

This assignment carries 10 marks in total.

***** ALL PRE-LAB ANSWERS MUST BE SUBMITTED THROUGH CANVAS BEFORE SUNDAY, SEPTEMBER 24, 11:59 PM.**

ZERO MARK WILL BE GIVEN TO LATE SUBMISSIONS OR NO SUBMISSIONS ***

Pre-Lab Assignment 1: Resistor Networks, Voltage and Current

LT Spice Simulation

INTRODUCTION:

LTspice is a free, high-performance SPICE simulator software developed by Linear Technology. It is widely used by engineers, researchers, and students for simulating analog and mixed-signal circuits, including power electronics, RF circuits, and microcontroller-based systems.

LTspice uses the SPICE (Simulation Program with Integrated Circuit Emphasis) simulation engine, which is a powerful tool for predicting the behavior of electronic circuits. It allows users to model the behavior of electronic components, such as resistors, capacitors, inductors, transistors, and operational amplifiers, and simulate the circuit's response to a variety of signals, including DC, AC, and transient signals.

LTspice offers a user-friendly graphical interface for creating and editing circuit schematics, as well as a comprehensive set of simulation and analysis tools. These tools include waveform viewers, frequency domain analysis, and Monte Carlo analysis, among others. One of the key advantages of LTspice is its ability to simulate complex circuits with high accuracy and speed, allowing users to quickly explore various design options and optimize their circuits for performance, cost, and reliability.

REQUIRED MATERIALS:

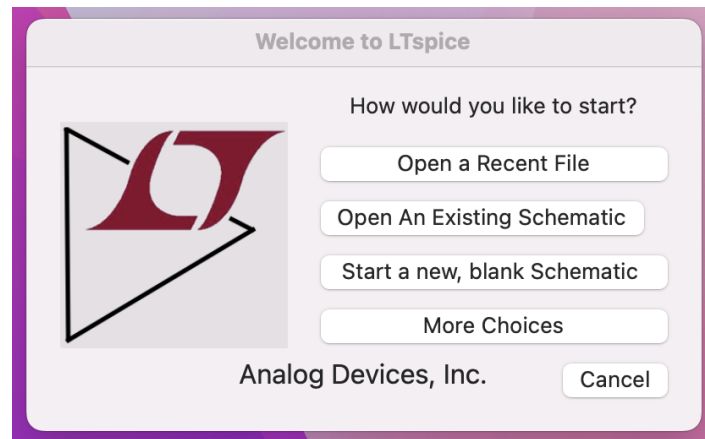
Hardware:

- Computer (Either Window or MacOS)

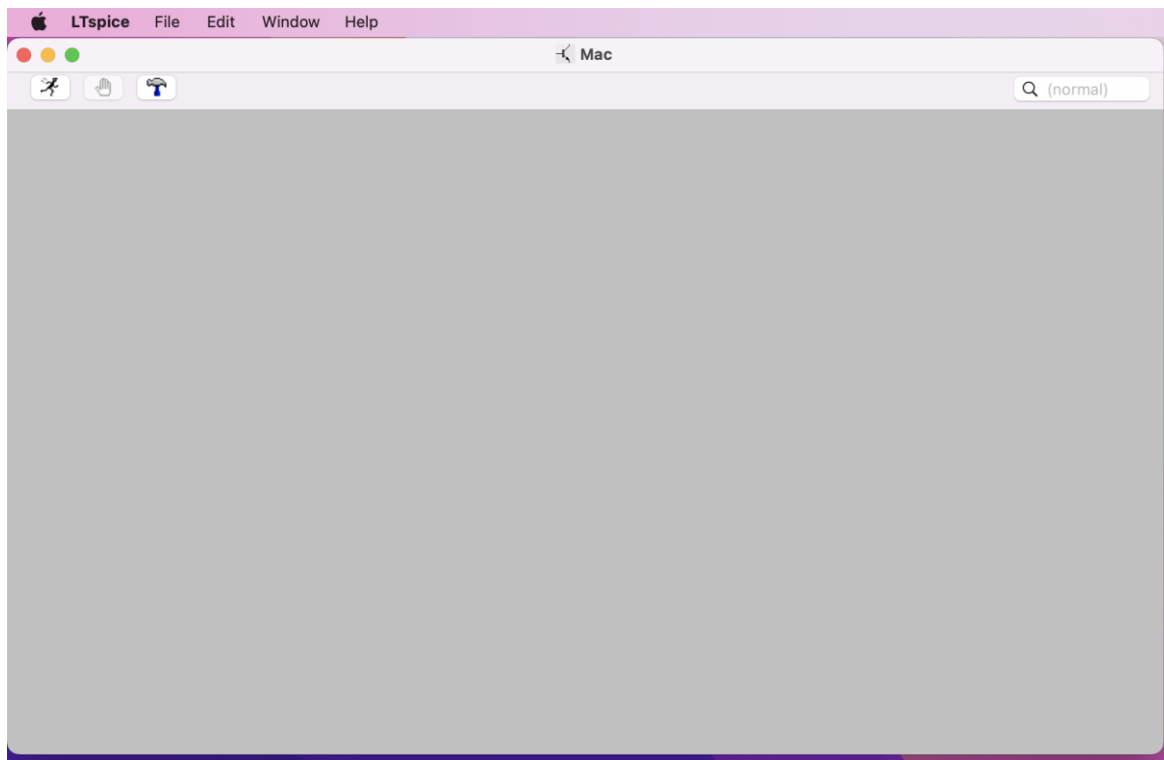
Software:

