EXPERIMENT NO 1

Objectives

The objectives of Experiment 1 are

- to learn how to read resistors by color bands,
- to learn how to build circuits on a breadboard,
- to learn how to measure resistance values using a multimeter and to calculate equivalent circuit resistance,

A. BACKGROUND

Resistors

A component that is specifically designed to have a certain amount of resistance is called a resistor. The principal applications of resistors are to limit current in a circuit, to divide voltage, and, in certain cases, to generate heat. Although resistors come in many shapes and sizes, they can all be placed in one of two main categories: fixed or variable.

Fixed Resistors

Fixed resistors are available with a large selection of resistance values that are set during manufacturing and cannot be changed easily. They are constructed using various methods and materials. Figure 1 shows several common types.

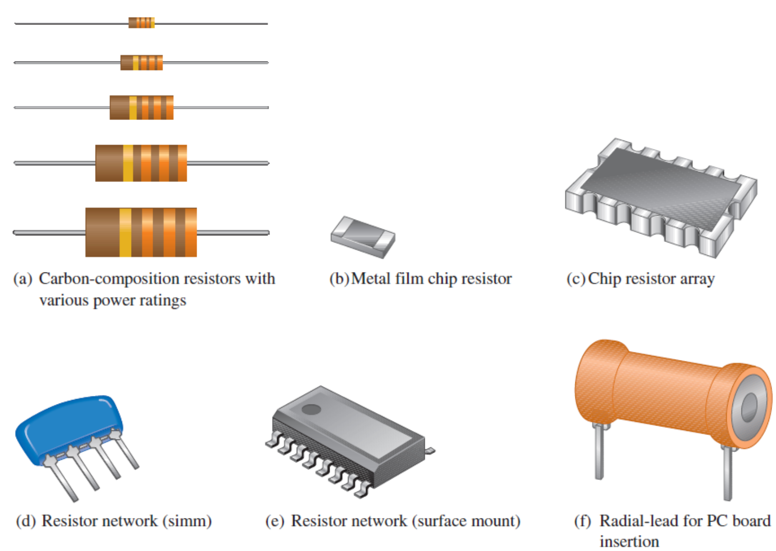


Figure 1. Typical Fixed Resistors

Resistor Color Codes

Fixed resistors with value tolerances of 5% or 10% are color coded with four bands to indicate the resistance value and the tolerance. This color-code band system is shown in Figure 2, and the color code is listed in Table 1. The bands are always closer to one end.

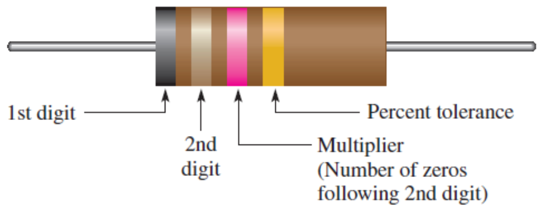


Figure 2. Color-code bands on a 4-band resistor.

The color code is read as follows:

1. Start with the band closest to one end of the resistor. The first band is the first digit of the resistance value. If it is not clear which is the banded end, start from the end that does not begin with a gold or silver band.
2. The second band is the second digit of the resistance value.
3. The third band is the number of zeros following the second digit, or the multiplier.
4. The fourth band indicates the percent tolerance and is usually gold or silver. If there is no band, it means 20% tolerance.
For example, a 5% tolerance means that the *actual* resistance value is within ±5% of the color-coded value. Thus, a resistor with a tolerance of ±5% can have an acceptable range of values from a minimum of 95 *Ω* to a maximum of 105 *Ω*.

Example: 470 k*Ω* → Yellow, Violet, Yellow

Table-1 Resistor 4-band color code

	Digit	Color
Resistance value, first three bands: First band—1st digit Second band—2nd digit Third band—multiplier (number of zeros following the 2nd digit)	0	Black
	1	Brown
	2	Red
	3	Orange
	4	Yellow
	5	Green
	6	Blue
	7	Violet
	8	Gray
	9	White
Fourth band—tolerance	±5%	Gold
	±10%	Silver

Breadboard

A breadboard holds circuit components in place and connects them electrically. A breadboard is shown in Figure 3. The breadboard has many strips of metal that run underneath the plastic top. The metal strips are arranged as shown in the Figure 3. These strips connect to the holes on top of the board. This makes it easy to connect components together when building a circuit.

