

# Parallel LC Circuit Worksheet

## Introduction

In the “Series LC Circuit Lab”, we learned that the impedance of the circuit at the resonant frequency was nearly zero which made it an impractical circuit to test and measure in the lab. In this lab, we consider a parallel LC circuit and its impedance.

## Discussion Overview

In order to find the impedance of the circuit below, we note that  $R_L$  and  $L$  are in series, and the combination of the two is in parallel with  $C$ .

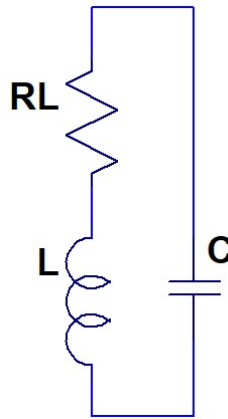


Figure 1 - Parallel LC Circuit

Therefore, the total impedance of the circuit can be found as follows

$$\frac{1}{Z_T} = \frac{1}{R_L + sL} + \frac{1}{sC} = \frac{1}{R_L + sL} + sC = \frac{1 + sR_L C + s^2 LC}{R_L + sL}$$

$$\Rightarrow Z_T = \frac{R_L + sL}{1 + sR_L C + s^2 LC}$$

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

$$Z_T = \frac{R_L + j\omega L}{1 - \omega^2 LC + j\omega R_L C}$$

As seen from the equation above, the resonant frequency is at the value where the denominator of the impedance is very small, or when

$$\omega = \frac{1}{\sqrt{LC}}$$

Therefore, at the resonant frequency,

$$Z_T|_{\omega=\frac{1}{\sqrt{LC}}} = Z_{T_o} = \frac{R_L + j\frac{1}{\sqrt{LC}}L}{1 - \left(\frac{1}{\sqrt{LC}}\right)^2 LC + j\frac{1}{\sqrt{LC}}R_L C} = \frac{R_L + j\sqrt{\frac{L}{C}}}{1 - 1 + j\sqrt{\frac{C}{L}}R_L} = \frac{R_L + j\sqrt{\frac{L}{C}}}{j\sqrt{\frac{C}{L}}R_L} = \frac{L}{R_L C} - j\sqrt{\frac{L}{C}}$$

And, finally, the magnitude of the total impedance at the resonant frequency is

$$|Z_{T_o}| = \sqrt{\left(\frac{L}{R_L C}\right)^2 + \frac{L}{C}}$$

Assuming  $R_L$  is small, equation above is approximately given by

$$|Z_{T_o}| \approx \frac{L}{R_L C}$$

Eq. 1

As seen in Eq. 1, the magnitude of the total impedance at the resonant frequency can become very large if  $R_L$  is small. For example, if we let  $L = 100\mu H$ ,  $C = 10\mu F$  and  $R_L = 0.08\Omega$ , the total impedance would be

$$|Z_{T_o}| = 125$$

Therefore, if we can build a current source that would push 100mA through the circuit above, we could get an oscillating 12.5V across the circuit.

$$V = I|Z_{T_o}| = 0.1 \times 125 = 12.5V$$

## Procedure

In this section, you will first use a current source in SPICE to drive a parallel LC impedance. You will examine the behavior of this simple circuit. Next using a simple transistor circuit, you will implement the current source and will again examine its behavior driving a parallel LC impedance. Lastly, you will build the circuit in the lab and make measurements to compare with your simulation results.

## Simple Current Source

Build the circuit shown in Figure 2. Note that the series resistance of the inductor will be specified as part of the inductor's parameters.

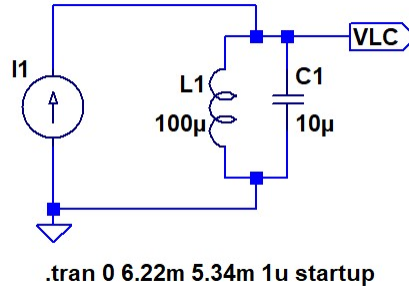


Figure 2 - LTSpice Parallel LC Circuit

Also, note that for the values above, the resonant frequency is

$$f = \frac{\omega}{2\pi} = \frac{1}{2\pi\sqrt{LC}} \approx 5.033\text{KHz}$$

Eq. 2

## LTSpice Model

- A. Capture the circuit in Figure 2 in LTSpice. To place a current source in the circuit search for “current” in the “Add Parts” window.
- B. When selecting a value for L1, chose a part with 100μH of inductance and a series resistance of  $R_{ser} \approx 0.083\Omega$  as shown below.

