Name: ______________________

Series LC Circuit Worksheet

Introduction

Understanding the behavior of reactive components such as capacitors and inductors based on their complex impedances is a critical skill in electrical engineering. In this lesson we will analyze the behavior of one such circuit using Laplace transforms.

Discussion Overview

As discussed in the lecture, LC circuits tend to oscillate at certain frequencies determined by the combination of the capacitor's and inductor's impedances. To better understand the frequency response of such circuits, we will examine the following circuit.

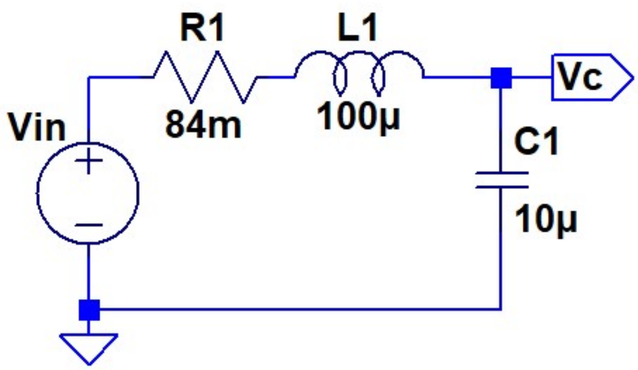


Figure 1 - Series LC Circuit

In order to determine the voltage Vc across the capacitor, we apply the voltage divider equation using the impedances of R1, L1 and C1. Note that resistor R1 represents the series resistance of the inductor; and L1 and C1 are treated as ideal inductor and capacitor respectively.

Recall that the impedance of an inductor is given as $Z_L = sL$, and that of a capacitor is given as $Z_C = \frac{1}{sC}$. Therefore,

$$V_C = V_{in}\frac{Z_C}{Z_C + R + Z_L} = V_{in}\frac{\frac{1}{sC}}{\frac{1}{sC} + R + sL} = V_{in}\frac{1}{1 + sCR + s^2CL}$$

To examine the behavior of the circuit in frequency domain, we let $s = j\omega$:

$$V_C = V_{in}\frac{1}{1 + j\omega CR - \omega^2 CL} = V_{in}\frac{1}{1 - \omega^2 CL + j\omega CR}$$

The magnitude of the voltage across the capacitor is, therefore, given by

$$|V_C| = \left|V_{in}\frac{1}{1-\omega^2 CL + j\omega CR}\right| = |V_{in}|\frac{1}{\sqrt{(1-\omega^2 CL)^2 + (\omega CR)^2}} \qquad \textbf{Eq. 1}$$

As seen in Eq. 1, Vc is a function of frequency ω $(= 2\pi f)$. We will now examine how Vc changes as a function of frequency. This is called the frequency response of the circuit.

Frequency Response

We will examine Eq. 1 for three different regions in frequency domain; low frequencies, high frequencies and the so called "corner frequency".

Low Frequencies

First, we note that for lower frequencies close to zero, the denominator in Eq. 1 is approximately one.

$$\left.\sqrt{(1-\omega^2 CL)^2 + (\omega CR)^2}\right|_{\omega\to 0} \approx 1$$

Therefore, for lower frequencies, the voltage across the capacitor is approximatley the same as the input voltage:

$$V_C|_{\omega\to 0} \approx V_{in}$$

We note that this behavior is in line with our expectations for very low frequency signals including DC. For low frequencies, an inductor acts like a short while a capacitor acts like an open circuit which would lead to $V_C = V_{in}$.

High Frequencies

The second region of interest is for very large frequencies. For large frequencies, the denominator in Eq. 1 is dominated by ω, and as ω gets larger, the fraction goes to zero.

$$\left.\sqrt{(1-\omega^2 CL)^2 + (\omega CR)^2}\right|_{\omega\to\infty} \approx \sqrt{(\omega^2 CL)^2 + (\omega CR)^2} \to \infty \Rightarrow \left.\frac{1}{\sqrt{(1-\omega^2 CL)^2 + (\omega CR)^2}}\right|_{\omega\to\infty} \approx 0$$

Therefore, for large frequencies, the voltage across the capacitor tends towards zero:

$$V_C|_{\omega\to\infty} \approx 0$$

This behavior, again, is in line with our expectations. For large frequencies, an inductor acts like an open circuit while a capacitor acts like a short leading to $V_C = 0$.

