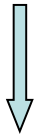
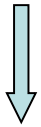
Transient Analysis using LTSpice

Why LTSpice ?



1. It is **free**, Developed by Linear Technology
2. Fast
3. Easy to learn



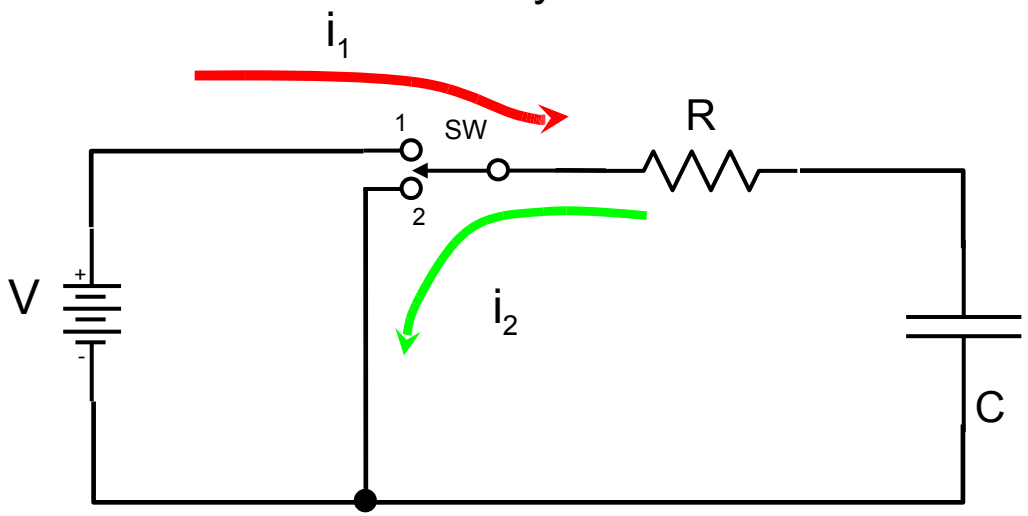
Download for free from the following link:

http://www.linear.com/designtools/software/?gclid=CI2Gg9uasLsCFQuUfgodpwQAJQ#LTspice

Also download the user's guide and go through it briefly

Sajjad Haidar

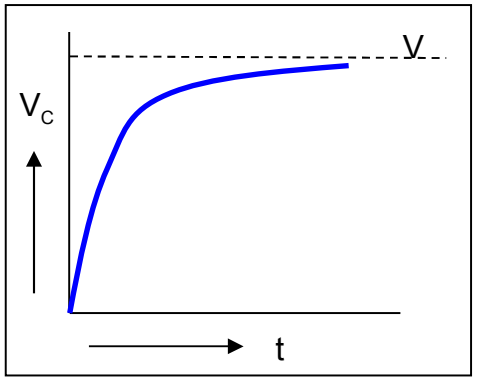
Transient Analysis: RC Circuit

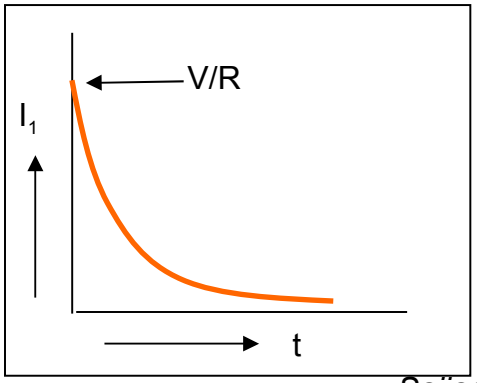


When switch in **Position:1**
Capacitor voltage ⇨ $V_C = V\left(1 - e^{-\frac{t}{RC}}\right)$

