Lab on The Series RL, RC and RLC Circuit and Resonance

Purpose:

1. To measure the value of a capacitor C and Inductor L using a series RC & RL circuits, in series with a sinusoidal voltage source.

2. To study the phase relationships between Voltage and Current for R, L and C.

3. To measure the resonance frequency of a series RLC circuit and compare with the theoretical value.

Introduction

This online lab exercise will allow you to simulate the time dependence of the current in a series RL circuit, a series RC circuit and finally a series RLC circuit, in each case driven by an *alternating voltage source*, using the online circuit-building tool at www.falstad.com/circuit. You will be able to view the time dependence of the voltage and current (and their phase relationship) for all 3 components in the circuit.
The simulator also allows you to change the values of R (resistance), C (capacitance), and L (inductance) and observe resulting changes in the resonant frequency of the series RLC circuit.

Software

This lab runs in any web browser. Google Chome is recommended. It can be run online or downloaded for offline viewing. The web address is: www.falstad.com/circuit

You can download an offline version as follows:

Mac: http://falstad.com/circuit/CircuitJS1.dmg

MS Windows: http://falstad.com/circuit/circuitjs1.zip

Downloading the program allows it to work faster and you may encounter fewer delays caused by network speed. If you do not want to download it, you can work online on the website. Whether running online or offline, the actual procedure of conducting the simulation and analysis is the same.

Running the experiment

Open the program: circuitjs1.exe (Or circuitjs1.dmg for mac, Or use the online version).

Part 1, Measuring C

1) Click *Circuits* in the top menu and click A/C circuits, then select Capacitor.

2) Let's set the capacitor value, C = 2 μF. To do this, move the mouse over the capacitor till its color becomes light blue, then right click it and edit and put the value as 2 micro Farads, i.e. type 2u (and it already knows that the unit is F:Farad) and click apply then click *OK*.

Now, let's set the resistance value, R = 200 Ohms. To do this, move the mouse over the Resistor till its color becomes light blue, then right click it and edit and put the value as 200 (and it already knows that the unit is Ω) and click apply then click *OK*.   This

1

Finally, let's set up an AC voltage source with amplitude 1 Volt and frequency 200 Hz, as follows: Move the mouse over the *Function Generator* (the A/C voltage source), till its color becomes light blue, then right click it and change the value of *Max Voltage* to 1 (and it already knows that the unit is v: Volts). Keep the type of the wave form as is (A/C), and put the value of the frequency as 200 (and it already knows that the unit is Hz: Hertz). Keep the *Phase offset* (degrees) as is (the default of zero), and then click apply then click *OK*.

3) Let's set up the way in which the waveforms are displayed at the bottom of the screen (the "Dock). In the graph area, at the bottom, click the gear icon (or right click in that area), and set the *scroll speed* to 2 ms/div (using the arrow).

4) To view the properties of the resistor, voltage source and capacitor:
- move the mouse over the Resistor until its color becomes light blue, then right click it and edit and select *view in scope*.
- Do the same for the function generator (the A/C voltage source).
- The scope for the capacitor voltage is already available at the bottom (the Dock). If you have deleted by accident, view it in the scope as well by doing the same procedure as for the resistor and function generator.

Make the Simulation speed *medium*, in the upper right corner, then click *Run/Stop*, and after about 4 cycles, click the *Run/Stop* again to stop the running.

5) Move the mouse to:

- the graph of the capacitor output in order to find the value of the voltage at a peak (say the third peak), and *also record the time* at that peak, $t_{V,C}$. **Note that** if you want to increase the number of decimal places in the time reading you can decrease the **scroll speed** per div(ision) in the properties of each graph (this makes the time scale more sensitive).
- the resistor graph to do the same thing, i.e. find the peak value of the voltage across the resistor, and the time at that peak, $t_{V,R}$. Make sure that you examine the **same peak for the capacitor and the resistor.** In other words, if you are looking the 3rd set of peaks for the capacitor, you must also look at the 3rd peak for the resistor. This is so that you can compare times within the same cycle.

We expect that $C = \frac{V_{R,max}}{V_{C,max}} \times \frac{1}{\omega R}$ where $\omega = 2\pi f$, f is the frequency, in this procedure $f = 200$ Hz.

