

**ELECTRIC CIRCUITS**  
**LABORATORY MANUAL**  
**(ECE-235 LAB)**

# GUIDE LINES FOR THE EXPERIMENTS AND REPORT PREPARATION

## 1. Preparation for the experiment:

Before conducting the experiment, the student is required to have read the experiment background and procedure from the experiment manual and studied the related theory. The lab instructor may, during the experiment, ask students questions pertaining to the procedure and theory. The lab instructor may give negative points to and even prevent an unprepared student from conducting the experiment. Tardy students may not be allowed to perform the experiment.

## 2. Laboratory teams:

The class will be divided in teams of three or four students. The composition of the teams (which students will team up) is left to the preference of the students, but the lab instructor makes the ultimate decision as to each team's composition.

Each lab experiment requires a report. The lab reports are due on the next lab meeting. The lab report for the final experiment is due a week after the final lab meeting.

Each team submits one report per experiment (unless otherwise required). The grade of the report is given to all members of the team. Late reports are penalized by taking 5 points off per each day past the due date of the report. The other grade components of the experiments are given to the students individually. If a student misses or is dismissed from an experiment, the grade of that student shall be zero for that experiment.

## 3. Preparation of the report:

The report must be produced electronically (e.g. using MS-WORD). Tables showing data or results, as well as figures and graphs should be produced electronically and embedded in the main body (e.g. using MATLAB or MS-Excel, MS-Visio). Include captions and titles for figures, graphs and tables as well as numbers for equations. The preferred language style is the use of the present tense and third person. The report must contain the following sections:

a) **Cover page:** Include number and title of the experiment, date the experiment was performed and the names of the team members.

b) **Objective:** Give a short description of the purpose of the experiment.

c) **Theoretical background:** Give a brief description of the relevant theory.

d) **The experimental procedure:** Summarize what was done for each experiment procedure. Do not copy or repeat the procedure description from the lab manual. Report the measurement and other experimental data. Tabulate measurements if necessary. Include table number and title over tables.

(e) **Analysis of experimental data:** Analyze the data. Compare with theoretical results. Produce graphs using MATLAB or MS-Excel and embed the graph figure into the main body of the

report. Include figure number and caption. Label axis. Show units. Tables and graphs should appear inserted in the text close to the place they are first mentioned and in the same section. Add remarks and calculations on each procedure if necessary.

f) **Conclusions:** Summarize the experiment and the results. Discuss the factual knowledge gained.

# **INTRODUCTION TO ELECTRIC CIRCUITS LAB (ECE-235 LAB)**

## **Objectives:**

- 1- To introduce the students to the basic electrical equipments in the lab.
- 2- To be able to deal with some of the frequently used instruments and equipment; like the digital multimeter and DC Power supply.

## **Introduction: DC Power Supply**

The DC power supply is used to generate either a constant voltage (CV) or a constant current (CC). That is, it may be used as either a DC voltage source or a DC current source. You will be using it primarily as a voltage source. Recall that DC is an acronym for direct current.

The voltage produced by the power supply is controlled by the knob labeled **voltage**. The current is limited by adjusting the knob labeled **current**. As long as the circuit does not attempt to draw more current than the value set by the current knob, the voltage will remain constant. This is often a difficult concept for students to grasp. Current limiting allows the power supply to be set such that it will not generate more current than it is safe. This can be useful as a safety feature, preventing electrocution due to accidental contact with terminals. In addition, current limiting can prevent damage to equipment and parts which may be unable to handle excessive currents.

## **Procedures:**

### **Part1: Voltage Measurement.**

- 1- Turn on the DC Power supply.
- 2- Make sure that the current knob is little bit above the minimum value.
- 3- Adjust the voltage knob at 6.5 V.
- 4- Measure the voltage value using the digital multimeter, and write down the measured value.
- 5- Find the percentage of error for the measured values.
- 6- State the reasons of the error.

### **Part 2: Resistance Measurement**

**Background on Graphite Resistance:** The standard resistor color code is shown on the back of the door to the lab. Here is a quick synopsis: Most resistors have four colored bands. The first

three bands indicate the nominal value of the resistor and the fourth band indicates the tolerance in value.

<b>color</b>	<b>value</b>
<b>Black</b>	<b>0</b>
<b>Brown</b>	<b>1</b>
<b>Red</b>	<b>2</b>
<b>Orange</b>	<b>3</b>
<b>Yellow</b>	<b>4</b>
<b>Green</b>	<b>5</b>
<b>Blue</b>	<b>6</b>
<b>Violet</b>	<b>7</b>
<b>Grey</b>	<b>8</b>
<b>White</b>	<b>9</b>

**Figure 1: the value of the colors**

The tolerance band is typically either gold or silver. A gold tolerance band indicates that the measured value will be within 5% of the nominal value. A silver band indicates 10% tolerance. For example a resistor with color code brown-black-red-silver indicates a nominal value of 1 k $\Omega$ . The first two bands (brown-black) produce the mantissa (10) and the third band (red) is the exponent of ten ( $10^2 = 100$ ). So the value is  $10 \times 100 = 1 \text{ k}\Omega$ . Since the tolerance band is silver, we can expect the measured value of the resistor to be between 900  $\Omega$  and 1100  $\Omega$ .

### Procedures:

#### PART A: Resistance measurement.

- 1- Pick up 2.2K $\Omega$  and 75 $\Omega$ .
- 2- Read the value of each resistance using the color method; show your steps in details.
- 3- Measure the value of each resistance using the digital multimeter.
- 4- Find the percentage of error for each resistance alone.

#### PART B: Measuring the Resistance of Your Body

1. Holding one probe between the thumb and forefinger of each hand, measure the resistance of your body between your hands. Squeeze the probes tightly so that good contact is established. Record the value of your body's resistance.
2. Considering that a current of 100-200 mA through your heart will almost certainly kill you, how much voltage across your hands would be lethal?



# EXPERIMENT 1:

## ELECTRICAL MEASUREMENTS

This experiment demonstrates the measurements of voltage, current, and resistance. The ohmmeter, the RLC-bridge, and two arrangements involving the voltmeter and the ammeter are presented for the measurement of ohmic resistance.

### I. BACKGROUND

#### *1.1 Types of Electrical Measurements.*

Measurements performed on an electric circuit include the circuit current, voltage, power, and resistance. The measurement of the current and voltage are basic as other quantities can be obtained such as power and resistance-power can be measured from the product of voltage and current and resistance can be measured from the voltage to current division (Ohm's Law).

Electrical measurements are classified into two major types, each using and requiring different instrumentation:

(a) DC measurements indicate the average value of a time-varying quantity. DC instruments are used only in circuits where the current is unipolar (dc), thus it has a non-zero average value. Figures A1 and A2 show two time-varying quantities that have non-zero average value. The dotted line in each case shows the indication of a dc instrument measuring the quantity. Equation (1) is the formula used to calculate the average (also called dc) value of a periodic wave form. Note that, if the quantity is constant with time, its instantaneous value is also its average (dc) value. That is the case, for example, when a circuit is supplied by a battery.

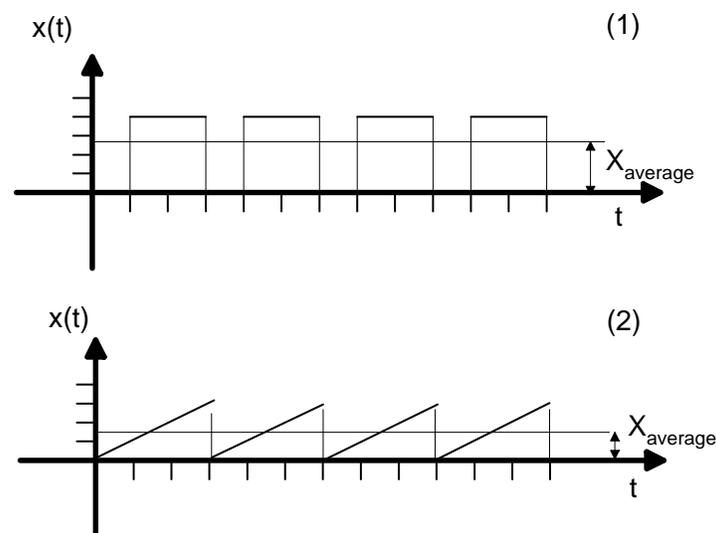


Fig. A Periodic wave forms and their average value (dc value).

$$X_{average} = \frac{1}{T} \int_0^T x(t) \cdot dt \quad (1)$$

Where T is the period of the wave form of the measured quantity.

(b) AC measurements indicate the rms (root mean square) value of a time-varying (usually periodic) quantity. Circuits that operate with ac current can only be measured by ac instruments. A dc instrument used in an ac circuit will indicate zero (why?). More about ac measurements will be presented in Experiment 7.

### 1.2 Current Sensing in DC Measurements.

The measurement of both the current and voltage requires sensing (measurement) of current. Many analog instruments sense current employing the d'Arsonval meter. Figure B shows a diagram of this meter.

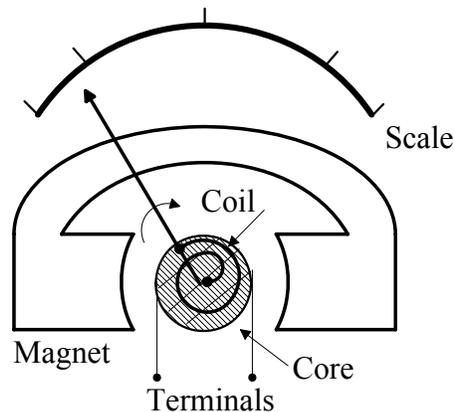


Fig. B. Configuration of the d'Arsonval instrument for the sensing of current.

The core is an electromagnet surrounded by a permanent magnet. The current that flows in the terminals of the electromagnet coil generates a torque on the core which is directly proportional to the current. This forces the needle to move. The motion of the needle is restrained by a mechanical coil (spring). The torque of the spring is directly proportional to the deflection of the needle. Therefore, the deflection of the needle is directly proportional to the current at the terminals of the instrument.

Current sensing instruments are rated at a maximum current and a maximum voltage. Thus, the instrument can safely operate in measurements that do not exceed its ratings. The ratio of the rated voltage to the rated current of the current sensing instrument is its internal resistance-this appears in series with its terminals.

### 1.3 The Ammeter.

A dc ammeter can be created employing the d'Arsonval current sensor, as shown in Figure C. The ammeter must be inserted in series with the current it measures. An internal arrangement of resistors is used to divide the current so that the sensor sees only a fraction of the circuit current. With reference to Figure C, this arrangement consists of several scaling resistors and a selection

switch. By selecting one of the scaling resistors, the portion of the measured current "seen" by the sensor varies.

The scaling resistors determine the range of the current the ammeter can measure. The resistance of a scaling resistor is chosen so that the current of the sensor is the rated (maximum) value when the circuit current is at the upper limit of the range. The different ranges are indicated on the scale of the instrument.

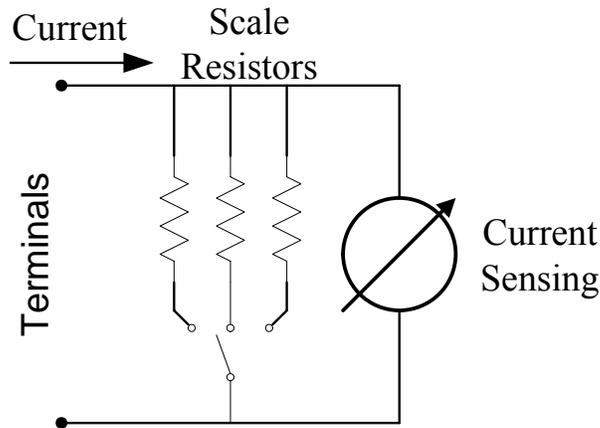


Fig. C. An ammeter employing current sensing and scaling resistors.

#### 1.4 The Voltmeter.

The voltmeter is connected in parallel to the measured voltage. Thus, it should insert a large resistance so that the circuit is not disturbed. Figure D shows a voltmeter created using a current sensor, series resistors and a selection switch. The current sensed by the sensor is proportional to the measured voltage. The resistance of the scaling resistors determines the range of voltage measurement. Their value is chosen to limit the current into the sensor.

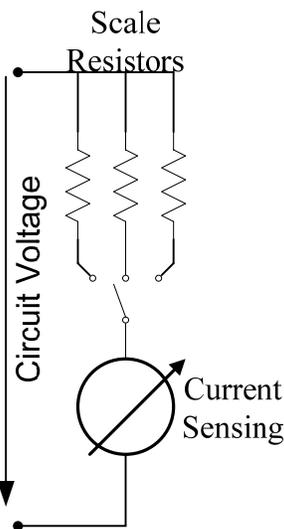


Fig. D. A voltmeter employing a current sensor and series resistors.

### 1.5 Measurement of Resistance.

There are two methods to measure resistance: (a) Directly employing the Wheatstone bridge. (b) Indirectly, by measuring current and voltage. The first method will be discussed in Experiment 2. The indirect method uses an ammeter and a voltmeter arranged in two possible configurations shown in Figures E and F. An external source drives a constant current into the resistor. With reference to Figures E and F, the ammeter is in series with the measured resistor and the voltmeter in parallel.

Each configuration E and F gives different error in the measurement of the resistor. In the arrangement of Figure E, the voltmeter measures the voltage drop across the unknown resistor and, also, across the internal resistance of the ammeter. If,  $I$ , is the indication of the ammeter and  $V$  the indication of the voltmeter, the estimated value,  $R_m$ , of the unknown resistor is given by (2):

$$R_m = \frac{V}{I} = \frac{(R_a + R_x) \cdot I}{I} = R_a + R_x \quad (2)$$

Where,  $R_x$  is the actual value of the measured resistor.

