ENGR 065 Circuit Theory

Laboratory 9

RL and RC Circuits

Objectives

- Observe the transient and steady-state responses of RL and RC circuits.
- Learn how to measure time constant of first order circuits.
- To learn how to simulate RL/RLC circuit with SPICE

Equipment and components

- A desk computer
- SPICE software

Preliminary Work

1. Review the lecture slides about RL/RC Circuits and Appendix 1 (included at the end of this document)
2. Review what you have learned in the previous labs about online SPICE simulator. We will use this tool to analyze the behavior of circuits with resistors, capacitors and inductors
3. Read the Lab Procedure (below) and **complete the tables.**

Procedure

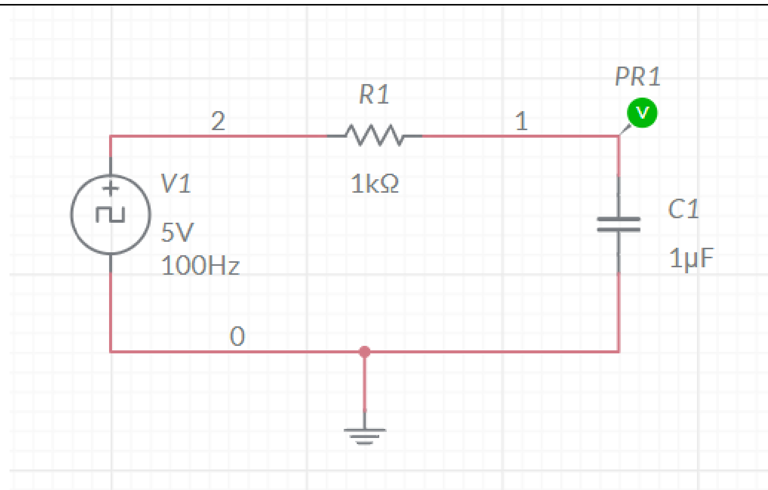
1. The first objective of this assignment is to simulate the behavior of the RC circuit shown below with a resistor 1 kΩ, a capacitor of 1μF and a square wave voltage generator.

<table>
<tr>
<td>
Create the RC circuit in the online SPICE simulator (check previous lab for details on how to use the online simulator)

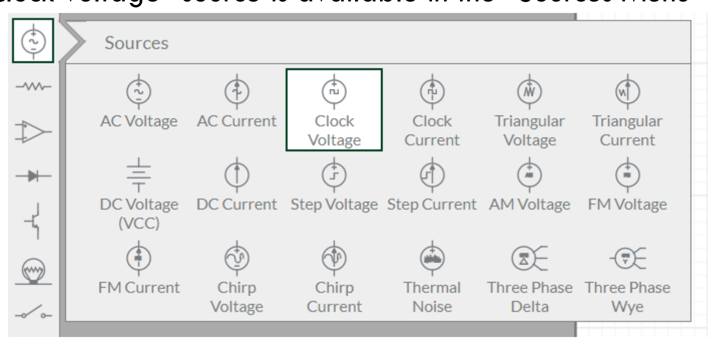
Connect in series a

- 1kΩ resistor
- 1μF capacitor
- "Clock voltage" source (this is a source that generates a square wave)

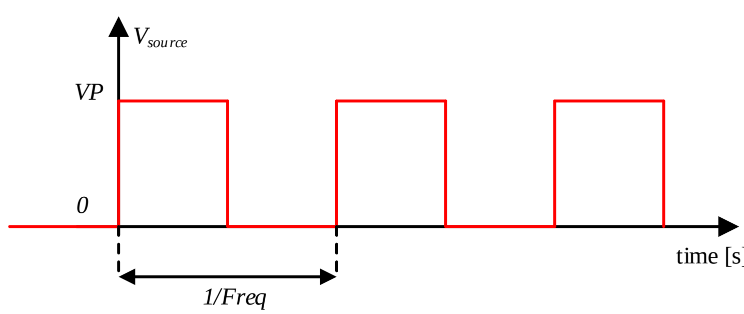
Connect also

- the ground to node that interconnects the voltage source and the capacitor
- A sensor to measure the capacitor voltage
</td>
<td>


