BE 330 Lab 2: Simulating Circuits with LTspice or MultiSim (Live)

Logistics

Lab on 1/27. Lab report is due on 2/3. No pre-lab is due today.

Read the entire lab instructions before starting the lab assignment. You should do this for every lab, every week. It is important to read the entire lab first because often, you will be connecting multiple components together, and you need to understand each component before connecting them together. If you just try to serially read and complete the instructions, you will get stuck because vital information for completing specific steps may appear later in the instructions.

There will be many breakout rooms that you may self-assign yourselves to, to identify the people you would like to work with and help each other with the debugging. You are highly likely encouraged to ask your peers before looking for a TA, peer instructor, or the professor.

If you need to refuel, please stay away from the electronics, wash your hands, and consume your food or drink. You do not want to consume heavy metals or fry your $300 instrument.

Your webcam should be on throughout the whole lab, but feel free to turn off your webcam for a short while if you need to use the restroom, tend to an unexpected guest, etc. Not turning on the webcam will not allow you to be checkout (which means that you may lose major points for your lab).

Introduction

Circuit simulation is one of the oldest applications of computer simulation. SPICE ("Simulation Program with Integrated Circuit Emphasis"), the original circuit simulation program, was first released in 1973. Forty-eight years later, SPICE is still the *de facto* standard for circuit simulation. Whether you are using LTSpice, MultiSim Live, or the full version of MultiSim, the core of the software you are using is still SPICE.

SPICE is initially inspired the need to obtain realistic input/output characteristic circuit without building a breadboard circuit. This situation is extremely common in building integrated circuit (IC) chips, for which breadboarding is highly impractical due to the high amount of circuitry components involved. Building IC chips often require high-quality photolithographic masks and product failure can be catastrophically costly.

The success of SPICE-based simulation has extended its application into simulating board-level designs due to their relative simplicity and low cost. We will use SPICE-based simulations to guide us through the labs this semester. The simulations can be effectively used to give you an idea about how your results should look like without resorting to long mathematical derivations, especially when performed before the labs.

Objectives

1. Learn about the interface of LTspice or MultiSim (Live).
2. Learn about using LTspice or MultiSim (Live) to simulate simple AC and DC circuits.
3. Learn to extract useful information from the simulation.
4. Learn to compare simulation results with theoretical results.

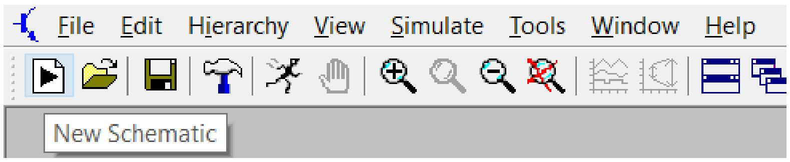
Task 1: Resistors in Series

Open your own simulation software. You will be building a very simple circuit that measures the voltage across one of the two resistors in series with a DC power supply. I do not have the MultiSim full version now, so I will show you the step-by-step guides for MultiSim Live and LTspice.

Step 1 – Making a new circuit

If you are using the MultiSim Live, go to [Multisim Live Online Circuit Simulator]() and the top right-hand side should be a "create circuit" button. Click on that button and set up a new circuit.

If you are on LTSpice, click on the "New Schematic" button instead (see below, the icon that has a play button inside a sheet of paper).