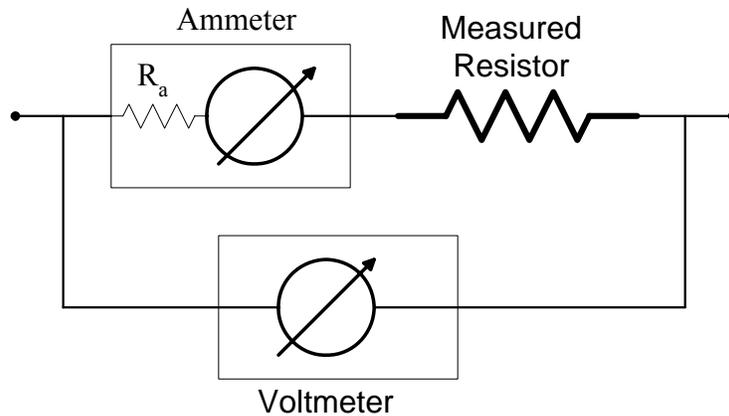


Fig. E. Series arrangement for the measurement of resistance.

Thus, the internal resistance of the ammeter acts as a parasitic resistor and introduces error in measurement.

In the arrangement of Figure F, the ammeter measures the current in the unknown resistor as well as the current in the voltmeter. Thus, the estimated resistance is:

$$R_m = \frac{V}{I} = \frac{V}{V/R_v + V/R_x} = R_x \cdot \frac{R_v}{R_v + R_x} \quad (3)$$

The resistance of the voltmeter acts, in this case, as a parasitic resistance to produce the error in the resistor measurement.

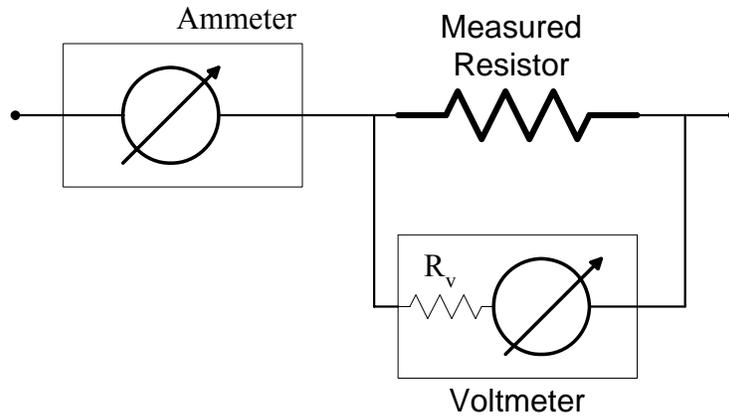


Fig. F. Parallel arrangement for the measurement of resistance.

*The Ohmmeter.* A simplified schematic diagram of the ohmmeter is shown in Figure G. The instrument employs a current sensor and a battery. The battery drives a constant current into the resistor measured by the current sensor. The value of the resistance is indicated on the instrument scale (sensor scale) from Ohm's law dividing the battery voltage by the current. The variable resistor in Figure G is adjusted so that only rated current flows into the sensor when the instrument terminals are shorted. The maximum deflection of the scale is, therefore, graded to zero ohms and the minimum deflection of the scale (open terminals) is graded to infinite ohms.

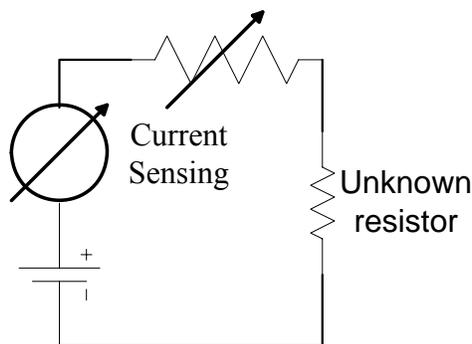


Fig. G. The simplified schematic diagram of an ohmmeter.

## II. INSTRUMENTATION

Power supply, Tektronix CPS250. Multi-meter, Tektronix CDM250, Metex M-3800.

## III. PROCEDURE:

### A. Familiarization with the Equipment.

1. Become familiar with the power supply, the analog and digital meters, and the resistor color code.

*B. Use of the Ohmmeter.*

4. Pick three resistors rated (according to their code)  $47\ \Omega$ ,  $4.7\ \text{k}\Omega$ , and  $680\ \text{k}\Omega$ .
5. Measure the resistance of the resistors using the ohmmeter.

*C. Measurement of Resistance Using an Analog Ammeter and a Voltmeter.*

7. Use the resistor rated  $47\ \Omega$  in the arrangement shown in Figure 1a. Set the voltage of the supply to 6 V. Measure the voltage and the current indicated in the figure.

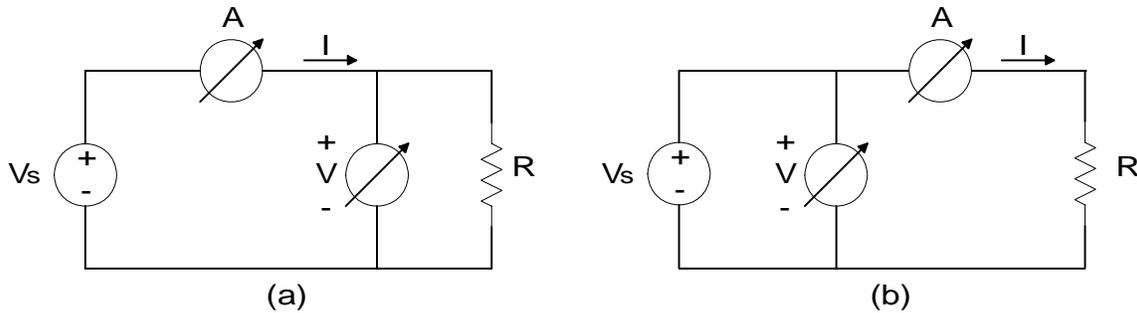


Fig. 1. The two arrangements of the voltmeter and ammeter for measuring resistance.

8. Repeat Procedure 7 using the arrangement of Figure 1b. Tabulate your measurements as in Table 1.

Table 1  
Measurement of resistance using a voltmeter and an ammeter

	Circuit 1a	Circuit 1b
V		
I		

**IV. REPORT:**

*A. Theoretical Development.*

1. (a) Design an ammeter with three scales:  $0\text{--}200\ \mu\text{A}$ ,  $0\text{--}2\ \text{mA}$ , and  $0\text{--}10\ \text{mA}$ . Use a current sensor rated at  $200\ \mu\text{A}$ ,  $100\ \text{mV}$ . What resistance does the instrument insert in the circuit at each scale?  
 (b) Using the same current sensor as in (a), design a voltmeter with two scales:  $0\text{--}10\ \text{V}$  and  $0\text{--}100\ \text{V}$ . What is the maximum current the instrument draws from the circuit for each scale?
2. Calculate the average value of the wave forms shown in Figure A. Use (1). Assume that each division on the time axis corresponds to  $1\ \text{s}$  and each division on the y-axis corresponds to  $1\ \text{V}$ . Also calculate the average value of  $20\cos(\omega t)$ .

*B. Resistance Measurements.*

3. Compare and discuss the measurements from Figures 1a and 1b. What is in each case the % error between the measured and rated value of the resistor?

## EXPERIMENT 2:

### OHM'S LAW AND APPLICATIONS

Ohm's law and its applications are investigated in this experiment. The V-I characteristic of linear resistors is derived. Applications of Ohm's law include voltage and current division. Measurements of the equivalent resistance of a resistive arrangement are performed.

#### I. BACKGROUND

##### *I.1 Ohm's Law.*

Ohm's law states that the voltage and current in a resistor are directly proportional. Resistors that obey the Ohm's law are called linear or ohmic resistors. In an ohmic resistor the ratio of the resistor voltage to the resistor current is independent of the voltage and current. This ratio is defined as the resistance of the resistor and it is measured in  $\Omega$  (Ohms). Equation (1) expresses the Ohm's law. This equation is the terminal equation of a linear resistor.

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

Equation (1) can also be written as in (2):

$$I = G \cdot V \tag{2}$$

Where  $G=1/R$  is the conductance of the resistor (measured in Siemens, S). Representation of the terminal equation of a resistor in the form of (1) is called resistance or impedance representation. Representation in the form of (2) is called conductance or admittance representation.

Resistors whose resistance varies with the voltage or the current in the resistor are called non-linear resistors. Non-linear resistors are described by a non-linear relation between their voltage and current.

##### *I.2 The Voltage versus Current Characteristic of Linear Resistors.*

The graph of the voltage v. the current of a linear resistor is a line called the V-I characteristic. With reference to Figure A, the V-I characteristic of the resistor always passes through the origin. Its slope is the resistance of the resistor. Its reciprocal slope is the conductance of the resistor. An ohmic resistor characteristic occupies only the first and third quadrant of the V-I plane. Thus, an ohmic resistor dissipates energy at any point of its characteristic.

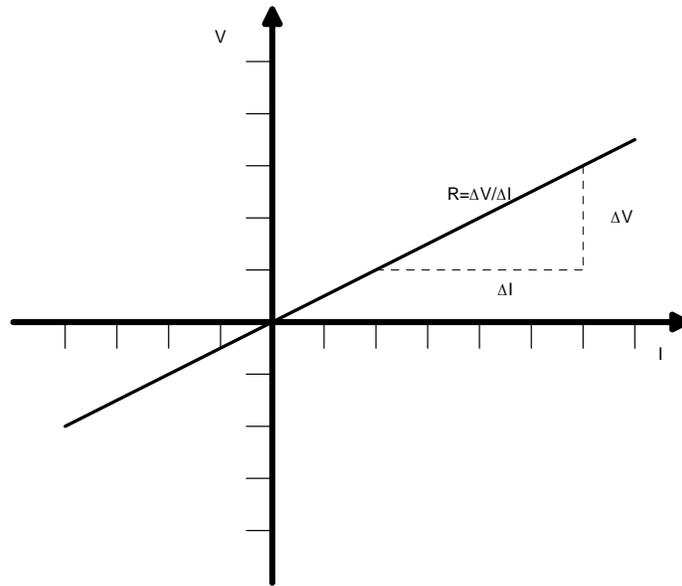


Fig. A. The V-I characteristic of a linear resistor.

### 1.3 Series and Parallel Combination of Resistors.

The equivalent resistance of resistors connected in series equals the summation of the resistance of each resistor. With reference to Figure B, the equivalent V-I characteristic of series resistors is obtained by the vertical summation of the V-I characteristics of the resistors. The slope of the equivalent characteristic is, thus, the summation of the slopes of each of the added characteristic.

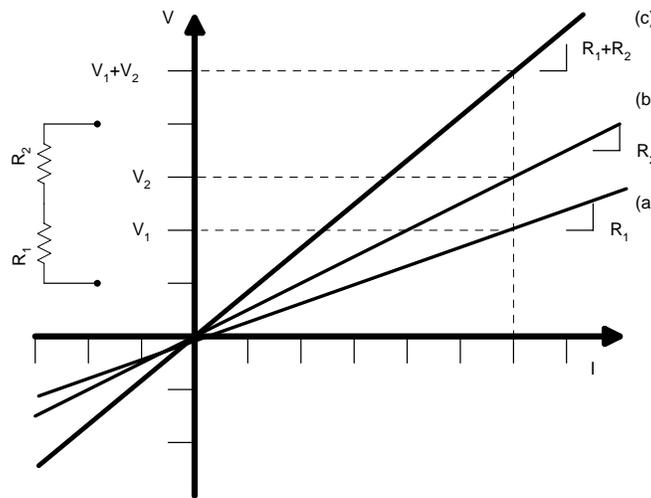


Fig. B. The combined V-I characteristic of series resistors. (a) and (b) The V-I characteristics of the resistors. (c) The combined V-I characteristic.

The voltage across the terminals of a series arrangement is distributed among the resistors proportionally to the resistance of each resistor (voltage division).

The equivalent conductance of resistors connected in parallel equals the summation of the conductance of each resistor. With reference to Figure C, the V-I characteristic of the parallel combination of resistors is obtained by the horizontal summation of the V-I characteristics of the resistors. The reciprocal slope of the V-I characteristic of parallel resistors is the summation of the reciprocal slopes of the V-I characteristics of the resistors.

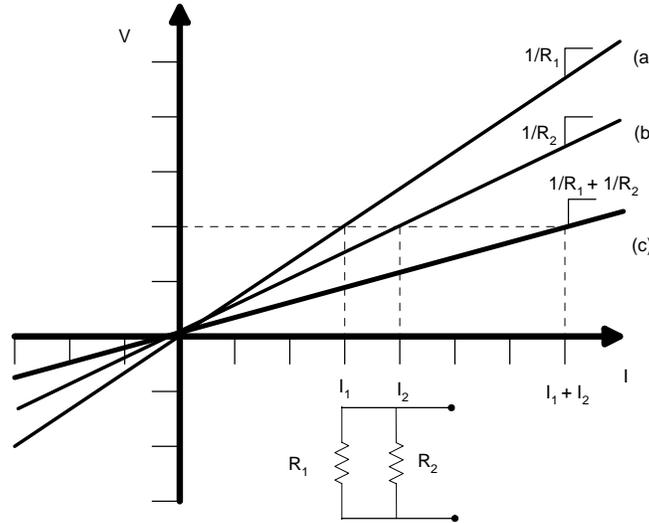


Fig. C. The combined V-I characteristic of parallel resistors. (a) and (b) The V-I characteristics of the resistors. (c) The combined V-I characteristic.

Parallel resistors divide the current at the terminals of the arrangement proportionally to their conductance (current division).

#### 1.4 Direct Measurement of Resistance. The Wheatstone Bridge.

Direct measurement of resistance is achieved by comparing the unknown resistance to a standard (known) resistance. Figure D shows the arrangement known as the Wheatstone bridge that is used for the direct measurement of resistance. In this arrangement,  $R_x$  is the unknown resistor and  $R_3$  is a variable resistor (rheostat).

