

Experiment 2: Wireless Power Transfer

Phasor analysis, mutual inductance, and ac power analysis

ECE 202: Circuits II

Contents

I.	Introduction.....	2
II.	Background.....	2
	a. Coupled Inductors in WPT	2
	b. Phasor Transform.....	3
	c. AC Circuit Power Analysis.....	6
	d. Impedance Matching.....	7
III.	Prelab Exercises	9
IV.	Laboratory Experiment	11

I. Introduction

Wireless power transfer (WPT) has gained considerable attention in recent years as a means to transfer energy wirelessly over short distances. WPT provides user conveniences in mobile device charging, safety in hazardous environments, and the potential to expand driving range in electric vehicles.

Though other approaches exist the most common implementation of short-distance WPT uses a pair of loosely coupled inductors to transfer power through magnetic fields. Though not rigorously defined, “short-distance” refers to “near-field” coupling, which occurs when the transmitting and receiving coils have a diameter larger than the distance between them.

In this experiment, we will investigate a low-power WPT setup and examine some of the parameters that need to be considered in designing such a system.

The goals of this experiment are

- Applying phasor analysis and ac power analysis to an experimental circuit
- Designing impedance matching circuits
- Observing impedances changing with frequency
- Measuring time-domain signals and correlating them to their phasor transformation
- Investigating changes in coupling between inductors

II. Background

a. Coupled Inductors in WPT

Inductive power transfer (IPT) is a type of WPT that employs a pair of coupled inductors to transfer energy across a small gap. An example coil, and the one used in this experiment, is shown in Figure 1. The coil has 35mm diameter and 3mm height; it consists of 27 turns of wire above a high permeability magnetic material. The magnetic material (the dark grey disc in Figure 1) acts as a near short-circuit for the magnetic field which shields area below the coil from magnetic field and increases the inductance of the coil.

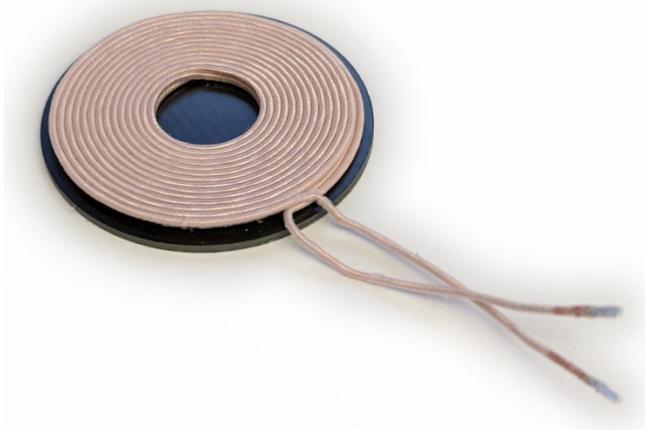


Figure 1. Abracon AWCCA-RX350300-101 WPT coil

A diagram of the magnetic field of a pair of these coils in close proximity is shown in Figure 2. The diagram shows a 2D slice, of two vertically-aligned coils, through the center of each coil. The two coils have 35mm diameter, and are separated by 9mm. Many of the flux lines in Figure 2 enclose (at least in portion) both

coils. This indicates that the two inductors are coupled: flux produced by current through one coil will result in a time-varying $d\Phi/dt$ in the other coil, thereby inducing a voltage on the second coil. In Experiment 1, we modeled the inductance, voltage gain, and coupling of a pair of coupled coils.

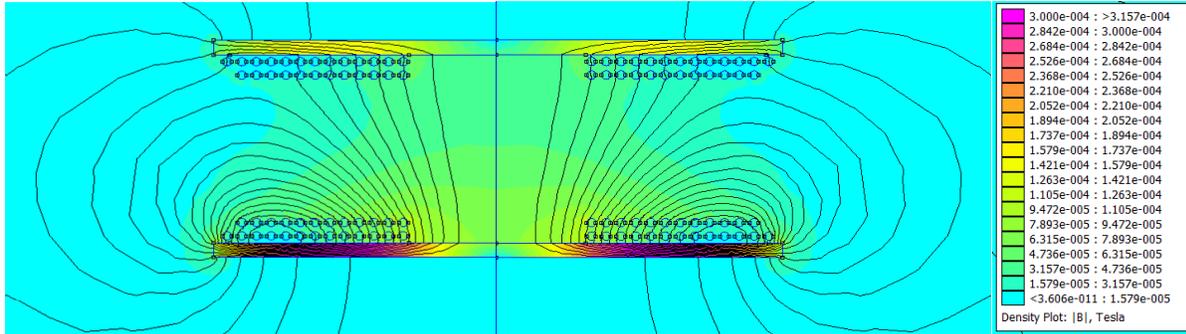


Figure 2. Finite element simulation of the magnetic field of two WPT coils with a distance of 9mm between them.

