The RC Time Constant (An LTspice Tutorial)

Outline

LTspice is another circuit simulation package different than that at falstad.com/circuit. Why do we need another circuit simulator? Really we don't, but if we were to have only a single circuit simulator, LTspice is the one. We started with falstad's simulator because 1) it had a very shallow learning curve, 2) no installation required and 3) we wanted you to not fear it and put it to use on your accord while doing homework problems! If LTspice had been taught first, there would likely have been much hesitancy to build something with it.

In this tutorial, we will take a practical approach to learning: Build something while learning the features of the tool! To keep it familiar, we will repeat a task we completed previously in falstad, investigating an RC time constant. As such, much of the intro of *this* document is identical to that *previous* exercise. Feel free to skim if you want or read the material again as a refresher. After that brief refresher, we'll have you complete the installation of LTspice while building circuits and answering a few questions.

Timing Mechanisms

Typically, the path for charging a capacitor and the path for discharging the capacitor is current-limited by a resistor, the value of which can be changed to alter the rate at which charging or discharging can occur. This ***timing mechanism*** is exploited in the construction of oscillators that create controlled voltage swings at a given frequency, systems (filters) that respond favorably only to signals that change at particular rates/frequencies, and also electronic timing to delay a response. The calculation of the "time constant" and, even better, "rise time" of many of these devices is very simple, requiring a simple multiplication! Let's learn more…

Prerequisites

- Familiarity with capacitors and their function as energy-storage devices by charge separation.

Parts Needed

- A computer, preferably Windows, but MAC should be workable. We should also have LTspice installed on the lab stations at the University of Illinois which are accessible remotely.

Learning Objectives

- You will be able to **provide** a definition and formula of the RC **time constant** as well as the 10%-to-90% **rise time**.

- You will be able to **predict** how a change in the capacitor or resistor values will alter the time constant of a circuit.

Capacitor Charging and Discharging

When a capacitor is charged by a constant (DC) voltage supply of V_{DD}, the time-domain voltage across the capacitor is given as

$$v(t) = (V_i - V_{DD})\left(e^{-\frac{\mathrm{t}}{RC}}\right) + V_{DD}$$

where C is the capacitance being charged, V_i is the initial voltage on the capacitor at time $t = 0$, and R is the series resistance within the charging path. If the initial voltage is $0\ V$ and the voltage supply is $9\ V$, then this equation simply becomes

$$v(t) = \left(1 - e^{-\frac{\mathrm{t}}{RC}}\right)V_{DD}$$

The voltage across the capacitor would be asymptotically approaching $9\ V$, although *most* of the charging occurs in a short time span <u>on the order of</u> the time constant which is the product $R\ times\ C$.

$$\tau = RC$$

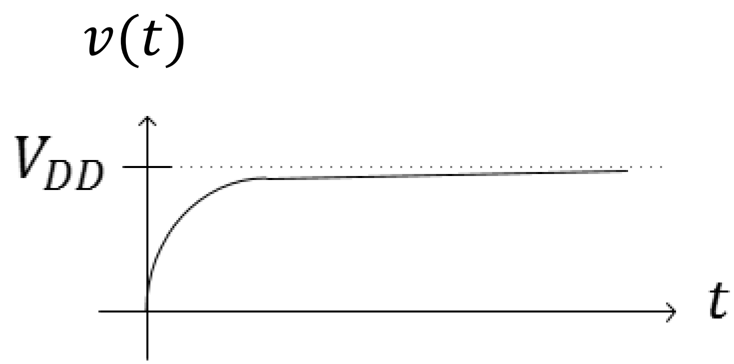


Figure 1*: The waveform* $v(t)$ *across a capacitor while charging to* V_{DD}.

When a capacitor is being discharged to ground voltage (0 V), the time-domain voltage across the capacitor is given by

$$v(t) = V_{start}e^{-\frac{t}{RC}}$$

where we are assuming the voltage across the capacitor is V_{start} at the beginning of the discharge. As before, the decay of the capacitor to 0 volts follows an asymptotic path with much of the decay occurring, again, in the order of magnitude of RC.

Notes:

Equation Reference: ECE210 textbook, page 97, *Analog Signals and Systems* by Kudeki and Munson.

“on the order of”: In engineering, this may also be an expression meaning *on the same “order of magnitude as” something else*. Order of magnitude is usually expressed in powers of 10. For example, if $\tau = 5\ s$, then most of the time it takes to charge is surely contained in the range $[0.5\ s, 50\ s]$.