The bridge consists of two voltage dividers formed by resistors  $R_1$  and  $R_3$  and by resistors  $R_2$  and  $R_x$ . The voltage across  $R_3$  and  $R_x$  are given by (3). The voltage  $V_o$  at the output terminals of the bridge is the difference between the two previous voltages and is given by (4).

$$V_3 = \frac{R_3}{R_1 + R_3} \cdot V_B, \quad V_x = \frac{R_x}{R_x + R_2} \cdot V_B \quad (3)$$

$$V_o = V_3 - V_x = \left[ \frac{R_3}{R_3 + R_1} - \frac{R_x}{R_x + R_2} \right] \cdot V_B \quad (4)$$

The value of  $R_3$  varies by known steps until the voltage at the output of the bridge is zero. At this state the bridge is balanced. The condition to balance the bridge is derived from (4) and it is given in (5). The unknown resistance is given by (6).

$$\frac{R_x}{R_2} = \frac{R_3}{R_1} \quad (5)$$

$$R_x = \frac{R_2 R_3}{R_1} \quad (6)$$

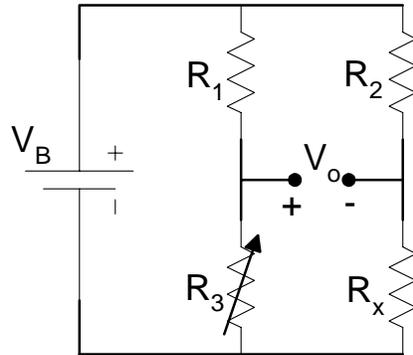


Fig. D. Arrangement of the Wheatstone bridge.

## II. INSTRUMENTATION

Power supply, Tektronix CPS250. Multi-meter, Tektronix CDM250, Metex M-3800. RLC-bridge, GenRad 1657. Decade resistor box 1432-L.

## III. PROCEDURE

### A. Measurement of the V-I Characteristic of Ohmic Resistors.

1. Pick two resistors rated at 1 k $\Omega$  and 3 k $\Omega$ . Measure their values on the RLC-bridge.
2. Construct the circuit of Figure 1 using the 1 k $\Omega$  resistor.

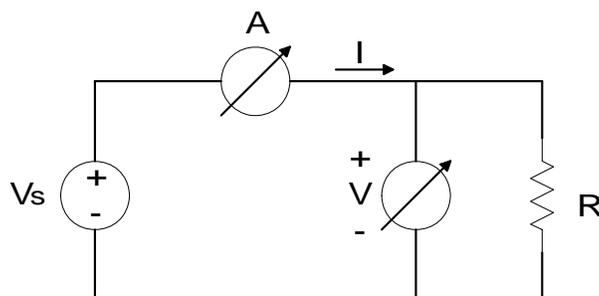


Fig. 1. Arrangement to obtain the V-I characteristic of a resistor.

3. Vary the voltage of the supply by increments of 2 V until 6 V. For each setting of the supply voltage measure the voltage and current indicated in Figure 1. Tabulate your measurements in Table 1.

Table 1.

The V-I measurements for the 1 kΩ resistor

	Step 1	Step 2	Step 3
V			
I			

*B. Series and Parallel Combinations of Resistors.*

- Pick three resistors rated at  $R_1=1\text{ k}\Omega$ ,  $R_2=2\text{ k}\Omega$ , and  $R_3=3\text{ k}\Omega$ . Measure their values in the RLC-bridge.
- Construct, one at a time, Arrangements a and b. Set the supply to 6 V.
- For each arrangement, measure the indicated variables. Tabulate your results.

Arrangement a:

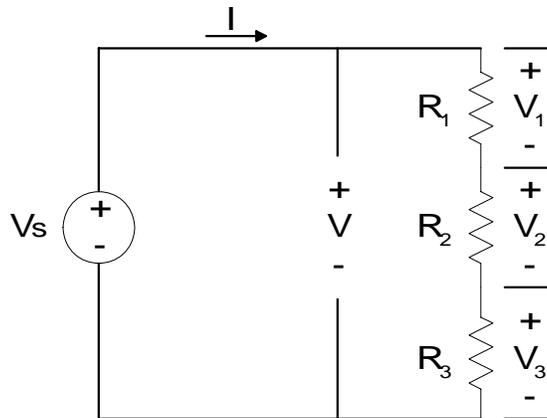


Fig. 2. Series resistors. Voltage division.

Table 2.  
Measurements for Arrangement of Figure 2.

V	I	V <sub>1</sub>	V <sub>2</sub>	V <sub>3</sub>

Arrangement b:

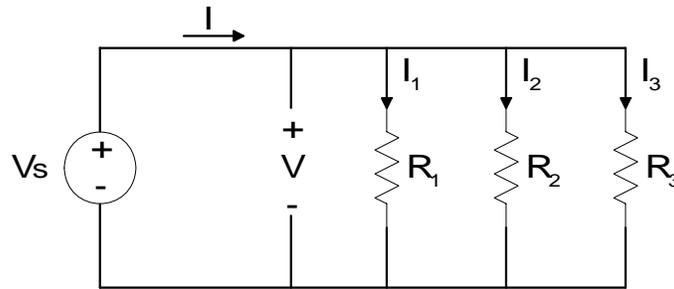


Fig. 3. Parallel resistors. Current division.

Table 3.  
Measurements for Arrangement of Figure 3.

V	I	$I_1$	$I_2$	$I_3$

*C. Wheatstone Bridge.*

- (a) Construct the Wheatstone bridge in Figure D. Use a 9 V source and  $R_1=50\text{ k}\Omega$ ,  $R_2=40\text{ k}\Omega$ , a resistance box and a third resistor for  $R_x$ . Measure the value of  $R_x$  using the ohmmeter.
- (b) Connect a voltmeter at the output of the bridge. Adjust the resistance box to balance the bridge. Record the value of the box resistance.

IV. REPORT:

*A. Theoretical Development.*

- (a) Discuss Ohm's law. What is a rheostat? What is a potentiometer? Describe the use of each.
- (b) Provide the formulae for voltage and current division between two resistors.

*B. Measurement of the V-I Characteristic of Ohmic Resistors.*

- Plot the V-I characteristic of the 1 k $\Omega$  resistor from the measurements in Table 1. Use a millimeter paper. Use appropriate scale and label both axes. Use linear interpolation.
- Estimate the resistance from the slope of the line in Step 2. Compare this value with the resistance you measured on the RLC-bridge.

*C. Series Connected resistors.*

- Use the measurements of Table 2 to verify the KVL in the circuit of Arrangement a.
- Compare these measurements with voltage division.
- Calculate from the measurements the power consumed by each resistor and the power delivered by the supply. Discuss your results.

*D. Parallel Connected Resistors.*

- Use the measurements of Table 3 to verify the KCL in the circuit of Arrangement b.
- Compare these measurements with current division.

9. Calculate from the measurements the power consumed by each resistor and the power delivered by the supply. Discuss your results.

*E. Wheatstone Bridge.*

10. Calculate from the measurements the value of resistor  $R_x$ . Compare it with its rated value.

## EXPERIMENT 3:

### ANALYSIS OF NETWORKS

The purpose of this experiment is to introduce students to the nodal voltage and mesh current methods for solving circuits. pSpice and MATLAB are introduced for solving the circuit.

#### I. BACKGROUND

##### *I.1 Development of the Nodal Equations.*

The nodal equations express the KCL in terms of the nodal voltages of the circuit. The KCL at each node is written assuming that all branch currents leave the node. The nodal equations of the circuit of Figure A are:

$$\begin{aligned} \frac{V_1}{R_1} + \frac{V_1 - V_2}{R_2} &= J_1 \quad \text{at Node 1} \\ \frac{V_2}{R_3} + \frac{V_2 - V_1}{R_2} &= J_2 \quad \text{at Node 2} \end{aligned} \tag{1}$$

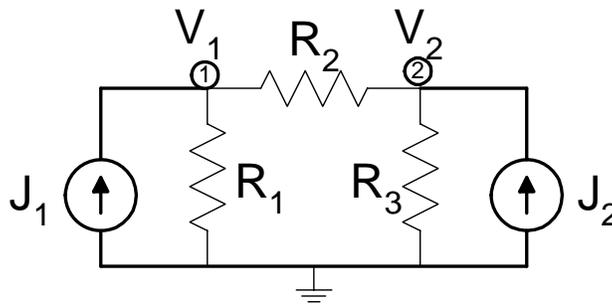


Fig. A. A two node network with current injections.

Equation (1) can be written as:

$$\begin{aligned} \left( \frac{1}{R_1} + \frac{1}{R_2} \right) \cdot V_1 - \frac{1}{R_2} \cdot V_2 &= J_1 \\ -\frac{1}{R_2} \cdot V_1 + \left( \frac{1}{R_2} + \frac{1}{R_3} \right) \cdot V_2 &= J_2 \end{aligned} \tag{2}$$

The general characteristics of the nodal equations are: The diagonal coefficients represent the total conductance directly connected to the node. The off diagonal coefficients represent the negative of the total conductance connected between any two nodes.

The nodal equations in (2) can be cast in a matrix form as in (3).

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix} \cdot \begin{bmatrix} V_1 \\ V_2 \end{bmatrix} = \begin{bmatrix} J_1 \\ J_2 \end{bmatrix} \quad (3)$$

$$y_{12} = y_{21}$$

The matrix in (3) is symmetric and is called the nodal admittance matrix of the circuit. It is often referred to as the  $Y_{\text{bus}}$ . The elements of this matrix are the coefficients in (2).

### 1.2. Physical Interpretation of the Nodal Admittance Matrix.

With reference to Figure B, if Node 2 is shorted to the ground and a voltage  $E_1$  is applied to Node 1, then the elements  $y_{11}$  and  $y_{21}$  of the nodal admittance matrix are:

$$y_{11} = \frac{I_1}{E_1} \quad y_{21} = \frac{I_2}{E_1} \quad (4)$$

Where  $I_1$  and  $I_2$  are the currents measured into Nodes 1 and 2 respectively. Similarly for Node 2.

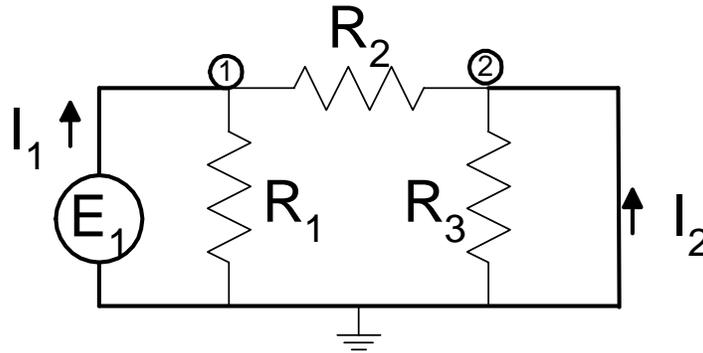


Fig. B. Physical interpretation of the nodal admittance matrix.

### 1.3 The Nodal Impedance Matrix.

The solution of (3) yields:

$$\begin{bmatrix} V_1 \\ V_2 \end{bmatrix} = \begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}^{-1} \cdot \begin{bmatrix} J_1 \\ J_2 \end{bmatrix} = \begin{bmatrix} z_{11} & z_{12} \\ z_{21} & z_{22} \end{bmatrix} \cdot \begin{bmatrix} J_1 \\ J_2 \end{bmatrix} \quad (5)$$

The matrix in (5) that relates the nodal currents to the nodal voltages is the nodal impedance matrix. This matrix is the inverse of the nodal admittance matrix and it is also symmetric. The nodal impedance matrix is often referred to as the  $Z_{\text{bus}}$  matrix and has many applications in the study of power networks especially in the study of faults and protection.

The elements of the nodal impedance matrix can be measured experimentally. With reference to Figure C, the elements  $z_{11}$  and  $z_{21}$  of the matrix are derived by (a) applying a voltage  $E_1$  to Node 1, (b) measuring the resulting current  $I_1$  and voltage  $E_2$ . Then according to (6):

$$z_{11} = \frac{E_1}{I_1} \quad z_{21} = \frac{E_2}{I_1} \quad (6)$$

It can be seen from (6) that the element  $z_{11}$  is also the Thévenin impedance seen from Node 1 (between the node and the ground).

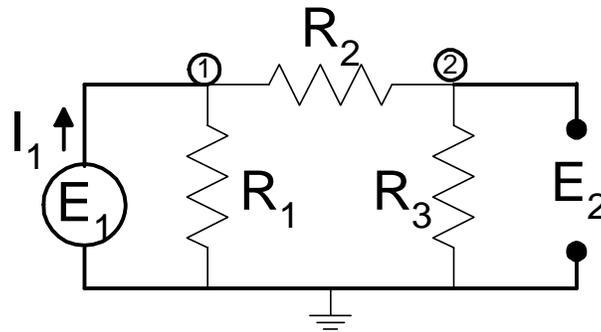


Fig. C. Experimental measurement of the nodal impedance matrix.

## II. INSTRUMENTATION:

Power supply, Tektronix CPS250. Multi-meter, Tektronix CDM250; Metex M-3800. RLC-bridge, GenRad 1657.

## III. PROCEDURE:

### A. Verification of the Nodal Equations.

1. Construct the circuit of Figure 1. Use the following resistors.

$R_0=510 \Omega$ ,  $R_1=1k$ ,  $R_2=6.8k$ ,  $R_3=3.3k$ ,  $R_4=5.1k$ ,  $R_5=2.k$ ,  $R_6=10k$ .

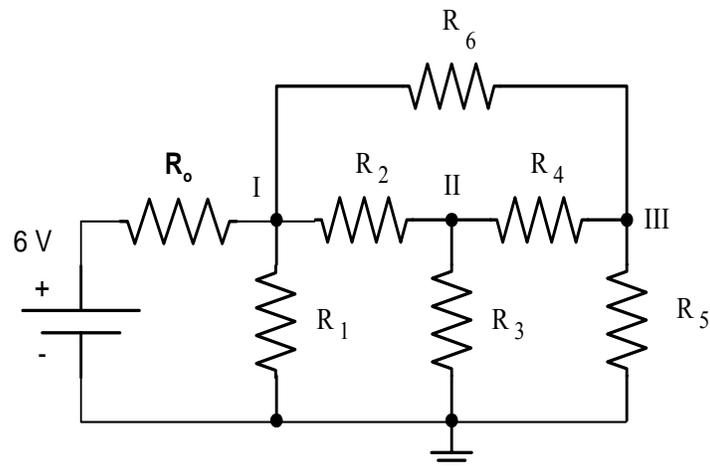


Fig. 1. The circuit for the nodal analysis.

