BRAC UNIVERSITY
DEPT. OF COMPUTER SCIENCE AND ENGINEERING
COURSE NO.: CSE250
Circuits and Electronics Laboratory

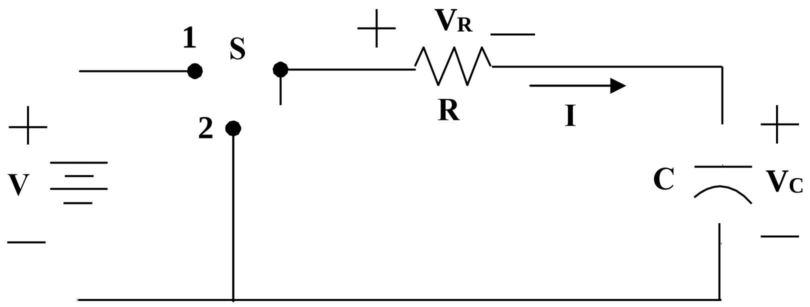
EXPERIMENT NO.: 5
(SIMULATION)

Name of the Experiment: Study of the transient behavior of RC circuit.

OBJECTIVE:
The objective of this experiment is to study the Transient Response of the RC circuit with step Input. In this experiment, we shall apply a square wave input to an RC circuit separately and observe the respective wave shapes and determine the time constants.

THEORY:
The transient response is the temporary response that results from a switching operation and disappears with time. The steady-state response is that which exists after a long time following any switching operation.

Let us consider an RC circuit shown in the figure.



CHARGING PHASE:
When the switch is connected to position 1, applying KVL we can write

$$C = Ri + \frac{1}{C}\int idt \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \ (1)$$

If the capacitor is **initially uncharged**, the solution of equation (1) is----

$$i = \frac{V}{R} e^{-\frac{t}{\tau}} \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots.. \ (2)$$

Therefore, the voltage across the resistor and capacitor is given by

$$V_R = V\, e^{-\frac{t}{\tau}} \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots.. \ (3)$$

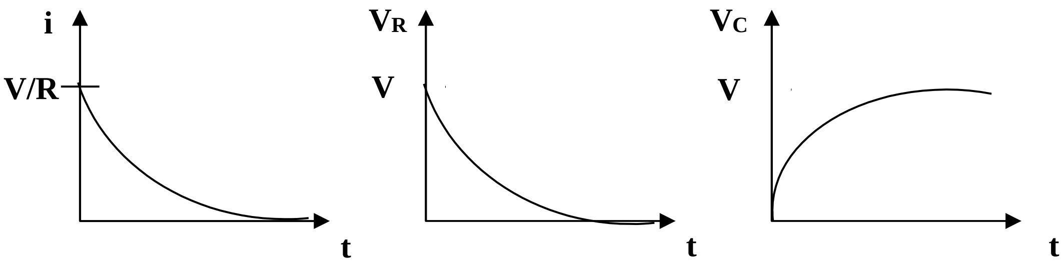
$$V_C = V - V_R = V\,(1 - e^{-\frac{t}{\tau}}) \ \ldots \ (4)$$

Where **τ = RC** and is called the time constant of the circuit.

The time constant is the time at which the response decays to 36.8% of its initial value.

At any rate, **the response decays to 1% of its initial value (i.e. reaches a steady state) after 5τ.**

Equations (2), (3) & (4) are plotted below:



It is seen from the curves that the voltage across the capacitor rises from zero to V volts exponentially and the charging current is maximum at the start i.e. when C is uncharged, then it decreases exponentially and finally ceases to zero when the capacitor voltage becomes V.

