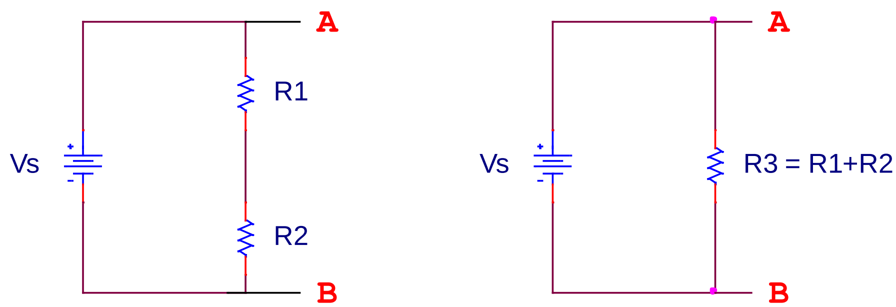


Figure 1: Two Resistors in Series

We can extend this to any number of resistors in series, as seen in **Figure 2**. The equivalent resistance is the sum of all the resistors in series.

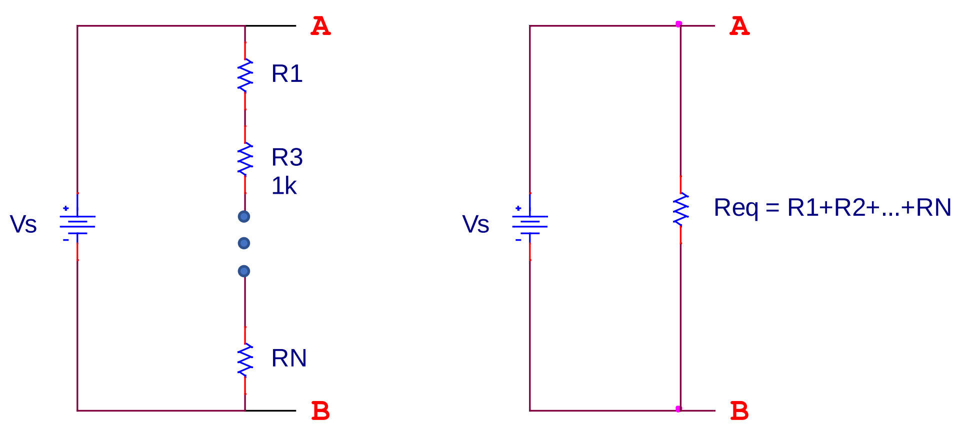


Figure 2: N Resistors in Series

Important Note: Two resistors are in series only if the end of one resistor is directly connected to one end of the other resistor. There cannot be anything else connected in between the two resistors.

If any number of resistances are connected in parallel, we need to take the inverse of the sum of the inverses, $R_{Tot} = \frac{1}{\sum_1^N \frac{1}{R_i}} = \left[\frac{1}{R_1} + \frac{1}{R_2} + \cdots + \frac{1}{R_N}\right]^{-1}$. This is summarized in **Figure 3** with two resistors in parallel. If we measure the voltage at nodes A and B (use the two leads of Channel 1 and Channel 2 of your instrumentation board) and apply Ohm's Law to get the total current, the total current will only be the same in the two circuits if they have the same equivalent resistance. In the figure, this is shown by the relationship $R_3 = \left[\frac{1}{R_1} + \frac{1}{R_2}\right]^{-1}$, where R1 and R2 are in parallel.

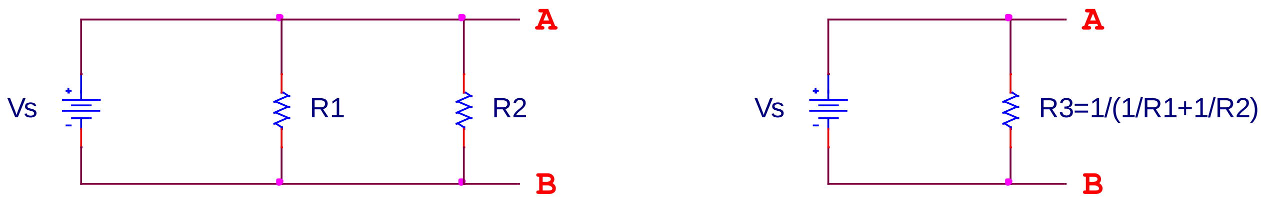


Figure 3: Two Resistors in Parallel

We can extend this to any number of resistors in parallel, as seen in **Figure 4**. The equivalent resistance is the inverse of the sum of the inverses.