- LT spice (<https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html>)

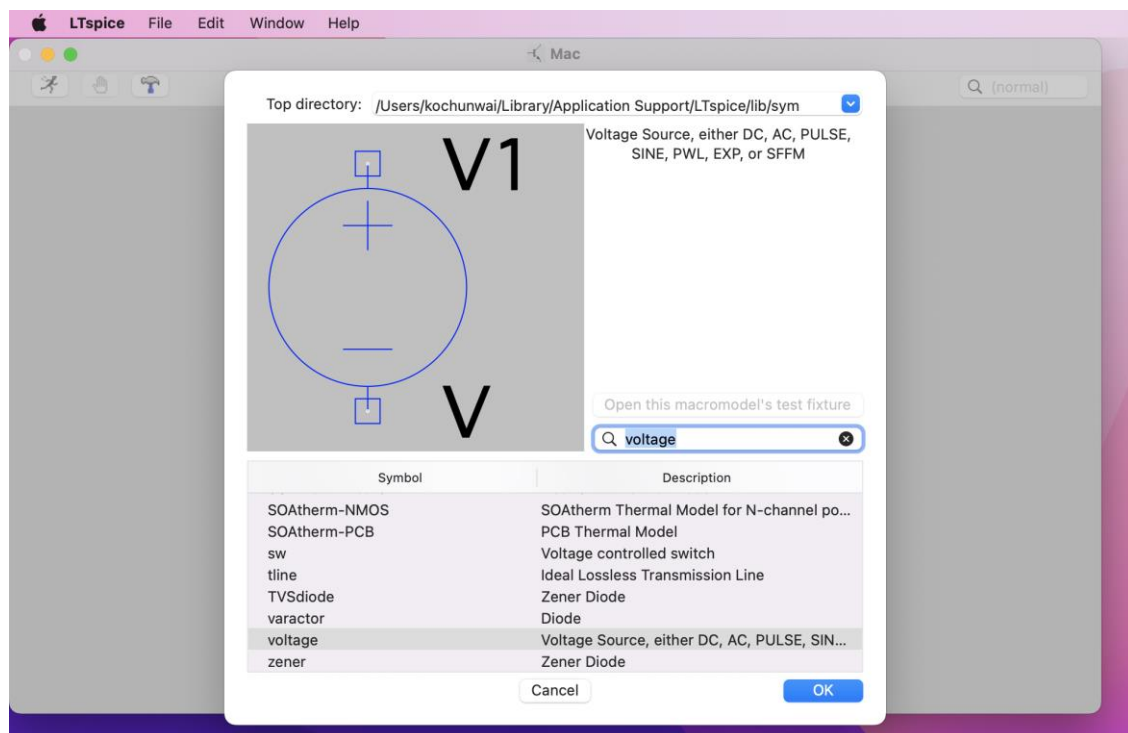
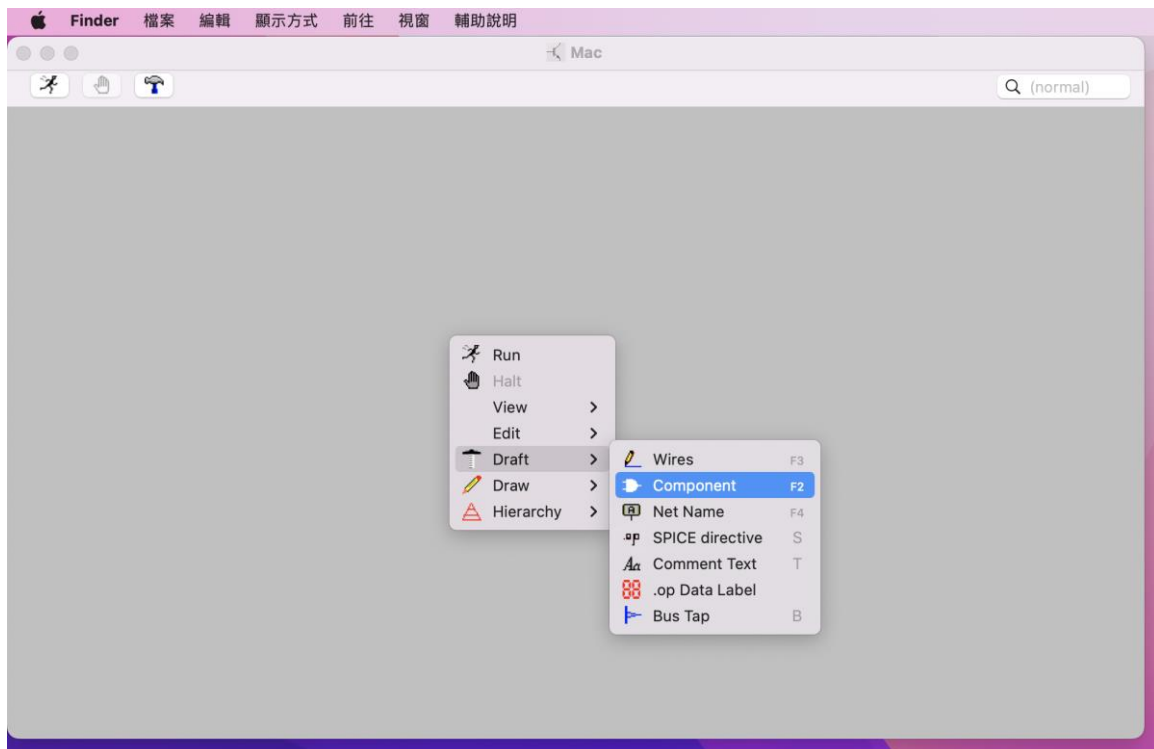
Step 1: Install LT spice and open the software.



