

# EXPERIMENT NO: 7(a)

Author : Gaurav Supal, Vineeta Parmar  
Date : March 14, 2016

## Aim:

Analysis of RC series circuit using eSim.

## Theory:

In a RC circuit a resistor and a capacitor is connected in series to a voltage source such as a battery. The battery is connected in series with a resistor will produce a constant current. The same battery in series with a capacitor will produce a time-varying current, which decays gradually to zero as the capacitor charges up. The time it takes for these transient currents to decay depends on the resistance (R) and capacitance (C). The resistor resists the flow of current; it thus slows down the decay. The capacitance measures capacity to hold charge.

$$\tau = R \cdot C$$

## Procedure:

1. Create the schematic of the RC Circuit as shown in Figure-1.
2. Annotate the schematic.
3. Test Electric rules.
4. Generate the netlist.
5. Insert analysis for transient analysis from 0 to 100 ms with a step time of 10 ms.
6. Insert Source Details.
7. Convert KiCad netlist to Ngspice netlist.
8. Simulate the Ngspice netlist using Ngspice simulator.

## Source Parameters:

Following are the Pwl parameters

1. Enter Value (t1 v1 t2 v2 ..) = 0m 0 0.5m 5 50m 5 50.5m 0 100m 0

## Schematic Diagram:

The circuit schematic of RC series circuit in eSim is as shown below:

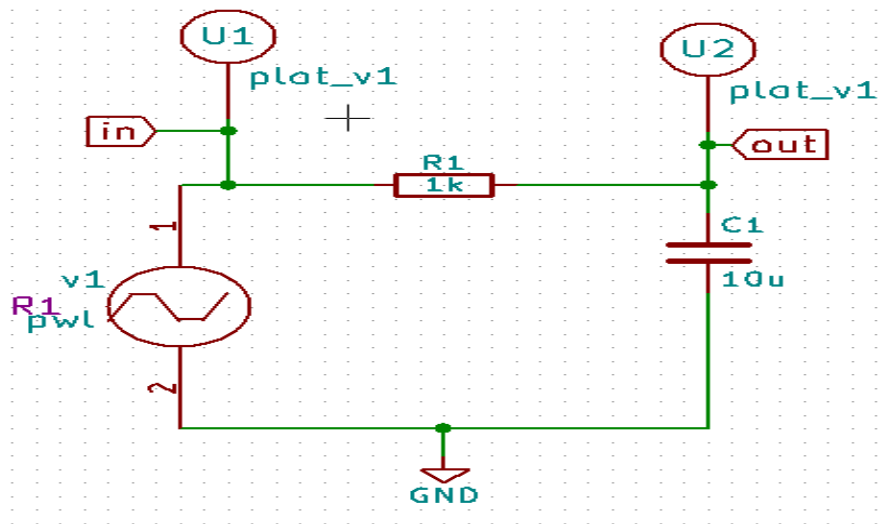


Figure 1: RC circuit

## Simulation Results:

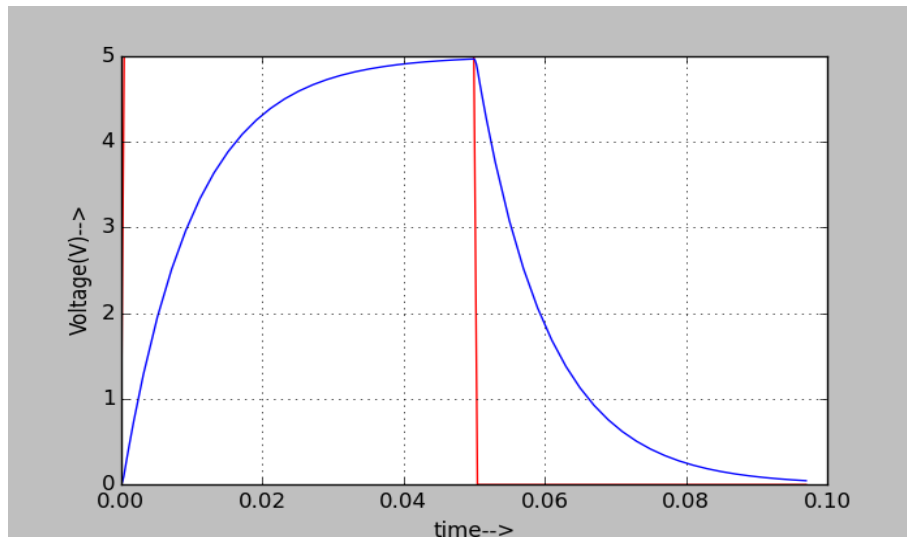


Figure 2: Python Plot

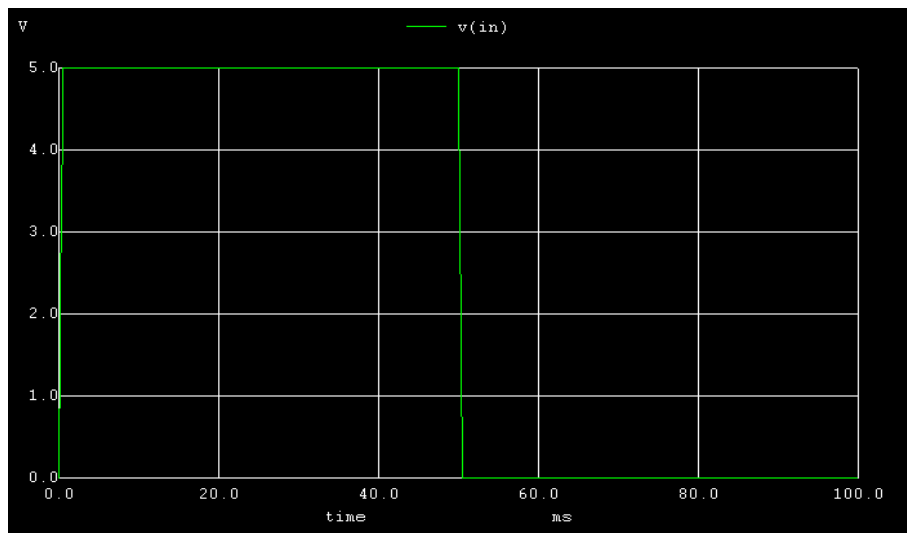


Figure 3: Ngspice Input Plot

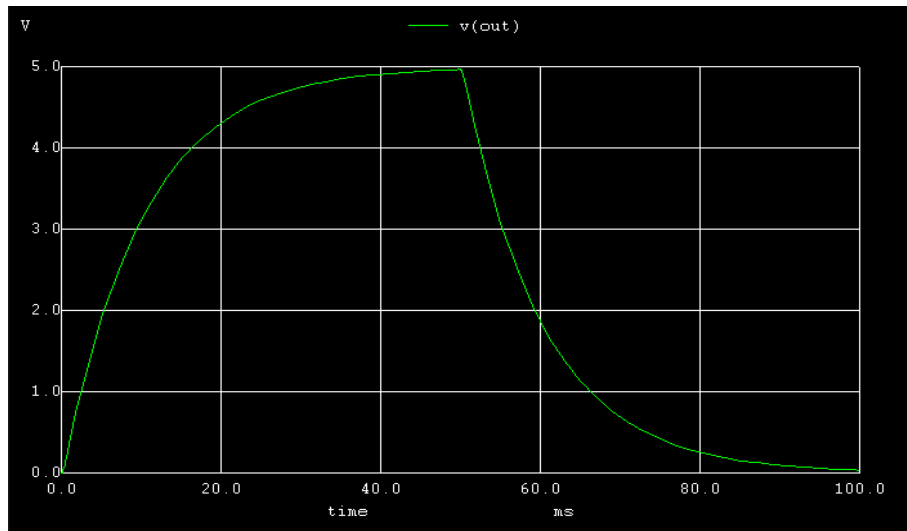


Figure 4: Ngspice Output Plot

## Conclusion:

Thus, we have studied the series RC circuit using eSim and we get the appropriate waveforms.