Individual inductors, such as those that make up WPT coils, are often characterized by their quality factor, Q . Quality factor is defined at a specified frequency f , as

$$Q = \frac{\omega L}{R} \quad (1)$$

where R is the parasitic series resistance of the (non-ideal) inductor. Higher quality factor inductors lose a smaller percentage of their energy to heat per period of sinusoidal current flowing through them. When used in WPT applications, it is critical to operate with high quality factor to prevent the coils from overheating. One way to do this is to increase the operating frequency, which increases the numerator in (8). However, for reasons explained in latter courses, the resistance R increases with frequency. At high enough frequency, this increase is quite rapid, and outpaces the increase in the numerator of (9), establishing a limited range over which increased frequency can benefit Q . In this lab, we will be working at low power levels, such that heating and field strength are not a concern. However, the main limitation for current commercial wireless phone chargers is the low power levels necessary to prevent the onboard coils from overheating.

The changing Q with frequency is also one (of many) reasons why it is desirable to operate WPT systems with single-frequency, sinusoidal currents and voltages. If the system operates at a single frequency, it can be tuned to operate with high efficiency. If nonsinusoidal waveforms are present, the coils may exhibit a suboptimal Q -factor. Because we will intentionally operate with single-frequency sinusoidal waveforms, we can model a WPT system accurately using the phasor transform.

b. Phasor Transform

The phasor transformation leverages Euler's formula to simplify the process of solving differential equations presented by linear, time-invariant circuits where only the steady-state behavior is needed, and all sources and signals are sinusoidal. Euler's formula states

$$e^{j\theta} = \cos(\theta) + j\sin(\theta) \quad (2)$$

Thus, if we have some real signal (voltage or current), $u(t) = A\cos(\omega t + \varphi)$,

$$u(t) = \text{Re}\{Ae^{j(\omega t + \varphi)}\} \quad (3)$$

and the derivative of the signal

$$u'(t) = \omega A \cos(\omega t + \varphi + 90^\circ) = \operatorname{Re}\{\omega A e^{j(\omega t + \varphi + \pi/2)}\} \quad (4)$$

The right-hand-side of (11) can be rewritten, breaking apart the exponent and noting that $e^{j(\pi/2)} = j$ as

$$\operatorname{Re}\{\omega A e^{j(\omega t + \varphi + \pi/2)}\} = \operatorname{Re}\{(j\omega) A e^{j(\omega t + \varphi)}\} \quad (5)$$

Comparing (12) and (10), we notice that the time-derivative operator equates to a multiplication by $j\omega$ inside the $\operatorname{Re}\{\cdot\}$ operator when we apply Euler's formula. All of the preceding is just mathematical manipulation, expressing the same things in different forms. To generate a phasor transformation, we now make a limiting assumption: if we know all the signals in the circuit are real, and all signals are sinusoids about the same frequency ω , we can simplify our notation by dropping the $\operatorname{Re}\{\cdot\}$ operator and the $e^{j\omega t}$ term, as they will show up in every signal. Note that this operator and term *are still present*, we just don't bother writing them. Then, our example signal can be expressed in phasor notation as

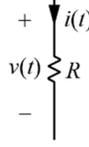
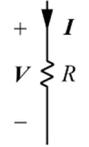
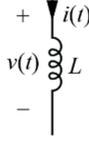
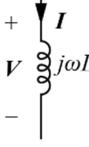
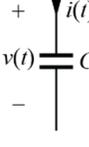
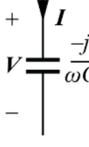
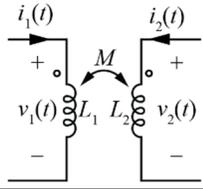
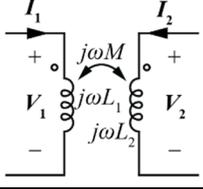
$$u(t) = A \cos(\omega t + \varphi) \xrightarrow{\text{Phasor Trans.}} \mathbf{U} = A e^{j\varphi} \xrightarrow{\text{shorthand}} \mathbf{U} = A \angle \varphi \quad (6)$$

Where the phasor signals are denoted with bold symbols. Note that (13) does not use “=” between terms; this is a transformation which only applies if there are no transients present and a single frequency from all independent sources.

The benefit of this transformation comes in the power of the relationship identified earlier: a time-derivative of the time-domain signal becomes multiplication by $j\omega$ in the phasor transformed signal. In linear circuit analysis, we incur addition/subtraction of signals in KVL and KCL equations, multiplication or division by constants (e.g. R , L , and C) and the time-derivative operator applied to signals with inductors and capacitors are present. The latter results in differential equations that need to be solved in order to solve the circuit operation. In the phasor domain, addition/subtraction and multiplication/division by a constant are unchanged, but the time-derivative operator becomes just multiplication by a constant, $j\omega$. Thus, instead of solving differential equations, we can solve algebraic equations to determine circuit operation.

