

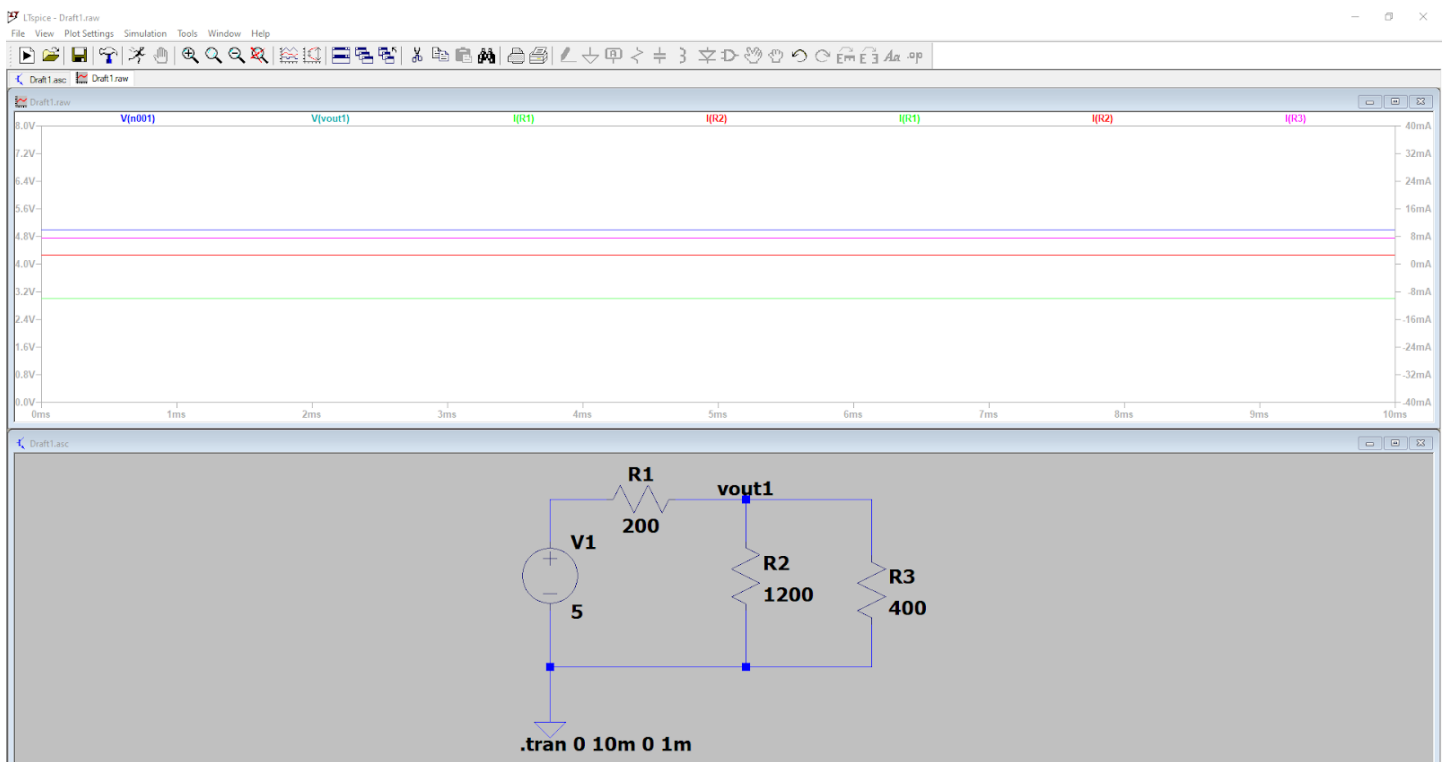
Class:	EE221 Circuits II 1001			Semester:	Fall 2023
Points		Document author:	Narek Kalikian		
		Author's email:	kalikn1@unlv.nevada.edu		
		Document topic:	Postlab 1		
Instructor's comments:					