## References:

<http://en.wikipedia.org/wiki/RC-circuit>

# EXPERIMENT NO: 7(b)

## Aim:

Analysis of RL series circuit using eSim.

## Theory:

In a simple RL circuit the resistor, R and inductor, L are connected in series with a voltage supply of V volts. Since both resistance and inductor are connected in series, so the current in both the elements and the circuit remains the same. Let VR and VL be the voltage drop across resistor and inductor.

Applying Kirchhoff voltage law ( i.e sum of voltage drop must be equal to apply voltage) to this circuit we get,

$$Vr^2 + Vl^2 = V^2$$

## Procedure:

1. Create the schematic of the RL Circuit as shown in Figure-1.
2. Annotate the schematic.
3. Test Electric rules.
4. Generate the netlist.
5. Insert analysis for transient analysis from 0 to 100 ms with a step time of 10 ms.
6. Insert Source Details.
7. Convert KiCad netlist to Ngspice netlist.
8. Simulate the Ngspice netlist using Ngspice simulator.

## Source Parameters:

Following are the Pwl parameters

1. Enter Value (t1 v1 t2 v2 ..) = 0m 0 0.5m 5 50m 5 50.5m 0 100m 0

## Schematic Diagram:

The circuit schematic of RL series circuit in eSim is as shown below:

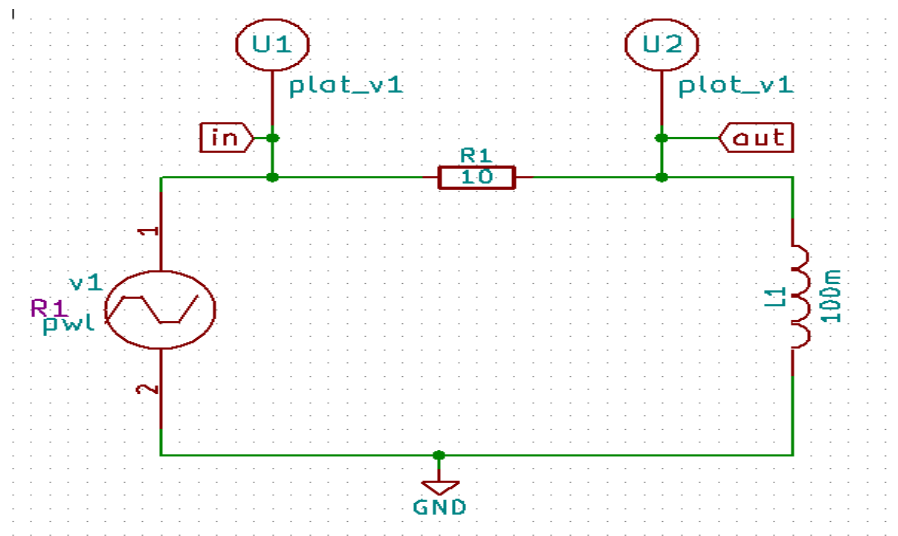


Figure 1: RL circuit

## Simulation Results:

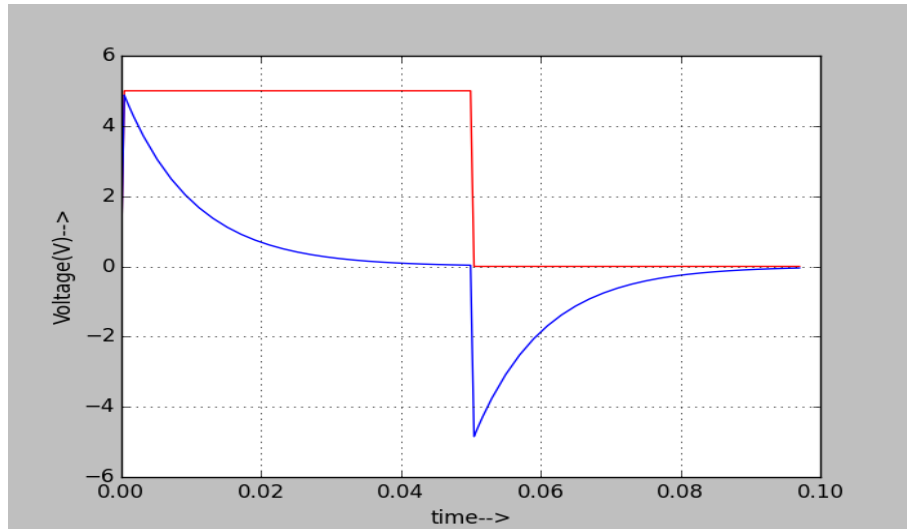


Figure 2: Python Plot

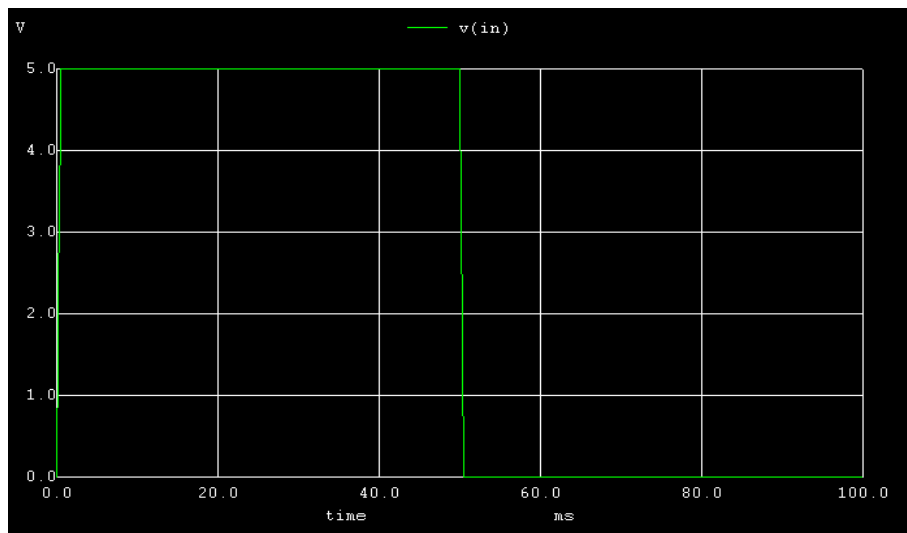


Figure 3: Ngspice Input Plot

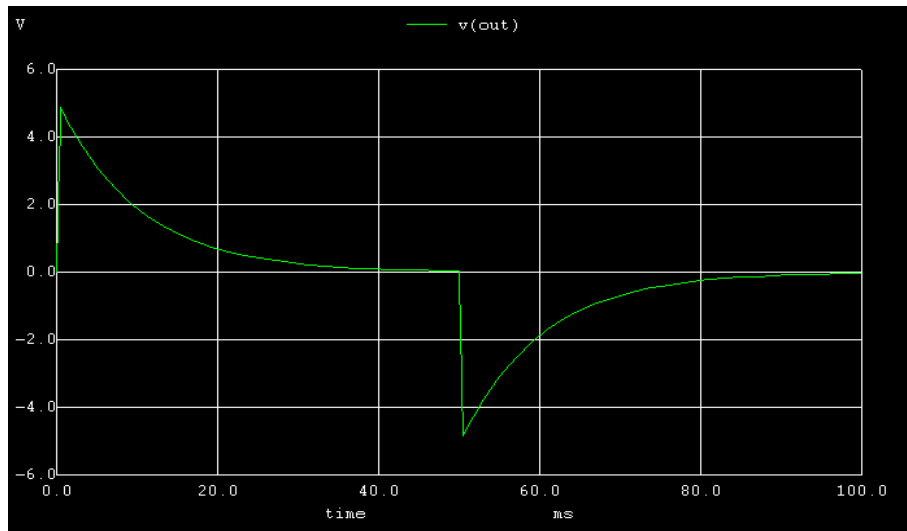


Figure 4: Ngspice Output Plot

## Conclusion:

Thus, we have studied the series RL circuit using eSim and we get the appropriate waveforms.

