Lab 3: Equivalent Networks and Superposition

Goals for Lab 3 –
For this lab you will be exploring some of the circuit solving techniques you have been studying in lecture. Specifically, you will build two DC circuits to experimentally test these concepts. The first procedure will verify Thévenin's equivalent circuit. The second procedure will verify the superposition principle. After you verify that the superposition principle works, you will also test the conditions under which superposition can be applied. In this lab, you will also learn more about how SPICE/Multisim can be used to model a circuit.

Theory

Superposition –
The Superposition principle should have been covered in lecture by this time. If you need more detail on it before beginning this lab, please read the appropriate sections of your text. Simply put, superposition holds that if you have a linear circuit with multiple independent sources, you can solve the circuit by solving a circuit with each source in the circuit alone. Then, summing the individual results, with respect to polarity, should give you the same result as if you solved the circuit with all independent sources present. In this experiment, you will be testing this principle.

Linear versus non-linear devices –
So far, you have encountered mainly resistors when building circuits. Resistors are generally modeled as linear devices in this course. However, there are other devices that are non-linear. For example, operational amplifiers (Op-Amps) have both a linear and a non-linear range of operation.

In this lab, while testing the superposition principle for a non-linear circuit, you will have a diode in the circuit. Figure 3.1 below shows more detail on diodes:
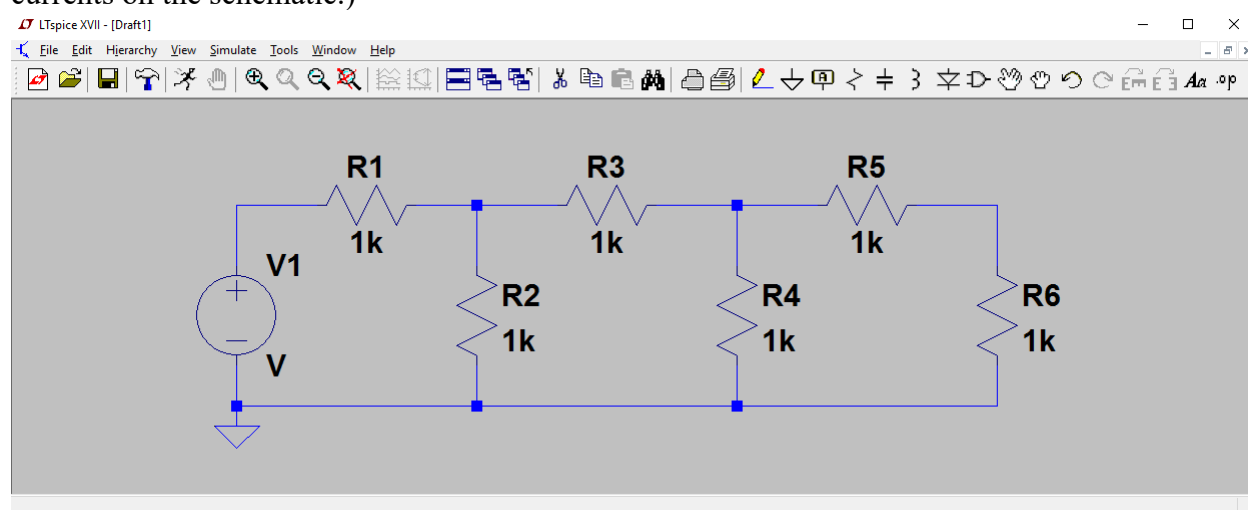

Figure 3.1

You will not study diodes in detail in this course, but you will extensively study and use diodes in ECEN 325 and other courses. There will be occasions in the lab, throughout the semester, where you will use diodes in your circuits. Therefore, it is a good idea to become familiar with them. If you have not studied diodes before, the important information you need to know is that current only flows through a diode from the positive (anode) terminal to the negative (cathode) terminal. Current cannot flow from the negative side of a diode to the positive side of a diode. Therefore, you can also consider a diode like a switch. When current is pushed from the positive to the negative terminals of a diode, the switch is closed. When current tries to flow from the negative to the positive side of a diode, the switch opens and no current is allowed to flow. Also, when current is flowing through a diode, there is a voltage drop of approximately 0.7V across the diode (measured from the anode to the cathode).

