American International University- Bangladesh
Department of Electrical and Electronic Engineering
COE 2102: Introduction to Electrical Circuits Lab

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

Abstract:

Multism is an electrical circuit simulation software with which circuits can be drawn, checked, and simulated for finding unknown circuit parameter-values and graphs. The software has component and device library which can be used for any types of construction, simulation, and demonstration of circuits.

Introduction:

In this lab, the students will learn about the circuit simulation software Multisim. They will also get accustomed to Multisim library. Besides, focus will be made over

1. Simulation of circuits by using components from the Multisim library,
2. Simulation of circuits by writing script files and to 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,

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

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

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

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

V1 = 0 V
V2 = 10V
TD = 0 S
TR = 0S
TF = 0S
PW = 1S

8K

V PULSE

8UF

V1 = 0 V
V2 = 10V
TD = 0 S
TR = 0S
TF = 0S
PW = 1S

8K

V PULSE

50H

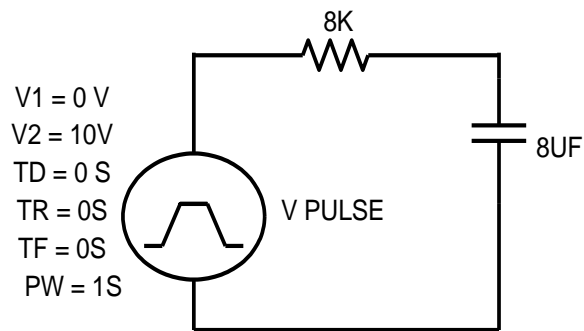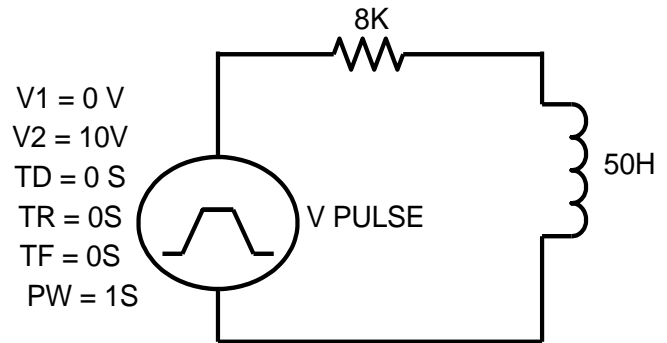Figire-1: RC circuit   Figire2: RL circuit

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 **Multisim** to generate the output of the circuits provided in this lab sheet. Compare the wave shapes given in the textbook with your results. Save the simulation results and bring it to the lab.

Apparatus:
- PC
- **Multisim** Simulating tool

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

Experimental Procedure (MULTISIM):

Simulating Circuits by using Components from the Multisim library

1. Open the Multisim software window: Start → Program → Multisim
2. Open component window from menu bar: Place → Components
3. Select Dc source/ Digital Clock/ Step Voltage source from components, a resistor, a capacitor (for RC) or an inductor (for RL) and a ground source.
4. For Clock or Pulse voltage: Place → Components → Sources → Signal_Voltage_Sources → Clock_Voltage or Pulse_Voltage.

5. Set the source, resistor, capacitor/inductor values properly.
6. Connect all the elements by using wire as necessary and label them properly.
7. Then go to Analysis and Simulation bar and change it to Transient: Analysis and Simulation → Transient. And select the Initial condition to set to zero.
8. Choose an End time that is appropriate, long enough (>5 time constant) to show multiple cycles of a wave and short enough to not have excessive number of cycles.
9. Choose the expected output variables from output window: Analysis and simulation → Output → Add output variable.
10. Insert new expressions if needed: Analysis and simulation → Output → Add expression.
11. Now run the simulation for the designed circuit and analyze the output from the simulation grapher view.
12. Perform the analysis as instructed.

Data Table:

RC Circuit (1 µF, 100Hz)

τ	Value Time Constant	% Charged	V_c
1τ			
2τ			
3τ			
4τ			
5τ			

RL Circuit (200 mH, 5Hz)

τ	Value Time Constant	% Storage	I_L
1τ			
2τ			
3τ			
4τ			
5τ			

Discussions:

i. In this experiment, RC, RL series circuits were constructed.
ii. Value time constant was modified as required and V_c, I_L were measured. The obtained data was inserted into the table.
iii. Relevant calculation was done using the experimental data.
iv. The analysis was completed effectively using **τ = R×C and τ = L/R**

v.   Every mentioned step should be completed properly to make sure the simulation works properly.

Questions for report writing:

1. Set the value of C= 1µf, R=1kΩ for RC, L= 200mH, R= 10Ω for RL circuit 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 Comment on the result.

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