# BASICS OF ELECTRICAL CIRCUITS LABORATORY

# EXPERIMENT SHEET

JANUARY 2013

# EXPERIMENT 1: USE OF OSCILLOSCOPE, MULTIMETER AND SIGNAL GENERATOR

## Objective

In this experiment, the subject of measuring the physical quantities in an electrical circuit, like current, voltage and resistance, will be examined. Initially, measuring instruments for these physical quantities will be presented, then, a simple electrical circuit will be set up for the experiment and voltage-current values on this circuit will be measured. It will be checked that whether the measured values satisfy Kirchhoff's current and voltage laws or not. In addition, how to utilize the oscilloscope, which is a device of great importance for measuring electrical signal quantities in the time domain, for measuring the amplitude or frequency of voltage-current signals will be demonstrated.

## Foreknowledge

The basic quantities in an electrical circuit are the voltage and current values among each circuit element (branch). To measure these quantities, we use devices which are actually other electrical circuits themselves. In other words, if our desire is to measure a current value on an electrical circuit, we need to make that specific current flow through another circuit (the measuring instrument) which is capable of telling us the current value flowing through itself. This means we need to connect the measuring instrument in "series" with the circuit. On the other hand, if our desire is to measure a voltage value on an electrical circuit, we need to make that specific voltage applied to another circuit (the measuring instrument) which is capable of telling us the voltage value among itself. This, in turn, means we need to connect the measuring instrument in "parallel" with the circuit. These measuring instruments which will be presented below are called "multimeter" and "oscilloscope". Multimeter is a measuring device which operates as an ammeter or voltmeter with respect to the position of the switch on it. It can also measure the resistance values of circuit values (That is why it is sometimes called **A**(mpere)**V**(olt)**O**(hm)**METER**). Oscilloscope on the other hand, measures only voltage quantities; however, it shows a certain voltage value on a circuit as a function of time.

Since each measuring device has its own internal resistance, when we try to measure a current (voltage) quantity on a circuit element, this internal resistance is connected to the element in series (parallel), and changes the electrical quantities on the circuit. Owing to this fact generally the aim in the measuring problem is to make the original system, on which the quantities desired to be measured are found, affected by the measuring device as little as possible. We can check the amount of energy siphoned by the measuring device from the actual system to understand this fact. Energy consumed by an element is given by the expression $\int_0^\infty v(t)i(t)dt$. Therefore, if we want the measuring device to siphon as little energy as possible from the circuit, the voltage or current value across it must be as small as possible. For the voltmeter, we prefer to make the current flowing through it smaller since the other way would change the quantity desired to be measured (in this case voltage). For this, in accordance with the relation $i = \frac{v}{R}$ the internal resistance of the voltmeter is high. On the contrary, for the ammeter, in accordance with the relation $v = iR$ the internal resistance is low for the purpose of decreasing the voltage value among the device. It is for these reasons that the internal resistances related to the current inputs are very low and the internal resistances related to the voltage inputs are very high in a multimeter.

**Multimeter**

Among the process of experiments, we will use two types of multimeters called “analog” and “digital” respectively. In Figure 1.1 an image of an analog multimeter is given. With the switches on the right and left side of the multimeter, the properties of the signal to be


**Figure 1.1**

measured are specified and the scale is adjusted. For example, if a DC current signal is to be measured, the left switch must be adjusted to “DC” and the right switch must be adjusted to one of the seven levels seen on the right side to make the indicator show a feasible value on the display. Thus, the scale is adjusted. If for example the value to be measured is around 3 mA, as long as the switch on the right side is adjusted to 5 mA – DC, indicator will show a value on the display. The DC current to be measured is read from the 0 - 50 DC display by scaling the value 50, where the number 50 is proportionate to 5 mA. As an example, if the indicator points the number 27 on the display, since the switch on the right side is adjusted to 5 mA, it should be read as $\frac{5}{50}27 = 2.7$ (2.7 mA). (The measured values for AC quantities are the root mean square (RMS/effective) values).

An image of a digital multimeter like we will use is given in Figure 1.2. As it is seen on the figure, again we need to specify with a switch what we are going to measure. Then if we intend to measure a voltage or a resistance quantity we need to plug one of the probes (probe is the pointed wire that makes the contact between the measuring device and the circuit) to the measuring device's COM input, and the other one to the V input. For measuring current quantities we need to plug one of the probes to the COM input again and the other one to the A input. Current input A is limited to 10 amperes as safety fuse for the device. Throughout the experiments, we should not let currents whose values are higher than 10 amperes flow through this input.


**Figure 1.2**

**Testing Kirchhoff's current and voltage laws**

As you should know, there are two basic laws in circuit theory. According to Kirchhoff's current law (KCL), the sum of the outflowing currents from a specified closed surface on an electrical circuit is equal to the sum of inflowing currents. An equivalent statement can be given as "The algebraic sum of currents meeting at a node of an electrical circuit is zero." The other law called Kirchhoff's voltage law (KVL) states that the voltage among a circuit element is equal to the difference between two absolute voltage values of its two nodes. Here, the absolute voltages are specified according to a previously determined reference node in the circuit. An equivalent statement is “The sum of the voltage values of each branch in a loop of the circuit is equal to zero.”

**Experimental work**

Set up the circuit shown in Figure 1.3 on your testing board. Get the 5 V DC supply voltage from the source on the board. To test the validity of Kirchhoff's voltage law, measure the voltage quantities $V_1 - V_2$ and $V_2 - V_3$ with the multimeter and show that KVL is valid for the loop $L_1$.

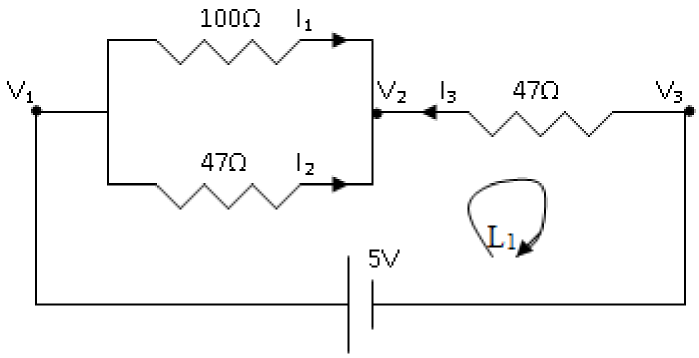



**Figure 1.3**

To test the validity of Kirchhoff's current law, measure the current quantities $I_1$, $I_2$ and $I_3$ with the multimeter. Show that KCL is valid for the node $V_2$.

**Oscilloscope**

The most important part of an oscilloscope is the cathode ray tube (CRT), similar to the television tubes, which can be seen in Figure 1.4 with its peripheral elements. CRT is a vacuum tube made of glass. The inner side of its surface is covered with a fluorescent substance.

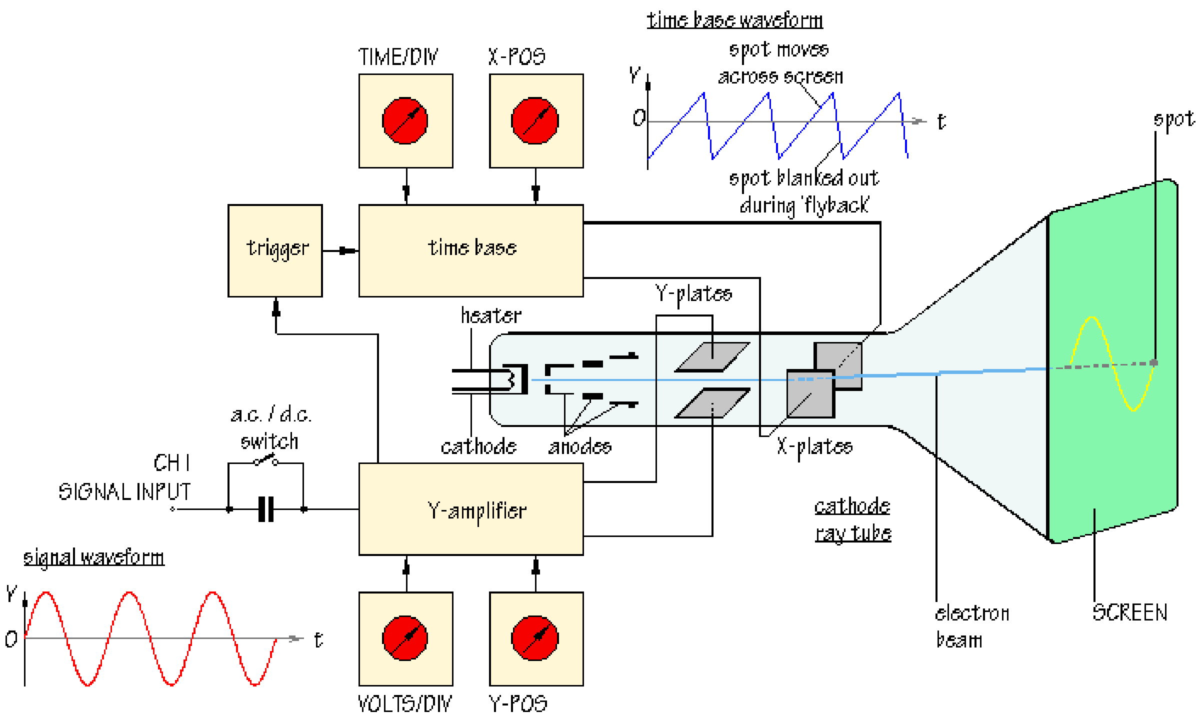



**Figure 1.4 Electron gun and its connections with its peripheral elements in an oscilloscope**

Main parts of a CRT can be listed as:

- Electron gun
- Vertical (Y) and Horizontal (X) diverting plaques
- Screen

Electron gun creates the electrons and the ability to control them. The cathode of the electron gun has high temperature and emits electrons. By the help of grid voltages, the light intensity of the electron beam appearing on the screen can be done. This can be achieved by applying negative voltage to the grid. The anodes have positive voltages. Owing to this, the focus adjustment and acceleration of the electron beam can be made with the anode. The electrons emitted from the electron gun which are focused, accelerated and whose intensities are adjusted reaches to the inner side of the screen by passing through vertical and horizontal diverting plaques. Because of the fluorescent substance on the inner side of the screen, the electron beam hitting on the inner screen appears as a spot. When there is no voltage applied to the vertical and horizontal plaques the spot appears on the middle of the screen. By applying voltage values between $20 - 50$ V to these plaques the spot can be moved to any part of the screen.

**Vertical and horizontal amplifiers**

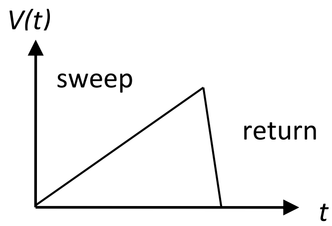
If the quantities intended to be measured by the oscilloscope are very low, only a very small image can be seen on the screen. For the signal to be measured to be seen on the screen in an appropriate size, the signal is amplified first, and then it is applied to the plaques. Thus, signals with low amplitudes can also be measured, in other words, the sensitivity or the resolution of the oscilloscope is increased.

