

Physics 281
Principle of Physics
Lab Manual

July 2002

Ling Jun Wang

Table of Contents

- Experiment 1. Ohm's Law
- Experiment 2. Resistors in Series and in Parallel
- Experiment 3. Temperature Dependence of Resistance
- Experiment 4. Wheatstone Bridge
- Experiment 5. Impedance Matching
- Experiment 6. RC Constants
- Experiment 7. Oscilloscope
- Experiment 8. I-V curve of Diode
- Experiment 9. Diode Power Supply
- Experiment 10. Magnetic field and Induction
- Experiment 11. Reactance
- Experiment 12. LRC Resonance

Experiment 1. Ohm's Law

Objective: To study Ohm's law for DC circuits; to learn basic construction and measurement of electric circuits; to learn curve fitting and basic error analysis.

Apparatus: voltmeter, ammeter (0-500 mA), resistor (100 Ω), and variable DC power supply (0-10V), computer and interface.

Theory: According to Ohm's law, the voltage across a resistor is proportional to the current passing through it:

$$V = I R \tag{1}$$

where the ratio of the voltage (V) over the current (I) is called the resistance (R). The unit of resistance is ohm, which is equal to one volt per ampere. For common conductors such as metals, the resistance is a constant if the temperature does not change significantly. At certain temperature, the resistance of a resistor depends on its geometric parameters (the length and the cross sectional area), its physical property (resistivity):

$$R = \rho L / A \tag{2}$$

where ρ , L and A are, respectively, the resistivity, the length and the cross sectional area of the resistor. A resistor is called a "linear device", as opposed to a "non linear device" such as a diode that does not obey the linear Ohm's law even without significant change in temperature. In this experiment we will study a regular resistor with constant resistance only.

Procedure:

1. Construct a circuit as shown in Fig. 1:

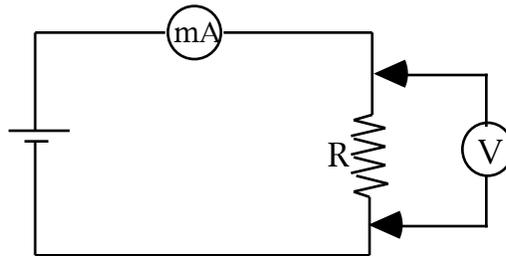


Figure 1.

Note: The voltmeter does not have to be connected to the circuit. You can measure the voltage across any two points by touching the points with the two leads of the voltmeter.

2. Measure the current for ten different voltages, with an increment of one volt. Record the current and the voltage in your data table. Calculate the resistance R_I at each different voltage, using Ohm's Law. You need to keep at least two significant digits for both the measured data and the calculated resistance. Use a data table as suggested in Table 1:

Trail	Voltage (V)	Table 1 Current (mA)	Resistance (Ω)	Deviation ($\%$)
1				
2				

3. Use a computer and interface with a voltage sensor and a current sensor in place of the voltmeter and the current meter to perform the same experiment outlined above. The data will be automatically taken and plotted. Use linear fit to obtain the resistance. Your instructor will tell you about the most recent version of the software in operation, and the folder in which has the program for this experiment. The software might be frequently updated, but should be pretty much self-explaining.

Data Analysis

1. Plot the voltage versus. current data in Table 1 in a linear-linear graph paper as shown in Figure
2. Find the slope which is the experimental value of the resistance.

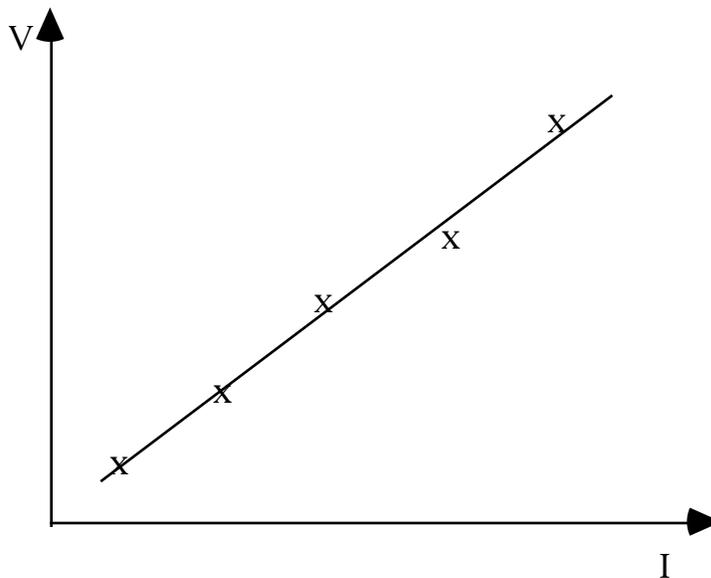


Figure 2

2. Calculate the average resistance R_{av} and the deviation for each trial: $\delta_i = R_i - R_{av}$. (See the suggested data table.)

3. Calculate the standard deviation in resistance as follows:

$$= \text{sqrt} [\sum (\delta_i)^2 / (n-1)] \quad (3)$$

where n is the number of data points, and δ_i is the random error. As you can see, the random error can be reduced to as small as your time allows you to increase the number of measurements.

4. Record the instrumental error in your measurements. Your measurement can not be made absolutely accurate, no matter how many measurements you might make and how small your random error is reduced to. The accuracy of your measurement is still limited by your instrument (meters in this case). For example, you will not be able to measure the thickness of a piece of paper with a meter stick. This type of error is called the instrumental error, and it is usually taken as one

half of the smallest division of the instrument. For digital instrument, the error is taken as one in the lowest digit.

Record the instrumental errors in the voltage and the current measurements.

5. Calculate the instrumental error in resistance as follows:

$$\frac{\Delta R}{R_{av}} = \frac{\Delta V}{V} + \frac{\Delta I}{I} \quad (4)$$

where ΔR , ΔV and ΔI are respectively the instrumental errors in resistance, voltage and current. Enter the mean values of V and I in the denominators in Eq. (4). The ratio $\Delta R/R$ is called the relative instrumental error. Calculate the instrumental error ΔR by multiplying the relative error by the average resistance R_{av} .

6. Combine the random error and the instrumental error to obtain the total error ΔR in your measurement of the resistance:

$$\Delta R = \sqrt{(\Delta R_{rand})^2 + (\Delta R_{inst})^2} \quad (5)$$

7. Use a computer program to plot your data and do a curve fitting. Compare the computer output with your result obtained above. Are they reasonably close?

Question

Can Eq.(4) be used in more general situation than Ohm's Law? Describe the condition under which Eq.(4) is applicable. You will need to use Eq.(4) in the future experiments.

Experiment 2. Resistors in Series and in Parallel

Objective:

1. To study the equivalent resistances of two fundamental connections of resistors - resistors in series and in parallel.
2. To practice more complicated construction of circuits.

Apparatus:

Three resistors with different resistances ranging from 50 to 200 ohms, voltmeter (or voltage sensor), ammeter (or current sensor), DC power supply, computer and interface.

Theory:

In the last experiment we studied Ohm's law for a single resistor. As a matter of fact, Ohm's law applies not only to a single resistor, but also to a part or the whole circuit network, as long as it is non active (no power supplies in that part of the circuit) and linear (no non-linear device in the circuit). For the example shown in Fig.(1), The battery is supplying a device contained in a box B. The device is a complex network of many resistors. To the power supply (say, the Electric Power Board), the whole device can be considered as a single resistor, the equivalent resistance of which is equal to the voltage of the power supply, V , divided by the current, I , flowing into the device. It is also called the "input resistance" (a special case of the more general "input impedance") of the device.

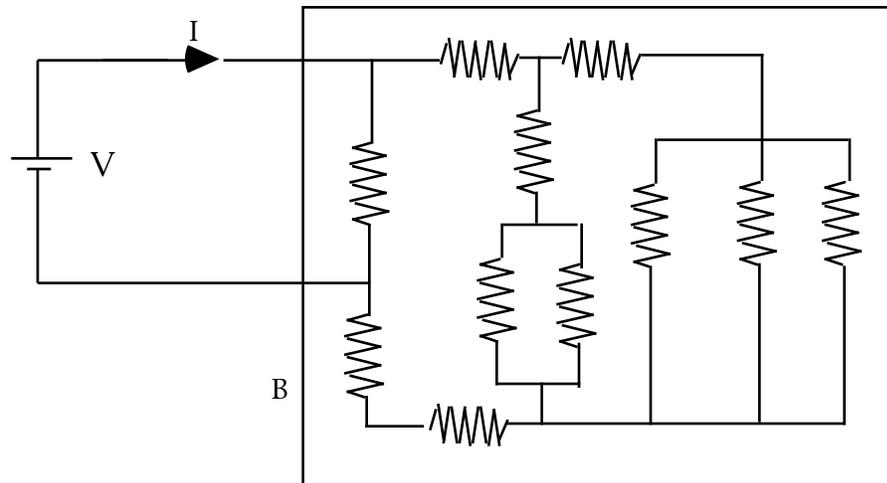


Figure 1

The equivalent resistance depends of the construction of the device, and can be calculated from the individual resistances if the construction is known. In this experiment, we will study the two simplest combinations: resistors in series and in parallel.

If two resistors R_1 and R_2 are connected in series, the current passing through the two resistors are equal, and the total voltage is equal to the sum of the individual voltages across the two resistors (Fig. 2).

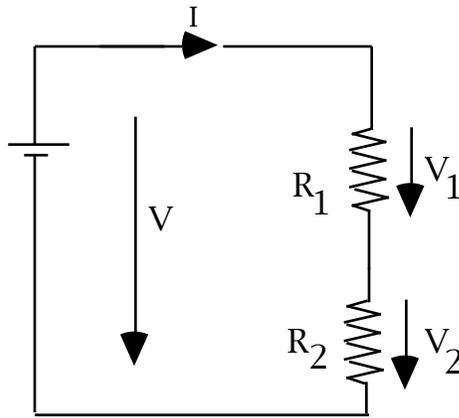


Figure 2

$$V = V_1 + V_2$$

Applying Ohm's law for all terms in the above equation, we have

$$I R = I_1 R_1 + I_2 R_2$$

Since $I = I_1 = I_2$, we obtain

$$R = R_1 + R_2 \tag{1}$$

Equation (1) says that the equivalent resistance of the resistors connected in series is equal to the sum of the individual resistances. It is not hard to see that this rule applies to any number of resistors connected in series.