2. Activate the source and measure the nodal voltages.

*B. Verification of the Mesh Equations.*

3. Measure the mesh currents, as shown in Figure 2.

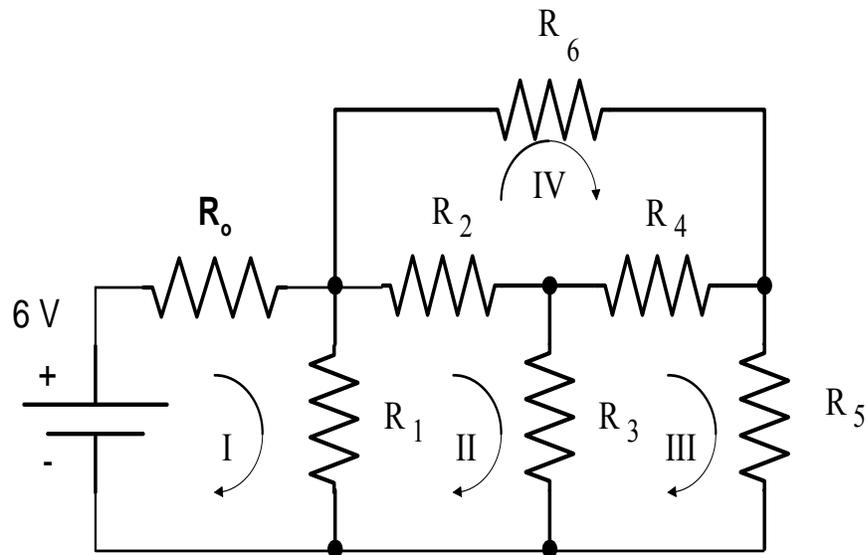


Fig. 2. The experimental setting for the mesh currents.

#### IV. REPORT

*A. Theoretical Development.*

1. Discuss the nodal and mesh method. When is one preferred over the other.
2. Write the nodal and mesh equations of the experiment circuit. Solve the equations using MATLAB.
3. Simulate the same circuit in pSpice. Compute the nodal voltage and mesh currents.

*B. Verification of Nodal Equations.*

4. Compare the measured values of the nodal voltages with the values from MATLAB and pSpice. Write the  $Y_{bus}$  and the  $Z_{bus}$  matrices of the circuit. What is the Thevenin resistance between Node II and ground?
5. Write the nodal equations and verify using the measurements the KCL at each node.

*C. Verification of Mesh Equations.*

6. Compare the mesh currents measured with the values from MATLAB and pSpice.
7. Write the circuit mesh equations and verify using the measurements the KVL in each mesh.

# EXPERIMENT 4:

## NETWORK THEOREMS

This experiment verifies some important network theorems: the Thévenin equivalent of a circuit, the maximum power transfer theorem, and the source superposition.

### I. BACKGROUND

#### *1.1 Representation of a Linear Resistive Circuit by a Thévenin Source.*

A linear resistive circuit seen (observed) from two of its terminals (a two-terminal circuit) is equivalent to a Thévenin or a Norton source connected between the two terminals. These are called Thévenin or Norton equivalents respectively of the circuit.

Definitions: If a circuit contains only ohmic resistors, dependent, and independent sources (that is no capacitors or inductors) the circuit is called a linear resistive circuit.

The terminal voltage and current of a two-terminal linear resistive circuit are related by a linear equation. The Thévenin or Norton equivalency imply that a two-terminal linear resistive circuit can be replaced by a Thévenin or Norton source connected between the circuit terminals. Equivalency means that the circuit and the Thévenin source have identical V-I characteristics at their terminals.

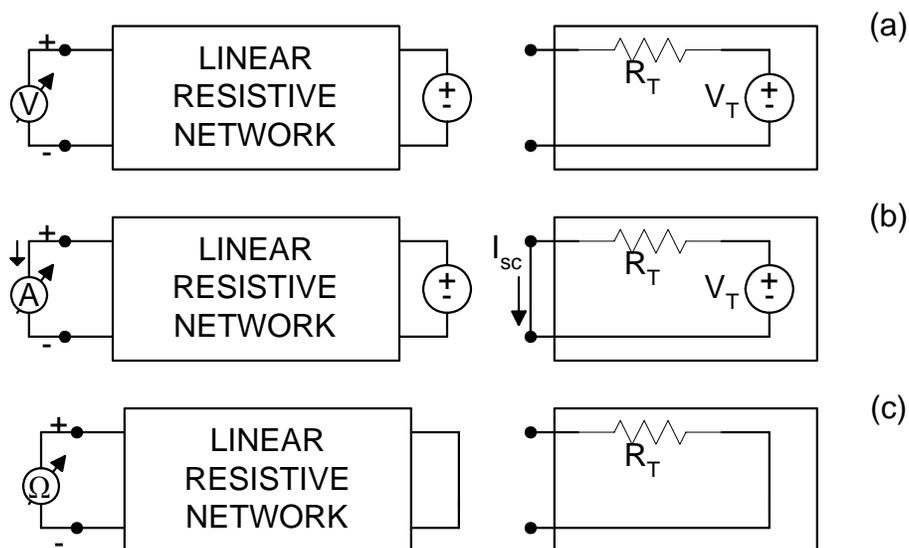


Fig. A. Derivation of the Thévenin equivalent. (a) The circuit and its equivalent Thévenin source. (b) The short circuit measurement. (c) The terminal resistance measurement.

Figure A(a) shows a two-terminal linear-resistive circuit. When observed from its terminals the circuit appears as a Thévenin source with voltage  $V_T$  (termed the Thévenin voltage) and

resistance  $R_T$  (termed the Thévenin resistance). The relation between the terminal voltage and current of the circuit is given by (1).

$$V = -R_T \cdot I + V_T \quad (1)$$

Figure B shows the terminal characteristic of (1). The intercept of this line on the V-axis is the Thévenin voltage. Figure A(a) shows that the Thévenin voltage can be measured directly across the open terminals of the network.

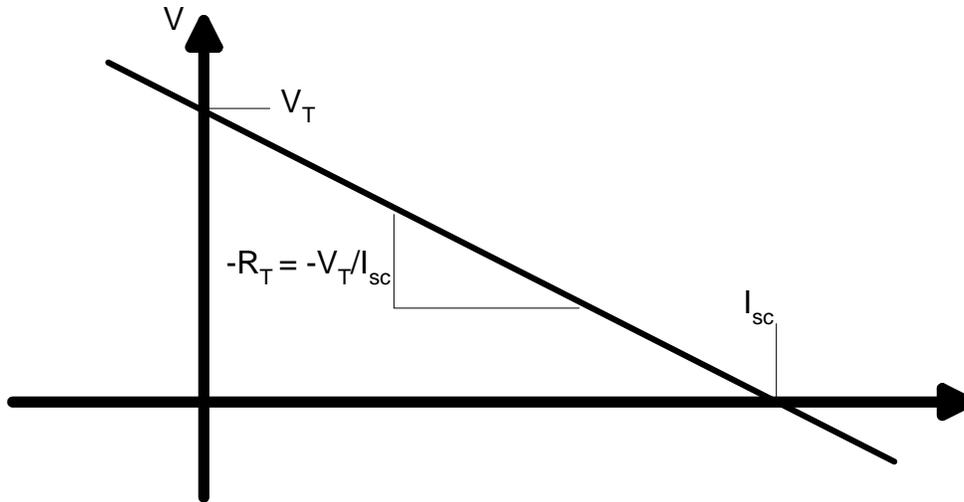


Fig. B. The V-I characteristic of the linear circuit.

The intercept of the circuit characteristic on the I-axis is the short circuit current (also equal to the Norton current). Figure A(b) shows that this current is measured when we short the terminals of the circuit. The short circuit current is given by (2).

$$I_{sc} = \frac{V_T}{R_T} \quad (2)$$

The slope of the characteristic is the opposite of the Thévenin resistance,  $R_T$ . The Thévenin resistance can be measured either (a) from the ratio of the Thévenin voltage to the short circuit (Norton) current:

$$R_T = \frac{V_T}{I_{sc}} \quad (3)$$

Or (b) with an ohmmeter across the circuit terminals, as shown in Figure A(c): all independent sources of the circuit are made zero (voltage sources are replaced by a short circuit, current sources by an open circuit). The resistance measured at the terminals is, then, the Thévenin resistance.

The experimental determination of the Thévenin equivalent of a circuit requires, therefore, these measurements: (a) Open circuit voltage (Figure A(a)); and one of the following: (b) short circuit current (Figure A(b)), or (c) terminal resistance (Figure A(c)).

### I.2. Y- $\Delta$ Resistance Transformation.

This is a useful transformation for the simplification of circuits, (e.g. derivation of Thévenin equivalents). Figure C(a) shows a  $\Delta$  arrangement of resistors. Figure C(b) shows its equivalent Y arrangement. The values of the resistors are given by (4) and (5).

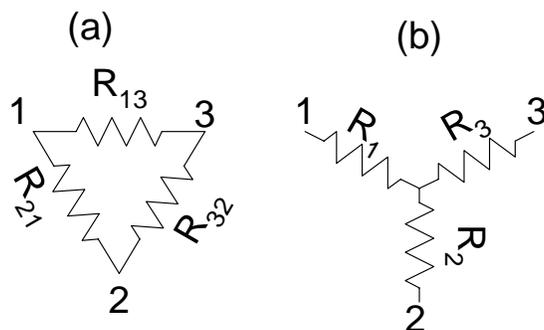


Fig. C. Y-D transformation. (a)  $\Delta$  Arrangement. (b) Y Arrangement.

$$\begin{aligned}
 R_{31} &= \frac{\Pi}{R_2} \\
 R_{21} &= \frac{\Pi}{R_3} \\
 R_{32} &= \frac{\Pi}{R_1} \\
 \Pi &= R_1 \cdot R_2 + R_2 \cdot R_3 + R_3 \cdot R_1
 \end{aligned}
 \quad \text{For a Y-}\Delta \text{ transformation, and} \tag{4}$$

$$\begin{aligned}
 R_1 &= \frac{R_{21} \cdot R_{31}}{\Sigma} \\
 R_2 &= \frac{R_{21} \cdot R_{32}}{\Sigma} \\
 R_3 &= \frac{R_{31} \cdot R_{32}}{\Sigma} \\
 \Sigma &= R_{31} + R_{32} + R_{21}
 \end{aligned}
 \quad \text{For the inverse } \Delta\text{-Y transformation.} \tag{5}$$

## II. INSTRUMENTATION

Power supply, Tektronix CPS250. Multi-meter, Tektronix CDM250, Metex M-3800. RLC-bridge, GenRad 1657. Decade resistor Box, 1432-L.

## III. PROCEDURE

### A. Experimental Determination of the Thévenin Equivalent.

1. Construct the circuit shown in Figure 1, where  $R_1 = 1.2 \text{ k}\Omega$ ,  $R_2 = 2.4 \text{ k}\Omega$ , and  $R_3 = 4.7 \text{ k}\Omega$ . Set the supply at 8 V. Perform the following measurements:
  - a. Measure the voltage across the open terminals a-b.
  - b. Short the terminals a-b through an ammeter. Measure the current.
  - c. Disconnect the supply from the circuit and replace it by a short circuit. Measure the resistance of Terminals a-b using an ohmmeter.

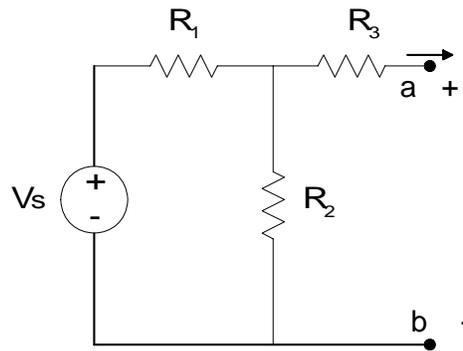


Fig. 1. Experimental arrangement.

*B. Maximum Power Transfer.*

2. Connect a variable resistor between the terminals a-b of the previous circuit. Vary its value in the range from 1.0 to 10  $\text{k}\Omega$  by increments of 1000  $\Omega$ . For each value of the resistance, measure the voltage, V, across it. Use Table 1.

Table 1.  
Measurements for the maximum power transfer.

<b>R</b>										
<b>V</b>										

*C. Source Superposition.*

3. Construct the circuit of Figure 2 by connecting a second source across the terminals a-b of the previous circuit. Use  $R_4 = 5.1 \text{ k}\Omega$ . Use the two outputs of the power supply for each of the sources.

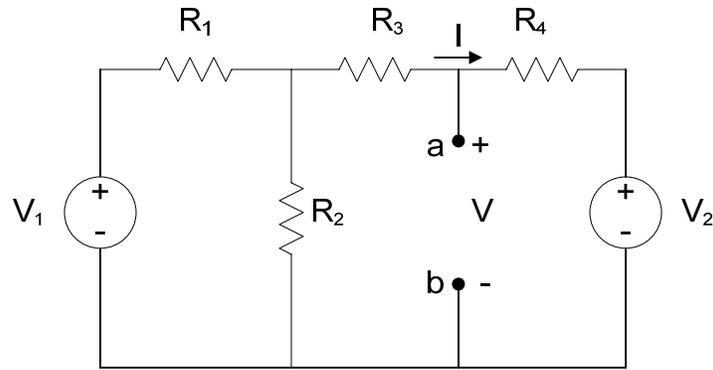


Fig. 2. The experimental arrangement with a Thevenin source.

4. Set  $V_1$  to 8 V and  $V_2$  to 6 V. Measure  $V$  and  $I$ .
5. Replace  $V_2$  by a short-circuit. Measure  $V$  and  $I$ .
6. Reconnect  $V_2$  and set its value to 6 V. Replace  $V_1$  by a short-circuit.. Measure  $V$  and  $I$ .