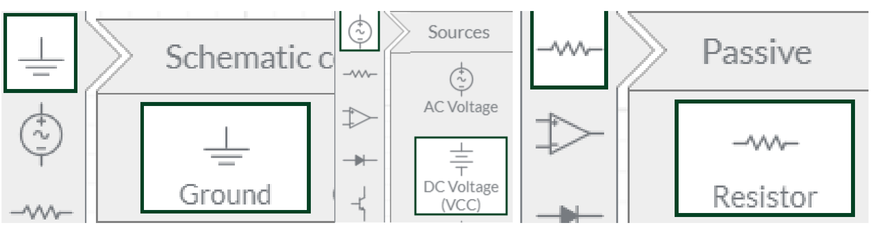


Step 2 – Adding components

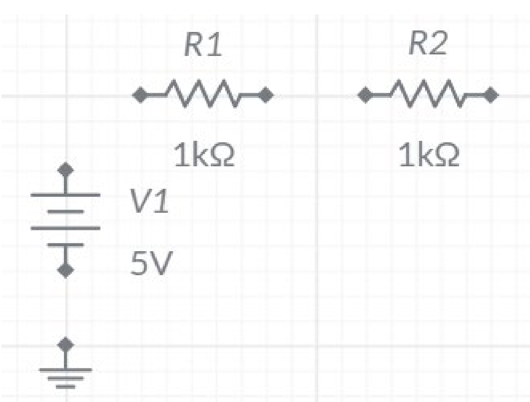
In this step, you will add a voltage source, two resistors, and a ground to your circuit canvas.

(MultiSim/Live users) The left row of buttons in MultiSim Live shows the available components in MultiSim Live. First, click on the component that you want to add. Then, hover your mouse on the circuit canvas. Click on any spot to put down your component. You should select (also see figures below):

1. The "Ground" out of the Schematic connectors;
2. "DC Voltage Source" out of the Sources button;
3. "Resistor" out of the Passive components.



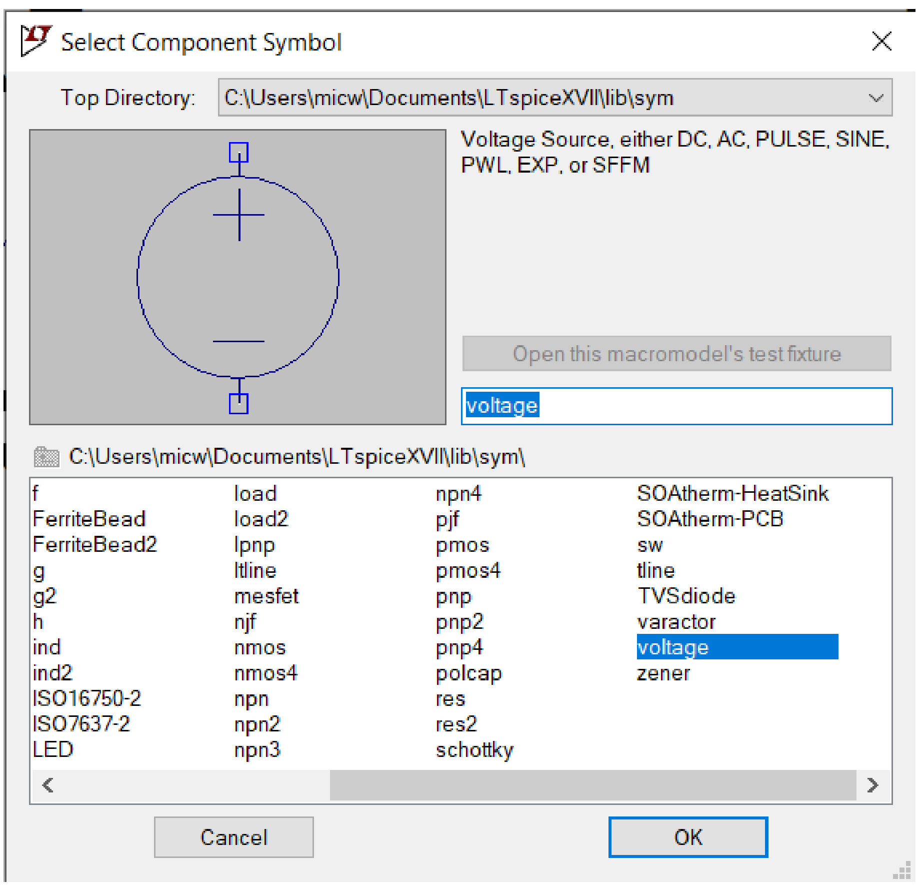
After adding the components, you should have a circuit similar to mine shown below.