DISCHARGING PHASE:

When the switch is connected to position 2, applying KVL we can write

$$0 = Ri + \frac{1}{C}\int i dt \text{ ………………….. (5)}$$

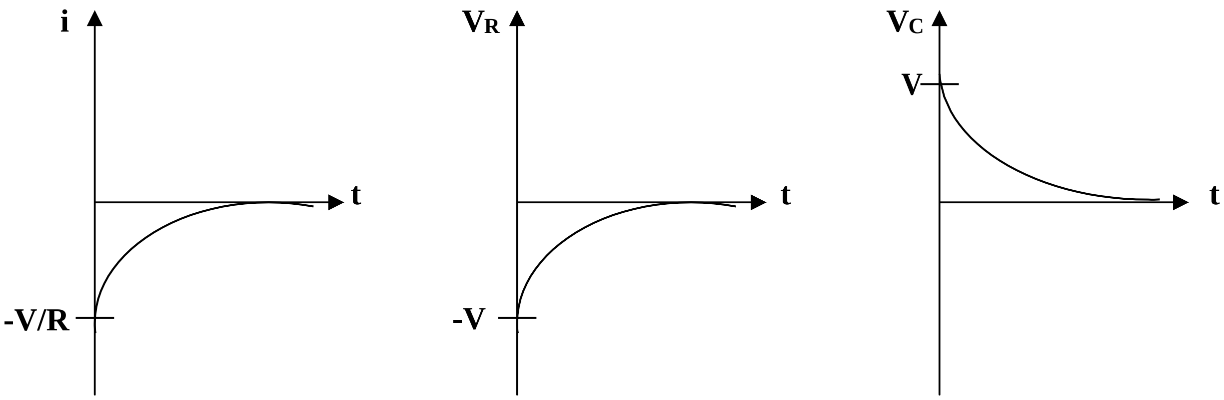
Since the voltage across the capacitor is now V, the solution of equation (5) is
Therefore the voltage across the resistor and capacitor is given by

$$i = -\frac{V}{R}e^{-\frac{t}{\tau}} \text{ ……………………….. (6)}$$

$$V_R = -V e^{-\frac{t}{\tau}} \text{ ………………….. (7)}$$

$$V_C = V e^{-\frac{t}{\tau}} \text{) ………………… (8)}$$

Equation (6), (7) & (8) are plotted below:



It is seen from the curves that the voltage across the capacitor falls from V to zero volts exponentially. The charging current is maximum at the start i.e. when the switch is just thrown to position 2, then it decreases exponentially and finally ceases to zero when the capacitor voltage becomes zero.

EXPERIMENTAL SETUP:

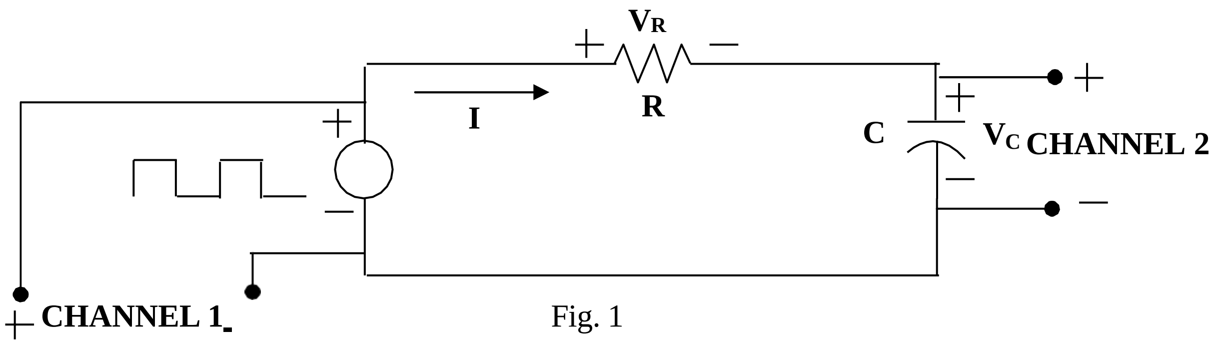


Fig. 1

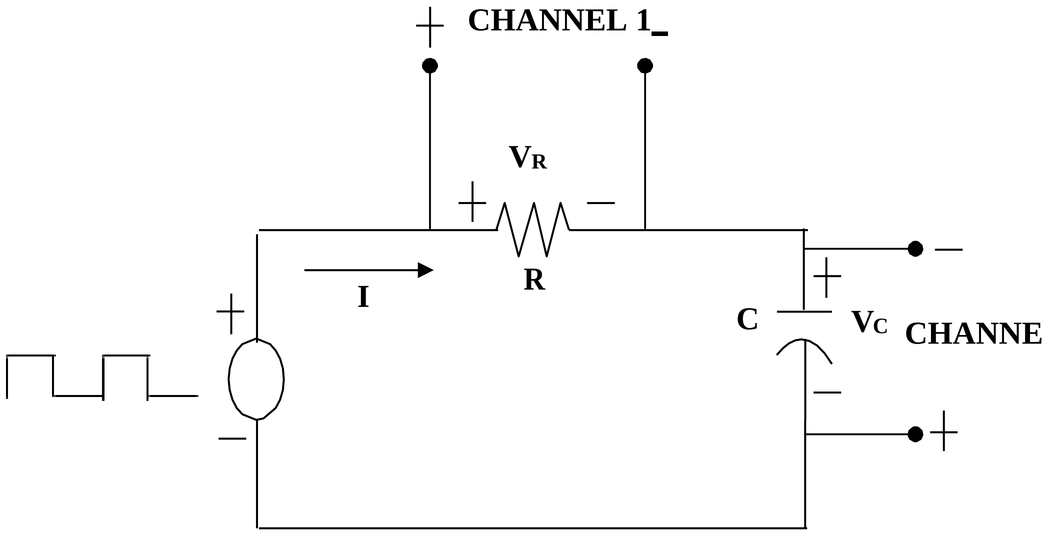
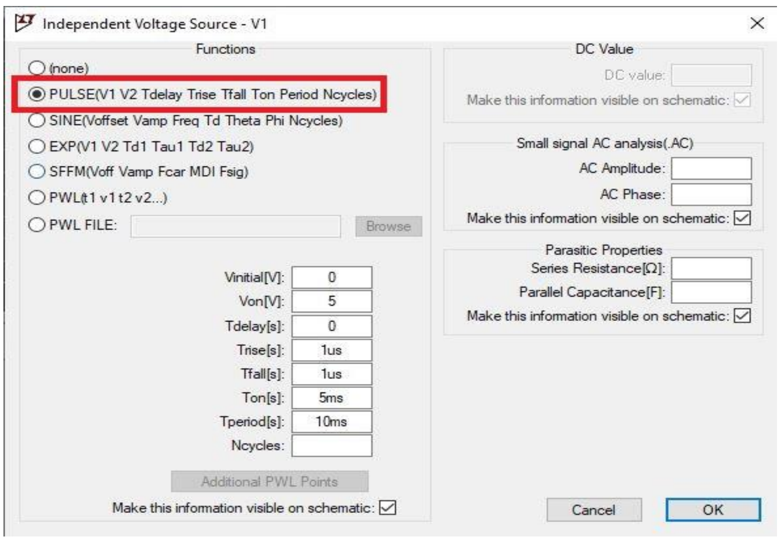


Fig. 2

CIRCUIT SIMULATION USING LTspice:

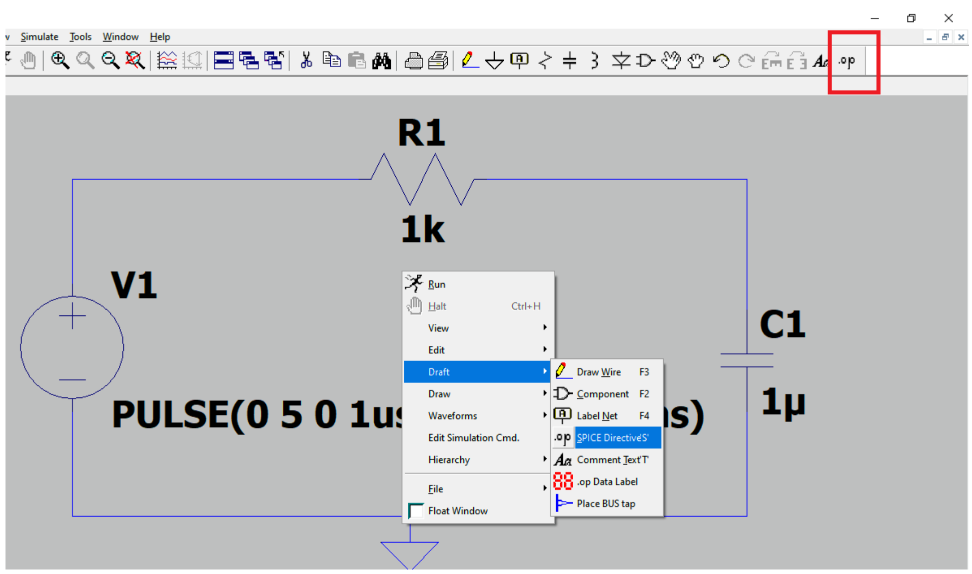
1. Draw the circuit shown in Fig. 1 in a new schematic in LTspice. Modify the components with their values and name the nodes. To name the nodes, ***Right-click on the wire/node → Label Net.* Do not forget to add a ground to the circuit.**

2. To modify the voltage source as a DC square wave voltage source, ***Right-click on the voltage source → Select Advanced → insert the values as below and click OK-*** for a 5V 100 Hz DC square wave voltage source.



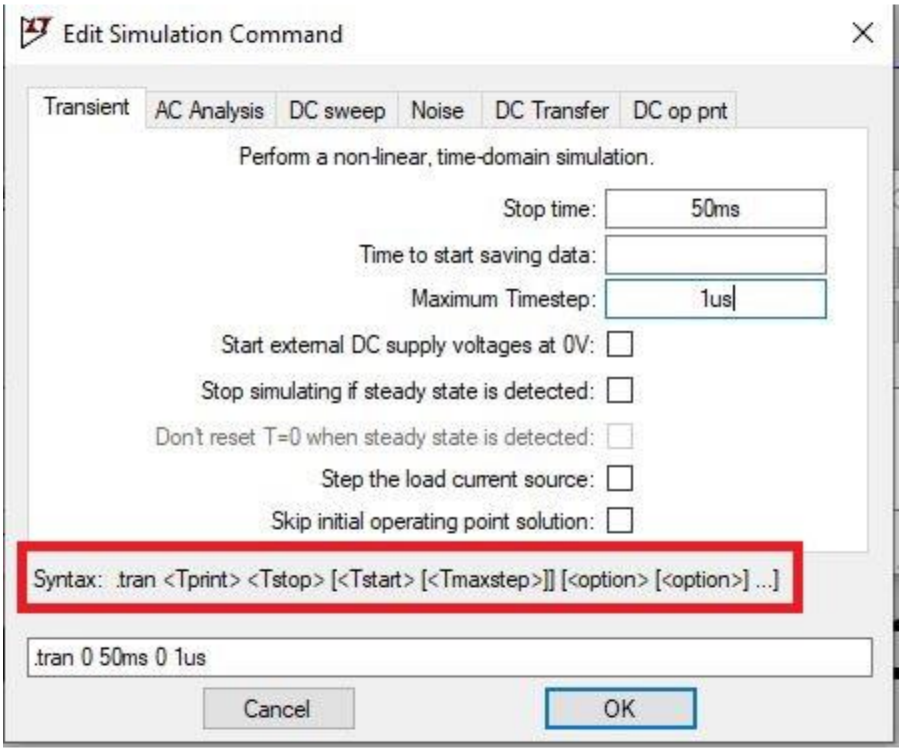
3. To see the responses we have to do ***'Transient Analysis'***. The transient analysis calculates a circuit's response over a period of time.

   To run the transient analysis we have to write the analysis command. Find ***the 'Spice Directives'*** option by ***Right-clicking on the schematic → Draft → Spice Directives or find it from the toolbar above.***



4. After clicking the ***'Spice Directives'***, the ***'Edit Text on the Schematic'*** window will appear. Now ***Right-click on the blank space on this window → Select 'Help me Edit' → Analysis Command.*** A window titled ***'Edit Simulation Command'*** will

appear. Insert values in the boxes as below and click OK. It will generate a transient analysis command. Place the command somewhere on the schematic.