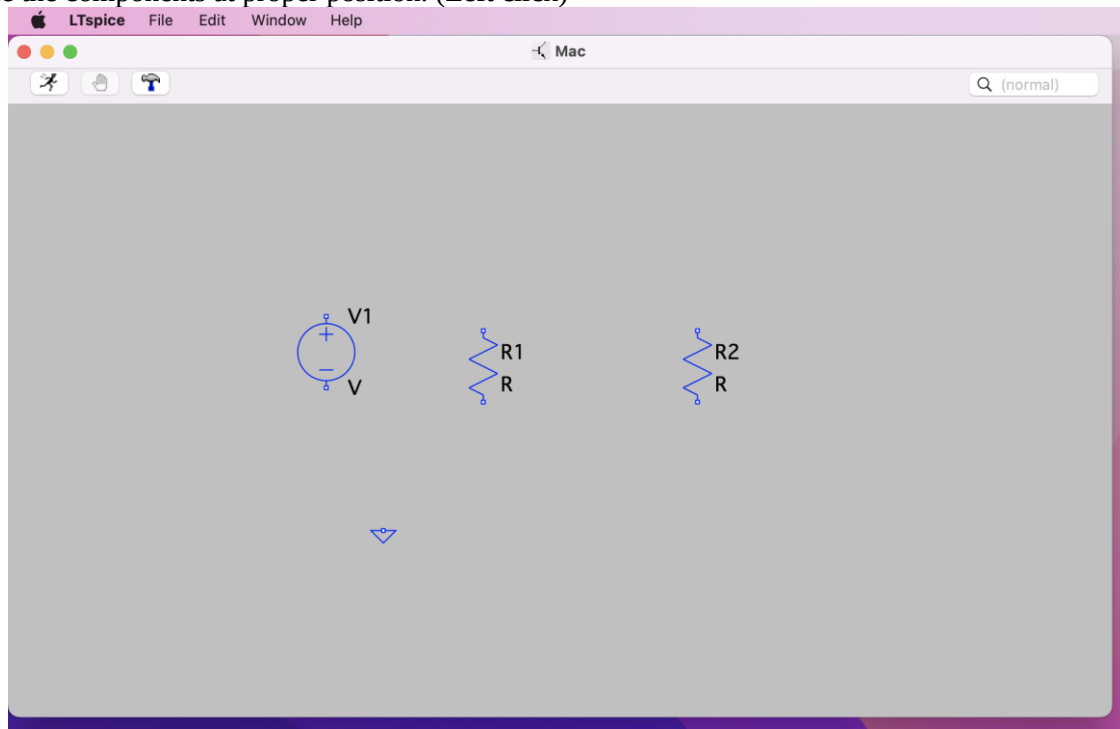
Step 2: Create a new schematic by clicking “start a new, blank Schematic”.