#### IV. REPORT

##### A. Theoretical Development.

1. (a) Discuss the Thévenin and Norton equivalents and source transformation. Give an example of a two-terminal circuit for which no-equivalent Thévenin source exists.
- (b) Prove the theorem of maximum power transfer.

##### B. Determination of Thévenin equivalent.

2. Calculate the Thévenin equivalent of Figure 1 from:
  - (a) The measurements in Procedures 1a and 1b.
  - (b) The measurements in Procedures 1a and 1c.
 Compare with the theoretical Thévenin equivalent.

##### C. Maximum Power Transfer.

3. Plot the power dissipated on the variable resistor vs the resistance in Procedure 2. Tabulate your calculations. At what value of the resistor does maximum power occur? How much is the maximum power? How close do these values correspond with the theoretical ones from the maximum power transfer theorem?

##### D. Source Superposition.

4. Do the measurements of the voltage and current from Procedures 4, 5, and 6 agree with the principle of source superposition? Explain.

## EXPERIMENT 5:

### FIRST ORDER R-L AND R-C CIRCUITS

The objective of this experiment is to observe the response of the first order R-C and R-L circuits. The experiment demonstrates a method for measuring the time constant.

#### I. BACKGROUND

##### *I.1 Measurement of the Natural Response of First Order Circuits.*

The natural response of an R-C circuit, shown in Figure A, is given by (1).

$$v(t) = v_o \cdot e^{-\frac{t}{\tau}} \quad (1)$$

Where,  $v_o$  is the value of the capacitor voltage  $v$  at  $t=0s$ ,  $\tau=RC$  is the time constant of the circuit. The time constant gives the rate at which the voltage decays to zero. In circuits, this decay response is due to ohmic losses.

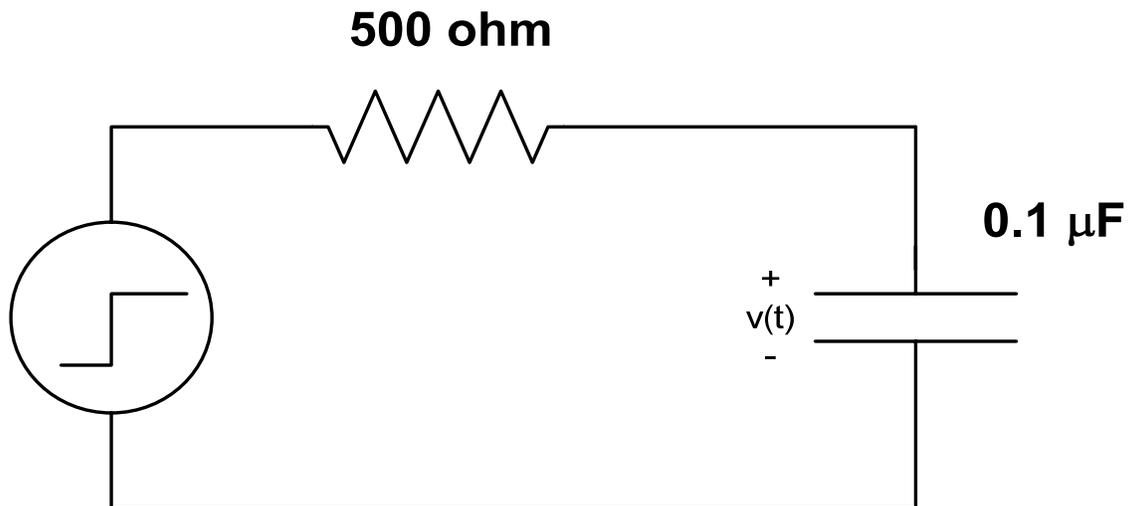


Fig. A. An R-C circuit.

The time constant can be measured by one of the following graphical methods.

(a) *The Tangential Line.* A line tangential at a certain point of the response curve is drawn. The line intersects the time axis exactly in one time constant from the point of tangent. Figure B shows the graph of the response from (1). Note that the response begins at  $t= 0.05$  s. With reference to Figure B, a line is drawn tangential to the curve at  $t= 0.05$  s (that is at the onset of

the decay). The line is extended until it intersects the time axis. This occurs at  $t = 0.25$  s. Therefore, the time constant of the response is

$$\tau = 0.25 - 0.05 = 0.2 \text{ s} \quad (2)$$

This method is suitable if a hard copy of the response graph is available.

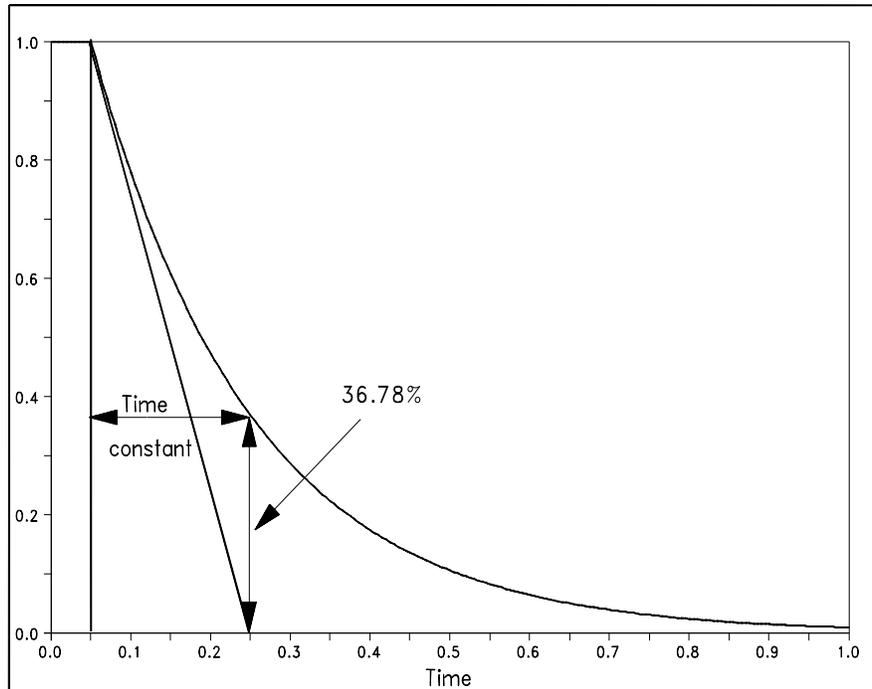


Fig. B. Measurement of the time constant of an exponentially decaying response.

(b) *The 63.22% Decay.* The second method is suitable for measurements on the oscilloscope. From (1), every time interval equal to one time constant the response decays by 63.22%. Equivalently, at the end of every interval equal to one time constant, the response is at the  $100 - 63.22 = 36.78$  % of its value at the beginning of the interval. This method of measurement is demonstrated in Figure B. The response begins its decay at  $t = 0.05$  s, at that point the value of the curve is 1. The response reaches 36.78 % of its initial value at  $t = 0.25$  s. Thus the time constant is

$$t = 0.25 - 0.05 = 0.2 \text{ s} \quad (3)$$

The results of (3) and (2) are in agreement.

### 1.2 The Step Response of First Order Circuits.

The step response of the first order circuit in Figure A is given by (4).

$$v(t) = v_f \left( 1 - e^{-\frac{t}{\tau}} \right) \quad (4)$$

Where,  $v_f$  is the final value of the response (also, called the constant steady state of the response). The step response of the circuit is characterized by the same time constant as the natural response. The time constant affects the step response in a similar manner as the natural response. Therefore, the methods discussed previously apply in this case, as well.

On the oscilloscope the time constant of the step response can be measured by measuring the time the output requires to reach 63.22% of its final value.

The natural and step responses of a first order R-L circuit, shown in Figure C, are the same as in (1) and (4) given for the circuit current  $i$ . The circuit time constant is  $\tau=L/R$ .

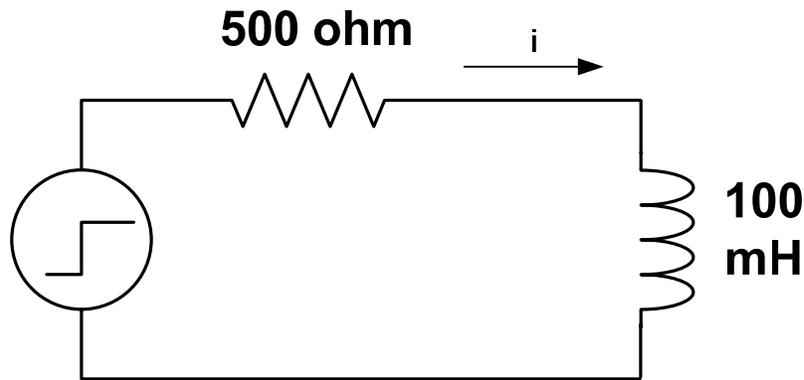


Fig. C. The experimental R-L circuit.

### 1.3 Oscilloscope Observations.

Non-storage oscilloscopes must display periodic wave forms, so that measurements and observations can be easily made. The natural response of a circuit can be observed along with its step response, if a square-pulse periodic excitation is applied to the circuit.

Figure D shows the response of a first order circuit to a periodic square pulse. The first part of the circuit response is the step response described by (4). The second part of the circuit response, when the excitation becomes zero, is the natural response described by (1). If the response is allowed to decay sufficiently, the following period will begin with an other step response followed by the natural response. Thus, the circuit response will alternate between step and natural responses. To achieve good results, therefore, the period of the excitation must be adjusted to 7 to 10 times the time constant of the circuit.

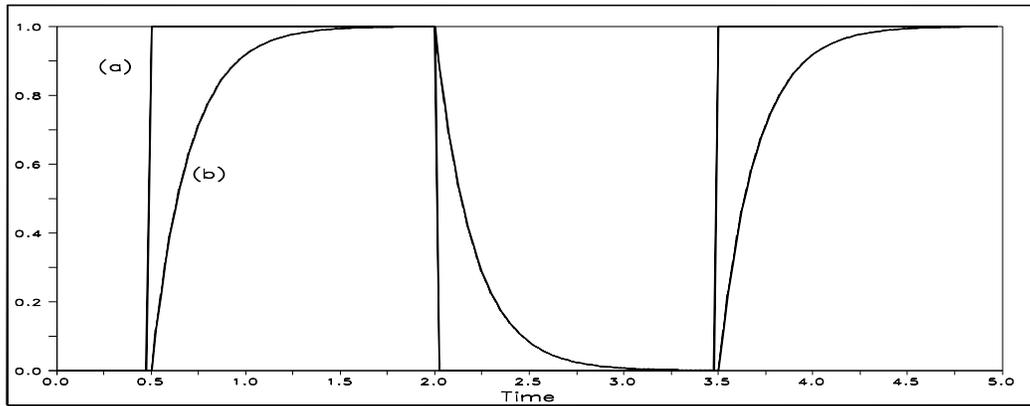


Fig. D. Circuit response to a square pulse excitation with large period. (a) The excitation. (b) The circuit response.

## II. INSTRUMENTATION

Function generator, Tektronix CFG 250. Oscilloscope, Tektronix 2205-20 MHz.

## III. PROCEDURE:

Become familiar with the signal generator and the oscilloscope.

### A. The R-C Circuit.

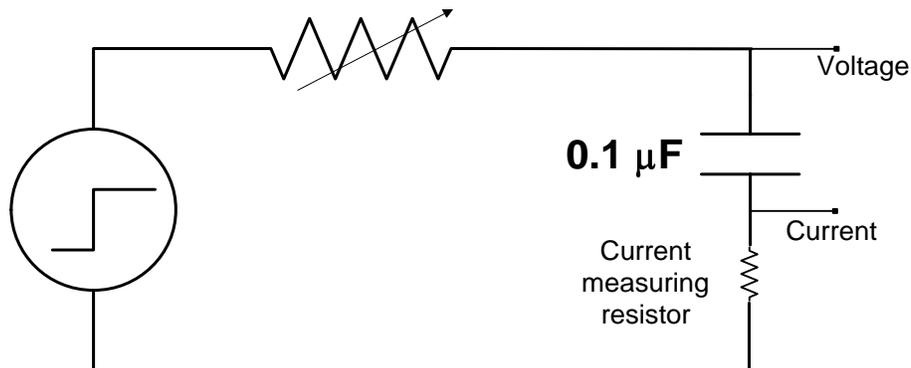


Fig. 1. The experimental R-C circuit.

1. (a) Construct the R-C circuit of Figure 1. Use a  $510\ \Omega$ .
- (b) Apply the square pulse generator to produce a periodic response of the circuit that appears clearly on the oscilloscope. The period of the source is adjusted to be at least 20 times higher than the calculated time constant of the circuit. This is necessary, so that the capacitor reaches steady state, in both charging and discharging intervals.
- (c) Observe on the oscilloscope the quantities shown in the figure. Measure the circuit time constant by the 63% decay method described above.

- (d) Plot on a millimeter paper one period of the voltage and current response. Use appropriate scale. Label the axes. Transfer measurements from the oscilloscope to the paper by inspection.
2. Change the resistance in Figure 3 to  $250\ \Omega$ . Repeat Procedure 1. Use the same millimeter paper and the same scale. Align the graphs. Do not change the setting of the source.

*B. The R-L Circuit.*

3. (a) Construct the circuit of Figure 2. Set the resistance box to  $500\ \Omega$ .  
 (b) Apply a square pulse voltage. Set the frequency of the source to correspond to 10 times the time constant of the circuit.  
 (c) Repeat the same step 1c for this circuit.
4. Change the resistance to  $250\ \Omega$ . Perform again step 1c.
5. Simulate the experiment circuit (Figure 2) in Pspice. For each value of R used above compute the circuit time constant.

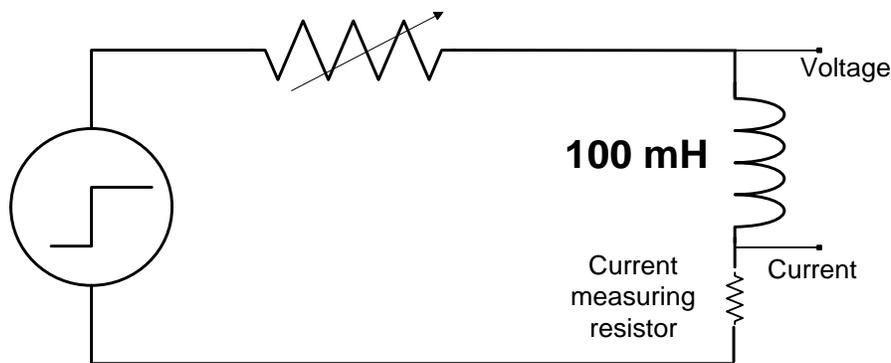


Fig. 2. The experimental R-L circuit.