Current ⇨ $i_1 = \frac{V}{R} e^{-\frac{t}{RC}}$

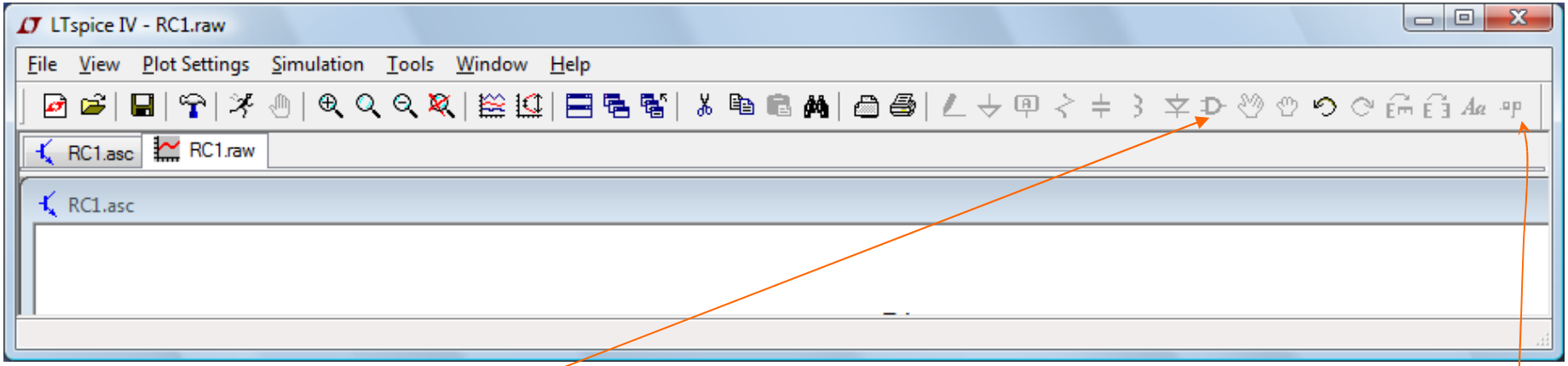
Position:2 ⇨ $i_2 = \frac{V_C}{R} e^{-\frac{t}{RC}}$





