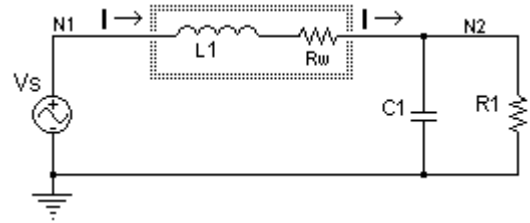


Experiment 20: Series-Parallel AC Circuit Measurements

Introduction

Refer to the circuit on the right. The impedance of this circuit is given by:

$$\mathbf{Z} = (R_w + j\omega L_1) + \mathbf{Z}_p, \text{ where: } \mathbf{Z}_p = \frac{-jR_1\left(\frac{1}{\omega C_1}\right)}{R_1 - j\left(\frac{1}{\omega C_1}\right)}$$



R_w is the resistance of the inductor. The voltage at node N_2 can be calculated using Ohm's Law. First calculate the circuit current, $\mathbf{I} = \mathbf{V}_s / \mathbf{Z}$.

Then calculate the voltage developed by that current across the impedance of the parallel combination of C_1 and R_1 . $\mathbf{V}_{N_2} = \mathbf{I} \cdot \mathbf{Z}_p$.

Objectives

A steady state sinusoidal voltage will be applied to a series-parallel connected RLC circuit. The amplitudes and phase angles of voltages and currents in the circuit will be measured. The measurements will be compared to theoretical calculations.

Procedure

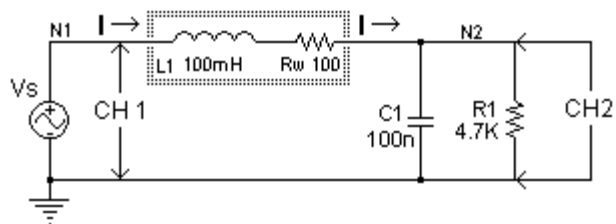
Equipment and Parts

Function Generator, Oscilloscope with 10X probes, and Breadboard.
Resistor: 4.7K, ¼ watt, 5%, Inductor: 100 mH, 5%, Capacitor: 0.1 μF, 5%.

1. Measure and record the values of the components, including the resistance of the inductor, R_w .

R_1 _____ R_w _____ L _____ C _____

2. Connect the circuit on the right. Connect oscilloscope channel 1 to N_1 and channel 2 to N_2 . Set the trigger to channel 1.
3. Set the generator to produce a 3.0 V peak-to-peak, 800 Hz, sine wave at node N_1 .



4. Measure and record the peak-to-peak magnitude and phase angle of the voltage at node N_2 (ch2). Record your results into the data table on the next page.

Data Table

Freq. Hz	V_{N1} p-p	VP_{N1} deg.	V_{N2} p-p	VP_{N2} deg.	I_{N2} p-p	IP_{N2} deg.
800.00	3.00	0.00				
1600.00	3.00	0.00				
3200.00	3.00	0.00				

- Set the function generator to produce a 3.0 V peak-to-peak, 1600 Hz, sine wave, as measured by channel 1 of the oscilloscope. Repeat step 4.
- Set the function generator to produce a 3.0 V peak-to-peak, 3200 Hz, sine wave, as measured by channel 1 of the oscilloscope. Repeat step 4.

Analysis

- Use the voltage measurements at node N2 to calculate the circuit current, I , at each frequency. Enter the results into the table above.
- Use your scientific calculator to calculate inductive and capacitive reactance at each frequency. Then calculate the impedance of the circuit, Z , at each frequency, using the measured values of your components.