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 current source to a “PULSE” source with the following parameters
  - a.  $I_1 = 0$

Name: \_\_\_\_\_

- b.  $I_2 = 1\text{V}$
- c. Delay = 0
- d. Rise time =  $1\mu\text{s}$
- e. Fall time =  $1\mu\text{s}$
- f. On time =  $0.888\text{ms}$
- g. Period =  $1.78\text{ms}$

Independent Current Source - I1

Functions

☐ (none)

☒ PULSE(I1 I2 Tdelay Trise Tfall Ton Period Ncycles)

☐ SINE(Ioffset Iamp Freq Td Theta Phi Ncycles)

☐ EXP(I1 I2 Td1 Tau1 Td2 Tau2)

☐ SFFM(Ioff Iamp Fcar MDI Fsig)

☐ PWL(I1 I1 t2 I2...)

☐ PWL FILE:  Browse

☐ TABLE(v1 i1 v2 i2...)

I1[A]:

I2[A]:

Tdelay[s]:

Trise[s]:

Tfall[s]:

Ton[s]:

Tperiod[s]:

Ncycles:

Additional PWL Points

Make this information visible on schematic: ☐

DC Value

DC value:

Make this information visible on schematic: ☐

Small signal AC analysis (AC)

AC Amplitude:

AC Phase:

Make this information visible on schematic: ☒

Parasitic Properties

This is an active load: ☐

Make this information visible on schematic: ☒

Cancel OK

- D. Setup the model to run a “transient” simulation with the following parameters
- Starting the print at time 0,
  - Ending at time 6.22ms,
  - Starting capture at time 5.34ms,
  - With a maximum simulation step size of  $1\mu\text{s}$ , and
  - 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 across the parallel LC circuit.
- F. What does the waveform look like?

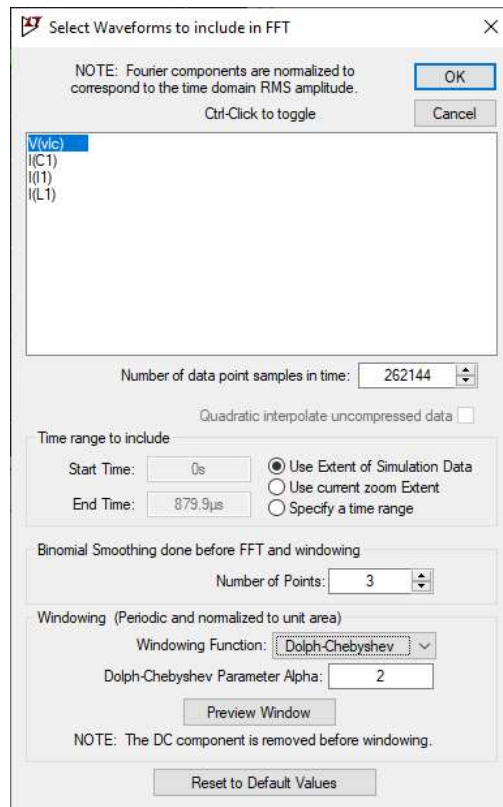
G. Estimate the frequency of the waveform and record it here

$$f = \text{_____} Hz$$

## 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 “Dolph-Chebyshev Parameter Alpha” to 2.



K. Click on OK to run the FFT.

Name: \_\_\_\_\_

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

$$f = \text{_____} \text{ Hz}$$

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 current source to a sine wave function and configure it with the values given below.

- DC offset = 0
- Amplitude = 100m
- Frequency = 5K
- AC Amplitude = 100m
- AC Phase = 0
- All the other parameters should be left blank

Independent Current Source - I1

Functions

☐ (none)

☐ PULSE(I1 I2 Tdelay Trise Tfall Ton Period Ncycles)

☒ SINE(I1 I2 Freq Td Theta Phi Ncycles)

☐ EXP(I1 I2 Td1 Tau1 Td2 Tau2)

☐ SFFM(I1 I2 Fcarr Fmod MDI Fsig)

☐ PWL(I1 I2 t1 t2 t3...)