Sajjad Haidar

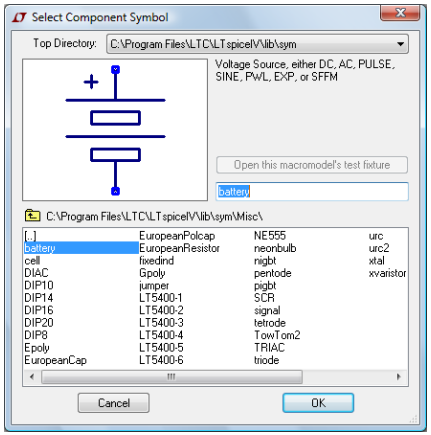
LT SPICE Simulation: Adding components

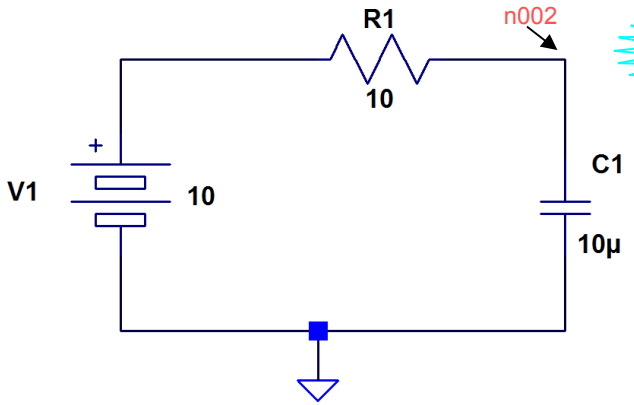


Add battery clicking on Component tool

Add resistor and capacitor

misc: battery

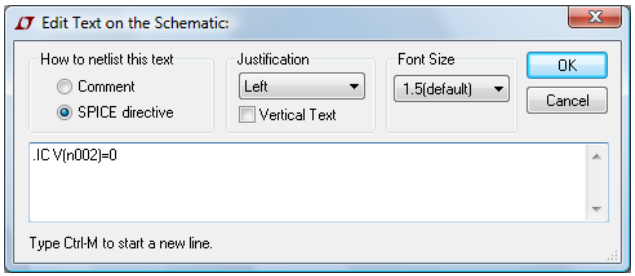




Add Spice directive

.IC V(n002)=0

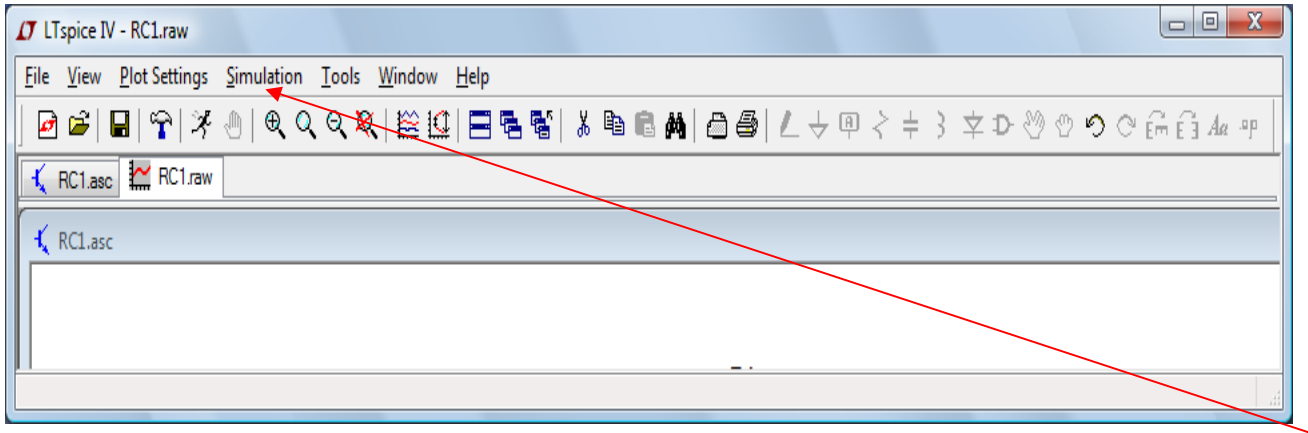
.tran .5ms



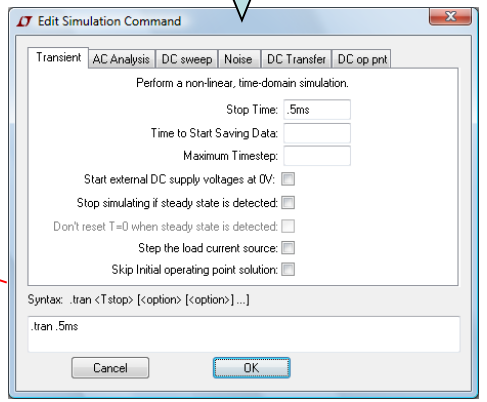
After drawing the schematic, touching
The cursor each wire will tell you the
Node number on the bottom left corner