To do so, we look at how the terminal equations of basic circuit elements behave on phasor-domain signals.

Table I: Basic Circuit Elements in the Time and Phasor Domains

Element	Time-Domain Symbol	Time-Domain Equation(s)	Phasor-Domain Symbol	Phasor -Domain Equation(s)
Resistor		$v(t) = i(t)R$		$\mathbf{V} = \mathbf{R}\mathbf{I}$
Inductor		$v(t) = L \frac{di(t)}{dt}$		$\mathbf{V} = j\omega L\mathbf{I}$
Capacitor		$i(t) = C \frac{dv(t)}{dt}$		$\mathbf{V} = \frac{-j}{\omega C}\mathbf{I}$
Mutual Inductors		$\begin{aligned} v_1(t) &= L_1 \frac{di_1}{dt} + M \frac{di_2}{dt} \\ v_2(t) &= M \frac{di_1}{dt} + L_2 \frac{di_2}{dt} \end{aligned}$		$\begin{aligned} \mathbf{V}_1 &= j\omega L_1 \mathbf{I}_1 + j\omega M \mathbf{I}_2 \\ \mathbf{V}_2 &= j\omega M \mathbf{I}_1 + j\omega L_2 \mathbf{I}_2 \end{aligned}$

Note that now all circuit elements exhibit an Ohm's-law-like behavior. That is, each of the elements' voltages and currents are related by a constant scale factor, with the exception of mutual inductance where there is also a multiplicative relationship with the coupled coil's current. Thus, we define Ohm's law for phasors as

$$\mathbf{V} = \mathbf{I}Z \quad (7)$$

where Z is a complex number called "impedance" of the element. For resistors, $Z_R = R$, for inductors $Z_L = j\omega L$, and for capacitors $Z_C = -j/(\omega C)$. Being a complex number, impedance has both real and imaginary parts,

$$Z = R + jX \quad (8)$$

where the real part, R , is a resistance and the imaginary part, X , is called reactance. Manipulations of impedances and phasors follow mathematics for working with complex numbers.

With this transformation in our toolkit, we can easily solve steady-state operation of circuits with single-frequency sinusoidal sources. All previous circuit analysis techniques work as they did for resistive circuits, including e.g. KVL/KCL, nodal and mesh analysis, source transformation, voltage/current dividers, etc.

The procedure for solving a circuit using the phasor transformation is as shown in Figure 3: rather than taking the hard path to a solution, which requires solving differential equations, we use the phasor transformation to convert the system of differential equations into a system of algebraic equations, which is much easier to solve.

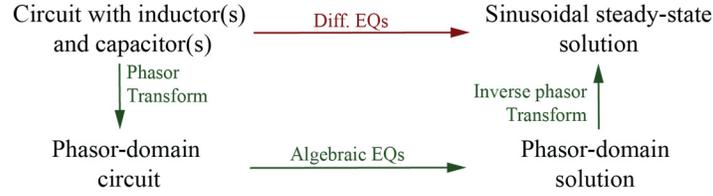


Figure 3. Transform-based solution to circuit equations

c. AC Circuit Power Analysis

The utility of the phasor transformation stems from the nature of sinusoids and their derivatives, and the fact that basic circuit analysis consists of only linear functions applied to signals (i.e. addition/subtraction, multiplication by a constant, and the time-derivative). The calculation of power, however, is not a linear function and thus is not preserved in the phasor transformation.

Consider a resistor with a nonzero dc current I_1 flowing through it. We know this will result in a power dissipation

$$P_1 = I_1^2 R \quad (9)$$

With a different nonzero dc current flowing through it, the power dissipation is

$$P_2 = I_2^2 R \quad (10)$$

If we apply both currents at the same time,

$$P_{1+2} = (I_1 + I_2)^2 R \neq P_1 + P_2 \quad (11)$$

Thus, superposition is violated. Because superposition is a characteristic of linear functions, we can conclude that the calculation of power is not a linear function.

Therefore, we need to return to the time-domain for calculations of power at any port in the circuit. Fortunately, we are able to develop a simple formula for how the correct calculation of power relates to the phasor voltages and currents that allows us to bypass having to convert back to the time domain in our analysis.

