# LABORATORY 1: Basic Analysis and Circuit Fundamentals

## Contents



*Written by S. Sawyer, A. Patterson and P. Paul*
*Rensselaer Polytechnic Institute*

*Spring 2025: 2/4/2025*
*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. Remember to **add your partners to your Gradescope submissions**.  (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. See the following documents for more information on Proof of Concepts:
   - Overview of labs and lab documents
   - Proof of Concepts Template – use this document for your Proof of Concepts submissions
   - Proof of Concepts Formatting Checklist
   - Good Proof of Concepts Example

   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!

## 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

# Intro to ECSE Laboratories: Alpha Experiments and Omega Explorations

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

- Apply circuit reduction concepts to resistive circuits to simplify the analysis to a simple equivalent circuit
- Simulate a simple resistive voltage divider using LTSpice
- 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.
- Measure and plot the I-V characteristic of a resistor
- Calculate the power dissipated by a resistor and the total power supplied to a resistive circuit
- Use nodal analysis to solve for nodal voltages in a resistive circuit
- Use matrix math to solve a set of linearly independent equations.
- 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: Simscape
- Omega Exploration: resistive sensors, Arduino or 555 timer, push buttons or switches

## Learning from Proof of Skills applied to this lab:

### Circuit Simulation (LTSpice):

- I can use operating point DC analysis to find voltages across a resistive circuit.
- I can perform a simulation, then label the nodes on the circuit schematic with their numerical voltage values.
- 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)
- I can build a resistive circuit and then measure DC current through ONE resistor using a DC source (OR find another way if needed depending on board i.e. Math function on oscilloscope or handheld multimeter!). Note: your circuit MUST consist of 2 or more resistors.
- I can use my cursor function to show specific voltage and time points.

### Matlab and Simulink:

- I can find the solutions for linearly independent equations using the matrix function on my personal calculator (TI-XX) AND Matlab, then compare them.

- I can import simulation data from LTSpice into MATLAB and plot it.
- I can import experimental data from Scopy or Waveforms into MATLAB and plot it.
- I can use regression in Matlab to fit a curve to my data and produce a mathematical function for that curve.

## PART A [Core] – Resistor Combinations and Voltage Dividers

### Series and Parallel Circuits and Voltage Divider Experiment

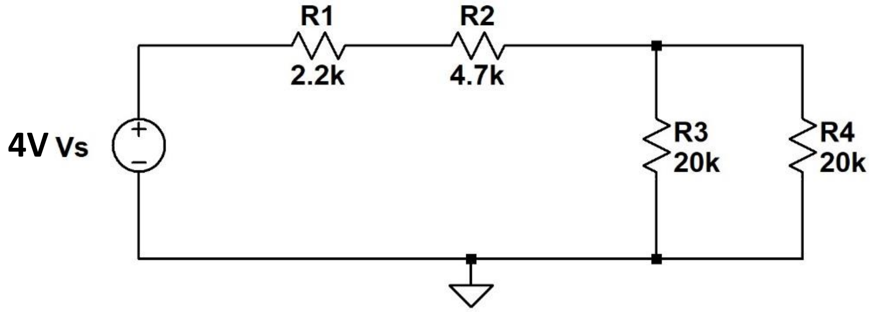



**Figure 1:** Simulate this circuit using LTSpice

*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**

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).

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 total 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!)

**Simulation (LTSpice)**

Use your LTSpice circuit schematic as the "Building Block" portion in the Proof of Concept document for every Concept. Also, use the appropriate LTSpice output as the simulation result under "Simulation" for each concept of your Proof of Concepts.

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. *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)**

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

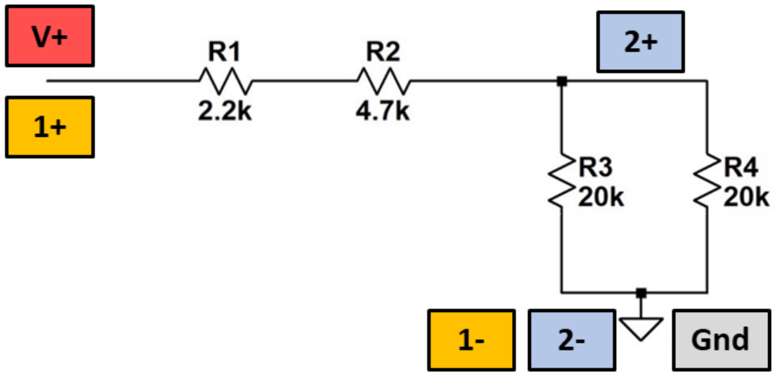



**Figure 2:** The circuit from Figure 1, labeled with the locations of the leads from an instrumentation board

1. 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.
2. Determine the equivalent resistance of the two resistors in series and the equivalent resistance of the two resistors in parallel.
3. 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…)
4. 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.

