www.simiode.org
SIMIODE A SYSTEMIC INITIATIVE FOR MODELING INVESTIGATIONS & OPPORTUNITIES WITH DIFFERENTIAL EQUATIONS

STUDENT VERSION
RLC Series Circuit
Analytical Solution and Modeling Software

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 capacitor is another passive two-terminal electronic component that stores electrical energy in an electric field. The effect of a capacitor is known as capacitance. A series RLC (Resistance Inductance Capacitance) small AC (Alternating Current) circuit consists of an AC source connected in series to a resistor (of resistance R), arranged in series with an inductor of inductance L, and in series with a capacitor of capacitance C (see Figure 1).

This modeling and validation experiment consist of two parts. Each part is described as follows:

Part I: Analytically solving the governing equation of the circuit.

Consider the RLC series circuit depicted in Figure 1.

Using Ohm's law [3] for the voltage drops across the resistor R, across the inductor L, and across the capacitor C, in combination with KVL (Kirchhoff's Voltage Law), applied to the entire circuit [8], one can derive the "governing" differential equation for the circuit in Figure 1. This equation, used as the basis for this modeling scenario, can be stated as:

$$v_{s(t)} = L(di_{s(t)}/dt) + R\, i_{s(t)} + \frac{1}{C}q\,, \tag{1}$$

where q is the amount of charge accumulated on the plates of the capacitor [1]. In most applications, we will be interested in determining the current$i_{s(t)}$. If we differentiate relationship from (1) with respect to$t$, and substitute $i_{s(t)}$ for dq/dt, we obtain:

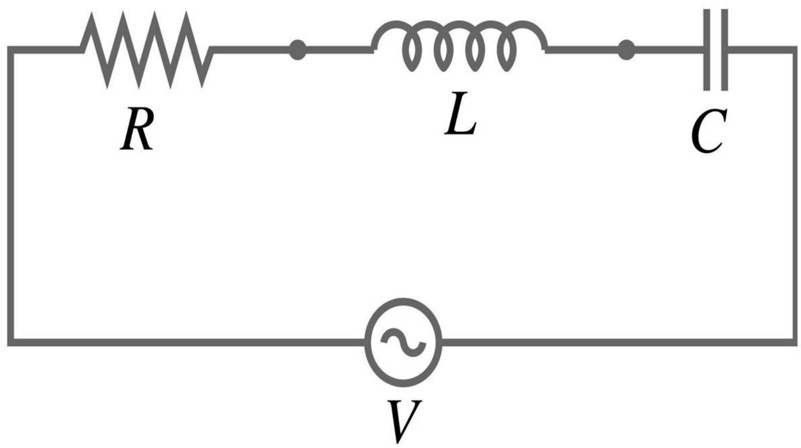

Figure 1. Schematic of the series RL circuit.

$$L(d^2 i_{s(t)}/dt^2] + R(di_{s(t)}/dt) + \frac{1}{C} i_{s(t)} = dv_{s(t)}/dt\,. \tag{2}$$

This is a second order linear ODE (Ordinary Differential Equation), having the current $i_{s(t)}$ as the dependent variable (a function of time, t). $i_{s(t)}$, determined here, will be compared with the current $i_{s(t)}$ computed using a numerical approach (Multisim on computer [4]).

In (2) the input source voltage $v_{s(t)}$ is a sinusoidal waveform. Moreover, for the purpose of our simulation, $i_{s(0)} = 0$A and $i'_{s(0)} = 0$A are the initial conditions (this assumption allows for the calculation of constants of integration). The values selected for the resistance R, inductance L and capacitance C, are carefully chosen so that they are the same component values used in Part II (virtual simulation) of the modeling scenario.

Employing the method of undetermined coefficients (procedure found in any standard textbook dedicated to ODEs), nonhomogeneous ODE (2) can be solved for $v_{s(t)} = \sin(100t)$ V, $R = 0.02\,\Omega$, $L = 0.001$H, and $C = 2$F (considering $i_{s(0)} = i'_{s(0)} = 0$A, as initial conditions), giving:

$$i_{s(t)} = \exp^{-10t}(10.08\cos(20t)5.57\sin(20t)) - 10.08\cos(100t) + 2.12\sin(100t)\,. \tag{3}$$

The form of $i_{s(t)}$ obtained in (3) represents the instantaneous value of the current through the RLC circuit.

In order to ensure validation, several sets of data should be considered for the circuit (for $v_{s(t)}$, R, L and C), while solving (3) for each set of parameters. The same values are to be used again in Part II.

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

Multisim software [4] integrates industry-standard SPICE [6] 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 saves development costs.

One of the attractive features of using Multisim is the almost non-existent learning curve. Free, limited time trials (for educational purposes) can be downloaded from [5]. Workstations - with Multisim already installed - from a dedicated electrical engineering lab, say, can also be used.

Using Multisim, as seen in [2], the circuit below is built, with the same source ($v_{s(t)}$), R, L and C values used in Part I of this modeling scenario.

