EEE202-03 Hardware Lab 1
Ayhan Okuyan
21601531
26.02.2018

EEE202 Hardware Lab 1

Introduction

The purpose of this laboratory exercise is to find the quality factor and inductance of a real inductor with two separate methods combined with getting familiar with using a SPICE software in our case, is LTSpice. Due to winding a physical wire, all real inductors appear to have internal resistances. However, LTSpice contains only ideal components. Hence, in order to make a real inductor, we will be using an ideal inductor in series with a resistance.

To begin the exercise, we first need to understand the meaning of a quality factor. Quality factor is the ratio of the energy stored in the inductor to the energy loss over one cycle. Thus, we can easily find the quality factor of a non-ideal inductor as:

$$Q = \frac{\omega * L}{R_L}$$

From this equation, we can clarify that, in order to find the quality factor Q, we must find L (inductance) together with R_L (the internal resistance). To find those quantities, two methods below will be used and demonstrated.

After the software lab is done, the next process, which is discussed in this report in detail is the physical application of these methods. Things that are expected from us are using our decided methods on two different inductors, one being a pre-made inductor in which I have chosen it to be 10µH, and another one that we wire onto a core. For that one, I have concluded that I would wind 15 turns of 0.5mm diameter wire onto a T38-8/90 core. Furthermore, we are asked to work on 4 different frequencies ranging from 1Mhz to 5Mhz.

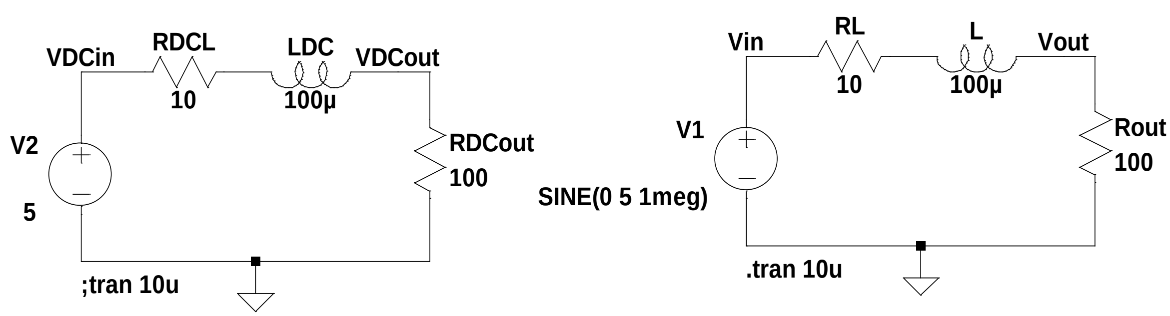
Software Implementation of Method 1

The first method takes advantage of two things. The first one is the fact that, in a circuit with only resistances and inductors, the only phase difference that will occur will directly be connected to the inductance. Thus, using the phasor domain will contain utmost importance to find the inductance. The second advantage is the inductors nature to act as open circuits when a DC voltage is applied on it.

Hence, applying a DC Voltage and calculating the voltage divider will be important for finding the internal resistance of the inductor.

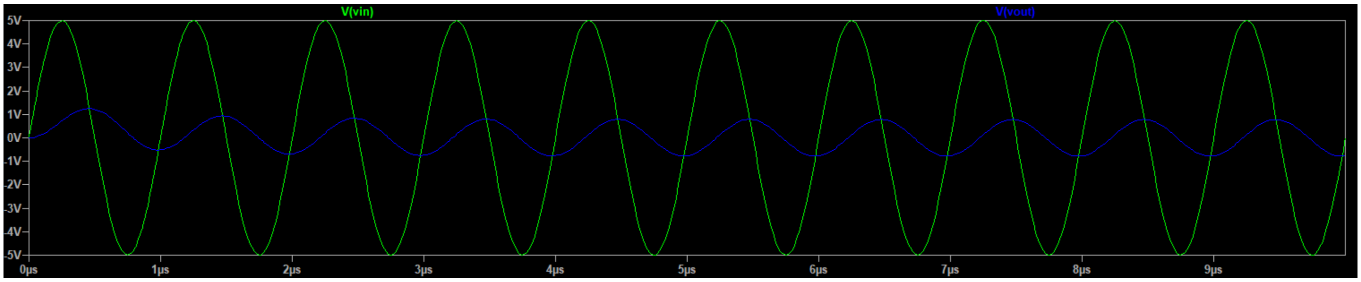
The materials that will be used consists of a voltage source, a real inductor and an external and known resistance. The important thing is that in order to find the inductance and resistance reliably, sine wave will be applied to the circuit following a DC Voltage. Finding the internal resistance through applying AC Voltage can be painfully hard in terms of precision.

In order to ensure the correctness of the designed circuit, an LTSpice pseudo-circuit is created an demonstrated with randomly picked values. The purpose is to be able to reach those randomly picked results through our calculations by accepting them as unknowns. Then the results obtained from the simulations will be compared to the theoretical values. The circuit that is designed for this method is shown below.