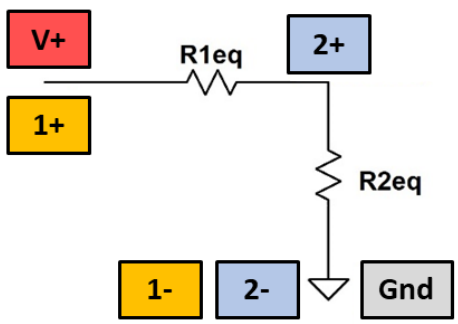



**Figure 3:** The circuit from Figure 2 with R1, R2, R3 and R4 replaced by equivalent resistances

**Part A: Proof of Concepts List**

1. **Prove the concept of a voltage divider in a series circuit**
2. **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 [Core] – Properties of Resistors

### I-V Characteristics of Resistors Experiment

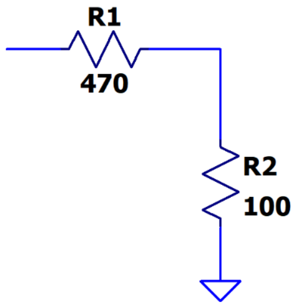



**Figure 4**: Set of two resistors for I-V measurements.

**Analysis**

1. Sketch the I-V characteristics for each of the resistors above by hand! Rock out to this ~1 minute video to learn how… https://youtu.be/ZDALtCd1M-U
2. Make the chart for R1 and R2 separately, then plot them together on the same x-y axis (current-voltage axis respectively) …you may use excel if you'd like or draw by hand. Either one is fine. You should have a line on your IV characteristic for each resistor.

**Simulation (LTSpice)**

1. Simulate the I-V characteristic of each of your resistors using parametric analysis. See https://iexploresiliconvalley.com/2019/02/03/ltspice-lesson-1-generating-iv-curves/
2. Export your I-V characteristics from LTSpice and plot them in Matlab with voltage on the x-axis and current on the y-axis.
3. Calculate the slope of your IV curve.

**Experimental Measurement (M2K or Analog Discovery Board)**

Using the circuit in Figure 5 you will measure the IV characteristic of a resistor. You will change V+ to obtain different voltages across resistors R1 and R2 and use an ammeter (a function of your multimeter) to measure the current through the circuit for each different DC voltage.

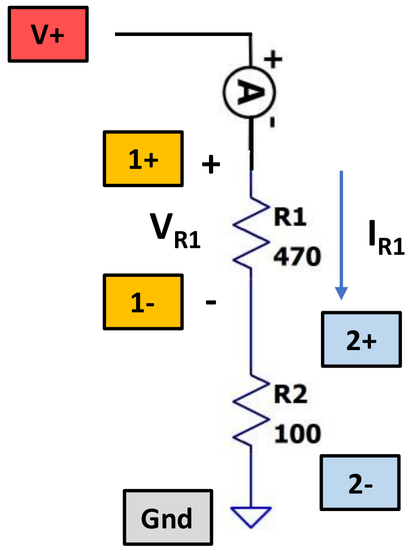



**Figure 5**: Circuit setup for measuring the IV characteristic of a resistor. The symbol with the "A" in a circle is that of an ammeter, which measures the current.

1. Build the circuit in Figure 5, including connecting the signal wires from your instrumentation board as shown. Be sure that you have both V+ (voltage source) and ground connected.

   To measure current, you will need to use the ammeter function on your multimeter. In the circuit in Figure 12, the ammeter is represented by the symbol of an "A" in a circle, which is in series with R1, R2, and the DC voltage source V+. *Note*: although you measure voltage by placing the measurement leads in parallel with the element whose voltage you want to measure, *you must place the ammeter in series with the element whose current you want to measure*. That means you'll need to leave a gap between the voltage source and R1 on your breadboard, then bridge that gap using the ammeter's measurement leads. See this short video on how to measure current using a multimeter.

2. Once your circuit is constructed, set the DC voltage source to 3V. Use channel 1 of your instrumentation board to measure the voltage across R1 and channel 2 to measure the voltage across R2 via your instrumentation board software's voltmeter. Record the values in the table under "R1 Voltage" and "R2 Voltage".

3. Use the multimeter to measure the current through R1 and R2. Take note of the units your multimeter is using to report current. Enter the current value in the table under "Current".

4. Repeat voltage and current measurements for R1 and R2 (separately) for values of V+ ranging from 3V down to 0V and fill out the table.

| V+ Voltage (V) | R1 Voltage (measured) | R2 Voltage (measured) | Current (measured) |
|---|---|---|---|
| 0 | | | |
| 0.5 | | | |
| 1 | | | |
| 1.5 | | | |
| 2 | | | |
| 2.5 | | | |
| 3 | | | |

5. Enter this data into MATLAB and plot the resistor voltage V (x-axis) vs the resistor current I (y-axis) for each of the resistors.
6. Fit a linear regression curve to the data for each of the resistors (if you've forgotten how, see Proof of Skills). From the slope of your fitted curves, determine the resistance of each of your resistors.
7. Using the ohmmeter function of your multimeter, measure the actual resistance of each of your resistors (measure them "in parallel" like you would measure the voltage across them). How close are their values to the resistances you determined from your IV measurement data and the linear regression?

## Power Consumption by Resistors Experiment

### Analysis

1. Solve for the voltage and current for each resistor in a circuit with a voltage source and multiple resistors.
2. Calculate the power consumed by each resistor in the circuit.
3. Calculate the total power supplied to the circuit by the power supply (i.e. voltage source).