Viewing and using the Netlist file from SPICE –
After you create a schematic in SPICE, you have the option of viewing the Netlist. If there is a problem with your circuit you do not understand, checking the SPICE Netlist (or the circuit file) can help you identify problems. If you are using LTspice, in order to view your Netlist, go to the View menu and click on, "SPICE Netlist." A new file will come up, named, XXX.net, in an editor for you to view. Each time you change your schematic, a new Netlist is created. See Appendix I for more details on working with LTspice.

For example, the schematic in Figure 3.2 was drawn in SPICE, and the simulation was run for bias point detail. (You should recall that the bias point detail allows you to display the voltages and currents on the schematic.)

Figure 3.2 – LTspice schematic for a sample circuit

The netlist that is created from this schematic is:
```
V1 N001 0 V
R1 N002 N001 1k
R2 N002 0 1k
R3 N003 N002 1k
R4 N003 0 1k
R5 N004 N003 1k
R6 N004 0 1k
```
Note how each part is given by its type and name (RX), the two nodes it is connected to (NXXX and NXXX), and the value of the component. For example, for the resistor labeled, R3, it is connected to node 3 (N0003) and node 2 (N0002) and it is a 1kΩ (1k) resistor. The ground node is indicated by 0. SPICE will not necessarily label the nodes the same way you would. However, from the list above, you should be able to draw the circuit.

Working with the schematics in SPICE can be very helpful. However, working with the Netlist and Circuit files that SPICE uses to generate output can help you understand more about how circuit simulation works.

The next step in your circuit analysis skills will be to take a physical circuit and write a Netlist for the circuit. See your TA if you need help with this part of the prelab.

Prelab –

Read Lab 3 before beginning the calculations for the prelab. Bold items must be turned in as part of your written prelab.

1. Refer to the circuit in Figure 3.3. **Solve for the voltage across the 1kΩ resistor using superposition.** Make sure to solve for the following 3 cases: Only V1 present, only V2 present, and both V1 and V2 present. **Fill out the column corresponding to Part D in Table 3.1. Show all steps to receive full credit.**
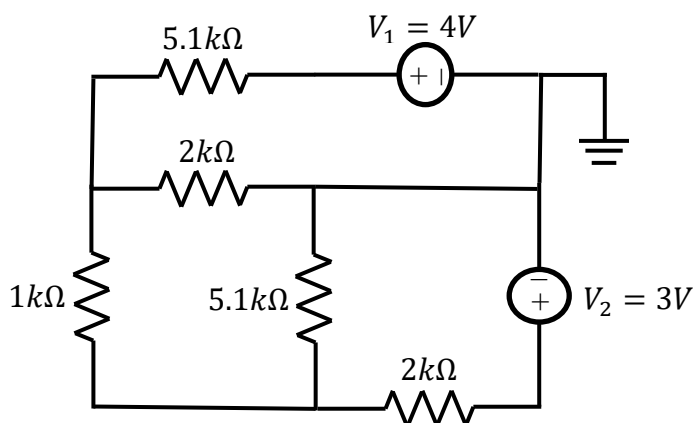
Figure 3.3

2. Simulate the circuit in Figure 3.3 in SPICE. **Print your schematic with all voltages and currents displayed to hand in to your TA.** You can test the superposition principle by running 2 more simulations: one with V_1 replaced with a short and one with V_2 replaced with a short. You do not need to print all of the schematics to turn in. However, you should run the simulations and fill out the entries in Table 3.1. Keep a copy of your results to use when you prepare your lab report.

3. Simulate the circuit in Figure 3.4. You will want to use part 1N4148 for the diode in SPICE. **Print the schematic with voltages and currents displayed to hand in to your TA.** Again, test the superposition principle by running 2 more simulations: first with V_1 replaced with a short and second with V_2 replaced with a short. You do not need to print all of the schematics to turn in. Finish filling out Table 3.1.
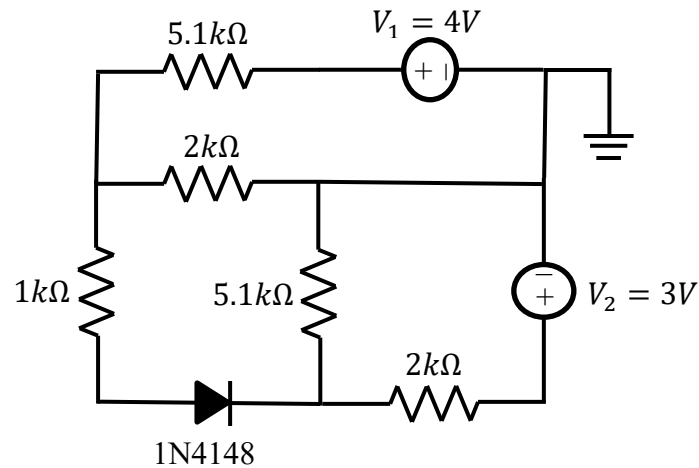
Figure 3.4

Table 3.1: Include this table, will all entries completed, to hand in as part of your prelab.