Name: ______________________

Resonance Frequency

The final region of interest is around the frequency $\omega = \frac{1}{\sqrt{LC}}$. For $\omega = \frac{1}{\sqrt{LC}}$, the term $(1 - \omega^2 CL)$ is equal to zero, and assuming that R is fairly small, this would lead to a very large value for the fraction in Eq. 1. We can find the exact value of V_C by plugging $\omega = \frac{1}{\sqrt{LC}}$ in Eq. 1:

$$V_C|_{\omega=\frac{1}{\sqrt{LC}}} = V_{in}\frac{1}{R}\sqrt{\frac{L}{C}}$$

which for $R \to 0$ would result in a large V_C.

As the name suggests, the circuit starts oscillating around the resonance frequency $\omega = \frac{1}{\sqrt{LC}}$.

Frequency Response Plot

The plot in Figure 2 shows the frequency response of the circuit in Figure 1 assuming $V_{in} = 1$. For the values of R, L and C given in Eq. 1, the resonance frequency is

$$f = \frac{\omega}{2\pi} = \frac{1}{2\pi\sqrt{LC}} = 5.03KHz \qquad \text{Eq. 2}$$

and

$$V_C|_{\omega=\frac{1}{\sqrt{LC}}} = V_{in}\frac{1}{R}\sqrt{\frac{L}{C}} \approx 31.5dB$$

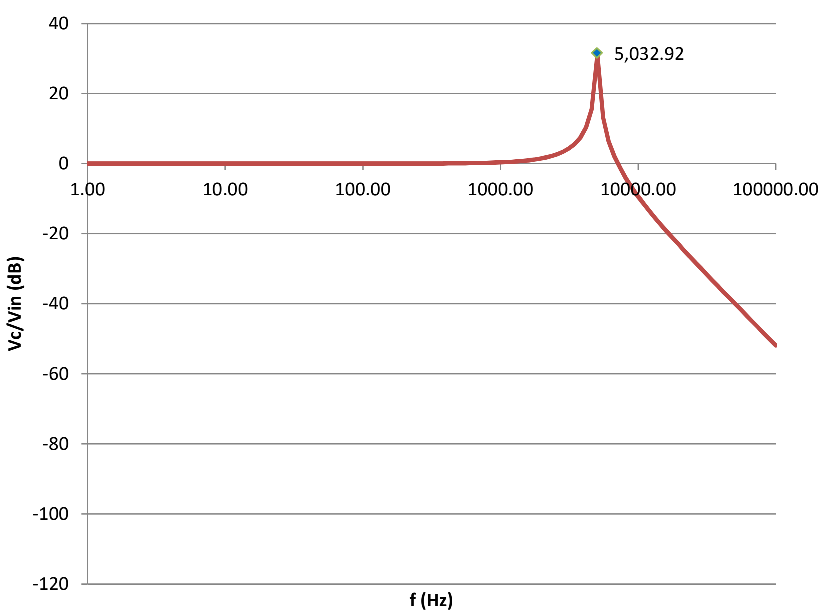


Figure 2 - Frequency Response of an LC Circuit

Procedure

In this section, you are asked to create a SPICE model and also build the circuit shown in Figure 3. Note that the series resistance of the inductor will be specified as part of the inductor's parameters.