On the other hand, if two resistors R_1 and R_2 are connected in parallel, the voltages across the two resistors are equal, and the total current is equal to the sum of the individual currents passing through the two resistors (Fig. 3).

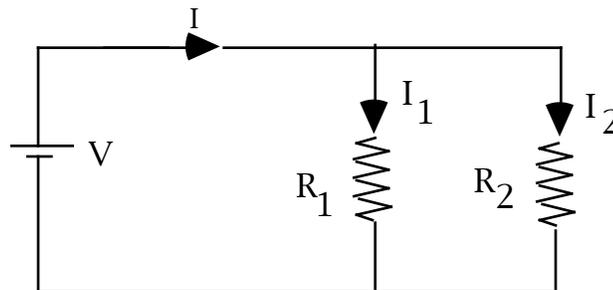


Figure 3

$$I = I_1 + I_2$$

Again applying Ohm's law to the terms in the above equation, we have

$$\frac{V}{R} = \frac{V_1}{R_1} + \frac{V_2}{R_2}$$

Since $V = V_1 = V_2$, we obtain the relationship between the equivalent total resistance R and the individual resistances R_1 and R_2 :

$$\frac{1}{R} = \frac{1}{R_1} + \frac{1}{R_2} \quad (2)$$

Likewise, the above relationship applies to any number of resistors connected in parallel.

Equations (1) and (2) can be used to find the equivalent resistance of more complex networks such as the ones shown in figures (4) and (5). It is left to the students to figure out the ways to do it in the following procedure.

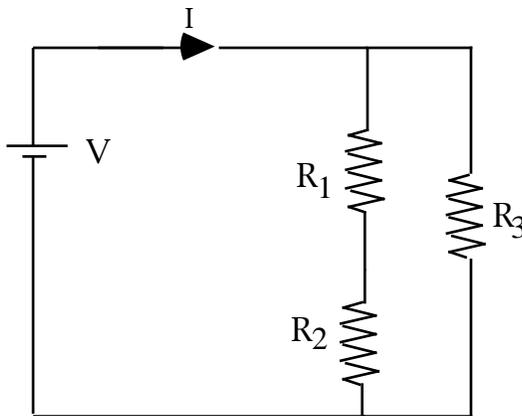


Figure 4

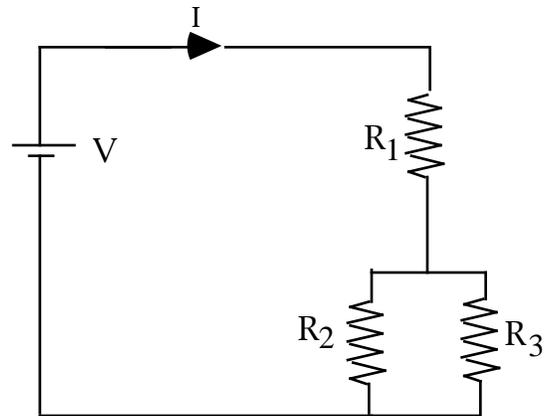


Figure 5

Procedure

1. Turn on the computer and interface. Use the voltage sensor and the current sensor to measure the voltages and currents. Get to the appropriate program to run this experiment with the help of your instructor. Use the method learned from the previous experiment to measure the resistances of the three resistors R_1 , R_2 and R_3 . Record these values in your data table. You do not have to make a hard copy of the plot, but do use the linear fitting program to obtain the resistances.
2. Construct a circuit in series like Figure 2, but with three resistors in series instead of two. Run the program again to obtain the total resistance of the three resistors in series. Record the value in your data table. Meantime, calculate the theoretical total resistance by making use of Eq. (1) and the individual resistances obtained in step 1. Calculate the difference between the theoretical and the experimental values. Find the percentage error.
3. Construct a circuit in parallel like Figure 3 with three resistors. Repeat the procedure stated in step 2 to find both the experimental and theoretical equivalent resistances and the percentage error.
4. Construct a mixed circuit of Figure 4. Repeat the measurement and calculation described above.
5. Construct a mixed circuit of Figure 5 and repeat the measurement and calculation.

Table 1

$R_1 =$ _____ , $R_2 =$ _____ , $R_3 =$ _____

Circuit	measured R (ohm)	calculated R (ohm)	difference (ohm)	percent error
Figure 2				
Figure 3				
Figure 4				
Figure 5				

Experiment 3. Temperature Dependence of Resistance

Objective:

To understand the non linearity nature of the resistivity and its dependence on temperature. To measure the temperature of a light bulb by measuring its resistance (the same working principle of the thermistors).

Apparatus:

A light bulb (3 to 5 v), computer and interface, voltage and current sensors.

Theory:

In the previous studies of Ohm's Law, the resistance is assumed constant. Such assumption is justified under certain conditions. One of the condition is that the environmental temperature is relatively constant. If the temperature changes significantly, the resistance would change. For metals, the resistance is caused by molecular vibration, and it is therefore greater at a higher temperature due to more violent vibration. For semiconductors, the dominant cause for change of resistance is the change in the concentration of the electrons and hole excited to the conduction band, and the resistance is lower at a higher temperature. The effect of temperature on semiconductors is too complicated and dramatic to be simply described by temperature dependence of their resistances. In this experiment we will study the temperature dependence of resistance of a metal -- the tungsten filament of a light bulb. The temperature dependence of metals can be expressed in a simple equation:

$$R_T = R_0 [1 + \alpha (T - T_0)] \quad (1)$$

where R_T is the resistance at the temperature T and R_0 is the resistance at the temperature T_0 . α is called the temperature coefficient. The temperature coefficients of the metals are carefully measured and tabulated in the various handbooks for reference. The temperature coefficient of tungsten is $4.5 \times 10^{-3} \text{ K}^{-1}$. ($1 \text{ }^\circ\text{C} = 1 \text{ }^\circ\text{K}$.) T_0 is usually the room temperature of $20 \text{ }^\circ\text{C}$. If both R_0 and T_0 are measured and α is known, the resistance R_T can be used as a measure of the temperature of the resistor.

Procedure:

1. Construct a circuit shown in Fig. 1.

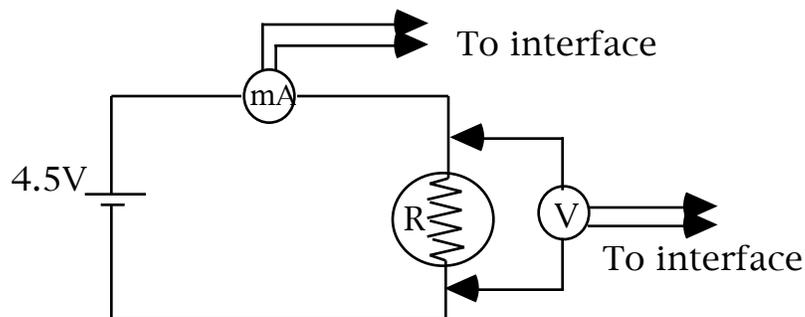


Figure 1.

2. Switch on the battery and adjust the voltage to about 0.5 v. Measure the current and the voltage across the bulb R with the voltage and current sensors interfaced to a computer. Then increase the voltage by 0.5 v and take another data point. Repeat the measurement for every increment of 0.5 v until the voltage reached (but do not exceed) 5 V.

3. Plot the voltage versus the current. You should get a smooth non linear curve. Try to fit the curve with a quadratic or exponential functions. Report this function and include the plot in your data analysis.

4. Create another column in your data table for the resistances of all the data points. Calculate and plot the resistances versus the current.

5. Use Eq. (1) to calculate the temperature change $T - T_0$ for the filament. (The temperature coefficient of tungsten is $4.5 \times 10^{-3} \text{ K}^{-1}$.) Estimate the final temperature of the bulb assuming the initial temperature $T_0 = 20 \text{ }^\circ\text{C}$. This should be a reasonable estimate of the working temperature when the bulb glows.

6. Create another column for the temperatures of all the data points, again assuming the initial temperature $T_0 = 20 \text{ }^\circ\text{C}$. Plot the resistance versus the temperature. Include the plot in your data analysis.

Questions

The initial temperature is not really known. We assume $T_0 = 20 \text{ }^\circ\text{C}$. How much would the assumed initial temperature affect your estimate of the final temperature of the filament if the initial temperature is, say, $100 \text{ }^\circ\text{C}$ instead ?

Experiment 4. Wheatstone Bridge

Objective:

To learn the operation of Wheatstone Bridge.

Apparatus:

Wheatstone bridge, resistors (in the 100 Ω range).

Theory:

We have learned from the previous experiments how to measure the resistance using a voltmeter and a current meter. The accuracy of such measurements is limited by the internal resistances of these meters. Ideally, we need the internal resistance of a voltmeter to be infinity, and that of an ammeter to be zero. But that is not possible and all the measurements with these meters will have unavoidable instrumental errors.

In this experiment we will learn a way to measure the resistance precisely by using an equipment called Wheatstone Bridge. The principle of the operation of it is depicted in Figure 1.