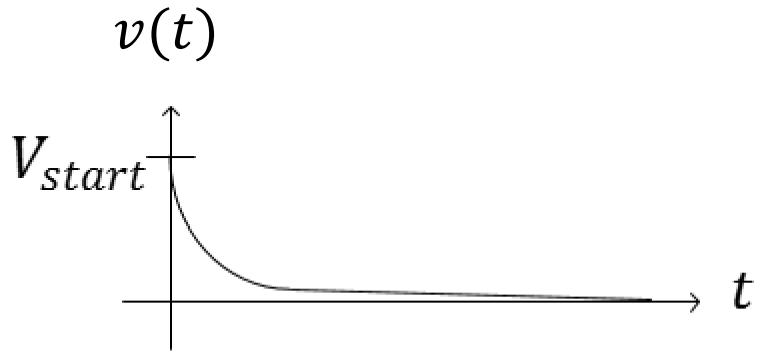
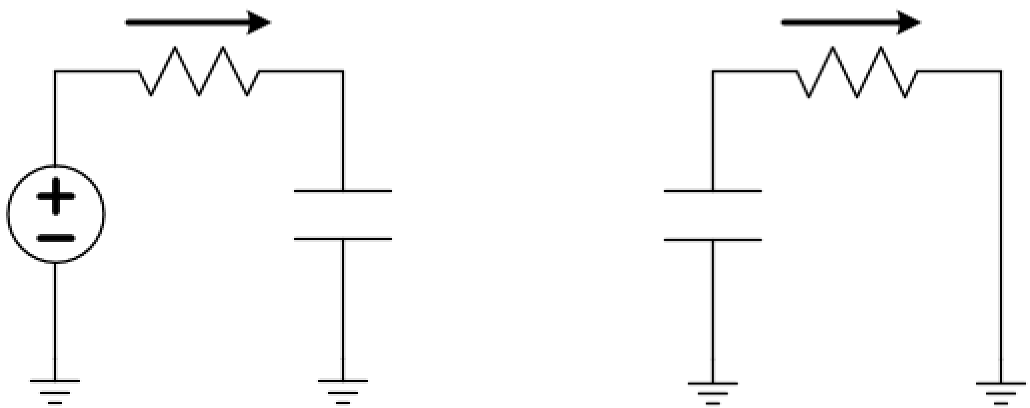


Figure 2*: The waveform $v(t)$ across a capacitor while discharging to ground voltage.*



(a) *(b)*

Figure 3*: Charging (a) and discharging (b) schematics for capacitor. The arrow shows the direction positive-valued current will flow as the capacitor charges and discharges, respectively.*

The time constant, while a very simple equation, is neither easy to remember nor easy to measure. Technically, it is the time an uncharged capacitor would take to charge to 63.8% of its final voltage value. A better metric would be the 10%-to-90% rise time, which, for a capacitor charging from 0 to V_{DD}, it would be the time between the point the voltage reaches 10% of V_{DD} until it reaches 90% of V_{DD}. Given a value for the capacitor, C, and the resistor, R, for Figure 3 (a), the capacitor will have a rise time given by

Notes:

$$\boxed{t_{rise} \approx 2.2\, RC}$$

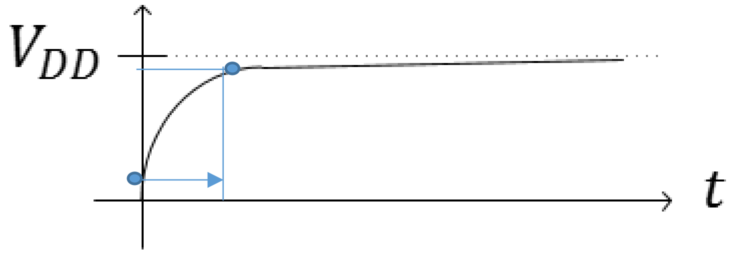
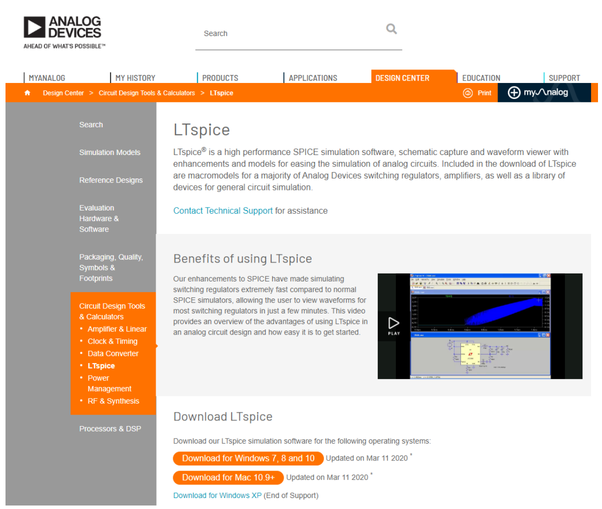


***Figure 4**: The 10%-to-90% rise time.*

LTspice

> **Note:** This guide is specific to the Windows 10 OS. For Mac users, please see https://www.youtube.com/watch?v=MsilYqaGPQw&feature=youtu.be and similar guides. Alternatively, the University's Windows based EWS machines can be accessed remotely via Citrix https://it.engineering.illinois.edu/ews/lab-information/remote-connections/connecting-citrix.