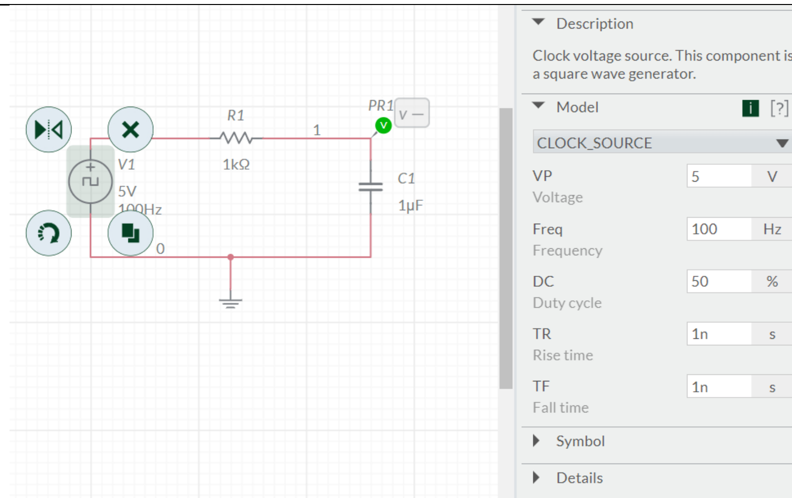
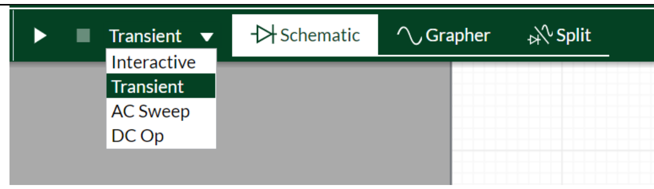
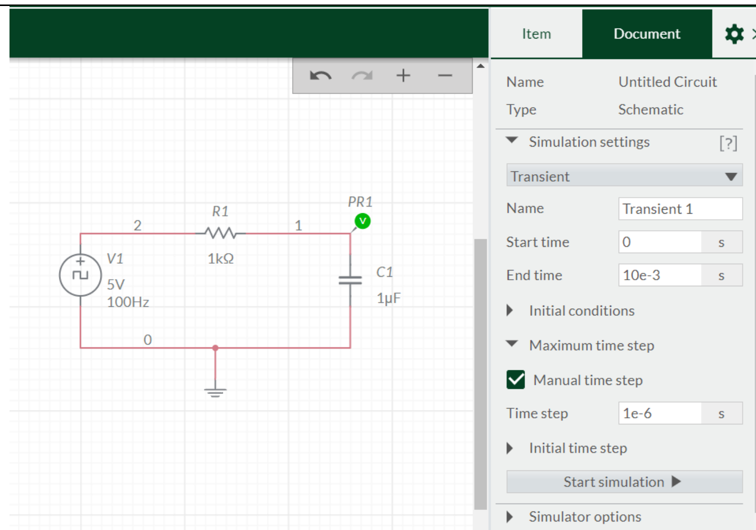
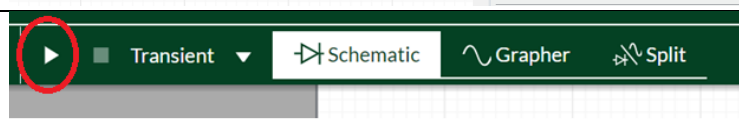
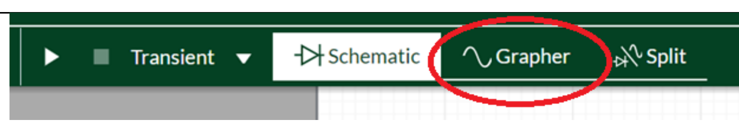
"Clock voltage" source is available in the "Sources Menu"