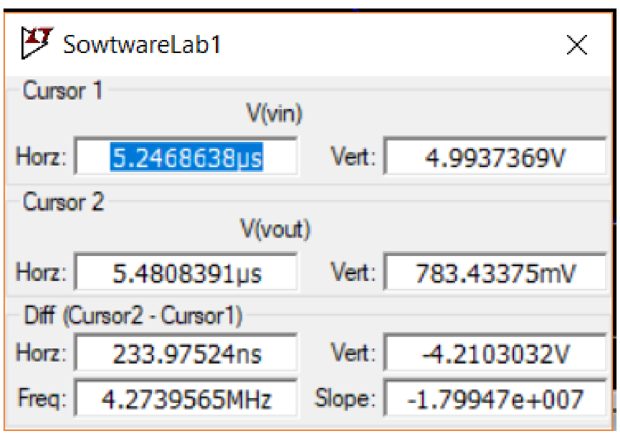
Capture 1 – The circuit designed for the first method. Although may seem as two different circuit, these circuits are in common through each of their components except the type of voltage generated by the voltage source. One source is creating a sine wave signal with 5V and 1Mhz while the other is generating 5V DC. As can be observed, the external resistor is 100Ω and the inductor is specified with 100μH and 10Ω of internal resistor.

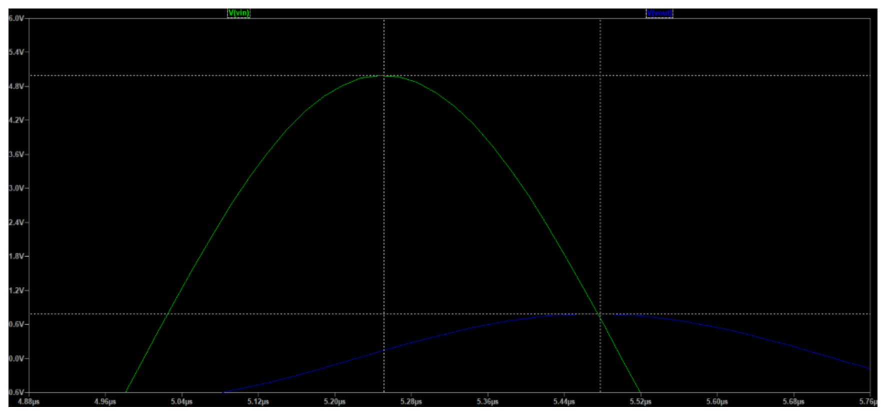
In order to find the phase angle, we have marked the Vin and Vout nodes and plotted their Voltage/Time graphs using the transient analysis function of the LTSpice. Observe that the right side of the spice drawing is used for demonstration with the sine wave applied. For the simulation we set the time interval between t=0s and t=10μs. These plots overlapped can be seen below.



Capture 2 - The Voltage vs Time Graphs of the nodes Vin and Vout. The green plot describes the behavior of the Vin, and respectively, blue curve stands for the Vout node. Easily, we can observe the phase difference between the two plots.

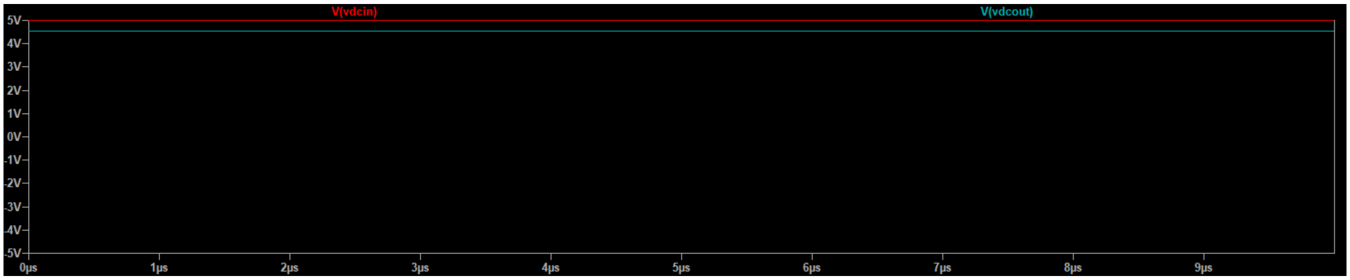
In order to find the phase difference that will be directing us to the phase angle. We have used the cursor option of the LTSpice in order to find the difference between peak points of the plots.





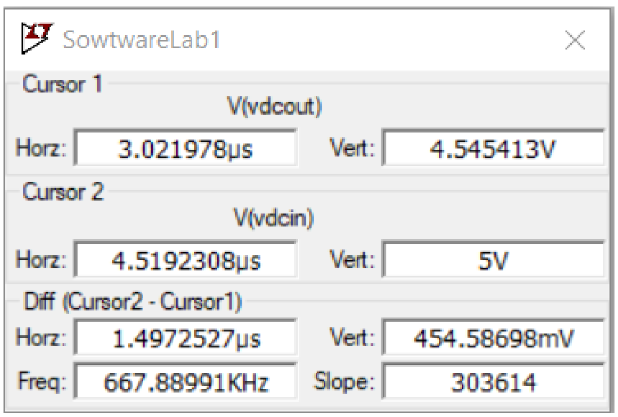
Capture 3 & 4 – The Up-Close View of Capture 2 with the Cursor Lines and The Screen with the Cursor Values. The most important values in the left screen is the Vert Values, which are the peak values of these curves and the Horz. value on the Diff section, which is the Δt, the phase difference between the two peak points.