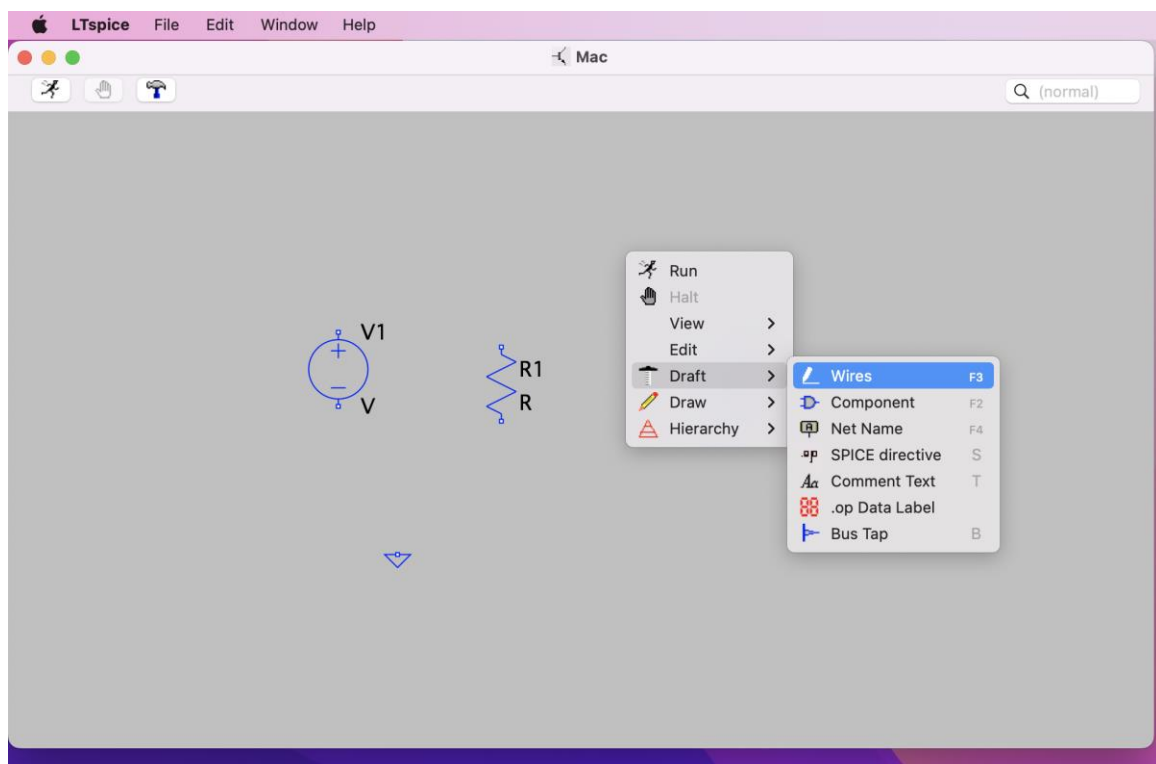
Step 3 Right click on the empty space and click “Component” which is in the directory of “Draft”. Add/Modify/Delete electrical components into the schematic.