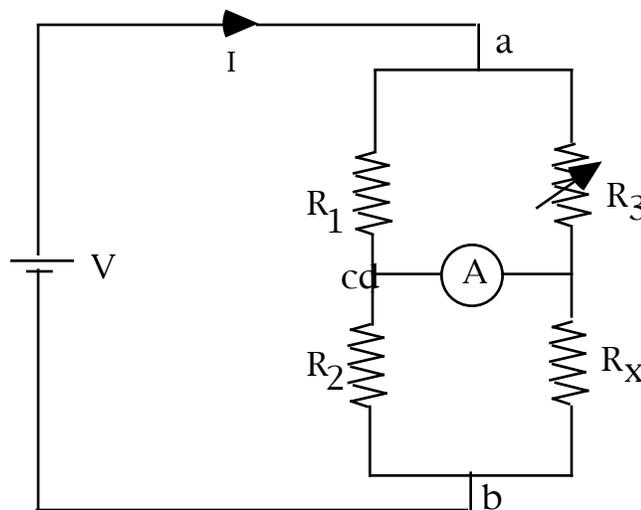


Figure 1. Wheatstone Bridge

As shown in the diagram, the unknown resistor R_x and the other three resistors R_1 , R_2 and R_3 form a circuit called a "bridge". A sensitive current meter connects the two points c and d. R_3 is a variable resistor. In operation, we vary the resistance of R_x such that the current between the points c and d is zero. This process is called "balancing" of the bridge. The current meter A is connected there exactly for balancing the bridge. When the bridge is balanced, the current passing through the resistors R_1 and R_2 is the same, and that passing through R_3 and R_x is the same. We then have the relationship:

$$R_1 / R_2 = V_{ac} / V_{cb} \quad (1)$$

$$R_3 / R_x = V_{ad} / V_{db} \quad (2)$$

Furthermore, since there is no current running between the points c and d, these two points are actually equipotential, i.e., $V_{ac} = V_{ad}$ and $V_{cb} = V_{db}$. We then have

$$R_1 / R_2 = R_3 / R_x \quad (3)$$

or,

$$R_x = (R_2 / R_1) R_3 \quad (4)$$

In a commercial bridge, the resistors R_1 , R_2 , and R_3 are made very precise. The ratio R_1/R_2 and the value of R_3 can be read directly from the front panel.

Since the accuracy of the measurement crucially depends on the smallness of the current through the ammeter, a galvanometer with high sensitivity is used. The galvanometer is therefore very delicate and can be easily damaged if excessive current is let to pass it. To prevent this, the galvanometer is first bypassed with a shunt resistor for a coarse adjustment so that the bridge is roughly balanced. This is done by pressing the Low Sensitivity button on the panel. Whence the bridge is balanced at the low sensitivity, the finer adjustment can be achieved by pressing the High sensitivity button. **NEVER PUSH THE HIGH SENSITIVITY BUTTON BEFORE THE BRIDGE IS BALANCED AT THE LOW SENSITIVITY.**

Procedure

1. Connect the unknown resistance and batteries (3 to 6V) to the bridge as indicated on the panel.
2. Choose the appropriate ratio and the decade resistors so the product will be close to the guessed value of the resistance being measured.
3. Depress the Low Sensitivity button and adjust decade resistors for zero galvanometer reading.
4. Depress the High Sensitivity button and if necessary, further adjust the decade resistors for zero galvanometer reading.
5. When the bridge is thus balanced, the unknown resistance is equal to the product obtained by multiplying the sum of the decade resistor readings by the ratio multiplier. Record the resistance.
6. Repeat the above procedures to measure five more resistors and record their resistances.

Question:

You might report the error of your measurements to be one in the last digit multiplied by the ratio. Is that the only error source in such measurements? What are the other possible error sources related to the measurement with a Wheatstone bridge?

Experiment 5. Impedance Matching

Objective:

To study impedance matching and the method of measuring the output impedance.

Apparatus:

DC power supply box with a battery (4.5 V) and an internal resistance of a few hundred ohms, resistor box, computer and interface, voltage and current sensors.

Theory:

Often times we need to connect two electronic networks (such as a battery to a resistor, a sampling circuit to an amplifier, a power amplifier to a speaker and so on). The best connection is achieved if the output impedance, including both the resistance and the reactance, of the source is equal to the input impedance of the load. This is known as impedance matching. There are two important advantages for impedance matching: When the impedance is matched, the power output is the greatest, and there is no reflection of the traveling waves. This principle is applied in electronics as well as in the cable telecommunication, microwave communication and optical fiber communication.

In this experiment, we will only look at the power aspect of impedance matching, and we will use the DC voltage only. Namely, we are going to study a special case of impedance matching -- resistance matching. As shown in Figure 1, the load resistor R is connected to a source with an internal resistance r . Do not expect the internal resistance to be small, because most electronic devices have significant output impedance and our experiment is a simulation of the electronic impedance matching.

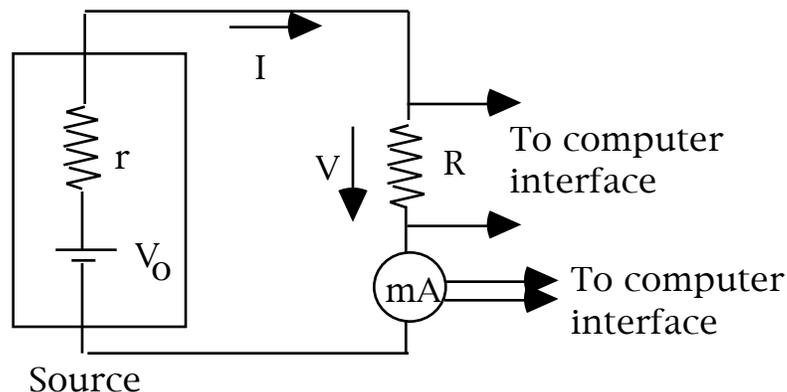


Figure 1

The current is given by

$$I = V_o / (r+R) \quad (1)$$

The voltage across the load resistor is

$$V = I R = R V_o / (r+R) \quad (2)$$

And the power delivered to the load resistor is

$$P = V I = R [V_o / (r+R)]^2 \quad (3)$$

Figure 2 is a plot of the power delivered to the load versus the ratio of the load resistance over the internal resistance of the source. The power is small when R is small because of the small voltage drop. In this case, most of the power is consumed by the internal resistance of the source. On the other hand, if the load resistance is too large, the power delivered is also small simply because the current gets smaller. When the ratio of R/r is equal to unity ($R=r$), the power delivered is the maximum. This is the condition of impedance matching.

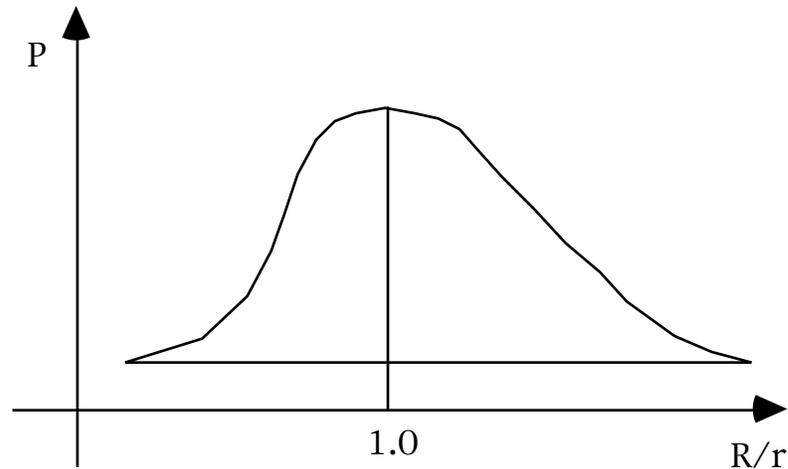


Figure 2

Procedure

1. construct a circuit as shown in Figure 1.
2. Start with a small load resistance $R = 100$ ohms. Measure and record the voltage across the load and the current passing through it, together with the corresponding resistance.
3. Increase the resistance by 100 ohms and repeat the measurement again until the maximum resistance of the box is reached.
3. Add another column of data for the power delivered, P , which is equal to the product of the voltage across the resistor box and the current. Plot the power versus the load resistance.
4. Find the load resistance corresponding to the maximum delivered power P_{\max} . This is the experimental value of the internal resistance of the source. Compare this value with the nominal value of the internal resistance obtained from your instructor. Find the percentage error of your measurement.

Discussion and Question

1. Include in your data analysis a mathematical proof (using calculus) that the maximum power delivered to the load is arrived at when the load resistance is equal to the output resistance of the source.
2. Describe a method to measure the output impedance of a source.

Experiment 6. RC Constants

Objective:

To study the RC circuits and the method of measuring the time constants. To learn the plotting and data analysis of the processes involving logarithmic functions.

Apparatus:

capacitor (about 24 μF), two resistors (about 1 M and 1 k Ω), DC power supply, double throw switch, computer and voltage sensor.

Theory:

A useful device for storing charge and energy is the capacitor, which consists of a pair of parallel conducting sheets, closely spaced but insulated from each other. The magnitude of the charge on either conductor is proportional to the voltage across the two conductors. The ratio of the charge to the voltage is called the capacitance:

$$C = Q / V \quad (1)$$

Capacitors have many uses ranging from storage of energy and charge, electronic tuning, filtering of high frequencies, constructing delay lines, oscillation tanks, pulse shaping, voltage doubling. AC coupling and so on.

As shown in Figure 1, the capacitor C charged to a voltage V_0 will start to discharge through the resistor R as soon as the switch is turned on.

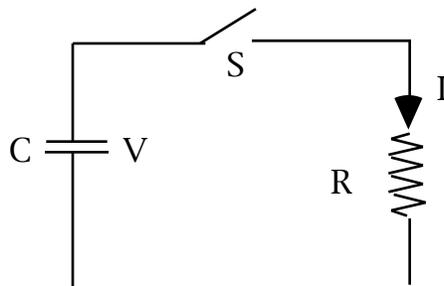


Figure 1

According to Kirchhoff's loop rule, the sum of the potential differences for a complete loop is zero:

$$V + I R = Q / C + I R = 0 \quad (2)$$

Since $I = dQ/dt$, we have a differential equation of the first order:

$$(RC) dQ + Q dt = 0 \quad (3)$$

with the initial condition that $Q_0 = C V_0$. The solution to Eq.(3) is

$$Q(t) = C V_0 \exp[- t / \tau] \quad (4)$$

and

$$V(t) = Q(t) / C = V_0 \exp[- t / \tau] \quad (5)$$

where $\tau = RC$ is called the time constant. It is the time for the voltage across the capacitor to reduce to $1/e$ (0.368) of its original value. After 3 time constants, only 5% of voltage is left. The voltage is practically zero after 10 time constants.