These data were important to collect in order to find the inductance. However, in order to find the internal resistance, we need to collect data from the leftmost circuit depicted in Capture 1. This is the exact same circuit fed with 5V DC. The plots for these nodes can be found below.



Capture 5 – The plot of the DC circuit. As expected the Vin (red) node of this graph can be seen as 5V, while the Vout (cyan) is below that level due to the inductor is acting only as the internal resistance inside.

With the help of the cursor, we find the exact values for these voltages which is also shown below.



Capture 6 – The values for the DC voltages. The vert data are important in order to find the internal resistance as this circuit forms a voltage divider.

Within these steps, we finish the data collection process. The next step is to calculate the values using the obtained values and compare them with the initial randomized values. Thus, the calculations done to find the quality factor is as below.

First step is to find the phase angle using the time difference we have obtained using the following function where ω is the natural frequency of the given signal and Δt is the time difference.

$$\Phi = \omega * \Delta t$$

$$\Phi = 2\pi * f * \Delta t$$

This equation enables us to have the phase angle. Then in order to work on the phasor domain, we write the transfer function between the two nodes.

$$\frac{Vout}{Vin} = \frac{Rout}{Rout + RL + j\omega L}$$

$$\frac{Vin}{Vout} = \frac{Rout + RL + j\omega L}{Rout}$$

$$\frac{Vin * Rout}{Vout} = \ Rout + RL + j\omega L$$

$$Rout * (\frac{Vin}{Vout} - 1) = \ RL + j\omega L$$

Thus, we have this equation that contains complex values both on the left and right-hand sides. One important thing is that, the Vin/Vout is a complex value such that it can be written as:

$$\frac{Vin}{Vout} = \frac{|Vin|}{|Vout|} * e^{j*\Phi}$$

Solving this equation for its complex values shall give us the inductance. Moving forward for the resistance, the followed method was simple as DC Voltage makes certain that there is not a changing voltage in the circuit, the inductor will not be effective on the circuit except its internal resistance (RL), which we can use a voltage divider equation to find.

$$\frac{(VDCin - VDCout)Rout}{VDCout} = \ RL$$

As we have found all the necessary equations and data, we can calculate the inductance, resistance and the quality factor easily. To make it even more easier, we can write and use a simple MATLAB code which is given below.

Editor - C:\Users\Ayhan\Desktop\Bilkent EE\Bilkent EE 2017-2018 Spring\EEE202\Lab1\Calculations\findInd.m

findInd.m

```
function [L, R, Q] = findInd(Rx, f, deltaT, Vin, Vout, DCVin, DCVout)
omega = 2*pi*f;
phi = omega*deltaT;
L = Rx*((Vin/Vout)*imag(exp(1j*phi)))/(omega);
R = findR(DCVin, DCVout, Rx);
Q = findQ(omega, L, R);
end

function R = findR(DCVin, DCVout, Rx)
R = Rx*(DCVin-DCVout)/DCVout;
end

function Q = findQ(omega, L, R)
Q = omega*L/R;
end
```

Capture 7 – MATLAB code to calculate the inductance, resistance and the quality factor. This is a three-piece function in order to make the process easily-readable. The main function “findInd” is the function that calculates the natural frequency, the phase angle and respectively the inductance by solving the imaginary part of the equation derived from the transfer function. Hence, “FindR” uses the voltage divider equation given above and finally “findQ” uses these values obtained to find and display the quality factor on the command window.

Furthermore, using this function will give us the result below.

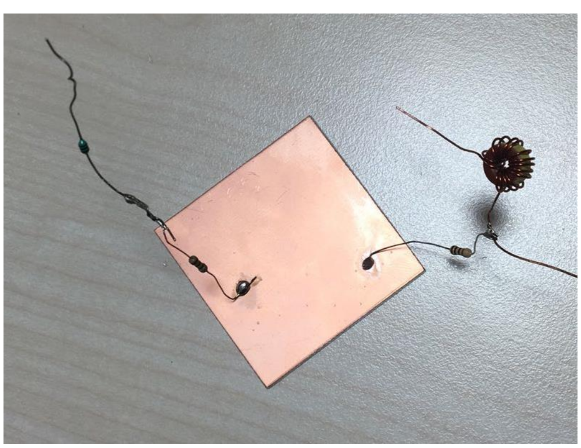
```
Command Window
>> [L R Q] = findInd(100, 1e6, 233.97524e-9, 4.9937369, 783.43375e-3, 5, 4.545413)
L =
   1.0093e-04
R =
   10.0010
Q =
   63.4125
```

Capture 8 – The MATLAB results for the inductance, resistance and quality factor for circuit in Capture 2.