1. Introduction / Theory of Operation

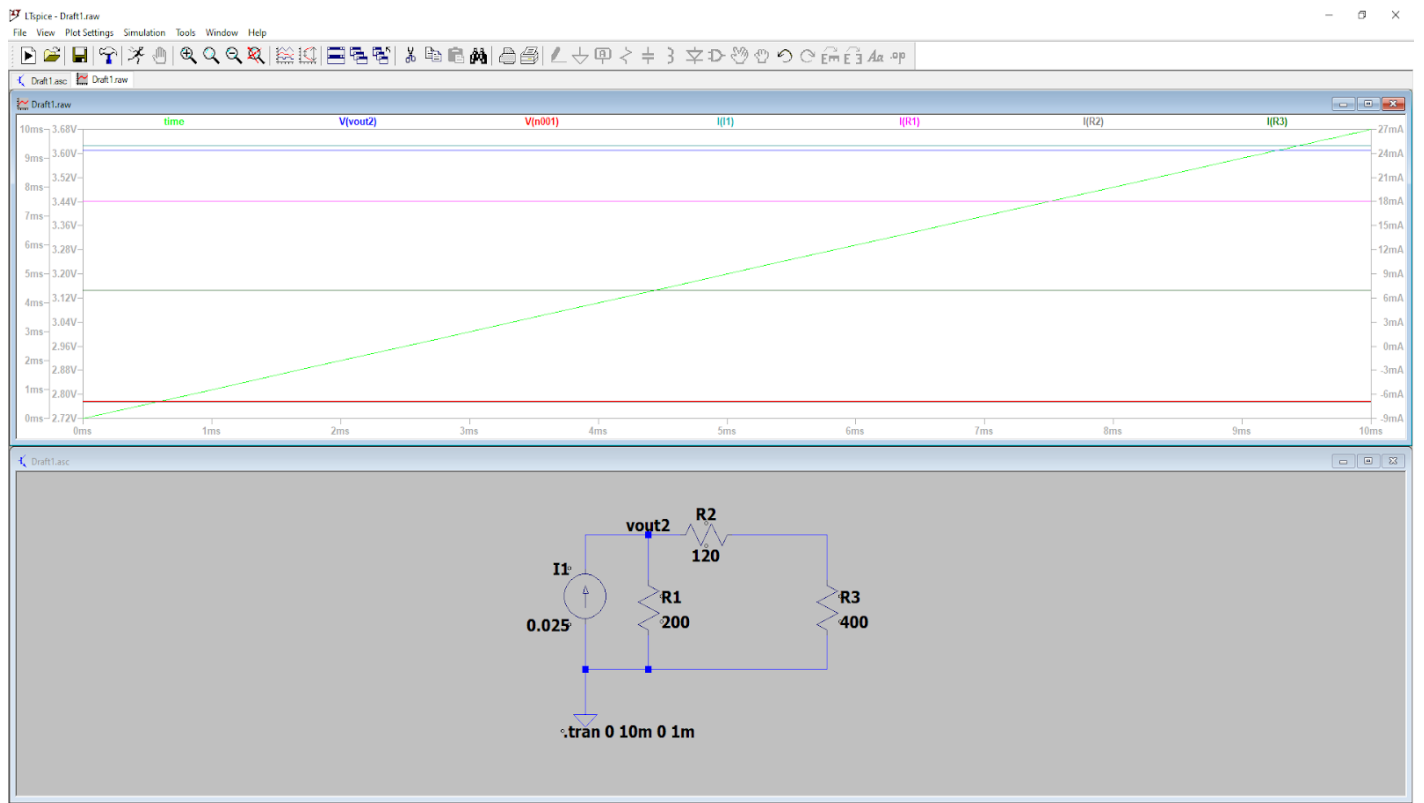
The objective of this lab was to become familiar with the SPICE software (LTspice) that we'll be using to simulate circuits digitally before implementing them physically. The elements that proceed consist of LTspice screenshots that show simulation results and tables to reflect those results compared to hand calculations done in the prelab.