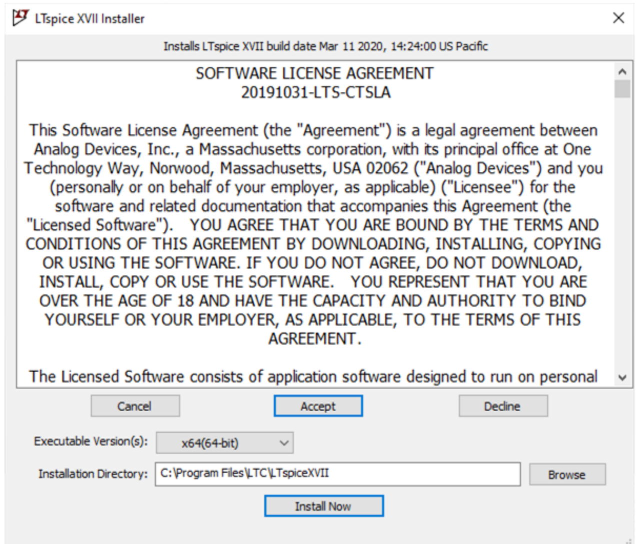
Find the LTspice downloads at https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html. Select appropriate platform (Windows or Mac). Download the LTspice installer. The current version of LTspice, as of September 16, 2020, is XVII.



Notes:

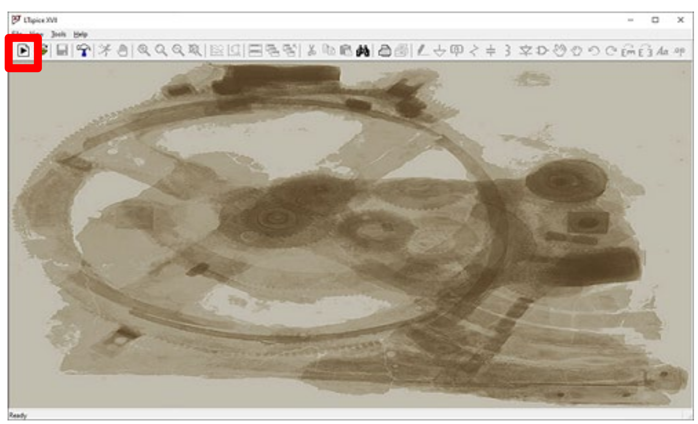
Notes:

Double click on the installer file to begin the installation. After reading through the license agreement, click “Accept”. Choose an appropriate directory and click “Install Now”. After installation is complete, the software should automatically open. If not, find the LTspice XVII executable and open it.

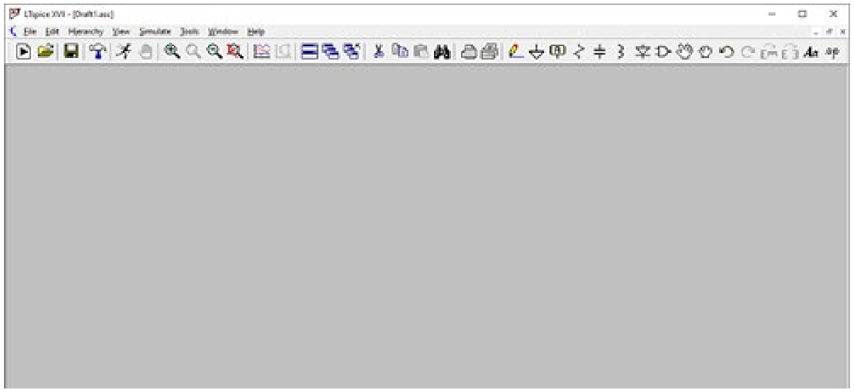


The Main Interface

Double click on the executable for LTspice XVII. The following window should appear with most options “grayed out”. Begin by creating an empty draft. To create a new schematic to add components to, click on the symbol. Alternatively, you may see as the new schematic symbol.