To compare the results, we see that the L value is given as 1.0093e-4 which is approximately equal to the 1.0000e-4 value we have assigned to it. The R value, which is the internal resistance value is given as 10.0010 which is precisely nearly equal to the 10Ω assigned to RL. Lastly, the Q, quality factor is given as 63.4125 and when this value is calculated from the initial values, it is equal to 62.819, which is also quite similar. This way, we can conclude that the method 1 works fine enough to be submitted as a method to find the quality factor.

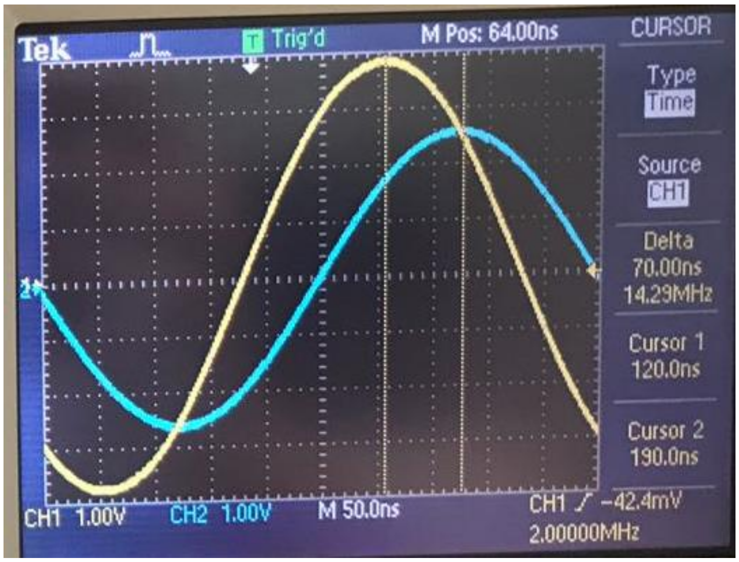
Hardware Implementation of Method 1

This method calculates the inductance and quality factor data through the phase difference that is formed between the input and output voltages. In order to find the phase differences, following circuits are constructed and soldered onto a 5x5cm PCB.



Capture 9 – The hardware circuits designed for Method 1 with both the winded inductor and the pre-made inductor. This circuitry is exactly the same when compared with the circuitry on c

After the circuits are built the only thing left is to observe the input and output voltages on the oscilloscope and record them. The only way to do that is to connect the channel one and two of the oscilloscope to Vin and Vout respectively and then calculate the time difference between them by using vertical cursors. The following picture shows the time difference and the respective cursors on the oscilloscope screen.



Capture 10 – The physical method on how to find the time difference between Vin and Vout. The cursors are displayed with dashed vertical lines. We can see that the delta value, which is equal to the time difference is presented on the right side of the screen.

Furthermore, as I explained in the Software Implementation, in order to get better results for the resistances, I have also applied DC Voltages to the circuits and used a basic voltage divider equation. Below is the oscilloscope screenshot of the DC supplied circuit, and the respective voltage values.

Capture 11 – The oscilloscope screen shot of the DC supplied circuit. As clearly seen, the input and output voltages differ due to the internal resistances of the inductances.

As the method is explained, the next step is to present the collected data and analyze them. Below you can see the data collected for both of the inductances.

The Winded Inductor with 4.73µH of Inductance						
Rx(Ω)	f(Hz)	Delta T(s)	Vin(V)	Vout(V)	DC Vin(V)	DC Vout(V)
100.00	1.00E+06	5.00E-08	6.90	6.70	5.05	4.98
100.00	2.00E+06	4.40E-08	7.10	6.30	5.05	4.98
100.00	3.00E+06	4.00E-08	7.50	5.80	5.05	4.98
100.00	4.00E+06	3.50E-08	7.90	5.40	5.05	4.98
The Pre-Made Inductor with 10µH Inductance						
Rx(Ω)	f(Hz)	Delta T(s)	Vin(V)	Vout(V)	DC Vin(V)	DC Vout(V)
100.00	1.00E+06	7.80E-09	6.80	6.10	5.03	4.98
100.00	2.00E+06	7.00E-08	7.70	5.30	5.03	4.98
100.00	3.00E+06	6.00E-09	9.00	4.80	5.03	4.98
100.00	4.00E+06	4.50E-08	8.80	3.80	5.03	4.98

This table shows all the needed values in order to find the inductance and internal resistance, thus, the quality factor. If we run it through the MATLAB code that is given in the Software Implementation, we get the following results.