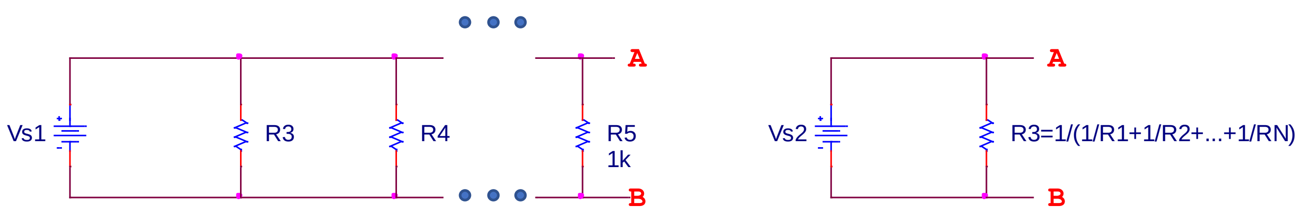


Figure 4: N Resistors in Parallel

Important Note: Resistors are only in parallel if both 'sides/ends' are connected together.

Voltage Dividers: In order to analyze the effect of the equipment, we need to understand a fundamental concept of circuit analysis called a voltage divider. When a voltage in a circuit is applied across two or more resistances, it divides up in a manner proportional to the magnitudes of the resistances. That is, a larger resistance will have a larger voltage drop across it and that voltage drop will be proportional to the magnitude of the resistance divided by the total resistance of a series circuit.

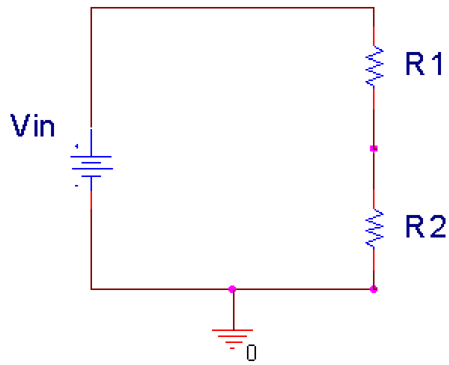


Figure 5

In **Figure 5** above, Vin is divided between R1 and R2. Mathematically, this can be expressed as:

$$V_{in} = V_{R1} + V_{R2} \qquad V_{R1} = \frac{R1}{R1 + R2} V_{in} \qquad V_{R2} = \frac{R2}{R1 + R2} V_{in}$$

In the voltage divider circuit, R1 and R2 are in series. Using the series expressions on the previous page, we can determine the total resistance 'seen' by the source, $R_{tot} = R1 + R2$.

Series and Parallel Circuits and Voltage Divider Experiment:

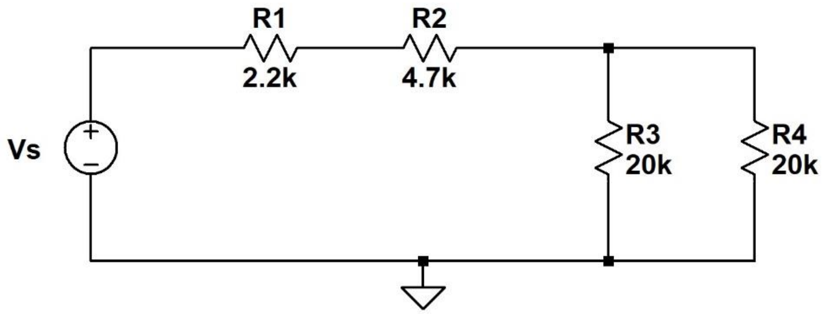


Figure A-1 Simulate this circuit using LTSpice

NOTE: YOU ARE FREE TO **CHOOSE ANY VALUE OF RESISTANCES** YOU WANT FOR THIS CIRCUIT. **YOU CAN ALSO REARRANGE COMPONENTS** IN THE CIRCUIT TO PROVE THE CONCEPTS IN THE CONCEPT LIST AT THE END OF THIS SECTION. JUST BE SURE TO SHOW YOUR SCHEMATIC IN THE PROOF OF CONCEPTS "BUILDING BLOCK" SECTION.

Use the circuit above (or a variation of it) to prove the concepts listed at the end of this part of the lab. Also include your answers to each of the questions below in the corresponding sections for your Proof of Concepts.