Sajjad Haidar

Run: Simulation



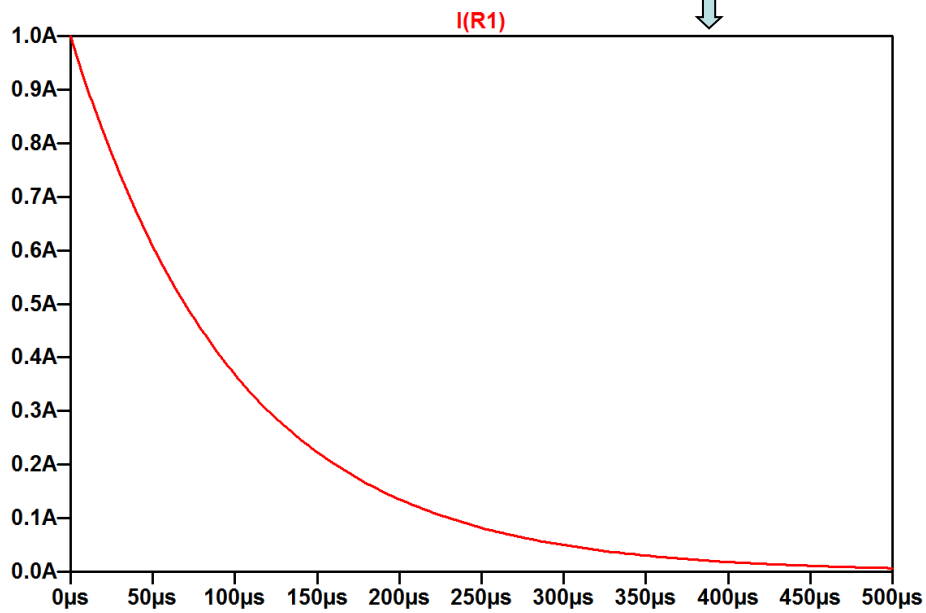
Run simulation
Set the stop time



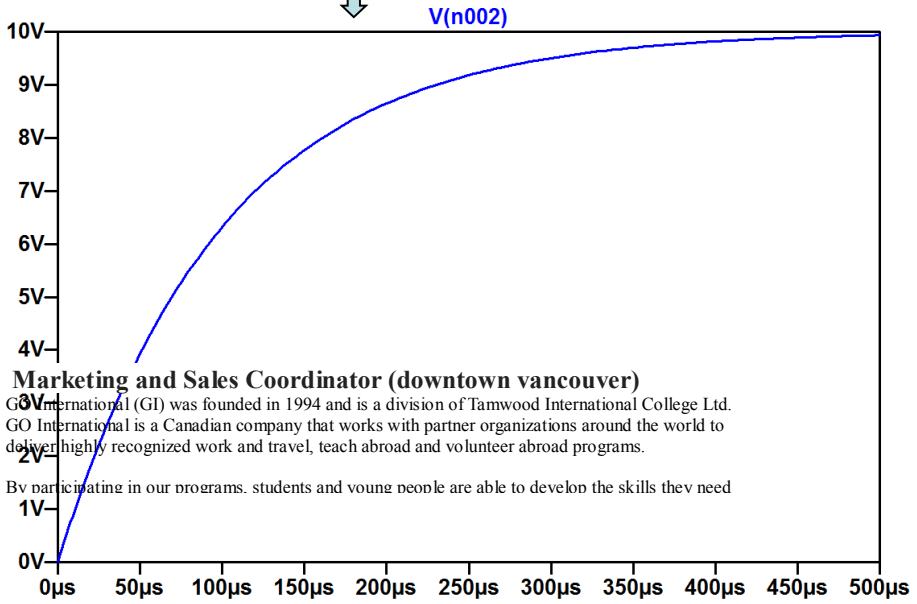
Important Note: You must add the initial condition: Spice Directive
.IC (n002)=0. it means there is no charge in the capacitor at t = 0

Press run button. And place voltage probe

Current probe



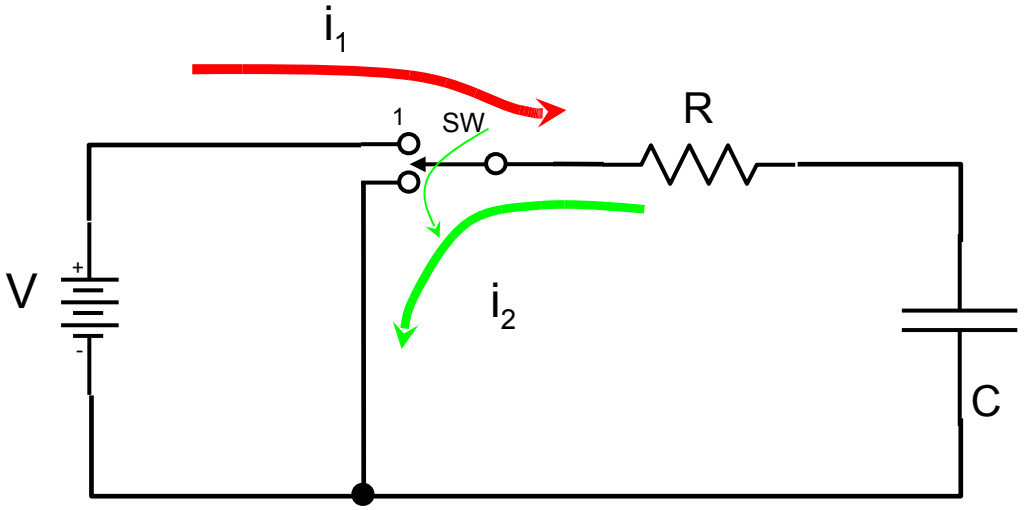
Current through resistor



Voltage across capacitor

Sajjad Haidar

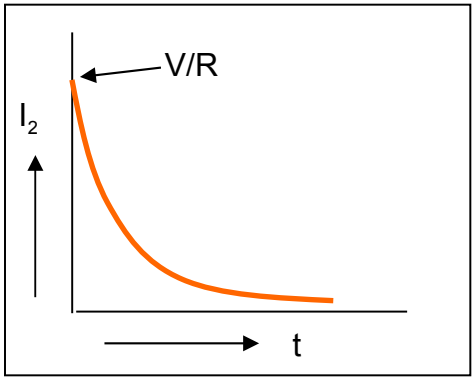
Switch ⇨ Position 2: Capacitor discharging