It can be shown in a like fashion that for a charging circuit of Fig. 2, the voltage across the capacitor is given by

$$V(t) = Q(t) / C = V_0 (1 - \exp[- t / \tau]) \quad (6)$$

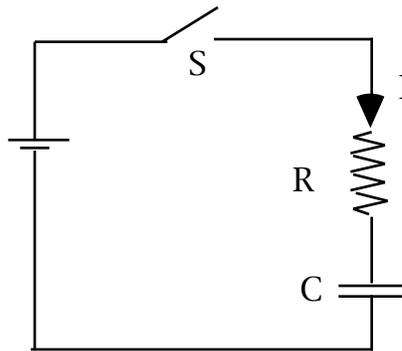


Figure 2

Fig. 3 shows the exponential curves of the discharging and the charging circuits.

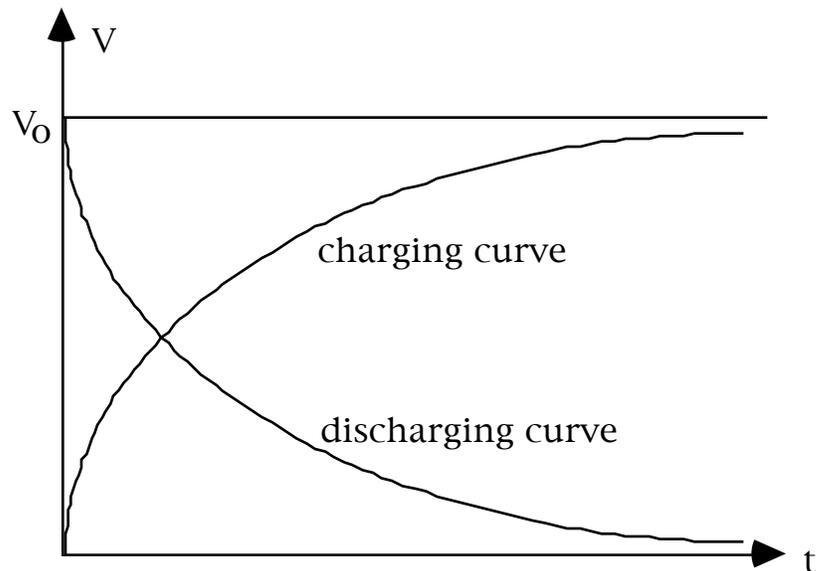


Figure 3

Procedures

1. Construct a circuit shown below.

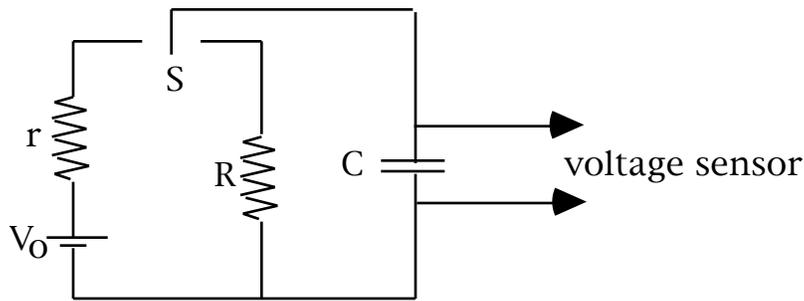


Figure 4

2. Set up the computer system and open the appropriate program to run "RC Circuits". Set the voltage axis to 5 V and the time axis to 60 seconds. You can always change the range such that the exponential decay curve fills up the whole screen.

3. Turn on the power supply and set at about 4 V. Switch to the power supply to charge the capacitor. Since the charging resistor r has small resistance (about a few $k\ \Omega$). It should take only a split of a second to charge up the capacitor to the full voltage of the power supply.

4. Then simultaneously switch the capacitor to the discharging resistor and click the "Start" or "collect" button to start collecting data. An exponential curve should show the remaining voltage across the capacitor. When the voltage is less than 10% or so, click "Stop" button.

5. Fit the curve to a natural exponential function of Equation (5). Find the decay constant from the results of the curve fitting. Calculate the resistance. The capacitance can be read from the capacitor.

6. Get a hard copy of the plot and the curve fitting data. Include it as a part of your data analysis.

7. The resistance obtained in step 5 is actually the combined resistance of the resistor R and the input impedance of the sampling circuit of the voltage sensor, which is usually quite high. You can measure the input impedance by measuring the RC constant. To do this, simply disconnect the resistor R and repeat steps 3 to 6. Since the R is not there, you need only to switch the capacitor to the power supply to charge up, then open the switch and click "Start" or "Collect".

Question

How do you estimate the error in resistance ?

Experiment 7. Oscilloscope

Objective:

To familiarize with the use of oscilloscope and the measurement of DC and AC voltages, amplitude, period and frequency. To learn the method of measuring high resistance through measurement of RC constant.

Apparatus:

Oscilloscope, probes, variable DC power supply, and function generator.

Theory:

The oscilloscope is a unique and probably the most important test instrument of an electrical engineer or any scientist working with electronic devices. The basic component of the oscilloscope is the cathode ray tube (CRT), as illustrated in Figure 1.

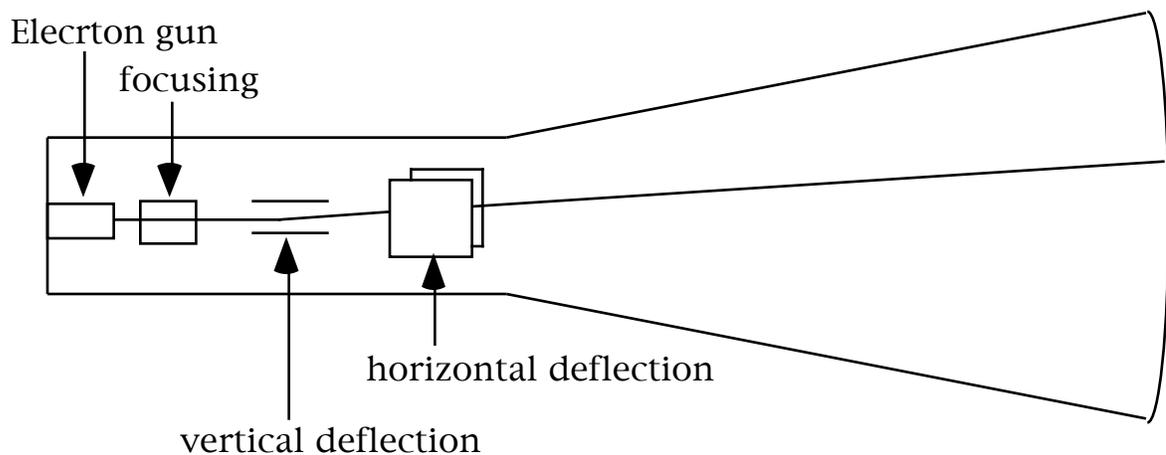


Figure 1. Construction of CRT

Electrons from a heated cathode or filament are accelerated through a potential difference into a focusing element to become a well collimated and focused beam. The electron beam enters into two sets of parallel deflection plates. The first pair of parallel plates is more sensitive and used for the signal channel. It causes vertical deflection. The second pair of plates causes horizontal deflection and is usually used for time sweeping. The electron beam coming out of the deflection plates heats the CRT screen with fluorescent coating and make a bright spot at certain position determined by the voltages applied across the two pairs of deflection plates. If both voltages are varying in time, the electron beam will trace a two dimensional curve on the screen.

Figure 2 shows one of such a two dimensional curve, a sinusoidal wave function. The horizontal movement of the electron beam is controlled by a voltage that increases as a linear function of time. This voltage is sometimes called the sawtooth voltage or ramp voltage. The curve traced by the electron beam on the screen is actually the waveform as a function of time of the signal applied across the vertical deflection plates. The time measurement (horizontal axis) is determined by the frequency of the sweeping voltage provided by a built-in sawtooth waveform generator, while the voltage measurement (vertical axis) is determined by the amplification of the internal amplifier. Both the sawtooth generator and the amplifiers are accurately calibrated by the

manufacturer so that the user can read the time and signal voltages directly from the knob settings on the front panel of the oscilloscope.

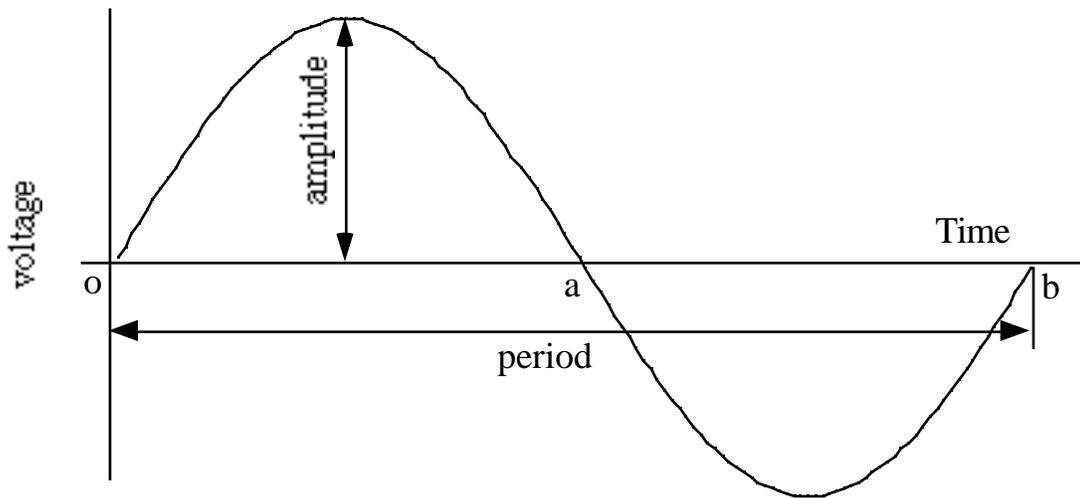


Figure 2. A sinusoidal waveform

In Figure 2, the time interval between points o and b is called the period because the waveform repeats itself again from the point b on. The maximum signal is called the amplitude of the waveform, which is measured from the time axis, which is supposedly at the vertical center of the waveform, to the peak of it. The period is usually very short (from a few nanoseconds to a few milliseconds for most measurements), and the whole curve of Figure 2 would cause a fluorescent flash for as long as the fluorescence lasts, which is about a second or so, too fast for any practical measurement. In order display a static waveform, the electronics is designed such that the electron beam keeps on repeating the same track on the screen. This requires that the electron beam always starts at the same point of the waveform. Namely, the horizontal sweep waveform and the signal waveform have to be synchronized. There are controls on the front panel for adjusting the trigger level and polarity to achieve stable synchronization. If the signals are not synchronized, the waveform will be seen running on the screen.