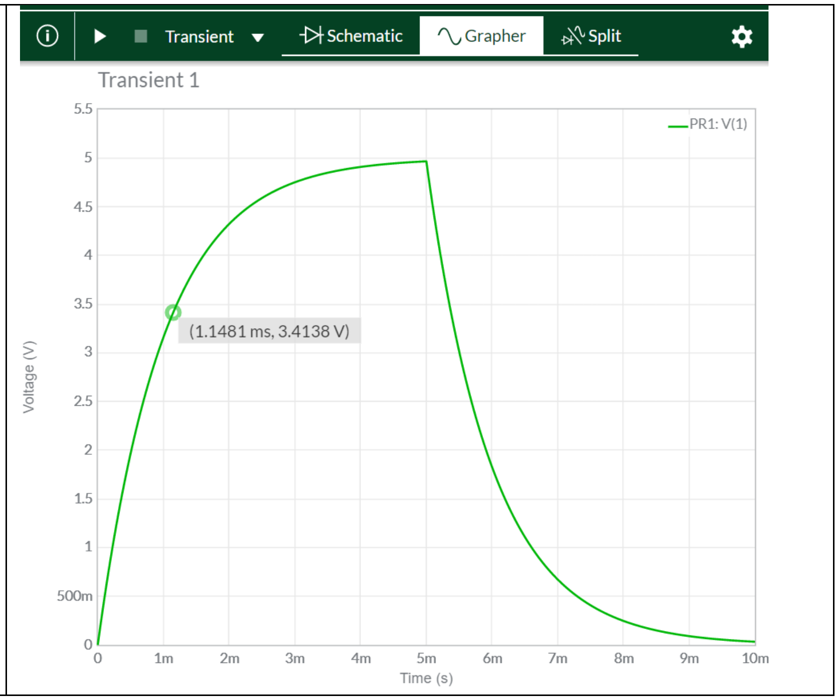


</td>
</tr>
<tr>
<td>
The "clock voltage" source generates a square wave. We can modify the

- VP = maximum value of the square wave [V]
- Freq = frequency of the wave [Hz]
</td>
<td>


</td>
</tr>
</table>

<table>
<tr>
<td>After selecting the “clock voltage”, modify its settings as follows:
• VP = 5V
• Frequency = 100Hz</td>
<td>
</td>
</tr>
<tr>
<td>Go to the top left side of the window and select “Transient” simulation mode</td>
<td>
</td>
</tr>
<tr>
<td>After selecting Transient Mode, a new simulation setting menu will emerge on the right side of the window. Use the following settings:
• Start Time =0 s
• End time = 10e-3s
• Manual time step: TimeStep = 1e-6s</td>
<td>
</td>
</tr>
<tr>
<td>Press Run Simulation</td>
<td>
</td>
</tr>
<tr>
<td>Press “Grapher” and analyze the capacitor voltage</td>
<td>
</td>
</tr>
</table>



2. Measure the time constant of the RC circuit by analyzing the voltage signal available in the "Grapher Window" (see also Appendix) . Fill out the table below.

Capacitance	Resistance	Calculated Time Constant*	Measured Time Constant
$1\mu F$	1 kΩ	1 ms	0.998 ms

3. Place two 1 μF capacitors in series. Repeat 1 and 2.

Capacitance	Resistance	Calculated Time Constant*	Measured Time Constant
0.5 uF	1 kΩ	0.5 ms	0.501 ms

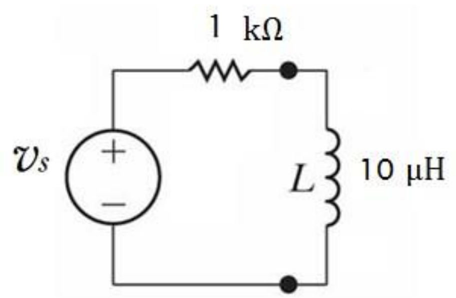
4. Place two 1 μF capacitors in parallel. Repeat 1 and 2. Note: to facilitate the measurement of the time constant you may decrease the function generator frequency to 50Hz.