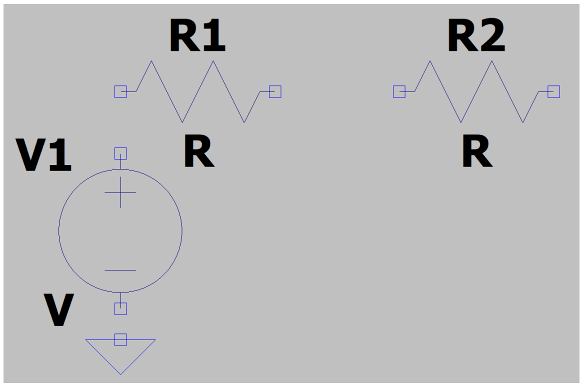
(LTspice users) You should be looking at the top row of buttons and will see something like this. From the left, the second button is your ground, and the fourth button is your resistor. You need to add one ground and two resistors to your circuit. To add components, follow the next steps:

1. Click on the corresponding button will allow you to add components that the button represents on the circuit canvas.
2. Click on the circuit canvas to add the active type of component.
3. Use (Windows) Ctrl+R to rotate and Ctrl+E to flip the circuit diagram. You probably will use the command key on Mac.
4. After adding each kind of component, I **highly suggest** you use a right-click to cancel adding the current type of component or otherwise your circuit canvas may start scrolling when your mouse reaches the edge of the circuit canvas. To recenter your schematic, use  button.

The eighth (8th) button from the left stands for "all components" and you will click on that button first if you want to add a voltage source. Search for "voltage" among the large amounts of symbols and click on "OK". Then, add one voltage source to your circuit canvas.



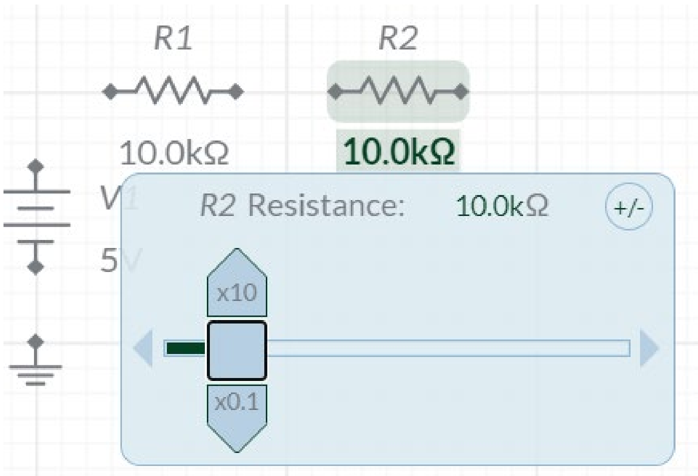
After adding all these components, your circuit should look like the picture below.



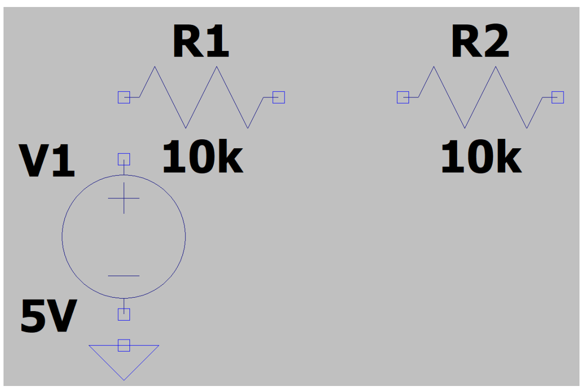
Step 3 – Changing component values

We are going to pretend that we will be using 5V power supplies and resistor values of 10kΩ.

(MultiSim/Live users) Click on the values (such as the 1kΩ) to change it. After changing the values, your circuit should look like the figure below.



(LTspice users) Right click on the values (which are R and V) on the schematics. Change the R into 10k and the V into 5V. You should have a figure looking like this after you finished changing them.