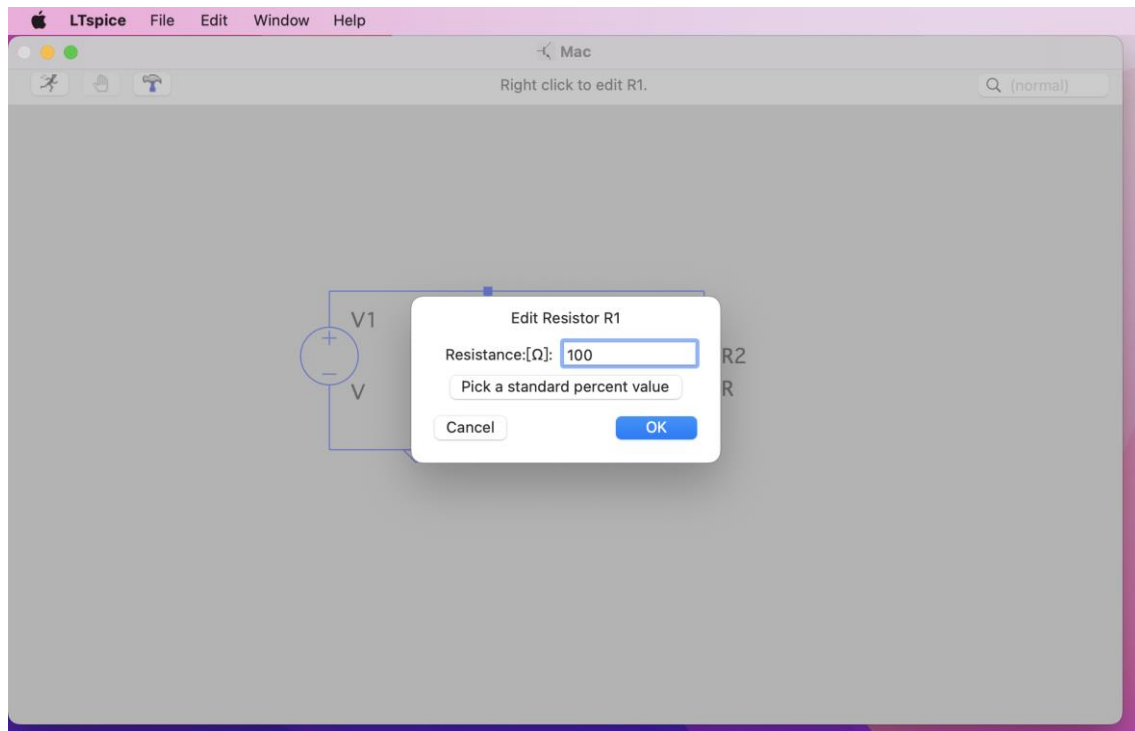
Step 4 Place the components at proper position. (Left click)



Step 5 Connect all components by wires.