IV. REPORT

*A. Theoretical Development.*

1. In theory discuss the natural and step responses of R-C and R-L circuits.
2. Define the duty cycle of a pulse. What is the duty cycle of a square pulse?
3. What is the difference in the setting of x1 and x10 of the oscilloscope probes?
4. How do we adjust the voltage and time scale of the oscilloscope? What is the purpose of calibrating a channel?
5. What is a waveform? What kind of waveforms are best observed and measured by oscilloscopes similar to the one used in the experiment?
6. What is a current measuring resistor? What was the value of the current measuring resistor used in the experiment? Is this value satisfactory?

*B. The R-C Circuit.*

7. (a) Explain the response observed in Procedure 1. Indicate which interval represents the natural and which interval represents the step response of the circuit.  
 (b) What is the circuit time constant measured for each value of R?

*C. Pspice Simulation of the R-L Circuit.*

8. Compare the time constant values obtained from the simulations with those obtained from the measurements.

# EXPERIMENT 6:

## SECOND ORDER RLC CIRCUITS

This experiment demonstrates the response of a series and a parallel RLC circuit. The over-damped, critically damped and under-damped responses are derived for each circuit.

### I. BACKGROUND

#### *I.1 The Natural Response of a Second Order Circuit.*

The natural response of second order circuit is derived from the solution of its characteristic differential equation (1).

$$\frac{d^2y}{dt^2} + 2 \cdot \beta \cdot \frac{dy}{dt} + \omega_o^2 \cdot y(t) = 0 \quad (1)$$

In (1) above,  $\beta$  is the damping coefficient and  $\omega_o$  is the frequency of undamped oscillations of the circuit (also called the resonance frequency for under damped circuits). The type of response derived from (1) is determined by the above two parameters. We distinguish these cases:

- (a)  $\beta < \omega_o$ . The response is under damped. The voltage and current oscillate with a diminishing amplitude.
- (b)  $\beta = \omega_o$ . The response is critically damped.
- (c)  $\beta > \omega_o$ . The response is over damped. The voltage and current decay exponentially.

The under damped response is characteristic of second order circuits. Hence, we emphasize it.

#### *I.2 Measurement of the Natural Under damped Response.*

The natural response of an under damped circuit is given by (2).

$$y(t) = Y_m \cdot e^{-\beta t} \cdot \cos(\omega_d \cdot t + \phi) \quad (2)$$

Where,  $\omega_d$  is the frequency of damped oscillations given by (3). The parameters  $Y_m$  and  $\phi$  are found from the initial conditions of the circuit according to (4).

$$\omega_d = \sqrt{\omega_o^2 - \beta^2} \quad (3)$$

$$y(0) = Y_m \cdot \cos(\phi)$$

$$\left. \frac{dy}{dt} \right|_{t=0^+} = -\beta \cdot Y_m \cdot \cos(\phi) - \omega_d \cdot Y_m \cdot \sin(\phi) \quad (4)$$

Figure A shows an under damped response. The envelope of the response is exponential with a time constant  $\tau = 1/\beta$ .

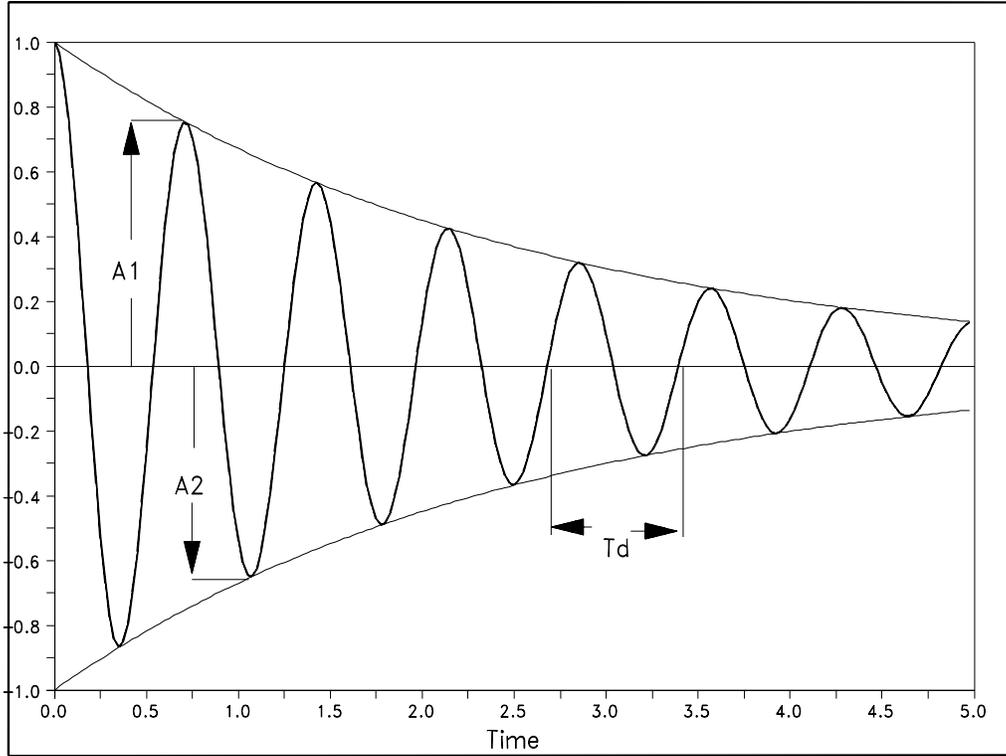


Fig. A. The responses of an under damped second order circuit.

The response can be also expressed in terms of the damping ratio according to (5).

$$y(t) = Y_m \cdot e^{-2\cdot\zeta\frac{t}{T_d}} \cdot \cos(\omega_d \cdot t + \phi) \quad (5)$$

Where the damping ratio is

$$\zeta = \frac{\beta \cdot T_d}{2} = \frac{\pi \cdot \beta}{\omega_d} \quad (6)$$

The damping ratio expresses the ratio between two successive peaks of the oscillation. With reference to Figure A, the ratio between the two successive peaks, A1 and A2, occurring at  $t_1$  and  $t_1 + T_d/2$  respectively (one half period apart), is, by using (5):

$$\frac{A2}{A1} = \frac{e^{-2\cdot\zeta\frac{t_1+T_d/2}{T_d}}}{e^{-2\cdot\zeta\frac{t_1}{T_d}}} = e^{-\zeta} \quad (7)$$

The damping ratio shows the speed with which the response decays. For damping ratio equal 1, the amplitude of the oscillation decays by 63.22% in a half cycle (that is between two successive peaks). Thus, the response decays with the same frequency as the frequency of the oscillations. If the damping ratio is 2, the response decays with twice the frequency of the oscillations, and so on. An equivalent way of looking at the damping ratio is this: the inverse of the damping ratio is the number of additional successive peaks it takes for the amplitude of the oscillation to decay by 63.22% from an initial peak. Thus, a response with a damping ratio of 0.5 will decay to 63.22% of an initial amplitude in 2 successive peaks (one full cycle of the oscillation).

### *1.3 The Step Response of an Under damped Second Order Circuit.*

The step response of an under damped second order circuit is shown in Figure B. The same measurements can be performed on this wave form as in the natural response. Three additional parameters of the response are of interest. These are explained in the following with reference to Figure B.

(a) *The response time,  $T_r$* , is the time lapsed from the onset of the excitation to the moment the response becomes equal, for the first time, to its steady state value (1 in the figure).

(b) *The overshoot time,  $T_{o,s}$* , is the time the response reaches its first peak.

(c) *Overshoot error,  $Y_{o,s}$* , is the amount the response rises in excess of its steady state value.

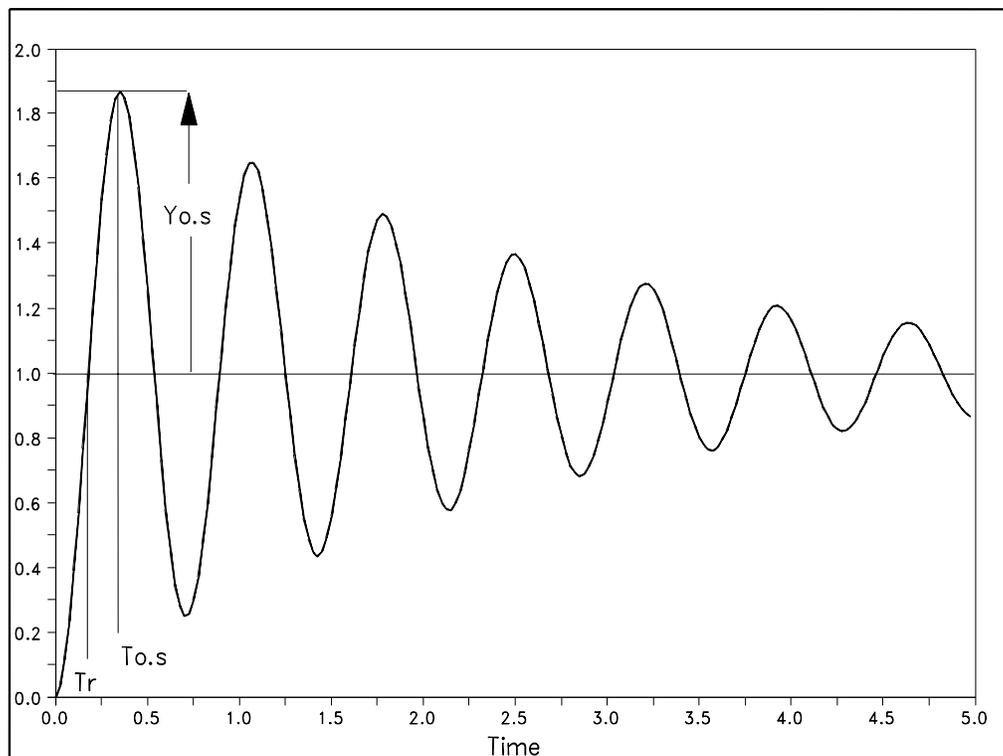


Fig. B. The step response of an under damped circuit.

## II. INSTRUMENTATION

Function generator, Tektronix CFG 250. Oscilloscope, Tektronix 2205-20 MHz.

## III. PROCEDURE

### A. The Series RLC.

1. (a) Construct the series RLC circuit in Figure 1. Observe the capacitor voltage.
- (b) Set the signal of the source to the square pulse mode. Adjust the frequency of the signal to 10 times below the resonant frequency of the circuit.
- (c) Adjust R to 10 k $\Omega$ . What type of response do we obtain for the voltage?
- (d) Sketch on a millimeter paper the positive half cycle of the voltage wave form.
- (e) Adjust R to obtain the critical response. Record R.
- (f) Sketch on the same millimeter paper the positive half cycle of the voltage response.
- (g) Adjust R to 500  $\Omega$ . What type of response do we obtain for the voltage?
- (h) Sketch on the same millimeter paper the positive half cycle of the current response.
- (i) Measure the frequency, damping ratio, response time, and overshoot error.
- (j) Simulate the circuit of Figure 1 in PSpice. Use the same values of R as above to obtain the three types of response. Compare the simulated wave forms with the oscilloscope waveforms.

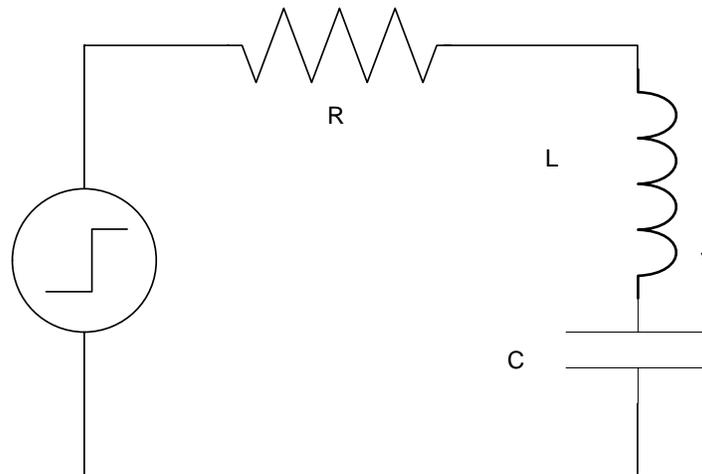


Fig. 1. The series RLC circuit.  $C=0.01 \mu\text{F}$ ,  $L=10 \text{ mH}$ .

### B. The parallel RLC.

2. (a) Simulate the circuit of Figure 2 in PSpice. Find the value of R resulting a critical response.
- (b) Set R to 2 k $\Omega$ . Obtain the voltage response.

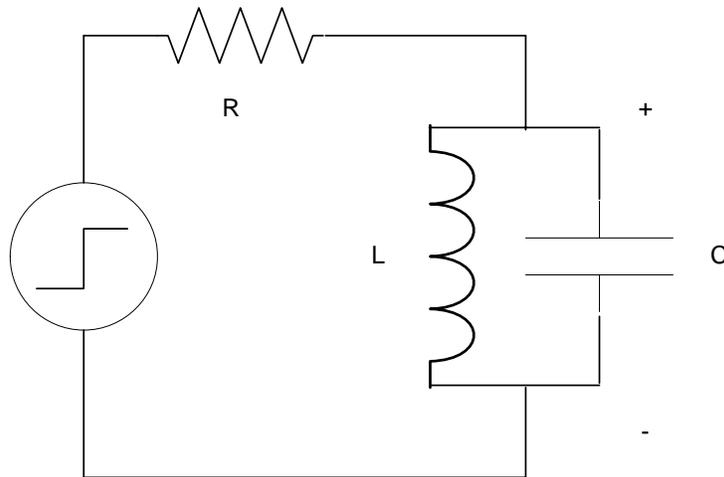


Fig. 2. The parallel RLC circuit.  $C=0.01 \mu\text{F}$ ,  $L=10 \text{ mH}$ .

