

## Laboratory 7

# Introduction to the Use of SPICE

## Objectives

- Learn what SPICE is and its basic functions.
- Learn how to use SPICE to construct a simple circuit.
- Learn how to use SPICE to simulate and analyze the circuit.

## Equipment and components

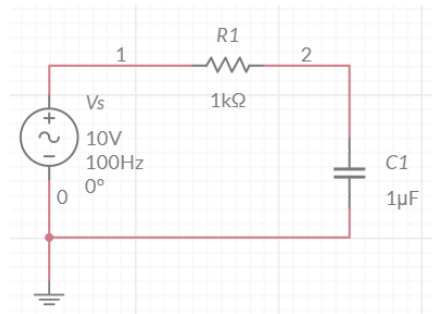
- A personal computer
- SPICE software

## Preliminary work

1. SPICE is an acronym for Simulation Program with Integrated Circuit Emphasis. SPICE is widely used in industries and universities, e.g. for integrated circuits design and predictive behavior of circuits. See this link for further details: <https://en.wikipedia.org/wiki/SPICE>
2. In this lab assignment you will use *Multisim Live*, an online SPICE simulator. Before the lab you should try to familiarize yourself with this tool by
  - Visiting <https://www.multisim.com/> and creating an online account (it is free!)
  - Watching the following introductory tutorials:
    - i. [https://www.youtube.com/watch?v=gBhIr2AcaR0&ab\\_channel=EEProfLady](https://www.youtube.com/watch?v=gBhIr2AcaR0&ab_channel=EEProfLady)
    - ii. [https://www.youtube.com/watch?v=p9wNpIWL8ko&ab\\_channel=GlenHartman](https://www.youtube.com/watch?v=p9wNpIWL8ko&ab_channel=GlenHartman)
  - Checking the online documentation of *MultiSim Live*, available in this link:
    - <https://www.multisim.com/help/getting-started/>

## Procedure

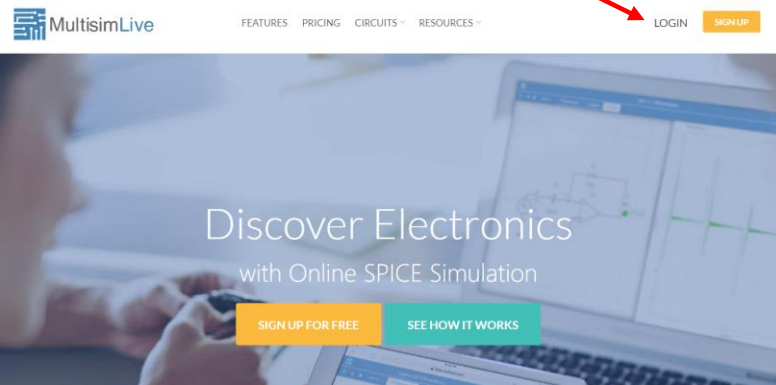
The objective of this assignment is to simulate and analyze the behavior of a circuit composed of a sinusoidal time-varying voltage source, a resistor (R) and a capacitor (C). This is known as a RC circuit (see figure below)



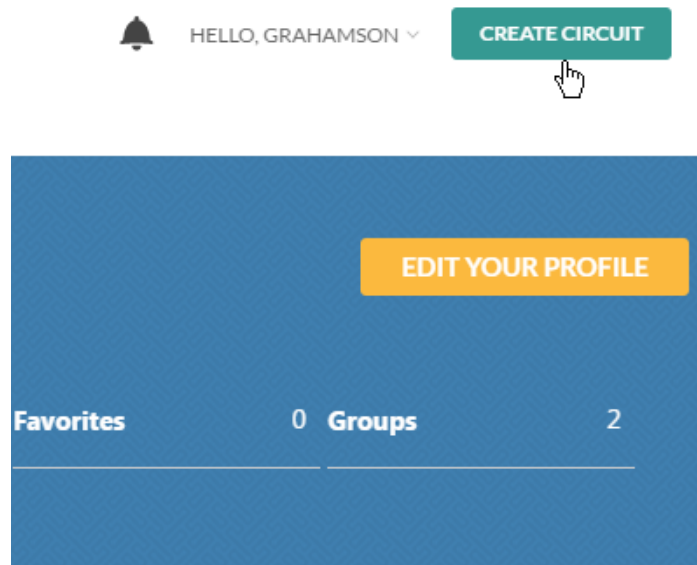
Execute the following steps to create the RC circuit:

Go to <a href="https://www.multisim.com/">https://www.multisim.com/</a>	
Press “Signup for free”	
Create an User Account	<p>Create an NI User Account</p> <p><small>Already have an account? <a href="#">Log In &gt;</a></small></p> <p>Alias <input type="text" value="johndoe77"/></p> <p>First Name <input type="text"/> Last Name <input type="text"/></p> <p>Role <input type="text" value="Please Select"/></p> <p>Company <input type="text"/></p> <p>Email Address <input type="text"/></p>

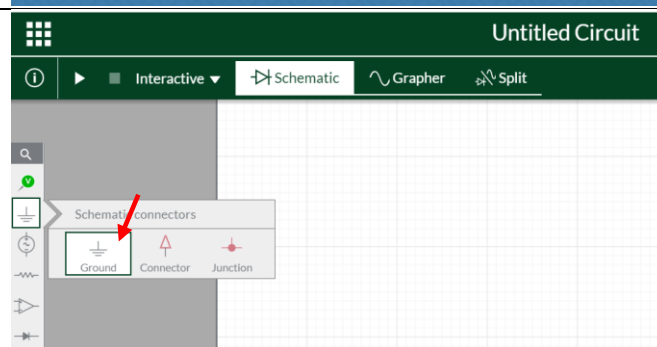
Once the account is created, you can login into the online simulator