Analysis

1. Which resistors are in series above? Which resistors are in parallel above?
2. Combine the two parallel resistors to make one resistor. Calculate the value.
3. Using the voltage divider equation calculate the voltage across R1 and R2.
4. Using the voltage divider equation calculate the voltage across R3 and R4.
5. Use Ohm's law to find the current through all resistors.
6. Using KCL, determine the relationship between the current through R2 and the current through R3 and R4. (Write at least one equation that defines this relationship. Yes, there is more than one!)
7. Use these calculations, including values and equations, in the Proof of Concepts document under "Analysis" section for the appropriate concepts (list shown in Part A: Proof of Concepts list in the gray box below).

Simulation (LTSpice)

1. Using LTSpice operating point DC analysis, find the voltage across each resistor and the current through each resistor. Label each voltage and current with the output results on the schematic showing only 3 significant figures.
2. Use the schematic as the "Building Block" portion in the Proof of Concept document for every Concept.
3. Use the appropriate LTSpice output as the simulation result under "Simulation" for each concept below.

4. ***Optional:*** Find at least one additional way to demonstrate voltage and current for resistor R1 using LTSpice simulation options (other than operation point DC analysis)

Experimental Measurement (M2K or Analog Discovery Board)

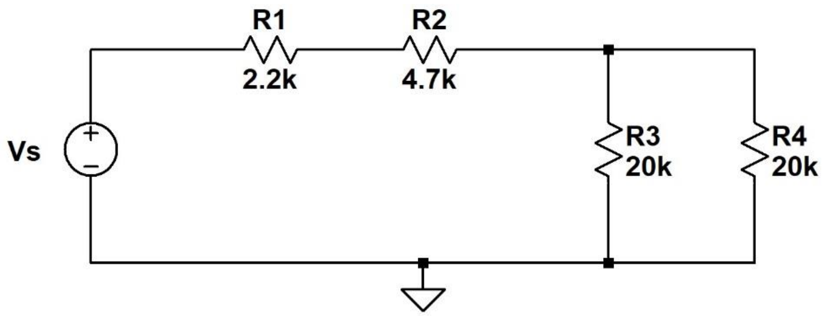


Figure A-2: Build this circuit with the exact values you simulated above *(which could be different than these shown in the diagram above…as long as the built circuit, simulation, and calculations all match you are good to go!)*

You should be able to quickly recognize the two resistors in series and the two resistors in parallel in the circuit shown in Figure A-2. Build the circuit on your breadboard and measure the voltage across the two resistors in series and across the two resistors in parallel. Figure A-3 is the same circuit as in A-2, but with the locations of the DC source (V+) and measurement (1+/- and 2+/-) connections of your instrumentation board shown.

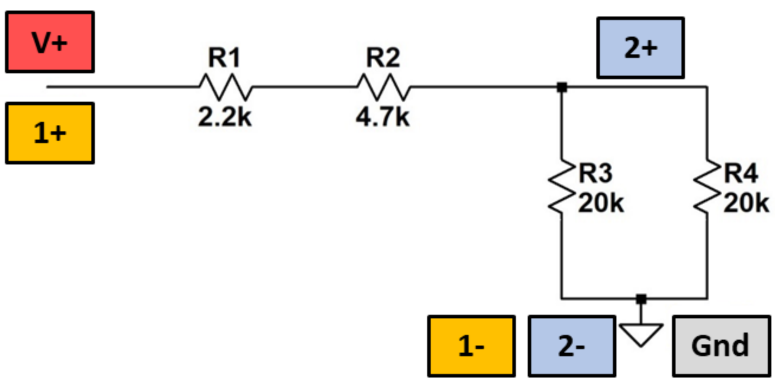


Figure A-3: The circuit from A-2, labeled with the locations of the leads from an instrumentation board

- Set the positive DC supply (V+) voltage to 4V and record the voltage measurements from Channel 1 (1+/-) and Channel 2 (+/-) using the voltmeter tool.
- Determine the equivalent resistance of the two resistors in series and the equivalent resistance of the two resistors in parallel.

- Redraw the simplified circuit with the equivalent resistances. (You can place this in your Proof of Concepts document for concepts it applies to in the list below…)
- Considering the two equivalent resistances, find the closest valued resistors in your parts kit and build the simplified equivalent circuit. (Keep your above circuit and build the new circuit somewhere else on your breadboard.) Measure the voltage across the two equivalent resistors.

