University of Basel
Department of Physics

Impedance Spectroscopy

Fortgeschrittenenpraktikum I/II

March 5, 2019

This experiment examines the impedance of fundamental components of electric circuitry, i.e. resistance, capacitance, inductance and their combination. The aim is to give the student a basic understanding of impedance with a focus on its dependence on frequency, and its technical relevance. Being a daily life object, a loud speaker will serve as an example to illustrate the concept. From a metrological point of view, the aim is to get familiar with the basic principles of the lock-in technique.

Contents

1 Introduction

When we apply a DC voltage to an electrical circuit, in general the magnitude of the voltage changes across the circuit, but when we apply an AC voltage to a circuit, not only the magnitude but also the phase changes. This change in magnitude and phase is called impedance Z, and it generalizes the idea of electrical resistance in a DC circuit. By measuring the impedance of a circuit, we can determine its resistance. This technique is used, as an example, in solar cell research, to determine the recombination resistance and the chemical capacitance of solar cells. We will use the same technique to examine the impedance of a resistor, an inductor, and a capacitor, and some basic combination of those, and at last a loud speaker.

2 Basics

2.1 Impedance

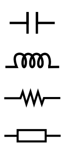
The impedance can be described with an amplitude and a phase. This leads to the idea to represent the impedance as a complex number $Ae^{i\theta}$, with A being the amplitude and θ the phase. If we apply an AC voltage $U(t) = U_0 e^{i\omega t}$ (ω is angular frequency) with a current $I = I_0 e^{i\omega t}$ to a resistor, we will measure an amplitude and a phase-shift, which leads to the voltage $U' = U_1 e^{i(\omega t+\theta)}$ and current $I' = I_1 e^{i(\omega t+\varphi)}$. Using Ohm's law $U = RI$ we find

$$Z = \frac{U'}{I'} = \frac{U_1 e^{i(\omega t+\theta)}}{I_1 e^{i(\omega t+\varphi)}} = \frac{U_1}{I_1} e^{i(\theta-\varphi)} \tag{1}$$

This is the general impedance of any AC circuit.

2.2 Impedance of basic circuit elements

An ideal resistor has a purely resistive impedance $Z_R = R$ without any imaginary component, while an inductor and a capacitor have a purely imaginary reactive impedance, denoted $Z_L = i\omega L$ and $Z_C = 1/i\omega C$, respectively. See Fig. 1.



Capacitor $Z_C = 1/i\omega C$
Inductor $Z_L = i\omega L$
Resistor $Z_R = R$
Resistor

Figure 1: Symbols of basic circuit elements and their impedances [A].

This also means, that the phase-shift of an ideal resistor is 0°, an ideal capacitor −90°, and an ideal inductor +90°.

In general, real circuit elements may not be represented by only their idealized counterpart. For example, a real resistor has a stray capacitance created by its electrodes, and especially for high frequencies it is better to model a real resistor with an ideal resistor and an ideal capacitance in parallel. Keep this in mind, when simulating the behavior of a circuits.

2.3 Parallel RC circuit

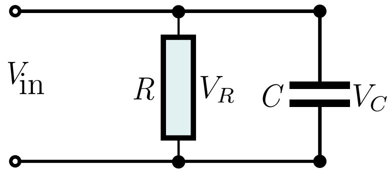


Figure 2: Parallel RC-circuit diagram [A].

For this simple circuit (Fig. 2) the impedance is

$$Z = \frac{1}{\frac{1}{R} + i\omega C}.$$

We can expect the following behavior: At low frequencies the resistor dominates, as the capacitor does not conduct. So we can expect the impedance to be dominated by R. We also expect the voltage and current to be in phase. On the other hand, for high frequencies the conduction of the signal will be mainly through the capacitor, so $1/i\omega C$ dominates, and the capacitor current is 90° out of phase with the voltage.

When measuring a real device, again it may be necessary to include further elements; in this case an additional series resistance representing the losses in the cables and devices could be included.