From the above formula, **calculate the "observed" value of capacitance, C** and compare it with the capacitance that you set, which was 2 µF.
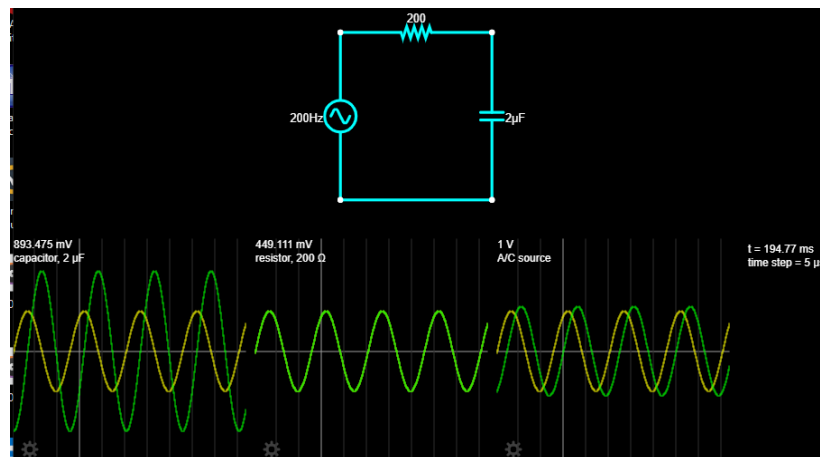

Figure 1: Example RLC circuit (top) with an AC voltage source. Graphs of the voltage across, and current through, the three components in the circuit are shown in the "Dock" below the circuit diagram.

6) Calculate the difference in the times you recorded in step 5 above for the peaks of V_R and V_C:

$$\Delta t = t_{V,R} - t_{V,C}$$

Using this time, find the phase difference (phase angle) between V_R and V_C by using the equation:

$$\Delta\phi = \omega\Delta t = 2\pi f \Delta t$$

You should find that, for the resistor the voltage curve and the current curve are in phase (i.e. 0 phase difference) and so you cannot "see" the voltage and current curves separately, as they overlap perfectly. But $I_R = V_R/R$, so I_R should be of different magnitude to V_R. The simulator plots V_R and I_R on top of each other which is probably a problem that sometimes occurs with the simulator. It is true that they are in phase, but they have different values. If you want to see I_R, right click the graph of R and select properties and uncheck show voltage and leave show current checked.

Part 2, Measuring L

1) You will now repeat the procedures of part 1, but with an inductor. Open the program, Click *Circuits* and click *A/C circuits* and select *inductor* (instead of *capacitor*). Set up a 0.1 Henry inductor by changing the value of the inductor to 0.1 (and it already knows that it is in H: Henry). The resistor can be 200 Ohms, as before. The power source should also have the same maximum voltage (1 Volt) and frequency (200 Hz) as before.

Observe the maximum in the voltage across the inductor, $V_{L,max}$ and, again, the maximum in the voltage across the resistor, $V_{R,max}$.

To "find" L, we can use the theoretical equation $L = \dfrac{V_{L,max}}{V_{R,max}} \times \dfrac{R}{\omega}$. Compare your finding with the value that you set (0.1 H).

2) As in Part 1, find the phase angle (phase difference) between the voltage of the inductor and the voltage of the resistor. Again, make sure to examine the same peaks in the "scope" output for an accurate comparison of the time within the same cycle. Which voltage is leading? Is V_L leading V_R or is V_R leading V_L?

Part 3, Resonance in a series RLC circuit

1) Click *Circuits* in the top menu, then *A/C* then select *Capacitor*. Set the values as shown in Figure 3. The max (peak) voltage for V_{source} should now be **5 Volts**.

We are going to delete the bottom wire and replace it by an inductor L. To do that, move the mouse to the bottom wire and right click and select *delete*. Then, Click *Draw* in the top menu and select *Passive components* and select *Add inductor*. Place it in the place of the wire that we deleted by clicking and dragging from the bottom end of the capacitor to the bottom end of the voltage source. **Make sure that the inductor windings are as shown in Figure 3; if not then right click the inductor and select swap terminals**. Change its value to 0.1 (it knows the unit: H).
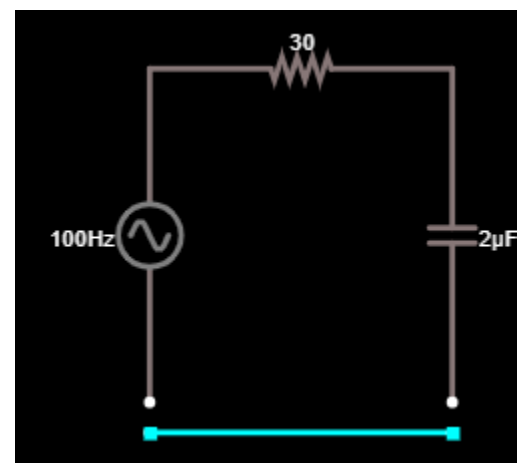

Figure 2: resistor and capacitance values for Part 3, shown before adding an inductor