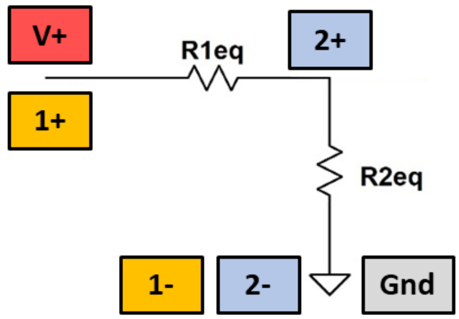


Figure A-4: The circuit from A-3 with R1, R2, R3 and R4 replaced by equivalent resistances

Part A: Proof of Concepts List

1. **Prove Ohm's Law, KCL, and KVL in a circuit.**
2. **Prove the concept of a voltage divider in a series circuit**
3. **Prove the concept of how current flows in a series circuit (same current through R1 and R2 for example? Or different current?)**
4. **Prove the concept of voltage across a parallel circuit (same voltage? Or different….)**
5. **Prove the concept of a *current divider* in a parallel circuit. *Hint: Try to search for this equation online. Use other current calculations and a simulation to confirm!***

Part B [Alpha/Omega] - Math as a COMPONENT you can use in a system….

Material covered: Voltage divider applications and design

Voltage Divider Applications and Design Background:

In the 2nd Year course that follows Intro to ECSE, ECSE 2010 Electric Circuits, we play a game called "Can a voltage divider save the world?" Each team comes up with as many real-world applications of a voltage divider as they can. As a class, the highest number of uses was 21.5 DIFFERENT ways a simple voltage divider can do real things in the world. You may choose to take the more defined Alpha experiment path to understanding the voltage divider as a useful component or explore on your own using the Omega design ideas below. Each path requires a Proof of Concept entry and full discussion.

Choose Your Adventure! Alpha Experiments or Omega Exploration

Alpha Experiments

Potentiometers

1. Use LTSpice Parametric analysis in a simple voltage divider circuit
2. Find the potentiometer in your ADALP2000 tool kit and build your circuit on the breadboard. Measure with your instrumentation board.
3. Discuss how a potentiometer is a voltage divider and how it might be used in common everyday applications

See below for more instructions

Omega Exploration

Design your own voltage divider application circuit using **one** of the following:

- **Temperature Sensing Circuit** – design a circuit that measures temperature
- **PWM Control of an LED** – design a circuit that controls the brightness of an LED via a potentiometer
- **Push Button DAC for Game Controller** – design a circuit that outputs different voltages based on button presses

Technical and functional requirements for your application are listed below.

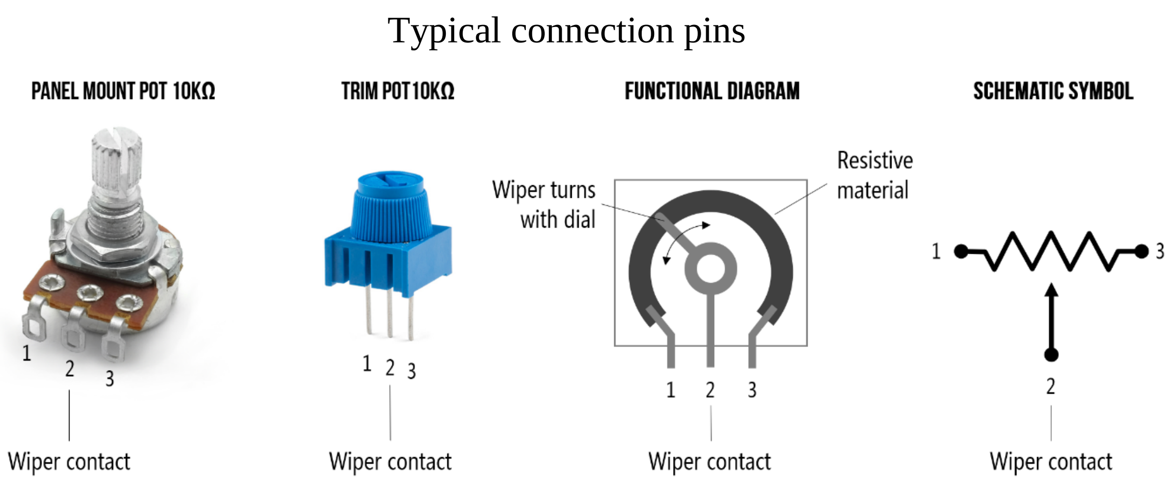
Part B: Proof of Concepts List

6. **Prove the Concept of a voltage divider in your application.**
   If you are doing an Omega Exploration, see the corresponding project description below for more details on your Proof of Concept.