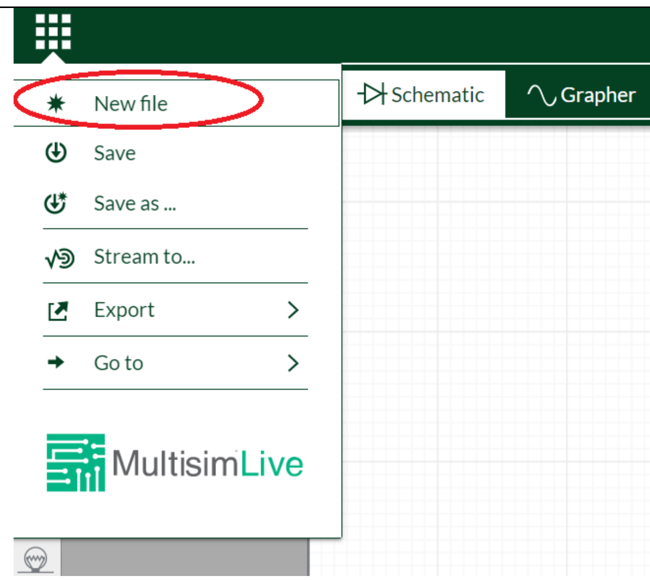
Capacitance	Resistance	Calculated Time Constant*	Measured Time Constant
2 uF	1 kΩ	2 ms	1.997 ms

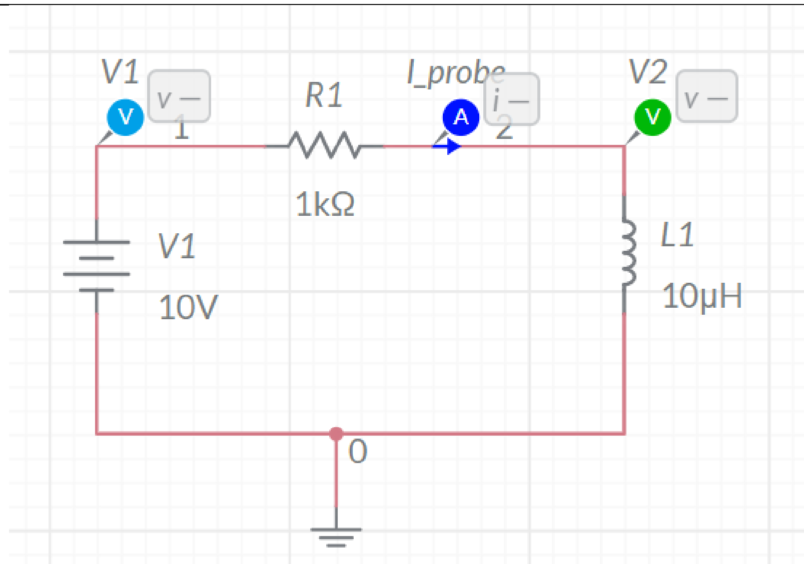
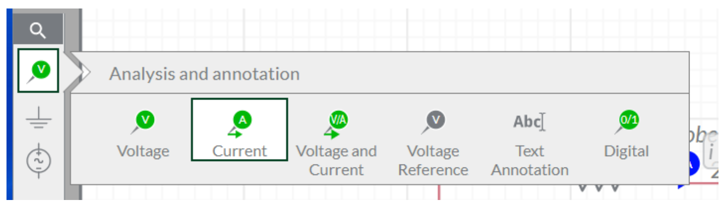
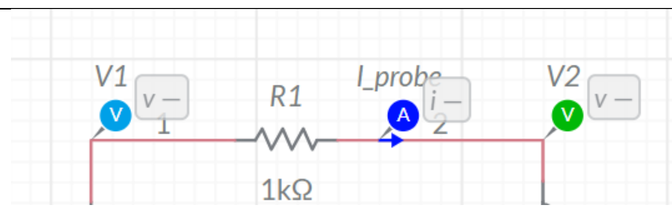
Does the time constant change with different capacitor combinations? Why?

