Class #11: Experiment 11
Nodal Analysis Part II

Purpose: The objective of this experiment is to introduce circuits with more than one source and obtain a linear relationship between the sources and a voltage output. The concept of superposition will be discussed and used to demonstrate linearity for multiple sources.

Background: Before doing this experiment, students should be able to

- Analyze simple circuits consisting of combinations of resistors.
- Measure nodal voltages using the Voltmeter
- Implement resistive networks on the protoboard
- Use both Alice Meter-Source and Voltmeter tools with the M1K Board
- Review the background for the previous experiments.
- Calculate the slope of linear data

Learning Outcomes: Students will be able to

- Perform nodal analysis when more than one voltage source exists in the circuit
- Do a DC sweep simulation of circuits using LTspice.
- Understand that a system with more than one variable can be linear
- Apply experimental results to determine the superposition equation relating sources to an output

Equipment Required:

- **ADALM1000 or M1K Board** (with Alice software)
- **Meter-Source Tool** (Alice)
- **Parts Kit**
- **LTspice** (DC Sweep)
- **Pencil**

Helpful links for this experiment can be found on the course website under Class #11.

Pre-Lab

Required Reading: Before beginning the lab, read over and be generally acquainted with this document and the other **required reading** materials.

Video Lecture: Before beginning the lab, watch the Class 11 videos for instructions on using the DC sweep simulation sweeps and a discussion of the linear characteristics of superposition.

Part A – Nodal Analysis

Background

When analyzing circuits with multiple sources, the process developed in the previous laboratories remains unchanged.

1) Identify the nodes
2) Define a ground
3) **For each voltage source, use voltage difference concepts to identify the voltage at a node**
4) For each remaining unknown nodal voltage, apply KCL to obtain a linearly independent equation
5) Transform the system of equation into matrix form.
6) Solve the matrix expression for the unknown voltages

Item number three represents a new observation. For each voltage source, the voltage at a node can be identified. In experiment 9, where only one voltage source was present, we were able to identify the voltage at a node. For the circuit in Figure A-1, with two sources, two of the nodes will have known voltages. In this case, we will leave the voltage values in symbolic form, V1 and V2. We consider these fixed values that can be assigned later.

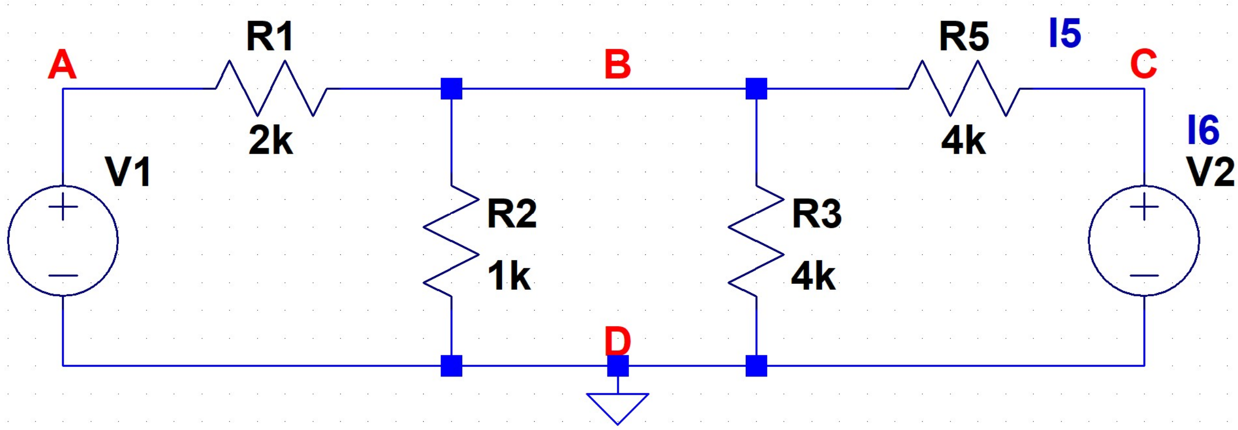

Figure A-1. Two Source Circuit Example

In Figure A-1, the nodes have been identified and node D is set to ground. The nodal voltage at A is determined by voltage differences and symbolically set as $V_A = V1$. Again we consider that a known value, even though it does not have an explicit voltage. Similarly, we can set $V_C = V2$. We can then apply KCL to get the equation,

$$\frac{V_B - V1}{2000} + \frac{V_B - 0}{1000} + \frac{V_B - 0}{4000} + \frac{V_B - V2}{4000} = 0$$