2.4 Parallel RLC circuit

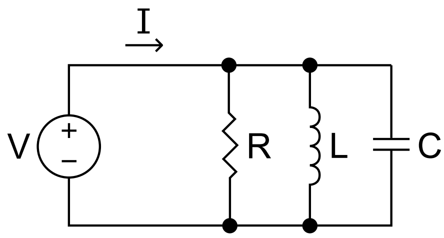


Figure 3: Parallel RLC-circuit diagram [A].

In an RLC circuit a resonance phenomenon may occur, where energy storage oscillates back and forth between L and C. In case of a parallel RLC circuit this is an anti-resonance, where the impedance peaks at the resonance frequency $\omega_0 = 1/\sqrt{LC}$. The quality factor, defined as the width of the resonance divided by the resonance frequency, is given by

$$Q = \frac{\Delta\omega_0}{\omega_0} = R \cdot \sqrt{\frac{C}{L}}.$$

Further, the impedance of the circuit is determined by

$$\frac{1}{Z} = \frac{1}{R} + \frac{1}{i\omega L} + i\omega C,$$

which reduces to $Z_0 = R$ for the resonance (using $\omega_0 = 1/\sqrt{LC}$). These formulas allow us to extract all three parameters R,L and C from an impedance measurement.

Also here a real circuit can behave differently from an idealized circuit. Specifically a real inductor may not be simply mimicked by an ideal inductor. A real inductor should be modelled by adding a small parallel resistor to the circuit.

For more detailed explanations of the RLC circuit, refer to [B], [C], and [D].

2.5 RLC-RL model of a loud speaker

Very often the impedance of a loud speaker is assumed to be constant over the audio frequency range (20 Hz to 20 kHz). This is sufficient for most applications, but we want to examine this assumption more carefully. A common model to describe a loud speaker is the RLC-RL model depicted in Fig. 4.

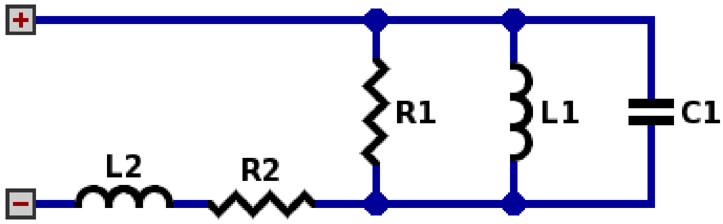


Figure 4: RLC-RL model of a loud speaker [E]

The easiest way to understand the behaviour of this circuit by considering an RLC anti-resonator in series with the RL components. The RLC circuit will give a peak in the impedance at its resonance frequency. The serial RL circuit has a dominance of R for low and of L for large frequencies, similar as for the RC circuit. The imaginary impedance $Z_L = i\omega L$ of the inductance will lead to a phase of 90° and an increase of the absolute impedance at high frequencies.

2.6 Simulating circuitry

To simulate your electric circuits you can use LTSpice, where **Spice** stands for **S**imulation **P**rogram with **I**ntegrated **C**ircuit **E**mphasis, and **LT** for the former semi-conductor manufacturer **L**inear **T**echnologies. Any other similar simulation program will do as well. You can download LTSPice from the developer's homepage, or use the version installed on the lab computer. By correctly choosing the involved (ideal) elements and their parameters, the software allows you to adequately simulate the impedance and phase of your circuits. A useful introductory video how to use LTSpice can be found on youtube.

3 Experimental Setup

The setup is very simple and consists only of a MFLI lock-in amplifier (Fig. 5) from Zurich Instruments, which performs the impedance measurement, a breadboard on which you build circuits, and a set of BNC cables to connect the breadboard to the lock-in amplifier. In order to understand the basics of the lock-in technique, refer to chapter 6.2 of the manual for the ZI MFIA, a device similar to MFLI. Also familiarize yourself with the principles of the impedance measurement in chapter 6.1, and finally follow the tutorial in chapters 3.1 and 3.2 to perform your first impedance measurements. The software interface of the MFLI is called *LabOne User Interface MF USB* and can be found in the Windows' start menu.