In an oscilloscope, the amplifying factors of the vertical and horizontal plaques can be adjusted with the switches VOLTS/DIV and TIME/DIV respectively. To make no mistake on the voltage and time measurements, the switches related to them should be in calibration position.

**Horizontal sweep circuit**

An important part of the oscilloscope is the horizontal sweep circuit. This part is an oscillator which generates a “saw tooth” signal as a function of time. When this signal is applied to the horizontal diverting plaques, and if there is no signal applied to the vertical plaques, the spot on the screen appears as a straight horizontal line (the time axis) in the middle. When there is a signal which is a function of time applied to the vertical plaques and if there is no signal applied to the horizontal plaques, a vertical line appears on the screen. When the saw tooth signal is applied to the horizontal plaques and a periodic signal, like a sinusoidal, triangle or square shaped signal, is applied to the vertical plaques the signal applied to the vertical plaques appears on the screen. When the signals applied both to the horizontal and vertical plaques are synchronized, the image seen on the screen appears as if it is stationery. Otherwise the image on the screen appears sliding towards left or right constantly.

**Figure 1.5 Horizontal sweep signal**




The voltage signal generated by the horizontal sweep circuit changes with time in a period as seen on Figure 1.5.

**Power supply**

There are two types of DC voltage sources which makes the oscilloscope operate. One of these supplies a voltage greater than 10 kV and it is used for the CRT to operate. The other one is a low voltage source and it is used for oscilloscope amplifiers and sweep voltage circuits.

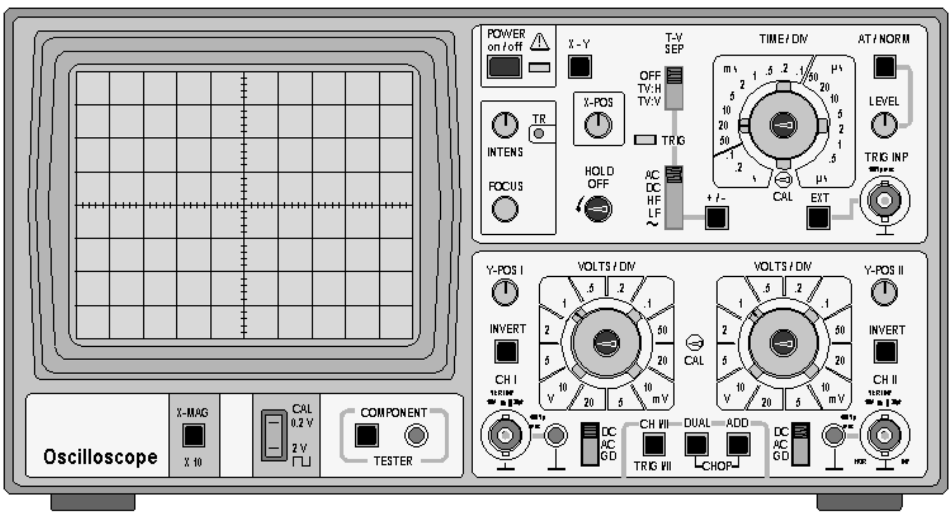



**Figure 1.6 Front panel of an oscilloscope**

**How to use the oscilloscope**

1. When the oscilloscope's ON-OFF switch is in OFF position, FOCUS and INTENSITY indicators should be pointing low levels.
2. The indicators for adjusting vertical and horizontal positions should be pointing a level around the middle.
3. The EXT button, which is used for synchronization with another signal, should be turned off unless synchronization with a different external signal is needed.
4. After taking care of the above, the power lead of the oscilloscope can be plugged in.
5. Bring the ON-OFF switch to ON position.
6. Wait until the spot appears on the screen. Adjust the light intensity on the screen with INTENSITY indicator until the image is clearly visible. If there is no line seen on the screen try to find the line by adjusting X-Y POSITION indicators. It is desirable to keep the INTENSITY indicator in a low level since when the light is too bright, it may result in the fluorescent substance to be damaged.
7. By adjusting the FOCUS indicator, make the line image more clearly seen.
8. By using X-Y POSITION indicators center the line.
9. By switching off the AT/NORM button, acquire the triggering to be done automatically.
10. Adjust the TIME/DIV indicator to a level lesser than or equal to 10 ms.

After all these steps, the oscilloscope is ready to be used for measurements.

**Measuring the voltage**

The oscilloscope is like a voltmeter. When there is the sweep voltage applied, the shape of the signal, i.e. a sinusoidal signal, applied to the vertical plaques can be seen on the screen. By appropriately interpreting the grids on the screen, the peak to peak amplitude value of the signal can be read. By this, the RMS value of the signal can also be evaluated.

**Measuring the current by using a test resistor**

The oscilloscopes are usually used for measuring the voltage. However, the current can also be measured indirectly. One way of measuring the current with an oscilloscope is to use a linear resistor whose resistance value is known. By measuring the voltage between its two nodes and making use of Ohm's Law, the current flowing through it can easily be calculated. A $1\ \Omega$ valued resistor which does not have parasitic inductance properties can be chosen. In that case, the observed voltage has the same shape and amplitude with the current intended to be measured.

**Measuring the time**

When there is the sweep voltage by adjusting the TIME/DIV switch the shape of the signal can be acquired on the screen. In this case, the time is given according to the following expression: time=(number of horizontal grids)×(time/div).

**Measuring the frequency**

Measuring the frequency of a periodic signal is possible by making use of the period of the sweep voltage. If the period of the periodic signal is equal to $T$, its frequency is $f = \frac{1}{T}$. After the period is read with the method described in "Measuring the time", frequency can be found by taking the inverse of the period in terms of multiplication.

**Function (signal) generator**

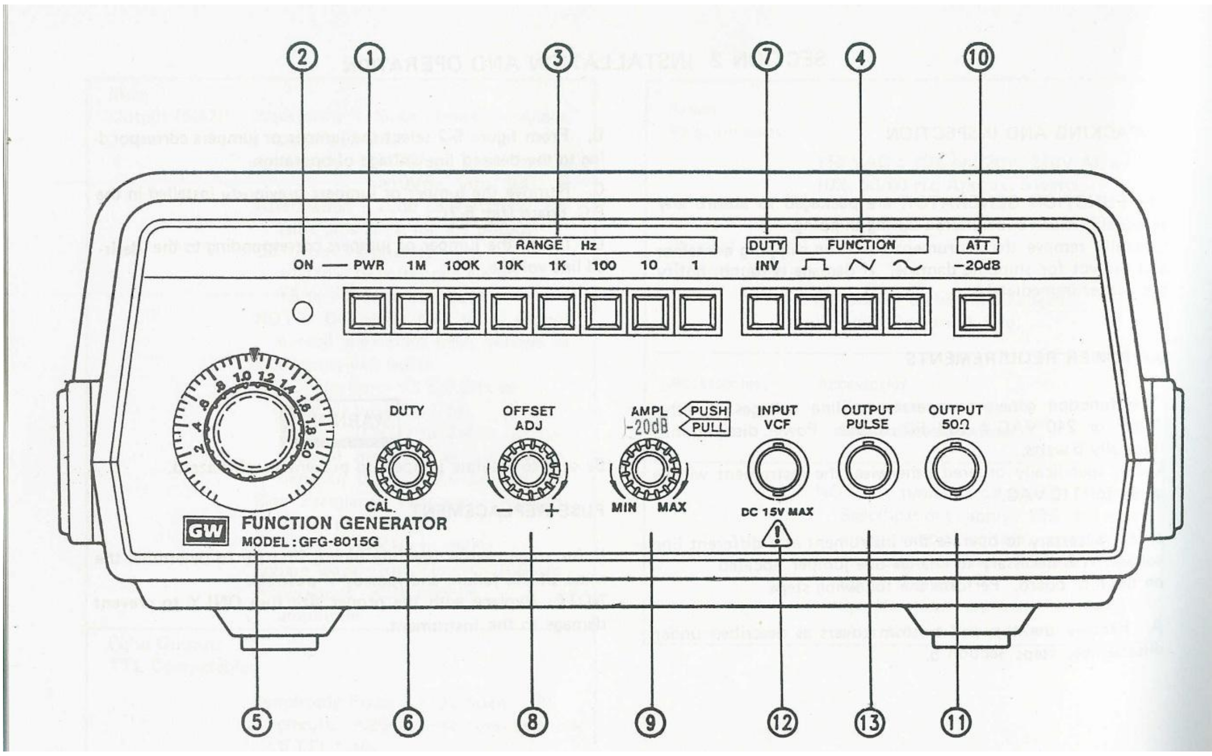



**Figure 1.7 Function generator**

The function generator is able to generate sinusoidal, square or triangle shaped signals in desired amplitude and frequency values in between its specified lower and upper bound.

Frequency adjustment is done by first choosing the rough range with the RANGE panel and then by fine-tuning with the FREQUENCY indicator.

Information about Figure 1.7 is below.

1. On/off button.
2. Light indicator which shows the function generator is operating or not.
3. Frequency level buttons.
4. Function shape buttons.
5. Multiplying coefficient (It multiplies the frequency value adjusted in RANGE panel with a number within the interval 0.2 and 2.0).

6. The indicator to control the time-axis symmetry of the signal (when the indicator points the CAL position, signal shape is symmetric).
7. The button to invert the time axis symmetry.
8. The indicator to specify the DC offset value of the output signal.
9. The indicator to specify the amplitude of the output signal.
10. When this button is on there is a $20$ dB attenuation on the output signal.
11. The output from where the sinusoidal, triangle or square shaped signal can be acquired.
12. The input used for scanning the frequency range externally (VCF: voltage-controlled frequency).
13. The output used to drive the TTL logic circuits.

**Experimental work:**

1. Adjust the oscilloscope, switch it on and connect it with the function generator.
2. Generate a sinusoidal and a square shaped signal with amplitudes of $1$ V and the frequencies $f = 800$ Hz and $f = 10$ kHz respectively. By making use of the vertical and horizontal amplifying coefficients, draw the signals seen on the screen for each case.
3. By setting up the circuit seen on Figure 1.8, for the sinusoidal source signal with the amplitude and frequency of $1$ V and $f = 1$ kHz respectively, find the current flowing through the circuit. By making use of your found current value and the resistance values of the resistors in the circuit, acquire the amplitude of the source signal generated by the function generator.

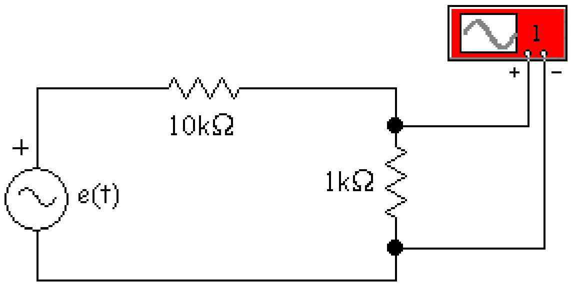



**Figure 1.8 Measuring the current flowing through the circuit with an oscilloscope**

# EXPERIMENT 2: MATHEMATICAL MODELS OF MULTI-NODE CIRCUIT ELEMENTS

## Objective

The objective of the experiment is to comprehend how to express the mathematical model of a multi-node circuit element.

