



American International University- Bangladesh
Department of Electrical and Electronic Engineering
 Introduction to Electrical Circuits Laboratory

Title: Transient Analysis of *RC* Series and *RL* series using PSPICE

Introduction:

In this experiment we apply a pulse waveform to the RC and RL series circuit to analyze the transient response of the circuit by using PSPICE simulating tool. The pulse width relative to a circuit's time constant determines how it is affected by an RC and RL circuits.

The purpose of this experiment is to

1. simulate the circuits by using components from the PSPICE library and,
2. analyze obtained graphs and results.

Theory and Methodology:

Time Constant (τ): A measure of time required for certain changes in voltages and currents in RC and RL circuits. Generally, when the elapsed time exceeds five time constants (5τ) after switching has occurred, the currents and voltages have reached their final value, which is also called steady-state response.

The time constant of an RC circuit is the product of equivalent capacitance and the Thevenin resistance,

$$\tau = R \times C \quad (1)$$

The time constant of an RL circuit is the equivalent inductance divided by the Thevenin resistance,

$$\tau = L/R \quad (2)$$

Time Period (T): Time required to complete one cycle is called Time Period or the length of each cycle of a pulse train is termed its time period (T).

Pulse width (t_p): The pulse width of an ideal square wave is equal to half of the time period.

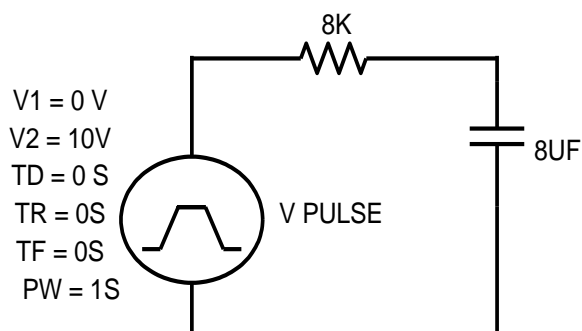


Figure-1: RC circuit

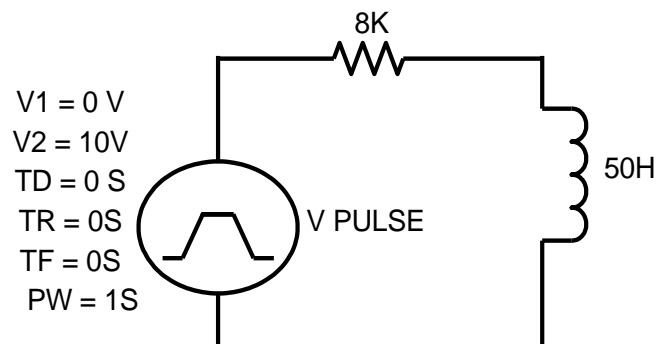


Figure2: RL circuit

Pre-Lab Homework:

Read about the characteristics of RC and RL series circuit during transient analysis from “Alternating Current Circuit” by George F Corcoran and use **PSPICE** to generate the output of the circuits provided in this lab sheet. Compare the wave shapes given in the text book with your results. Save the simulation results and bring it to the lab.

Apparatus:

- PC
- **PSPICE** Simulating tools

Precautions:

Connecting of circuit should be done properly and PSPICE simulating software should be properly installed using the information provided at the manual before starting the experimental work.

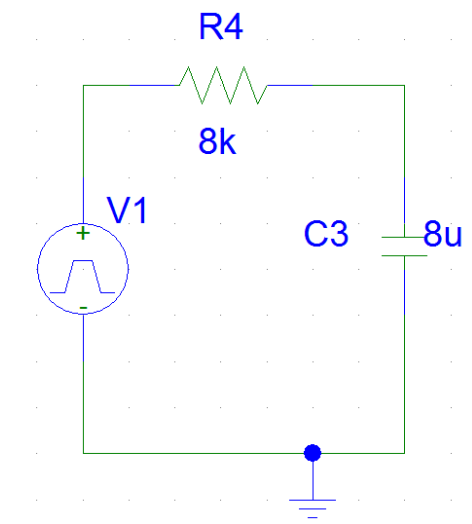
Experimental Procedure:

1. Open the PSPICE Design Manager window: Start → Program → MicroSim Eval 8.
2. Open schematic editor: Press Run Schematics icon from the bar on the left side of the screen.
3. Select: Draw → Get New Part, then select and place each of the circuit elements one by one (VPULSE for pulse type voltage source, R for resistor, C for capacitor and EGND for ground). Join the elements by using the wire as necessary.
4. Change the label and magnitude of each element by double clicking on them and editing as necessary.
5. Then go to Analysis → Setup → Select Transient and provide the necessary Values. And swlwct automatically Run Probe after Simulation.
6. To execute the analysis, select: Analysis → Simulate or press F11. To view the analysis result select: Analysis → examine output.
7. Select Analysis → Simulate.
8. Select Trace → Add and select the desired traces.
9. Perform the text analysis as instructed.

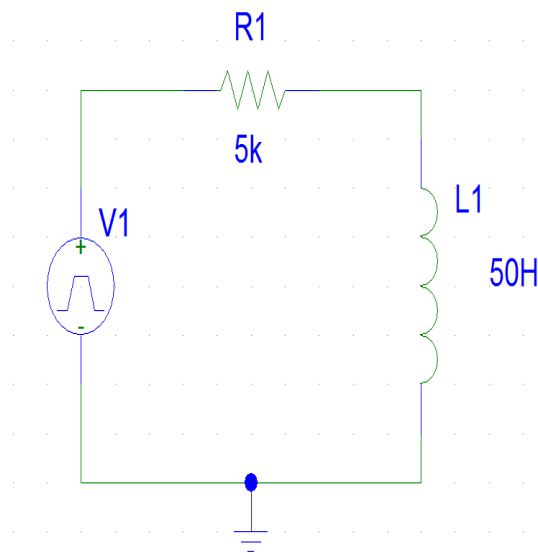
Simulation and Measurement:

In PSpice window, construct the virtual RC and RL circuits as shown below. Simulate the parametric wave shape for both the circuits. Compare the simulation results with your theoretical data and comment on the differences (if any).

RC Circuit:



RL Circuit:



Questions for report writing:

1. Set the value of $C = 100\mu\text{f}$, $L = 100\text{H}$, $R = 50\text{k}\Omega$ and simulate the circuits.
2. Calculate the value of τ and t_p for RL and RC circuit.
3. Verify the simulating result of the total circuit with theoretical result and also Comment on the result as a whole.

Discussion and Conclusion:

Interpret the data/findings and determine the extent to which the experiment was successful in complying with the goal that was initially set. Discuss any mistake you might have made while conducting the investigation and describe ways the study could have been improved.

Reference(s):

1. Robert L. Boylestad, "Introductory Circuit Analysis", Prentice Hall, 12th Edition, New York, 2010, ISBN 9780137146666.
2. R.M. Kerchner and G.F. Corcoran, "Alternating Current Circuits", John Wiley & Sons, Third Ed., New York, 1956.
3. Lamar University website, [Cited: 12.01.2014]
Available: <http://ee.lamar.edu/eelabs/elen2107/lab5.pdf>
4. Lamar University website, [Cited: 12.01.2014]
Available: <http://ee.lamar.edu/eelabs/elen2107/lab6.pdf>