Procedure

1. Turn on the power switch. (It could be quite a challenge to find one for some models!) You may need to wait for seconds to see something shown on the screen. Find the "Intensity" knob and adjust the intensity such that the figure on the screen is bright enough for you to see, but not too bright to burn up the screen. The problem is more serious if you see only a single dot on screen. Reduce the intensity immediately when this happens. Get help from your instructor if you are completely lost.

2. Adjust the "Focusing" knob to obtain the finest trace on the screen. The better focused beam is usually brighter. You need to reduce the intensity accordingly to protect the screen.

3. Set the time scan to the scale of 10 ms per division. Set the "Input" switch of both channels to "Ground". Adjust the "X position" knob so that the horizontal sweep line is centered on the screen. This is the time axis.

4. Adjust "Y position" knobs to see which channel is used. If the knob controls the Y position of the sweep line, that channel is used. Make sure Channel One is used and adjust the Y position so that the sweep line is centered vertically.

Apply a few volts of DC voltage to Channel One input, using a BNC probe. Notice the change of the vertical position of the time axis. Set the Y sensitivity knob to the more sensitive scale, but not too sensitive to send the sweep line off screen, for more accurate measurement. Record the DC voltage as read from the screen. This is how you measure the voltage of a waveform or pulse.

Now switch the polarity of the input voltage. What happens to the sweep line?

Switch the "Input" from "DC" to "AC". What happens to the sweep line? Why?

5. Connect the output of the function generator to the input of channel One. Turn on the function generator. Set the waveform to sinusoidal wave and the frequency to kHz range. Try to obtain a few periods of a stable sine wave on the screen by adjusting the time base and the triggering level. Make sure the triggering source is set at Channel One. This is probably the most challenging part of the experiment. If you have great difficulties in obtaining a stable waveform after reasonable effort, get some help from your instructor.

6. Measure the period and the amplitude of the sine wave. Record the numbers.

7. Now change the sensitivity of the y axis and measure the amplitude again. Do you get the same voltage? Which scale setting gives you the best measurement ?

8. Change the time base to the adjacent settings. You may need to readjust the triggering to stabilize the waveform. What happens to the waveform ? Measure the period again. Do you get the same value for the period ?

9. Now switch the waveform from sinusoidal to square wave. There is a switch on the waveform generator for choosing the waveforms. Measure the pulse height, the period and the pulse width of the squarewave. Record the data.

10. Repeat step 9 with a sawtooth waveform.

Experiment 8. I-V Curves of Non linear Device

Objective:

To study the method of obtaining the characteristics of a non linear device, and to learn the rectifying feature of the diodes.

Apparatus:

diode, adjustable DC power supply, resistor (200 ohms or so), computer and interface, voltage and current sensors.

Theory:

Resistors and capacitors that we studied in the previous experiments are called linear devices because the voltage across such a device is linearly proportional to the current passing through it. However, many devices that play important roles in electronics are non linear devices. Indeed, physics would be dull and life most unfulfilling if all physical phenomena were linear. As a matter of fact, a linear device is linear only under certain conditions and within certain limits, as we saw in the experiment of temperature dependence of resistance. In this experiment we will study one of the simplest but not the least important non linear device -- a diode, and obtain the I-V curve of it.

A diode is made of two pieces of semiconductors, an n-type semiconductor and a p-type semiconductor, in direct contact. The n-type semiconductor is electron-rich while the p-type is hole-rich. The region in which the semiconductor change from p-type to n-type is called a junction. This junction creates a potential barrier that favors the current to run one way and impede it to run the opposite way. When the positive terminal of a battery is connected to the p side of the junction, the current can easily pass through and the diode is said to be forward biased. If the positive terminal of the battery is connected to the n side, however, it is very difficult for the current to pass and the diode is said to be reverse biased. The direction of bias is usually marked on the casing of the device or by the asymmetrical shape of the casing itself.

Figure 1 shows the characteristics of a typical diode. It is a plot of the current versus the voltage. When the voltage is forward biased, the voltage across the diode is very small, about 0.5 volts for the germanium diode and about 0.7 volts for the silicon diodes. (The voltage drop across a diode is more important than its resistance in most applications.) The working current of a diode changes dramatically, ranging from less than one mA to more than a kA, depending on its application. When the diode is reverse biased, the current is very small, usually less than 100 μ A for germanium diodes and less than 10 μ A for silicon diodes, as long as the voltage is less than the avalanche breakdown voltage. If the reverse voltage exceeds the avalanche voltage, the current would increase dramatically like the snow current running down hill in an avalanche, and it usually kills the diode. It is therefore to stay safely below the avalanche voltage in your applications. The breakdown voltage is listed in the semiconductor catalogue books and should be on the data sheets attached to the purchased diode.

There is, however, a special type of diodes known as Zener diodes capable of recovering from the avalanche if the current is kept too great. It is used to provide voltage references because the avalanche voltage across a Zener diode is relatively stable within a great range of current. The maximum allowed operating current of a Zener diode is specified in the catalogue books or the data sheets.

The most important application for the regular diodes is rectification. A rectifier is a device that turns an AC voltage into a DC voltage, which we will study in the next experiment.

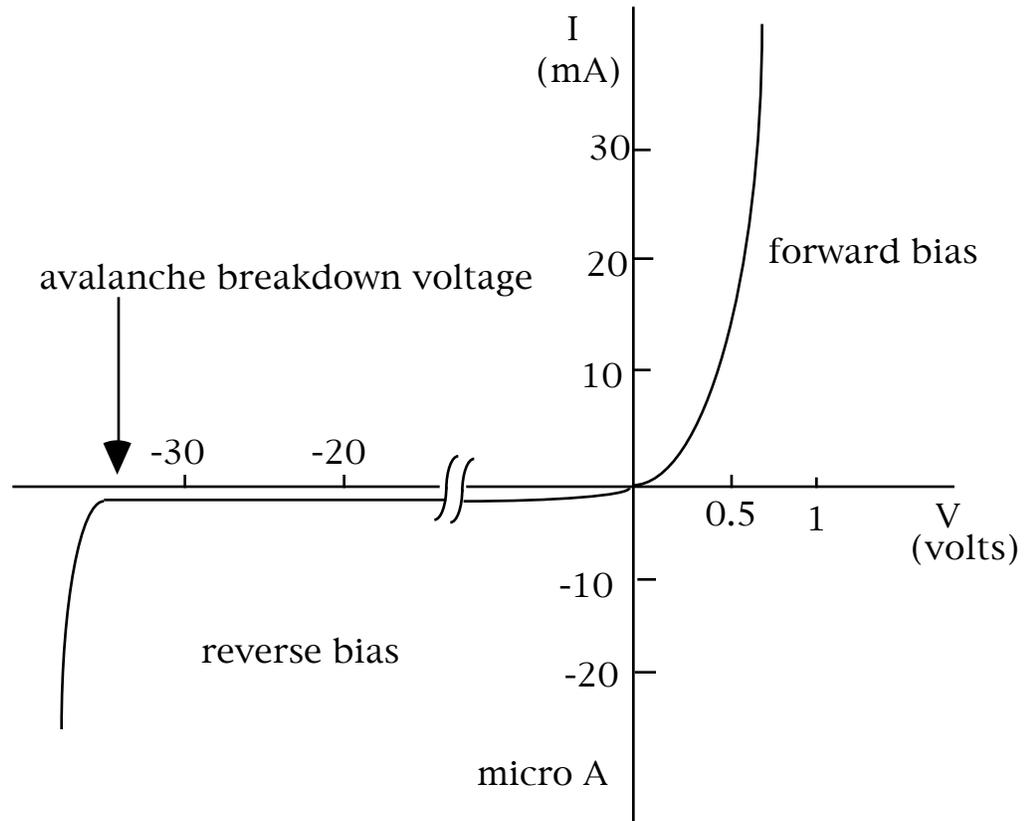


Figure 1

Procedures

1. Construct a circuit shown in Figure 2 and set up the computer system for the I-V Curve experiment.

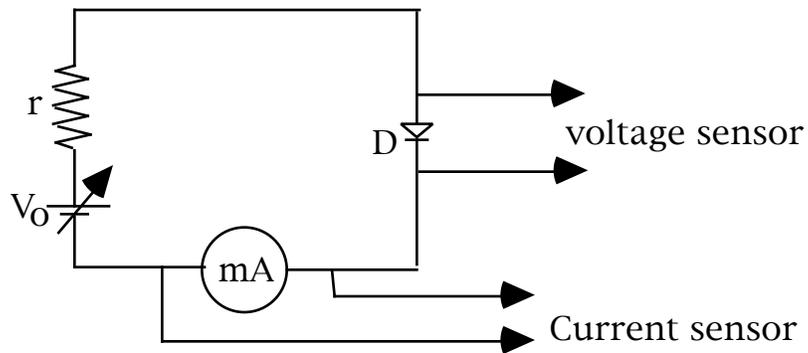


Figure 2

2. Turn on the power supply and give a voltage of about one volt. If the current is practically zero, then reverse the diode to forward bias it. The forward current should be a few mA depending on the resistance of the resistor.

3. Whence the diode is forward biased, then start from zero voltage and gradually increase to 5 Volts. The data should be automatically collected and a curve shown on the screen.