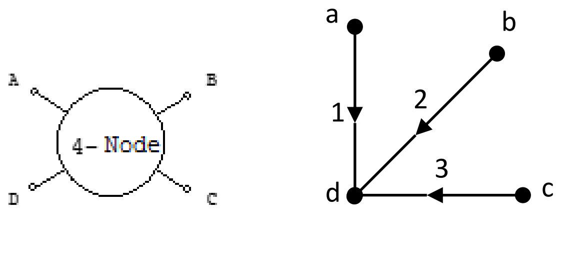
## Foreknowledge

In this part, brief information about how to acquire the mathematical model of a multi-node is given. While doing so, the 4-node circuit element which is going to be used in the experiment is considered.

All the properties of an n-node circuit element are exactly specified by the $(n-1)$ mathematical relation (equation) between the $(n-1)$ current and $(n-1)$ voltage quantities defined by a chosen node-graph of the element. These $(n-1)$ equations are called **the node equations of the n-node**.

The mathematical model of an n-node is expressed by the duo node-graph and node-equations. Due to the fact that a chosen node-graph is not unique, multiple equivalent mathematical models are possible for the same multi-node element.

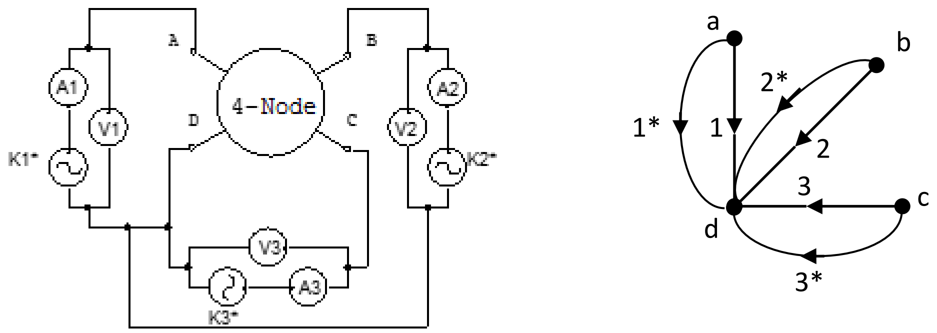
In this experiment, the node-equations of a 4-node circuit element consisting of resistors will be acquired by experimental measuring for a chosen node-graph T. After acquiring the node equations of the 4-node according to the node-graph T, if the inner structure of the 4-node is known (the topology of interconnection of the resistors), the values of the 2-node resistors which comprises the 4-node can be found by making use of the node equations acquired by measurements.




**a** **b**

**Figure 2.1**

To acquire the mathematical model of the 4-node given in Figure 2.1a by measurements, a node graph must be chosen and then the node equations according to this graph must be acquired by making use of the results of the measurements. When the graph T is chosen as seen on Figure 2.1b, measuring devices (ammeters and voltmeters) and sources (voltage or current sources) must be connected as seen on Figure 2.2a.




**a** **b**

**Figure 2.2**

In general, the types of sources to stimulate a multi-node circuit element are related to the inner topology of the multi-node, and they cannot be chosen arbitrarily. (For example, to acquire the mathematical model of a 3-node ideal transformer, if one of the stimulating sources is chosen as a voltage source, the other one must be chosen as a current source.) Depending on the situation, all of the stimulating sources can be voltage or current sources.

On the other hand, it is also possible that some of the stimulating sources are voltage sources and some of them are current sources.

By measuring the current and voltage values of the current and voltage sources connected in between the nodes of the element as seen on Figure 2.2a, node equations of this element can be formed. If all the sources stimulating the 4-node are voltage sources, node equations of the 4-node can be written as

$$\begin{bmatrix} i_1(t) \\ i_2(t) \\ i_3(t) \end{bmatrix} = \begin{bmatrix} G_{11} & G_{12} & G_{13} \\ G_{21} & G_{22} & G_{23} \\ G_{31} & G_{32} & G_{33} \end{bmatrix} \begin{bmatrix} v_1(t) \\ v_2(t) \\ v_3(t) \end{bmatrix} \quad (1)$$

The equations between the node-variables and variables related to the sources seen below can be easily derived from the graph in Figure 2.2b:

$$\begin{bmatrix} v_1(t) \\ v_2(t) \\ v_3(t) \end{bmatrix} = \begin{bmatrix} v_1^*(t) \\ v_2^*(t) \\ v_3^*(t) \end{bmatrix} \qquad \begin{bmatrix} i_1(t) \\ i_2(t) \\ i_3(t) \end{bmatrix} = - \begin{bmatrix} i_1^*(t) \\ i_2^*(t) \\ i_3^*(t) \end{bmatrix} \quad (2)$$

In the node equations seen on (1),

**a)** Taking $v_2(t) = v_3(t) = 0$ means to short-circuit the B-D and C-D nodes of the 4-node. In this case, stimulating the 4-node with only the voltage source $K_1^* = v_1^*(t)$ is enough. By measuring $i_1(t)$, $i_2(t)$ and $i_3(t)$ the elements of the coefficient matrix in (1) can be found as, $G_{11} = i_1/v_1 = -\, i_1^*/v_1^*$, $G_{21} = i_2/v_1 = -\, i_2^*/v_1^*$, $G_{31} = i_3/v_1 = -i_3^*/v_1^*$.

**b)** As in a), by taking $v_1(t) = v_3(t) = 0$, coefficients $G_{12}$, $G_{22}$, $G_{32}$ are acquired. (In this case the 4-node is stimulated only by the voltage source $K_2^* = v_2^*(t)$.)

**c)** By taking $v_1(t) = v_2(t) = 0$, coefficients $G_{13}$, $G_{23}$, $G_{33}$ are acquired. (In this case the 4-node is stimulated only by the voltage source $K_3^* = v_3^*(t)$.)

## Experimental work

### A)

The node equations of the 4-node element which will be given in lab are going to be derived as in (1) according to the node-graph $T_1$ seen on Figure 2.3. For this, after the measurement circuit is set up as seen on Figure 2.4, the measurement results are going to be written to Table 2.1.

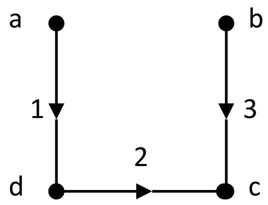



**Figure 2.3**

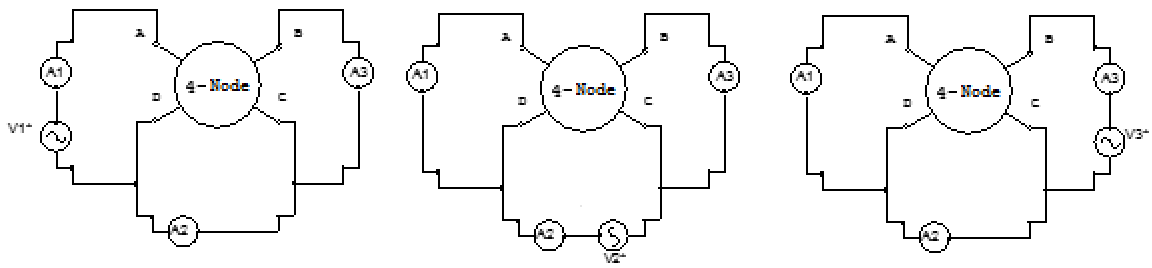



**Figure 2.4**

| $v_2 = 0, v_3 = 0, v_1 = 10V$ | | $v_1 = 0, v_3 = 0, v_2 = 10V$ | | $v_1 = 0, v_2 = 0, v_3 = 10V$ | |
|---|---|---|---|---|---|
| $i_1 =$ | $G_{11} =$ | $i_1 =$ | $G_{12} =$ | $i_1 =$ | $G_{13} =$ |
| $i_2 =$ | $G_{21} =$ | $i_2 =$ | $G_{22} =$ | $i_2 =$ | $G_{23} =$ |
| $i_3 =$ | $G_{31} =$ | $i_3 =$ | $G_{32} =$ | $i_3 =$ | $G_{33} =$ |

**Table 2.1**

**B)**

Repeat the same experiment by taking the graph $T_2$ of the 4-node as seen on Figure 2.5. For this, set up the measurement circuit given in Figure 2.6. By making use of Table 2.2, acquire the node equations of the 4-node according to $T_2$.

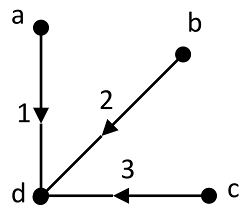



**Figure 2.5**

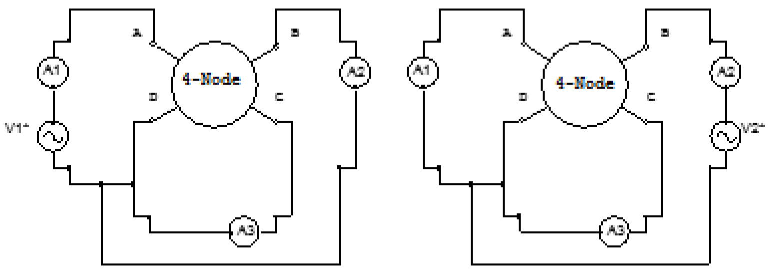



**Figure 2.6**

| $v_2 = 0, v_3 = 0, v_1 = 10V$ | | $v_1 = 0, v_3 = 0, v_2 = 10V$ | | $v_1 = 0, v_2 = 0, v_3 = 10V$ | |
|---|---|---|---|---|---|
| $i_1 =$ | $G_{11} =$ | $i_1 =$ | $G_{12} =$ | $i_1 =$ | $G_{13} =$ |
| $i_2 =$ | $G_{21} =$ | $i_2 =$ | $G_{22} =$ | $i_2 =$ | $G_{23} =$ |
| $i_3 =$ | $G_{31} =$ | $i_3 =$ | $G_{32} =$ | $i_3 =$ | $G_{33} =$ |

**Table 2.2**

**C)**

Node equations of the 4-node will be acquired according to the graph T3 given in Figure 2.7. As seen from the graph, only two current and two voltage measurements are needed to be done. For this purpose, set up the measurement circuit in Figure 2.8 and evaluate the values in Table 2.3. From it, write the node equations according to $T_3$ in the form given below.

$$\begin{bmatrix} i_1(t) \\ i_2(t) \end{bmatrix} = \begin{bmatrix} G_{11} & G_{12} \\ G_{21} & G_{22} \end{bmatrix} \begin{bmatrix} v_1(t) \\ v_2(t) \end{bmatrix} \quad (3)$$

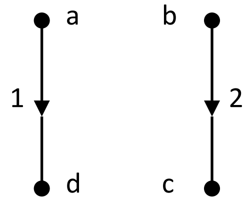



**Figure 2.7**

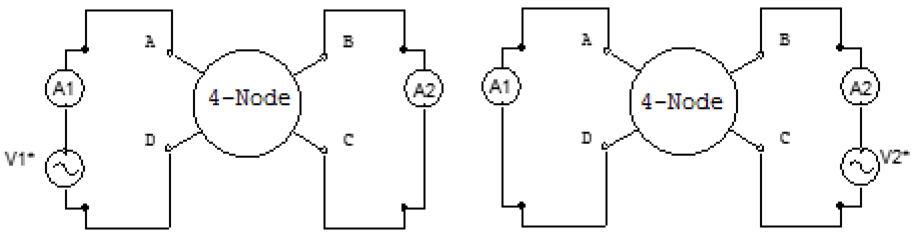



**Figure 2.8**

| $v_2 = 0, v_1 = 10V$ | | $v_1 = 0, v_2 = 10V$ | |
|---|---|---|---|
| $i_1 =$ | $G_{11} =$ | $i_1 =$ | $G_{12} =$ |
| $i_2 =$ | $G_{21} =$ | $i_2 =$ | $G_{22} =$ |

**Table 2.3**

# EXPERIMENT 3: STUDY OF MULTIPLICATIVITY-ADDITIVITY AND RECIPROCITY THEOREMS

## Objective

The aim of this experiment is to comprehend the multiplicativity-additivity (superposition principle) and reciprocity theorems.