Yes indeed, as seen within the simulation since the capacitance and time constraint has a direct correlation, additional capacitors series are introduced to the model, the equivalent capacitance decreases and in turn, the time constraints also decrease.

5. Construct the RL circuit shown below with a resistor of 1 kΩ and an inductor of 10 μH. Setup the voltage source with a constant voltage, $v_s = 10\ V$. Measure the current in the circuit and voltages across the resistor and the inductor.



Create the RL circuit in the online SPICE simulator by starting a New (Simulation) file	

Construct the RL circuit with: • 10V DC Source • 1 $k\Omega$ resistor • $10\mu H$ Add also • ground (the node that connects the DC source to the inductor) • Voltage probes • Current probe	 Current/Voltage probes available in the "Analysis and annotation probe" 
Press "run simulation"	
Observe the values of the current and voltage probes.	

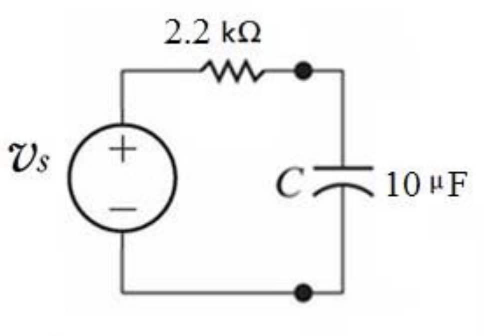
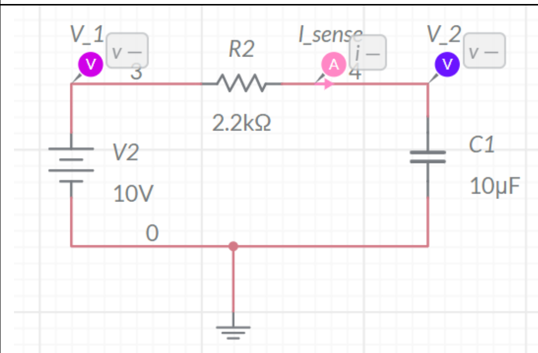
Fill out the following table:

	Calculated Value*	Measured Value
Current	0.01 A	0.01 A
Voltage across the inductor	0 V	0 V
Voltage across the resistor	10 V	10 V

What did you find from the above table? Explain your results.

I can see that the RL circuit first is in transient state by hindering the current through the inductor but eventually reach the steady state by allowing current through the resistors.

6. Construct a circuit shown below with a resistor of 2.2 kΩ and a capacitor of 10 μF. Setup the voltage source with a constant value $v_s = 10\ V$. Measure the current in the circuit and voltages across the resistor and the capacitor.

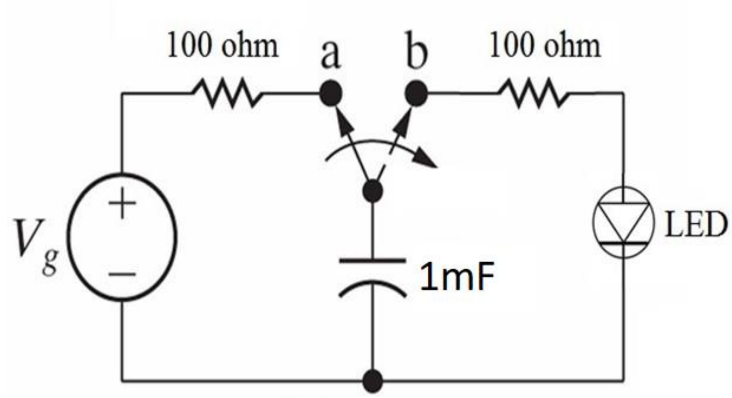
Circuit diagram	Implementation of the circuit in the online simulator
	

