Lab7 RLC Resonant Circuit Prelab Report

Name 杨润康 Student ID 2022533080

TA ______________ Score ______of 31.5pt

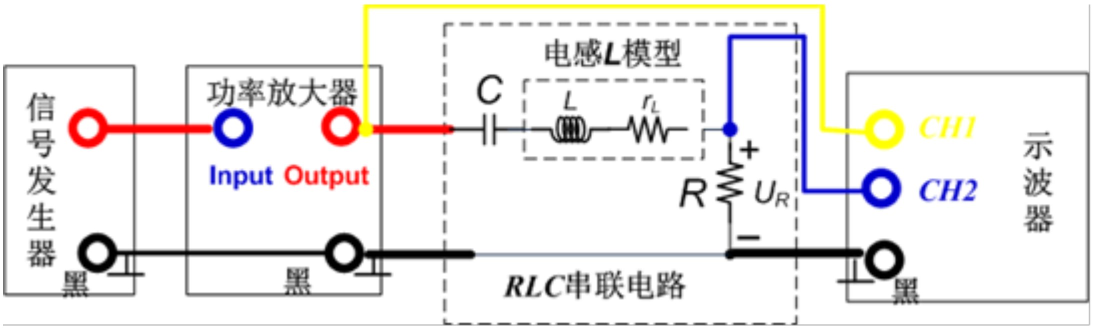


Fig.2 schematic diagram of RLC series circuit test

Part 1:

1. A shown in Figure 2, gain of the power amplifier is set to 1. Calculate the resonance frequency (not the angular frequency) f_0, cutoff frequencies f_1 and f_2, bandwidth and Q factor for each circuit. The R you're using in calculation should include the resistance of the inductor, because the inductor has some resistance, both magnitude and phase of the voltage across the inductor will not completely agree with the calculations presented. This will become more evident as the applied frequency is made lower. In addition, calculate U_{L0}, U_{C0} at resonance frequency to find out whether the magnitude of U_{L0}, U_{C0} is larger or smaller than Vin. Assume series resistance r_L of the 100mH inductor is about 42 Ω, **Show your calculation process.** __/15pt

C=0.5μF L=100mH R=300Ω	$f_0=\frac{1}{2\pi\sqrt{LC}}=711.763Hz$ $I=\frac{I_m}{\sqrt{2}}$ $R^2+(wL-\frac{1}{wC})^2=2R^2$ $w>0$ $\begin{cases} w_1=3077.91 Hz & w_2=6497.91 Hz \\ f_1=489.86 Hz & f_2=1034.17 Hz \end{cases}$ $f_{BW}=f_2-f_1=544.31 Hz$ $Q=\frac{f_0}{f_{BW}}=1.308$ $Z_C=\frac{1}{jwC}=447.214\angle-90\Omega$ $Z_L=jwL=447.214\angle 90^\circ\ \Omega$ $U_{L0}=U_{C0}=\frac{U_i}{R}\lvert Z\rvert=1.308U_{in}$
C=0.25μF L=100mH R=300Ω	$f_0=\frac{1}{2\pi\sqrt{LC}}=1006.584Hz$ $I=\frac{I_m}{\sqrt{2}}$ $R^2+(wL-\frac{1}{wC})^2=2R^2$ $w>0$ $\begin{cases} w_1=4841.65 Hz & w_2=8261.65 Hz \\ f_1=770.57 Hz & f_2=1314.88 Hz \end{cases}$ $f_{BW}=f_2-f_1=544.31 Hz$ $Q=\frac{f_0}{f_{BW}}=1.849$ $Z_C=\frac{1}{jwC}=632.456\angle-90\Omega$ $Z_L=jwL=632.456\angle 90^\circ\ \Omega$ $U_{L0}=U_{C0}=\frac{U_i}{R}\lvert Z\rvert=1.849U_i$