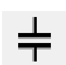
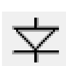
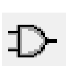
Some of the main icons will now be enabled.



This is where you will build a circuit schematic. The schematic can be saved as a *.asc file. Go ahead and call it RC_time_constant.asc and save it now.

Adding and removing components

Below are some more useful icons on the interface. Components such as resistors and capacitors can be added by clicking on the appropriate icon (for shortcuts, see the list after clicking on the **Edit** tab). Once clicked, hover to the area you wish to place the component. Left click to place the component. Once clicked, you can press **right click** or press **Esc** to exit. Use the mouse wheel or the microscope/zoom icons to zoom in or out.

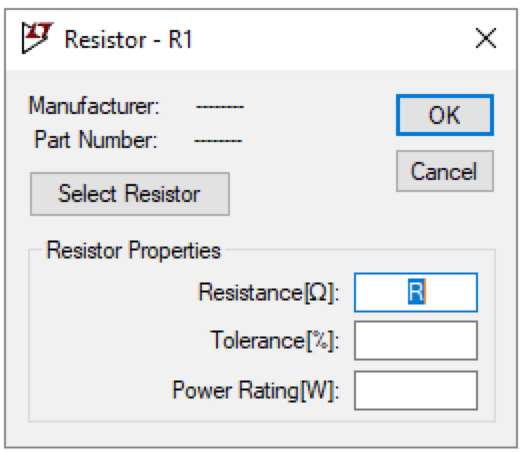
Icon	Description
	Add resistor.
	Add capacitor.
	Add diode.
	Add custom component, e.g. voltage source, transistors, ICs, etc.
	Delete component. Left click on component to remove, or left click and drag making a box to delete whole portions of your circuit.

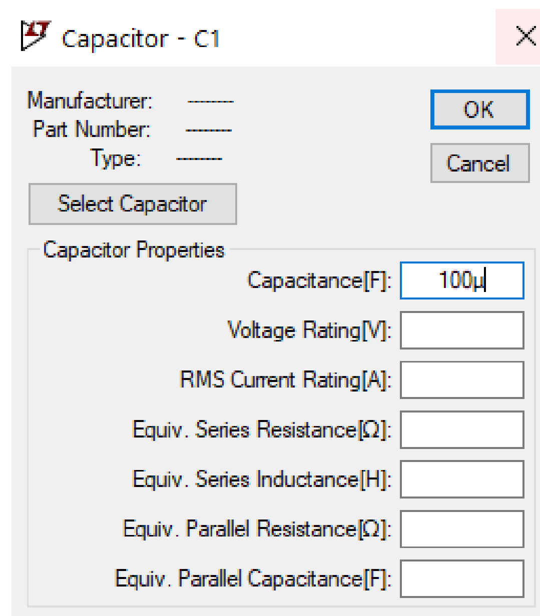
Notes:

Editing Component Properties

Right click on any component to modify its properties. By default, most devices will have a letter and not a value inserted that you will need to modify using a quasi-SI notation. The letters “k”, “m”, “u”, “n”, etc. can be used as multiplication factors. For example, you would enter “**100**” in the “Resistance” field to represent 100 Ω or “**100u**” for a capacitance of 100 µF.

Suffix (case insensitive)	Power
T	10^{12}
G	10^{9}
Meg	10^{6}
k	10^{3}
m	10^{-3}
u	10^{-6}
n	10^{-9}
p	10^{-12}
f	10^{-15}



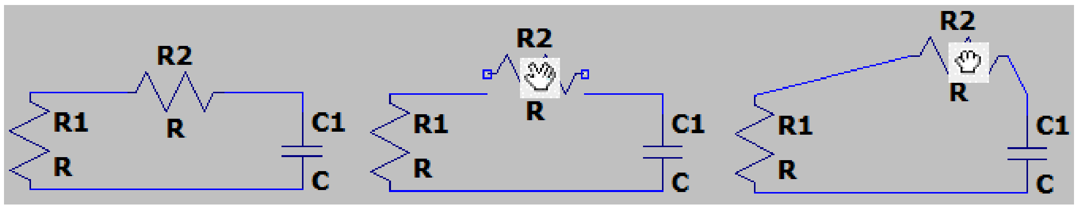


Notes:

Moving components

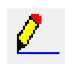
- To **move** a component, click on the open hand, , then left click on a component to move it. Click and drag to select an entire area. The closed hand, , drags the node or “wires” along with the components and is useful when moving a component along a wire rather than perpendicular to the wire on which it resides. You can also move labels as was done on the circuit above.