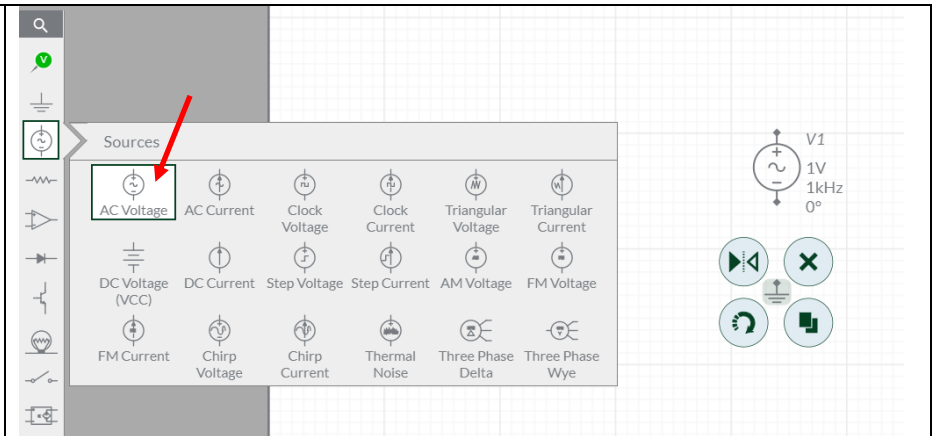
Create a circuit



Insert ground in your new circuit



Insert an AC voltage source in the circuit



Change the parameters of the voltage source

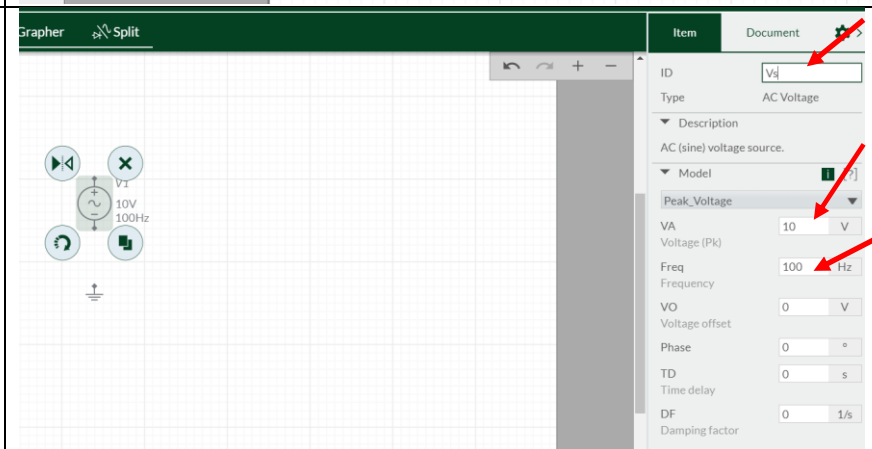
ID:  $V_s$

VA: 10V

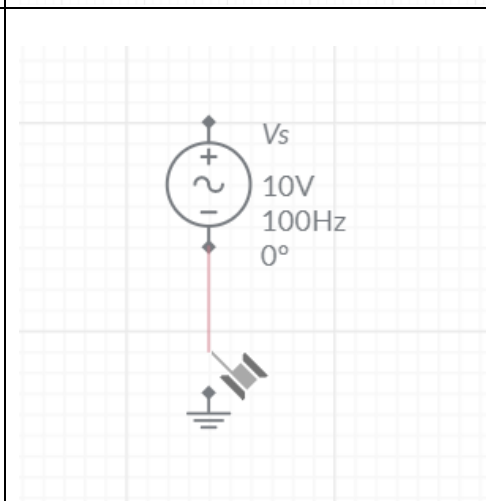
Freq: 100Hz

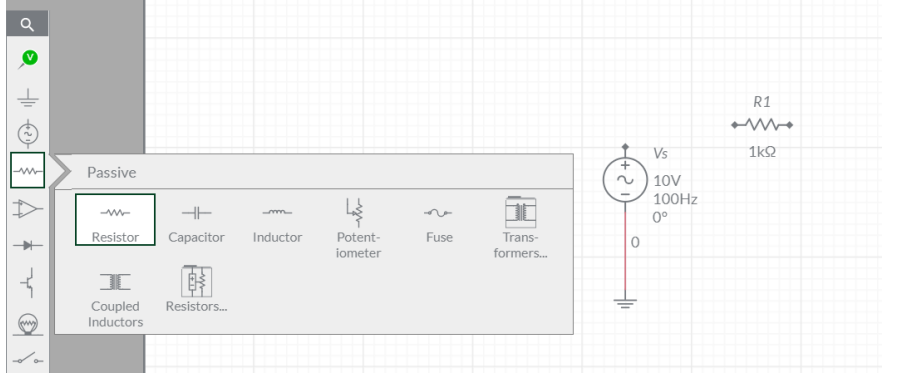
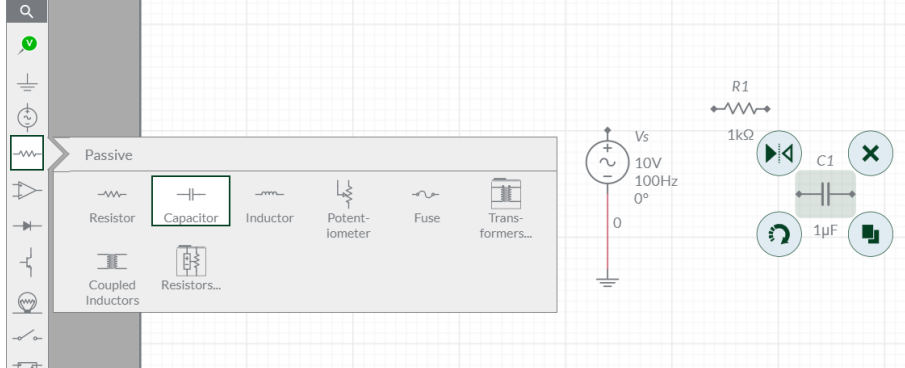
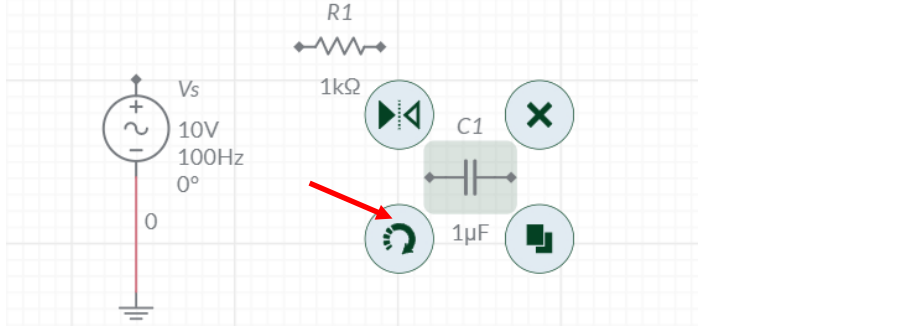
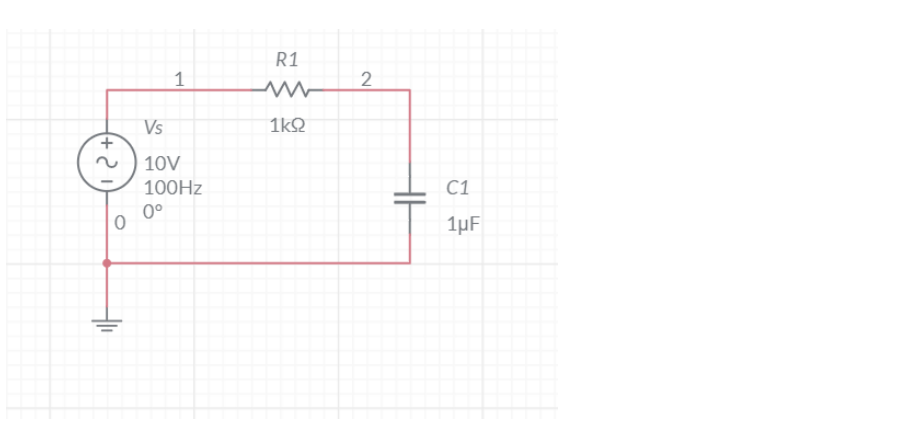
This will create a sinusoidal voltage source with 10V peak and frequency of 100 Hz, i.e.  