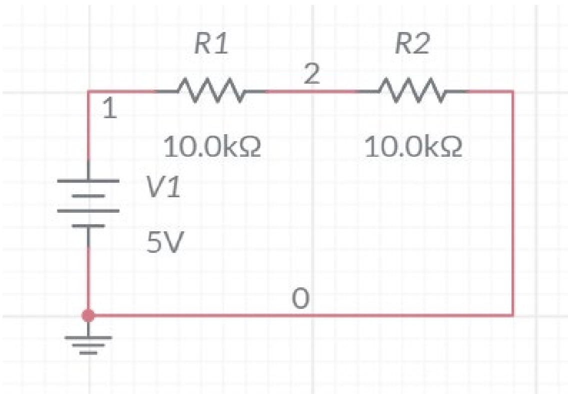


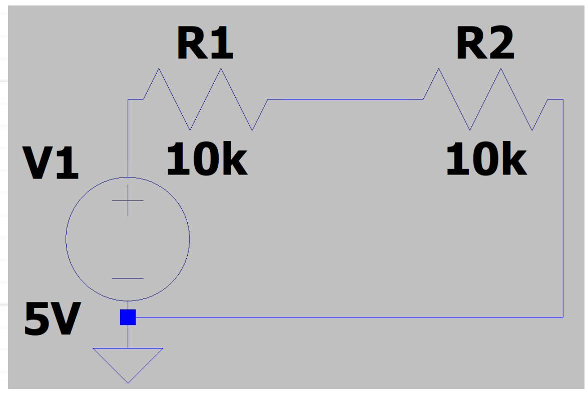
Step 4 – Connecting the wires

Now you would want to close the circuit.

- On MultiSim (Live), Click on the hanging nodes to add wires.
- On LTspice, use the pencil  (wire) tool to add wires.

After adding wires, you should be able to generate a circuit that looks like the example circuit below.





If you made a mistake:

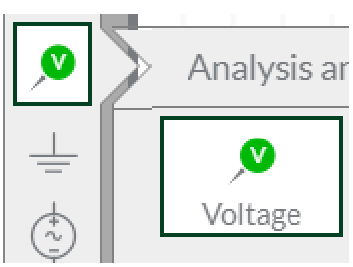
- On MultiSim (Live), left click on the wire and then you will be able to delete it.
- On LTspice, right click on the wire and there will be a pop-up menu that allows you to delete the wire or net. You may need to add components again if you deleted the net.

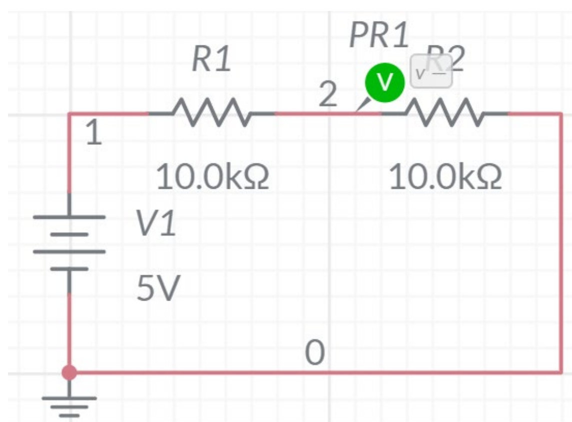
Step 5 – Adding your voltage probe

Now, you will tell the circuit simulator about the location(s) of where you want the simulation results. You will give the simulation software the directions by adding a voltage probe.

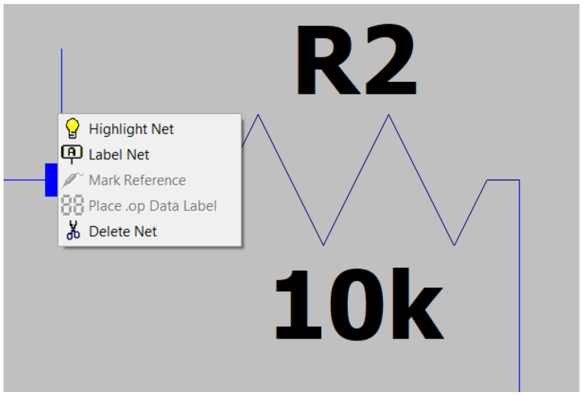
(MultiSim/Live Users)

Select the voltage probe from the left toolbar and put it right in front of the second resistor (R2). See pictures below. You should be measuring the voltage across R2.