Winded Inductor		
L	R	Q
5.06E-06	1.40	22.64
4.70E-06	1.40	42.10
4.69E-06	1.40	62.97
4.48E-06	1.40	80.19
Standard Inductor		
L	R	Q
8.35E-06	1.00	52.26
8.90E-06	1.00	111.49
9.00E-06	1.00	168.97
8.33E-06	1.00	208.70

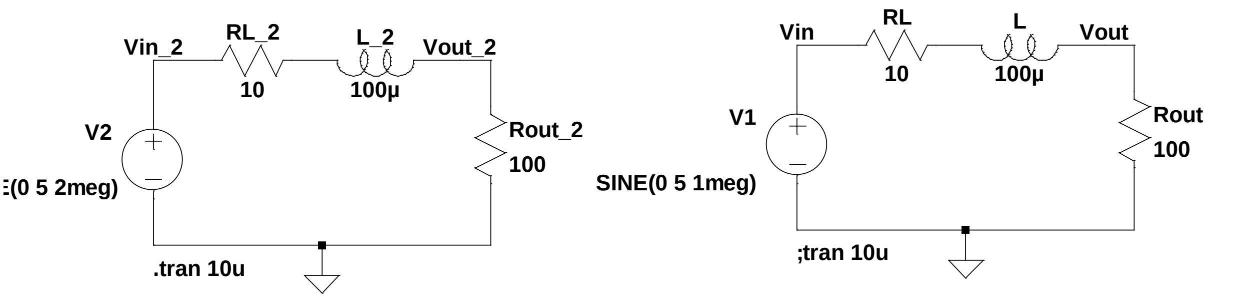
This table shows the inductance, resistance and the quality factor values, obtained from the corresponding rows of the previous graph. We can see that the average value for the winded inductor's inductance is equal to 4.73µH which is nearly equal to the calculated value with 4.68µH. However, we can see that the values tend to errata more due to the increasing frequency value. Also, the internal resistance comes up to be 1.4Ω which is also nearly the same with the calculated value which is 1.17Ω. However, the quality factor seems to errata more than the other quantities as it is calculated through the obtained values, which are obviously erroneous data. When It comes to the standard inductor, it becomes more obvious that the data have errors in it. The average inductance value is found as 8.65µH. As its proclaimed value was 10µH. Thus, this makes the average percentage error to be 13.5%. Hence, we can conclude that this method was a successful one in both software and hardware implementation.

Software Implementation of Method 2

Method 2, however is another case that must be separately investigated. The way I chose to achieve finding the quality factor this time can be considered convenient as we do not change the circuit at all. We use the same circuitry with one really important difference that changes how we look at the situation a lot. We find the voltage values of Vin and Vout on two different frequency settings. This will enable us to use the ratios of input and output voltages. Two equations with two unknowns described before as L and R_L will be easily solvable.

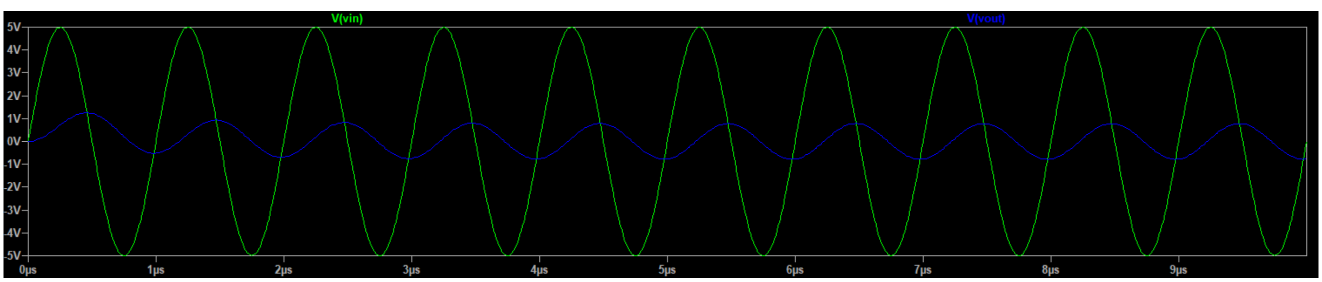
When the frequency is altered, reactance of the inductor changes accordingly as it is described as $j\omega L$. One thing to keep in mind as we progress is that R_L also changes its value according to the given frequency however, as the change is too insignificant when compared with the inductance value, we simply ignore this fact as we are only able to do finite precision calculations anyway. Like we have used throughout Method 1, we will use the same components: a voltage source, a non-ideal inductor and a resistor.

To confirm the validity of this method, we create the two circuits desired on LTSpice. Creating these two different circuits in two drafts can be valid however, I will be using only one draft and put the two circuits on the same sheet for easy calculations. The LTSpice drawing is given below.