### Simulation (LTSpice)

1. Simulate your circuit to determine the voltage and current for each resistor.
2. Calculate the power consumed by each resistor in the circuit.
3. Calculate the total power supplied to the circuit by the power supply (i.e. voltage source).

### Experimental Measurement (M2K or Analog Discovery Board)

1. Build your circuit on your breadboard.
2. Power the circuit with a voltage source, then measure the voltage across each resistor. Determine the current flowing through each resistor either by using a math channel on your instrumentation board's oscilloscope or measure the current using a multimeter.

3. Calculate the power being dissipated by each resistor.
4. Determine the power being supplied to your circuit by measuring the current flowing through the voltage source and the voltage being supplied by the voltage source.

**Part B: Proof of Concepts List**

3. **Prove that the slope of an I-V curve corresponds with Ohm's law for two different resistor values.**
4. **Prove the concept of conservation of energy by showing that the power supplied to a circuit is equal to the sum of the power consumed by its components.**

## PART C [Core] – Solving a Resistive Circuit using Nodal Analysis & Linear Algebra

### Nodal Analysis Experiment

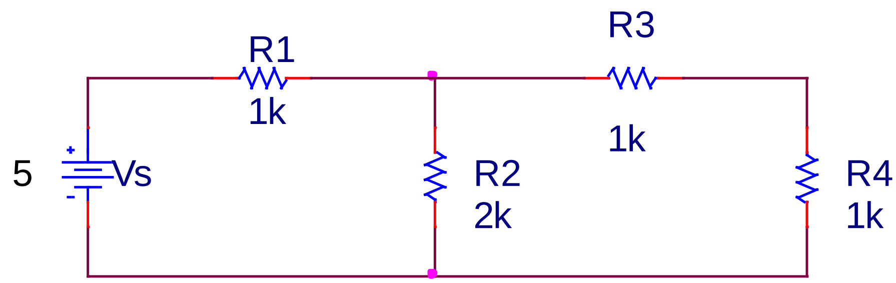



**Figure 6**: Four node circuit

In your Proof of Concepts, (Building Block) include the circuit schematic with each node labelled. You may choose any resistor values you want. You can also change the input voltage if you'd like.

**Analysis**

1. Using nodal analysis, solve for the voltages at each node in the circuit.

**Simulation (LTSpice)**

1. Simulate your circuit to determine the voltage at each node.

**Experimental Measurement (M2K or Analog Discovery Board)**

1. Build the circuit on your breadboard.
2. Power the circuit with a 5V voltage source, then measure the voltage at each node (with respect to ground).

**Part C: Proof of Concepts List**

5. **Prove the concept of solving for nodal voltages in a resistive circuit using nodal analysis.**

## Part D [Alpha/Omega] – Alpha Experiment and Omega Exploration

### Alpha Experiment: DC Motor Control with a PWM Signal

Motors use the current and voltage supplied to them to do work on mechanical systems. In the alpha experiment, you will explore the relationship between the power consumed by a DC motor and its rotation speed. You will also learn about controlling a DC motor via pulse-width modulation (PWM).

### Omega Exploration: Voltage Divider Design Background:

In the $2^{nd}$ 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. Explore a voltage divider application using the Omega design ideas below.

**Choose Your Adventure! Alpha Experiment or Omega Exploration**

### Alpha Experiment

Explore DC Motors and Power in Simscape

1. Complete training for "Simscape", a simulation platform within Simulink
2. Create a model of a DC motor and explore the relationship between power and its rotation speed
3. Control the DC motor using a PWM signal

**See below for more instructions**

### Omega Explorations

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.**

# Alpha Experiment Guide

Follow the instructions below to complete the alpha experiment for Lab 01. Instead of submitting a proof of concept for this experiment, you will answer questions regarding your simulation results and submit screenshots of them on Gradescope (assignment: "Lab 01 Alpha Experiment"). At the end of the Gradescope assignment, you will answer questions about what you conceptually learned by doing the experiment.

In this Alpha Experiment, you will learn how pulse-width modulation (PWM) can be used to control the speed and direction of a DC motor. You will also explore how parameters like the average voltage supplied to the motor and wheel mass affect the power consumed by the motor.

## Required Software

- Matlab
- Simulink
- Simscape
- Simscape Electrical

## Lab Instructions

### Pre-Lab Tasks

1. Ensure that the required software above is installed on your computer.
2. Complete the Simscape Onramp. If you can't find it under the "Learn" tab in Simulink, go to the Matlab learning website https://matlabacademy.mathworks.com/ to find it. It is recommended that you complete the training on the desktop version of Simscape. This should take about 1 hour, plus time to install Simscape.