With some rearranging, the equation that results from applying KCL will have the form,

$$V_{node} = aV_1 + bV_2$$

Where *a* and *b* represent the coefficient terms for each of the source voltages. In this equation, the voltage at the unknown node is a linear function of both source voltages. The coefficients *a* and *b* can be considered the slopes associated with the corresponding input voltage, V_1 and V_2 respectively. Since the expression is linear, we can find those coefficients experimentally.

$$V_B = \frac{1}{4}V1 + \frac{1}{8}V2$$

Experiment

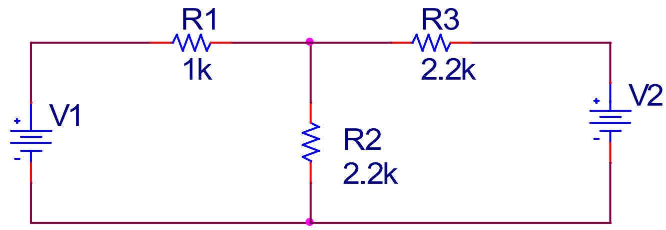

Figure A-2. Two Source Experiment

- Identify the nodes in the circuit
- Place a ground at a convenient location
- Use the symbolic values, V1 and V2, identify two different nodal voltages
- Apply KCL at the remaining unknown node
- Rearrange the expression to get the form

$$V_{node} = aV_1 + bV_2$$

Part B – LTspice and Linear expressions

Background

The KCL equation from the previous page, $V_{node} = aV_1 + bV_2$, indicates that the voltage at the node is dependent on the two source voltages and can be written as a linear combination. We can extend this to any parameter of interest in the circuit. For example, if we wanted to find the current through R1 as a function of voltages V1 and V2, we get a very similar expression, $I_{R1} = cV_1 + dV_2$. The coefficients are different, however, the form of the expression is the same. The linearity of the expression indicates that we can find the contribution from each voltage independently. This characteristic is called superposition, where we can find how each source contributes to the output term (for example V_{node}) by setting all other sources to zero and finding the result due to just one source. This process is repeated for each source. In LTspice, this can be done using the DC sweep simulation setting.

Referencing the circuit from Figure A-1, we can set the voltage of V2 to zero and then perform a DC sweep of the voltage V1.

1) Set the voltage of both V1 and V2 to zero. Even though V1 is set to zero, we will be changing that voltage using the DC sweep feature.

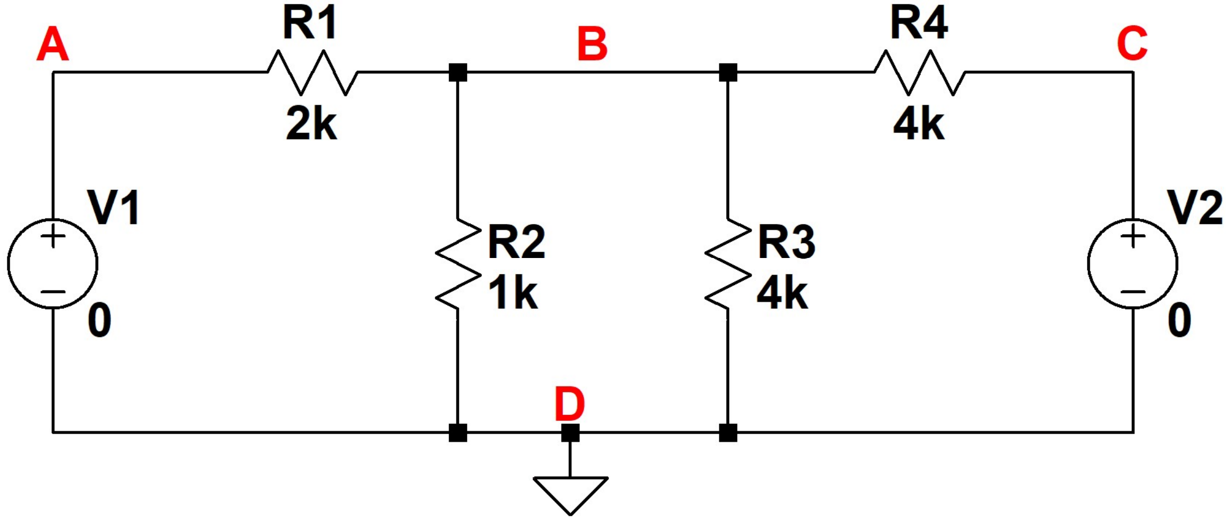

Figure B-1. Two Source Circuit Example with Voltage Sources set to Zero

2) Under the simulation tab, select Edit Simulation CMD
3) Select the DC Sweep tab
    a. Enter the name of the source: V1
    b. Choose the type of sweep: Linear
    c. Choose the start value: 0V
    d. Choose the stop value: 5V
    e. Choose the increment: 0.1V (small enough to get reasonable data)
    f. Enter 'OK' and place the command on the schematic