2. Description of Experiments

Experiment 1A - Thevenin Equivalent:

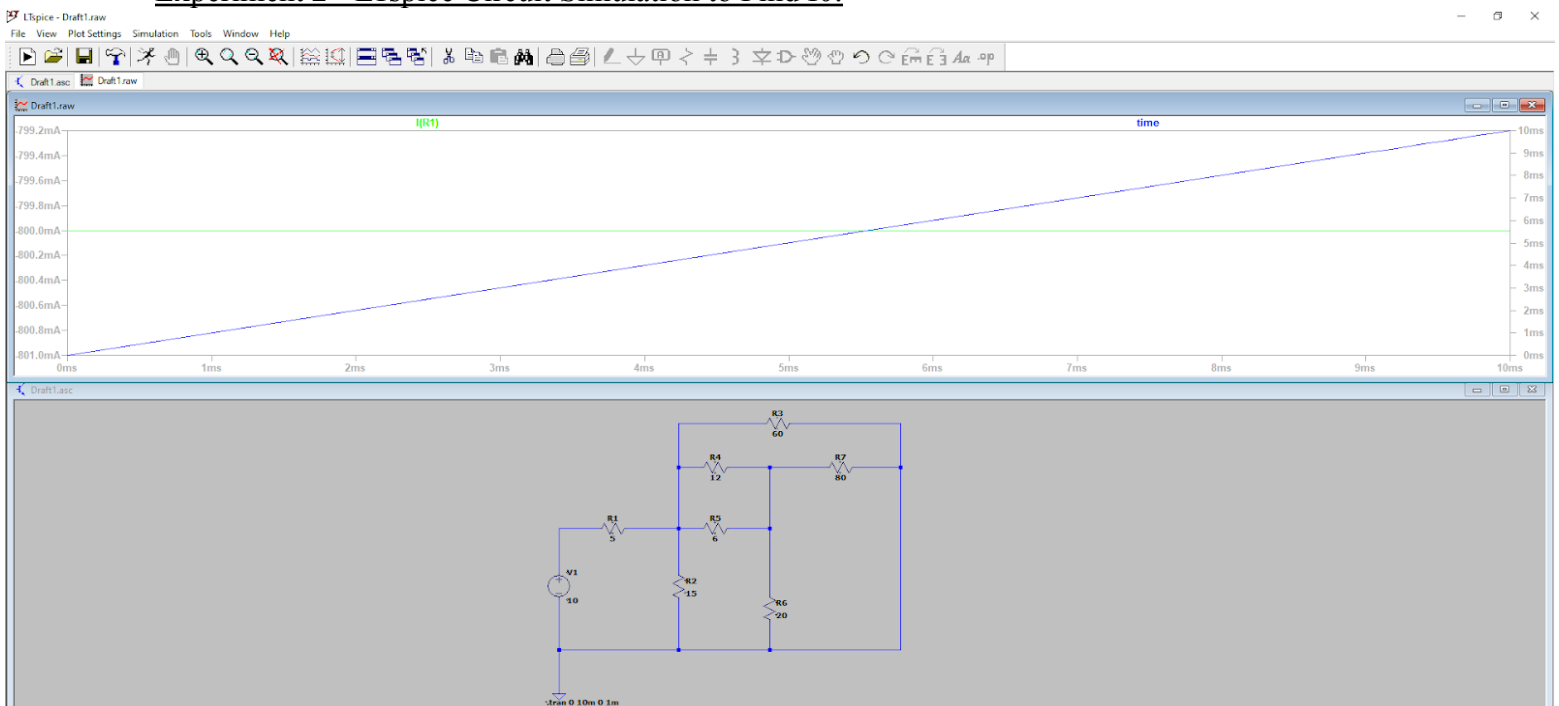


Experiment 1B - Norton's Equivalent:

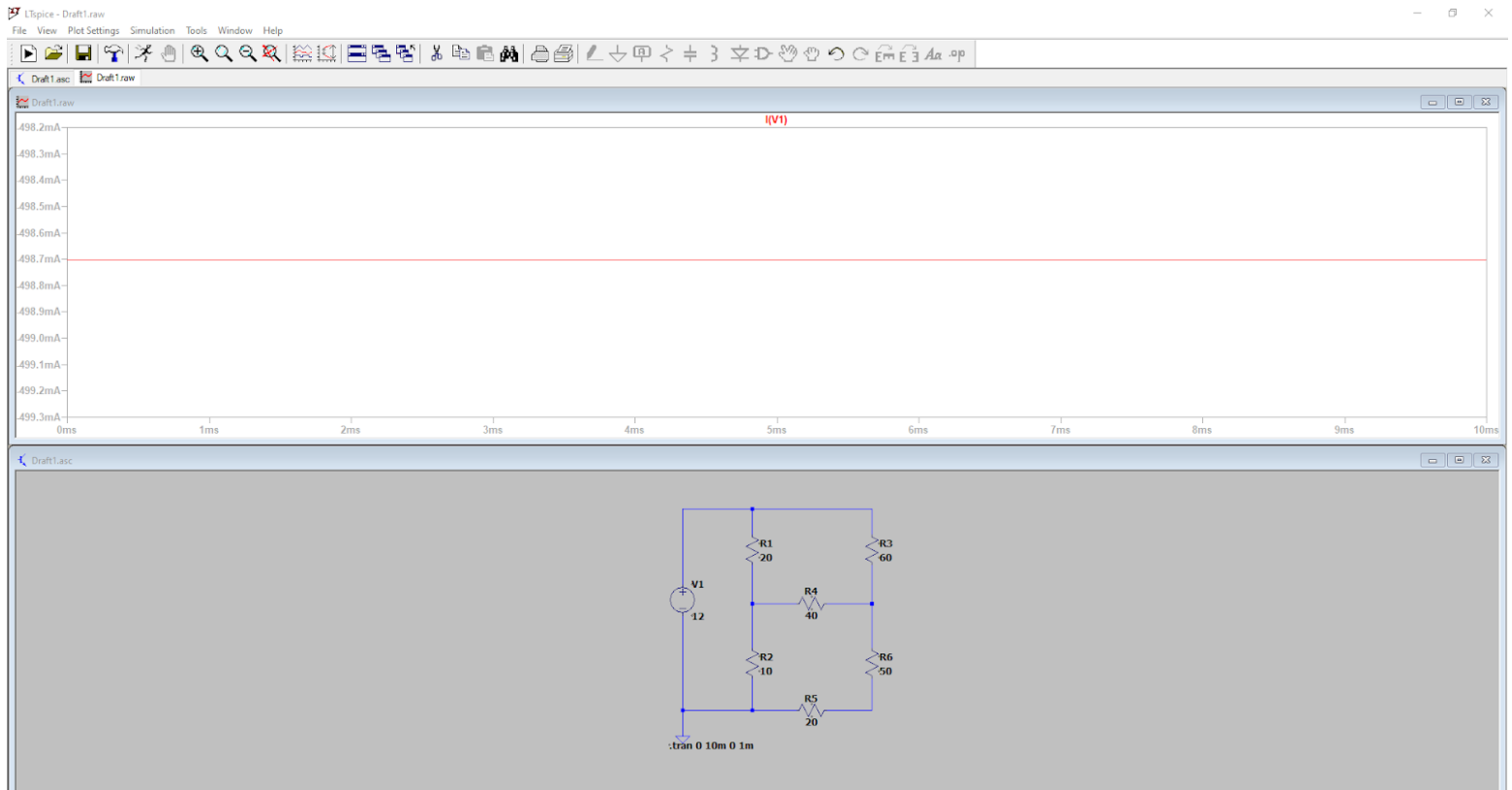


		Vout (V)	ir1 (mA)	ir2 (mA)	ir3 (mA)
Circuit 1 (1)	Hand Calc	3	10	2.5	7.5
	LTspice	3	10	2.5	7.5
Circuit 1 (2)	Hand Calc	3	15	10	10
	LTspice	3	15	10	10