4. Try to fit the curve into an exponential function and get the parameters. Include not only the straight part, but almost the whole length of the curve in your curve fitting.

5. Now reverse the diode so that it is reverse biased. Repeat steps 3 and 4. Remember to keep the voltage below 5 V. The current should be practically zero since the current sensor is not sensitive enough to detect a current less than 1 mA. Get a hard copy of the plot, but you do not have to do curve fitting for the reverse biased data.

6. Make a plot manually to combine the forward biased and the reverse biased data and make a single plot. This is characteristics curve of the diode.

Questions

1. How well does the data fit to the exponential function ?
2. Can you come up with an idea to make use of the characteristics of the diode?

Experiment 9. Diode Power Supply

Objective:

To study the two basic components of a simple power supply: the rectifier and the filter; and to learn how to measure and calculate the ripple factor -- one of the major parameters to measure the quality of a DC power supply.

Apparatus:

diode, AC power supply, filter resistor (200 Ω), load resistor (10 k Ω), capacitors (1 μ F and 22 μ F), oscilloscope.

Theory:

We have studied the characteristics of a diode in the last experiment, and understood that the non linear characteristics of a diode can be used to rectify an AC voltage into a DC voltage, namely, to allow the current flowing one way only. Such a rectifier can be used to obtain a low frequency signal from its high frequency carrier, or simply used to turn an AC power supply into a DC power supply. Such device is called an AC-DC converter or adapter.

As shown in Figure 1, a sinusoidal AC voltage is applied across the points a and c. A diode D is inserted between the AC power supply and the load resistor R. Because the diode allows the current to run from the point a to the point b only, there is no current when the potential at the point a is negative with respect to the point c. The negative current and voltage is simply cut out of the load resistor R. The waveform of such a rectified voltage is called a half-wave rectified voltage, which is the half of a sinusoidal waveform above the time axis.

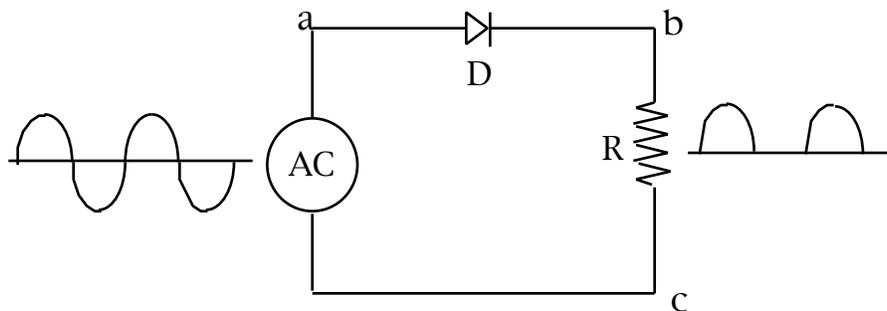


Figure 1

Figure 2 shows a half-wave rectified voltage. As it can be seen, the rectified half-wave form does not look like the DC voltage of a battery at all. The half wave rectified voltage can be called a "DC" voltage all right, because it stays positive all the time and gives a current that flows one way only. However, it is not constant and has an AC component that varies between the sinusoidal peak and zero, as indicated as V_{pp} in Fig. 2. We can define one half of this amount as the amplitude of the residual AC component in the rectified voltage:

$$V_{ac} = 0.5 V_{pp} \quad (1)$$

The DC component V_{dc} of the rectified voltage is indicated in Fig. 2 by a line above the time axis, which represents the average value of the rectified voltage over the whole period. If a battery having a voltage equal to this average voltage, V_{dc} is used to drive a DC motor, for instance, it should

provide an equivalent power as the rectified waveform does. Experimentally, this DC component can be easily measured by switching the signal input between DC coupling and AC coupling on the oscilloscope panel. When the switch is on the AC coupling, the waveform would move down vertically on the screen exactly by the amount V_{dc} .

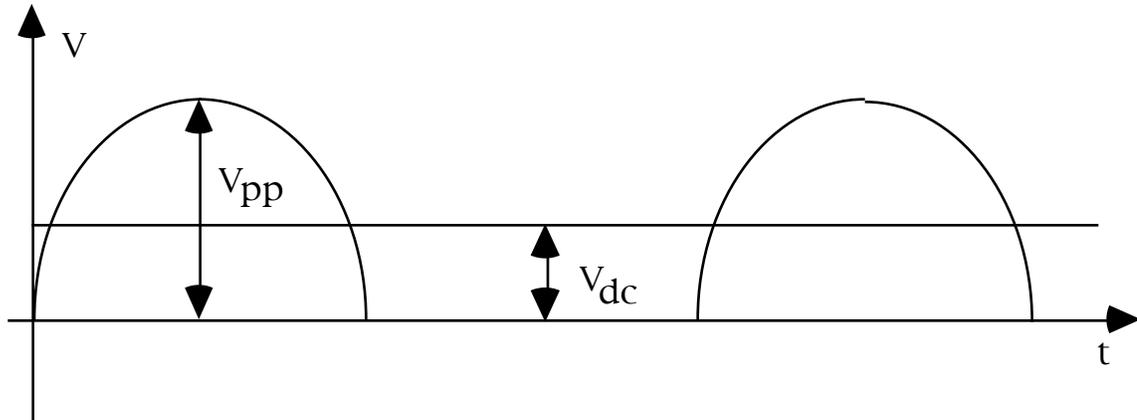


Figure 2

The waveform of Fig.(2) can not be used as a good DC power supply in most applications due to its significant AC component. To make the DC voltage more like the constant voltage of a battery, it is necessary to filter out the AC component. The simplest filter is a capacitor. As shown in Figure 3, a capacitor C is connected in parallel with the load resistor to absorb the fluctuation of the voltage. When the point a is at high voltage, not only the load resistor draws current, the capacitor draws current as well. The capacitor is charged to the peak voltage of the sinusoidal waveform. When the voltage at the point a start to decrease, the capacitor start to discharge through the load resistor. (Why not through the diode?) Namely, the capacitor functions temporarily as a "battery" until the sinusoidal AC voltage becomes higher than the voltage across the capacitor again. The charging and discharging processes repeat themselves every period so that the voltage across the load resistor is kept between certain minimum voltage V_{min} and the maximum voltage V_{max} . (See the waveform in Fig.3)The fluctuation of the voltage is called the ripple voltage, and it is defined as

$$V_{rip} = 0.5 (V_{max} - V_{min}) \quad (2)$$

The smaller the ripple voltage, the better the DC power supply. We can therefore define a parameter called the "ripple factor" as one of the criteria of the quality of the power supply:

$$\text{Ripple Factor} = (V_{rip} / V_{dc}) \times 100\% \quad (3)$$

Apparently, the larger the capacitance, the small the ripple factor and the better the power supply. In order to keep the voltage almost constant, the product of the capacitance and the load resistance needs to be much greater than the period of the AC voltage. (Why?)

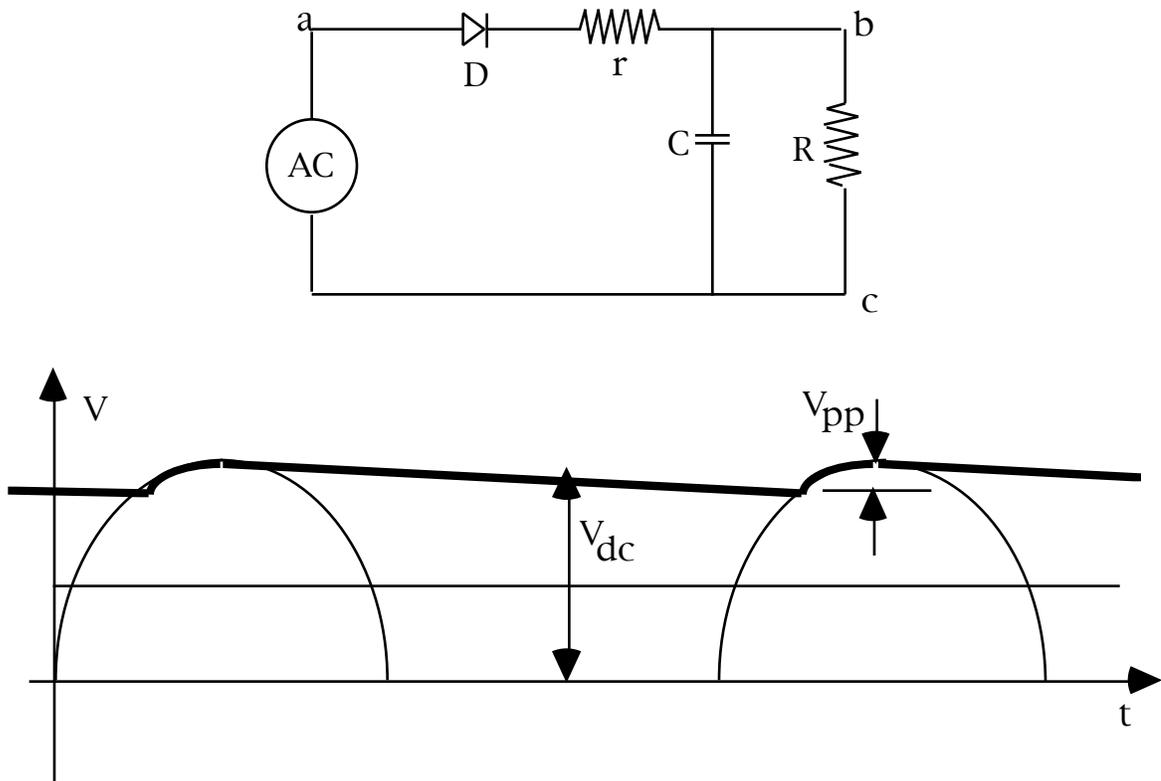


Figure 3

Because the capacitor can draw a current much greater than that passing through the load resistor, a protective resistor r is usually connected in series with the diode to protect the device. This resistor can also improve the filtering function. The disadvantage is that it reduces the maximum voltage and consumes power.