C=0.25μF L=100mH R=150Ω	$f_0 = \frac{1}{2\pi\sqrt{LC}} = 1006.584\ Hz$ $I = \frac{I_m}{\sqrt{2}}$ $R^2 + (\omega L - \frac{1}{\omega C})^2 = 2R^2$ $\omega > 0$ $\begin{cases} \omega_1 = 5437.00\ Hz & \omega_2 = 7357.00\ Hz \\ f_1 = 865.33\ Hz & f_2 = 1170.90\ Hz \end{cases}$ $f_{BW} = f_2 - f_1 = 305.58\ Hz$ $Q = \frac{f_0}{f_{BW}} = 3.294$ $Z_C = \frac{1}{j\omega C} = 132.456 \angle -90\ \Omega$ $Z_L = j\omega L = 132.456 \angle 90^\circ\ \Omega$ $U_{L0} = U_{C0} = \frac{U_i}{R}\lvert Z \rvert = 3.294\ U_{in}$

Part2: Simulating Circuits

Simulate the circuit in Figure 2 by MultiSim, in which consists of a 300Ω resistor, 100mH inductor and a 0.5μF capacitor (Power amplifier can be omitted when simulating). **Find the resonance frequency, cutoff frequencies, bandwidth and Q factor for each circuit.** To obtain accurate simulation results, perform the following steps：

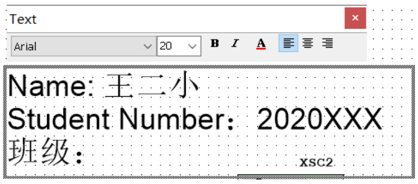
1. Under "Simulate->Analysis and Simulate", select "AC Sweep" and make sure everything is set how you want. The Frequency parameters tab contains the following:

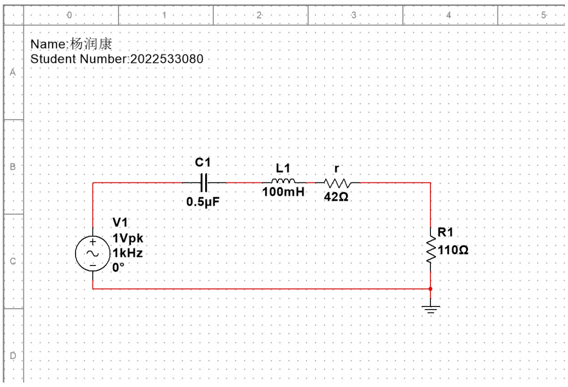
Start frequency	The start frequency for the sweep.
Stop frequency	The stop frequency for the sweep.
Sweep type	Defines how points to be calculated are distributed across the frequency range. Choices are Decade, Linear, or Octave.
Number of points	Determines the number of points to be calculated during the analysis. For Linear sweep, enter the total number of points between start and end. For Decade/Octave sweep, enter the number of points per decade/octave. The greater the number of points calculated, the more accurate the results will be. However, the simulation speed will be slower.
Vertical scale	Controls the y-axis scaling on the output graph. Choices are Linear, Logarithmic, Decibel, or Octave. To plot the output in decibels (dB), use the Decibel option in the Vertical scale option, rather than an expression such as 20*log(x) combined with the Linear option. Unless Decibel is selected, the magnitude plot only plots positive numbers. Negative numbers are captured by the phase plot. Therefore, a gain of -3dB appears as +3dB with a phase of -180 degrees.

2. Under "Frequency parameters", to investigate the frequency response of the circuit to sinusoidal excitation, specify the desired frequency range, assign reasonable values to

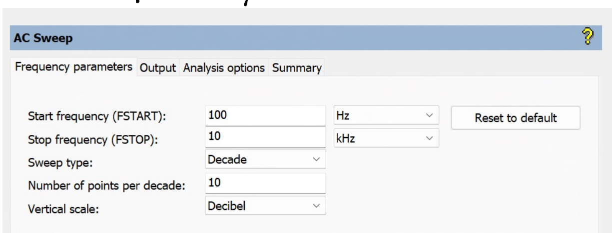
"Start frequency", "Stop frequency" and "Number of Points" to obtain accurate results.