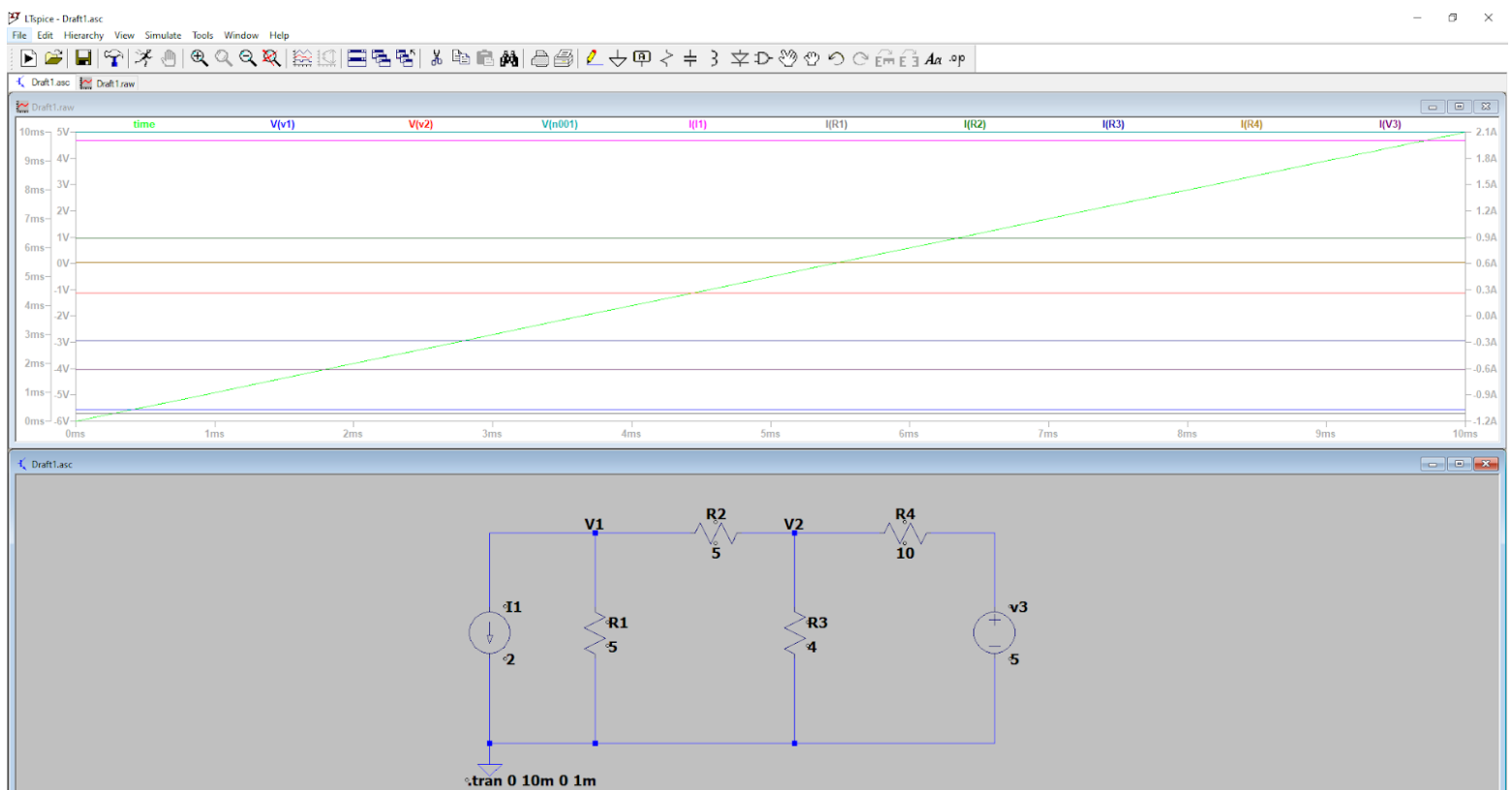
Experiment 2 - LTspice Circuit Simulation to Find I_0 :