$$v_s = 10 \sin(2\pi 100t)$$



Connect the voltage source to the ground  
 (Note: place the cursor next to the terminal of one component; wait one second; the cursor will change into “wiring mode”, allowing you to connect different electrical elements)

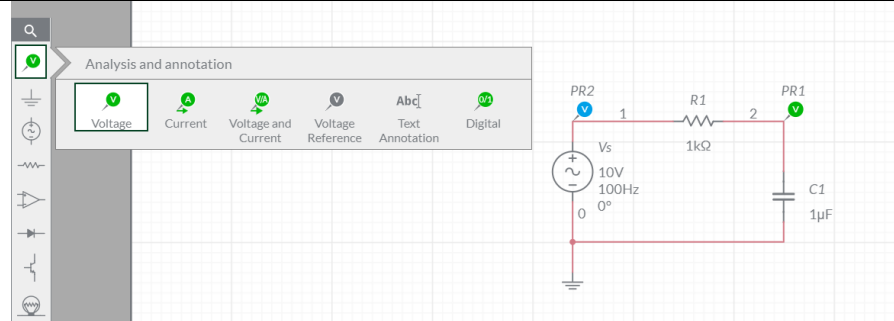


<p>Insert a resistor</p>	
<p>Insert a capacitor</p>	
<p>Rotate the capacitor by - Selecting the capacitor - Pressing the rotate button</p>	
<p>Connect the circuit elements as show in this figure</p>	

Add two voltage probes

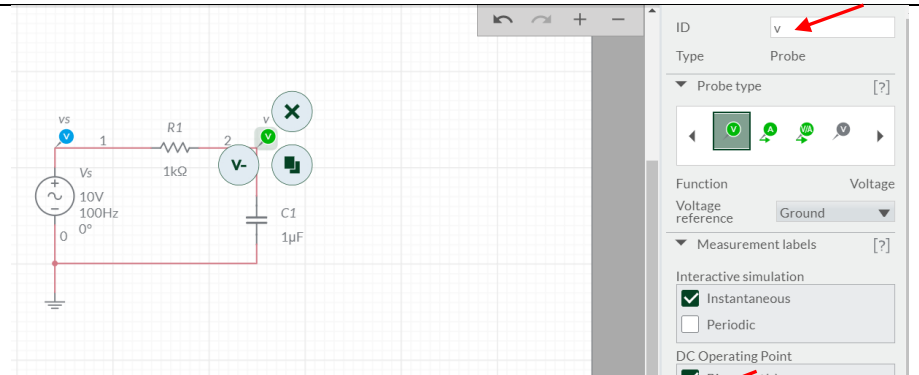
- Place one voltage probe at the “upper terminal” of the capacitor