Alpha Lab Experiment Guide

Mathematical Analysis

1. Find the 10kΩ potentiometer in your [ADALP2000 parts kit](#) as shown in the picture below. A potentiometer is a variable resistor, whose resistance you can change by turning a dial. It has three terminals or "legs" (labeled 1, 2 and 3) and two variable resistances: R12, the resistance measured between terminal 1 and 2, and R23, the resistance measured between terminals 2 and 3. The resistance measured between terminals 1 and 3 will always be the maximum or total resistance of the potentiometer, which in this case will be 10kΩ.



https://makeabilitylab.github.io/physcomp/arduino/potentiometers.html

2. Using an ohmmeter (either use a multimeter – ask a TA if you don't have access to one), measure the maximum and minimum resistance of either R12 or R23 (whichever you plan to use in your circuit) by turning the dial all the way in both directions.

3. You will use the circuit in Figure B-1 below for this Alpha Lab. R2 in the circuit below represents the potentiometer (either R12 or R23 – whichever you measured). Using circuit theory, calculate VR2 for the minimum resistance, maximum resistance, and three other resistances in between. You will compare this to simulations and experimental measurements.

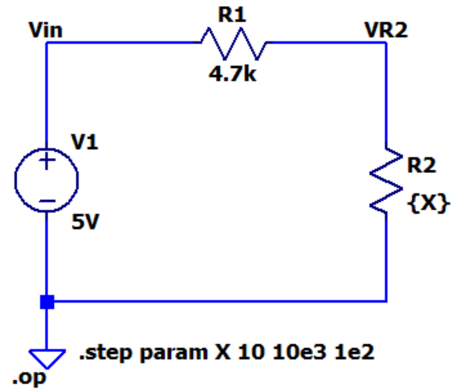
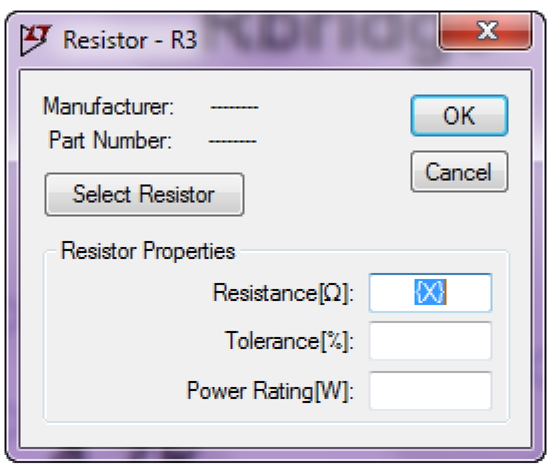


Figure B-1: Series circuit containing a 4.7kΩ (R1) and 10kΩ potentiometer (R2).

Simulation (LTSpice): *Parametric Analysis:* The ".step" command performs repeated analysis while stepping through specified values of a model parameter, global parameter or independent source.

1. Build the circuit in **Figure B-1** in LTSpice.
2. Define the component parameter by right clicking a resistor in your circuit, i.e. resistor R2 and entering "{X}" for the value of resistance (as shown in the diagram below).