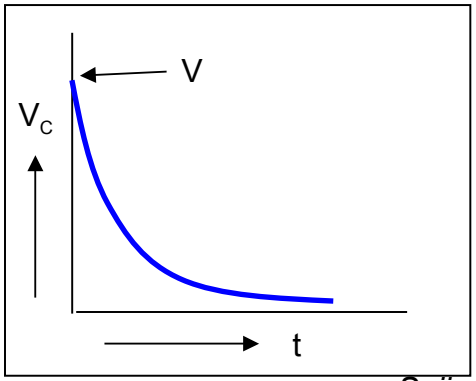
When switch in position in 1 for a long time, $V_C \sim V$

$V_{C(initial)} \Rightarrow V$

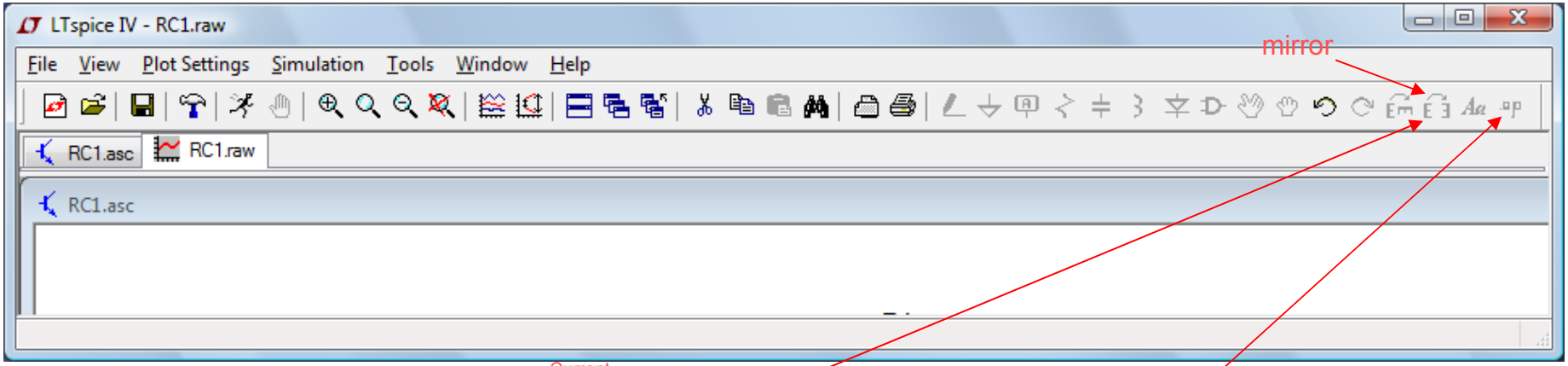
$$i_2 = \frac{V_{C(initial)}}{R} e^{-\frac{t}{RC}}$$



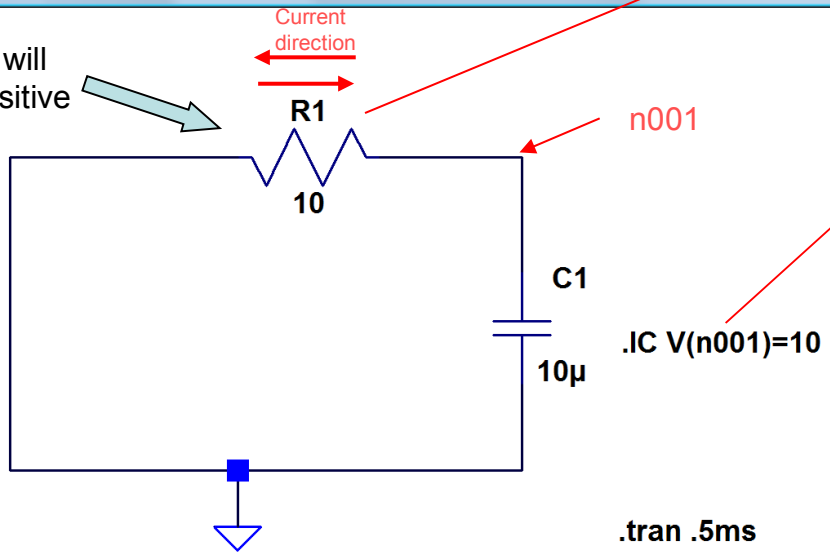
$$V_C = V_{C(initial)} e^{-\frac{t}{RC}}$$