- To rotate or mirror a component, ensure it is selected by using . Press **CTRL+R** to rotate the component or **CTRL+E** to mirror the component.



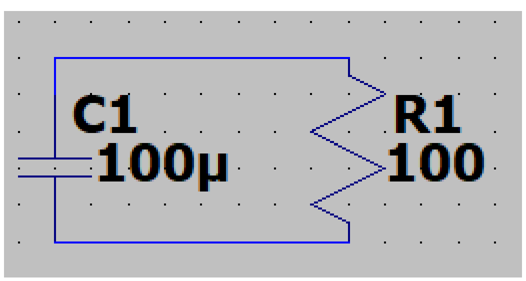
- To delete a component, press **Delete** or select the scissors, . Left click on a component to delete it, or click and drag to form a box and delete an entire area.

Adding Wire

Establish wires or nodal connections by using the pencil, . Continuously left click to add straight lines and establish “wire” connections. Right click to end.

Build an RC Circuit

Build the circuit shown below with the values as given.



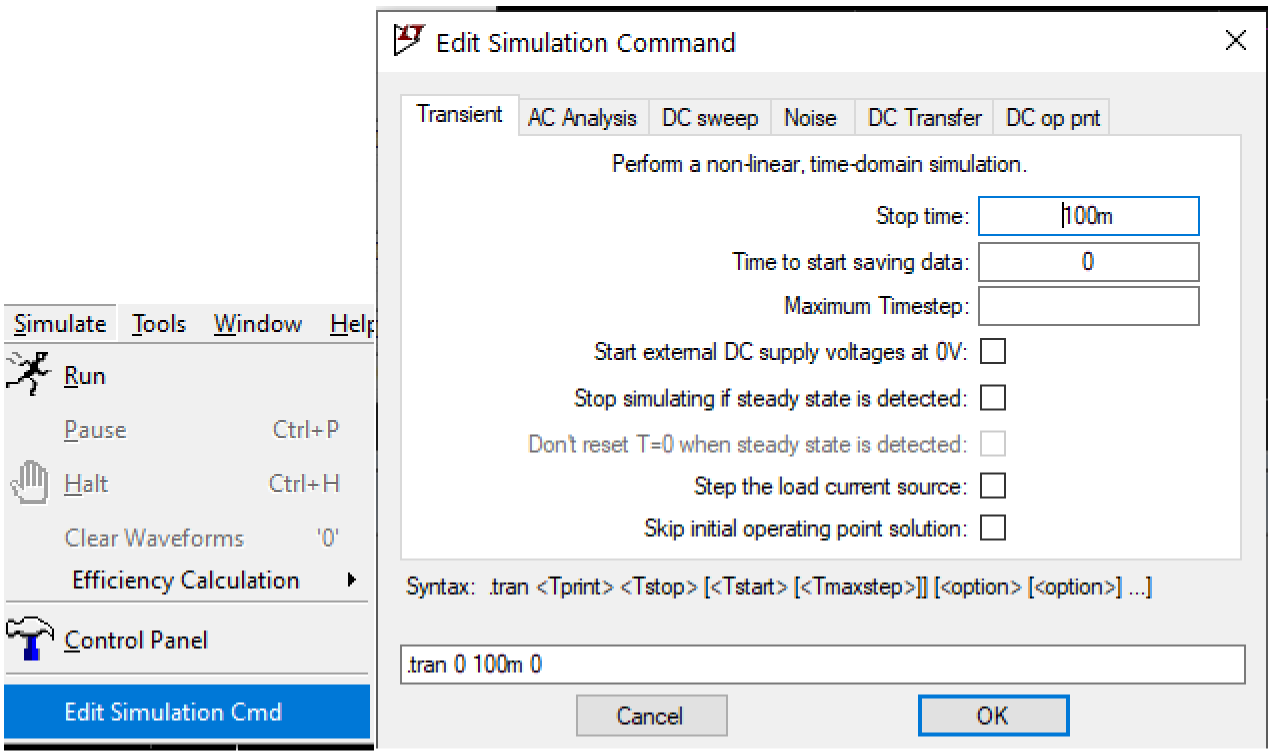
Notes:

Notes:

Use the mouse wheel or the microscope/zoom icons to zoom in or out on your view. Left-click on the schematic background to drag the schematic around on the page.