Proving that a circuit satisfies the properties of multiplicativity and additivity is equivalent to proving that the circuit is linear. When both of these properties exist, it is said that the circuit satisfies the “superposition principle”, i.e. the circuit, or in more general sense, the system is linear. In some physical systems this property is needed to be satisfied and such systems are designed accordingly. Communication systems, sound amplifiers, various measuring devices and a lot more system can be given as examples to systems which desired to be operated linearly.

Reciprocity is also a property which can be observed in various physical systems. Reciprocity is an important property in the subjects as electrical circuits and continuum mechanics (like elastic or piezoelectric mediums). A priori knowledge of the existence of this property in a circuit makes the analysis and the design of the circuit easier.

## Foreknowledge

**Multiplicativity Theorem:** Let the forced solution (response) and particular solution of the output of a circuit consisting of linear resistors, inductors, capacitors, linear multi-nodes, voltage and current sources shown by $y_f(t)$ and $y_p(t)$ respectively. If the functions (or amplitudes) of all the independent sources in the circuit are multiplied by a coefficient $k$, the forced and particular solution of the circuit would be $ky_f(t)$ and $ky_p(t)$ respectively.

Multiplicativity theorem can also be briefly stated as follows: In a circuit consisting of linear elements, if the amplitude of the independent sources are multiplied by $k$, the forced solutions related to all voltages and currents in the circuit are also multiplied by $k$.

**Additivity Theorem:** Let again the forced solution (response) and particular solution of the output of a circuit consisting of linear resistors, inductors, capacitors, linear multi-nodes, voltage and current sources shown by $y_f(t)$ and $y_p(t)$ respectively. In this circuit which has $r$ independent sources, if all the sources in the circuit except the $i^{th}$ one are cancelled out (their amplitudes are made zero, i.e. all the other voltage sources are made short-circuit and current sources are made open-circuit), let the forced and particular solutions for this case be defined by $y_{fi}(t)$ and $y_{pi}(t)$ respectively. In this case the forced and particular solutions of the original circuit are given below as:

$$y_f(t) = \sum_{i=1}^{r} y_{fi}(t)$$

$$y_p(t) = \sum_{i=1}^{r} y_{pi}(t)$$

Multiplicativity theorem can also be briefly stated as follows: The forced solution of a circuit consisting of linear elements can be written as the sum of the forced solutions related to each individual independent source in the circuit.

The validity of multiplicativity and additivity theorems will be discussed for RLC circuits. The state equations of a RLC circuit are in the general form:

$$\frac{d}{dt}x(t) = Ax(t) + Bu(t) \tag{1}$$

In these equations, $x(t)$ is the state vector (column matrix) representing the state variables and $u(t)$ is the input vector representing the currents and voltages of the independent sources and their derivatives. If the state variables in a circuit are known, the current and voltage values of all circuit elements can be acquired by making use of fundamental loop, cutset equations and node equations of the elements. That means, the vector consisting of the current and voltage values in interest can be written as

$$y(t) = Cx(t) + Du(t) \tag{2}$$

where $C$ and $D$ are matrices with real coefficients and $y(t)$ is called the output vector. The forced solution of the equation (1) can be given as

$$x_f(t) = x_p(t) - \Phi(t)x_p(0). \tag{3}$$

Here, $\Phi(t)$ is called the state transition matrix and it is related to the matrix $A$. $x_p(t)$ is the particular solution of the equation (1). It is clear that the particular solution satisfies the equation

$$\frac{d}{dt}x_p(t) = Ax_p(t) + Bu(t). \tag{4}$$

If all the amplitudes of the sources are multiplied by $k$ in the circuit taken in hand, to write the state equations of the circuit, it is enough to replace $u(t)$ with $ku(t)$ in equation (1). In this case, if the particular solution of the differential equation is defined as $x_y(t)$, the equation

$$\frac{d}{dt}x_y(t) = Ax_y(t) + Bku(t) \tag{5}$$

is satisfied. If $x_y(t)$ is replaced with $kx_p(t)$ in (5) and when also equation (4) is considered it can be seen that $kx_p(t)$ is the particular solution for the case when the amplitude of the sources are multiplied by $k$. Thus, the multiplicativity theorem is proved.

To prove the additivity theorem the input vector $u(t)$ consisting of sources given in (1) is written in the form:

$$\begin{bmatrix} u_1(t) \\ u_2(t) \\ u_3(t) \\ \vdots \\ \vdots \\ u_r(t) \end{bmatrix} = \begin{bmatrix} u_1(t) \\ 0 \\ 0 \\ \vdots \\ \vdots \\ 0 \end{bmatrix} + \begin{bmatrix} 0 \\ u_2(t) \\ 0 \\ \vdots \\ \vdots \\ 0 \end{bmatrix} + \cdots + \begin{bmatrix} 0 \\ 0 \\ 0 \\ \vdots \\ \vdots \\ u_r(t) \end{bmatrix} \tag{6}$$

Let each column vector on the right side of equation (6) are defined as $\widetilde{u_1}(t)$, $\widetilde{u_2}(t)$, $\widetilde{u_3}(t)$,…, $\widetilde{u_r}(t)$. In this case equation (1) can be written as

$$\frac{d}{dt}x(t) = Ax(t) + B\widetilde{u_1}(t) + B\widetilde{u_2}(t) + \cdots + B\widetilde{u_r}(t) \tag{7}$$

Let the particular solution of the equation

$$\frac{d}{dt}x(t) = Ax(t) + B\widetilde{u_\imath}(t) \tag{8}$$

be $x_{pi}(t)$. Since $x_{pi}(t)$ would satisfy the equation (8) following can be written:

$$\frac{d}{dt}x_{pi}(t) = Ax_{pi}(t) + B\widetilde{u_\imath}(t). \tag{9}$$

If this equation is written for each value of $i = 1, \dots, r$ and summed up

$$\frac{d}{dt}\sum_{i=1}^{r} x_{pi}(t) = A\sum_{i=1}^{r} x_{pi}(t) + B\widetilde{u_1}(t) + B\widetilde{u_2}(t) + \cdots + B\widetilde{u_r}(t) \tag{10}$$

is acquired. As seen from this equation the particular solution of equation (8) can be written as

$$x_p(t) = \sum_{i=1}^{r} x_{pi}(t) \tag{11}$$

Thus, the additivity property is expressed for the particular solution. When the expression in (11) is replaced in (3) the proof for additivity is complete.

**Reciprocity Theorem:** Before moving on to the theorem, let us consider the circuit in Figure 3.1a first to explain what the reciprocity property in a circuit means.

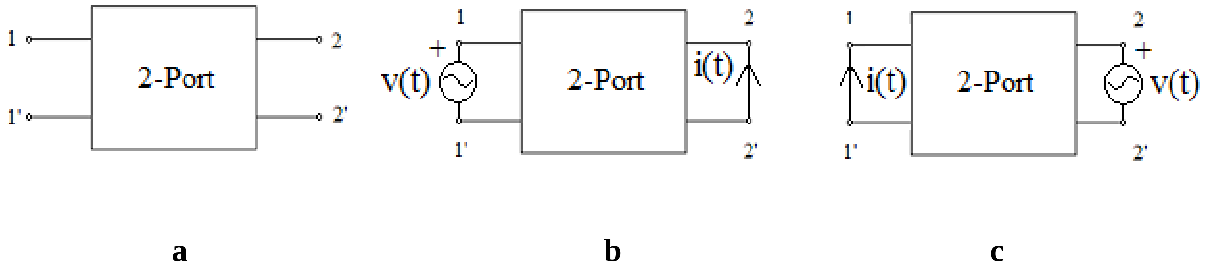



**Figure 3.1**

When the 2-2’ nodes of this circuit are made short-circuit as seen on Figure 3.1b and the source $v(t)$ is connected to the nodes 1-1’, let us call the current flowing through 2-2’ short circuit path as $i(t)$. If the current flowing through port-1 is equal to $i(t)$ when the source in port-1 is connected to port-2 as seen on Figure 3.1c, it is said that the circuit has the property of reciprocity.

Let us generalize this notion for multi-node circuits. For this consider the circuit in Figure 3.2 and let us connect independent sources to each of its ports.

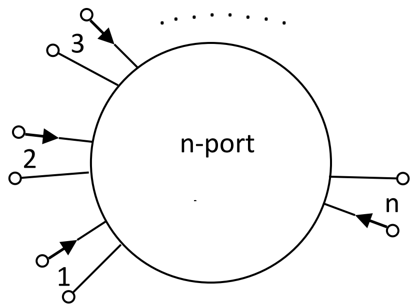



**Figure 3.2**

Let us define the voltages and currents in each of these ports as $v_1(t)$, $v_2(t)$, …, $v_n(t)$ and $i_1(t)$, $i_2(t)$, …, $i_n(t)$. Then, assume that these sources are disconnected and instead new sources are connected to each port. In this case, let us define the new port voltages and currents as $v_1'(t)$, $v_2'(t)$, …, $v_n'(t)$ and $i_1'(t)$, $i_2'(t)$, …, $i_n'(t)$. If the vectors $v_a$, $i_a$, $v_b$ and $i_b$ are defined as

$$v_a = \begin{bmatrix} v_1(t) \\ v_2(t) \\ \vdots \\ v_n(t) \end{bmatrix}, i_a = \begin{bmatrix} i_1(t) \\ i_2(t) \\ \vdots \\ i_n(t) \end{bmatrix}, v_b = \begin{bmatrix} v'_1(t) \\ v'_2(t) \\ \vdots \\ v'_n(t) \end{bmatrix}, i_b = \begin{bmatrix} i'_1(t) \\ i'_2(t) \\ \vdots \\ i'_n(t) \end{bmatrix} \tag{12}$$

and the equation

$$v_a^T i_b = v_b^T i_a \tag{13}$$

is satisfied, it is said that the circuit has the property of reciprocity.