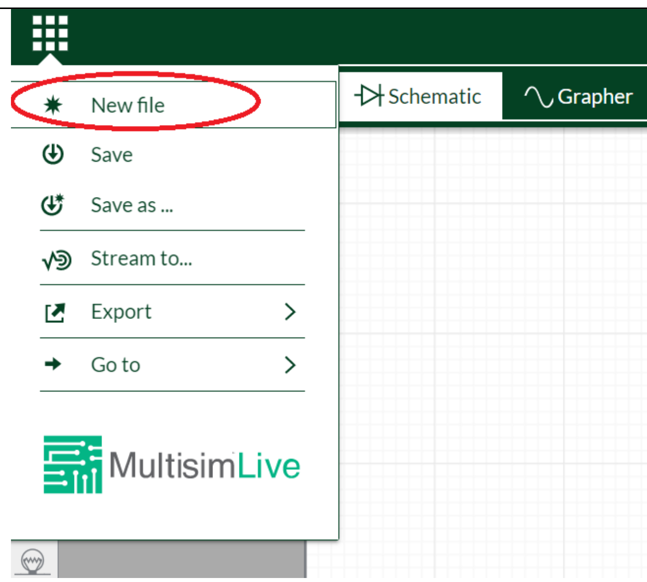
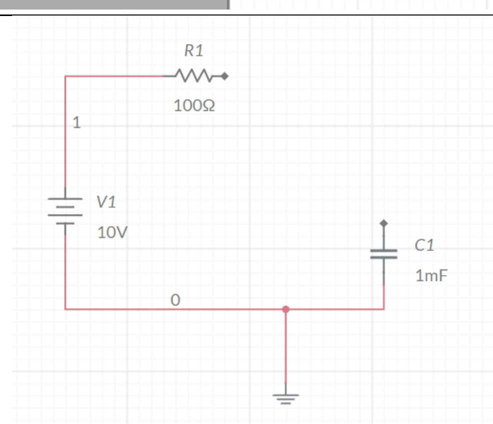
	Calculated Value*	Measured Value
Current	0 A	9.98 pA
Voltage across the capacitor	10 V	10 V
Voltage across the resistor	0 V	0 V

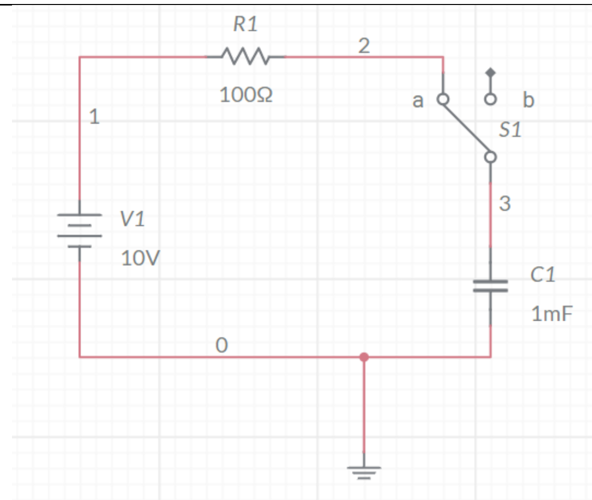
What did you find from the above table? Explain your results.

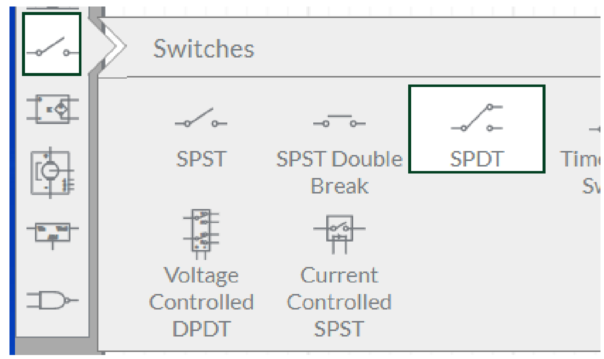
Current is initially at zero, but RC circuit will store up current within the capacitor reaching it maximum value while voltage in the resistor will decrease and vice version.