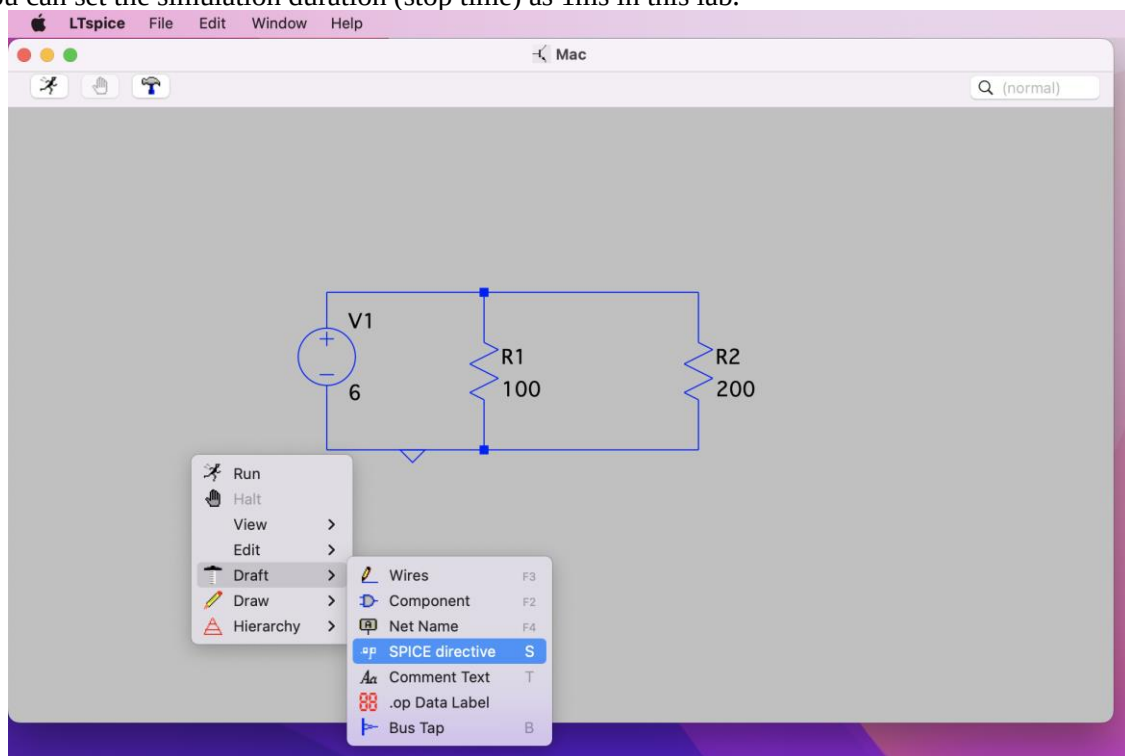


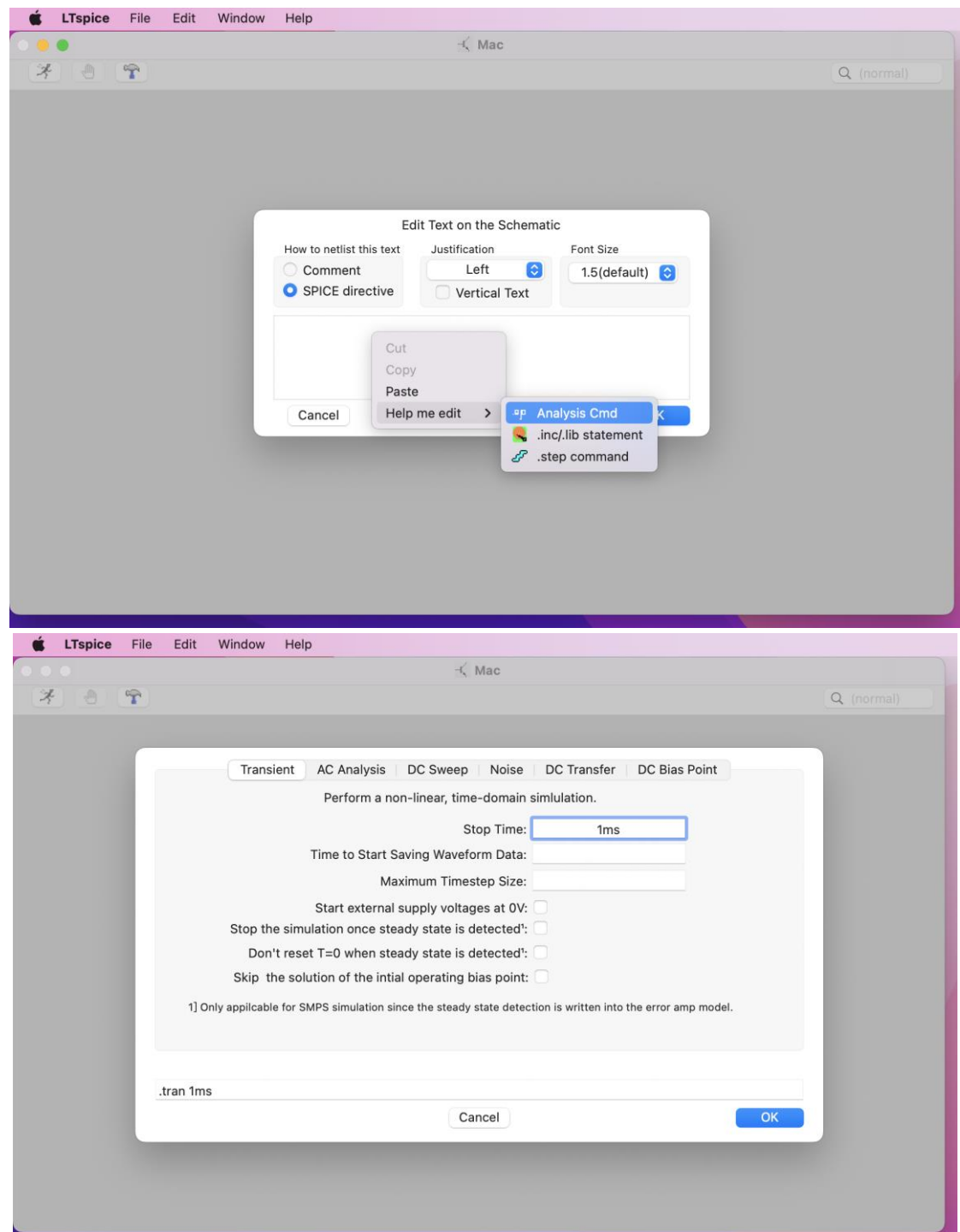
Step 6 Right click on the components to change its values.