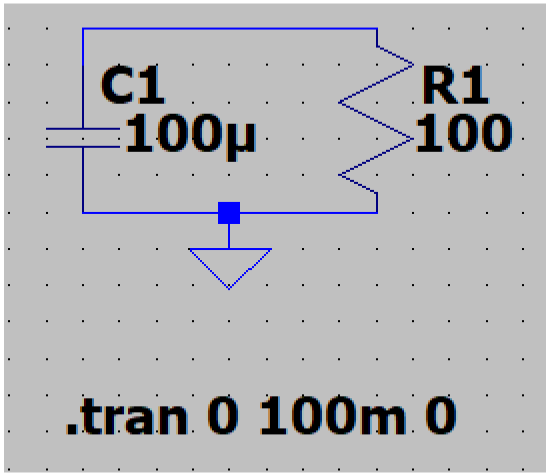
It would be interesting to simulate this circuit, but two more things must be done first. 1) We need to define *how* the circuit is simulated. We can actually do a lot more than just the traditional DC simulation. 2) We need to place a *ground* node  on the circuit somewhere. The ground node is needed to allow us to define voltages as though they were "single-ended" measurements. That is, that they can be measured using a single-wire probe instead of measured differentially.

To define the simulation mode, go to Simulate => Edit Simulation Cmd. In the popup window, choose the simulation to stop after 100 ms and allow it to start "saving data" at the start of the simulation. When you click "OK", your cursor will display the text from this window. You must still insert this command on your circuit schematic somewhere by left clicking to anchor it.

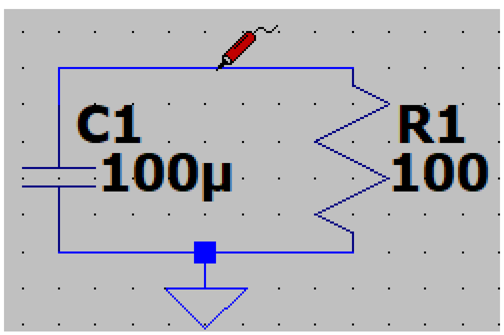


Notes:

Also place a ground node so that your circuit now looks like that below. Save your circuit schematic again (under the same name as before) by clicking CTRL-s.



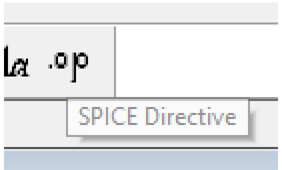
Press the Run Simulation icon, . The simulation will run and an empty plot window will appear. With the simulation window selected (click on it if needed), hover over the top node and a single-wire measurement “probe” will appear. Left-click to choose to display the voltage at that node (relative to the voltage measurement’s “black probe” which is always at the ground node).



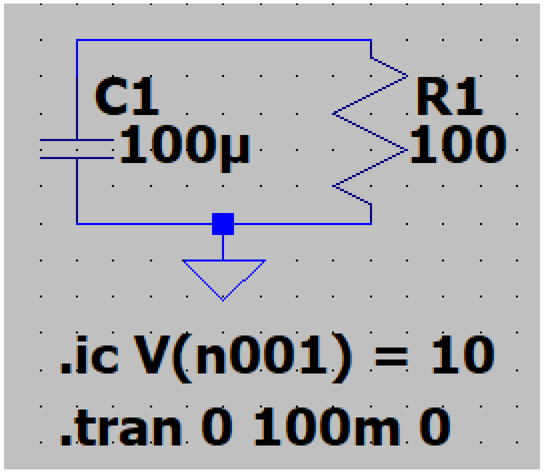
The plot of this voltage vs. time will appear on the plot window. There should be a title that says something like V(n001), meaning that the voltage for the node with the default label n001 is being displayed in that color. Our plot, however, is uninteresting. There was no voltage source and the capacitor was likely uncharged at the start of the simulation.

Notes:

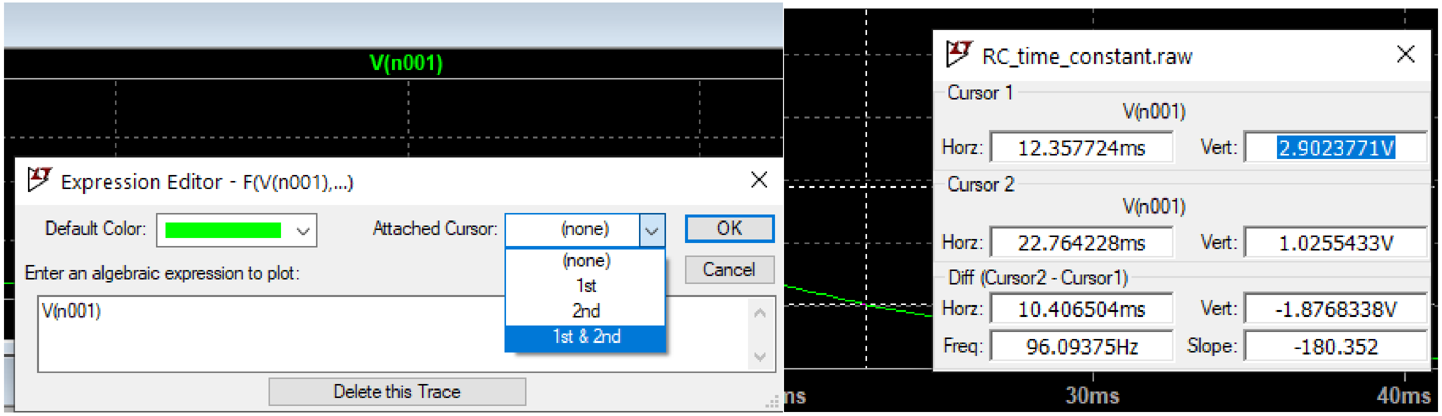
Let's change the initial conditions of the capacitor so that the simulator assumes it was charged already. To set initial conditions, you need to make the schematic window active and select the SPICE Directive tool labeled .op.