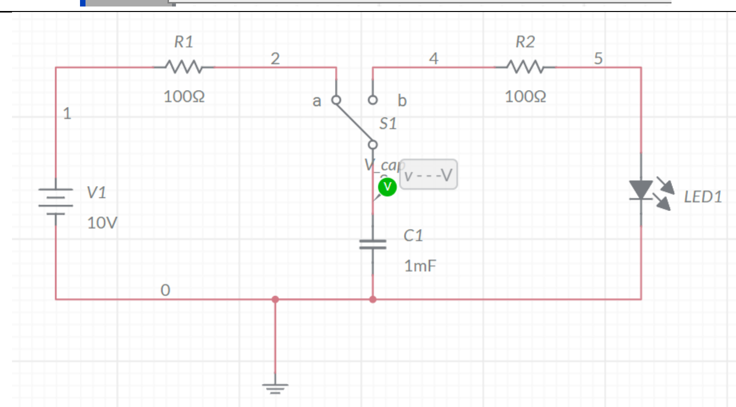
7. Construct a circuit shown below with 2.2 kΩ resistor and an LED. Setup the voltage source with $v_g = 10\ V$.

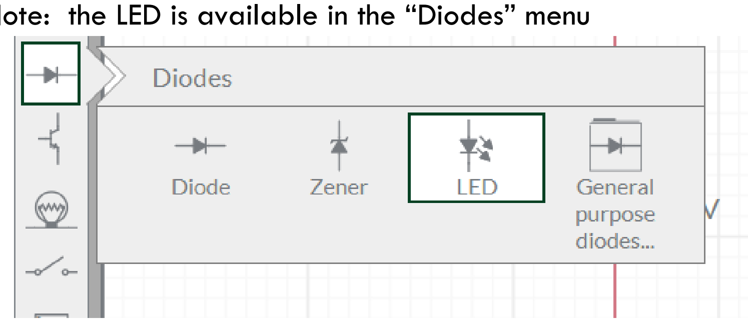


<table>
<tr>
<td>Create the circuit in the online SPICE simulator by starting a New (Simulation) file</td>
<td>
</td>
</tr>
<tr>
<td>Construct the circuit with the following elements
• 10V DC voltage
• 100 Ω resistor
• 1 mF capacitor
• Ground (node connecting the voltage source and the capacitor)</td>
<td>
</td>
</tr>
</table>

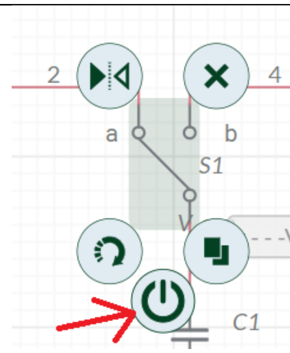
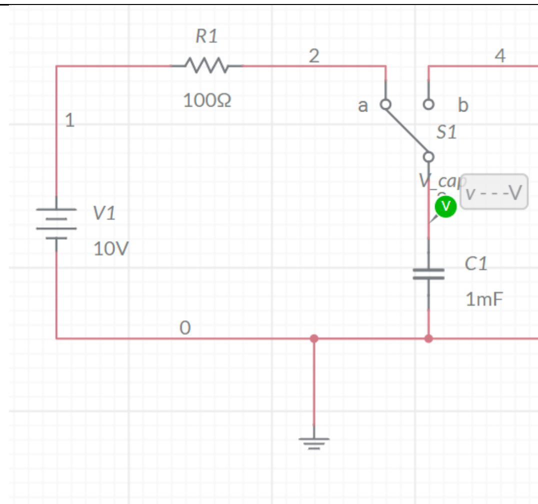
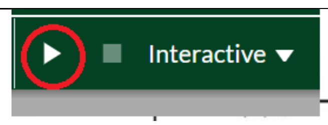
<table>
<tr>
<td>Add a SPDT (Single Pole, Double Throw) switch to the circuit as shown on the right figure</td>
<td>


Note: SPDT switch available in the “Switches” menu


</td>
</tr>
<tr>
<td>Connect a second 100Ω resistor and a LED in series with the terminal “b” of the switch (see figure on the right)</td>
<td>


Note: the LED is available in the “Diodes” menu


</td>
</tr>
</table>

Note: you can manually change the position of the switch (click in the switch and then press the "toggle option" (see red arrow on the right)	
Make sure that the switch is position "a"	
Run the simulation	

8. At the beginning, the switch is placed in the position **"a"**. What happens to the capacitor voltage? The capacitator charges the 10 v voltage source .