The second general definition given for reciprocity includes the definition given for a 2-port. If the case in Figure 3.1b is considered following can be written:

$$v_1(t) = v(t), v_2(t) = 0;\ i_1(t) = i_1(t), i_2(t) = i(t).$$

And for the case in Figure 3.1c;

$$v'_1(t) = 0, v'_2(t) = v(t);\ i'_1(t) = i(t), i'_2(t) = i'_2(t).$$

Thus,

$$v_a = \begin{bmatrix} v(t) \\ 0 \end{bmatrix}, i_a = \begin{bmatrix} i_1(t) \\ i(t) \end{bmatrix}, v_b = \begin{bmatrix} 0 \\ v(t) \end{bmatrix}, i_b = \begin{bmatrix} i(t) \\ i'_2(t) \end{bmatrix}$$

and it can be seen that the equation (13) is satisfied.

The definition of reciprocity of a 2-port is physically the fact that the effect of port-1 on port-2 is the same as the effect of port-2 on port-1.

**Experimental work**

**Multiplicativity and Additivity:** For the experimental inspection of this theorem the circuit seen on Figure 3.3 will be made use of.

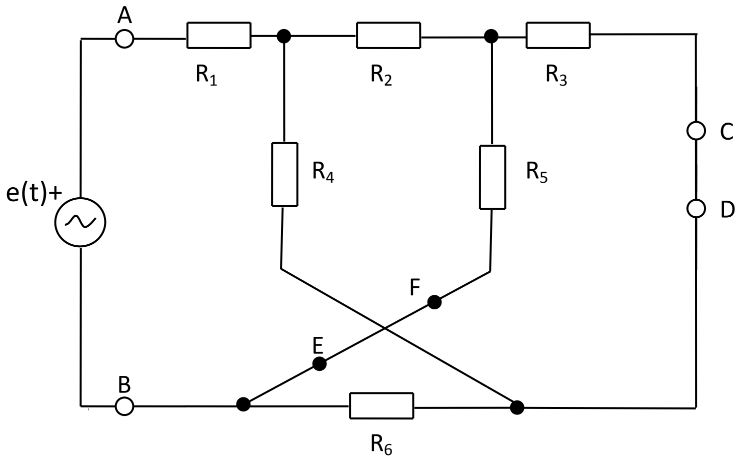



**Figure 3.3**

**A)** Set up the circuit seen on Figure 3.3. Measure the C-D short-circuit current and the voltage between nodes A-B. Change the amplitude of the voltage source, then measure the C-D short-circuit current note your measured values.

**B-1)** Connect a voltage source between nodes E-F of the circuit seen on Figure 3.3. When there are these two independent sources in the circuit measure the C-D short-circuit current.

**B-2)** Make the nodes E-F short-circuit again and measure the C-D short-circuit current of this circuit with one independent source.

**B-3)** Connect the voltage source in B-1 again between the nodes E-F and make A-B nodes short-circuit. Measure the C-D short-circuit current. Sum up the measured values in B-2 and B-3 and compare the result with the measured value in B-1.

**Reciprocity Theorem:** For the experimental inspection of this theorem the circuit seen on Figure 3.3 will be made use of.

**A)** Set up the circuit seen on Figure 3.3. Note the C-D short-circuit current. Connect the voltage source in between nodes A-B to the nodes C-D without changing its amplitude.

Measure the A-B short-circuit current. By commenting on the results, figure out if the circuit has the reciprocity property or not.

**B-1)** To prove the reciprocity of a circuit by using equation (13), set up the circuit seen on Figure 3.4.

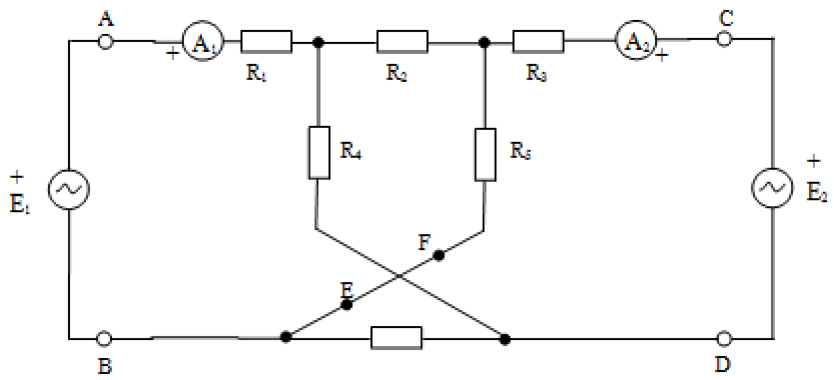



**Figure 3.4**

**B-2)** Adjust the sources $E_1$ and $E_2$ to some specific voltages $V_1$ and $V_2$. Measure the currents $i_1$ and $i_2$ flowing through the ammeters $A_1$ and $A_2$.

**B-3)** Change the voltage values of the sources $E_1$ and $E_2$ to $V_1'$ and $V_2'$. Measure the currents $i_1$ and $i_2$ flowing through the ammeters $A_1$ and $A_2$.

**B-4)** By making use of your measurements in B-2 and B-3 decide whether the circuit has the reciprocity property or not.

# EXPERIMENT 4: THÉVENIN'S AND NORTON'S THEOREMS

## Objective

The aim of the experiment is to practically comprehend Thévenin's and Norton's theorems.

## Foreknowledge

Two of the main methods used in circuit analysis are loop currents and node voltages methods. By using these methods, all the current-voltage pairs in a circuit of interest can be found. In circuit analysis and some other circuit applications, a linear time-invariant 2-port whose analysis was previously done is connected with other such 2-ports and new circuits are created this way. In such cases, instead of re-analyzing this newly acquired circuit, to make use of the previously acquired information about the 2-port for the analysis of this new circuit, Thévenin's and Norton's theorems are used.

Consider a linear circuit $N_A$ is connected to another circuit $N_B$ from its A-B nodes as seen on Figure 4.1. The voltage $v(t)$ between the nodes A-B and the current $i(t)$ drawn by the circuit $N_B$ wouldn't be changed if the Thévenin or Norton equivalent of the circuit $N_A$ is placed to the plant instead of the actual circuit. Placing an equivalent simple circuit to the plant instead of the circuit $N_A$ with a complex topology, makes the evaluations of the currents and voltages in question easier.

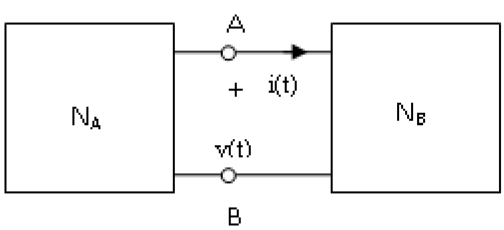
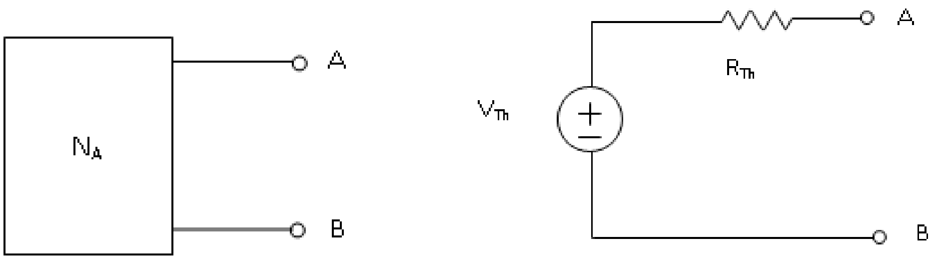



**Figure 4.1**

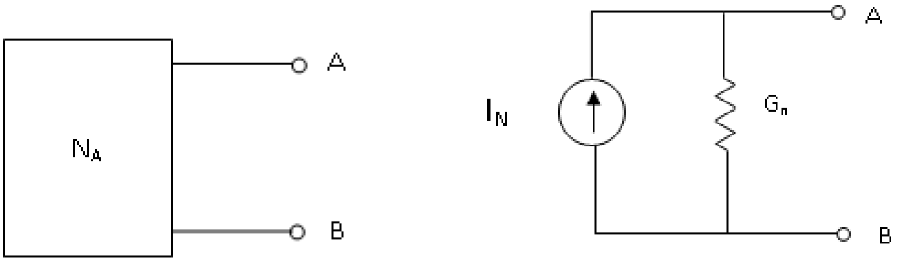
The Thévenin equivalent circuit is represented with a voltage source and a 2-node resistor as seen on Figure 4.2b. The amplitude of the voltage source $V_{th}$ in the equivalent circuit is equal to the measured open-circuit voltage between the nodes A-B of the circuit $N_A$ shown in Figure 4.2a. The voltage $V_{th}$ in the Thévenin equivalent circuit is specified by the independent voltage and current sources in the circuit $N_A$. On the other hand, the resistance $R_{th}$ in the Thévenin equivalent circuit is specified by the equivalent resistance seen through nodes A-B when the independent sources are cancelled out (when the independent voltage sources in the circuit $N_A$ are made short-circuit and current sources are made open-circuit).




**a** **b**

**Figure 4.2**

The Norton equivalent circuit is represented with a current source and a 2-node resistor as seen on Figure 4.3b.




**a** **b**

**Figure 4.3**

The amplitude of the current source $I_n$ in the equivalent circuit is equal to the measured current flowing through nodes A-B of the circuit $N_A$ shown in Figure 4.3a when these nodes are made short-circuit. On the other hand, the conductance $G_n$ is, as in Thévenin equivalent circuit case, equal to the equivalent conductance seen through nodes A-B when the independent sources are cancelled out.

**Experimental work**

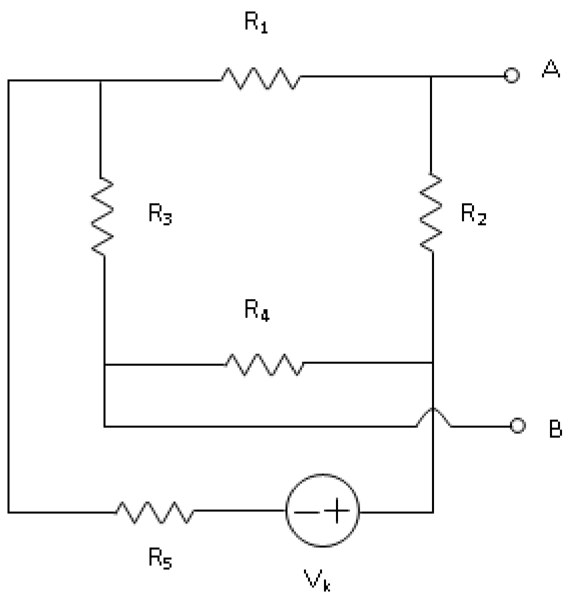



**Figure 4.4**

Set up the circuit seen on Figure 4.4 ($R_1 = 10k\Omega, R_2 = 1.8k\Omega, R_3 = 1.2k\Omega, R_4 = 33k\Omega,$ $R_5 = 4.7k\Omega, V = 10V$).