## REPORT:

### A. Theoretical Development.

1. In theory discuss the natural response of an RLC circuit.

### B. Recording of the Three Types of Response of the Series RLC.

2. Are the three values of  $R$  in Procedures 1c, 1e, and 1g consistent with the theory?

3. Are the experimental responses consistent with those predicted by the theory? Explain by comparing the wave form parameters with their theoretical values in each case.

### C. Under damped Response of the Series RLC Circuit.

4. Compare the oscilloscope wave form obtained in Procedure 1h with the PSpice simulation for the same circuit.

### D. Simulation of the Parallel RLC circuit.

5. (a) Discuss the simulation results from Procedure 2a.

(b) From the simulation in Procedure 2b, calculate the maximum energy of the inductor and capacitor. Plot both as function of time. Plot their summation as function of time. Discuss the results.

# EXPERIMENT 7:

## SINUSOIDAL STEADY STATE

This experiment demonstrates the properties of ac networks. The concept of impedance is discussed. Phasors are demonstrated through oscillograms.

### I. BACKGROUND

#### *I.1 AC Measurements. The RMS Value.*

AC instruments measure the rms (root mean square) value of the ac voltage and current. For a periodic wave form of period T, the rms value is given by (1):

$$X_{rms} = \sqrt{\frac{1}{T} \int_0^T x^2(t) \cdot dt} \quad (1)$$

The rms of a sinusoidal quantity is given by (2):

$$X_{rms} = \frac{X_m}{\sqrt{2}} \quad (2)$$

Where,  $X_m$  is the amplitude of the sinusoidal quantity. The distinction between dc and ac instrumentation is important. In circuits operating in sinusoidal steady state, a dc instrument will consistently indicate zero voltage and current.

The rms value of a current or voltage is, also, referred to as its effective value. This shows an equivalency between ac and dc quantities in the following sense:

An ac current (voltage) produces the same power dissipation on a resistor as a dc current (voltage), which has an average (dc) value equal to the rms (effective) value of the ac current (voltage). Thus, an ac current of amplitude 1.41 A is equivalent, in the previous sense, with a dc current of average value equal 1 A .

In practice, different notation is used to distinguish rms and average measurements. Thus, 10 V AC means that the rms value of the voltage measured is 10 V. While, 15 V DC means that the average value of the voltage measured is 15 V. Additional notation is available for amplitude description of ac wave forms. Thus, 20 mA p-p (red 20 mA peak-to-peak) means that the current varies by 20 mA between two successive peaks. For a sinusoidal current this means that its amplitude is 10 mA.

#### *I.2 Measurements of Phasors and Impedance in the Sinusoidal Steady State.*

The currents and voltages of a circuit that operates in sinusoidal steady state are represented by phasors. With reference to Figure A, the impedance expresses the relation between the phasor of the terminal voltage and the phasor of the terminal current of a two-terminal RLC combination.

Conversely, every passive element or two-terminal combination of passive elements possesses an impedance, which expresses the Ohm's law as follows:

$$\bar{V} = \bar{Z} \cdot \bar{I} \tag{3}$$

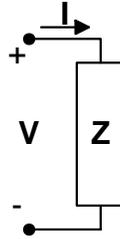


Fig. A. Phasors and impedance in sinusoidal steady state.

The magnitude of the phasor of a quantity can be measured by measuring the rms value of the quantity. The phase of the phasor can be measured using the oscilloscope. Figure B shows the oscillogram of the current and voltage at the terminals of an RLC arrangement. It is important to indicate or note the scale of the oscillogram. Notice that the oscilloscope probes measure voltage (unless a current probe is available). Appropriate scaling is required to obtain the correspondence between the oscilloscope vertical divisions and the measured current.

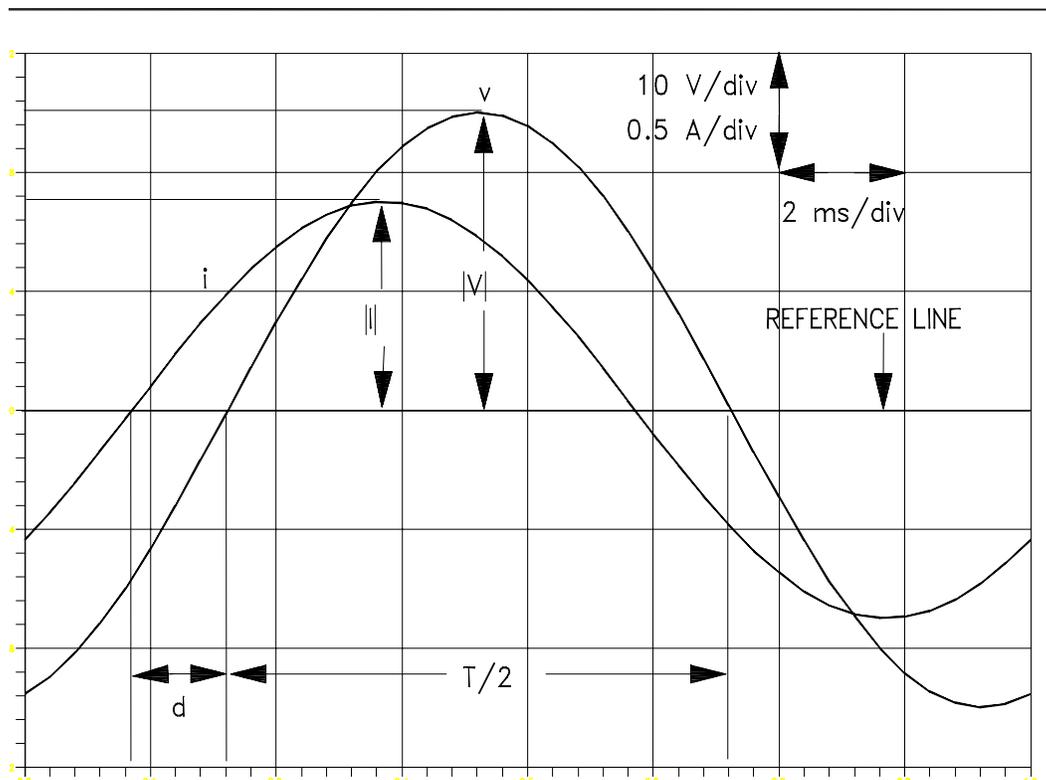


Fig. B. The oscillogram of voltage and current.

For the example of Figure B, the period of the oscillations is, by reading the scale,  $(T/2) = 4$  divisions  $\times 2\text{ms/division} = 8\text{ms}$ . Therefore, the period is 16 ms. This corresponds to a frequency of 62.5 Hz.

The amplitude of the voltage oscillation on the oscilloscope is, by reading the scale from the reference line, 2.5 divisions. Therefore, the amplitude of the voltage is  $2.5 \text{ div} \times 10\text{V/div} = 25 \text{ V}$ . The rms value of the voltage is, therefore, 17.68 V.

Likewise, by reading the scale, the current oscillation on the oscilloscope has an amplitude of 1.8 divisions and the actual current has an amplitude of 0.9 A. Its rms value is 0.636 A.

The zero crossing displacement between voltage and current is  $d = 0.8\text{div} \times 2\text{ms/div} = 1.6 \text{ ms}$ . This corresponds to a phase difference of  $\Delta\phi = 360^\circ \times 62.5 \text{ Hz} \times 1.6 \text{ ms} = 36^\circ$ , with the current zero crossing leading the voltage zero crossing.

From this measurements, the phasors of the voltage and current and the impedance of the RLC arrangement can be calculated. Assuming the voltage as reference:

$$\begin{aligned}\bar{V} &= 25 \angle 0^\circ \text{ V} \\ \bar{I} &= 0.9 \angle -36^\circ \text{ A} \\ |\bar{Z}| &= \frac{|\bar{V}|}{|\bar{I}|} = \frac{25}{0.9} = 27.78 \Omega \\ \angle \bar{Z} &= \angle \bar{V} - \angle \bar{I} = 0^\circ - 36^\circ = -36^\circ\end{aligned}\tag{4}$$

Note that since the current is leading the voltage, the current's phase angle with respect to the voltage is positive. The impedance angle is the angle between current and voltage. Hence, the calculation of (4) results in negative impedance angle. This means the RLC arrangement is capacitive.

Its equivalent resistance is:  $R = |Z| \cos(-\Delta\phi) = 22.47 \Omega$ .

Its equivalent reactance is:  $X = |Z| \sin(-\Delta\phi) = -16.33 \Omega$ , or  $16.33 \Omega$  Capacitive.

## II. INSTRUMENTATION

Function generator, Tektronix CFG 250. Oscilloscope, Tektronix 2205-20 MHz.

## III. PROCEDURE:

### A. Measurements of Capacitive Impedance.

1. Construct the circuit of Figure 1. Set the signal generator to the sinusoidal mode. Set the frequency to 60 Hz and the amplitude to 10 V.
2. Display on the oscilloscope the current and voltage of the capacitor.

3. Vary R from 0 to 1 k $\Omega$  in increments of 100  $\Omega$ . For each value of R measure the amplitude of the voltage and current. With the voltage as reference, also measure the phase of the current. Use Table 1.

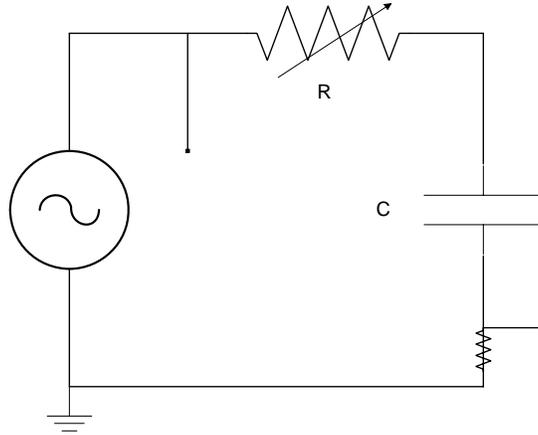


Fig. 1. The experimental RC circuit. C= 10  $\mu$ F.

Table 1.  
Impedance measurements in the RC circuit.

<b>R</b>										
<b> V </b>										
<b> I </b>										
<b><math>\phi</math></b>										

*B. Measurement of Inductive Impedance.*

4. (a) Construct the circuit of Figure 2. Set the source to the same values as in the previous circuit.  
 (b) Vary R from 0 to 1 k $\Omega$ . For each value of R measure the amplitude of the voltage, the amplitude of the current, and the phase of the current to the voltage. Use Table 2.

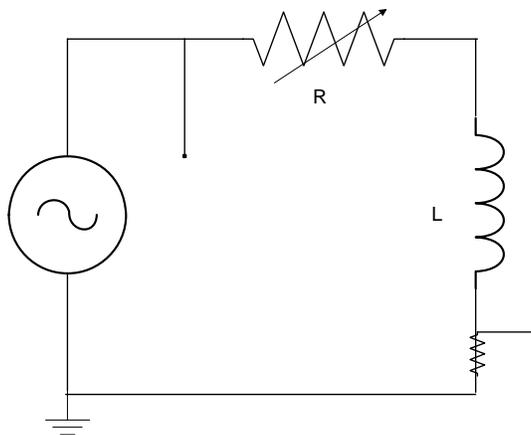


Fig. 2. The experimental RL circuit. L=100 mH.

Table 2.  
Impedance measurements in the RL circuit.

<b>R</b>										
<b> V </b>										
<b> I </b>										
<b>∠I</b>										

*C. Measurement of the Impedance of an RLC Arrangement.*

5. Construct the circuit of Figure 3. Maintain the settings of the source. Measure the magnitude of the supply voltage. Measure the magnitude and phase (with respect to the supply voltage) of the inductor current, the capacitor current and the supply current.

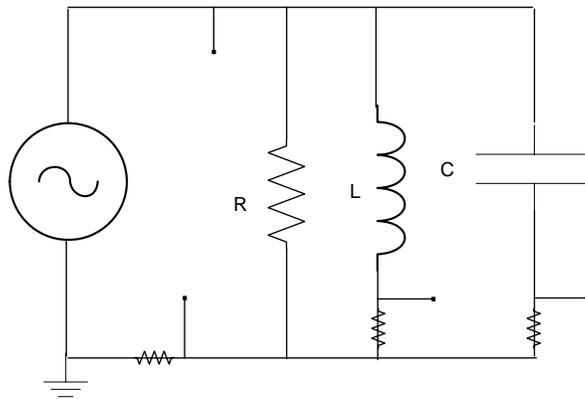


Fig. 3. The experimental parallel RLC circuit.  $R= 100 \Omega$ ,  $L= 100 \text{ mH}$ ,  $C= 10 \mu\text{F}$ .

*D. Compensation of Source Reactance.*

6. (a) Construct the circuit of Figure 4 with the capacitor disconnected. Maintain the source at the settings of the previous procedure. Measure the phasors of the supply voltage and current and the phasor of the voltage across the resistor. Use the voltage across the resistor as reference.  
(b) Connect the capacitor. Repeat the previous measurements. Also, measure the phasor of the capacitor current.

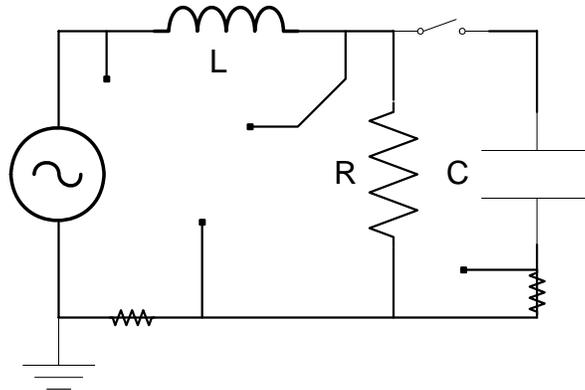


Fig. 4. The experimental circuit to observe compensation.  $R= 100 \Omega$ ,  $L=100 \text{ mH}$ ,  $C=10 \mu\text{F}$ .