### DC Motor Drive with DC Voltage Source

1. Open Matlab, then start Simulink either by typing "simulink" into the command window or clicking on the "Simulink" button in the "Home" tab of the Matlab window.
2. Open a blank model in Simulink and add the following blocks, so that it looks like the model in
   a. DC Voltage Source ($V_s$) (**Simscape > Foundation Library > Electrical > Electrical Sources**)
   b. Electrical Reference
   c. Solver Configuration
   d. DC Motor
   e. Current Sensor: you will also need to add a PS-Simulink Converter and a Scope (name the scope "Motor Current")
   f. Voltage Sensor: you will also need to add a PS-Simulink Converter and a Scope (name the scope "Motor Voltage")
   g. Inertia (name it "Wheel Mass")
   h. Mechanical Rotational Reference
   i. Ideal Rotational Motion Sensor: you will also need to add a PS-Simulink Converter and a Scope (name the scope "RPM")

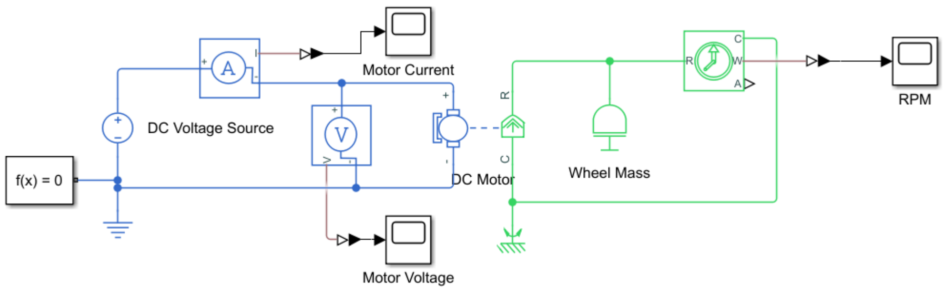
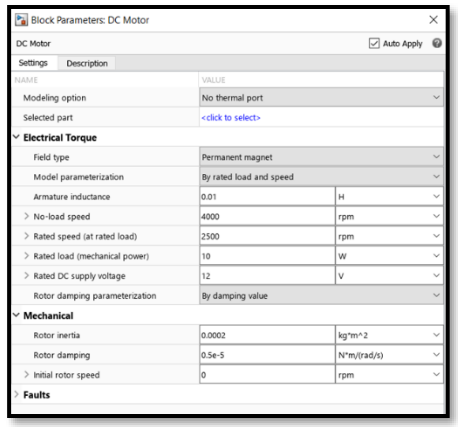



**Figure 7**: Model of DC motor with an inertial load driven by a DC voltage source

3. Add a "product" block to the model and use it to multiply current and voltage signals to create a power measurement. Add a scope to view the power vs. time. Name the scope "Motor Power".
4. Open the DC Motor block and use the following instructions to set the parameters:
    a. Expand the *Electrical Torque*
        i. In Model Parameterization section, change to "**By rated load and speed**"
        ii. Set the Armature inductance to **0.01 H**
        iii. Set No-load Speed to **4000 rpm**
        iv. Set Rated speed (at rated load) to **2500 rpm**
        v. Set Rated load (mechanical power) to **10 W**
        vi. Set Rated DC supply voltage to **12 V**
    b. Expand the *Mechanical*
        i. Set Rotor inertia to **0.0002 kg*m^2**
        ii. Set Rotor damping to **0.5e-5 N*m/(rad/s)**




5. Open the Inertia and set the inertia parameter to **0.0002 kg*m^2**.
6. Set simulation stop time to 15s.
7. Run the simulation for $V_s$ = 2.5V. Inspect the rotational speed speed ("RPM" scope) and power of the motor and answer the following questions on Gradescope.
    a. What is the final RPM of the motor?
    b. What is the peak power delivered to the motor?
    c. What relationship do you see between the power delivered to the motor and its rotational speed vs. time? How does this relate to the motor's acceleration and force?
8. Run the simulation again for $V_S$ = 1V and $V_S$ = 5V.
    a. What do you observe about the relationship between the peak power delivered to the motor and its final speed for different voltages?
9. Increase the inertia attached to the motor to **0.001 kg*m^2** and run the simulation again for $V_S$ = 2.5V. Answer the following questions:
    a. What is the effect of changing the mechanical load on the peak power delivered to the motor?
    b. What is the effect of changing the mechanical load on the final rotational speed achieved by the motor?

c. What is the effect of changing the mechanical load on the time it takes the motor to reach its final rotational speed?

**DC Motor Drive with a PWM Signal: Background on Pulse Width Modulation (PWM)**
*Pulse Width Modulation* (PWM) is a technique used to control power delivered to a device by changing the width of a square wave pulse, which switches between a constant voltage ON (high state) and constant voltage OFF (low state). PWM is advantageous for controlling DC motor speed for several reasons. First, DC motors do not respond the same way at low voltages and currents as they do at higher values, so it is easier to precisely control the speed by keeping the voltage values constant, but changing the amount of time that voltage is supplied to the motor. Secondly, motors require relatively large amounts of current to function, which devices like microcontrollers cannot supply themselves. Instead, the microcontroller sends a PWM signal to another switching device (such as an H-bridge) that turns the motors on and off, but with higher currents and voltages from a different power supply.

The percentage of the period that the PWM signal is ON (high) is known as the **duty cycle**. Several different duty cycles are shown in Figure 8 below.

