

## Complex Impedance

**Purpose:** Examine and understand the frequency dependent behavior of the complex impedance for passive circuit elements (resistors, capacitors, and inductors). Gain experience working with test equipment, PSpice, and transferring data to Excel for plotting.

### Equipment Required

- 1 – Computer with Capture, Pspice, and Excel
- 1 - Digital Multimeter (DMM)
- 1 - Function Generator
- 1 - Oscilloscope
- 1 - RCL Meter (shared)
- 1 - Protoboard
- 1 - 100 $\Omega$  Resistor
- 1 - 1k $\Omega$  Resistor
- 1 - 0.1 $\mu$ F Capacitor
- 1 – 2.2mH Inductor

### Prelab

In your lab notebook, write a formula for the complex impedance for an inductor with parasitic resistance. Write formulas for the magnitude and phase of the complex impedance of an inductor.

In excel, set up two sheets in one workbook. Label sheets capacitor and inductor. On the top of the capacitor sheet, put in labels and values for Capacitor(=0.1 $\mu$ F) and resistor Rcd (=1k Ohm; see fig. 2). On the top of the inductor sheet, put in labels and values for Inductance(=2.2 mH), parasitic resistance (assume =10 ohm for now), and Rcd(=100 Ohm).

For both sheets:

- 1) A few rows down, create a table with six columns by labeling columns: Freq\_theory, Mag\_theory, Phase\_theory, Freq\_Pspice, Mag\_PSpice, Phase\_Pspice.
- 2) Populate the Freq\_theory column in each sheet with frequencies from 20Hz to 2 MHz at 5 value per decade (Hint: that's a proportional spacing of  $10^{0.2}$  per row).
- 3) In the Mag\_theory and Phase\_theory columns in each sheet (for all frequencies), have excel calculate the theoretical magnitude and phase for the complex impedance of respective circuit element, based on the frequency, capacitance/inductance, and parasitic resistance in that sheet. (use the formula  $\text{atan2}(x,y)$  for the inductor's phase)
- 4) Create two graphs in each sheet: one log-log of the Mag\_theory versus frequency and one log-linear of the Phase\_theory versus frequency.
- 5) Print out input values and table and tape them into your lab notebook.
- 6) Print out both charts and tape them into your lab notebook.

For both sheets:

- 1) Create a second table below the first with seven columns by labeling the

columns: Freq\_plan, Freq\_expt, Voltage\_DUT, Voltage\_R, Phase\_RtoDUT, Mag\_expt, and Phase\_expt.

- 2) Populate the frequency plan with frequencies from 10 to 2MHz, use a 1, 2 5 pattern for data taking.
- 3) Bring spreadsheet to lab and load it onto your lab computer at the start of lab.

Read the rest of lab before coming to class.

When you get to lab, after you have the excel workbook running on your computer, have your lab TA check your notebook and computer. He will check that you have values, tables, and charts taped into your lab notebook and that you have excel file loaded and up and running.

You will bring the data from PSpice into your excel worksheets and use your worksheets to enter your experimental measurements and calculate the experimental values for magnitude and phase of the inductor and capacitor.

### Background

**Impedance** is a measure of the “opposition” to the flow of electrons in a circuit in response to an applied voltage. For a resistor (or set of resistors), the impedance is just the resistance, which is the real valued and frequency independent. In circuits with reactive elements (capacitors and inductors), impedance takes on real and imaginary values and becomes frequency dependant. A circuit with a complex impedance,  $Z(\omega)$ , has a phase shift between the applied voltage to the circuit and the current through the circuit generated by the applied voltage. The complex impedance ( $Z$ ) of a circuit is the ratio of the complex voltage  $V(\omega)$  across and complex current  $i(\omega)$  through a circuit:  $Z(\omega) = V(\omega) / i(\omega)$

The impedances of a resistor, a capacitor, and an inductor are: (subscript indicates element):

$$\text{Resistor: } Z_R = R \qquad \text{Capacitor: } Z_C = \frac{1}{j\omega C} \qquad \text{Inductor: } Z_L = j\omega L$$

Note the use of electrical engineering notation for complex numbers, where  $j = \sqrt{-1}$ . This is the common usage in electronics, to avoid confusion with the symbol for current,  $i$ . Then dealing with complex impedances, voltages, and currents, it is convenient to represent a complex value  $A(\omega)$  by its magnitude and its phase, where  $A(\omega) = |A(\omega)| e^{j\theta_A(\omega)}$ .