☐ PWL FILE:

☐ TABLE(v1 i1 v2 i2...)

DC offset[A]:

Amplitude[A]:

Freq[Hz]:

Tdelay[s]:

Theta[1/s]:

Phi[deg]:

Ncycles:

Make this information visible on schematic: ☐

DC Value

DC value:

Make this information visible on schematic: ☐

Small signal AC analysis (AC)

AC Amplitude:

AC Phase:

Make this information visible on schematic: ☐

Parasitic Properties

This is an active load: ☐

Make this information visible on schematic: ☒

Name: \_\_\_\_\_

*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
- Type of sweep: Decade
  - Number of points per decade: 100
  - Start frequency: 1
  - Stop frequency: 1Meg

*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 1MHz (stop frequency) with 100 points between each decade.*

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

$f = \text{_____} \text{ Hz}$

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

## Implemented Current Source

Build the circuit shown in Figure 3.

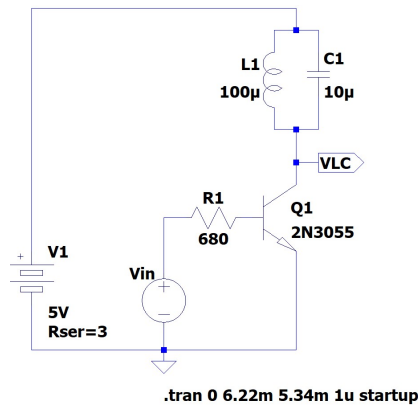
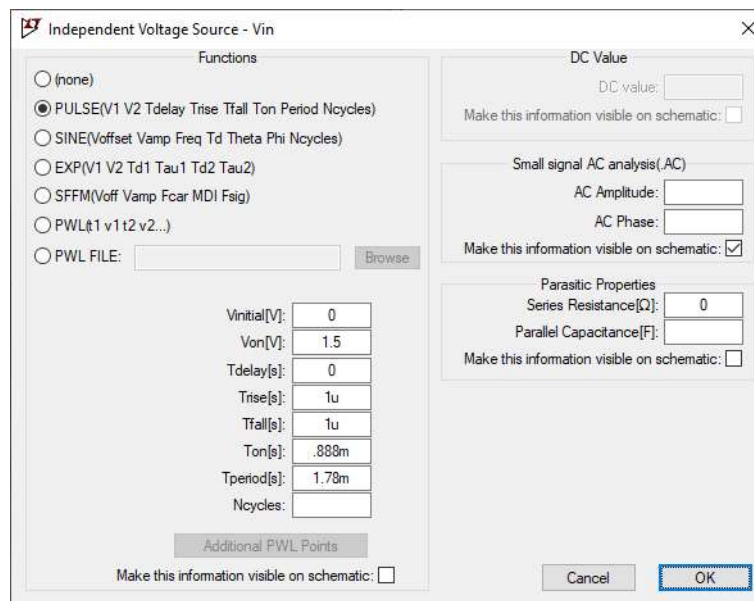


Figure 3 - LTSpice Parallel LC Circuit

## LTSpice Model

- A. Set your voltage source to a “PULSE” source with the following parameters
  - a. Initial Voltage = 0
  - b. On Voltage = 1V
  - c. Delay = 0
  - d. Rise time = 1μs
  - e. Fall time = 1μs
  - f. On time = 0.888ms
  - g. Period = 1.78ms





- B. Setup the model to run a “transient” simulation with the following parameters
- Starting the print at time 0,
  - Ending at time 6.22ms,
  - Starting capture at time 5.34ms,
  - With a maximum simulation step size of 1 $\mu$ s, and
  - Setting the external voltage sources to 0 at the “startup”

*Below is the syntax for your reference:*

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

- C. Run the simulation and display the waveform for the voltage across the parallel LC circuit.
- D. What does the waveform look like?