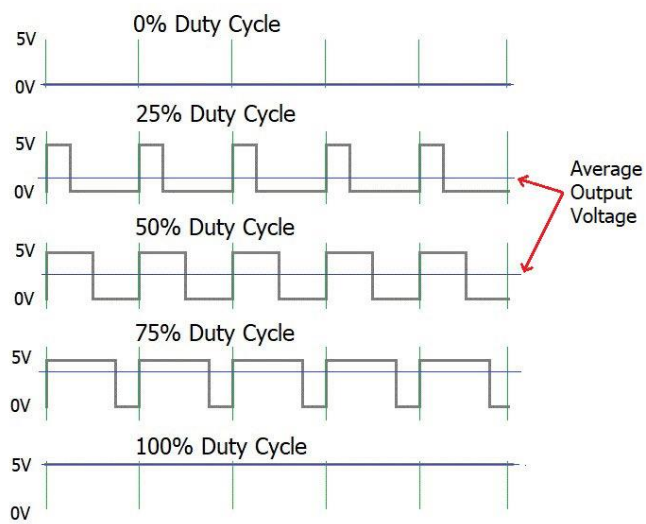



**Figure 8**: PWM in relation to Duty Cycle and Average Voltage Output
Source: https://circuitdigest.com/tutorial/what-is-pwm-pulse-width-modulation

The ON voltage and duty cycle together determine the average voltage delivered to the load. The relationship between the PWM signal's ON time, period and the duty cycle is shown below in Figure .

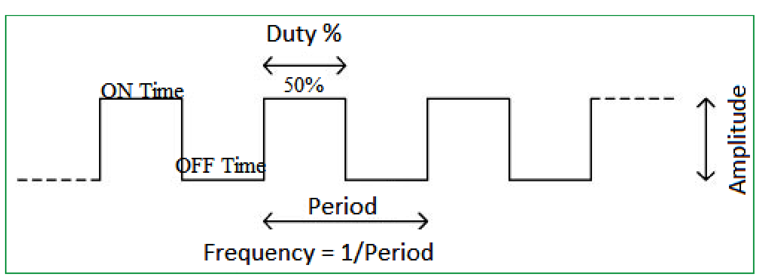



**Figure 9**: Relationship between duty cycle, pulse width and period
Source: https://circuitdigest.com/tutorial/what-is-pwm-pulse-width-modulation

The following formula is used to determine the duty cycle of the PWM signal:

$$\textbf{Duty Cycle (\%)} = \left(\frac{\boldsymbol{Time\ On}}{\boldsymbol{Square\ Wave\ Period}}\right) * \mathbf{100},$$

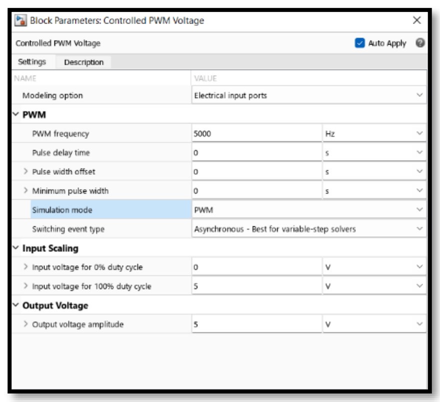
which can then be used to calculate the average voltage supplied to the load:

**Average voltage = Duty Cycle (%) x Supply Voltage**

**Experiment Instructions**

In this part of the lab, you will control the motor directly by using a PWM signal. This is not a common practice, since microcontrollers cannot supply enough power to run the motor (an H-bridge is usually used), but it will help you understand the effect of the PWM signal on motor voltage, current and rotational speed. The "Controlled PWM Voltage" block takes a reference voltage (the average voltage to be outputted) as an input and automatically calculates the pulse width and duty cycle needed to provide that average voltage at its output.

1. Add a "Controlled PWM Voltage" block between the DC voltage Source and the Current Sensor as shown below in Figure 10. Do the following to enter the settings in the Controlled PWM Voltage block:
    a. Extend the PWM section
        i. Set PWM frequency to **5000 Hz**
        ii. Set Simulation mode to "PWM"
    b. Extend the Input Scaling section
        i. Make sure the Input voltage for 0% duty cycle is set to **0 V**
        ii. Make sure the Input voltage for 100% duty cycle is set to **5 V**
    c. Extend the Output Voltage section
        i. Make sure the Output voltage amplitude is set to **5 V**



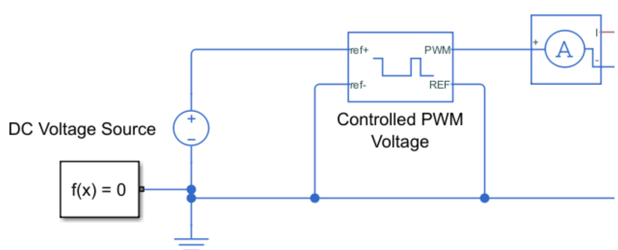




**Figure 10**: Model with Controlled PWM Voltage block and DC Voltage Source for PWM reference voltage added

2. Set the DC Voltage Source value to $\boldsymbol{V_s} = \mathbf{2.5V}$ and run the simulation.
    a. Does the final motor speed with the PWM signal match the final motor speed when the motor was being driven by a DC voltage source with $V_s = 2.5V$?
    b. Inspect the voltage, current, and power scopes. Do they show the same relationships as you saw before with the DC voltage source? If not, what is different?

3. Calculate the duty cycle that would be needed to provide an average voltage of 1V to the motor. Then, using the duty cycle, calculate the pulse width (pulse time) that would be needed to supply an average voltage of 1V to the motor using a PWM signal of frequency of 5000 Hz (see "Background on Pulse Width Modulation" above for details on how to calculate these values). Set the DC Voltage Source value to $\boldsymbol{V_s = 1\mathrm{V}}$ and run the simulation.