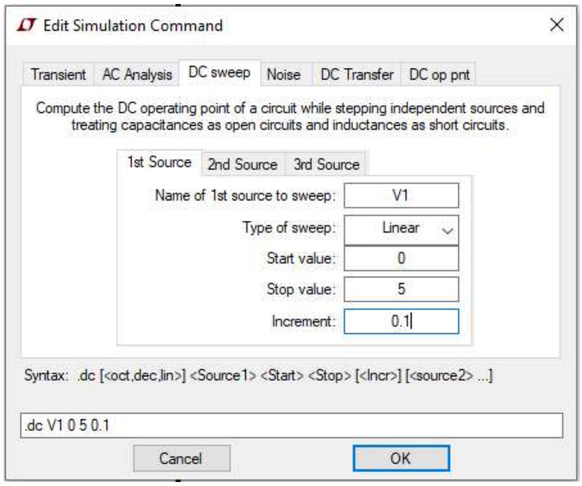

Figure B-2. Example of Simulation Settings

4) Run the simulation
5) Click on node B to see the voltage at node B as a function of voltage V1 (with voltage V2 = 0)
6) The slope of the line is then the coefficient *a* in the linear expression.

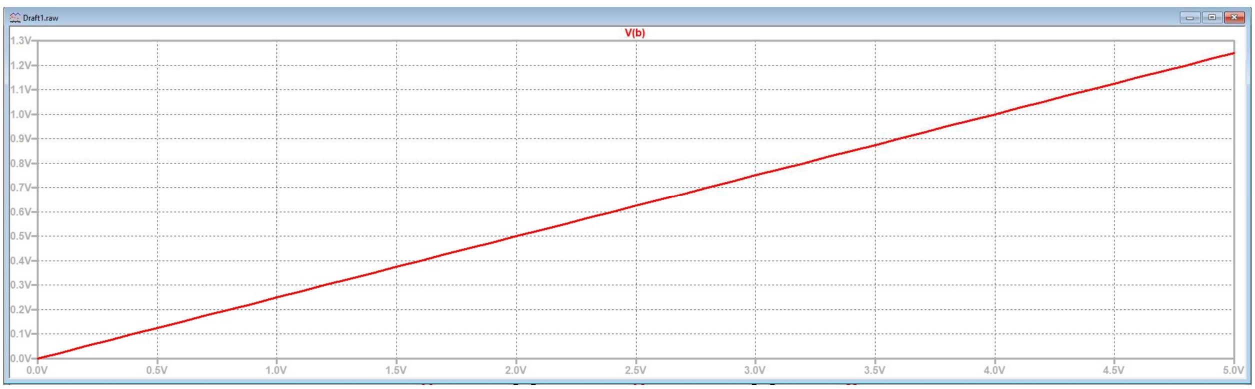

Figure B-3. Voltage at B as a Function of Source Voltage V1 (V2 = 0)

We see that the slope of the line is 0.25, which is the same value calculated in part A. We would repeat this process for source V2, setting V1 = 0 and setting the sweep voltage to V2 to get the *b* coefficient in out expression.

Experiment

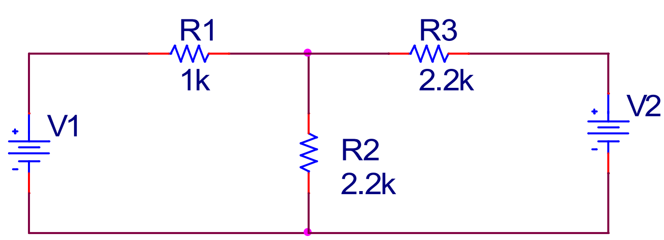

Figure B-4. Two Source Circuit for LTspice

- Implement the above circuit in LTspice and use the DC sweep parameter to find the ‘slope’ of the V1 contribution to the voltage at the unknown node.
- Repeat the process to find the contribution from V2.
- Compare your results to the Part A calculations. Are they the same?