V_L (voltage over the 1kΩ resistor)	Calculation Part D (No diode)	PSPICE Part E (No diode)	PSPICE Part F (with diode)
1. V_1 & V_2 Present			
2. V_1 only			
3. V_2 only			
4. Add line 2 & 3			
5. % difference between line 1 and line 4			

For which columns does the superposition principal hold?

SAVE YOUR SPICE SIMULATIONS AND CALCULATIONS! You will need some of the values in your lab report!

Procedure –

Parts used:
> Two 1.5V batteries
> 3 -- 1.0 kΩ resistors, ¼ W
> 2 --2.0 kΩ resistors, ¼ W
> 1 -- 3.3 kΩ resistor, ¼ W
> 2 --5.1 kΩ resistors, ¼ W
> 1 -- 1N4148 diode
> A selection of colored 24 gauge connection wires, at least 7 strands.

It is a good idea to always verify your component values before you begin doing an experiment. This will allow you to identify one possible source of error easily. If you are using the AA batteries as a voltage source, check the actual voltage produced by your batteries.

Task #1 – Verify the superposition principle
Construct the circuit of Figure 3.3. You can use your batteries for the 3V source and the variable voltage source as the 4V source. Make the following measurements with your voltmeter:
1. Measure the voltage across the 1kΩ resistor and record in your lab notebook.
2. Remove the voltage source V_2 (the 3V source) from the circuit and replace it with a short circuit. Measure the voltage across the 1kΩ resistor. Call it $V_{L,1}$ and record it in your lab notebook.
3. Put V_2 back in the circuit and remove the voltage source V_1 (the 4V source) and replace it with a short circuit. Measure the voltage across the 1kΩ resistor. Call it $V_{L,2}$ and record it in your lab notebook.

Does the principle of superposition apply to this circuit? Complete the calculations indicated in Table 3.2 and discuss any significant differences between measured and theoretical values that you see.

Task #2 – Check the superposition principle validity for a non-linear device
For Task #2, you will repeat the procedure as outlined in Task #1. However, use the circuit shown in Figure 3.4 where we have added a diode to the circuit. For more information on using the 1N4148 diode, see the theory section of this lab. Be sure to place the diode in the circuit with the polarity indicated in Figure 3.4 so that the current will flow in the correct direction.
1. Measure the voltage across the 1kΩ resistor with both voltage sources V_1 and V_2 present. Record in your lab notebook (Table 3.3).
2. As you did for Task #1, measure the voltage across the 1kΩ resistor with only V_1 present. Record your lab notebook (Table 3.3).
3. Measure the voltage across the 1kΩ resistor with only V_2 present. Record in your lab notebook (Table 3.3).

Report Requirements –
1. Be sure to have a title page with the usual items.
2. You should write Task #1 & Task #2 Together–
    A. Procedure – same as above.

B. Data Tables w/ results – reproduce Table 3.2 & Table 3.3
C. Sample calculations – Show the calculation method used to fill out the columns in Table 3.2. If you fill out the calculations in column in Table 3.3 correctly, you will receive 5 extra points. You do this by using the diode in the following way: IF there CAN be a 0.7V drop over the diode, then there IS ONLY a 0.7V drop over the diode. If there CANNOT be a 0.7V drop over the diode, then the diode functions like an open switch. Feel free to see one of the TAs during office hours if you need more details.
D. Discussion –Discuss how superposition works in Task #1 and why superposition doesn't work in Task #2. Think about how the diode works. We have said that the diode is a non-linear element. Discuss why resistors are linear elements and diodes are not. (What makes something linear?)

Table 3.2: Task 1

Parameter	Measured	Calculated	% difference	SPICE	% difference (SPICE to measured)
V_L					
$V_{L,1}$					
$V_{L,2}$					
$V_{L,1} + V_{L,2}$					

Table 3.3: Task 2:

Parameter	Measured	Calculated	SPICE	% Error
V_L				
$V_{L,1}$				
$V_{L,2}$				
$V_{L,1} + V_{L,2}$				