   Open the "Motor Voltage" scope and zoom in until you can see the details of the square wave being supplied to the motor. Measure the pulse width to verify your calculations above. Also verify that you obtain the same final motor speed as you did when directly supplying a 1V DC signal to the motor.

   a. What was your calculated pulse width?
   b. What pulse width did you measure?

4. As you should have seen, although the PWM signal changes the characteristics of the motor current, voltage, and power, it does not affect the plot of motor speed vs. time. This means that controlling the motor via the average voltage of a PWM signal is equivalent to controlling the motor using a constant DC voltage source, except that the simulation with the PWM signal takes much longer. The simulation speed can be improved by changing the Simulation mode of the "Controlled PWM Voltage" block. Open the "Controlled PWM Voltage" block and change the Simulation mode from "PWM" to "Averaged", which will only output the average voltage instead of a train of pulses. Since the voltage is now treated as constant, Simulink does not have to perform as many calculations and the motor speed vs. time will remain the same as in the "PWM" mode case.

5. Run the simulation with ref+ = 2.5V and 1V, and confirm that your motor speed vs. time scope results have not changed compared with the "PWM" simulation mode.

   a. Does your simulation run faster in "averaged" mode than it did in "PWM" mode?

## Summary

A DC motor converts electrical energy into mechanical energy. The power delivered to the motor is directly related to the angular acceleration of the motor, peaking when the motor's rotor just begins to rotate and gradually decreasing to zero as the motor reaches a constant speed (zero acceleration). Changing the mass attached to the motor's rotor does not affect the peak power delivered to the motor, but it takes the motor longer to reach its final rotational speed. Pulse-width modulation (PWM) provides precise control of the motor's speed by adjusting the duty cycle of the on/off time of the pulse, allowing for more stable control than simply varying the DC voltage.

## 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: $|V_{out}| \leq$ 5V
- ☐ *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 ($T =$ 5C, $T = T_{ref}$ and $T =$ 40C).
  - 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*:
  - Input voltage: 5V
  - Voltage divider of the input stage should provide an adjustable voltage via a potentiometer between approximately 0V and a maximum voltage as determined by the application
- ☐ *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 or switches as circuit inputs
- ☐ *Operating conditions*:
  - Any combination of buttons can be pressed simultaneously
  - Output voltage range: $0 < V_{out} \leq 5$V
- ☐ *Output requirements*:
  - 16 unique output voltages – one output voltage per combination of buttons pressed
  - Minimum output voltage: 0V
  - Maximum output voltage: $V_{out} \leq 5$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 the concept of a voltage divider in a series circuit**
2. **Prove the concept of a current divider in a parallel circuit.**

**Part B:**

3. **Prove that the slope of an I-V curve corresponds with ohm's law for two different resistor values.**
4. **Prove the concept of conservation of energy by showing that the power supplied to a circuit is equal to the sum of the power consumed by its components.**

**Part C:**

5. **Prove the concept of solving for nodal voltages in a resistive circuit using nodal analysis.**

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

You will be graded on the following Standards. For each Proof of Concept, you will receive points for technical proficiency (can I demonstrate the concept?) and points 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. **Prove the concept of a voltage divider in a series circuit**
2. **Prove the concept of a current divider in a parallel circuit.**
3. **Prove that the slope of an I-V curve corresponds with ohm's law for two different resistor values.**
4. **Prove the concept of conservation of energy by showing that the power supplied to a circuit is equal to the sum of the power consumed by its components.**
5. **Prove the concept of solving for nodal voltages in a resistive circuit using nodal analysis.**

# Appendix: Background Information

## Part A Background: Series and Parallel Circuits and Voltage Dividers

### Resistors in Series

How to mathematically combine resistances is a fundamental concept we need to understand in order to analyze the circuits. Combining resistances means representing two or more resistances as a single resistance, which behaves in the same way that the original collection of multiple resistances does.

Two resistors are in series 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 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 A1 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.

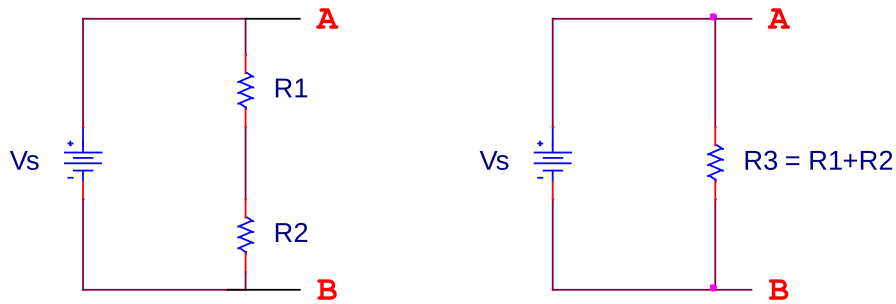



**Figure A1:** Two Resistors in Series

We can extend this to any number of resistors in series, as seen in Figure A2. The equivalent resistance is the sum of all the resistors in series.