In the popup window, enter **.ic V(n001) = 10** which will set the initial condition (**.ic**) of the voltage (**V**) at node **001** to a value of **10** volts. Don't forget that little period at the beginning! You will need to left-click to anchor this command on your schematic.

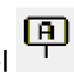


You are now ready to simulate again. This time, your capacitor will have been assumed charged to 10 volts in the simulation at the start time and will discharge through the 100 Ohm resistor. This should be clear in the plot window. To measure the fall time, you can add two cursors to your plot. To do so, right-click on the name in the plot title and choose Attached Cursor: 1^{st} and 2^{nd}. Two cursors will be added to your plot. Hover over each cursor and left-click to drag them to the 10% and 90% points on the voltage plot. The cursors will also provide a window where you can see their locations as well as the difference between those locations. If you need to adjust the scale of the voltage, right-click on the y-axis label and set the top and bottom voltages as well as the "tick" or spacing between the horizonatal scale markers. You can similarly change the scale on the time axis.



Important Message: All questions are repeated at the end of this document so that you can scan and submit fewer pages to GradeScope.

Question 1: Record the cursor data and calculate t_{fall}.

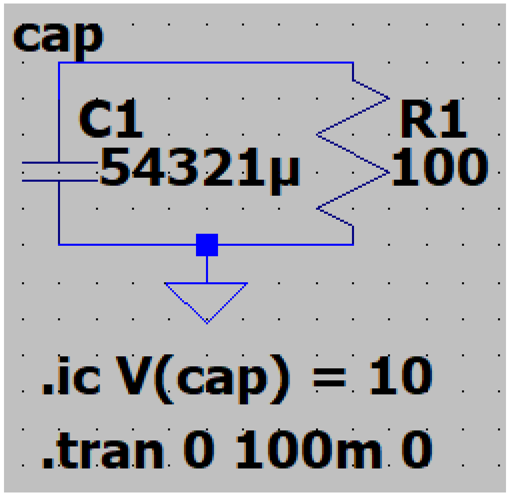
The node name n001 is very non-descriptive. To make this better, add a node label  to the node at the top of the capacitor and call it **cap**. Because you have over-ridden the original node name, you will also need to edit the initial condition statement.

Interesting Effect: By defining a node label, that same node label can be used in multiple locations in a circuit and those nodes will be, in essence, all joined by an invisible wire…kind of like how power rails are hidden inside the breadboard! This is useful to keep diagrams clean of excess wires, but cause accidental shorts if you use cut-and-paste while building a circuit in the simulator and you fail to redefine a copied node label.

Notes:

Notes:

Change the value of the capacitor to the last ***five*** digits of your UIN, in μF. Your circuit schematic should be very similar to the figure below.