So as not to keep adding more inductors, in Draw select the last item **Select/Drag Sel.** Now the mouse is back to normal.
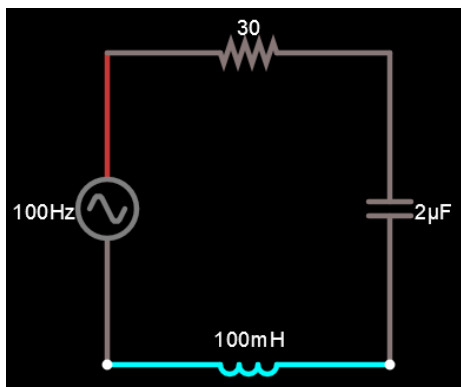

Figure 3: the completed series RLC circuit, showing the 0.1 Henry inductor on the bottom

2) Click the settings of the graph in the bottom Dock and change the *Scroll speed* to 2ms/ div. Move the mouse to the inductor until its color becomes light blue, and right click it and select *view in scope*, and do the same for the resistor. This is to show the voltage wave form across the inductor (green curve) and also the current wave form in the inductor (yellow curve) and voltage and current for R. *What is the relation of this current in R to the current in L and in C in this circuit? Why?*

3) Run the simulator by clicking *Run/Stop*. Keep running the simulation **until the curves become smooth.** You can increase the speed of the simulation using *Simulation speed* at the top right of the window. Stop the simulation by clicking *Run/Stop*.
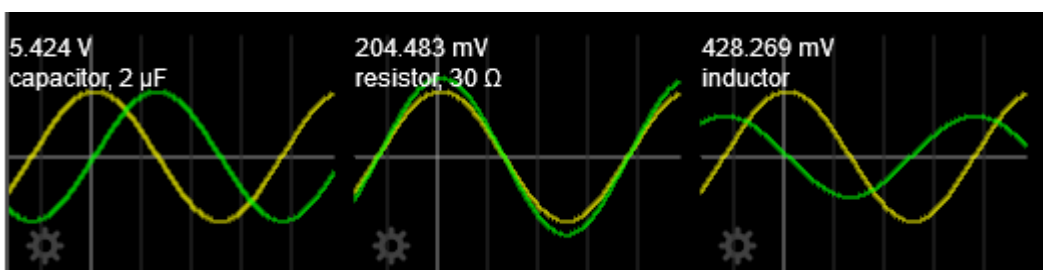

Figure 4: Scope traces for the three components in Part I3

Measurement

- Find the peak value of the current (yellow curve) by placing the mouse on the peak of the yellow current curve and reading the value of the current displayed by the simulator at the peak, where you have placed the mouse. (Remember: the current through the circuit is the same for all 3 devices since they are in series!)
- Repeat this measurement of peak current with the frequency of the source changed to *300 Hz*. Each time, record the peak (Maximum) of the **current** curve. **You need to run, stop and then re-run for each separate measurement.**
- Then repeat for f= 356 Hz, f= 400 Hz and 500 Hz. Finally try at f= 350 Hz and then 360 Hz. Use the table below to log the variation in peak current, I_{max}.

Analysis

(a) Plot a graph of the peak value of the current versus frequency. You can add more values of the frequency if you want to, by taking more measurements as above. Find the "resonance" frequency from your graph – the frequency at which the peak current is maximum – and compare with the calculated resonance frequency for this circuit using

$$X_{L\ at\ res} = X_{C\ at\ res} \quad \rightarrow \quad f_{resonance} = \frac{1}{2\pi\sqrt{LC}}.$$

(b) For any frequency (say 100 Hz), find the phase difference angle between the voltage across L (V_L), and the voltage across C (V_C), by recording the times of corresponding peaks (same peaks, say the first peak after the vertical line of the y axis **in both graphs**) using the mouse and then using $\Delta\phi = \omega\Delta t = 2\pi f\Delta t$. As in the rest of this lab, you must use same corresponding peaks to be able to find corresponding phase difference. To emphasize, same corresponding peaks mean: if you use the first +ve peak of V_C after the vertical line of the y

4

axis you must also use the first +ve peak of the V_L after the vertical line of the y axis to record the correct corresponding times to find Δt. You can *run* and *stop* the simulator until you are able to see simultaneously displayed in the scopes the same corresponding peaks for V_C and V_L voltages (green curves for C and L), as in the example shown in Figure 4. Which is leading V_L or V_C? **Note that** if $\Delta t= 0$ s, then something is wrong. Review step 1 in part 3 concerning the windings of the inductor.

(c) What is the phase difference between the I_R, the current through the resistor and the V_R, the voltage across R?

f (Hz)	Max (peak) current
100	
300	
350	
356	
360	
400	
500	

f$_{resonance}$ from graph = ……………. Hz

f$_{resonance}$ from calculation = ……………….. Hz

Show your work for f$_{resonance}$ calculation:

You can copy this table and values into your lab report along with all the other requirements of the lab report: the graphs of all parts, calculations,….etc.