In the time domain, for voltage and current at a port

$$\begin{aligned} v(t) &= V_A \cos(\omega t + \varphi_V) \\ i(t) &= I_A \cos(\omega t + \varphi_I) \end{aligned} \quad (12)$$

The power $p(t) = v(t) \cdot i(t)$ is

$$p(t) = \frac{V_A I_A}{2} [\cos(2\omega t + \varphi_V + \varphi_I) + \cos(\varphi_V - \varphi_I)] \quad (13)$$

which consists of a sinusoid at 2ω , twice the frequency of both $v(t)$ and $i(t)$, and a constant (i.e. not time-varying) dc term. Taking the average over one period, the average power is

$$P = \frac{V_A I_A}{2} \cos(\varphi_V - \varphi_I) \quad (14)$$

In terms of our phasor transformation, this same value can be written as

$$P = \frac{1}{2} \text{Re}\{VI^*\} \quad (15)$$

Where I^* is the complex conjugate of I , obtained by negating the imaginary part of I . The factor of $\frac{1}{2}$ out front can be removed if rms values for the voltage and current are used,

$$P = \text{Re}\{\mathbf{V}_{\text{rms}}\mathbf{I}_{\text{rms}}^*\} \quad (16)$$

where the rms of a signal is defined about a time t_1 and window of time T , as

$$u_{\text{rms}} = \sqrt{\frac{1}{T} \int_{t_1 - \frac{T}{2}}^{t_1 + \frac{T}{2}} u(t)^2 dt} \quad (17)$$

For periodic signals, the rms over any positive integer number of complete periods is the same as the rms over all time.

If we generalize from (23), we can defined complex power

$$S = \mathbf{V}_{\text{rms}}\mathbf{I}_{\text{rms}}^* = P + jQ \quad (18)$$

which has a real component, P , that is the average (real) power, and an imaginary component, Q , that is reactive power. Complex power is given in units of VA, real power in W, and reactive power in VAR. Power factor is a measure of how close the load is to a resistive load, and is given by

$$PF = \frac{P}{|S|} = \cos(\varphi_V - \varphi_I) \quad (19)$$

Power factor is in the range $0 \leq PF \leq 1$; a power factor of unity indicates that the voltage and current are perfectly in phase, so the net load at that port is resistive.

d. Impedance Matching

Generically, impedance matching is the process of selecting or designing subcircuits to transform a fixed (required) impedance into another value which is more advantageous to circuit operation. Impedance matching may be used between a fixed source and load to maximize power transfer, improve signal-to-noise ratio, correct power factor, increase efficiency, etc.

In wireless power transfer applications, impedance matching plays a critical role in circuit operation. This is because the systems operate at high frequency (100's of kHz to 10's of MHz), and use large, inductive coils to generate the magnetic fields. Because of the large frequency and inductance, the reactance of the coils, $X_L = \omega L$ is large, and results in relatively small current amplitudes when a fixed voltage is applied, and significant reactive power.

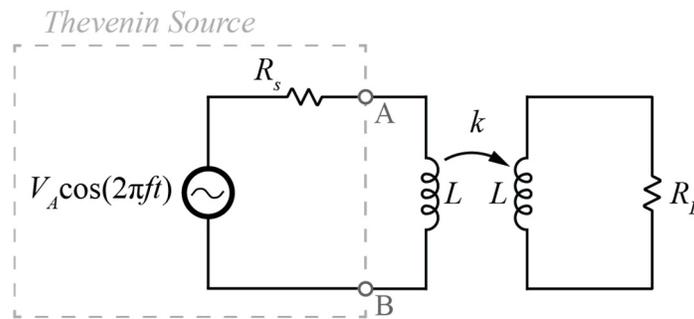


Figure 4. Uncompensated WPT coils with Thévenin Source

For example, consider a pair of identical coils ($L_1 = L_2 = L$) with load resistance R_L as shown in Figure 4. The equivalent impedance between nodes A and B is

$$\begin{aligned} Z_{eq} &= j\omega(L - M) + j\omega M \parallel [R_L + j\omega(L - M)] \\ &= R_L \frac{k^2}{1 + \left(\frac{R_L}{\omega L}\right)^2} + j\omega L \left(1 - \frac{k^2}{1 + \left(\frac{R_L}{\omega L}\right)^2}\right) \end{aligned} \quad (20)$$

which was obtained by replacing the two coils with a T-network, then simplifying. The repeated term,

$$\frac{k^2}{1 + \left(\frac{R_L}{\omega L}\right)^2} \ll 1 \quad (21)$$

Because $0 < k < 1$, and $(R_L/\omega L)^2 > 0$. Thus, the Thévenin source sees an impedance Z_{eq} that has a small real part, but the reactance is nearly that of the coil. As an example, for $k = 0.3$, $R_L = 50 \Omega$, $L = 25 \mu\text{H}$, and $\omega = 2\pi(300 \text{ kHz})$, the resulting impedance is $Z_{eq} = 2.1 + j45.1 \Omega$. As predicted, the reactance is much larger than the resistance. Thus the power factor seen by the Thévenin source is $\text{PF} = 0.05$, lagging, and the power delivery is well below the maximum possible or, equivalently, the power losses in the circuit resistances are much larger than needed to deliver the same power.