**A)** Acquiring the Thévenin and Norton equivalent circuits

**1)** To find the voltage $V_{th}$ belonging to the equivalent circuit in Figure 4.2b, measure the open-circuit voltage between nodes A-B.

**2)** To find the resistance $R_{th}$ belonging to the equivalent circuit in Figure 4.2b, make the voltage source $V_k$ short circuit and measure the resistance value between nodes A-B.

**3)** To find the current $I_n$ belonging to the equivalent circuit in Figure 4.3b, connect an ammeter between the nodes A-B and measure the short-circuit current.

**4)** Calculate the equivalent conductance $G_n = 1/R_{th}$ and draw the Thévenin and Norton equivalent circuits.

**B)** Verifying the equivalent circuits

**1)** Connect a $R = 1k\Omega$ valued resistor to the nodes A-B of the circuit in Figure 4.4 and measure the current and voltage value among this resistor.

**2)** Set up the Thévenin equivalent circuit related to the circuit in Figure 4.4, connect a $R = 1k\Omega$ valued resistor to the nodes A-B and measure the current and voltage value among this resistor.

**3)** Find the Thévenin equivalent of the circuit in Figure 4.4 by using one of the circuit analysis methods (Node voltages / Loop currents).

**4)** Compare your results in 2 and 3 with your measurements in 1.

**C)** Set up the Thévenin equivalent circuit related to the circuit in Figure 4.4 and connect the resistor $R_L$ between the nodes A-B. Fill in Table 4.1. By commenting on your acquired values, find the needed relation between $R_L$ and $R_T$ to transfer the maximum amount of power to $R_L$.

| Load Resistor | Calculated $I_{RL}$ | Calculated $V_{RL}$ | Calculated $P_{RL}$ | Measured $I_{RL}$ | Measured $V_{RL}$ | Measured $P_{RL}$ |
|---|---|---|---|---|---|---|
| $R_L = 0.1R_{th}$ | | | | | | |
| $R_L = R_{th}$ | | | | | | |
| $R_L = 10R_{th}$ | | | | | | |

**Table 4.1**

# EXPERIMENT 5: EXAMINING TRANSIENT BEHAVIORS IN RL, RC AND RLC CIRCUITS

## Objective

In this experiment the aim is to increase the knowledge about the transient behaviors of 1$^{st}$ and 2$^{nd}$ order electrical circuits.

## Foreknowledge

To specify the behavior of an electrical circuit as a function of time, the equations related to the circuit must be acquired and solved. Circuit equations, in their general form, include integrals, derivatives and algebraic relations. Since solving such equations is hard, the state variable method in which the equations contain no integrals is preferred in analyzing electrical circuits.

Let us consider a circuit which consists of only independent sources, resistors, capacitors and inductors. In such a circuit, the node equations related to the independent sources and resistors are algebraic whereas the node equations related to capacitors and inductors are differential. In state variable method, a group containing linearly independent capacitor voltages and inductor currents is chosen as the set of state variables. By writing the other quantities in the circuit in terms of state variables, circuit equations contain no integrals. If the currents and voltages related to the state variables are known, other circuit quantities can be evaluated by merely using algebraic equations. Therefore, it can be said that to figure out the behavior of the state variables means to figure out the behavior of the circuit. In state variable method, by using the node equations related to the circuit elements and the loop/cutset equations, a set of differential equations as (1) is acquired. The vector $x(t)$ seen on (1) represents the state variables and $e(t)$ represents the independent sources.

$$\frac{d}{dt}x(t) = Ax(t) + Be(t) \tag{1}$$

The solution of this set of differential equations is found in two steps. Firstly, the part of the solution influenced by the term $Ax(t)$ in (1) is found by solving the following homogeneous differential vector-matrix equation.

$$\frac{d}{dt}x(t) = Ax(t). \tag{2}$$

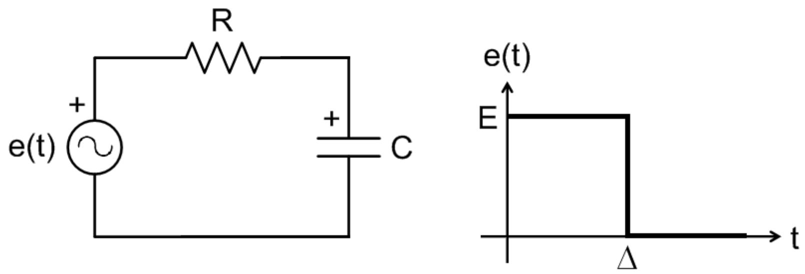
The solution of the equation (2) contains arbitrary coefficients. The values of the state variables at the instant when the circuit begins to operate (at the time instant $t = 0$) are called the initial conditions. By making use of the fact that those initial conditions must also be satisfied by the solution, the arbitrary coefficients in the mathematical solution are specified. This is the zero-input solution (response) related to the circuit. Zero-input solution specifies the behavior of the circuit when the impact of independent sources is zero i.e., it specifies the behavior only influenced by the initial conditions.

By solving the equation (2), the influence of the term $Ax(t)$ on the solution of equation (1) is found. To find the complete solution of (1), the influence of the term $Be(t)$ on the solution must also be found. At this stage, this part of the solution is found by using one of the differential equation solving methods. As in the case of homogeneous solution, this part of the solution also contains some arbitrary constant coefficients. These constants are found again by making use of the initial conditions. Thus, the zero-state solution (forced solution/response) related to the circuit is found. The zero-state solution specifies the behavior of the circuit when the initial conditions are zero i.e., it specifies the behavior only influenced by the outer independent sources. The sum of the zero-state solution with the zero-input solution previously acquired yields the complete solution of equation (1).

The circuits whose zero-input solutions for $t \to \infty$ are equal to zero, are called asymptotic stable circuits. In an asymptotic stable circuit, after a specified time from the beginning of the circuit's operation, complete solution is practically (with a good amount of approximation) equal to the zero-state solution. The transient solution, even though it is influential in the beginning, becomes smaller and nearly zero in time. The permanent solution is the solution which will exist as long as there are sources in the circuit. In this experiment, by considering RC, RL and RLC circuits, the transient behavior of them when stimulated by a square-wave source will be examined.

**RC Circuit**

Let us consider the RC circuit in Figure 5.1a.




**a** **b**

**Figure 5.1**

The state equations of this circuit are given as follows:

$$\frac{d}{dt}v_c(t) = -\frac{1}{RC}v_c(t) + \frac{1}{RC}e(t). \tag{3}$$

Let the voltage source $e(t)$ is chosen as a unit step function given in Figure 5.1b. In this case, the RC circuit must be analyzed separately for the cases $e(t) = E$ $(t < \Delta)$ and $e(t) = 0$ $(t \geq \Delta)$.

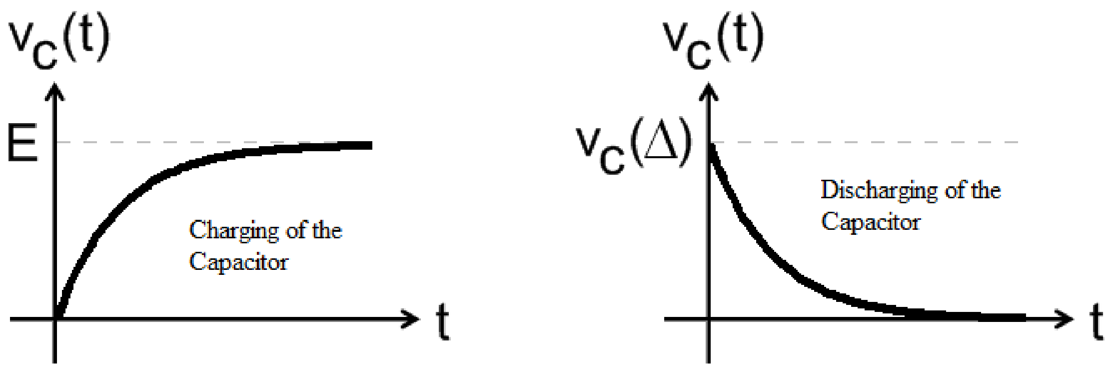
The solution of the differential equation (3) for $e(t) = E$ is:

$$v_{C_{cmp}}(t) = \underbrace{v_c(0)e^{-t/RC}}_{v_{c_{zi}}(t)} + \underbrace{E(1 - e^{-t/RC})}_{v_{c_{zs}}(t)} \tag{4}$$

and for $e(t) = 0$:

$$v_{C_{cmp}}(t) = \underbrace{v_C(\Delta)e^{-t/RC}}_{v_{c_{zi}}(t)}. \tag{5}$$

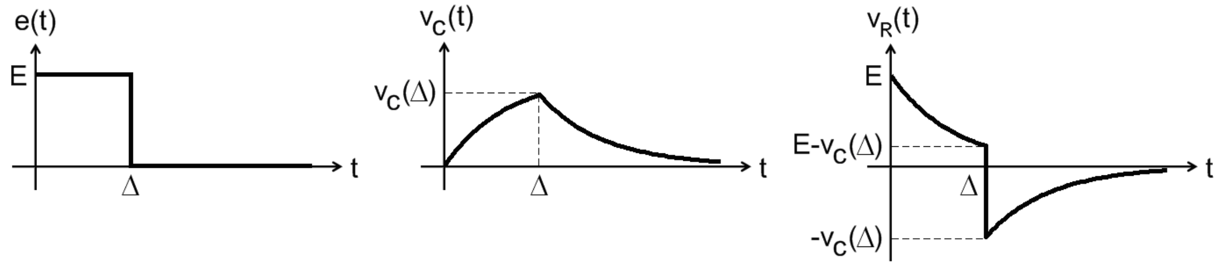
The product $RC$ which specifies the exponential terms in (4) and (5) is called the time constant and represented by $\tau$. If the unit of resistance is Ohms and the unit of capacitance is Farads, the unit of the time constant is seconds. The time constant specifies for what time the zero-input solution is influential. The terms specifying the zero-input solution are $v_c(0)e^{-t/RC}$ and $v_C(\Delta)e^{-t/RC}$. For values of $t$ greater than $5RC$, since in this case $e^{-t/\tau} < 0.01$, zero-input solution is considered to be zero. Since in equation (4) the zero-state response is $E(1 - e^{-t/RC})$, after $5\tau$ seconds, the capacitor voltage is approximately $E$. This phenomenon is called the charging of the capacitor. On the other hand, since there are no terms related to the sources in equation (5), the zero-state solution is equal to zero. That means the voltage of the capacitor will be equal to zero after $5\tau$ seconds. This phenomenon is called the discharging of the capacitor. The charging and discharging graphics of the capacitor are in the form seen on Figure 5.2.