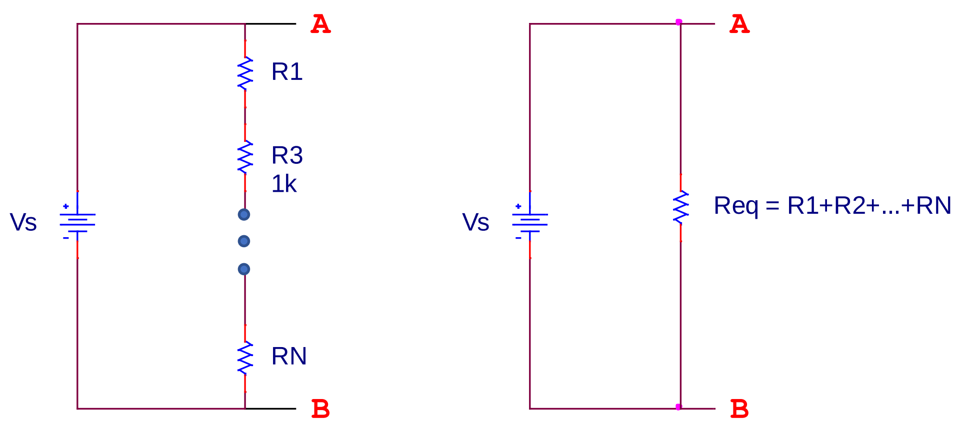



**Figure A2:** N Resistors in Series

**Resistors in Parallel**

Resistors are in parallel if both ends are connected together. 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 A3 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 $(R_1||R_2)$.

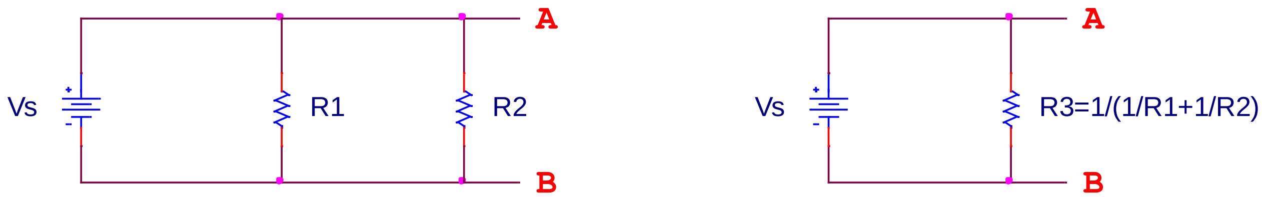



**Figure A3:** Two Resistors in Parallel

We can extend this to any number of resistors in parallel, as seen in Figure A4. The equivalent resistance is the inverse of the sum of the inverses.

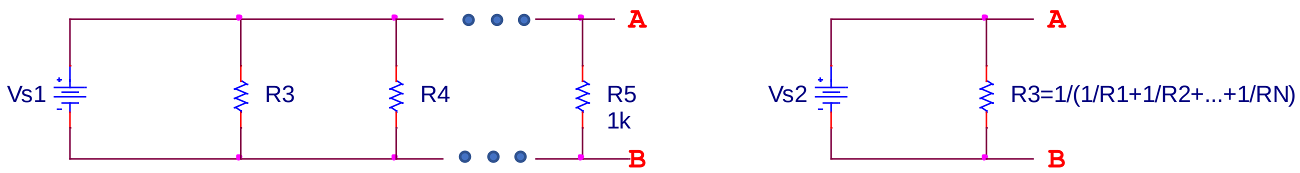



**Figure A4:** N Resistors in Parallel

**Voltage Dividers**

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.

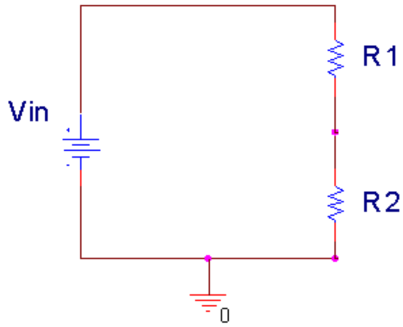



**Figure A5:** 2-Resistor Voltage Divider

In Figure A5 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, $R_{tot} = R1 + R2$.

## Part B Background: I-V Characteristics of Resistors

The relationship between current and voltage in a resistor is defined by Ohm's law, V = IR (Voltage = Current * Resistance). This equation is a linear relationship, following the classic expression, y = mx, where *m* is the slope. However, I-V characteristics are typically represented with voltage, V, as the x-axis and with current, I, as the y-axis, which requires us to rearrange Ohm's law to match this convention: $I = V/R$. As a result, the slope of a resistor's I-V characteristic is $1/R$.

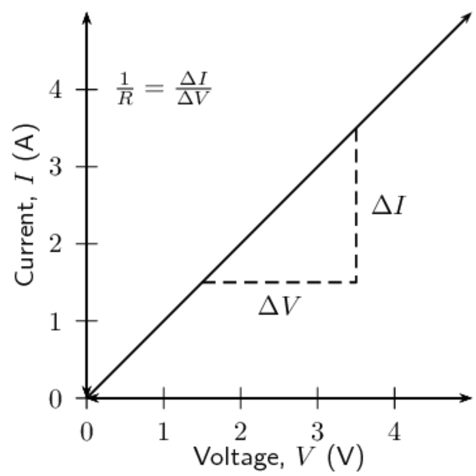



**Figure A6**: I-V characteristic of a resistor: Ohm's law.

## Part B Background: Power Consumption by Resistors

Components in circuits consume electric power either by using it to do work on some other system or by dissipating it as heat. The power consumed by a resistor is converted to heat and is equal to the voltage drop across it times the current flowing through it:

$$P = IV$$

Due to conservation of energy, the amount of power supplied by your power supply or function generator must be equal to the total power consumed by every component in the circuit. The power supplied to the circuit by the power supply is also given by the product of the current it is supplying to the circuit and the voltage of the supply: $P_{supply} = I_{supply}V_{supply}$.