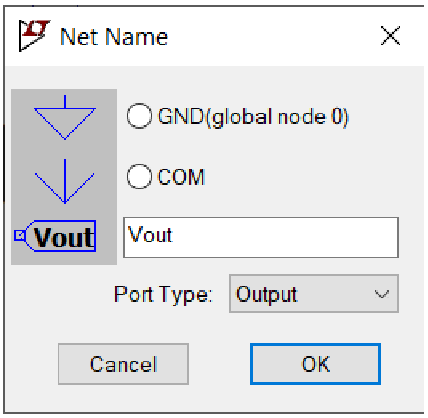




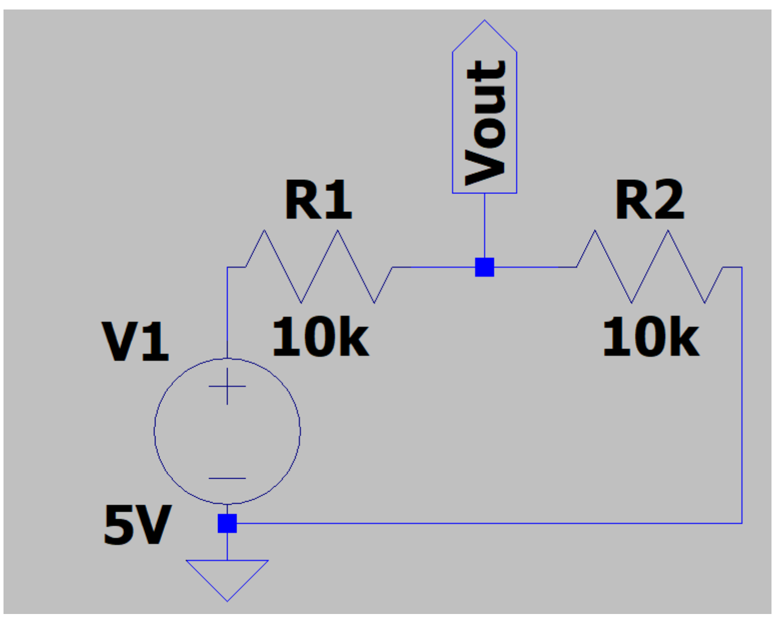
(LTspice users) Use the pencil (wire) tool to add a hanging wire leading from R2. Right-click the newly added wire and select “Label Net” from the pop out menu.



Then, select the Port Type to be “Output” and give it a name as “Vout”.



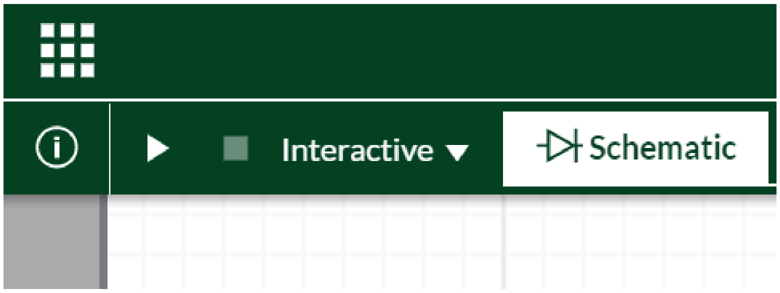
If it does not automatically place your Vout node, make sure to click on the hanging wire so that the hanging wire becomes an output node. Your circuit should look like this.

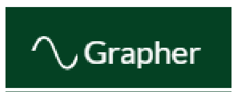


Step 6 – Simulate your circuit

Finally, you will be able to obtain some results!

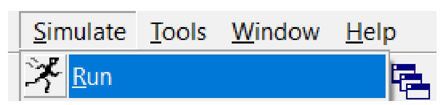
(MultiSim/Live users) Click on the "play" button to start simulation, then immediately click on "stop". Since we are performing DC simulations, the result is not dependent on time anyway.



Switch to the Grapher 

tab and record your results.