If we add a reactance $X_c = -j45.1 \Omega$ in series with the primary coil, we can cancel out this series reactance and produce a purely resistive load for the Thévenin Source, resulting in unity power factor. Note, however, that we won't achieve maximum power transfer unless

$$R_s = \text{Re}\{Z_{eq}\} = R_L \frac{k^2}{1 + \left(\frac{R_L}{\omega L}\right)^2} \quad (22)$$

In wireless power transfer applications, there are four common impedance matching arrangements that are used in most applications. Each compensates both the transmitter coil and receiver coil with a single capacitor, and is characterized by the connection between them, as shown in Figure 5. The series-series (SS) scheme connects one capacitor in series with each coil; the series-parallel (SP) connects one capacitor in series with the primary coil and one in parallel with the secondary coil; the parallel-series (PS) and parallel-parallel (PP) follow the same naming convention. Each has different characteristics in terms of the voltages, currents, and maximum power loads that are produced.

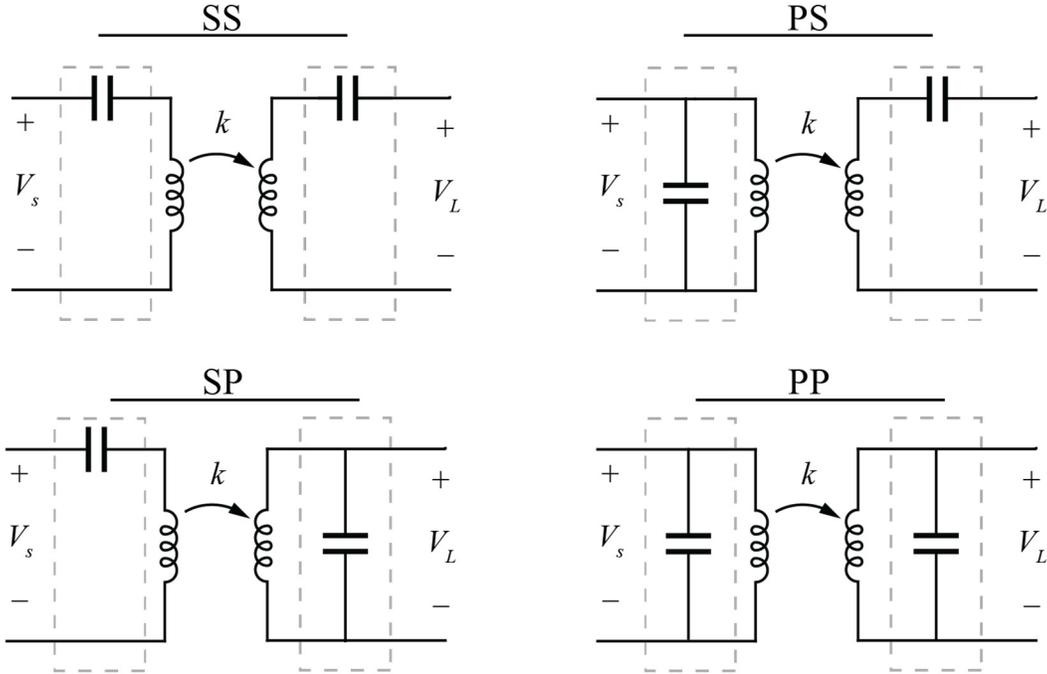


Figure 5. Common compensation schemes for WPT coils

Many more complex matching schemes exist, but they focus on performance metrics beyond our study in this experiment, e.g. presenting a large impedance for all frequencies other than the WPT frequency, or minimizing variations as the load power changes. In this experiment, we will focus on just these four candidate impedance matching schemes.

III. Prelab Exercises

Complete the following exercises prior to coming to the lab. Turn-in your prelab work on canvas prior to starting the Laboratory Exercises.

In this prelab, you will complete calculations and simulations to design the WPT system you want to test in the lab. We will assume that the power source for our WPT system is modeled by a sinusoidal Thévenin source with 10 V amplitude and 50 Ω Thévenin resistance.

The starting point for our design is summarized in Table II.

Table II: Starting Values for System Design

f	L	k (@ 5mm)	R_L	V_{th}, R_{th}	R_p
500 kHz	11 μ H	0.3	25 Ω	10 V, 50 Ω	2 Ω

PE1 Prelab Exercise 1: Maximum Power Calculation

If the source is a sinusoidal voltage source with amplitude $V_{th} = 10$ V and 50 Ω Thévenin resistance, calculate the maximum average power that can be delivered to any load, i.e. what is the maximum average power of $p_L(t) = i_L(t)v_L(t)$ that can be achieved in the circuit of Figure for any Z_L ?