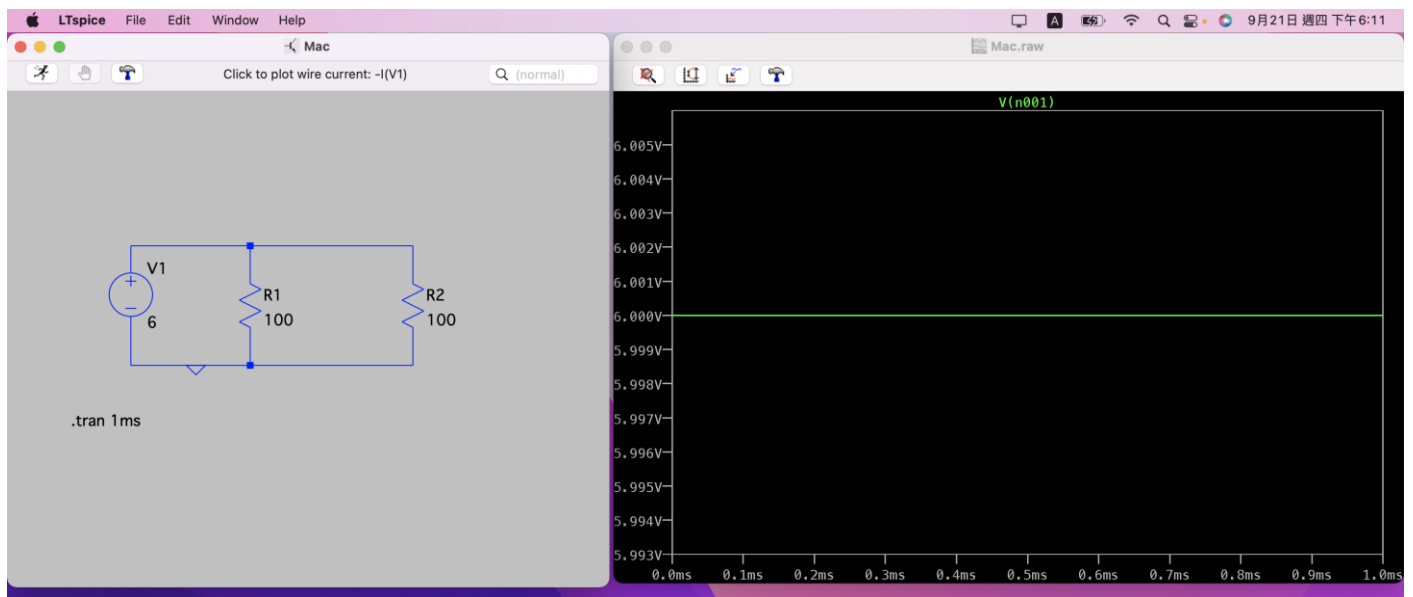
Step 7 Run the simulation.

- You can set the simulation duration (stop time) as 1ms in this lab.





Step 8 Place your cursor on the wires or the components. Click on the left to see the simulated value on the trace panel.



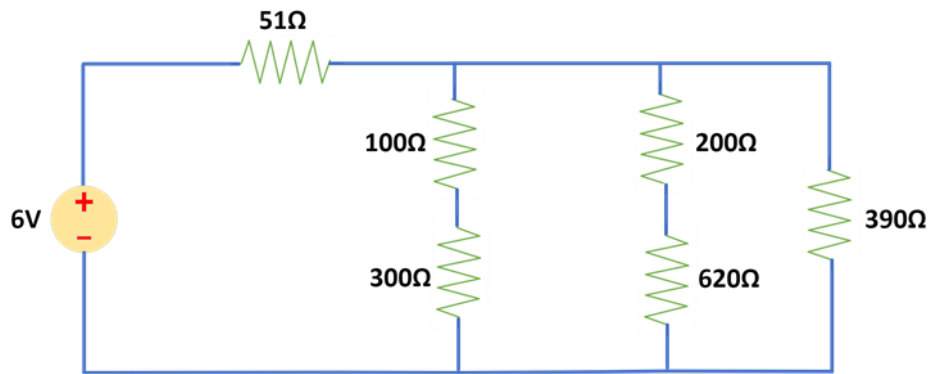


Fig 1

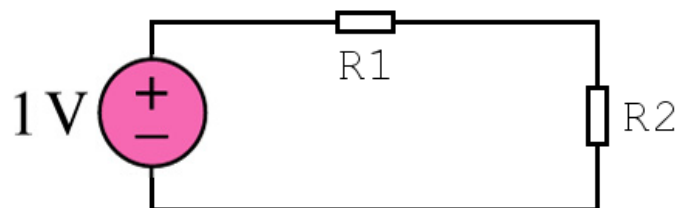
Please note that **only the numerical values** to the answers for the following questions are required for submission on Canvas. There is no need to include the unit in your answer submission.