*E. Measurement of Parallel Resonance.*

7. Construct the circuit of Figure 5. Set the supply to 1 V in the sinusoidal mode.
  8. Vary slowly the frequency of the source until the amplitude of the supply current becomes minimum. For this point record the phasors of the supply voltage (reference), supply current, inductor current, and capacitor current.
  9. Take the same measurements as in 8 for five frequencies below the frequency you recorded in 8.
  10. Repeat 9 for five frequencies above the frequency in 8.
- For your measurements in 8, 9 and 10, use Table 3.

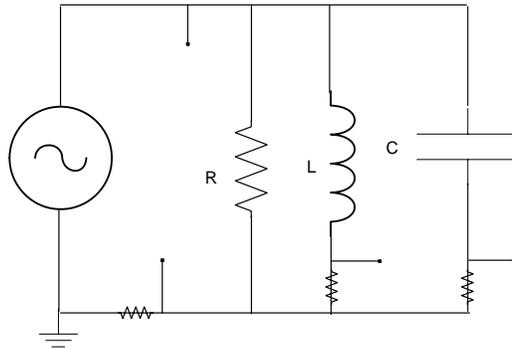


Fig. 5. The experimental setting to record resonance.  $R= 100 \Omega$ ,  $L=100 \text{ mH}$ ,  $C= 10 \mu\text{F}$ .

Table 3.  
Measurements of resonance.

<b>f</b>											
<b> V </b>											
<b> IL </b>											
<b>∅IL</b>											
<b> IC </b>											
<b>∅IC</b>											
<b> Is </b>											
<b>∅Is</b>											

**IV. REPORT:**

*A. Theoretical Development.*

1. In theory explain the meaning of impedance. Explain the basic power calculations in ac circuits.

*B. Measurement of RC and RL Impedance's.*

2. For each of the circuits in Figures 1 and 2 and using Tables 1 and 2, respectively, perform the following calculations.
  - (a) Calculate the impedance of the circuit using the measurements.

- (b) Calculate the theoretical value of the impedance of the circuit for the values of R in Tables 1 and 2. Tabulate magnitude and angle.
- (c) Plot vs R on millimeter paper the impedance magnitude from a and b. Use different axes on the same paper and plot vs R the impedance angle from a and b. Use one graph paper for Circuit 1 and another graph paper for Circuit 2. Compare the experimental with the theoretical graphs.
- (d) How does the behavior of each circuit change as R increases?

*C. Phasor Diagrams. Verification of KVL and KCL.*

- 3. (a) Draw on millimeter paper and on scale the phasor diagram of Circuit 3 using your measurements. Compare with theoretical values.
- (b) Analyze the supply current to its active and reactive components using the supply voltage as reference. Compare with your measurements. What is the real and reactive power of the supply? What is the reactive power of the inductor and capacitor?

*D. Phasor Diagrams. Application to Reactance Compensation.*

- 4. Draw the phasor diagram of Circuit 4 with and without the capacitor. Use your measurements. Explain with appropriate theory the difference of these diagrams. What is the active and reactive power of the source?

*E. Parallel Resonance.*

- 5. (a) For the circuit of Figure 5, draw on separate graphs the magnitude and angle of the circuit impedance vs the frequency of the source. Use your measurements. Mark the resonance frequency on the graph. Compare with the theoretical value. What is the magnitude and angle of the circuit impedance at resonance? In what frequency range is the impedance inductive? In what frequency range is the impedance capacitive?
- (b) At the resonance frequency, draw the phasor diagram of the circuit. Discuss it. What is the reactive power of the capacitor and inductor? Compare the supply current with the resistor current. Compare the inductor and capacitor currents.
- (c) Draw the locus of the phasor of the supply current for different frequencies. In what frequency range is the current leading the supply voltage? In what frequency range is the current lagging the supply voltage?

# INTRODUCTION TO THE USE OF PSPICE

## 1. INTRODUCTION

PSPICE is a PC version of SPICE which runs on personal computers and offers many improvements over SPICE. SPICE is an acronym for Simulation Program with Integrated Circuit Emphasis. PSPICE is widely used in industry and universities. Though developed for IC design, it can be a useful tool for both the student and practicing circuit designer with many non-IC circuits.

Versions of PSPICE are available for use in the Electrical Engineering PC lab (E-136) across from the SUN lab in the E-building and also in **E-237** across from the EE department office. The program goes by the names Microsim Eva1 6.3, Microsim 7.1, and Microsim Release 8. To run PSPICE click the "Schematics" section under those names. Also you can download latest version from <http://www.orcad.com~comvanv/move.asp>

<http://math.uww.edu/~harrisb/courses/cs412/vspice/PSpice8.0/>.

## 2. PRODECURE

To begin building your circuit, go to "Draw" on the menu bar and select "Get new part". Select the parts you need from the menu that will appear. Table 1 shows some of parts list. All parts from the list are "generic", meaning that they all have the same attributes (all resistors are 1 k Ohms and all sources are 0V). To change the generic elements into useful ones, click on an element until it turns red then double click on it. A list of attributes for that element should appear. Change the attributes you need to change and close the attribute box. Arrange your circuit elements by clicking and dragging and using the "Flip" and "Rotate" commands in the "Edit" menu. Connect parts using "Wire" from the "Draw" menu.

While building your circuit, some housekeeping issues may need to be dealt with. Value labels on the elements can really make your circuit look like a train wreck, so move them but never erase them. For later use, label the nodes by clicking on a wire attached to the node until it turns red then double click on it. The program will then allow you to name the wire. These names will then appear in the output file as the names of your nodes, so you can easily find your nodal voltage.

They Call It	It Is
R	Resistor
L	Inductor
C	Capacitor
VDC	DC Voltage Source
IDC	DC Current Source
VAC	AC Voltage Source
IAC	AC Current Source
Gnd_Earth	Earth Ground
E_Poly	Voltage Controlled Voltage Source
F_Poly	Current Controlled Current Source
G_Poly	Voltage Controlled Current Source
H_Poly	Current Controlled Voltage Source

TABLE 1. PARTS LIST

### 3. Simulations

#### 1) Independent Sources

To begin the simulation, make sure all unused parts are cleared from the workspace and select "Create Netlist" from the "Analysis" menu. Upon successful completion of the netlist, select "Setup" from the "Analysis" menu and check the box marked "Bias Point Detail" and exit that window. Then select "**Run** PSPICE" or "Simulate" from the analysis menu. If run was successful, select "Examine Output" from the "Analysis" menu.

**NOTE:** Make sure to include a ground in your schematic. If you get errors saying something in your circuit is "floating", it is either unused parts still on the workspace or a lack of a ground.

#### ASSIGNMENT 1

#### 2) Dependent Sources

Dependent sources in PSPICE are handled with the E-Poly, F-Poly, G - Poly, and H - Poly. E-Poly is a voltage controlled voltage source. F-Poly is a current controlled current source. G-Poly is a voltage controlled current source. H-Poly is a current controlled current source. To use any of these sources, the controlling variable must be "sampled" by the primary side of the sources. The primary side is on the left hand side of each source. This "sampling" is achieved by connecting the primary side of the source directly to the controlling variable. This means that if the source is voltage controlled, the primary side of the

source is connected in parallel with the controlling variable, like a voltmeter. If the source is current controlled, the controlling current should flow through the primary side of the source, similar to an ammeter.

In this attributes menu, each source has a changeable coefficient. Set this coefficient to reflect the coefficient of your dependent source. For example, if the source magnitude was supposed to be 12V, then the coefficient should be changed to 12.

## ASSIGNMENT 2

### 3) Probe and Transient Analysis

Probe is an attachment to PSPICE which allows the user to graph time dependent and frequency dependent outputs. To use this feature, a few simple adjustments need to be made to your procedure for simulating circuits. First, build your circuit as you normally would. Prior to netlisting and simulating, go to the "Analysis" menu and choose "Setup". A variety of simulation parameters will be shown. The one we have been using so far is called "Bias Point Detail". Make sure the box next to this option is checked. The new feature we will be concerned with is "Transient Analysis". Click on the button for this option and set the "Final Time" and "Print Step" to approximate values. For best results, use a "Final Time" and "Print Step" that plots between 100 and 1000 points. Exit from these windows and return to your circuit. While not absolutely necessary, the next step could save some time. Go to the "Markers" menu and select appropriate markers for what you would like to see graphed and place them into your circuit. Your selections are "Voltage/Level", "Voltage Differential" and "Current into pin". These markers will tell "Probe" to graph these variables automatically without any further input from you. With the markers in place, run the simulation. In Probe, you can add traces by going to the "Trace" **menu** and selecting "**Add**". However, if repeated analyses are required, using markers will save a considerable amount of time and frustration.

a. VPULSE: **A** single pulse or pulse train, with rise and fall time that can be specified. This can be used to produce pulse, square wave, triangle, and sawtooth sources.

V1 = initial value

V2 = pulsed value

TD = time delay

TR = rise time

**TF** = fall time

PW = pulse width

PER = period

## ASSIGNMENT 2

b. VPWL: **A** piece-wise linear description of a source which can be used to approximate nearly any wave form.

Tn = nth time value

Vn = nth voltage value

## ASSIGNMENT 3

c. VSIN: **A** sinusoid in which the start can occur after time zero. It may be exponentially damped if desired

Voff = DC offset of sine wave

Vamp1 = Amplitude of output sine wave

Freq = Frequency of sine wave in Hertz

**TD** = Time Delay, set this to zero

DF = Damping Factor, .set this to zero

Phase = Phase angle of the waveform in degrees, zero is referenced to a sine wave

# INTRODUCTION TO MATLAB

Matlab is powerful Software with a variety of applications. It can give solutions to many numerical problems in a far less time than other language programs like C++ or FORTRAN. With matlab it is possible to plot 2-D and 3-D graphs, solve matrices, integrals, differential equations, animation plots, etc.

This text is an introduction on how to use matlab to plot 2-D graphs, and solve matrices in order to solve problems related to ECE 235. The student is advised to read matlab handbooks for further applications.

## 1. BASIC COMMANDS

To get to matlab either double click the matlab icon on the desktop or go to programs and select matlab on the programs menu. A window will appear on the screen. There are two different ways of writing a matlab program. You can write your program directly on the screen or open a file where to write your program. For graph plots and systems of equations solutions you may not need to open a file because the program takes only a few lines. However, you may need to open and write a file if you want to keep your program for further use, or if your program is relatively long.

### 1.1 Writing Your Program on the main window

To write a program on the main window, just write the instructions. Unlike C++ programming, when using matlab each instruction may or may not be followed by a semi column. When not using a semi column each instruction will be executed after you press the <ENTER> key. This may not be good if the instruction executes many steps. It is therefore better to often end your instructions by a semi column. After the last instruction just press <ENTER> and your program will run automatically.

### 1.2. Writing your program in a file

Open a new file and write your instructions. Save your file as **filename.mat** and go back to the main window. At the prompt just write the name of your file (without the **.mat** part) and the program will run automatically. The solution will appear on the main window or a separate window will appear on the screen with the solution.

### 1.3. Arithmetic operations

<i>Operation</i>	<i>Symbol</i>	<i>Example</i>
addition, $a + b$	+	$5 + 3$
subtraction, $a - b$	-	$23 - 12$
multiplication, $a \times b$	*	$3.14 * 0.85$
division, $a : b$	/ or \	$56/8 = 8 \setminus 56$
power, $a^b$	^	$5^2$

### 1.4. Some Elementary Math functions.

cos(x)	cosine
sin(x)	sine
tan(x)	tangent
exp(x)	Exponential: $e^x$
log(x)	Natural logarithm
log10(x)	Common (base 10) logarithm
real(x)	complex real part
imag(x)	complex imaginary part

## 2. SOME EXAMPLES

### 2.1. Graph plotting

```
>> x = linspace(0,2*pi, 30); %First create 30points between 0 and 2*pi
>>y=sin(x); %find the sine of the points in x
>>plot(x,y); % generate the plot
>>xlabel('angle, x'); ylabel('sin(x)'); % x and y labels
>>title('Graph of y=sin(x)'); % title
```

If plotting many functions on the same graph, you may need to identify each plot. You can do that by adding an instruction at the location:

```
>>x = linspace(0,2*pi, 30);
>>y=sin(x);
>>z=cos(x);
>>plot(x,y);
.>plot(x,z);
>>xlabel(' angle, x') ;
>>text(2.5,0.7, 'sin(x)'); % identifies the sine curve at the location (2.5, 0.7)
>>text(2, 0,'cos(x)'); % identifies the cosine curve at the location (2,0)
```

## 3. SOLVING A SYSTEM OF LINEAR EQUATIONS

Let us suppose we want to solve the following system of linear equations:

$$\begin{aligned}x_1 + 2x_2 + 3x_3 &= 366 \\4x_1 + 5x_2 + 6x_3 &= 804 \\7x_1 + 8x_2 &= 351\end{aligned}$$

We can write the above system in matrix form:

$$\begin{bmatrix} 1 & 2 & 3 \\ 4 & 5 & 6 \\ 7 & 8 & 0 \end{bmatrix} \begin{bmatrix} x_1 \\ x_2 \\ x_3 \end{bmatrix} = \begin{bmatrix} 366 \\ 804 \\ 351 \end{bmatrix}$$

We then write matlab commands:

```
>>A=[1 2 3; 4 5 6; 7 8 0]; % writes the coefficient matrix
>>B=[366; 804;351];
```

Note: if not writing the semi **column** the following will appear on the **main window**:

```
B=  
    366  
    804  
    351
```

To find the values of  $x_1$ ,  $x_2$ ,  $x_3$  write the following command:

```
x = inv(A)*B;
```

The values of  $x_1$ ,  $x_2$ ,  $x_3$  will then be displayed

```
x=  
25.0000  
22.0000  
,99.0000
```

To run a program located in a given path such as a:/filename.mat: just write;

```
chdir a:filename. mat;
```

and the program will run automatically.

To learn about more commands please read any matlab manual.