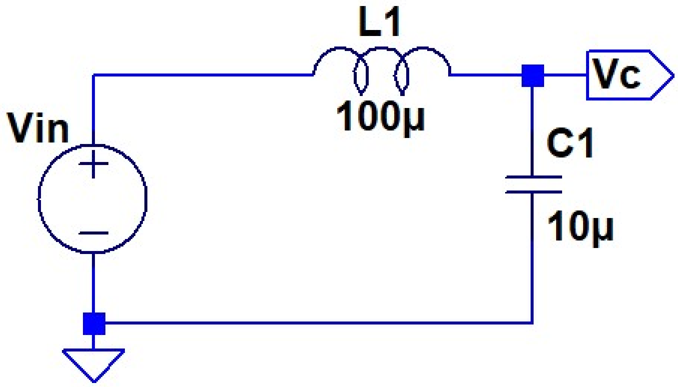
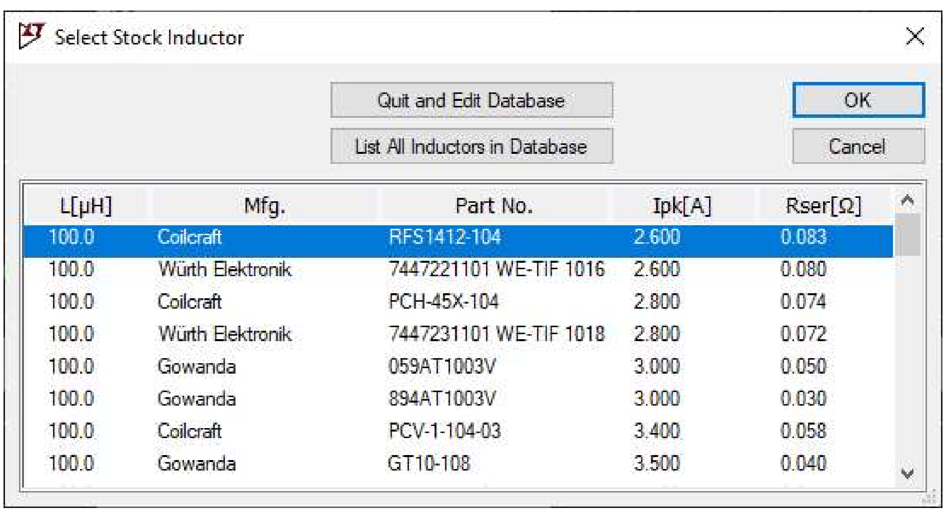


Figure 3 - LTSpice LC Circuit

LTSpice Model

A. Capture the circuit in Figure 3 in LTSpice.
B. When selecting a value for L1, chose a part with 100μH of inductance and a series resistance of $Rser \approx 0.083\Omega$ as shown below.



Select Stock Inductor

Quit and Edit Database

List All Inductors in Database

OK

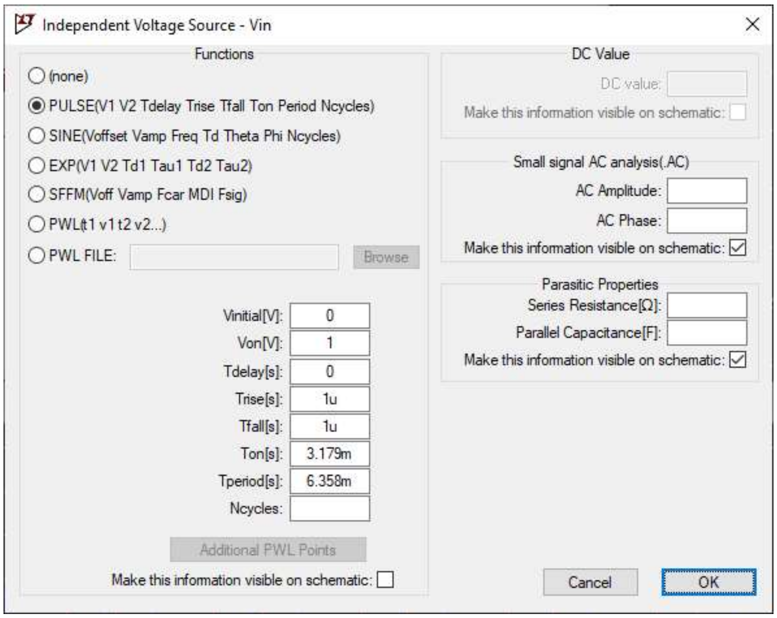
Cancel

L[µH]	Mfg.	Part No.	Ipk[A]	Rser[Ω]
100.0	Coilcraft	RFS1412-104	2.600	0.083
100.0	Würth Elektronik	7447221101 WE-TIF 1016	2.600	0.080
100.0	Coilcraft	PCH-45X-104	2.800	0.074
100.0	Würth Elektronik	7447231101 WE-TIF 1018	2.800	0.072
100.0	Gowanda	059AT1003V	3.000	0.050
100.0	Gowanda	894AT1003V	3.000	0.030
100.0	Coilcraft	PCV-1-104-03	3.400	0.058
100.0	Gowanda	GT10-108	3.500	0.040