Procedure

1. Construct a circuit as shown in Fig.(1). Observe the AC waveform before the diode and the rectified waveform across the load resistor. Include a sketch of the rectified waveform in your report. Measure the DC and AC components of the rectified voltage. Calculate the ripple factor.
2. Now insert the RC filter in the circuit as shown in Fig.(3), using the capacitor of $1\ \mu\text{F}$. Observe the waveform across the load resistor again. Note that waveform would look like the bold curve shown in Fig.(3). The sinusoidal shape is no longer observable. Measure the DC and AC component of the voltage with the capacitor filter. Now the DC component is relatively easy to measure. But AC voltage is too small to measure directly on the same screen with the DC voltage. To get more accurate measurement of the AC component, switch the input coupling to "AC" to show the AC component only, and use the Y-offset adjustment to center the waveform vertically. Now you can use more sensitive input scale to enlarge the waveform vertically for more accurate measurement of the AC voltage. Calculate the ripple factor for the circuit with filter.
3. Replace the $1\ \mu\text{F}$ capacitor with the $22\ \mu\text{F}$ capacitor. Repeat Step 2 and find the ripple factor for the filter with the larger capacitor.

Questions

1. Can you design a circuit of full-wave rectifier ?
2. Why is the time constant of the filter equal to RC instead of rC ? (R is the load and r is the charging resistor before the capacitor.)

Experiment 10. Magnetic Fields and Induction

Objective:

To study the special dependence of the magnetic field of a solenoid.

Apparatus:

Solenoid, magnetic field sensor, computer and interface, supporting posts and meter stick, power supply and resistor.

Theory:

When an electric charge is moving, it generates a magnetic field in the space. Since a current is a continuous flow of moving charges, it will generate a magnetic field around it. The magnetic field of a current element is given by Biot-Savart Law:

$$d\mathbf{B} = \mu_0 / (4\pi r^3) I d\mathbf{L} \times \mathbf{r} \quad (1)$$

Where I is the current; \mathbf{B} , \mathbf{L} and \mathbf{r} are, respectively, the magnetic field vector, the length vector of the current element and the displacement vector from the current element to the point of interest. The bold faced letters represent vectors. The constant $\mu_0 = 4\pi \times 10^{-7} \text{ N / A}^2$.

Experimentally, we can measure the magnetic field of a macroscopic current with finite size, instead of an infinitesimal current element. Eq.(1) is therefore of theoretical significance only. To obtain the magnetic field of a current of finite size, it is necessary to integrate Eq.(1) over the total length of the current, which is somewhat a tedious but straightforward process. Inside a solenoid, the magnetic field along the axis is given by

$$B = 0.5 \mu_0 n I [b (b^2 + R^2)^{1/2} + a (a^2 + R^2)^{1/2}] \quad (2)$$

where R is the radius of the solenoid, n is the number of turns per unit length, a and b are the distances from the point of interest, which is the origin of the coordinates in Figure (1), to the ends of the solenoid. The direction of the magnetic field is along the X-axis.

The magnetic field of a solenoid is shown in Figure (1). Inside the solenoid, the magnetic field is nearly a constant. It starts to decrease dramatically towards the ends of the solenoid. It is sometimes called the "edge effect". The magnetic field quickly reduces to practically zero outside the solenoid. In the central part of the solenoid, if the solenoid is very long, namely, $a \gg R$ and $b \gg R$, Eq.(2) can be approximated by a much simpler formula:

$$B = \mu_0 n I \quad (3)$$

At the ends of the solenoid, when either a or b is zero,

$$B = 0.5 \mu_0 n I \quad (4)$$

The direction of the magnetic field along the axis of the solenoid is always parallel to the axis. Elsewhere, however, the direction of the magnetic field changes depending on the position. The magnetic field vector generally has three components, and it usually is obtained numerically with computer if the geometrical parameters of the solenoid are given. For a picture of the magnetic field lines, refer to your text book of general physics.

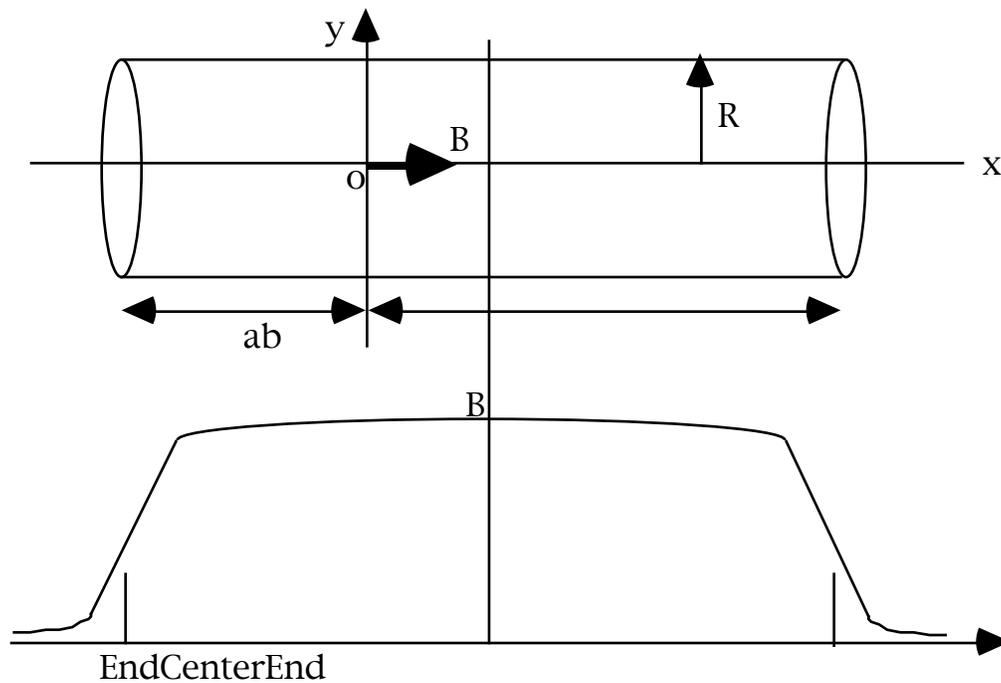


Figure 1. The magnetic field of a solenoid

Procedure

1. Set up the solenoid and the Gauss meter sensor as follows:

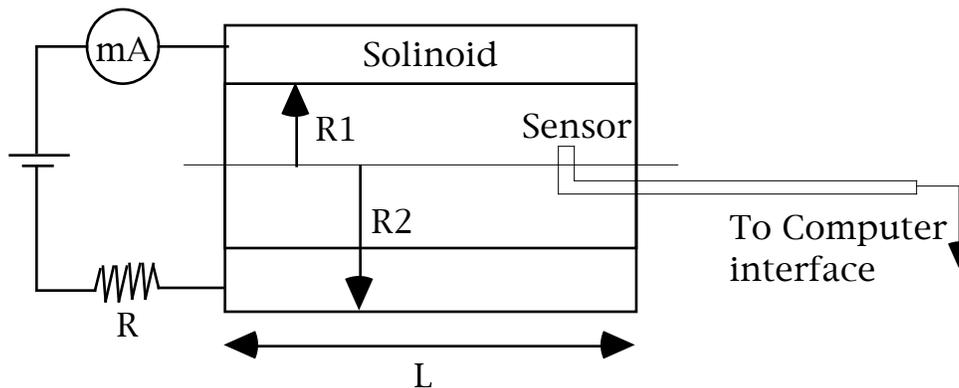


Figure 2. The experimental Set-up

Measure the inner and outer radii R_1 and R_2 . Enter the average radius into Eq.(2) for your theoretical calculations. Measure the length L of the solenoid. Record the total number of turns, N , which is written on the solenoid. Divide to total number of turns N by the total length of the solenoid to obtain n

$$n = N / L \tag{5}$$

2. Connect the solenoid to a DC power supply of 15 V. Divide this voltage by the total resistance of the solenoid and the resistor to obtain the current I through the solenoid. The resistance of the solenoid is also labeled on it.

Now you have all the information for your theoretical calculations.

3. Set up your semiconductor field sensor such that the tip, not the stem, of the sensor is centered at the axis of the solenoid. The stem of the sensor is but a glass sleeve of the electrical wire and therefore has no significance. The output of the sensor goes to the computer interface.

4. Turn on the computer and the interface, and open the software for data taking.

5. Measure the magnetic field along the axis of the solenoid. You need to measure the position of the sensor and manually enter into the computer to keep the data point. Starting from the center of the solenoid and map out the magnetic field along the axis.

6. Plot the experimentally measured magnetic field along the axis of the solenoid. Compare the curve with the theoretical curve as calculated by Eq. (2). How good is the agreement between theory and experiment ?

Question

What are the sources for discrepancies between theoretical and experimental results?

Experiment 11. Reactance

Objective:

To study Ohm's law of AC circuits, and the frequency dependence of reactance.

Apparatus:

Solenoid, capacitor, resistor, oscilloscope, signal generator.

Theory:

Ohm's law plays a very important role in the theory of DC circuits. Does it work in AC circuits? Yes, it does, if the resistance R is replaced by the impedance, which is determined by both the resistance and the reactance of the inductors and the capacitors. What are the impedance and the reactance? We will study these concepts in the following two experiments.

The reason that Ohm's law takes more complicated form in AC circuits is that the current passing through a capacitor or inductor is not in phase with the AC voltage applied across them. Physically, it reflects the fact that capacitors and inductors are devices that store energy, not consuming it. Such storage function makes the current advance (in the case of capacitors) or lag (in the case of inductors) the voltage by one quarter of a period. To see this, let us consider the inductor circuit of Figure 1.