Part C – M1K Board and Alice Meter-Source Tool

Background

We can find the coefficients experimentally as well. The process is very similar to that used in simulation, where we set one of the sources to zero and then step the other source through a series of voltages, measuring the voltage at node B (figure B-1) for each step. Plot the results and find the slope to determine the coefficients. Since there are two sources, we need to use both Channels, A and B in Split I/O mode. The setup is shown in Figure C-1. The left source (V1) is connected to CH A pinout of M1K. The right source (V2), is connected to CH B pinout of M1K. Since we are measuring nodal voltage at the node connecting al four resistors, we will connect this node to Ain (or Bin) and measure the respective CA-V (or CB-V) voltage with respect to ground using the meter source tool.

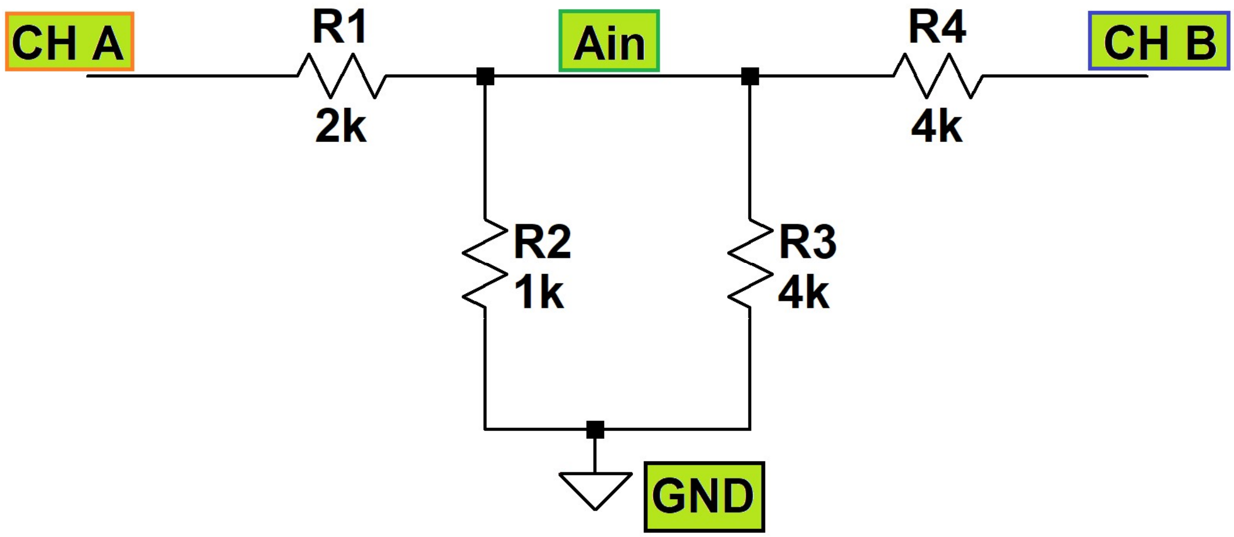

Figure C-1. Experimental Setup for Example Circuit

Experiment

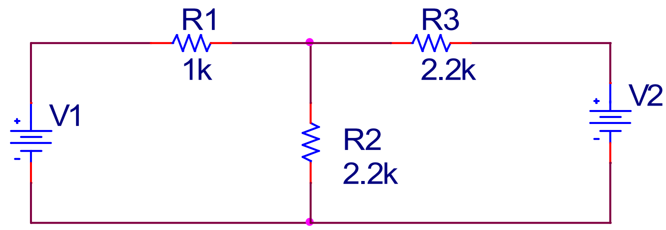

Figure C-2. Two Source Circuit for Experimental Measurements

- Build the above circuit on your protoboard, using CH A for source V1 and CH B for source V2. Don't forget to add the ground connection from the M1K Board
- In the Alice meter-source tool, put both channels in Split I/O mode
- Turn on both Channel A and Channel B
- Set to Channel B to 0V, and step Channel A from 0V to 5V by 0.5V increments. At each step, measure the voltage at the unknown node using Ain read CA-V (or Bin pin read CB-V).
- Plot your results and determine the slope.
- Set to Channel A to 0V, and step Channel B from 0V to 5V by 0.5V increments. At each step, measure the voltage at the unknown node using Ain read CA-V (or Bin pin read CB-V).
- Plot your results and determine the slope.
- How do your results compare to the previously calculated coefficients?

Having determined the coefficients, we can easily determine the expected voltage for any combination of inputs. For each of the following voltage settings, calculate the expected nodal voltage and then experimentally check the results. In your measurements, there may be small differences from the expected values. You should note any small errors and give some thought as to why they might exist. We will visit the concepts of noise and error later, investigating how they may affect our results.

- V1 = 2V, V2 = 2V
- V1 = 1V, V2 = 3V
- V1 = 1.5V, V1 = 2.5V