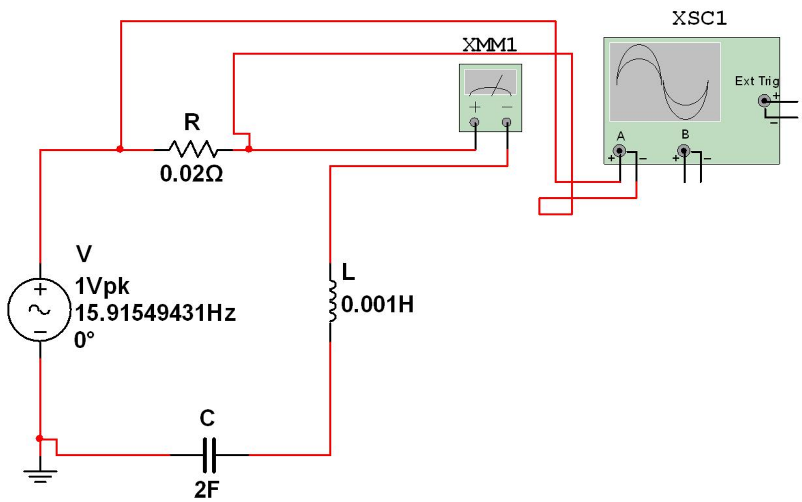

Figure 2. Multisim schematic of the series RLC circuit (shown in Figure **??**

A digital multimeter (XMM1) is used to collect RMS (root-mean-square [7]) current $i_{s(t)}$ measurements (determined to be at 7.258A, which is equivalent to 10.263A peak!!), to cross-check and cross-reference data. The multimeter is connected "in series", 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 [7] values of the voltage across resistor R, as well as indicate how the voltage across R behaves.

For ease of contrasting the data gathered via the procedures suggested, peak current values are determined (peak voltage values are acquired by XSC1, where Ohms Law [8] is used) by analyzing the data collected from Multisim.

Specifically, while in Multisim, the simulation is turned on, and stopped, once a steady value is displayed by the multimeter and a "clean" waveform is outputted by the oscilloscope. On the display of the oscilloscope XSC1, any of the two cursors is dragged to any of the peaks of the voltage visualized (voltage across resistor R); this instance will also represent the very point in time used to determine the peak current from (3).

Cursor 2 (blue) from Figure 3 indicates a peak voltage across R of approx. 204.399mV (instance

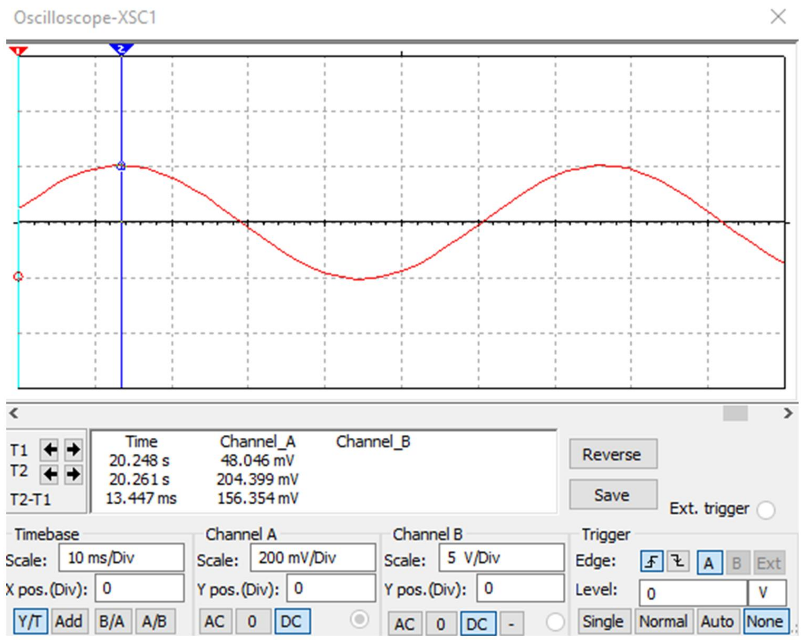

Figure 3. Screenshot of oscilloscope (peak voltage across R is displayed).

occurring at time t = 20.261s).

The peak current through the circuit is obtained indirectly, by applying Ohm's Law [8] at the level of the resistor. The observed peak voltage displayed by the XSC1 is divided by R, and is 10.219A, whereas (3) yields 10.299A (time t = 20.261s, displayed on the XSC1, when the peak voltage across R is obtained, is plugged in (3); radians are used to evaluate sine and cosine), current magnitude that validates the analytical analysis of the RLC circuit against the numerical modeling via Multisim.

Several iterations are recommended, using the same circuit values implemented in the "versions" used in Part I above.

REFERENCES

[1] Capacitor Guide. `http://www.capacitorguide.com/electric-field`. Accessed on 1 March 2019.

[2] Ganescu, V. 2019. *Building an RLC Circuit in Multisim*. `https://use.vg/6DYAzp`. Accessed on 5 March 2019.

[3] Maker Pro. `https://maker.pro/custom/tutorial/basics-of-ohms-law`. Accessed on 25 February 2019.

[4] Multisim. http://www.ni.com/en-us/shop/electronic-test-instrumentation/application-software-for-electronic-test-and-instrumentation-category/what-is-multisim.html. Accessed on 27 February 2019.

[5] Multisim Academic. https://lumen.ni.com/nicif/US/GB_ACADEMICEVALMULTISIM/content.tt xhtml. Accessed on 27 February 2019.

[6] Rohm Semiconductor. https://www.rohm.com/electronics-basics/spice/what-is-spice. Accessed on Accessed on 27 February 2019.

[7] Testguy. https://testguy.net/content/270-Peak-vs-Average-vs-RMS-Voltage. Accessed on 1 March 2019.

[8] Wikiuniversity. https://en.wikiversity.org/wiki/Electric_Circuit_Analysis/Kirchhoff's_Voltage_Law. Accessed on 25 February 2019.