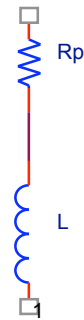
In this laboratory exercise, you will study these relationships by comparing the theoretical, simulated, and experimental magnitudes and phases of the impedance at different frequencies for capacitors and inductors. Experimentally, you will derive the complex impedance of the elements by measuring the magnitude and phase of the voltages and currents of each circuit element, first in a simulator (Pspice) and then in the actual circuit elements. You will do this by measuring only voltages, as that's what oscilloscopes do. The currents will be derived from the voltages. After reducing the data, you will analyze the differences between the theoretical, simulated (Pspice), and actual impedances.

## Procedure

### 1. Measure component values

#### **(Record all measurements in your notebook)**

Using the Capacitor-Inductor analyzer, measure and record the actual values for your C and L. Using the DMM, measure and record the resistances of all the resistors and the parasitic resistance,  $R_p$ , of your inductor (as shown in the model of Fig. 1). Use a 1k Ohm for Rcd with the capacitor and a 100 Ohm for Rcd with inductor in all cases. Change the values in your excel sheets to match these measured values.



### 2. Empirical measurement via PSpice simulator

To get a start on simulating circuits with the popular Cadence PSpice software, to get a theoretical understanding of impedance, and to see the contrast between theory and experiment, you will start by entering the test circuit in figure 2 into Cadence Capture and the run PSpice to analyze and empirically measure the impedance of the devices. Set the internal resistance of the function generator,  $R_{th}$ , to 50 Ohms. The following steps will get you set up with the first circuit.

Figure 1

#### Setting up a schematic

-Refer to circuit diagram in Figure 2 for placement, rotation, and value of parts. Start with the capacitor as the Device Under Test (DUT)

-Open DESIGN ENTRY CIS

-Choose the default suite. For me, it was Librarian XL.

-When CIS window opens, size it to left 2/3 of screen.

-Create new project: FILE->NEW->PROJECT

Type in name, choose "analog or mixed A/D"

Set location by browsing to a folder in your MY DOCUMENTS folder.

Select and click OK, and then click OK again.

-To "create PSpice project", choose "blank project"

When Schematic window opens, scale to upper 2/3 of CIS window

-Click on "Place Part" icon

Click "Add library" icon. Ctrl-click both "pspice/analog" and "pspice/source"

Click "open"

Type "V" and double-click "Vac/source"

Place "Vac" on schematic, then press "esc" to stop placing parts.

-Double click on "Vac" icon and size properties window in bottom 1/3 of CIS window.

Change "AC MAG" to 100 VAC and "Apply"

Click "Page1" tab to return to your schematic

-Click on "Part Placement" icon

Make sure "analog" library is chosen

Type "R" and double-click "R" for resistor

To rotate part on schematic, press "R".

Place resistor(s) on schematic. Press "esc" when done.

-Click on "wire" icon

Use "wire" tool to connect "Vac" to resistor. Press "esc"

(from now on, press "esc" to finish placing parts, as needed)

-Click on "ground" icon

Under "Libraries" select "SOURCE" and choose and place "0"

-Click on "part placement" icon

Choose "C" for capacitor, click OK, and place capacitor for test. Add wires.

- Click on “off page connection” icon; under “Libraries” select “CAPSYM”  
Place connectors, change their labels, and wire them into circuit.
- Double click on resistor or capacitor values (not the icons)  
Type in new values, as per lab instructions.  
Use a “u” for a “mu”  
(Or double click on part icon to edit values in properties window)

#### Setting up and Running Pspice

Choose: PSPICE->NEW SIMULATION PROFILE

Name the profile and choose inherit “none”

Under “Analysis” choose “AC Sweep”, Log, 10 to 1000000 Hz, 5 points/decade

Click “OK”

Click on “voltage” and “current” probe icons

Place probes as needed on circuit (must be on a component pin).