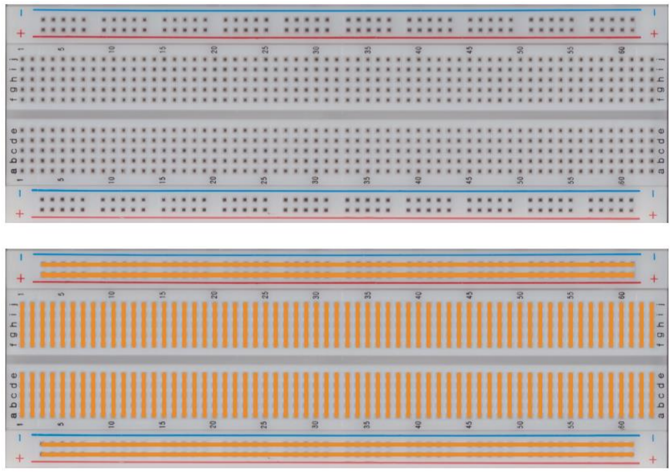


Figure 3. Breadboard and breadboard connection pattern

In Figure 4, schematic of a circuit is given, and it is shown how to build it on the breadboard.

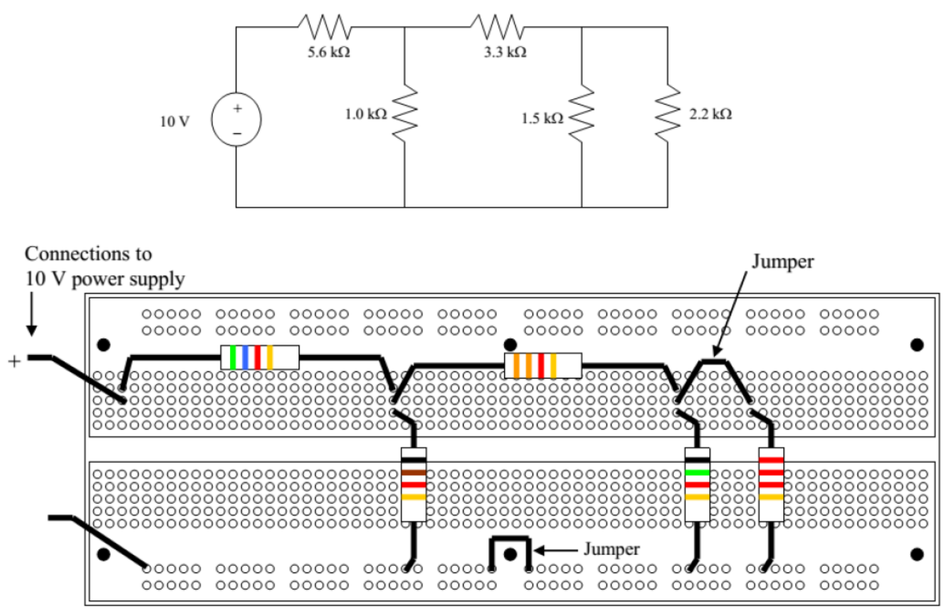


Figure 4.

Multimeter:

The multimeter, also called a volt-ohm meter (or VOM), is the basic tool for anyone working in electronics. You can see a fairly typical modern multimeter in Figure 5. You use a multimeter to take a variety of electrical measurements — hence the term "multi."

With this one tool, you can:

- Measure AC voltages
- Measure DC voltages
- Measure resistance
- Measure current going through a circuit
- Measure continuity (whether a circuit is broken or not)

Figure 5. A simple multimeter

Voltmeter:

A voltmeter measures electrical potential between its terminals. Voltmeters are always placed in parallel with the circuit or circuit element where the voltage measurement is desired. Since the voltage across two or more parallel elements is the same, the voltage measured by the meter will be the same as the element to which the meter is connected. When using a non-auto-ranging meter, select the highest possible range and reduce the range as necessary until the desired level of accuracy is reached. Always start with a range higher than the expected value to prevent damage to the meter.