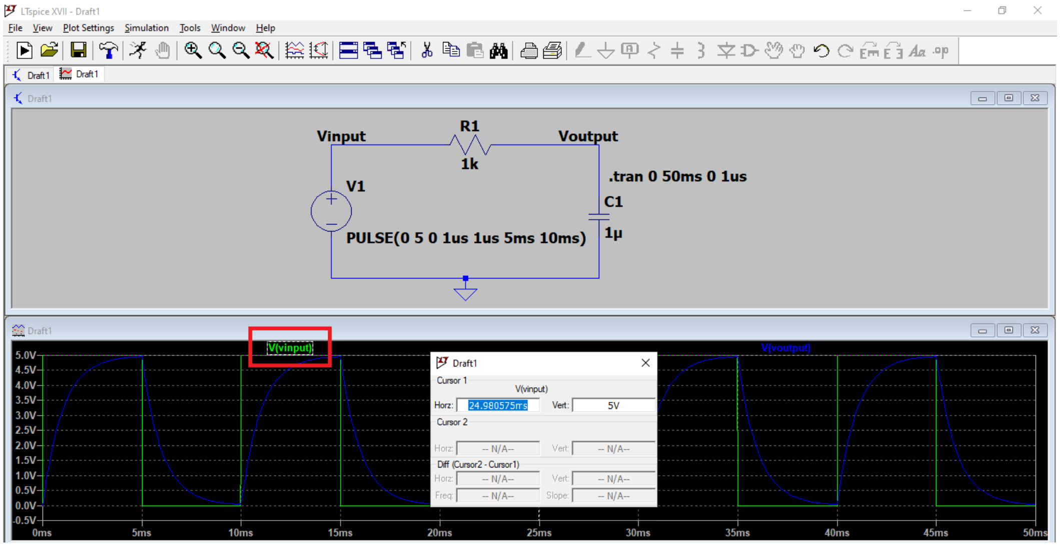


[Notice the '.tran' syntax for transient analysis.]

5. To run the simulation, click ***'Run'***. Find the ***'Run'*** button from the above toolbar or by just Right-clicking on the schematic.

6. After clicking the ***'Run'*** button a plot window will appear. In this window we can see responses and waveshapes of voltage and currents with respect to time. To see a plot ***Right-click on the plot window → Add trace → Select any voltage or current → OK.***
[We can also add trace by simply using marker on the schematic. When the run is complete a cursor will appear if we place the mouse cursor on a wire or component of the circuit.]

7. To extract data from a plot/response, use the data cursor. A cursor for a particular trace will appear by clicking on the name of that trace. The data point of the cursor can be moved by the arrow keys from the keyboard.



The axes properties (Range) can be changed by Right-clicking on the horizontal (x-axis) and the vertical (y-axis).

8. Save the Schematic by ***clicking File → Save as → 'Name.asc'*** and the plots by ***clicking File → Save plot settings → 'Name.plt'*** for future use and analysis.

LAB TASKS:

1. Built and simulate the circuit in Fig. 1 in LTspice and plot Vin, Vc, V_R and current I.
2. From the plot note max, min, peak to peak values for Vin, Vc, V_R **.** Also measure time constant τ and 5τ.

REPORT:

1. Draw the waveforms of **V_C**, **V_R**, and **I** for up to **50 ms** for the specified V, R, and C values. Give the waveforms the proper value labels.
2. Show measured values from **Lab Task 2** in a tabular form.
3. Built and simulate circuit shown in Fig.1 with a **5V 100 Hz square source**, **R = 1 KΩ** and **C = 0.47 µF and C = 4.7 µF** separately.
4. Draw the waveforms for the (3), note and, show the data mentioned in Lab Task 2 in a tabular form. Also comment on the change in result for different circuit.