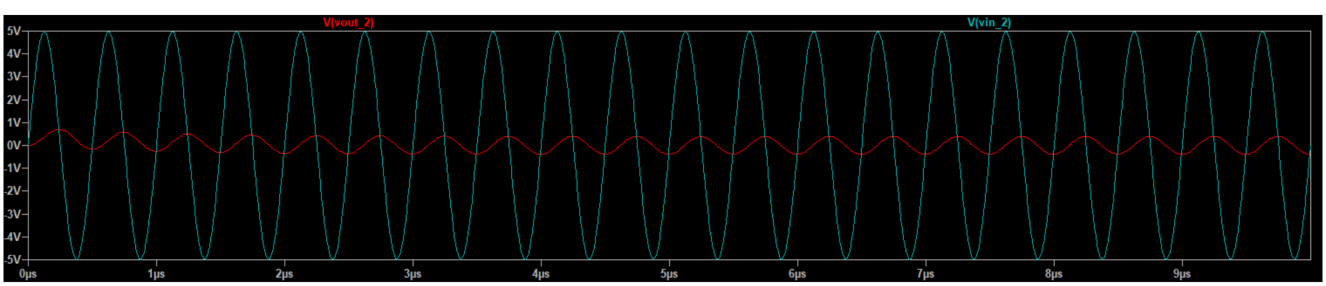


Capture 9 – The LTSpice drawing for Method 2. These drawings represent two nearly identical RL circuits with 100µH inductances, 10Ω of internal resistances and 100Ω of external resistances. The only difference is that the first circuit (on the right) generates 5sin(1e6t) while the second circuit (on the left) generates 5sin(2e6t) as their input voltages respectively.

Again, in parallel with the 1st Method, we have assigned some initial and same values to the circuits, however, we will act as if they are unknowns and compare if the values that are found through the calculations are in sync with these initial values. Then, simulations regarding the circuits are run in order to find their Vin/Vout ratios respectively. The simulation curves of Vout and Vin for Circuits 1 and 2 are given below.



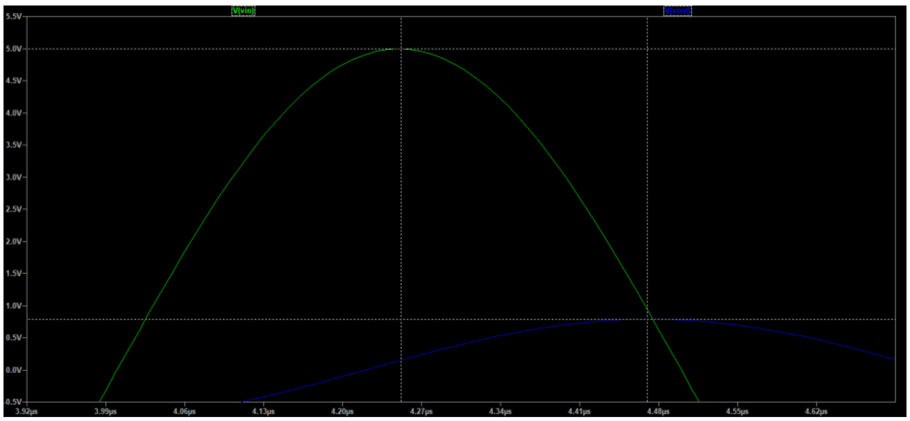
Capture 10 – The simulation of Circuit 1. This simulation is exactly the same with capture 2. As the green curve shows Vin and the blue curve shows Vout. We can see that there is a time difference between Vout and Vin curves essentially caused by the inductor that is put in.

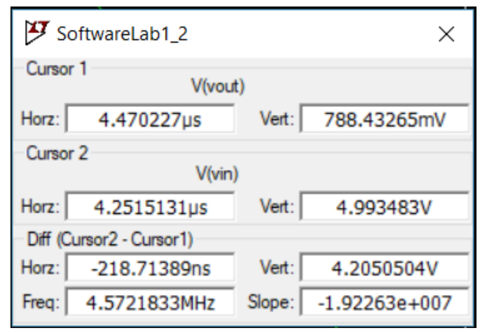


EEE202-03 Hardware Lab 1
Ayhan Okuyan
21601531
26.02.2018

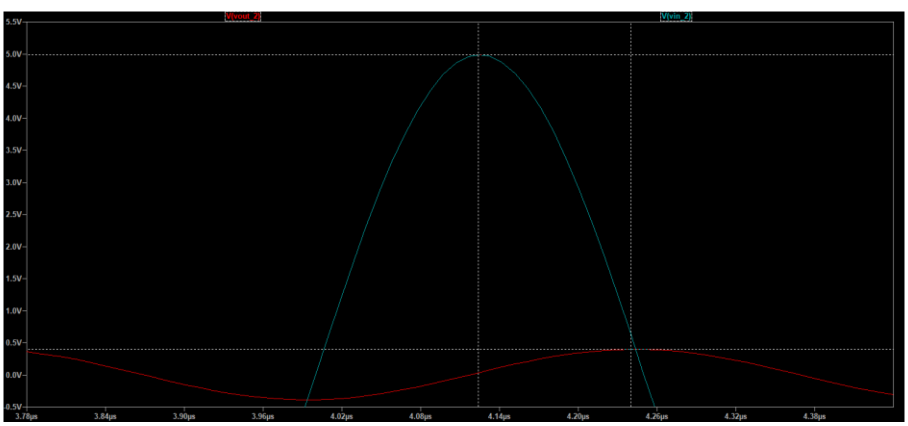
Capture 11 – The simulation of Circuit 2. This simulation is different from Circuit 1 clearly seen as there is a more frequent wave figure. Mark that both simulations capture the time between t=0 to t=10µs. The cyan curve shows Vin and the red curve shows Vout. We can see that there is a time difference between Vout and Vin curves essentially caused by the inductor that is put in again.

