www.simiode.org

SIMIODE A SYSTEMIC INITIATIVE FOR MODELING INVESTIGATIONS & OPPORTUNITIES WITH DIFFERENTIAL EQUATIONS

STUDENT VERSION

RL Series Circuit - Analytical Solution vs Modeling Software vs Physical Setup

Virgil C. Ganescu
Department of Mathematics
York College of Pennsylvania
York PA USA

STATEMENT

An inductor in an electrical circuit "opposes" the flow of current through it. An ideal inductor has no resistance or capacitance associated with its coil windings. A series RL (Resistance Inductance) small AC (Alternating Current) circuit consists of an AC source connected in "series" with a resistor (of resistance R) arranged in "series" with an inductor of inductance L.

This modeling and validation experiment consists of three parts.

Part I: Analytically solving the governing equation of the circuit

Consider the RL series circuit in Figure 1.

Ohm's law [4] for the voltage drops across the resistor R and across the inductor L, in combination with KVL (Kirchhoff's Voltage Law), applied to the entire circuit [2], gives the above circuit's "governing" differential equation. The equation used as the basis for this modeling scenario can be stated as:

$$v_{s(t)} = \left(\frac{d_{is(t)}}{dt}\right) + R_{i_{s(t)}}\,. \tag{1}$$

This is a first order linear ODE (ordinary differential equation), having the current $i_{s(t)}$ as the variable (function of time). $i_{s(t)}$, determined in Part I, will be compared with the current $i_{s(t)}$ computed using a numerical simulator, Multisim, in Part II. These results may be compared with the measured $i_{s(t)}$ from the actual (physical) circuit in Part III.

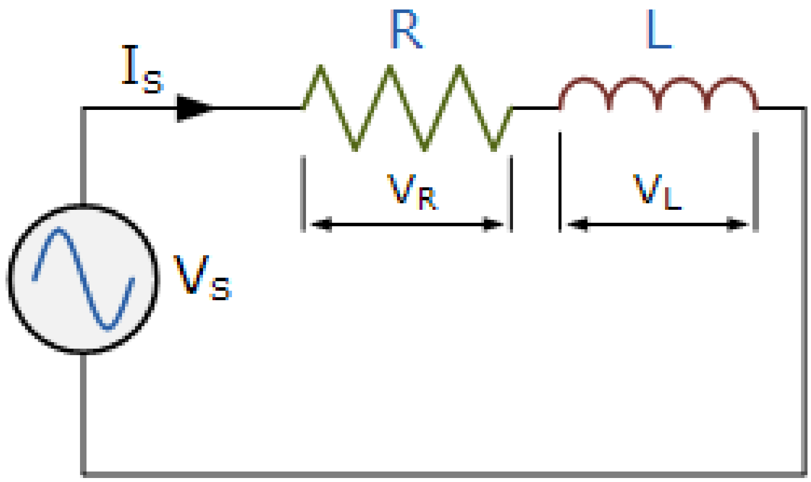

Figure 1. Schematic of the series RL circuit

From a more specific stand point, the relationship above can be re-written as:

$$\frac{d\,i_{s(t)}}{dt} + \frac{R}{L} i_{s(t)} = \frac{1}{L} v_{s(t)}\,, \tag{2}$$

which fits the very form of a first order linear ODE.

The differential equation (2) is solved ([5]), and yields (an instantaneous value at time t) an $i_{s(t)}$ given by:

$$i_{s(t)} = e^{-\frac{Rt}{L}} \int \left(e^{-\frac{Rt}{L}} \frac{v_{s(t)}}{L} \right) dt + C\,, \tag{3}$$

where, the input source voltage $v_{s(t)}$ is a sinusoidal waveform. Moreover, for the purpose of our simulation, $i_{s(0)} = 0$A is the initial condition. This allows for the calculation of constant C in (3) above, while the values selected for the resistance R and inductance L, are carefully chosen so that they are the same component values used in Part II (virtual simulation), as well as in Part III (actual, physical circuit) of the simulation. Equation (3) is therefore solved for $v_{s(t)} = \sin(100t)V$, $i_{s(0)} = 0$A, $R = 1\Omega$ and $L = 0.01$H, giving:

$$i_{s(t)} = 0.5 \left(\sin(100t)\cos(100t) + e^{-100t} \right)\,. \tag{4}$$

The form of $i_{s(t)}$ obtained in (4) represents the instantaneous value of the current through the RL circuit.

Part II: Determining $i_{s(t)}$ using modeling software Multisim

Multisim ^{rmTM} software [6] integrates the industry-standard SPICE software [7] simulation with an interactive schematic environment to instantly visualize and analyze electronic circuit behavior. By adding powerful circuit simulation and analyses to the design flow, Multisim helps researchers and designers reduce printed circuit board prototype iterations and save development costs.