Figure 5: MFLI Lock-in Amplifier from Zurich Instruments [F].

4 Measurements

Perform all your measurements in a 4-terminal configuration. Cover a frequency range from 10 Hz to 510 kHz, except for the loud speaker. Use a couple of hundreds of data points to get nice and smooth curves. Store all your data using useful filenames, so you can evaluate them later.

1. Measure the impedance of individual resistors with $R = 100\,\Omega$, $1\,\mathrm{k}\Omega$ and $1\,\mathrm{M}\Omega$.
2. Measure a parallel RC circuit with $R = 100\,\Omega$ and $C = 1\,\mu\mathrm{F}$.
3. Upgrade to a parallel RLC circuit with $R = 1\,\mathrm{M}\Omega$, $L = 10\,\mathrm{mH}$ and $C = 1\,\mu\mathrm{F}$ and examine it.
4. Measure the impedance of the provided loud speaker in the audible frequency range of 20 Hz to 20 kHz.
5. *If you like to see some cool AC circuit techniques, you can play around a bit:*
   RC circuits, depending on their configuration, can be used as very basic frequency filters. You may use it to filter the output from e.g. your smartphone, and play the result, which is first amplified, on the provided speaker. You can low-pass filter the signal (use $1\,\mathrm{k}\Omega$ and $1\,\mu\mathrm{F}$), so high tones are suppressed, or you can employ a high-pass filter (use $1\,\mathrm{k}\Omega$ and $33\,\mathrm{nF}$), achieving the opposite. First, you need to power the amplifier with the provided 12 V power supply, and level the volume to a reasonable value with the blue potentiometer.

5 Data Analysis

For all the circuits mentioned above, plot the magnitude as well as the phase-shift of the impedance as a function of frequency in one diagram. The left y-axis should represent the magnitude of the impedance, and the right y-axis should represent the phase-shift (scale $\pm 100°$). Plot the frequency (x-)axis in a log scale. Further, create Nyquist plots for all of them, i.e. plot the real part of Z vs the imaginary part. Learn how to interpret these plots and extract useful information on your circuits.

You can use Python or MATLAB, or any other scientific plotting software that you like, to generate plots. MATLAB is installed on lab computers. Python is free software and you can download and install it on your own computer. Export the data from LabOne into a format that your plotting software can interpret.

Simulate all circuits with LTSpice, try to find an adequate model for each, and explain why you chose the model that you used. You may have to refine your model by iterating the values of the used elements. Add the simulated impedance and phase to your plots, compare them to measurements and draw conclusions. Especially for high frequencies do not be discouraged, if you cannot model everything perfectly. High-frequency circuit analysis is complicated! For each point on the list in the previous section perform the following tasks:

1. Extract the values of the circuit elements from your measurement, and compare them to their nominal value (e.g. did you measure $100\,\Omega$ for the $100\,\Omega$ resistor?) Compare the results for each of the three resistors. Do they differ? If yes, in what frequency range? Can you model this adequately in LTSpice? Can you get further information from your model?
2. What information can you extract from the Nyquist plot? (As an example, this analysis would be useful if you try to characterize an interface in a solar cell, for which you do not know R and C.)
3. Extract ω_0 and Q, as well as R, L, and C from your measurement. Do these values agree with your model in LTSpice?
4. Extract all parameters of the RLC-RL model of the loud speaker from your measurement in combination with LTSpice simulations. Is the model for the loud speak realistic?

References

[A] Wikipedia

[B] Wikipedia on RLC circuits

[C] Electronics Tutorials on RLC circuits

[D] The Arts of Electronics by P. Horowitz and W. Hill

[E] Dave Barber's homepage on loud speakers and impedance

[F] Zurich Instruments