If you want to save your file, click on the top left corner (nine squares) and save your file. You can also export your circuits to other formats including pictures.

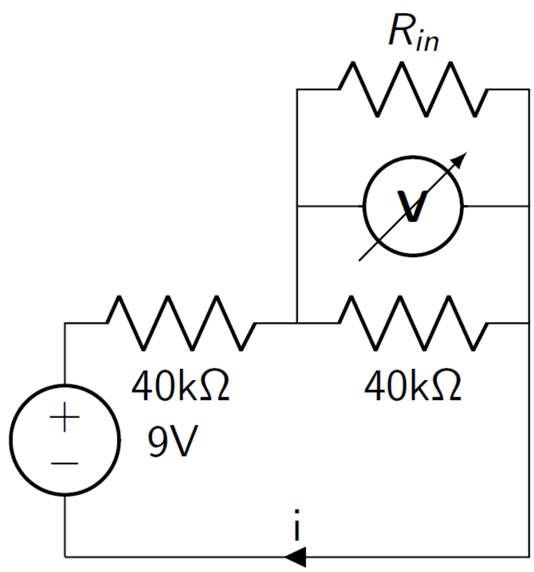
(LTspice users) From the Simulation menu, go to "Run". 

Then, switch to the "DC op pnt" tab and click "OK". You will get your results in the pop-up window. Save your file.

Task 2: Simulation of input impedances

We will study the effects of input impedances during Friday's lecture. You will perform simulations on the effects of non-ideal voltmeters. Namely, you will be simulating the following circuit.

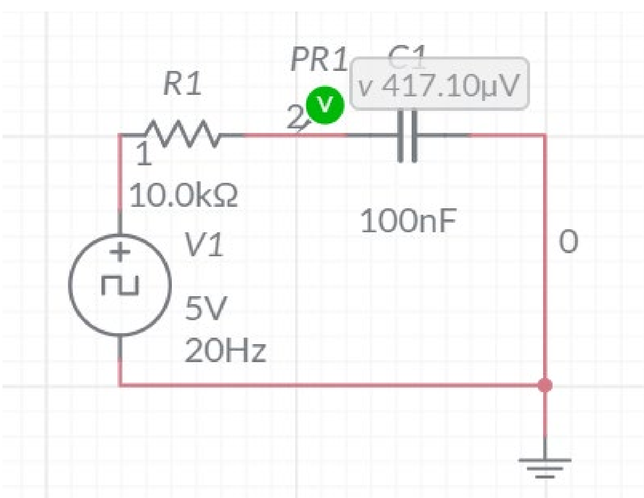
Use the knowledge that you just learned, simulate the reading on the voltmeter for R_{in} = 100kΩ and 1MΩ. The R_{in} represents the input impedance of the voltmeter.



