Faculty of Engineering
Department of Electrical & Computer Engineering (ECE)

Summer Semester - 2016

Electronics - II (ECE 411)

Experiment No: 02

"Transient Response Analysis of RC Series Circuit"

Submitted By:

_____

University ID No:

Name of Faculty: Ankit Patel

Introduction:

Cellular phones and computers are just two examples of some of today's extremely complex electronic systems. Such devices contain millions of circuit components, and simple trial and error is not an effective way of ensuring that the final product will work properly. As a result, designers often use circuit simulators to verify the performance of a circuit before fabrication.

The most popular component level circuit simulator available today is called SPICE (**S**imulation **P**rogram with **I**ntegrated **C**ircuit **E**mphasis), which was developed at University of California, Berkeley, in the 1970s under the guidance of Prof. Pederson. Today vendors offer many different versions of SPICE that differ mainly in the user interface but are internally very similar to the original "Berkeley SPICE". The new SPICE is called, MultiSim.

Circuit simulation with MultiSim involves two steps:

a. Enter in the circuit schematic (with MultiSim's Graphical User Interface)
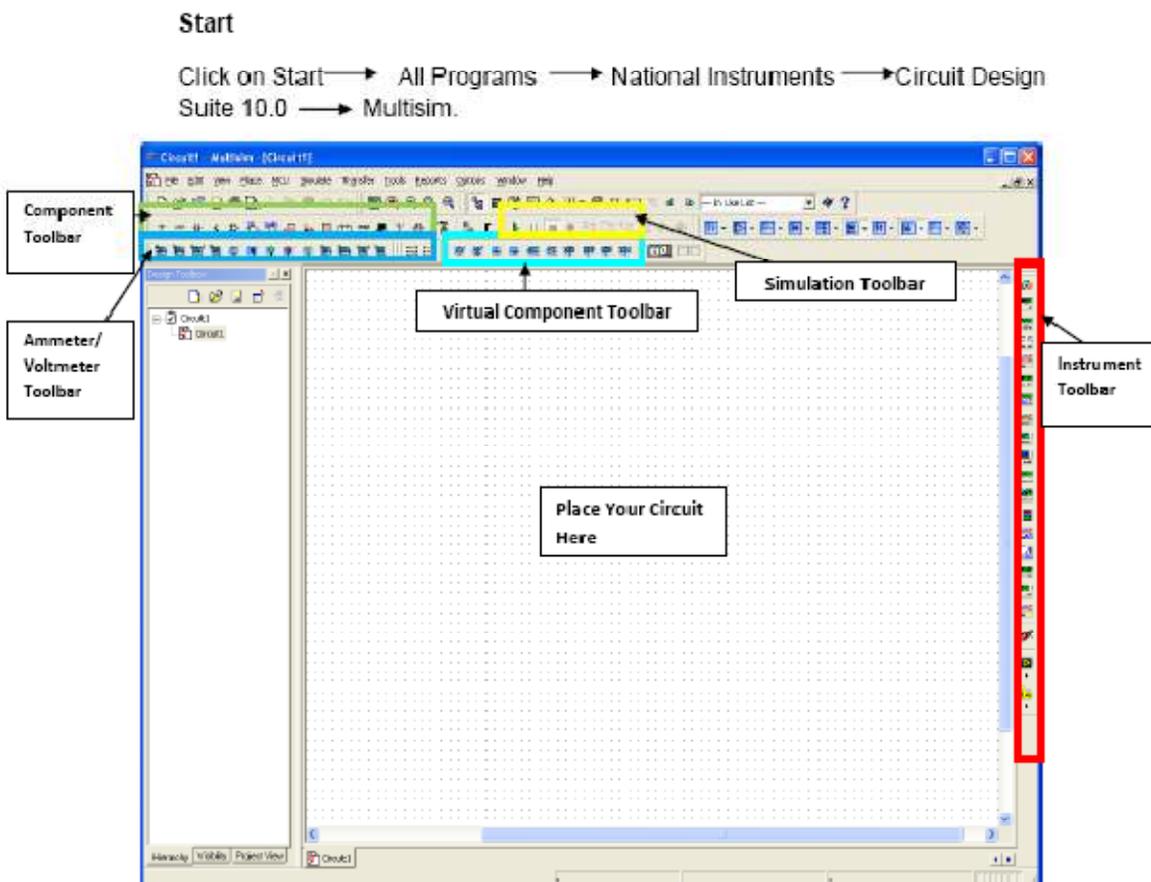b. Choose the type of analysis and run the simulation

Fig.1 Schematic Diagram of Main Screen of MultiSim

Transient – A circuit changes from one DC configuration to another DC configuration (a source value changes or a switch flips). Determine the DC state (current, voltage, etc.) before the change. Then determine what happens after the change. Over time, the circuit will settle into a new DC state, where the capacitors are again open circuits. In between will be an interval during which currents and voltages are changing as the capacitors charge or discharge. Since this lasts for only a "short" time, this is known as a *Transient Effect.*

The Transient Response (also known as Natural Response) is the way the circuit responds to energies stored in storage elements, such as capacitors and inductors. If capacitors have energy stored within it, then that energy can be dissipated/absorbed by a resistor. How that energy is dissipated is the Transient Response.
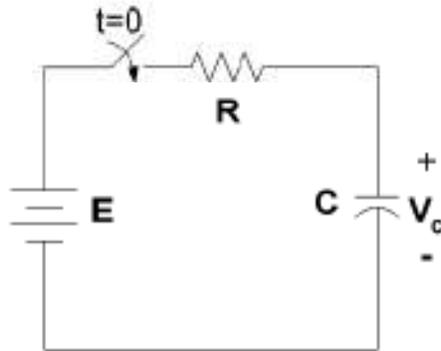

Fig. 2 RC Series Circuit

The capacitor in the circuit shown above is assumed to have no initial voltage. When the switch is closed, the capacitor begins to charge. The voltage $V_c(t)$ across the capacitor for t=0 is given as,

$$Vc\ (t) = E\ (1\text{-}e^{\text{-}t/RC})$$

The current through the capacitor is expressed as,

$$I_c = \frac{E}{R} e^{-\frac{t}{RC}}$$

Procedure:
- Build the circuit on MultiSim Electronic Workbench.
- Run the transient analysis. The next steps are as under.
  - Select Simulate on Main Menu
  - Select Analysis
  - Select Transient Analysis
  - Select Analysis Parameters
    - Initial conditions – Set to zero
    - Start time – 0 second
    - End time – 120 second
- Select output variables

- Select simulate
- Observe the wave shape