(You will see that + or – current will depend on which pin is probed.)

Click “RUN” icon. Observe behavior of voltages and currents.

Choose PLOT->ADD PLOT to start a plot for currents

Click a “I( )” in legend below plot and delete it from plot (use delete key).

Select other plot

In schematic, select and right click on I-probe, and choose “view results”.

Convert to Bode plots by

Double clicking y-axis on both plots and choose log and OK.

(Beware, if you click RUN again, these settings will be reset)

To keep plot settings

Choose PSPICE-> EDIT SIMULATION PROFILES

Set “probe window” to “last plot” and click OK. Now settings won't reset.

#### Play with Simulator (write observations in your notebook)

**Vary “C” values and rerun simulator. What do you observe?**

**Vary the value of Rcd. What do you observe?**

**Vary the value of Rth. What do you observe?**

End with measured C value and Rth=50ohm.

#### Set Simulator to measure impedance (write observation in your notebook)

Select the current plot and choose TRACE -> ADD TRACE

Modify “trace expression” to  $(V(Vs)-V(Vdut))/Rcd$  and click OK.

**Compare and Contrast this trace with the current probe measurement.**

Modify “trace expression” to  $Rcd * V(Vdut) / (V(Vs) - V(Vdut))$  and click OK.

**What does this formula represent?**

#### Retests Simulator (write observations in your notebook)

**Vary “C” values and rerun simulator. What do you observe?**

**Vary the value of Rcd. What do you observe?**

**Vary the value of Rth. What do you observe?**

**Why do we need to measure Rcd, but not Rth?**

End with measured C value and Rth=50ohm.

#### Plotting Phase

Add phase plot: PLOT-> ADD PLOT. Make sure new plot is selected

On schematic, PSPICE->MARKERS->ADVANCED-> PHASE OF V (or I)

Place probe at nodes (note probe measurement appears on plot)

Drag probe to other locations or add more probes and observe.  
(Note current phase depends (180 degree shift) on which node is probed.)

#### Outputting Data

Choose FILE->EXPORT->TEXT

Set “. . .” to send file to MY DOCUMENTS folder

(After first save, this selection is remembered)

Run Microsoft Excel and choose FILE->OPEN and open saved data file

Click “finish” to finalize import of data file.

**Copy these values into your capacitor worksheet under columns for capacitor magnitude and phase.**

**Add PSpice data to your plots in Excel. Use line and marker values that distinguish between the Excel and PSpice plots, even if overlapped.**

**Do the Excel and PSpice plots overlap for both magnitude and phase?**

#### PSpice for the Inductor

Replace capacitor with inductor and its parasitic resistance and redo the PSpice measurements for the inductor and export the results.

**Copy imported PSpice outputs for the inductor into your Excel sheet.**

**Do the Excel and PSpice plots overlap for both magnitude and phase?**

### 3. Determine actual device impedances

The math capability of the oscilloscope allows you to display a trace that is directly related to the circuit current. You can also use the measurement capabilities of the oscilloscope to measure voltages, frequencies, and phases (and much more).

**Record your estimates of the errors on each type of measurement you make.**

The procedure for measuring the impedance of a circuit element on the oscilloscope uses the Math and Measurement capabilities of the scope. Set up the circuit in Figure 2. Set up the oscilloscope to probe  $V_s$  and  $V_{dut}$ . Use Math to set up a trace of  $V_s - V_{dut}$ . Use Measurements for measure frequency,  $V_s$ ,  $V_s - V_{dut}$ , and phase  $V_s - V_{dut}$ .