One of the attractive features of using Multisim centers on the almost non-existent learning curve. Free, limited time trials (for educational purposes) can be downloaded from [3]. Workstations - with Multisim already installed - from a dedicated electrical engineering lab, say, can also be used.

Using Multisim, as seen in [1] the circuit skhown in Figure 2 is built, with the source ($v_{s(t)}$), R, and L values used in Part I.

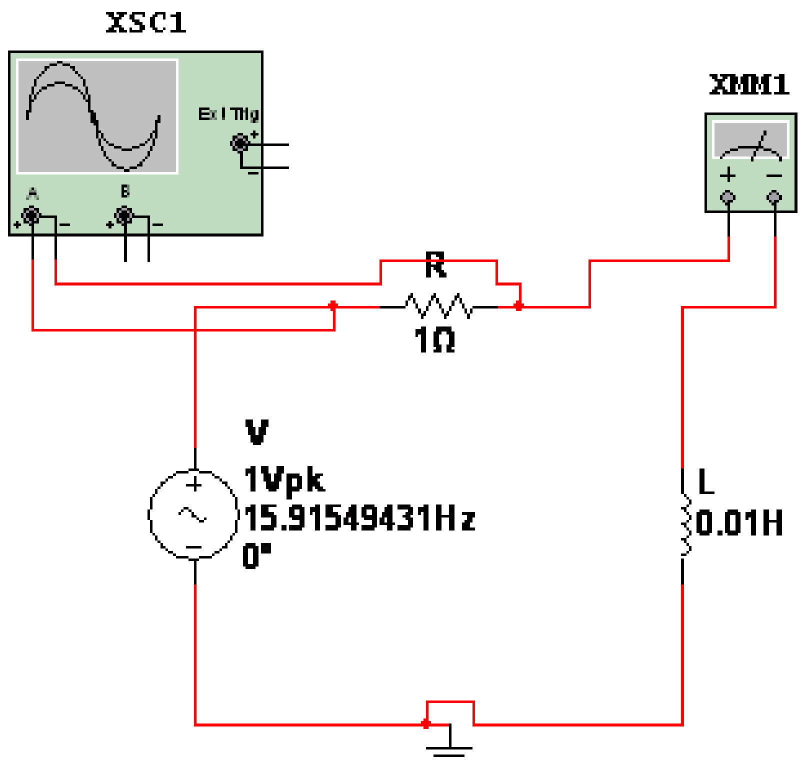

Figure 2. Schematic of the series RL circuit.

A digital multimeter (XMM1) is used to collect RMS (root-mean-square [8]) current, $i_{s(t)}$, measurements (determined to be at 499.188 mA); these measurements can be compared with the RMS current measurements taken in Part III (physical setup) of the simulation. The multimeter is connected in series with (as part of) the circuit.

Additionally, in order to visualize the signal through the circuit, an oscilloscope (XSC1) is utilized. The oscilloscope is connected for voltage measurements across the resistor R (in parallel with the resistor). XSC1 will show peak [8] values of the voltage across resistor R, as well as indicate how the voltage across R behaves.

For ease in contrasting the data gathered via the procedures suggested in Part I and Part II, PEAK current values are determined (PEAK voltage values are acquired by XSC1 using Ohm's law [4] is used) by analyzing the data acquired from Multisim.

Specifically, while in Multisim, the simulation is turned on, and stopped once stabilized. On the display of the oscilloscope XSC1, any of the cursors is dragged to any of the PEAKS of the voltage visualized (voltage across resistor R). This also represents the very point in time used to determine the peak current from (1).