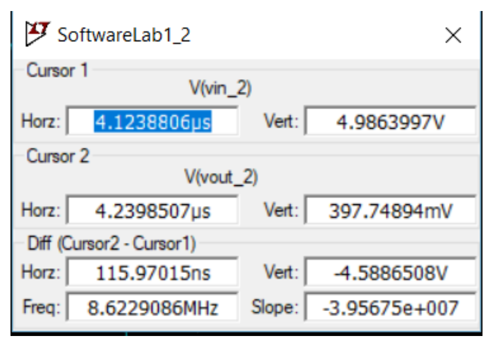
To find each ratio separately, we use the cursor feature. The up-close view of the peak points with cursors and the corresponding value screens are represented below in Captures 12-15.





Capture 12 & 13 – The up-close view of the Circuit 1 curve. Intersections of the dashed lines, which are the cursors are seen at the peak points of the corresponding sinusoids. Also, on the right, we can see the time and voltage values of Circuit 1. As seen clearly, the time difference between the two peaks can be read as 218.71389ns and the two peak voltages are respectively 4.993483V and 788.43265mV.





Capture 12 & 13 – The up-close view of the Circuit 2 peak points. To the right, we can see the time and voltage values of Circuit 1. As seen clearly, the time difference between the two peaks can be read as 115.97015ns and the two peak voltages are respectively 4.9863997V and 397.74894mV.

Since data collection process is done, we can move onto the mathematical analysis that will be used in the solution. First of all, we will be using the transfer function of this circuit which is given below. The transfer function can easily be derived from the voltage divider idea.

$$\frac{Vout}{Vin} = \frac{Rout}{Rout + RL + j\omega L}$$

As we are not interested in the phasor domain for this problem, we are only concerned about the absolute value of the transfer function described as:

$$\frac{Vout}{Vin} = \frac{Rout}{\sqrt{(Rout + RL)^2 + (\omega L)^2}}$$

$$(Rout + RL)^2 + (\omega L)^2 = \frac{Rout^2 * Vin^2}{Vout^2} = (\frac{Rout * Vin}{Vout})^2$$

$$(Rout + RL)^2 + (2\pi * f * L)^2 = (\frac{Rout * Vin}{Vout})^2$$

Considering that we have two equations at the same type, we can interpret this system of linear equations as

$$(2\pi * f_1 * L)^2 = \left(\frac{Rout * Vin_1}{Vout_1}\right)^2 - (Rout + RL)^2$$

$$(2\pi * f_2 * L)^2 = \left(\frac{Rout * Vin_2}{Vout_2}\right)^2 - (Rout + RL)^2$$

Thus, if we subtract the lower equation from the upper one, I can get to the L value easily.

$$(2\pi * f_1 * L)^2 - (2\pi * f_2 * L)^2 = \left(\frac{Rout * Vin_1}{Vout_1}\right)^2 - \left(\frac{Rout * Vin_2}{Vout_2}\right)^2$$

$$L^2 = \frac{\left(\frac{Rout * Vin_1}{Vout_1}\right)^2 - \left(\frac{Rout * Vin_2}{Vout_2}\right)^2}{4\pi^2 * (f_1 - f_2) * (f_1 + f_2)}$$

$$L^2 = \frac{\left(\frac{Rout * Vin_1}{Vout_1}\right)^2 - \left(\frac{Rout * Vin_2}{Vout_2}\right)^2}{4\pi^2 * (f_1 - f_2) * (f_1 + f_2)}$$

$$L = \frac{Rout}{2\pi}\sqrt{\frac{\left(\frac{Vin_1}{Vout_1} - \frac{Vin_2}{Vout_2}\right) * \left(\frac{Vin_1}{Vout_1} + \frac{Vin_2}{Vout_2}\right)}{(f_1 - f_2) * (f_1 + f_2)}}$$

Hence, we can find the inductance value doing the algebra. When L is found, we can calculate the L by just placing the L into its location on one of the equation in the system. Then the RL value becomes

$$(Rout + RL)^2 = \left(\frac{Rout * Vin_1}{Vout_1}\right)^2 - (2\pi * f_1 * L)^2$$

$$RL = \sqrt{\left(\frac{Rout * Vin_1}{Vout_1}\right)^2 - (2\pi * f_1 * L)^2} - Rout$$

$$RL = \sqrt{\left(\frac{Rout * Vin_1}{Vout_1} - 2\pi * f_1 * L\right) * \left(\frac{Rout * Vin_1}{Vout_1} + 2\pi * f_1 * L\right)} - Rout$$

Therefore, by discovering both the values of L and RL, we can easily find the quality factor by simply dividing them and multiplying them with ω. As we have concluded the mathematical processing of our method, we can run a simple MATLAB function to solve the problem and find the corresponding answer. Hence, the MATLAB function findInd2 is given down below.