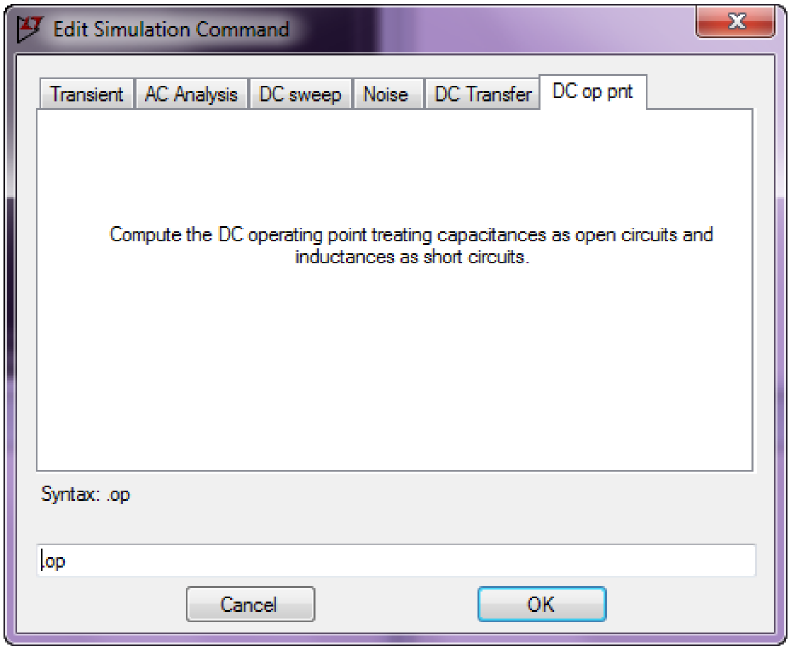


Note: eventually you will have to build the circuit you designed and you will use a potentiometer with a range of resistances from 0Ω to 10kΩ as your variable-valued resistor.

3. Add a ".step" command using a SPICE directive (press "s") which specifies the steps for a parameter.

Example: ".step param X 1 10k 1k" steps the parameter X from 1 ohm to 10k ohms in 1k ohm increments.
You may change the increments to a value that will give you more points.

4. Add ".op" in the SPICE directive. (Click ".op" at far right on the toolbar and add ".op" to the text field in the box that pops up, then place it anywhere on the schematic)
5. Run the simulation (click the "Running man"), go to "DC op pnt tab" and click "ok".



6. Run the simulation again. (click the "Running man"). The simulation pop-up window should show, but it won't have any traces. However, the x-axis should show the range of resistor values you specified in the ".step" directive.
7. To specify a differential voltage probe across a resistor (for example R2 in **Figure B-2**), click the node to the left of the resistor (a red probe should appear), hold, then release the click on the node to the right of the resistor (a black probe should appear).
8. The trace V(N00x, N00y) for the voltage across your chosen resistor (i.e. VR2) should appear (where n is some number label of node). Label the trace directly on the diagram accordingly (i.e. "VR2").
9. Pick points on the trace that correspond to the resistances you used for your calculations in the mathematical analysis section above and then compare them to the values you calculated via circuit theory.

Measurement (M2K or Analog Discovery Board)

1. Build the circuit in **Figure B-1** on your breadboard. Remember that only two of the potentiometer legs should be connected to the circuit and that you should use the same two terminals as you measured for your mathematical analysis.
2. Measure the voltage across the potentiometer (VR2) for the five resistances you chose for your mathematical analysis. Using an ohmmeter, turn the dial on the potentiometer until the ohmmeter shows the resistance you want to set the potentiometer to, then plug it into the circuit, and measure the voltage across R2. Repeat this procedure for all five resistances. *Note: be careful not to touch the dial on the potentiometer as you're placing it into your circuit!*
3. Verify that your measured voltages agree with both your mathematical analysis and simulation results.

Omega Exploration Application Requirements

All technical projects in engineering have a set of specifications or requirements that must be fulfilled for the project to be considered successfully completed. In ECSE 1010 - Introduction to ECSE, we will give you the specifications and you must design a circuit (or circuits) that satisfy those requirements. In a later course, ECSE 2010 - Electric Circuits, you will have the freedom to define your own Omega Exploration projects and specifications for success. The focus of Omega Explorations in Intro to ECSE is to allow you practice the design and documentation process on a smaller scale first, which will help to prepare you for more complex open-ended design projects in later courses.

A short description and the technical specifications for each Omega Exploration option are listed below.

Temperature Sensing Circuit

A thermistor is a sensor that changes its resistance based on its temperature. You will design and build a circuit whose voltage output will allow you to estimate the temperature of the thermistor.

Requirements:

- ☐ *Circuit architecture*:
  - Choose one: Wheatstone bridge **OR** single voltage divider. In your presentation, you must discuss the advantages and disadvantages of both approaches.
  - Must include a thermistor (circuit input stage)
  - Must include a potentiometer for calibration
- ☐ *Operating conditions*:
  - Input temperature range: 5C < T < 40C
  - Output voltage range: $|V_{out}| \leq 5\text{V}$
- ☐ *Output voltage*:
  - $V_{out}(T_{ref}) = V_{ref}$: at a reference temperature T_{ref} of your choice between 20C and 25C (roughly room temperature), your circuit must output an expected reference voltage V_{ref}.
- ☐ *Experimental Verification / Test Cases*:
  - Measure V_{out} for at least three temperatures ($T = 5$C, $T = T_{ref}$ and $T = 40$C).
  - Verify that the V_{out} matches your mathematical analysis & simulations for these temperatures.