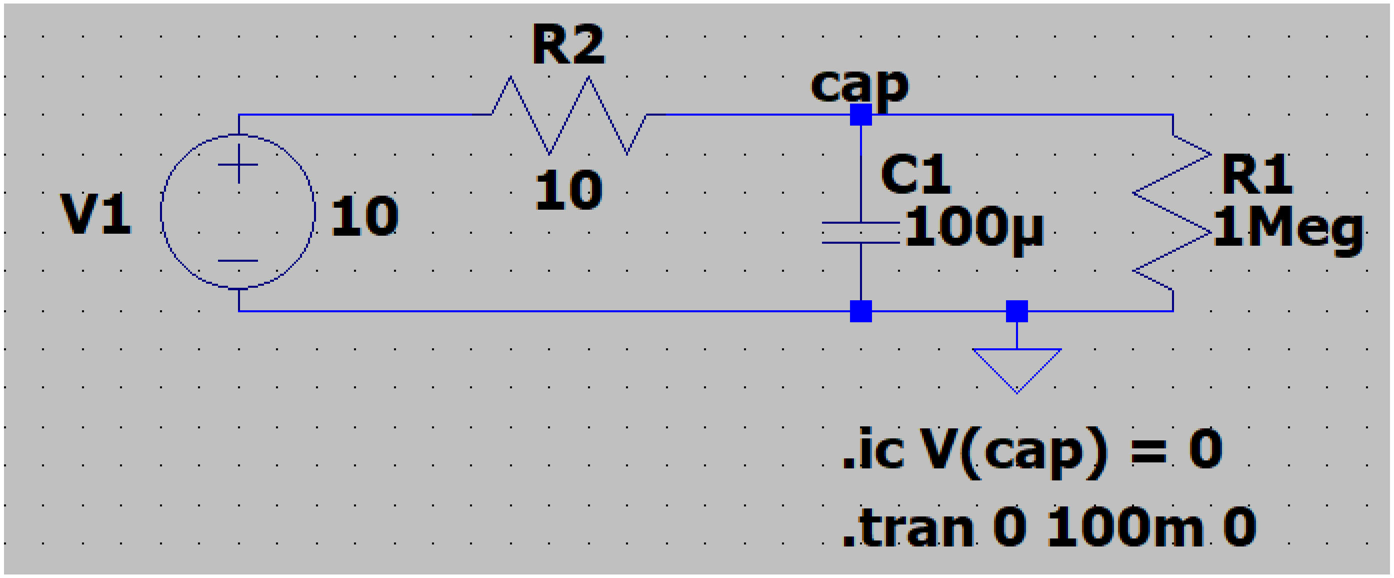
Question 2: Predict how long your simulation should be adjusted to run to easily display the fall time of the capacitor. Explain your prediction using what you know about time constants.

Run the simulation. Note that you will need to probe the **cap** node to display the voltage vs. time plot because you renamed that node.

Question 3: Again, calculate t_{fall}. Make adjustments to your simulation time and plot parameters as needed to accommodate this measurement.

Question 4: Collect a screenshot of your final plot with the cursors correctly set and the cursor window visibly showing the time difference representing t_{fall}. Include this plot with this list of answered questions in your submission for this assignment.

Finally, let's add a voltage source (set to 10 volts) to the circuit and charge the capacitor (reset to 100 μF) through a 10 Ohm resistor. We will not remove the discharging path, but, instead, change the value of the discharging resistor to a very large value (1 MOhm).

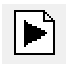
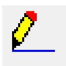
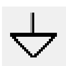
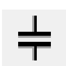
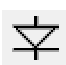
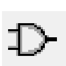


Simulate this circuit and view V(cap).

Question 5: Explain, using concepts from ECE 110, why the discharge path has little-to-no effect on t_{rise}.

Notes:

Quick Reference

Icon	**Description**
	Run simulation. Prior to running, you will need to specify some simulation parameters through **Simulate** > **Edit Simulation Cmd**.
	Create new schematic (.asc file).
	Establish wires or nodal connections. Continuously left click to add straight lines and establish "wire" connections. Right click to end.
	Add ground node. Think of this node as hosting the negative probe of any voltages you wish to measure as you move the positive (+) probe of your voltmeter/oscilloscope around the circuit.
	Label another node or point in circuit. This is useful in place of wire connections in complex circuits as any two nodes with the same name are actually connected by an "invisible" wire! It is also useful for plotting the voltage between that node and the ground node **by name**.
	Add resistor.
	Add capacitor.
	Add diode.
	Add custom component, e.g. voltage source, transistors, ICs, etc.
	Delete component. Left click on component to remove, or left click and drag making a box to delete whole portions of your circuit.

Notes:

Name:____________________ UIN: | | | | | | | | | | Section AB/BB: | |

Notes:

Question 1: Record the cursor data and calculate t_{fall}.

Question 2: Predict how long your simulation should be adjusted to run to easily display the fall time of the capacitor. Explain your prediction using what you know about time constants.

Question 3: Again, calculate t_{fall}. Make adjustments to your simulation time and plot parameters as needed to accommodate this measurement.

Notes:

Question 4: Collect a screenshot of your final plot with the cursors correctly set and the cursor window visibly showing the time difference representing t_{fall}. Include this plot with this list of answered questions in your submission for this assignment.

Question 5: Explain, using concepts from ECE 110, why the discharge path has little-to-no effect on t_{rise}.