## Part C Background: Nodal Analysis

### Nodal Voltages

In previous experiments we have discussed the relationship between voltage and current in a resistor (Ohm's law). The voltage we use in Ohm's law is always a voltage difference, i.e. the voltage on one side of the resistor minus the voltage on the other side of the resistor.

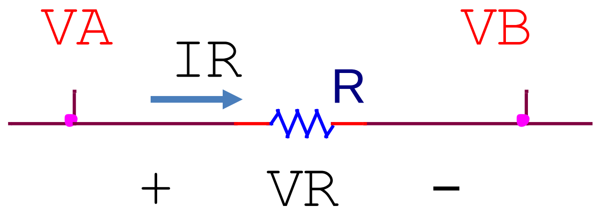



**Figure A7**: Voltage of resistor between nodes VA and VB

In Figure A7, the voltage across the resistor is $V_R = V_A - V_B$, as we saw in the previous laboratory. Likewise, the current through the resistor is $I_R = V_R/R$. We can then use the voltage difference expression and rewrite the current through the resistor as, $I_R = \frac{V_A - V_B}{R}$ . This form of the equation is used to set up a system of equations to find the voltage at every node in a circuit.

Important note: current in a resistor has direction and will always flow from a higher voltage node to a lower voltage node. In Figure A7, the '+' side of the resistor is assumed to be a higher voltage, indicating that current will flow to the 'right'. These designations are called the *polarity* of the voltage and current. It is possible that VB is a higher voltage than VA, which would result in VR being negative (using the expression $V_R = V_A - V_B$). If the voltage across a resistor is negative, then the current will also be negative since we always use the same polarity designations.

**Kirchhoff's Current Law (KCL)**

In words, KCL indicates that the total amount of current entering a node must be equal to the total amount of current leaving a node. In other words, there is no accumulated charge (electrons) at a node. Visually, we can represent that concept in Figure A8. Node VA has three paths (connections to various components. If we draw currents leaving the node, as shown in the figure, we can express KCL mathematically as,

$$I_1 + I_2 + I_3 = 0$$

In this expression, we notice that at least one of the currents must be negative. In other words, at least one of the currents must be entering the node. As mentioned above, a negative current just means we 'guessed' the wrong direction when we drew the arrow.

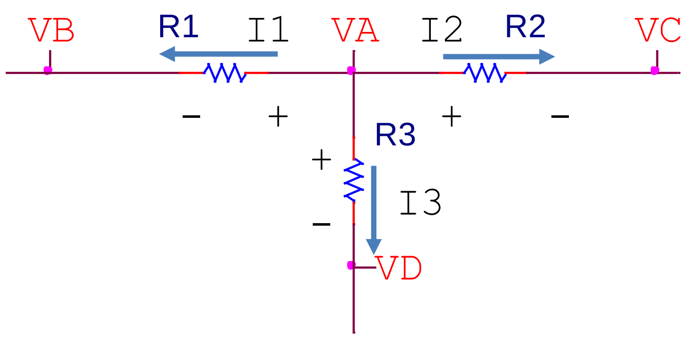



**Figure A8**: KCL applied at node VA with current leaving node

We can apply KCL to a node connected to resistors, as shown in Figure A8. Again, we assume currents are leaving the node. Based on that current polarity, we must assign a consistent voltage polarity, with the + side on the VA side of each resistor and the – side on the other node for each resistor (VB for R1, etc.). Applying KCL, we get the same expression as seen on the previous page. We can now apply Ohm's law to each of those currents and use voltage difference concepts to include the nodal voltages,

$$I_1 = \frac{V_A - V_B}{R1} \qquad I_2 = \frac{V_A - V_C}{R2} \qquad I_1 = \frac{V_A - V_D}{R3}$$

We can substitute these Ohm's law expressions back into the KCL expression, to obtain an equation

$$\frac{V_A - V_B}{R1} + \frac{V_A - V_C}{R2} + \frac{V_A - V_D}{R3} = 0$$

We now have one equation with four unknowns (VA, VB, VC, and VD). In order to find the values of each of those nodal voltages, we would need three more *linearly independent* equations.

**Summarizing Nodal Analysis**

To solve for the voltages at the nodes of a circuit, a summary of the process is as follows.

1. Label all the nodes in the circuit.
2. Pick a node to be common ground and set the voltage at the node to be zero.
3. For each voltage source, use voltage differences to find the voltage at another node (note, the voltage source needs to be connected to ground).
4. At each of the remaining unknown nodal voltage, apply KCL to get an equation
5. Implement the system of equations as a matrix expression, $Ax = b$.
6. Solve the matrix expression