University of Basel
Department of Physics

Impedance Spectroscopy

Fortgeschrittenenpraktikum I/II

March 30, 2018

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 dependency on frequency, and its technical relevance. Being a daily life object, a loud speaker will serve as an illustrative example. 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 on a resistor, the magnitude of the voltage will change. But if we apply an AC voltage on the resistor, not only the magnitude but also the phase will change. This resistance is called impedance Z. By measuring the impedance of a circuit, i.e. the magnitude and phase, we can determine its resistance. This is e.g. used in actual solar cell research to determine the recombination resistance and the chemical capacitance of solar cells. We will use it to examine the basic behavior of the impedances of R, L and C, some basic combination of them, 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 $Ae^{i\vartheta}$, with A Amplitude and ϑ phase. If we apply a voltage $U = U_0 e^{i\omega t}$ (ω: frequency) with a current $I = I_0 e^{i\omega t}$ to a resistor, we will measure an amplitude and a phaseshift, which leads to the voltage $U' = U_1 e^{i(\omega t + \vartheta)}$ and current $I' = I_1 e^{i(\omega t + \varphi)}$. With Ohm's law $U = RI$ we receive

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

This is the general impedance of any AC-circuit.

2.2 Impedance of the 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. Compare Fig. 1 for the corresponding element symbols. This also means, that the phase of an ideal resistor is 0°, of an ideal capacitor −90°, and 90°

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].

of an ideal inductor.

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

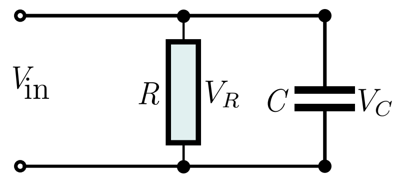

Figure 2: Parallel RC-circuit diagram [A].

2.3 Parallel RC circuit

For this simple circuit (cf. 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 and current and voltage to be in phase. In turn for high frequencies the capacitor will do the main part of the conduction, 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

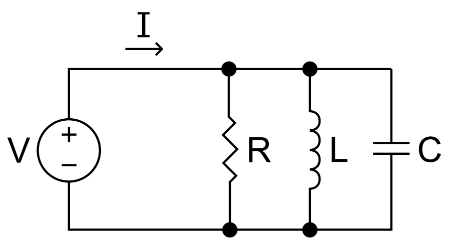

Figure 3: Parallel RLC-circuit diagram [A].

In an RLC circuit a resonance phenomenon may occur, where energy swaps 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 to extract all three parameters R,L and C from an impedance measurement.

Also here a real circuit can differ significantly from the case of idealized elements. Specifically the inductance may not be simply mimicked by an ideal inductance, but also has a relatively low, parallel resistance.

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, however, we want to examine this in a bit more detail. A common model to describe a loud speaker is the RLC-RL model depicted in Fig. 4. The impedance of this circuit is easiest understood by considering an RLC

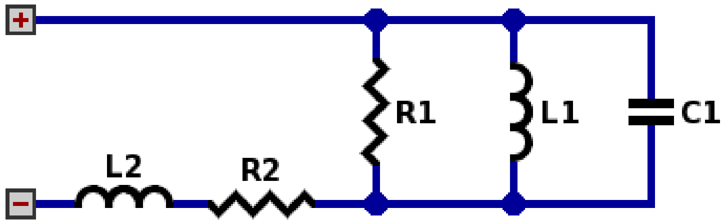

Figure 4: RLC-RL model of a loud speaker [E]

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 on 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 measurements, a breadboard to built up the circuits, and a set of BNC cables to connect the circuits to the MFLI. To understand the basics of the lock-in technique please refer to chapter 6.2 of the manual of the ZI MFIA, a similar device as the MFLI. Also get familiar with the principle 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 nicely smooth curves. Store all your data using useful filenames, so you can evaluate it later.

1. Measure the impedance of single 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 practical implications of the circuitry, 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 Analyzation

Plot for all measured circuits the absolute value of the impedance and the phase vs frequency. Do this in one diagram for each circuit, where the left y-axis serves as impedance axis, and the right one as phase axis (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.

It is recommended to use Matlab for plotting, as the LabOne interface allows to directly export to the matlab data format. It further allows to implement the plotting procedure in a script, which, once written, makes the plotting a task of a few clicks. Matlab 2017 is installed on the lab computer, in case you do not have access to it yourself.

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 the measurement and draw conclusions. Especially for high frequencies do not fall into despair, if you cannot model everything perfectly. For each point in the enumeration from the previous section do the specific tasks listed below:

1. Extract the values of the circuit elements from your measurement, and compare them to their nominal value (E.g. is the 100 Ohms really a 100 Ohms as denoted?) Compare the results of your three resistors. Do they differ? And if yes, in what frequency range? Can you model this adequately in LTSpice? Can you get further information from this modeling?
2. What information can you extract from the Nyquist plot? (This would e.g. be important, if you would 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 fit, if you model it 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