Editor - C:\Users\Ayhan\Desktop\Bilkent EE\Bilkent EE 2017-2018 Spring\EEE202\Lab1\Calculations\findInd2.m

findInd.m | findInd2.m

```
function [L, R, Q] = findInd2(f1, f2, Vin1, Vout1, Vin2, Vout2, Rout)
L= (Rout/(2*pi))*sqrt(((Vin1/Vout1-Vin2/Vout2)*(Vin1/Vout1+Vin2/Vout2))/((f1+f2)*(f1-f2)));
R= sqrt(((Rout*Vin2/Vout2)-(2*pi*f2*L))*((Rout*Vin2/Vout2)+(2*pi*f2*L)))-Rout;
omega = 2*pi*f2;
Q= omega*L/R;
end
```

Capture 14 – The function findInd2 takes input as the frequencies and voltages of both equations also the test resistor. Then first, computes the inductance, followed by the reactance, and finally the quality factor.

Running this simple function on the Command Window and the result is given below, which seems to be the result of our laboratory experiment.

```
Command Window
>> [L2 R2 Q2] = findInd2(1e6, 2e6, 4.993483, 788.43265e-3, 4.9833997, 397.76894e-3, 100)
L2 =
   9.9328e-05
R2 =
    9.8310
Q2 =
   62.8350
```

Capture 15 – The result of the experiment. As can be seen from the picture, found inductance value is very close to the theoretical value which is 99.328e-5H while the real value was 100.000e-5H. Also, the resistance value is equal to 9.8310 which is also close to the real value which is 10. Furthermore, the quality factor of the inductor resulted as 62.8350 which is also nearly equal to the theory in 62.819.

This concludes the usability of the second method as all variables seemed to be correct in some amount of error as this is finite precision mathematics. Overall, both of the methods worked more or less to give the intended result. Hence, we can conclude this as a successful software lab.

Hardware Implementation of Method 2

This method takes advantage of the different Vout, Vin ratios that come from the transfer functions. Furthermore, it uses different frequency values, thus the expected error for this method is definitely higher than the first method as this time we use four different measurements for each finding, making the error quadruple. In this method we use the oscilloscope again in the same circuitry, but this time in order to read the values from the measure section of the oscilloscope. The collected data for each of these different voltages are given below.

The Winded Inductor with 4.73µH Inductance						
f1	**f2**	**Vin1**	**Vout1**	**Vin2**	**Vout2**	**Rout**
1.00E+06	2.00E+06	6.90	6.70	7.10	6.30	100.00
2.00E+06	3.00E+06	7.10	6.30	7.50	5.80	100.00
1.00E+06	4.00E+06	6.90	6.70	8.80	3.80	100.00
1.00E+06	3.00E+06	6.90	6.70	7.50	5.80	100.00
The Pre-Made Inductor with 10µH Inductance						
f1	**f2**	**Vin1**	**Vout1**	**Vin2**	**Vout2**	**Rout**
1.00E+06	2.00E+06	6.80	6.10	7.70	5.30	100.00
2.00E+06	3.00E+06	7.70	5.30	9.00	4.80	100.00
1.00E+06	4.00E+06	6.80	6.10	8.80	3.80	100.00
1.00E+06	3.00E+06	6.80	6.10	9.00	4.80	100.00

This table shows the collected data that is going to be used in the analyzation of the inductance values.

When these values are put through the corresponding MATLAB code for the second Method, the following table is collected.

Winded Inductor		
L	**R**	**Q**
4.20E-06	0.46	57.37
4.50E-06	2.61	21.67
8.50E-06	12.03	4.44
4.40E-06	1.79	15.44
Standard Inductor		
L	**R**	**Q**
8.56E-06	-0.26	-206.86
6.66E-06	39.10	2.14
8.34E-06	1.61	32.55
8.48E-06	2.10	25.37

As we can see in this table, while the inductor values are mostly consistent, the resistance values tend to jump between numbers more than expected. However, when the constructed method is observed, we see that, in order to find the resistance, we also used the before-found inductance value, which enables us to come across chaotic errors. 5.4µH is the corresponding average inductor value for the winded inductor, which is somewhat similar to the 4.73µH we have calculated. To analyze the method as a whole. It is pretty good in terms of finding the inductance, but if it is required to find the resistance, I believe Method 1 is better in terms of consistency.

Conclusion

To conclude the software lab, this was overall a successful software lab and both of the methods were successful and usable in the hardware lab. There were some errors of course throughout, however these errors were not theoretical errors but finite precision errors. The most important error that I have encountered was the LTSpice's inability to find maximums and minimums precisely, thus It made even harder to work with the small numbers with only very little availability for mistakes. This laboratory exercise taught me about Spice Software in the form of LTSpice and I have become familiar with it in a short notice. The way how I chose to implement the given lab assignment was focused on the RL circuits, the phasor domain, phase angle and transfer functions.

Furthermore, if we look at the hardware lab, Things got more confusing in terms of finding good and usable data than in the software lab. The confusing part was that the high frequency that we have used through the experiment led us to more erroneous results. Although the results that we have used through the software lab were nearly identical to the calculated values. This time when physical factors were involved, it was harder to collect data and one of my methods became nearly unusable in terms of finding internal resistance. Overall, it was an instructive lab assignment as it has taught me how much difference can be observed between the virtual life(LTSpice) and the real life(Hardware Lab).