**Make measurements at 3 frequencies per decade from 20Hz to 2MHz for the capacitor and inductor (use 1, 2, 5 method). Record all values to 2-3 significant figures in the columns in your excel sheet. Make full measurements for capacitor and inductor.**

Analyze your voltage data in excel to compute the magnitude of the impedance. **Record Excel formula(s) you use in your notebook.** Use your phase measurements to get the phase of the impedance. **Note in your notebook what you had to do to get from experimental measurements of phase to your measured phase of the impedance.**

**Add your experimental measurements of impedance magnitude and phase to your theory/simulation plots.**

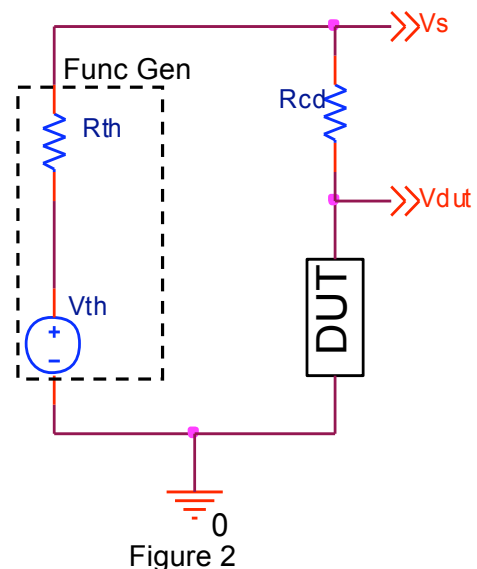


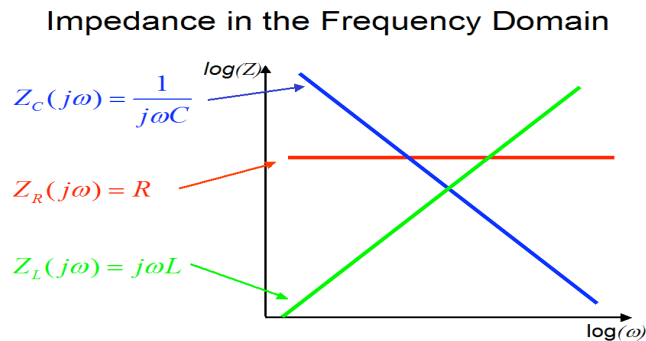
Figure 2

**Set the resistors, capacitor, and inductor you used for this lab aside with your names on them, so you can use them in next lab.**

### Data analysis

1. You should have four graphs with plots of Pspice derived, theoretical, and measured impedance magnitudes and phases for your DUTs (inductor and capacitor). For magnitude, both axes should be logarithmic, with the frequency on the horizontal axis and impedance on the vertical axis. Your phase plots should be linear in phase and logarithmic in frequency.
2. Print out the graphs, clearly label each curve, and **tape the graphs into your lab book. Tape your final excel sheet tables into your lab notebook.**

3. The theoretical (ideal) impedance magnitude curves, ignoring any parasitic behavior, should be similar to Figure 3. **Compare and contrast the all your traces in each graph.**



4. Compute the expected fractional errors in the theoretical versus experiments plots of impedance magnitude of your capacitor given your estimate of the measurements errors in frequency with the oscilloscope and in capacitance using the RCL meter. **Plot the absolute value of your computed fractional error versus frequency (log-log). Note how this error compares with deviations between your theoretical and experimental curves.**

### Conclusion (Include results and answers in your lab notebook)

1. In PSpice, we used the formula  $Rcd \cdot V(Vdut) / (V(Vs) - V(Vdut))$  to get the impedance and on the oscilloscope we use the Math function to subtract  $Vs - Vdut$  and then use the magnitude of the math to calculate impedance. Why didn't we just measure the magnitude of  $Vs$  and subtract the magnitude of  $Vdut$  to use in our calculations?
2. In procedure 1 you measured the parasitic resistance of the inductor. At what frequencies does this resistance begin to create a significant divergence between the ideal and actual behavior of this particular inductor? Give a simple formula (as a function of inductance and parasitic resistance) for the frequency at which the ideal and actual behavior diverge
3. You might have observed that at high frequencies the measured magnitude of the impedance of the inductor deviated from the theoretical impedance. The inductor model in Fig. 1 explains only the low frequency behavior. Draw a model of an inductor that would explain the behavior at high frequencies.
4. Discuss the observed behavior of the measured impedance phase angles of the capacitor and the inductor at very low and very high frequency.