LT SPICE Simulation: Capacitor Discharging



Mirror resistor R1, which will give you the current in positive direction

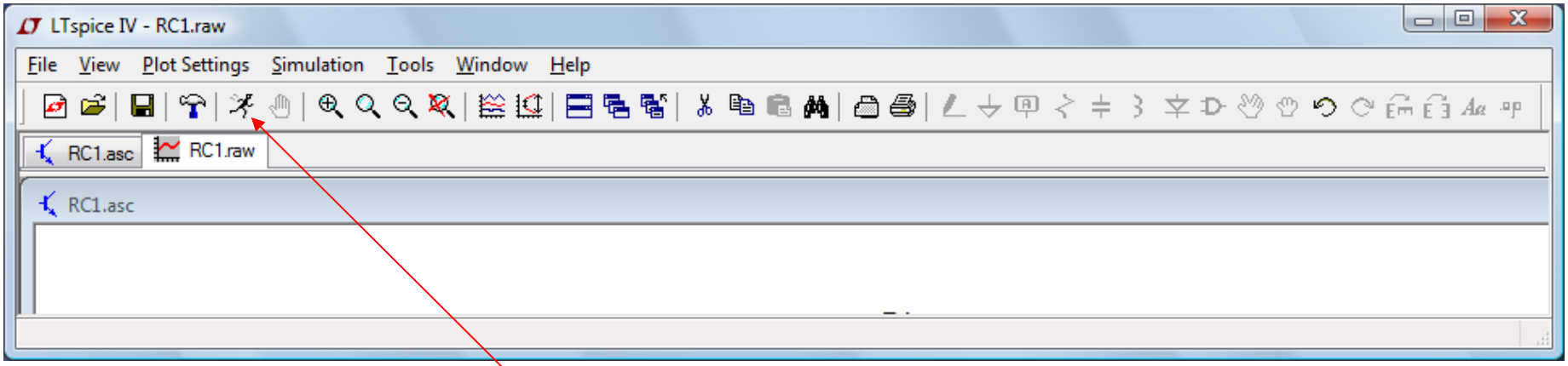


Set the Spice directive :
.IC V(n001)=10
As the capacitor voltage is now 10 V

Removing the battery
Sets node: n001 at the capacitor

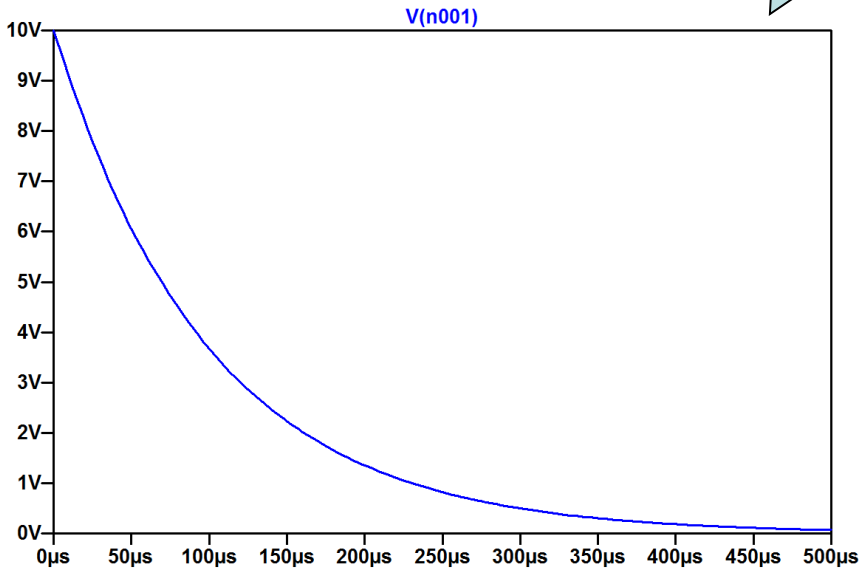
Sajjad Haidar

LT SPICE Simulation: Capacitor Discharging

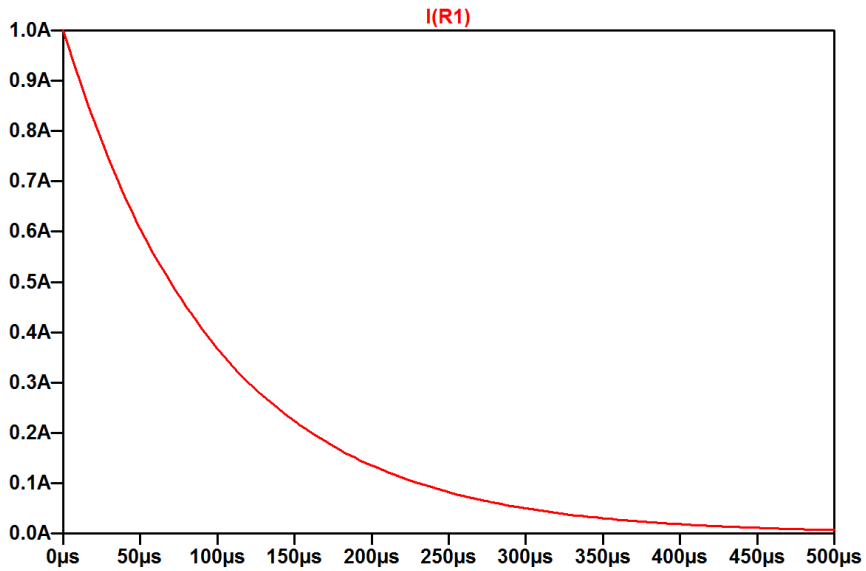


Run and click current and Voltage probe

Capacitor voltage