3. Under "Vertical scale", select either "Linear" or "Decibel" which can best express your data.
4. Select the output in the output tab.
5. Take screenshots of your simulation schematic in the below. You should "Place-> Text "in your schematic which include your name and student number, as shown in the Figure below. __/5pt



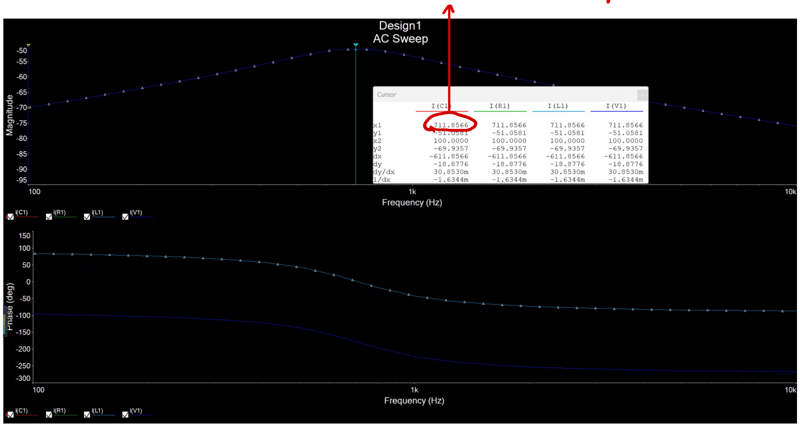


start and stop frequency I set in the "AC sweep"

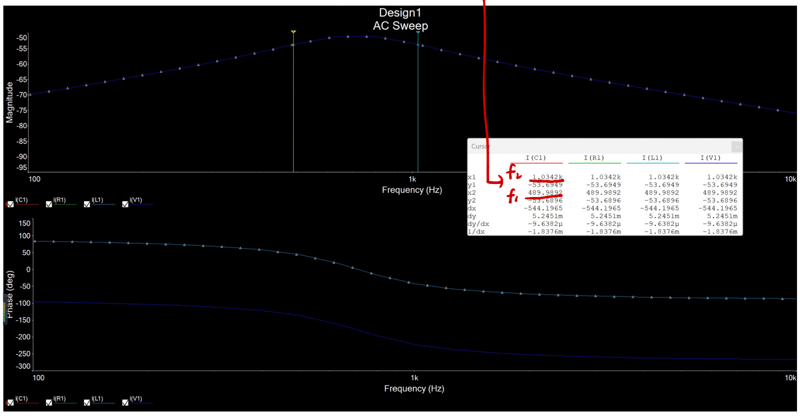


6. Finally, save and run the simulation. And you should take screenshots of the plots and data for both the phase and magnitude. With those outputs shown, you should use cursors to indicate the key parameters when the RLC circuit at the resonant state. __/5pt

resonance frequency f_0



cutoff frequency



Bandwith: $f_{BW} = f_2 - f_1 = 544.21$ Hz

$$Q = \frac{f_0}{f_{BW}} = \frac{711.8566}{1034.2 - 489.9892} = 1.308$$

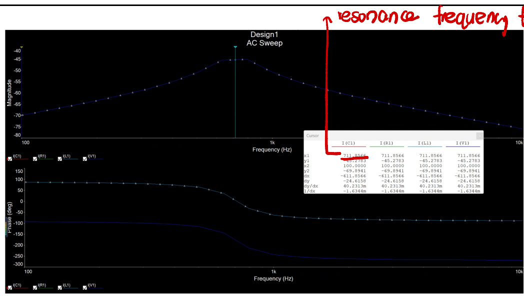
7. Note down the resonance frequency, cutoff frequencies, bandwidth from your simulation results. And show how you get Q factor from the simulation results. __/6.5pt

C=0.5µF

L=100 mH

R=110Ω

resonance frequency f_0



Bandwidth $f_{BW} = f_2 - f_1 = 240.4143$ Hz

$$Q = \frac{f_0}{f_{BW}} = \frac{711.8566}{842.1418 - 601.7275} = 2.961$$

cutoff frequency