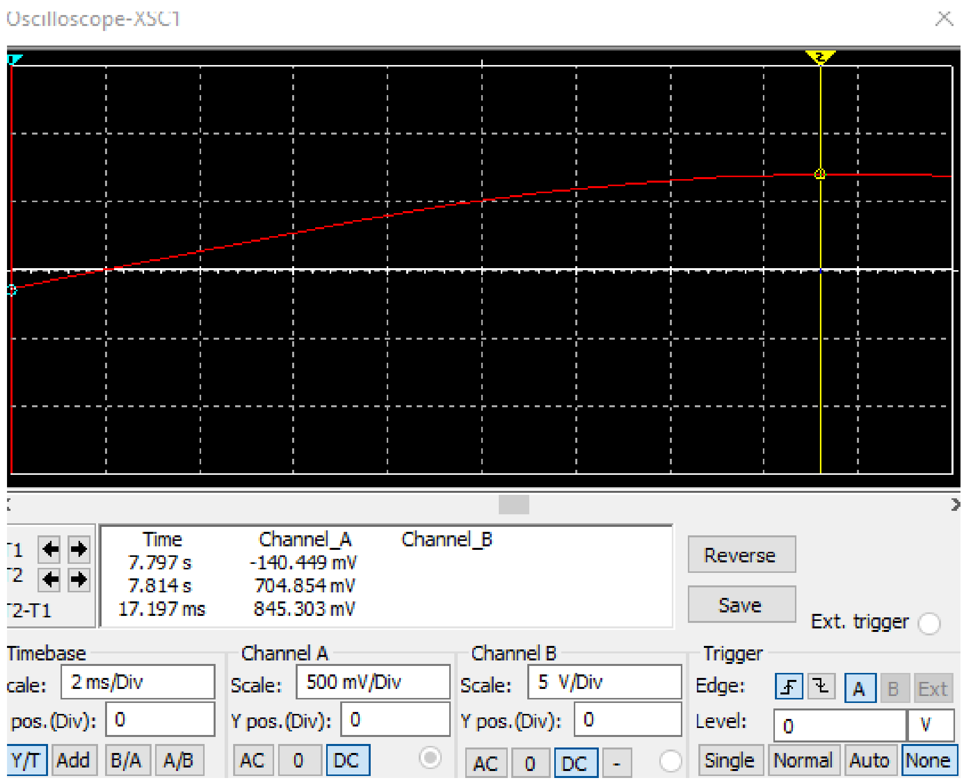

Figure 3. Screenshot of oscilloscope (PEAK voltage across R is displayed. Cursor 2 (yellow) from Figure 3 indicates a PEAK voltage across R of approximately 704.854 mV (instance occurring at time $t = 7.814$ s).

The PEAK current through the circuit is obtained indirectly, by applying Ohm's law [4] at the level of the resistor (the observed PEAK voltage). Radians are used to evaluate sin and *cos*. This data validates the analytical analysis of the RL circuit against the numerical modeling via Multisim.

Part III: Determining $i_{s(t)}$ from an actual/physical prototype of the circuit

This part reiterates the realistic applicability of modeling via ODEs, while reinforcing the advantages associated with numerical simulations, e.g., using Multisim, one can consider various tolerances to the components used as well as the fluctuating temperature of operation.

In a dedicated electrical engineering lab, the circuit from Figure 2 is constructed. The function generator used (to supply the input voltage $v_{s(t)}$) has to be set on sinusoidal output. The voltage, frequency, resistance and inductance values used have to be the same values implemented in Parts I and II of the experiment.

In order to keep the analysis concise, limited to strictly basic measurement equipment, and to make use of yet another industry specific apparatus, it is suggested that RMS current values be recorded with a (preferably digital) multimeter, connected in the same manner as XMM1 from Figure 2.

RMS current through the physical circuit is measured at 498.9mA, which matches the RMS value indicated by XMM1 in Part II of the simulation. Moreover, converting this 498.9mA RMS to PEAK [8], we obtain 705.851mA for the current value, which is in line, yet again, with the results obtained in Parts I and II.

REFERENCES

[1] Ganescu, V. C. 2019. Building a Circuit in Multisim. `https://app.vidgrid.com/view/yPwnkhhgvEVq/?sr=vT2IRl`. Accessed 2 March 2019.

[2] Wikiversity. 2018. Electric Circuit Analysis/Kirchhoff's Voltage Law. `https://en.wikiversity.org/w/index.php?title=Electric_Circuit_Analysis/Kirchhoff\%27s_Voltage_Law&oldid=1959596`. Accessed 2 March 2019.

[3] *Multisim*. 2019. Free Academic Evaluation of *Multisim*. `https://lumen.ni.com/nicif/US/GB_ACADEMICEVALMULTISIM/content.xhtml`. Accessed 2 March 2019.

[4] *MakerPro*. 2019. The Basics of Ohm's Law. `https://maker.pro/custom/tutorial/basics-of-ohms-lawhttps://maker.pro/custom/tutorial/basics-of-ohms-law`. Accessed 2 March 2019.)

[5] Nagle, R. K., A. D. Snider, and E. B. Saff. 2018. *Fundamentals of Differential Equations, 9th Edition*. Boston: Pearson.

[6] National Instruments. What is Multisim? 2019. `http://www.ni.com/en-us/shop/electronic-test-instrumentation/application-software-for-electronic-test-and-instrumentation-category/what-is-multisim.html`. Accessed 2 March 2019.

[7] Rhome Semiconductors. 2019. SPICE Model (Whatd is SPICE?) `https://www.rohm.com/electronics-basics/spice/what-is-spice`. Accessed 2 Marchi 2019.

[8] testguy.net. 2019. Peak vs. Average vs. RMS Voltage. `https://testguy.net/content/270-Peak-vs-Average-vs-RMS-Voltage`. Accessed 2 March 2019.