TI-89 example at 1600 Hz:

$$X_{L1}: 2 \cdot \pi \cdot 1600 \cdot .1 = 1005 \text{ ohms. } X_{C1}: 1 / (2 \cdot \pi \cdot 1600 \cdot .1E-6) = 995 \text{ ohms.}$$

$$Z: 100 + 1005i + (-4700 \cdot 995i) / (4700 - 995i) = (306.2 \angle 9.91).$$

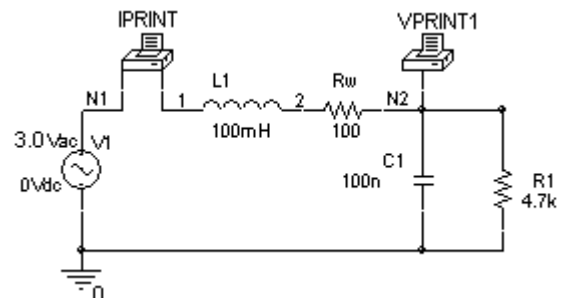
- Use Ohm's law to calculate the circuit current, I , at each frequency.
 $I: 3 / (306.2 \angle 9.91) = (9.80 \angle -9.91).$
- Simulate the circuit. Compare the simulated results to your calculations. They should agree exactly. Express the percent difference between your calculated and measured current and between your calculated and measured node 2 voltage.

PSpice Example

Connect the circuit as shown on the right.

The printers, "IPRINT" and "VPRINT1", are in the "SPECIAL" library.

Enable the printers by double clicking on each printer to open the property editor. Type "ok" under AC, MAG, and PHASE.



Simulation Settings: Analysis type: AC SWEEP/Noise, Sweep Type: Log/Octave, Start Frequency: 800, End frequency = 3200, Points/Octave: 1.

Output File Results:	FREQ	IM(V_PRINT2)	IP(V_PRINT2)
	8.000E+02	2.088E-03	5.550E+01
	1.600E+03	9.800E-03	-1.001E+01
	3.200E+03	1.965E-03	-8.428E+01
	FREQ	VM(N2)	VP(N2)
	8.000E+02	3.824E+00	-1.156E+01
	1.600E+03	9.536E+00	-8.806E+01
	3.200E+03	9.722E-01	-1.682E+02

LTspice Example

Connect the circuit on the right. Use your measured part values.

Right click on V1 and set "AC amplitude" to 6 under "Small signal AC analysis" in the dialog box that opens.

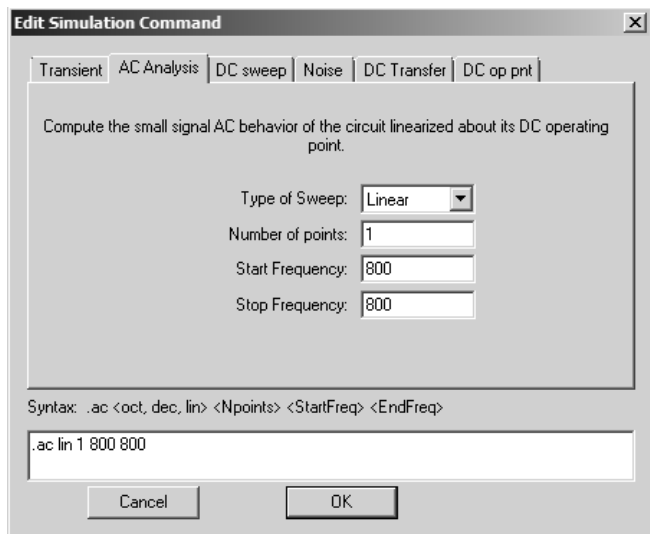
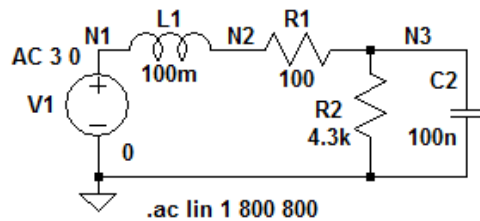
Labeling the nodes (N1, N2, and N0) makes it easier to interpret the results.

Click on **Simulate** in the main menu.

Select Edit Simulation Cmd to open the dialog box shown on the right.

Select AC Analysis and enter values as shown. Note that this is for one frequency. If you specify more than one frequency the output of the simulation will be a graph instead of a list.

You will need to run the simulation three times, once for each frequency. Run the simulation. A window will open listing the simulation results.



```

--- AC Analysis ---
frequency:      800          Hz
V(n3):          mag:   3.80461 phase:  -12.2394°      voltage
V(n1):          mag:         3 phase:         0°      voltage
I(V1):          mag:  0.00210717 phase: -127.067°    device_current

```

The results for the voltages should agree with your expectations. The currents need to be interpreted according to LTspice conventions.