With reference to the circuit shown in Fig 1,

1. Determine the resistance seen by the 6V source (R_{eq}) in Ω .
2. Determine the voltage across the 51Ω resistor in V.
3. Determine the voltage across the 390Ω resistor in V.
4. Determine the current through the 300Ω resistor in mA.
5. Determine the current through the 620Ω resistor in mA.

LAB TASK: PART 2

Step 1 Draw the circuit below using LT spice given that $R1 = 6.8k\Omega$ and $R2 = 8.2k\Omega$.



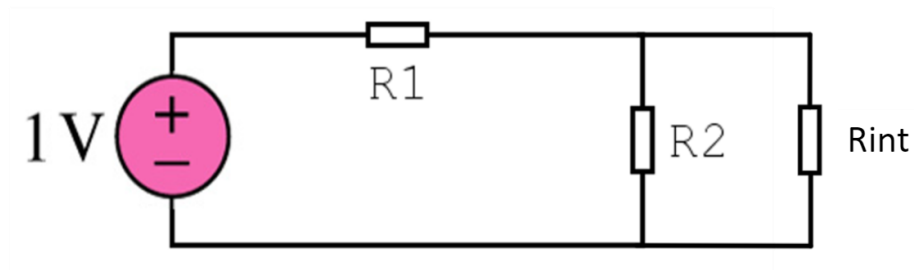
Step 2 Find the voltage across $R2$.

Voltage across $R2$, $V_{R2} = \underline{\hspace{2cm}}$ V

Step 3 Find the voltage across $R2$ if **both** resistors ($R1$ and $R2$) are 1000 times larger?

Voltage across $R2$, $V_{R2} = \underline{\hspace{2cm}}$ V

Step 4 Draw the circuit below using LT spice given that $R1 = 6.8k\Omega$ and $R2 = 8.2k\Omega$. R_{int} refers to the internal resistance of the voltmeter ($R_{int}=10M\Omega$).



Step 5 Find the voltage across R2.

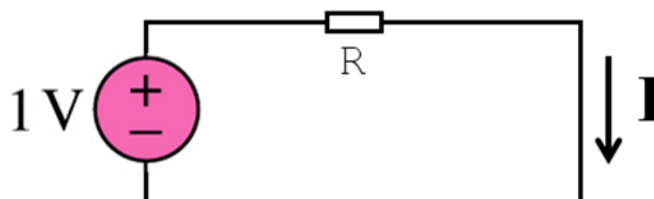
Voltage across R2, $V_{R2} = \underline{\hspace{2cm}}$ V

Step 6 Find the voltage across R2 if **both** resistors ($R1$ and $R2$) are 1000 times larger?

Voltage across R2, $V_{R2} = \underline{\hspace{2cm}}$ V

LAB TASK: PART 3

Step 1 Draw the circuit below using LT spice given that $R1 = 24\Omega$.



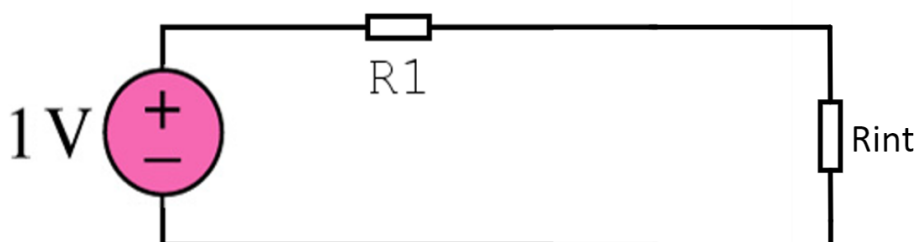
Step 2 Find the loop current

loop current, $I = \underline{\hspace{2cm}}$ A

Step 3 Replace the resistor in the circuit so that now $R = 24k\Omega$, and then find the loop current again.

loop current, $I = \underline{\hspace{2cm}}$ A

Step 4 Draw the circuit below using LT spice given that $R1 = 24\Omega$. R_{int} refers to the internal resistance to the ammeter ($R_{int} = 0.5\Omega$).



Step 2 Find the loop current

loop current, $I = \underline{\hspace{2cm}}$ A

Step 3 Replace the resistor in the circuit so that now $R = 24k\Omega$, and then find the loop current again.

loop current, $I = \underline{\hspace{2cm}}$ A