**a** **b**

**Figure 5.2**

From Figure 5.1, it can be seen that the voltage source, capacitor and the resistor forms a loop. It means when $v_C(t)$ is known $v_R(t)$ can also be found. The voltages of the source, capacitor and resistor can be seen as a function of time in Figure 5.3.




**a** **b** **c**

**Figure 5.3**

Let the voltage source in Figure 5.1 be chosen as a square-wave with $T_1 = T_2$ as in Figure 5.4a. In this case, there can be three different forms of behavior depending on the value of $T$ in the circuit.

***i)*** If $t \ll \tau$, the capacitor is charged during the pulse ($T_1$) and it is discharged during the pulse gap ($T_2$). Since the time constant is small, the capacitor can be charged and discharged completely. The function of this voltage can be expressed as:

$$v_C(t) = \begin{cases} E\left(1 - e^{-t/RC}\right); 0 \le t < T_1 \\ v_C(T_1)e^{-t/RC}; \qquad t \ge T_1 \end{cases} \tag{6}$$

In Figure 5.4a for $t \ll T$ and in Figure 5.4b for $5\tau = T/2$ the change in the voltages of the capacitor and the resistor are seen respectively.

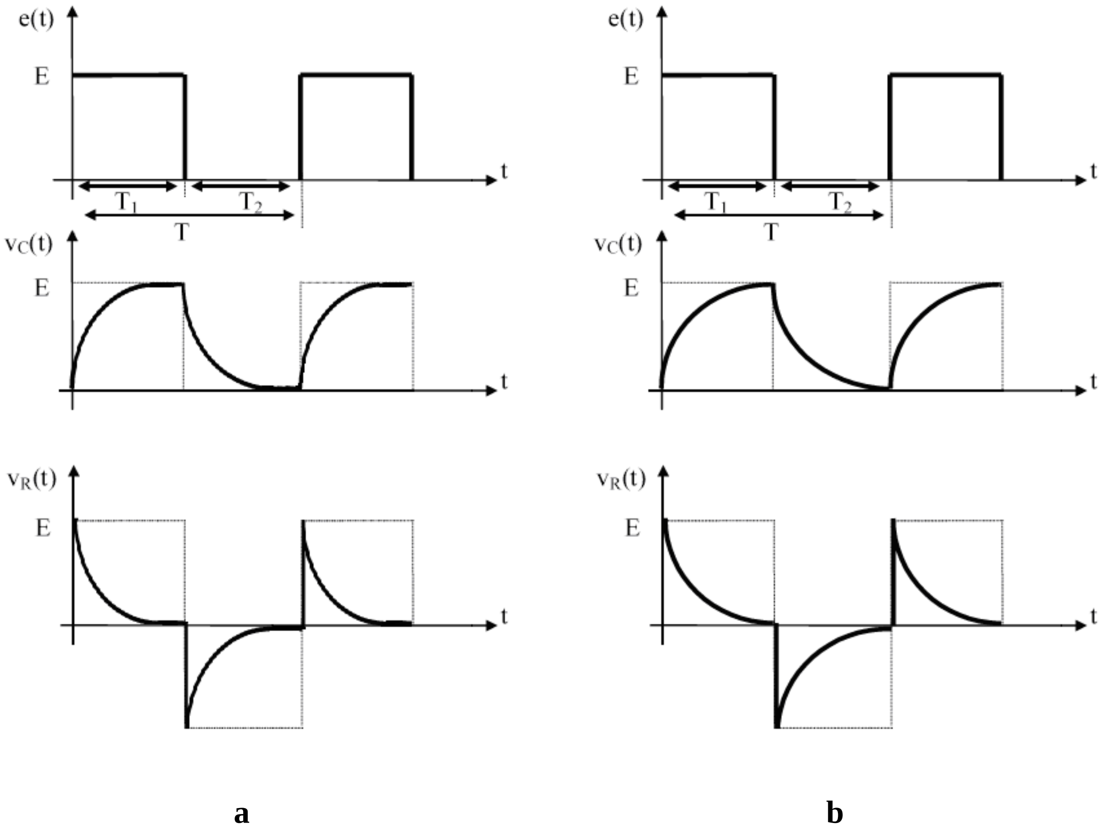



**Figure 5.4**

***ii)*** If $5\tau > T/2$, the change in $v_C(t)$ will be like seen on Figure 5.5a. With the first pulse, the capacitor will be charged to a specific amount, in the duration of the pulse gap ($T_2$) the capacitor will not have the time to discharge completely and it will start to charge again with the new pulse. Since the voltage of the capacitor after one period is greater than the previous, the capacitor will be charged to a greater amount with each iteration. In other words, the remaining amount of voltage among the capacitor will be increased with each iteration since at each time the capacitor starts to discharge itself from a greater amount of voltage. This is the case for the initial pulses. After a while the voltage change between the nodes of the capacitor will become periodical. This case is seen on Figure 5.5b.

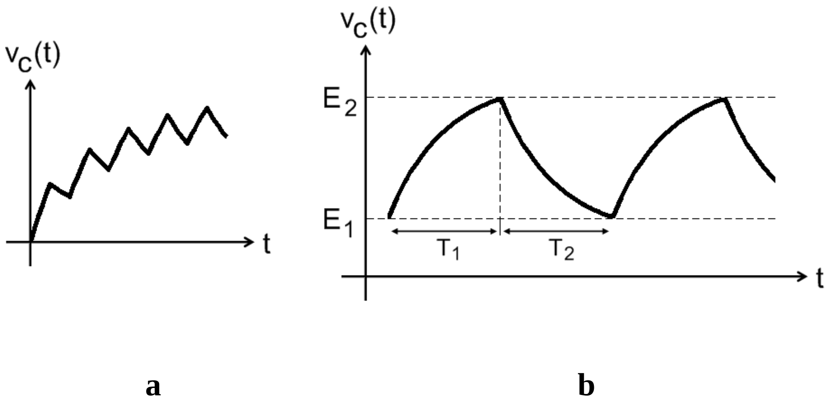



**Figure 5.5**

***iii)*** If $\tau \gg T$, the exponential terms in (4) and (5) will behave almost entirely linearly. Because of this reason, Figure 5.5b will take the form of Figure 5.6.

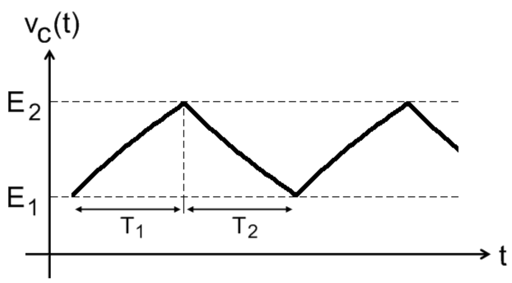



**Figure 5.6**

**RL Circuit**

Let us consider the RL circuit in Figure 5.7.

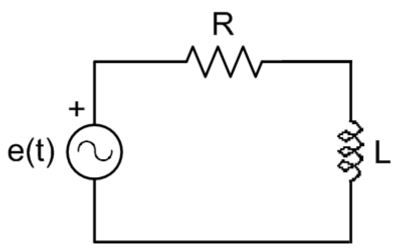



**Figure 5.7**

The state equations of this circuit are given as follows:

$$\frac{d}{dt}i_L(t) = -\frac{R}{L}i_L(t) + \frac{1}{L}e(t) \qquad (7)$$

Since the equations (7) and (3) are in the same form, the observations made for the RC circuit is valid also for the RL circuit. However, the time constant in the RL circuit is $\tau = L/R$ whereas it is $\tau = RC$ in the RC circuit.

**RLC Circuit**

Let us consider the RLC circuit in Figure 5.8.

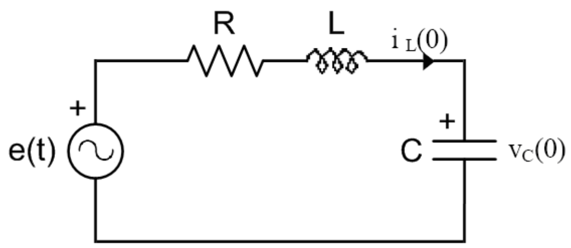



**Figure 5.8**

The state equations of this circuit are given as follows:

$$\frac{d}{dt}\begin{bmatrix} v_C(t) \\ i_L(t) \end{bmatrix} = \begin{bmatrix} 0 & 1/C \\ -1/L & -R/L \end{bmatrix}\begin{bmatrix} v_C(t) \\ i_L(t) \end{bmatrix} + \begin{bmatrix} 0 \\ 1/L \end{bmatrix} e(t) \qquad (8)$$

This equation is in the form of (1); however $A$ in this equation is a matrix whereas it is a scalar in (3). When equations (4) and (5) are examined, it is seen that the solution of (3) contains the term $e^{At}$. Similarly the solution of (8) also contains the term $e^{At}$. The key point of finding the solution of this differential equation is to evaluate this matrix-exponential $e^{At}$.

The characteristic equation of this equation set will specify the structure of the solution. The characteristic equation of (8) is:

$$p^2 + 2\xi\omega_0 p + \omega_0^2 = 0 \qquad (9)$$

Here;

$$\omega_0^2 = \frac{1}{LC}, \xi = \frac{R}{2}\sqrt{\frac{L}{C}} \tag{10}$$

The roots of the equation (9) may be one of the following three depending on the value of $\xi$:

(a) If $\xi > 1$, $R > 2\sqrt{\frac{L}{C}}$ and the roots are real.

(b) If $\xi = 1$, $R = 2\sqrt{\frac{L}{C}}$ and the roots are real and equal.

(c) If $0 \leq \xi < 1$, $0 \leq R < 2\sqrt{\frac{L}{C}}$ and the roots are a complex-conjugate pair.

For $v_C(0) = 0$, $i_L(0) = 0$ for all three cases the form of the function $v_C(t)$ can be roughly seen on Figure 5.9.