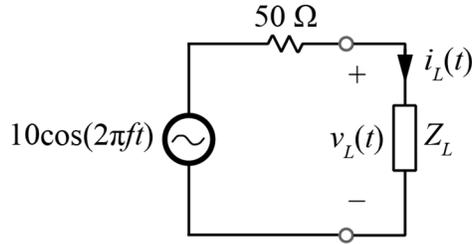


Figure 6. Thévenin source with unknown load

PE2 Prelab Exercise 2: Complex Power Calculation

Consider the WPT circuit of Figure 7. Circuit model for prelab exercises 1 through 3, with parameter values in Table II. By hand, solve for $v_L(t)$, $i_s(t)$, $v_s(t)$, and calculate the complex power S_L dissipated by R_L and the complex power S_s supplied by the Thévenin source (i.e. at the port of $v_s(t)$). Calculate the power factor of the load seen by the Thévenin source.

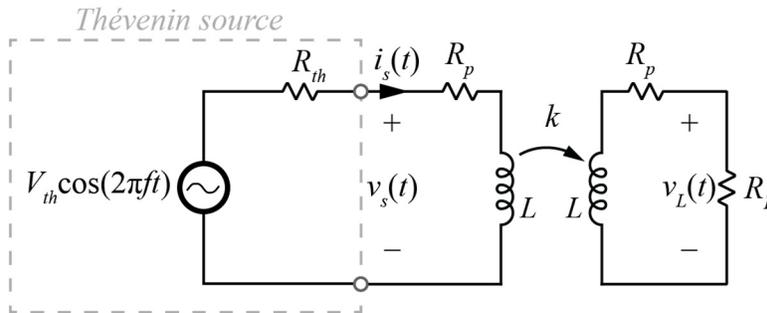


Figure 7. Circuit model for prelab exercises 1 through 3

PE3 Prelab Exercise 3: LTSpice Load Sweep

Using LTSpice, simulate the circuit of Figure 7 with the same parameter values. Sweep the load resistance between 1 Ω and 50 Ω and turn-in a plot of the output power P_L versus R_L .

Note: see the “Simulation Tools” section of the “Materials” page on the course website if you are unfamiliar with how to sweep variables in LTSpice

PE4 Prelab Exercise 4: LTSpice Frequency Sweep

Again with a fixed load resistance of $R_L = 25 \Omega$, sweep the frequency of the source from 100 kHz to 500 kHz. Turn in a plot of the output power P_L versus f .

PE5 Prelab Exercise 5: Compensation Design

Consider the compensation schemes of Figure 5. With only two 10nF capacitors, a load resistance between 1 Ω and 50 Ω, and a frequency between 100 kHz and 1 MHz, decide on a compensation circuit, load resistance, and frequency that will give the maximum power P_L . You may use any combination of

MATLAB, LTSpice, and hand calculations to accomplish this task. Report your selected compensation, R_L , and f , and show justification for your choice.

*Note: You **do not** need to find or derive the global maximum. Rather, find the design that you think is worth building in the experiment.*

Turn in your prelab by submitting it to Canvas prior to your group's experiment. Prelab format does not require any narrative; it can be written up like a normal homework rather than as a "report".

IV. Laboratory Experiment

In this experiment, we will be testing a WPT system made from the WPT coils shown in Figure 1. You will use the following components from the ECE 202 supplemental kit

- 2 PCB coils
- 1 coil holder
- 2 10nF capacitors
- 1 50 Ω potentiometer
- 2 1 Ω resistors

coil holder and an image showing how to place the WPT coils in the holder are shown in Figure 8. The coil holder keeps the coils aligned and separated by 5mm, yielding a coupling factor of slightly greater than $k = 0.3$.

The measured inductance, resistance, and quality factor of the coils are shown in Figure 9. Note that, as mentioned earlier, the resistance increases at high frequency, resulting in a peak quality factor at around 5 MHz. At this point, the resistance of the coils is $R_p = 8 \Omega$.