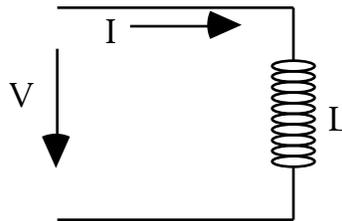


Figure 1. The voltage and current across an inductor

We assume the current passing through the inductor L to be

$$I = I_0 \sin(\omega t) \quad (1)$$

where $\omega = 2\pi f$, f being the frequency. According to Faraday's law, the voltage V across the inductor is given by

$$V_L = L (dI/dt) = I_0 \omega L \cos(\omega t) = V_{L0} \sin(\omega t + 90^\circ) \quad (2)$$

With $V_{L0} = I_0 \omega L = I_0 X_L$ (3)

where $X_L = \omega L$ (4)

X_L is called the inductive reactance. Similarly, for a capacitor we have

$$V_C = V_{C0} \sin(\omega t - 90^\circ) \quad (5)$$

With $V_{Co} = I_o / (\omega C) = I_o X_C$ (6)

where $X_C = 1/(\omega C)$ (7)

X_C is called the capacitive reactance. Eqs.(3) and (6) are the AC forms of the Ohm's law, because they show the linear relationship between the amplitudes (or rms values) of the voltage and the current. Eqs.(2) and (5) show that the voltage across an inductor advances the current by a phase angle of 90° , while the voltage across a capacitor lags the current by 90° .

In this experiment we will test the linear relationship of Eqs. (3) and (6), and the frequency dependence of the reactance, Eqs. (4) and (7).

Procedures

1. Assemble the circuit shown in Fig. (2).

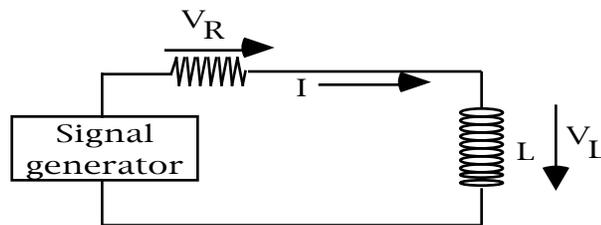


Figure 2

2. Record the values of L and R .
3. Turn on the signal generator and choose sinewave. Set the frequency at about a few tens of kHz. Measure the period T of the sinusoidal wave with the oscilloscope and calculate the frequency f . Keep this frequency unchanged through step 8. Use **Data Table 1** to record your data.
4. Set the amplitude of the output of the signal generator to maximum, and measure the amplitude of the voltage across the inductor, V_L . Record the value.
5. Measure the amplitude of the voltage across the resistor, V_R . Divide V_R by the resistance R to obtain the current I passing through the circuit. Record both V_R and I .
6. Reduce the amplitude by about 10% and repeat the steps 4 and 5 to get another set of data.
7. Repeat step 6 until the voltage becomes too low to obtain reasonably good data. You should be able to get more than 6 data points.
8. Plot the voltage across the inductor, V_L , versus the current I . Find the slope X_L . Calculate the inductance using the measured frequency measured in step 3. Compare the measured value of the inductance with the nominal value as read from the inductor. Calculate the percentage error.
9. Now you are going to test Eq.(4), the dependence of the inductive reactance as a function of frequency. Use **Data Table 2** to record your data. The frequency is now a variable and must be changed in your data taking. Use the maximum amplitude all the time. For a certain frequency, measure the current I and the voltage across the inductor as described in steps 4 and 5, and measure the frequency ($1/T$) using the oscilloscope. Calculate the inductance X_L ; at that frequency. This gives one data point. Then change the frequency and repeat the measurement to get another data point. You should get about ten data points in the frequency range between a few hundred Hz to a few hundred kHz.
10. Plot the inductance X_L as a function of frequency f . What is the slope? Theoretically, it is supposed to be 2π . Calculate the percentage error in the measured slope as compared to the theoretical value 2π .

11. Now replace the inductor with a capacitor and repeat the above steps. You will need to replace the equations with the ones appropriate to a capacitor, of course. To get a linear plot of the capacitive inductance, plot X_C as a function of the period T instead of the frequency f . The theoretical slope should be $1/(2\pi C)$.

Data Table 1

R = _____ L = _____
 T = _____ f = _____

Trial	V_R	I	V_L
1			
2			
.			
.			

Data Table 2

Trial	T	f	V_R	I	V_L	X_L
1						
2						
.						
.						

Experiment 12. Electric Resonance

Objective:

To study electric resonance and learn how to obtain the band width and the quality factor.

Apparatus:

Solenoid, capacitor, resistor, oscilloscope, signal generator.

Theory:

For a circuit containing a resistor, a capacitor and an inductor connected in series, as shown in Fig.1, the current I is related to the voltage V of the AC power supply by

$$I/V = 1/Z \quad (1)$$

with

$$Z = \frac{1}{\sqrt{R^2 + (X_L - X_C)^2}} \quad (2)$$

where R , X_L and X_C are the resistance, the inductive and the capacitive reactances respectively. Z is called the impedance of the circuit.

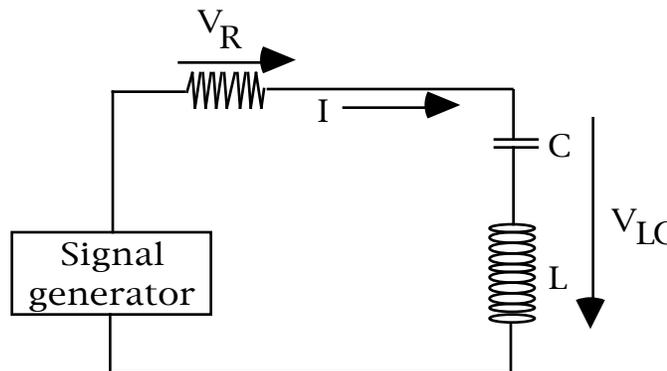


Figure 1

Since the impedance contains the reactances, it is a function of the frequency. If we plot I/V versus the frequency f , we get a Lorentzian curve depicted in Fig.(2). The curve approaches zero when the frequency approaches either infinity (when X_L is infinity) or zero (when X_C is infinity). The impedance at the two extremes is infinity which results in zero current. The current is non zero at any other finite frequency, with a peak at certain frequency f_0 called the resonance frequency.

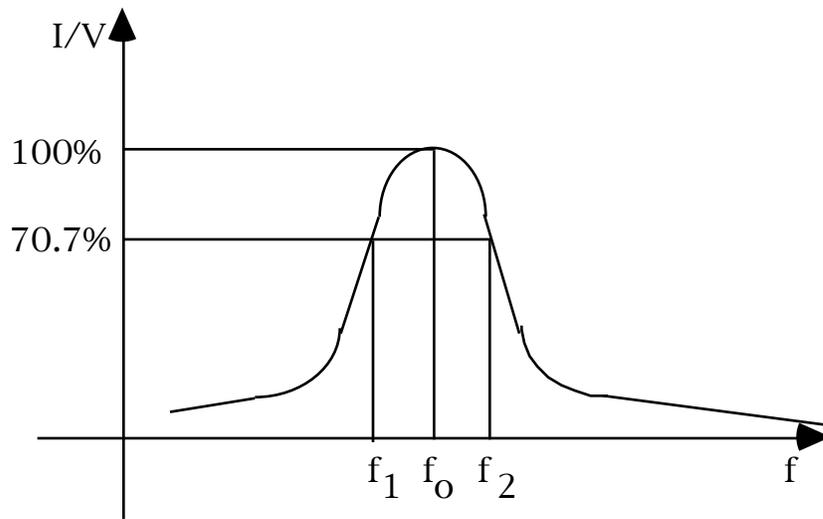


Figure 2. The resonance curve

The resonance frequency f_0 can be obtained by inspecting Eq.(2). Since the resonance takes place when the impedance Z is the minimum, we let the total reactance equal to zero:

$$X_L - X_C = \omega L - 1 / (\omega C) = 0$$

$$\omega = \frac{1}{\sqrt{LC}} \quad (3)$$

$$f_0 = \frac{1}{2\pi \sqrt{LC}} \quad (4)$$

In telecommunication, it is also desirable to know how many different frequency channels can be placed within certain band without interference, i.e., how "wide" are the resonance peaks. Since the resonance peak covers theoretically the whole frequency spectrum and no clear cut board lines can be defined, the width of the resonance peak is defined to be the difference between two frequencies f_1 and f_2 when the amplitude of the current (or voltage) falls to 70.7% of its maximum value. This corresponds to a reduction of power by a factor of two.

$$\Delta f = f_2 - f_1 \quad (5)$$

The quality factor Q is defined as the ratio of the resonance frequency over the width:

$$Q = f_0 / \Delta f \quad (6)$$

The quality factor Q is a measure of the tuning quality of a resonance circuit. The higher the quality factor, the smaller the band width Δf , and therefore the less interference between the adjacent channels.

Procedure:

1. Construct a circuit shown in Fig. (1). Measure and record the resistance R . You will need the resistance to calculate the current.
2. Read the capacitance C and the inductance L from the devices. Record these parameters. Calculate the theoretical resonance frequency f_0 using Eq. (4). This expected resonance frequency may not necessarily exactly equal to the experimental resonance frequency due to the experimental errors. But the expected value does give your a guidance as to what frequency range you should be working in to get the entire resonance curve.
3. With the guidance from step 3, set the signal generator in the range covering the expected resonance frequency. measure the total voltage across the capacitor and the inductor, V_{LC} , with the oscilloscope. The amplitude of this voltage should change while you are changing the frequency of the generator. Change the frequency slowly while watching the change of the amplitude of V_{LC} until you find the frequency at which the voltage V_{LC} is the minimum. This frequency is the experimental value of the resonance frequency. But do not trust the frequency reading of the front panel of the signal generator. Use the oscilloscope to measure the period of the waveform and calculator the frequency which is the reciprocal of the period. The oscilloscope measurement is more reliable than the frequency value read from the signal generator.
4. Having obtained the experimental resonance frequency, you can then map out the resonance curve. You need to obtain about half dozen points of each side of the resonance peak. For each frequency point, you need to measure both the resonance voltage V_{LC} and the current I . The current I can be obtained by measuring the voltage across the resistor, again using the oscilloscope, divided by the resistance measured at the beginning of the experiment.
5. Plot the ratio (I/V_{LC}) versus the frequency f . You should get a resonance curve as shown in Fig.(2). Use the method shown in Fig.(2) to obtain the band width Δf and the quality factor Q .