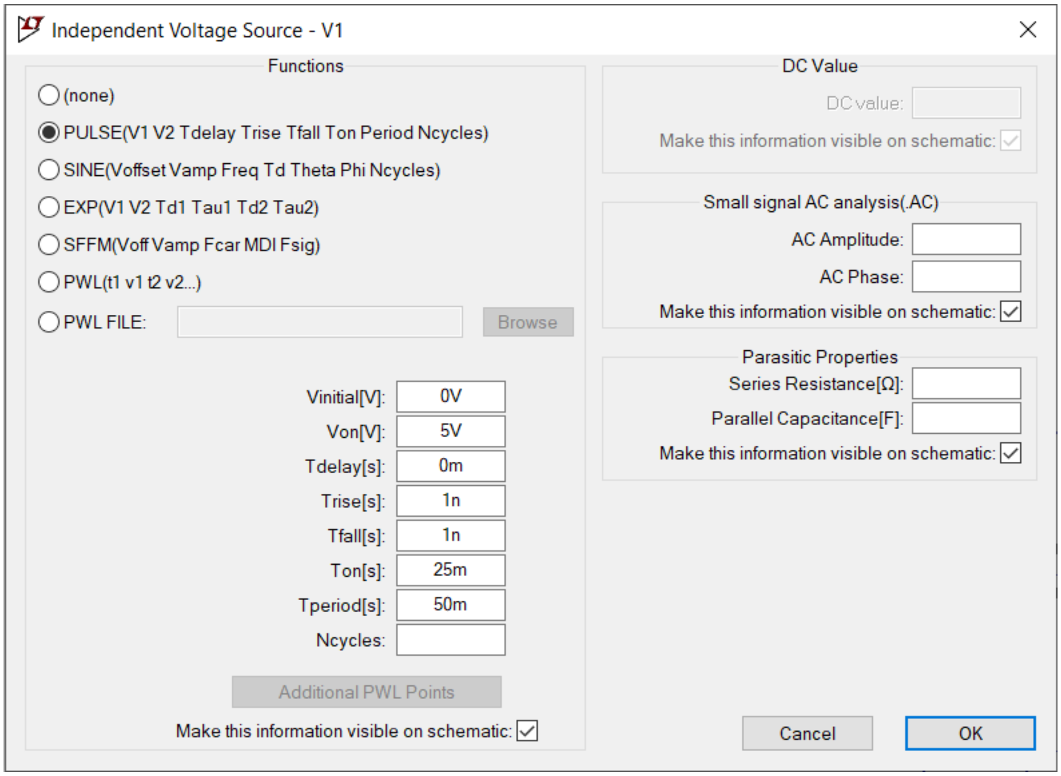
Task 3: Simulating an RC circuit

For this task, you will be simulating the forced response of RC circuits. You will be using a resistor of 10kΩ and a capacitor of 0.1μF. You will be using an alternating voltage source of a square wave at 20Hz. The amplitude of the square wave should be a clock-like signal that switches between 0V (when low) and 5V (when high). Find the voltage across the capacitor with respect to time.

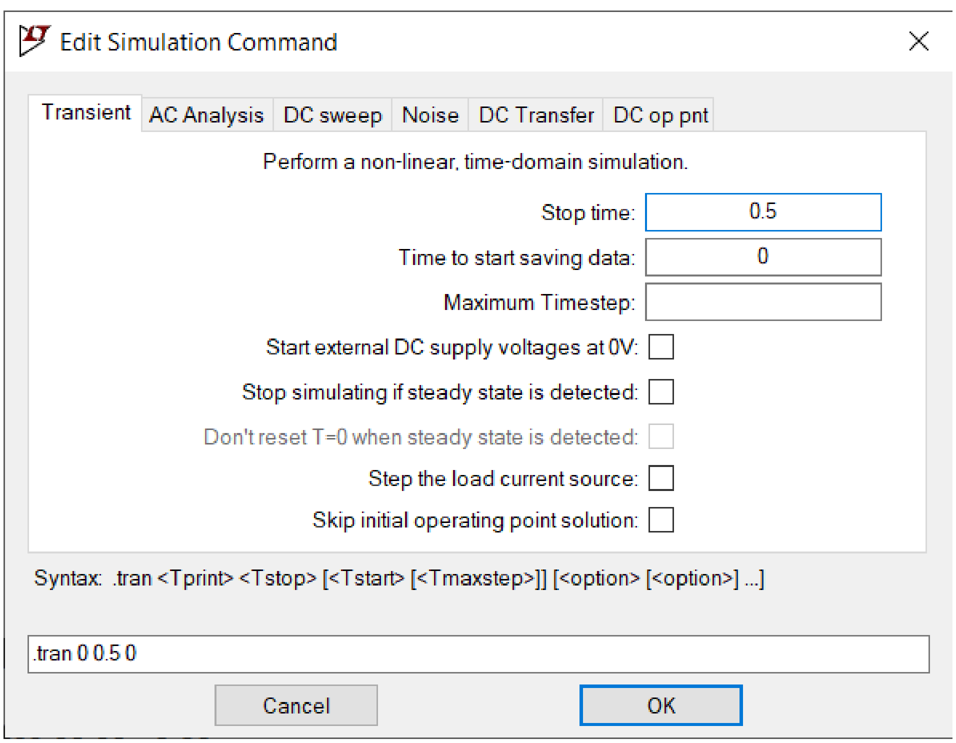
(MultiSim/Live users): You should have a circuit that looks like mine below. The voltage source you select should be a “Clock Voltage”. In the Grapher tab, make sure to select x and y axis ranges to display about two or three full cycles and can show all the graph (I’m using t between 0 and 100m, and voltage between -1 to 6 V for plotting).