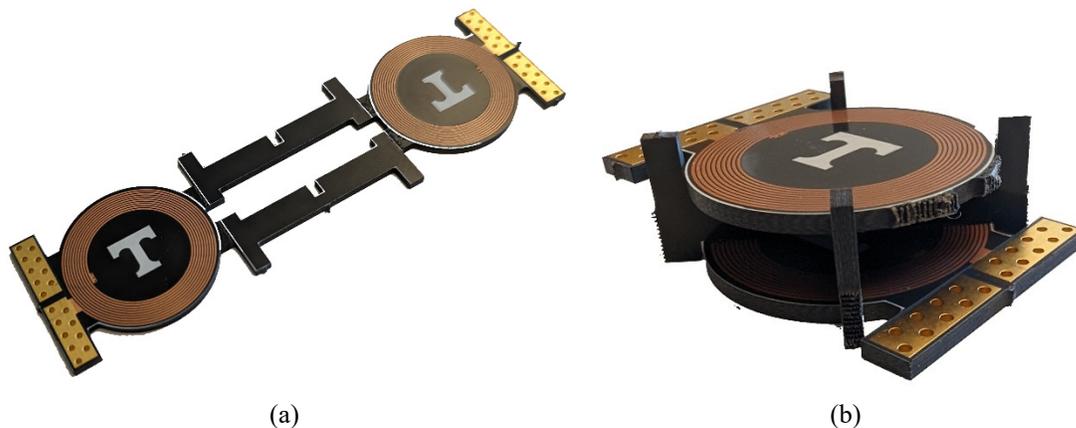


Figure 8. WPT Coil assembly (a) and two WPT coils in the coil holder (b)

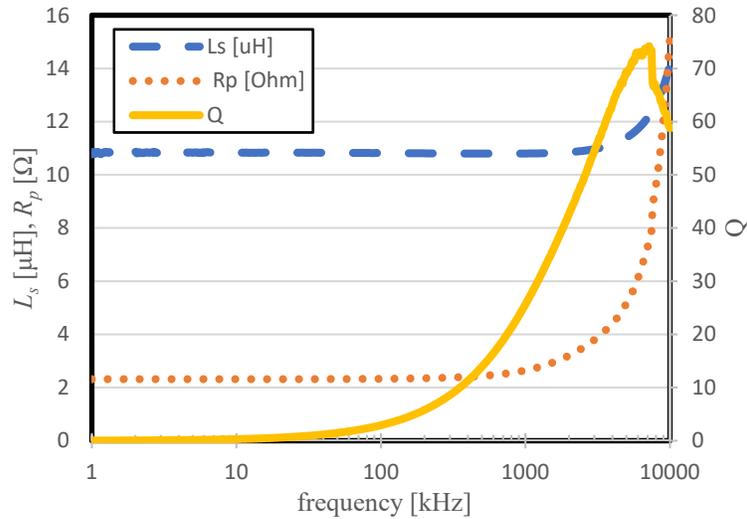


Figure 9. Measured inductance, resistance, and quality factor of one WPT coil.

The power source for our WPT system is the function generator in the lab. The function generators operate very similarly to a Thévenin source, with a Thévenin resistance of $50\ \Omega$ and a maximum amplitude of $V_A = 2\ \text{V}$, as shown in Figure 10.

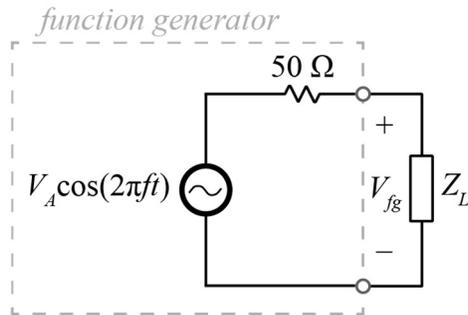


Figure 10. Function generator Thévenin equivalent model

Remember to take pictures of setups and oscilloscope waveforms where necessary for your lab report as you complete these experiments

LE1 Lab Exercise 1: Uncompensated Coil Measurement

The potentiometer will act as our variable load. Adjust the potentiometer to $25\ \Omega$. You can use a multimeter to measure the resistance between the middle and (either of the) outer pins while turning the dial until you reach the desired resistance.

Turn on the function generator and set the output to a sinusoid, $2\ \text{V}$ amplitude, with a frequency of $500\ \text{kHz}$. Connect the coils, $1\ \Omega$ resistor, and $25\ \Omega$ potentiometer (R_L) as shown in Figure 11. The two $1\ \Omega$ resistors are used to allow current measurement, using voltage probes connected across them.

Using the oscilloscope, measure the amplitude and phase of $v_s(t)$, $i_s(t)$, and $v_L(t)$.

Note: be aware that the negative leads of all probes are shorted together internally in the oscilloscope. Make sure that you do not short out any part of your circuit by attempting to connect two probe negatives to two different points in the circuit.

Compute the complex power supplied by the source, the real power going to the load, and the source power factor. Compare your results to your prelab predictions.

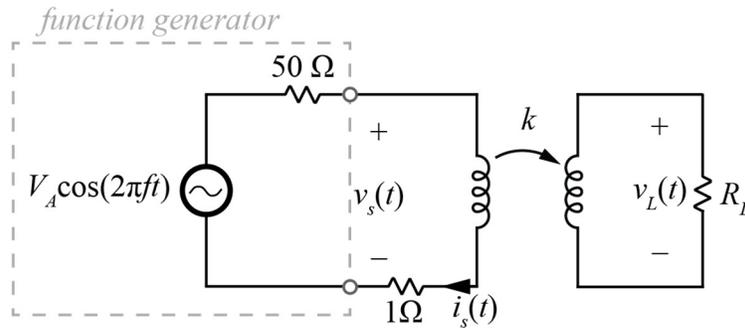


Figure 11. Experimental circuit connections for Lab Exercise 1.