- Place another voltage probe at the “+” terminal of the voltage source



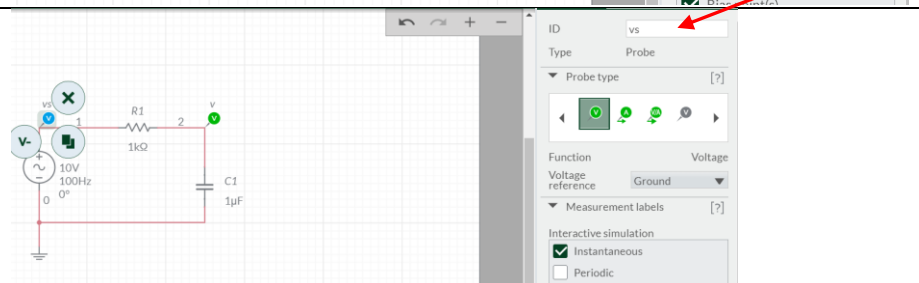
Change the ID of the first voltage probe (connected to the capacitor) to

ID: v

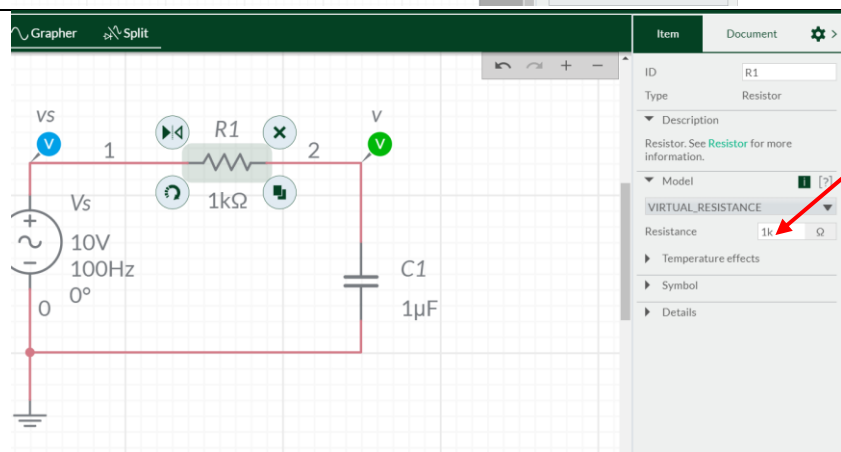


Change the ID of the second probe to vs

ID: vs

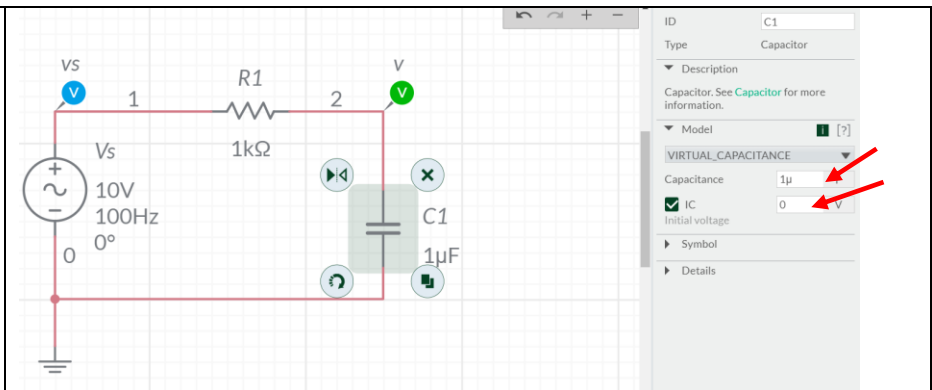


Make sure the resistance value of R1 is 1kΩ (select the resistor and then check the right window pane, “Resistance”)