## References:

<http://www.electrical4u.com/rl-series-circuit/>



# EXPERIMENT NO: 7(c)

## Aim:

Analysis of RLC series circuit using eSim.

## Theory:

When a resistor, inductor, and capacitor are connected in series with the voltage supply, the circuit so formed is called series RLC circuit. Since all these components are connected in series, the current in each element remains the same,

$$IR + IL + IC = I(t) \text{ where } I(t) = IM \sin \omega t$$

Applying Kirchhoff voltage law to this circuit we get,

$$L \frac{dI(t)}{dt} + QI(t) + 1/C I(t) = V(t)$$

Series RLC circuits are classed as second-order circuits because they contain two energy storage elements, an inductance L and a capacitance C.

## Procedure:

1. Create the schematic of the RLC circuit as shown in Figure-1.
2. Annotate the schematic.
3. Test Electric rules.
4. Generate the netlist.
5. Insert analysis for transient analysis from 0 to 100 ms with a step time of 10 ms.
6. Insert Source Details.
7. Convert KiCad netlist to Ngspice netlist.
8. Simulate the Ngspice netlist using Ngspice simulator.

## Source Parameters:

Following are the Pwl parameters

1. Enter Value (t1 v1 t2 v2 ..) = 0m 0 0.5m 5

## Schematic Diagram:

The circuit schematic of RLC series circuit in eSim is as shown below:

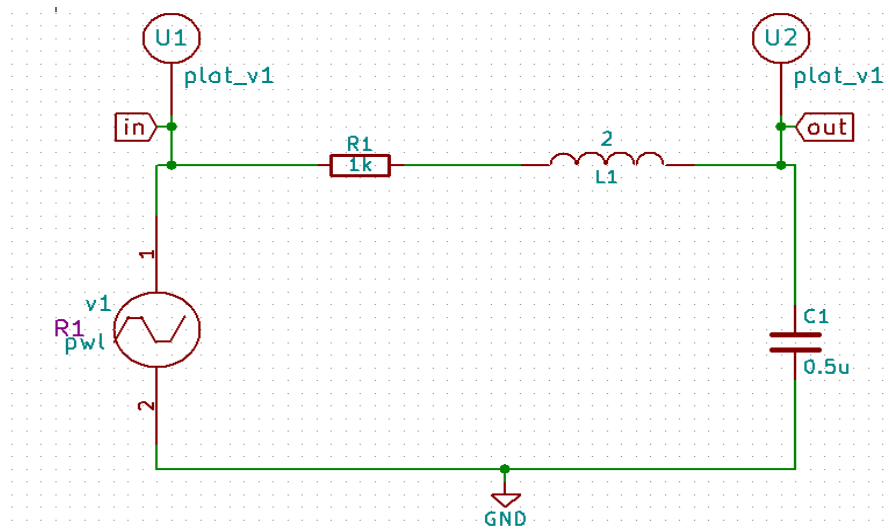


Figure 1: RLC circuit

## Simulation Results:

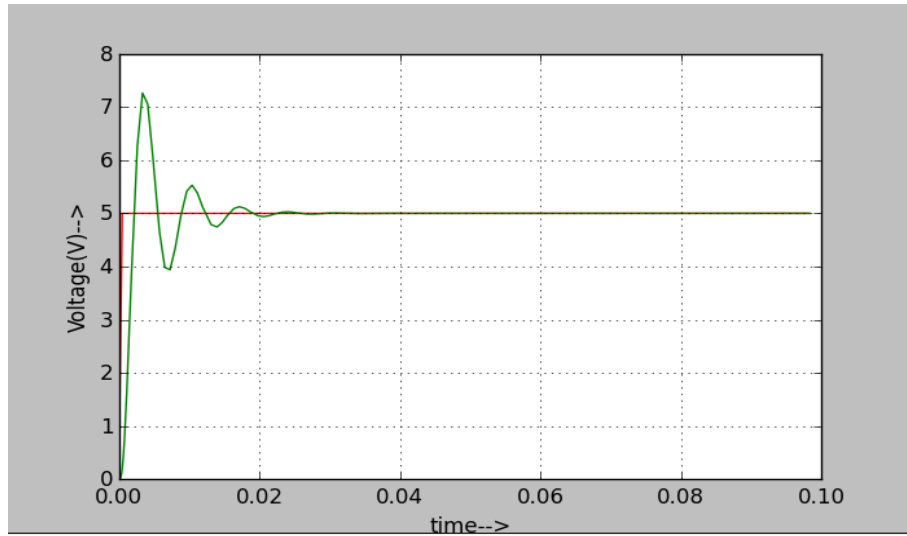


Figure 2: Python Plot

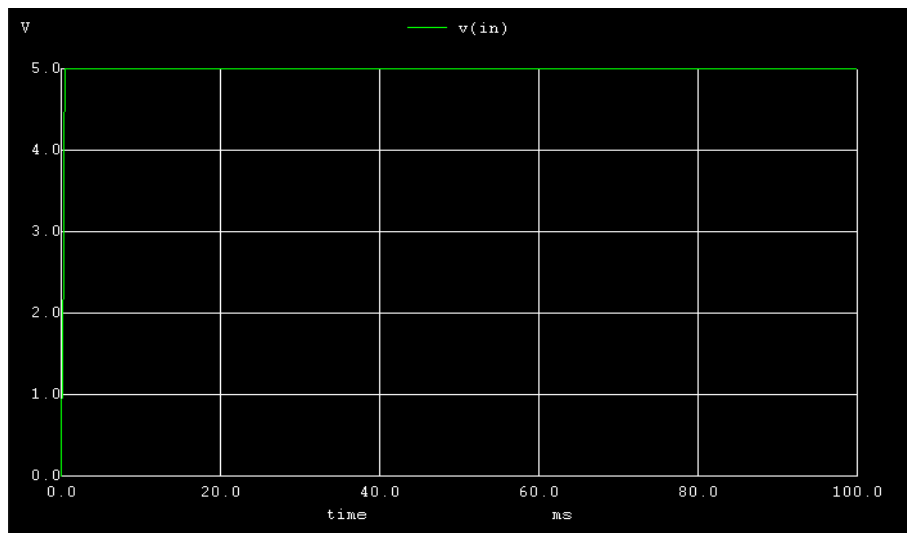


Figure 3: Ngspice Input Plot

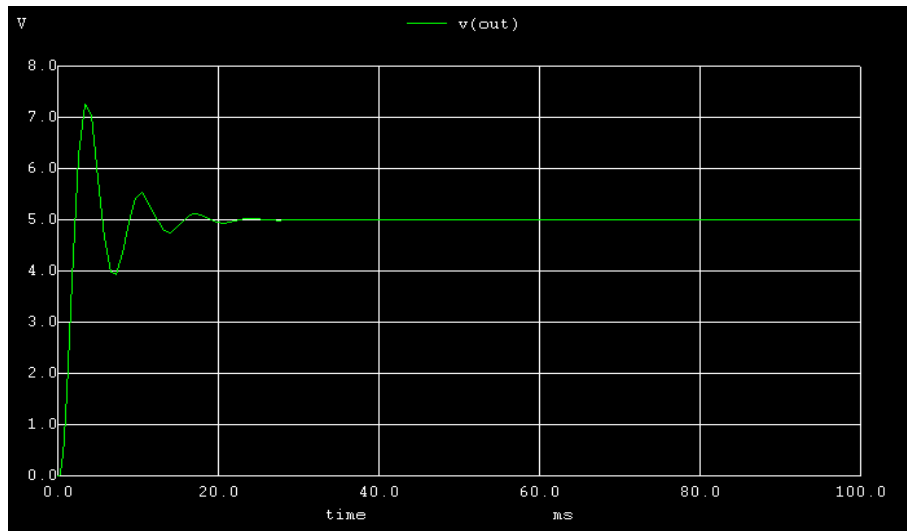


Figure 4: Ngspice Output Plot

## Conclusion:

Thus, we have studied the series RLC circuit using eSim and we get the appropriate waveforms.

## References:

<http://www.electrical4u.com/rl-series-circuit/>