LE2 Lab Exercise 2: Compensated Coil Measurement

As a group, discuss your results from the final prelab exercise. Decide on a *single* compensation scheme to test experimentally. Once you have decided, place the two capacitors in your circuit in the appropriate configuration from Figure 5. With the capacitors in place, but still at $f = 500 \text{ kHz}$ and $R_L = 25 \Omega$, measure and compute the new complex power supplied by the source, real power going to the load, and source power factor. Compare each to the uncompensated system measurements.

LE3 Lab Exercise 3: Compensated Frequency Sweep

With the same circuit from the previous exercise, vary the frequency up and down on the function generator, while observing the load voltage $v_L(t)$ on the oscilloscope. Find the frequency at which this voltage is maximized, limited within the range $100 \text{ kHz} < f < 1 \text{ MHz}$, or to a narrower range. With the load resistance still at a fixed $R_L = 25 \Omega$, the maximum amplitude $|v_L(t)|$ corresponds to the maximum power. Record the frequency at which this occurs and the value of the power consumed by R_L .

LE4 Lab Exercise 4: Compensated Load Sweep

Add an additional 1Ω resistor in series with the load, as shown in Figure 12. This resistance is used to measure the load current. On the oscilloscope, measure both $v_L(t)$ and $i_L(t)$ and use the “math” function of the oscilloscope to display the product of the two, $p_L(t) = v_L(t) \cdot i_L(t)$.

At the frequency found in Exercise 3, vary the load resistance R_L from its minimum to maximum values (roughly 1Ω to 50Ω) by turning the potentiometer. Find the point at which maximum power is delivered to the load. Record the average value of the power, then remove the potentiometer and, using a multimeter, measure the resistance. Add 1Ω to your measurement for the “effective” load resistance, include the resistor used to measure current, and record this value.

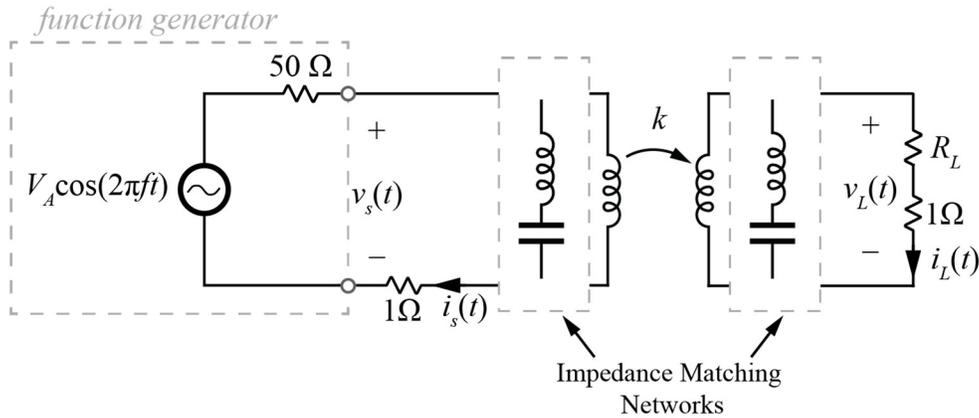


Figure 12. Experimental circuit connections for Lab Exercise 4.

LE5 Lab Exercise 5: Efficiency Measurement

At the load resistance from Exercise 4 and frequency from Exercise 3, measure and compute the complex power supplied by the source, the real power going to the load, and the source power factor. Additionally, compute the efficiency of your circuit at this operating point,

$$\eta = \frac{P_L}{P_s} \times 100 \quad (23)$$

OE1 Optional Exercise 5: Efficiency Maximization

This exercise is completely optional, and is purely extra credit. Not completing it will not negatively impact your grade. If you choose to complete it, you have the opportunity to earn up to 10% extra credit on this lab report.

Using any variation of load resistance, frequency, or orientation of the coil the holder (5mm or 7mm gap), find the maximum efficiency operating point of the system. Report the efficiency, load resistance, compensation scheme, and operating frequency. Extra credit earned on this exercise will be awarded based on how close your measured result is to the true maximum of the hardware.

In your report, make sure to include the results of *all* measurements and calculations requested in the lab. Whenever the exercise ask for a “comparison”, the comparison should be discussed explicitly in the report. When measurements are asked for, you should show the results of the measurements with oscilloscope waveforms (preferred) or tabular data.