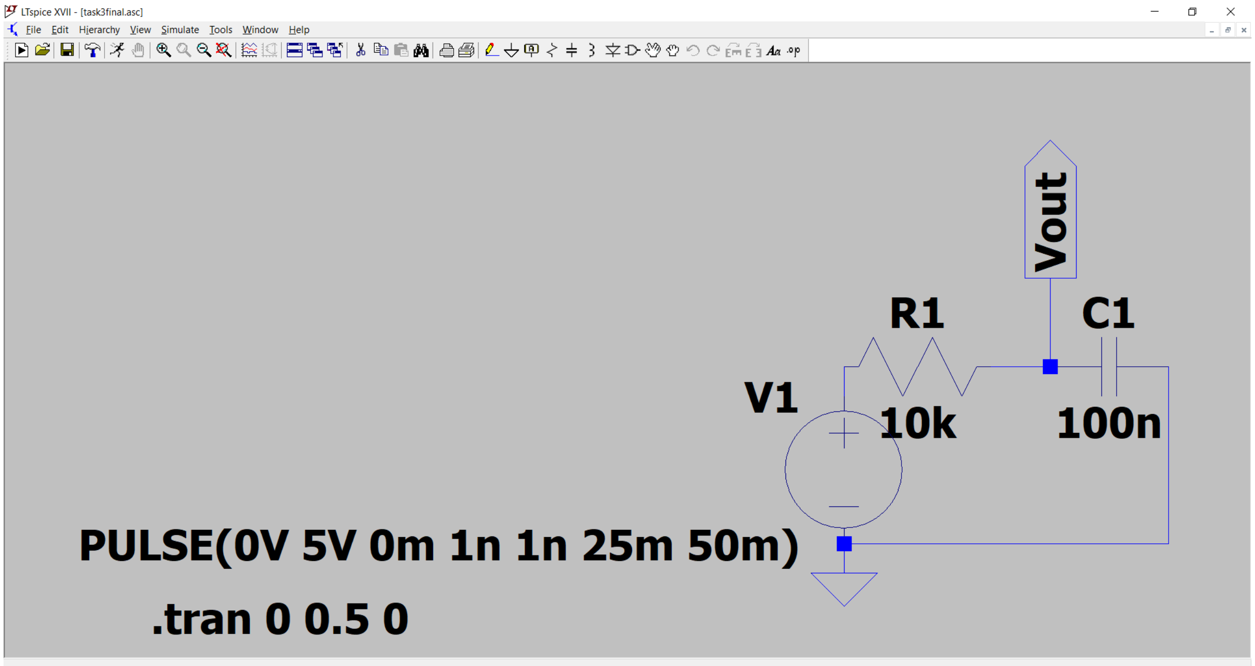
(LTspice users): After you assemble your circuit (that looks like the sample circuit below), make sure that your voltage source is defined as follows. To define the voltage source, click on the voltage source and select “Advanced”. If your voltage source is defined correctly, your voltage source should say PULSE(0V 5V 0 0 0 25m 50m). This means that your voltage source has a low of 0V and a high of 5V with 1 ns rise/fall time, stays on 5V for 25 ms in a period and a period of 50 ms (thus stay on 0V for 25ms in a period).



In addition, when you are about to run the simulation, select the "transient" analysis with an end time of 0.5 s such as below.



Your circuit should look like this for a full configuration.



Lab Report

1. For task 1, report the simulated voltage across R2 and compare it with your theoretical calculations.
2. For task 2, report your simulated values and compare the values with respect to the calculated values from Friday's lecture. Briefly discuss whether low or high input impedance is preferred and why.
3. For task 3, attach a voltage-time graph across the capacitor. From your graph, calculate the time constant of the RC circuit and compare it to the theoretical results.