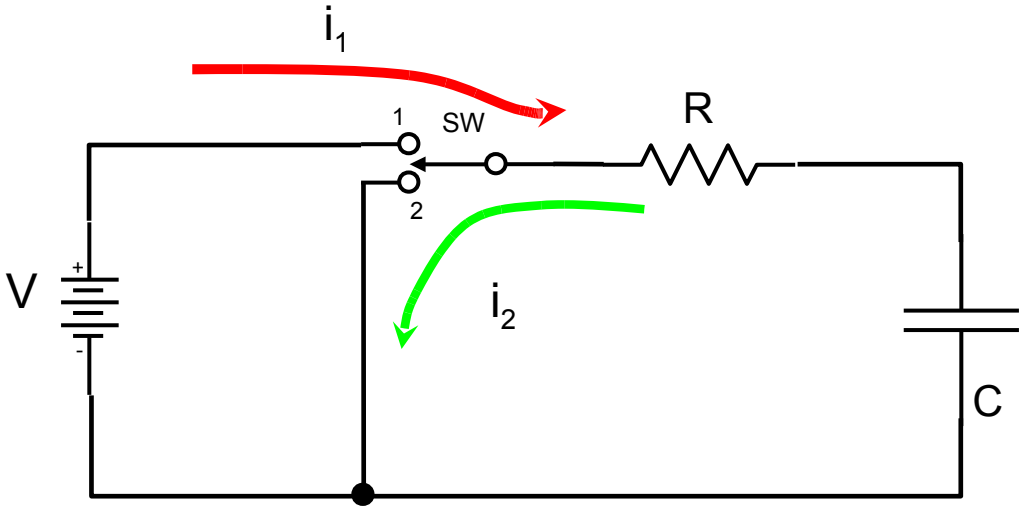


Discharging current



Sajjad Haidar

Calculating Power and Energy



When switch in position 1 for a long time, or in steady state capacitor voltage becomes equal to battery voltage. Energy Stored in capacitor can be expressed as:

$$E = \frac{1}{2} C V^2$$

When the switch in position 2, power dissipated by the resistor, R is:

$$P = i_2^2 R = \left(\frac{V}{R} e^{-t/RC} \right)^2 R = \frac{V^2}{R} e^{-2t/RC}$$

Total energy consumed by R is:

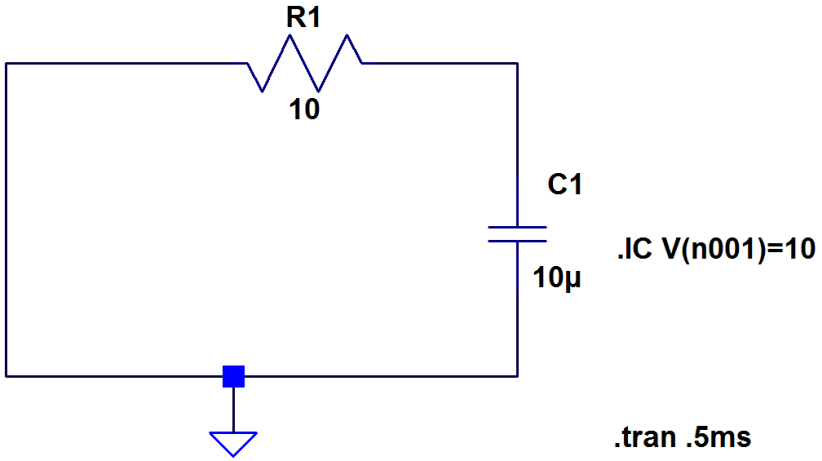
$$E = \int p.dt = \frac{V^2}{R} \int_0^\infty e^{-2t/RC} .dt$$

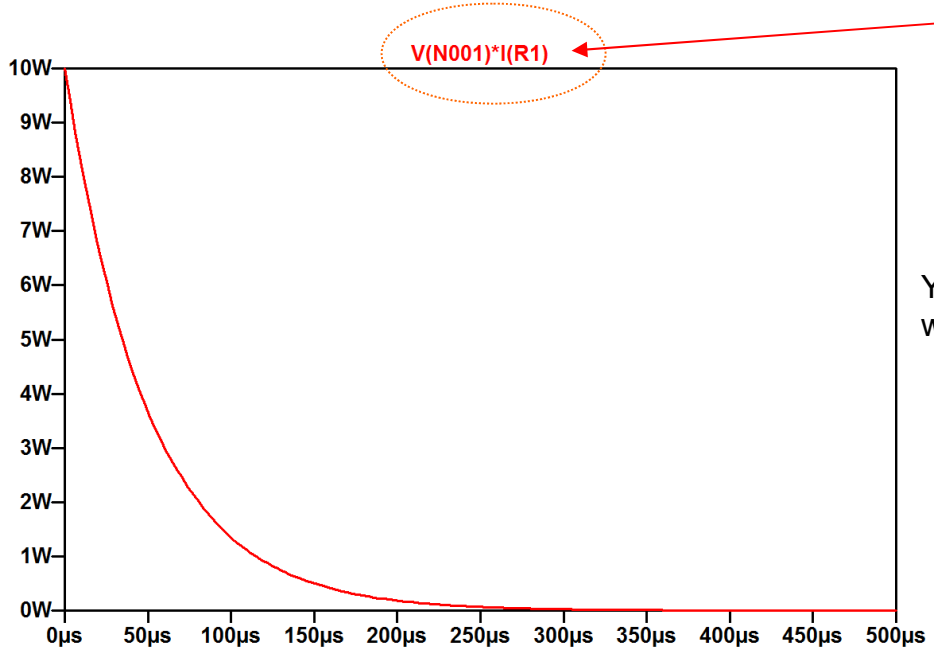
Sajjad Haidar

LTSpice to find Power and energy

Press down **ALT** and put the cursor on R1 (You will See a thermometer icon) and click

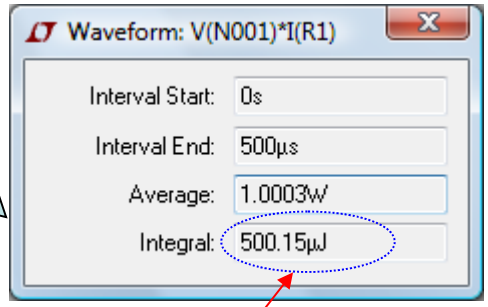
You will get the power data as shown





Press down CTRL and place the cursor on V(N001)*R1 as shown and click

You will get the window like this



Energy

Sajjad Haidar