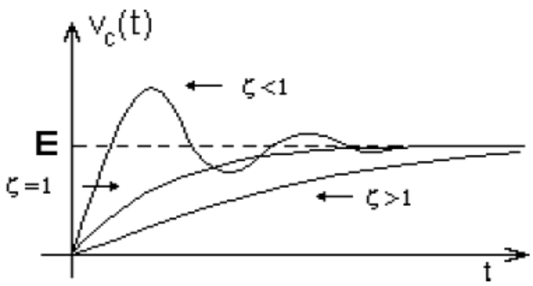



**Figure 5.9**

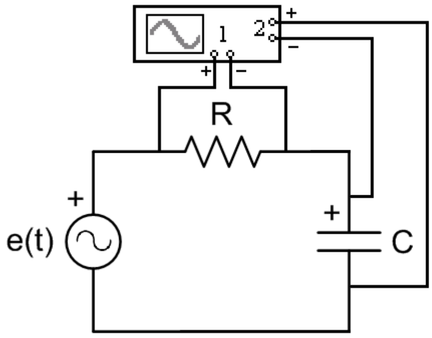
The form of the solution for cases (a), (b) and (c) will be as given below:

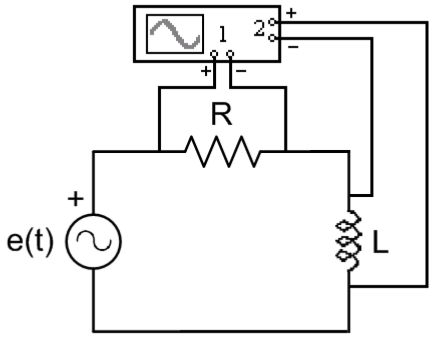
(a) $x(t) = K_1e^{-\alpha_1 t} + K_2e^{-\alpha_2 t}$

(b) $x(t) = K_1e^{-\alpha t} + K_2te^{-\alpha t}$

(c) $x(t) = e^{-\alpha t}(K_1cos\beta t + K_2sin\beta t)$

**Experimental work**




**a** **b**

**Figure 5.10**

**1)** Set up the circuit given in Figure 5.10a by taking the resistance value as $100\Omega$, and the capacitance value as $1\mu F$. Apply a $0-5V$ symmetric square-wave to the input of the circuit. Observe $v_c(t)$ by changing the frequency of the square-wave. Specify the frequency where you get the closest form seen on Figure 5.4b (case of complete charging and discharging). Verify this frequency value with your calculations. By changing the period of the square-wave to $T = 10RC$, $T = RC$ and $T = RC/10$, draw the forms you see on the screen of the oscilloscope for $v_C(t)$ and $v_R(t)$ with noting the peak values.

**2)** Reduce the amplitude of the square-wave to its half, repeat the measurements in 1) and comment on the results.

**3)** Set up the circuit given in Figure 5.10b by taking the resistance value as $600\Omega$, and the inductance value as $60mH$. Apply a $0-5V$ symmetric square-wave to the input of the circuit. Observe $v_R(t)$ by changing the frequency of the square-wave. Specify the frequency where you get the closest form seen on Figure 5.4b (case of complete charging and discharging). Comment on the difference between the value you get by calculations and the value you get by measurements. By changing the period of the square-wave to $T = 10L/R$,

$T = L/R$ and $T = L/10R$, draw the forms you see on the screen of the oscilloscope for all three cases with noting the peak values.

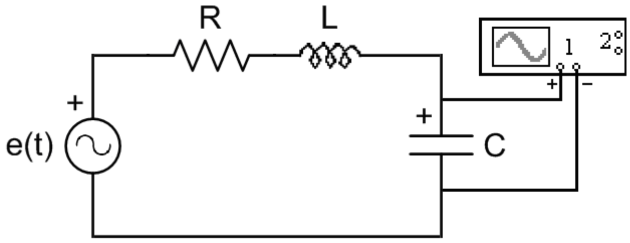



**Figure 5.11**

**4)** Set up the circuit given in Figure 5.11 by taking the capacitance value as $1\mu F$, and the inductance value as $60mH$. Apply a $0 - 5V$ symmetric square-wave with the frequency of $100Hz$ to the input of the circuit. By taking $R = 20\sqrt{\frac{L}{C}}$, $R = 2\sqrt{\frac{L}{C}}$ and $R = 0.2\sqrt{\frac{L}{C}}$, draw the forms you see on the oscilloscope screen for each case. Comment on your resulting graphs. For $R = 20\sqrt{\frac{L}{C}}$, explain how changing the value of the capacitance affects the wave-form.

# EXPERIMENT NO. 6: SINUSOIDAL STEADY STATE ANALYSIS

## Objectives

Use phasor analysis to obtain steady state solution of RLC circuits that are excited with sinusoidal ac sources.

## Finding Sinusoidal Steady State Solution

The complete solution for the state equation of a stable dynamic circuit can be written as the sum of two parts: transient solution and steady state solution. The part that decays to zero in time is called as “transient solution”, and the part that contributes to complete the solution as long as the sources exist is called as “steady state solution”.

$$\underbrace{x(t)}_{\text{Complete Solution}} = \underbrace{\phi(t)\left(x_0 - x_{ö}(0)\right)}_{\text{Transient Solution}} + \underbrace{x_{ö}(t)}_{\text{SteadyStateSolution}} \tag{1}$$

$$x_{SSS}(t) = x_{CS}(t)_{\ t\to\infty} \tag{2}$$

The solution of an RLC circuit which is excited by a single sinusoidal source or multiple sinusoidal sources having the same frequency is defined as sinusoidal steady state solution.
A sinusoidal signal can be expressed as in eq. (3).

$$A(t) = A_m\cos(\omega t + \varphi) = \sqrt{2}\,A_{eff}\cos(\omega t + \varphi) \tag{3}$$

Here, $A_m$, $A_{eff}$, ω and φ correspond to amplitude, effective value, angular frequency and phase, respectively.
The phasor A, which is a complex quantity, that corresponds to the signal A(t) is defined as below:
(Since the multimeters measure effective values, the phasor expression below also uses effective value term.)

$$A = A_{eff}\, e^{j\varphi} \tag{4}$$

Using (3) and (4),

$$A_{eff}\cos(\omega t + \varphi) = \mathrm{Re}[A\, e^{j\omega t}] \tag{5}$$

can be found.

Differentiating both sides in (5) and employing linearity properties of Re and $\frac{d}{dt}$ operators, one can reach to eq. (6).

$$\frac{A_{eff}}{A_m}\frac{d}{dt}A(t) = \mathrm{Re}[j\omega * A\, e^{j\omega t}] \tag{6}$$

It can be deduced from eq. (6) that if Phasor X stands for x(t), than jω*X stands for dx/dt.

Thus, a transformation is done from the derivative operator to an algebric expression. By this way, solving an ordinary differential equation set in time domain is reduced to solving a linear equation set in phasor domain that involves complex numbers.
On the other hand, it is easy to show that network theorems are also valid in phasor domain which means that the analysis of the circuits can also be done in phasor domain.

As a conclusion, it is a simpler task to analyze circuits and find sinusoidal steady state solutions in phasor domain than analyzing the circuits in time domain and solving differenatial equations.

**The Concept of Impedance and Admittance in RLC Circuits**

The impedance and admittance concepts are widely used during circuit analysis in phasor domain. Impedance is the ratio of (port) voltage to (port) current of a one port element in phasor domain.

Utilizing eq. (6), V-I relations of L and C elements are given as below,

$$V_L = j\omega L * I_L \quad , \quad I_C = j\omega C * V_C \tag{7}$$

Thus the impedances of L and C elements are defined as

$$\frac{V_L}{I_L} = Z_L = j\omega L \quad , \quad \frac{V_C}{I_C} = Z_C = \frac{1}{j\omega C} \tag{8}$$

Admittance is also defined as Y = 1/Z.

Since L and C elements can be considered as resistors with complex values in phasor domain, steady state analysis of an RLC circuit in phasor domain is equivalent to resistive circuit analysis in time domain.

**Remarks on Complex Numbers**

The complex number $V = a + jb$ has the amplitude $|V| = \sqrt{a^2 + b^2}$ and phase $\phi(V) = \arctan(b/a)$. In other words, V complex number can be expressed as $V = |V|e^{j\phi}$. This representation makes calculations easier in some cases. For example, it is easy to show that $\left|\frac{V_1}{V_2}\right| = \frac{|V_1|}{|V_2|}$ and $\phi\left(\frac{V_1}{V_2}\right) = \phi(V_1) - \phi(V_2)$.

During the experiment, the value you measure with the multimeters will be the amplitude of the phasor V, i.e. $|V|$.

**Procedure**

**1.** Build the circuit in fig. 6.1. Using the signal generator, generate an e(t) sinusoid which has an amplitude of 5V and frequency of 1kHz. Set the element values as R = 1kΩ and C = 100nF.

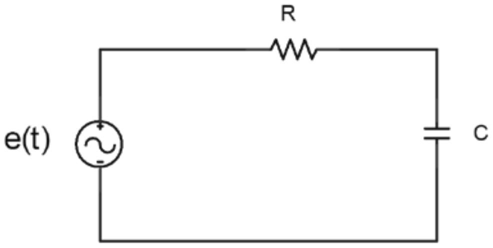



Figure 6.1

**a)** Measure the amplitudes $|V_C|$ and $|V_R|$. Obtain $|I_C|$ using the expression of $|V_R|$. Calculate the value of C from the ratio of $\frac{|I_C|}{|V_C|}$.

**b)** Write down the parametric expression of the phasor $V_C$ (in terms of E, R,C and ω). Write down the expression of $|V_C|$ in the same way and using the element values, calculate the numeric value of $|V_C|$ and compare the result with the measured value.

**c)** Find a relationship between $|V_C|$, $|V_R|$ and E.

**2.** Build the circuit in fig. 6.2 and repeat the same steps in 1.

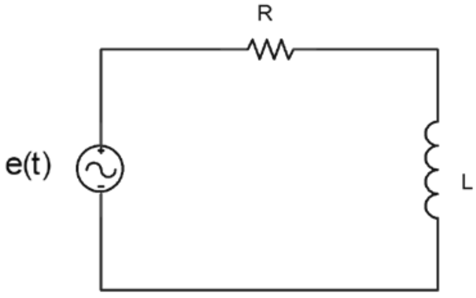



Figure 6.2

**3.** Observe e(t) and $v_L(t)$ signals simultaneously on the oscilloscope screen for the circuit in fig. 6.2.

**a)** Make use of the scales of the oscilloscope screen and calculate an approximate value for the phase difference of these two signals.

**b)** Employing $\frac{V_L}{E}$ expression, find the analytical expression of phase difference between e(t) and $v_L$(t). Calculate the numeric value and compare with the estimated one.

**c)** What is the phase difference between $v_R$(t) and e(t), $v_R$(t) and $v_L$(t)?

**4.** Build the circuit in fig. 6.3.

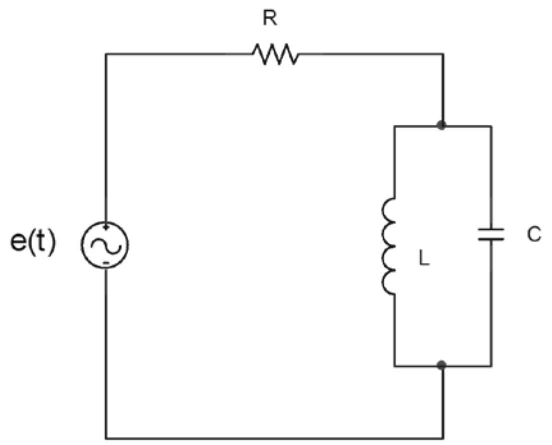



Figure 6.3

**a)** Sweep the frequency of e(t) input signal and determine the frequency at which the voltage drop on L//C terminals gets maximum. Draw an approximated graph of voltage vs frequency.

**b)** Write down the expression of ω dependent impedance, $Z_{L//C}$ . Find an equation for the frequency where this impedance gets maximum. Calculate the frequency value and compare it with the measured one.