Note that selecting this part, sets the inductor's series resistance to 83mΩ

C. Set your voltage source to a "PULSE" source with the following parameters
    a. Initial voltage = 0
    b. Von = 1V

c. Delay = 0
d. Rise time = 1μs
e. Fall time = 1μs
f. On time = 3.179ms
g. Period = 6.358ms



D. Setup the model to run a “transient” simulation with the following parameters
   a. Starting the print at time 0,
   b. Ending at time 3.179ms,
   c. Starting capture at time 0,
   d. With a maximum simulation step size of 1μs, and
   e. Setting the external voltage sources to 0 at the “startup”

   Below is the syntax for your reference:

   .tran <Tprint> <Tstop> [<Tstart> [<Tmaxstep.]] [<options>]

E. Run the simulation and display the waveform for the voltage Vc.

F. What does the waveform look like?

Name: ______________________

G. Estimate the frequency of the waveform and record it here

$f =$ ____________________ Hz

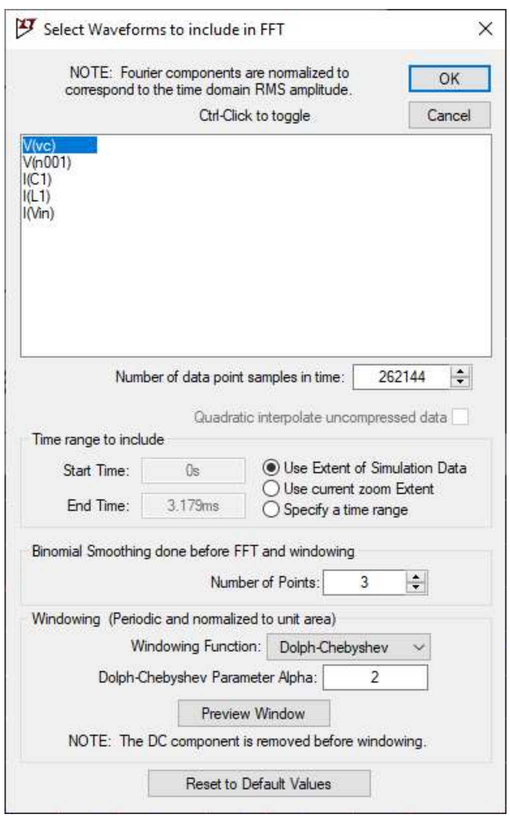
Frequency Response

In this section, we will use the FFT tool of LTSpice to plot the frequency response of the voltage across the capacitor. FFT stands for Fast Fourier Transform, and it is a mathematical algorithm for extracting the various frequency components present in a signal. (The details of the FFT algorithm are beyond the scope of this level.)

H. Right click on the waveform window and select View → FFT

I. From the FFT window, select V(vc) to display

J. Leave all the settings as default except for the following:
   a. Set the “Windowing Function” to “Dolph-Chebyshev”, and
   b. Set the “Dolp-Chebyshev Parameter Alpha” to 2.



K. Click on OK to run the FFT.

You should now see a waveform similar to the one given in Figure 2.