9. Then move the switcher to the position "**b**" (you can click on the switch during simulation mode to change its position). What happens to the capacitor voltage? Check also the color of the LED. Does the LED become red? Explain why.

The capacitor is slowly discharging indicating that the voltage within the LED has reached its maximal potential and will beginning to decrease - inducing the dimming of the LED.

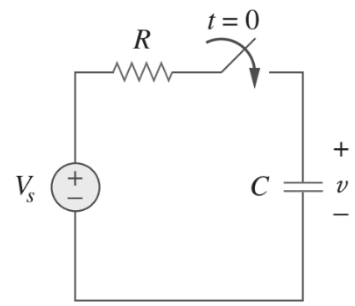
Questions and conclusions

• Summarize your findings and explanations in response to the questions posed in this lab.

Appendix 1 – Measuring the Time Constant of a RC Circuit

As discussed during the ENGR065 lectures, the step response of a RC circuit can be represented by a first-order differential equation:

$$\tau \frac{dv}{dt} + v = V_s, \quad v(0) = V_0, \qquad t \geq 0$$



Where v is the capacitor voltage $[V]$, $\tau = RC$ is the time constant $[s]$, R the resistance value, C the capacitance [F], V_s is the source voltage [V] and V_0 the initial capacitor voltage [V].

If we assume $V_0 = 0\ V$, the solution of this equation is given by

$$v(t) = -V_s e^{-t/\tau} + V_s = V_s(1 - e^{-\frac{t}{\tau}})$$

The resulting voltage waveform is depicted in the following figure.

After $t = \tau$ seconds, the capacitor voltage reaches $v(t = \tau) = V_s(1 - e^{-1}) \approx 0.63V_s$. By measuring the time it takes to reach $0.63V_s$ you will be able to estimate the time constant τ (see figure below).

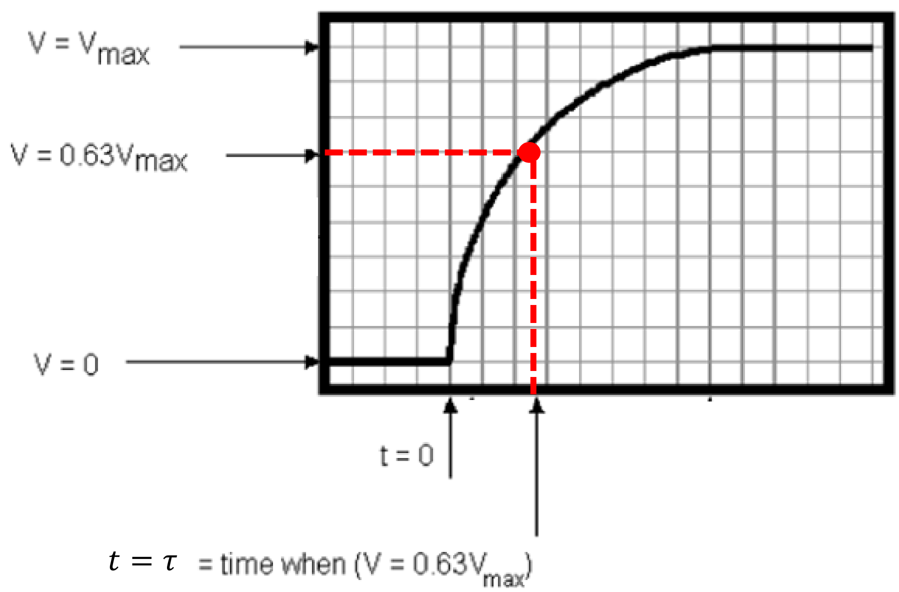


Figure – Capacitor voltage waveform; note that $V_{max} = V_s$ (Source: https://www.webassign.net/question_assets/unccolphysemI1/lab_3/manual.html)