Additional Note: you will need to *actually* measure temperature in order to complete this project. Your multimeter has the ability to measure temperature.

Pulse-Width Modulation (PWM) Control of an LED

In some cases, it is easier to control the voltage delivered to a device by sending rapid voltage pulses at a calculated rate and delivering an average voltage, rather than delivering the same steady voltage. This control method is called pulse-width modulation (PWM). You will design a system that controls the brightness of an LED via changes in the PWM signal, in response to turning a potentiometer at the input.

Requirements:

- ☐ *Circuit architecture*
  - Choose one: microcontroller (such as an Arduino) **OR** 555 Timer in Astable Multivibrator Mode to generate the PWM signal. In your presentation, you must discuss the advantages and disadvantages of both approaches.
  - Must use voltage divider with a potentiometer as the input
  - Must use an LED (with current limiting resistor) to indicate average output voltage level
- ☐ *Operating conditions*:
  - Voltage divider of the input stage should provide an adjustable voltage between approximately 0V and 5V via the potentiometer
- ☐ *Output requirements*:
  - Output voltage:
    - Must be a PWM signal
    - Minimum duty cycle: 0%
    - Maximum duty cycle: 100%
    - PWM signal duty cycle must be adjustable in real time via the potentiometer at the input
  - LED brightness should vary with the average voltage of the PWM signal
- ☐ *Experimental Verification*:
  - Demonstrate that the LED brightness varies between “off” and a maximum brightness by turning the potentiometer at the input.
  - Demonstrate the PWM voltage duty cycle varies between 0% and 100% via turning the potentiometer at the input.

Push Button DAC for Game Controller or Counter

A digital-to-analog converter (DAC) takes digital values (0 or 1) as inputs and outputs an analog, or continuous, range of voltages. In this application, you will design and build a circuit that outputs 16 different voltages based on the combination of buttons that is pressed.

Requirements:

- ☐ *Circuit architecture*:
  - Choose one: R-2R Ladder Network **OR** Voltage-mode Binary-Weighted Resistor DAC. In your presentation, you must discuss the advantages and disadvantages of both approaches.
  - Must include 4 push buttons as circuit inputs
- ☐ *Operating conditions*:
  - Any combination of buttons can be pressed simultaneously
  - Output voltage range: $0 < V_{out} \leq 5\text{V}$
- ☐ *Output requirements*:
  - 16 unique output voltages – one output voltage per combination of buttons pressed
  - Minimum output voltage: 0V
  - Maximum output voltage: $V_{out} \leq 5\text{V}$
  - Spacing between output voltages must be uniform and in the range: 0.1V - 0.3V.
- ☐ *Experimental Verification / Test Cases*:
  - By pressing the push buttons, demonstrate all 16 unique output voltages.
  - Verify that these voltages match your mathematical calculations & simulation results.

SUMMARY of Concepts

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

PART A:

1. **Prove Ohm's Law, KCL, and KVL in a circuit.**
2. **Prove the concept of a voltage divider in a series circuit.**
3. **Prove the concept of how current flows in a series circuit.**
4. **Prove the concept of voltage across a parallel circuit.**
5. **Prove the concept of a current divider in a parallel circuit.**

PART B:

6. **Demonstrate/Prove the Concept of a voltage divider in your application.**

How You'll Be Graded (with Standards Based Assessment):

You will be graded on the following Standards. For each Proof of Concept, you can receive up to 1 point for technical proficiency (can I demonstrate the concept?) and up to 1 point for the quality of your documentation (can I clearly document the work I did in proving the concept?). Please ensure to achieve each standard. If you do not, you can resubmit to the missing standard later in the semester. CLEARLY mark the changes you make in your Proof of Concept submission by either Tracking Changes in Word or highlighting changes by writing comments in a different color and/or changing the color of the updated work.

Lab 01 Standards

1. **I can prove Ohm's Law, KCL, and KVL in a circuit.**
2. **I can prove the concept of a voltage divider in a series circuit.**
3. **I can prove the concept of how current flows in a series circuit.**
4. **I can prove the concept of voltage across a parallel circuit.**
5. **I can prove the concept of a current divider in a parallel circuit.**
6. **I can prove the concept of a voltage divider in my application (alpha or omega).**