Experiment 3 - LTspice Circuit Simulation to Find I0:



Experiment 4 - LTspice Circuit with 2 Sources Simulation (Superposition to find Voltage and Current):



	Circuit 4	V1 (V)	V2 (V)	ir1 (mA)	ir2 (mA)	ir3 (mA)	ir4 (mA)
1.)	Calc	0.55	1.12	110	110	280	390
	LTspice	0.55	1.12	110	-110	-280	-390
2.)	Calc	-6.11	-2.22	-122	780	-550	220
	LTspice	-6.11	-2.22	-122	-780	550	-220
3.)	Calc	-5.55	-1.12	-112	890	-280	610
	LTspice	-5.55	-1.12	-112	-890	280	-610

3. Encountered Problems

I didn't encounter any problems during this lab as it was all very straightforward and it laid out the instructions well for using the SPICE software correctly.

4. Questions and Answers

- 1.) **Question 1:** Simulating in SPICE is very much recommended because it allows for engineers to figure out if the proposed circuit will work as intended before fully committing to the implementation of a real circuit. This is beneficial for a number of reasons. It will eliminate waste, increase product yield, and save us money in the long run. It also allows engineers to make smarter and faster design decisions moving forward and can also provide safety benefits.
- 2.) **Experiment 1 - Question 1:** Yes, they meet KCL and KVL because the expected results acquired from completing the hand calculations match exactly with the results of the experiment (LTspice simulations).
- 3.) **Experiment 1 - Question 2 (Thevenin/Norton):** The voltage remains the same but we will get different results for the current readings using Thevenin's Equivalent as opposed to Norton's.
- 4.) **Experiment 2 - Question 1 (Delta or Y Circuits):** The experiment does not include any delta or Y circuits. This is due to the fact that resistors R6 and R7 are connected to the same two nodes.

- 5.) **Experiment 2 - Question 2 (Req and $10/i_0-5$):** All hand calculations done and submitted with prelab. I found R_{eq} to be $7.5\ \Omega$ and $I_0(i_0)$ to be 800 milliamps. So, $(10/i_0) - 5 \rightarrow 12.5 - 5 = 7.5$ amps (750 mA).
- 6.) **Experiment 3 - Question 1 (Req Calculation):** Calculated and submitted in prelab. I found R_{eq} for this circuit to be $24\ \Omega$.
- 7.) **Experiment 3 - Question 2 ($12/I_0$):** Both in my hand calculations and in LTspice I found I_0 to be 0.5 A (500 mA). $12 / 0.5 = 24$, which matches the result I found in the previous question for R_{eq} .
- 8.) **Experiment 4 - Question 1 (Superposition Law):** If we reference the table in experiment 4, we see that the current at r_2 added with the current at r_3 will result in the current at r_4 . From this we can conclude that by evaluating the circuit with only one source on and subsequently finding the voltage and current, we can calculate the actual voltages and currents when both sources are turned on.

5. Conclusions

I learned how to use SPICE software, specifically LTspice, which will be useful for a long time to come. I'm sure I will use it a lot this semester and in the future to simulate virtual circuits before moving on and creating the real, physical circuits. I found out that this is important for many of the aforementioned reasons and it will always be a good rule of thumb to simulate my circuits in SPICE first. Specifically within the software, I learned how to implement different types of components, how to move and rotate them as needed, and how to ground and run them.