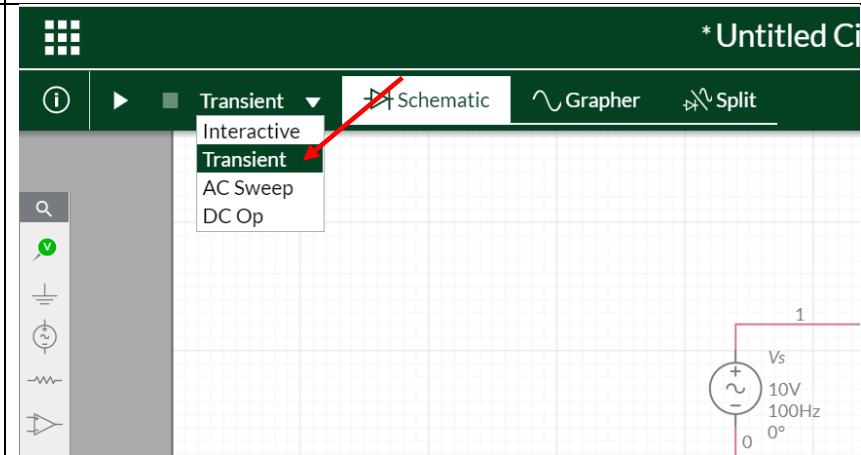


Make sure the capacitance value is  $1\mu F$  (select the capacitor and then check the right window pane. “Capacitance”)

Select the check box “IC” (this will fix the initial voltage of the capacitor)



Change simulation mode to Transient

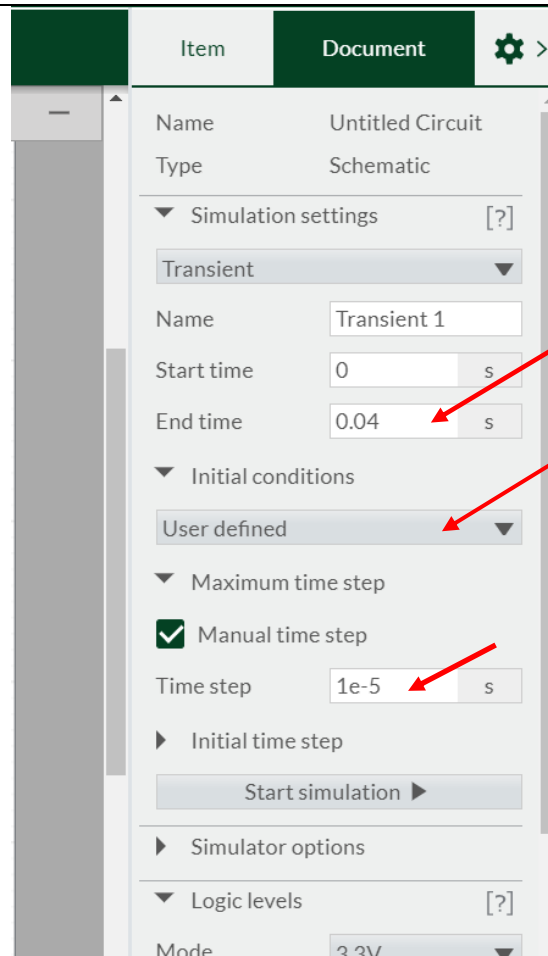


Change the simulation settings on the right side of the window:

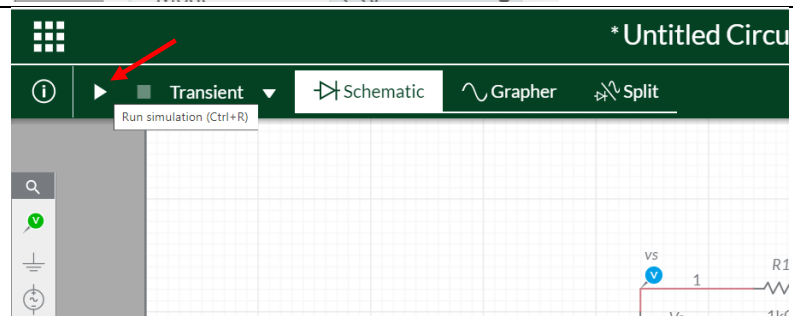
End Time: 0.04s

Initial condition: user defined

Manual time step:  $1e-5$ s

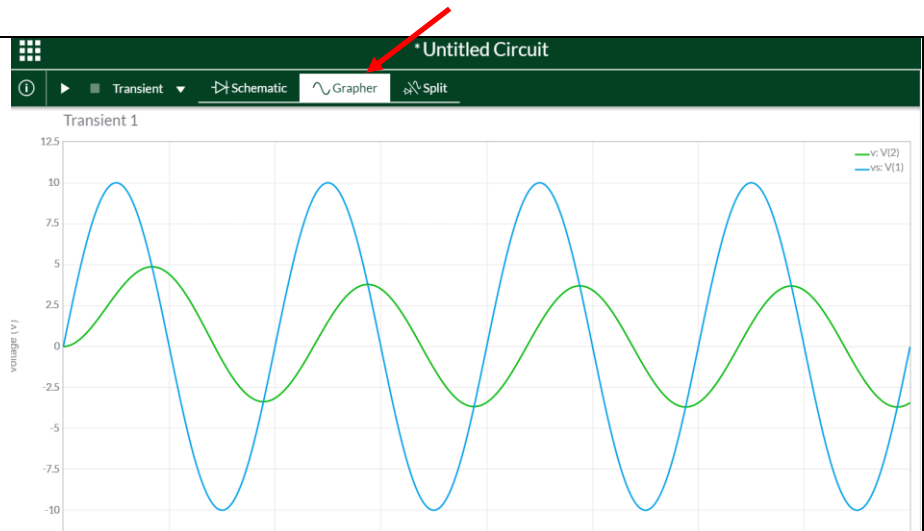


Press "Run Simulation:

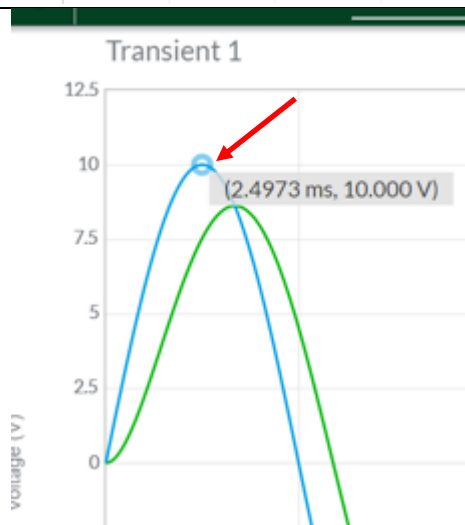




Check the grapher tab for the results

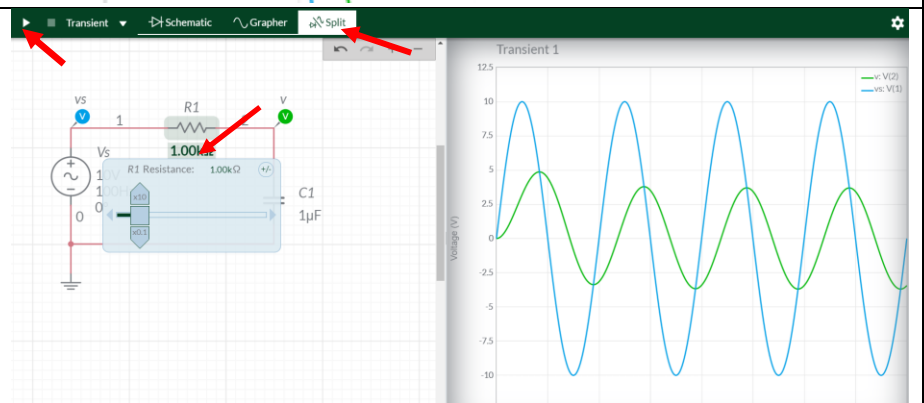


You can use the cursor to check the values of the voltage signals



You can also easily check the effect of different values of resistance in the behavior of the circuit:

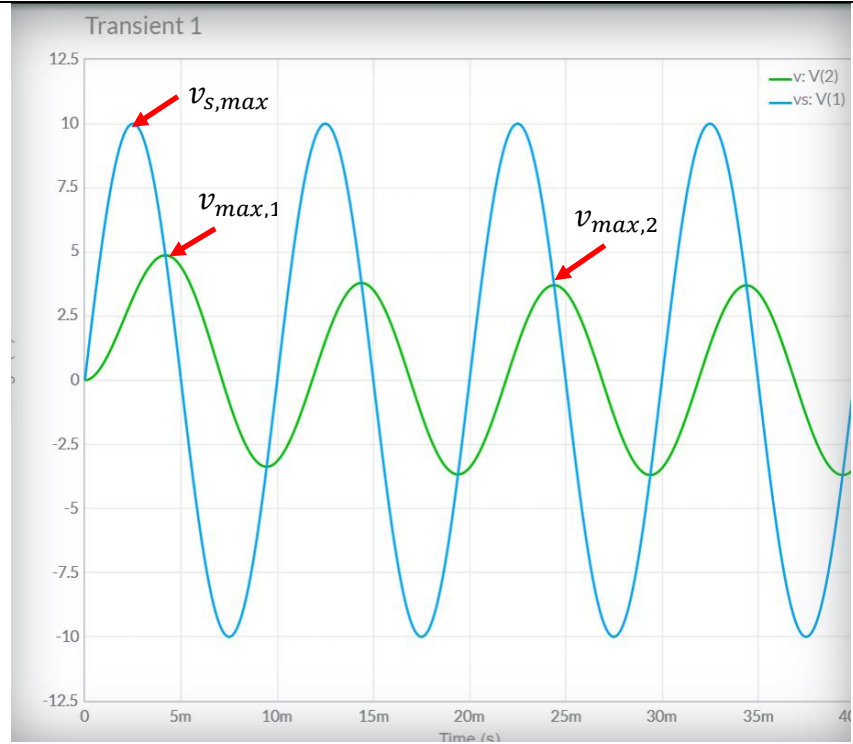
1. Press “1.00k $\Omega$ ”. This will open a small window which allows quick modification of the resistance value
2. Press “Split” option. This will let you see side-by-side the circuit and the voltage signals.
3. Press Simulate



Note down the values of  $v_{s,max}$  = peak voltage of the voltage source

$v_{max,1}$  = peak voltage of capacitor during the first cycle

$v_{max,2}$  = peak voltage of the capacitor during the third cycle



- After creating the circuit, you will perform a sensitivity analysis by varying the resistance  $R1$  and evaluating the peak voltages in the voltage source ( $v_{s,max}$ ) and capacitor ( $v_{max,1}$  and  $v_{max,2}$ ); check the previous step for the definition of these voltages. Complete the table below with the obtained results:

$R1$ ( $\Omega$ )	50	100	200	500	1000	1500	2000	2500	3000	4000
$v_{s,max}$ [V]										
$v_{max,1}$ [V]										
$v_{max,2}$ [V]										

- Plot  $v_{s,max}$ ,  $v_{max,1}$  and  $v_{max,2}$  in terms of  $R$ .
  - Do the peak values of the voltage source ( $v_{s,max}$ ) and the capacitor response ( $v_{max,1}$ ,  $v_{max,2}$ ) change when you modify  $R1$ ? If yes, how do they change?
- Is the frequency of the capacitor response (voltage probe  $v$ ) different from that of the voltage source (probe  $vs$ )? It might be useful to check Chap. 9.1 from the course textbook [1] to refresh the concepts of frequency in sinusoidal signals.
- Summarize your findings of this lab in a report. In the report you should also include:

- Screenshot of the circuit created with the online SPICE simulation
- The sinusoidal voltage signals (voltage source and capacitor) obtained when  $R1 = 1k\Omega$ .

## References

[1] C. Alexander and M. Sadiku “Fundamentals of Electric Circuits”, 7th Edition, 2021, McGraw-Hill