- E. Estimate the frequency of the waveform and record it here

$f = \underline{\hspace{2cm}} \text{ Hz}$

## 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. Save your circuit as a new schematic.

- F. Change the voltage source to a sine wave function and configure it with the values given below.
- DC offset = 1.25
  - Amplitude = 0.5
  - Frequency = 5K
  - AC Amplitude = 0.5
  - AC Phase = 0
  - All the other parameters should be left blank

Name: \_\_\_\_\_

Independent Voltage Source - Vin

Functions

- ☐ (none)
- ☐ PULSE(V1 V2 Tdelay Trise Tfall Ton Period Ncycles)
- ☒ SINE(Voffset Vamp Freq Td Theta Phi Ncycles)
- ☐ EXP(V1 V2 Td1 Tau1 Td2 Tau2)
- ☐ SFFM(Voff Vamp Fcar MDI Faig)
- ☐ PWL(t1 v1 t2 v2...)
- ☐ PWL FILE:  Browse

DC offset[V]: 1.25

Amplitude[V]: 0.5

Freq[Hz]: 5K

Tdelay[s]:

Theta[1/s]:

Phi[deg]:

Ncycles:

Additional PWL Points

Make this information visible on schematic: ☐

DC Value

DC value:

Make this information visible on schematic: ☐

Small signal AC analysis(.AC)

AC Amplitude: 0.5

AC Phase:

Make this information visible on schematic: ☐

Parasitic Properties

Series Resistance[Ω]: 0

Parallel Capacitance[F]:

Make this information visible on schematic: ☐

Cancel OK

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

- G. Change the Transient directive to the following AC Analysis directive
- Type of sweep: Decade
  - Number of points per decade: 100
  - Start frequency: 1
  - Stop frequency: 1Meg

*Below is the syntax for your reference*

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

- H. Run the simulation and probe VLC.
- I. Measure the frequency at which the response peaks and record it below.

$f = \text{_____} \text{ Hz}$

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

## 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

$$C = \text{_____} F$$

$$L = \text{_____} H$$

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

$$R_L = \text{_____} \Omega$$

- 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:

- Amplitude = 0.75V ( $V_{pk-pk} = 1.5V$ )
- Offset = 0.75V
- Frequency = 560Hz

- E. Use the waveform generator as the  $V_{in}$  input source to your circuit.

- F. Use a power supply as the  $V_1$  source for your circuit.

- Set the output voltage to 5V
- Set the current limit to 200mA

- G. Connect the oscilloscope to observe the VLC voltage. Here are the suggested initial settings:

- Ch. 1 volts/div = 0.5V
- Horizontal sec/div = 1ms
- Trigger set to the leftmost location on the screen (~8ms)

- H. What does the waveform look like?

- I. Estimate the frequency of the waveform and record it here

$$f = \text{_____} Hz$$

Name: \_\_\_\_\_

- J. Use the “Math” function to plot the FFT of the signal.
- K. Measure the frequency at which the response peaks and record it below.

$f =$  \_\_\_\_\_  $Hz$

- L. How does this value compare with the calculated value in Eq. 2 or the estimated value in step I above?

- M. Change the waveform generator output to a sine wave with the following parameters:

- Amplitude = 0.5V ( $V_{pk-p} = 1V$ )
- Offset = 1.25V
- Frequency = 100Hz

- N. Measure the peak to peak voltages at VLC and record them in the table below for the frequencies given in each row:

Frequency (Hz)	Amplitude at $V_{LC}$	$\frac{V_{LC}}{V_{in}}$	$10\log\left(\frac{V_{LC}}{V_{in}}\right)$
100			
500			
1.0K			
2.0K			
3.0K			
4.0K			
4.5K			
5.0K			
5.5K			

Name: \_\_\_\_\_

Frequency (Hz)	Amplitude at $V_{LC}$	$\frac{V_{LC}}{V_{in}}$	$10\log\left(\frac{V_{LC}}{V_{in}}\right)$
6.0K			
7.0K			
8.0K			
10.0K			
100.0K			
1.0M			

O. Plot the values of  $10\log\left(\frac{V_c}{V_{in}}\right)$  vs frequency below.