Name: ______________________

L. Measure the frequency at which the response peaks and record it below.

$f =$ ____________________ Hz

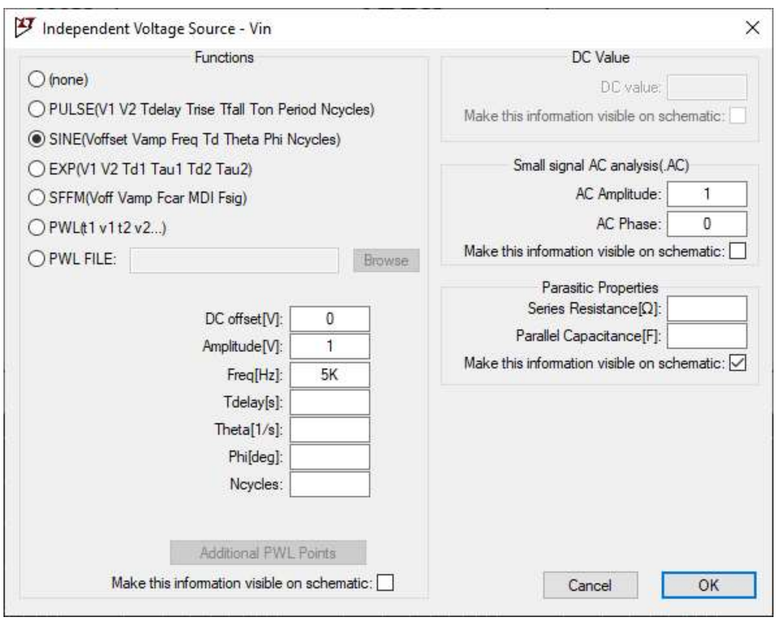
M. How does this value compare with the calculated value in Eq. 2 or the estimated value in step G?

N. Save your schematic

AC Analysis

In this section, we use the AC Analysis of SPICE to simulate the circuit over a range of frequencies and examine the frequency response of the circuit.

O. Save your circuit as a new schematic
P. Change the voltage source to a sine wave function and configure it with the values given below.
   a. DC offset = 0
   b. Amplitude = 1
   c. Frequency = 5K
   d. AC Amplitude = 1
   e. AC Phase = 0
   f. All the other parameters should be left blank



Note: For AC Analysis, the parameters under "Small signal AC analysis (.AC)" need to be set.

Q. Change the Transient directive to the following AC Analysis directive
   a. Type of sweep: Decade
   b. Number of points per decade: 100
   c. Start frequency: 1
   d. Stop frequency: 100K

   Below is the syntax for your reference

   .ac <oct, dec, lin> <Npoints> <StartFreq> <EndFreq>

   Note: Setting the above parameters, you are directing SPICE to run the simulations by sweeping the frequency of your voltage source (sine wave) from 1Hz (start frequency) to 100KHz (stop frequency) with 100 points between each decade.

R. Run the simulation and probe Vc.
S. Measure the frequency at which the response peaks and record it below.

$f =$ ____________________ Hz

T. How does this value compare with the calculated value in Eq. 2 or the estimated value in step G?

U. How does the frequency response plot compare with the one given in Figure 2?

Name: ________________________

Lab Build and Measurements

In this section, you are asked to build the circuit in Figure 3 and make measurements using an oscilloscope.

A. Note the values of your inductor and capacitor and record them here

$C =$ ____________________F

$L =$ ____________________H

B. Measure the series resistance of the inductor and record it here

$R_L =$ ________________Ω

C. Build the circuit in Figure 3 on a breadboard.

D. Set the waveform generator on your oscilloscope to a square wave with the following parameters:
   a. Amplitude = 0.5V ($V_{pk-p} \;\; = 1V$)
   b. Offset = 0.5V
   c. Frequency = 160Hz

E. Use the waveform generator as the input source to your circuit.

F. Connect the oscilloscope to observe the voltage across your capacitor. Here are the suggested initial settings:
   a. Ch. 1 volts/div = 0.5V
   b. Horizontal sec/div = 2ms
   c. Trigger set to the leftmost location on the screen (~-16ms)

G. What does the waveform look like?

H. Estimate the frequency of the waveform and record it here

$f =$ _____________________Hz

I. Use the “Math” function to plot the FFT of the signal.

Name: ______________________

J. Measure the frequency at which the response peaks and record it below.

$f =$ ____________________ Hz

K. How does this value compare with the calculated value in Eq. 2 or the estimated value in step H above?