Ammeter:

An ammeter measures the current that flows between its terminals. An ammeter is always placed in series with the circuit or circuit element where the current flow is of interest. Since the current in each element of a series circuit is the same, the current flow through the meter will be the same as the current flow to the element of interest. Never connect an ammeter in parallel unless you intend to measure the short circuit current of a circuit or circuit element and you have made sure that destructive current levels won't be reached. When using a non-auto-ranging meter, select the highest possible range and reduce the range as necessary until the desired level of accuracy is reached. Always start with a range higher than the expected value to prevent damage to the meter.

Ohmmeter:

An ohmmeter measures the electrical resistance between its terminals. An ohmmeter is connected to the circuit or circuit element of interest after the element of interest has been isolated from the rest of the circuit. The element of interest has to be isolated from the rest of the circuit so that its resistance value isn't obscured by the resistance values of the other circuit components connected to the element of interest. Never connect an ohmmeter to an energized circuit or the meter could be destroyed.

Measurement Errors:

Absolute Error

In general, the result of any measurement of physical quantity must include both the value itself and its error. The result is usually quoted in the form

$\pm\Delta X = X_0 - X_{measured}$

where X_0 is the best estimate of what we believe is a true value of the physical quantity and ΔX is the estimate of absolute error (uncertainty). ΔX indicates the reliability of the measurement, but the quality of the measurement also depends on the value of X_0.

Fractional Error

Fractional error is defined as $\frac{\Delta X}{X_0}$

Fractional error can be also represented in percentile form: $\frac{\Delta X}{X_0} \times 100$

B. PRELIMINARY WORK:

1. Study how to read resistor values using color bands.
2. Study working principles of voltmeter, ammeter and ohmmeter.
3. Study how to use multimeter which you purchased. Read its user manual.
4. Study how to connect circuit components in series and parallel on breadboard.
5. The values of resistors are given in Table-2. Write the corresponding color bands into related fields in this table.

Table-2	The resistance	1st color	2nd color	3rd color	4th color
	560 Ω ± 10%				
	20 kΩ ± 5%				
	33 Ω ± 20%				
	47 kΩ ± 5%				
	1 kΩ ± 20%				

6. The color bands are given in Table-3. Write the corresponding resistors into related fields in this table.

Table-3	1st color	2nd color	3rd color	4th color	The resistance
	Brown	Grey	Black	No band	
	Green	Blue	Violet	Silver	
	Yellow	Violet	Brown	Gold	
	White	Green	Black	Gold	
	Red	White	Orange	No band	

7. Calculate equivalent resistance of $R_{a\text{-}b}$ for Fig. 7, Fig. 8, and Fig. 9. Write your results into related fields in Table-4.

C. EXPERIMENTAL PART:

1. Build the circuit given in Fig. 7 on breadboard. Measure equivalent resistance seen between a and b terminals using multimeter and write the result into Table-4. Repeat this process for Fig. 8 and Fig. 9 and compare measured and calculated equivalent resistance values. Calculate absolute and % errors for each one and write them in Table-4.

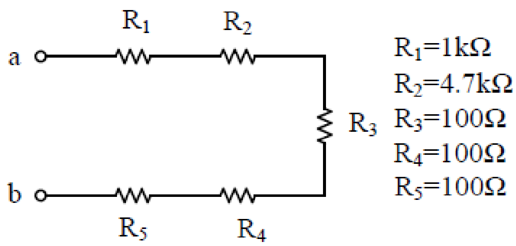


Figure 7. Voltage Divider Circuit

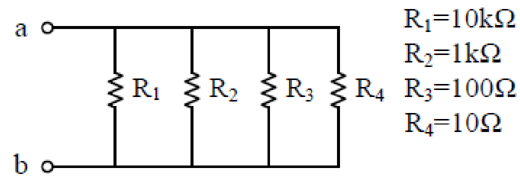


Figure 8. Current Divider Circuit

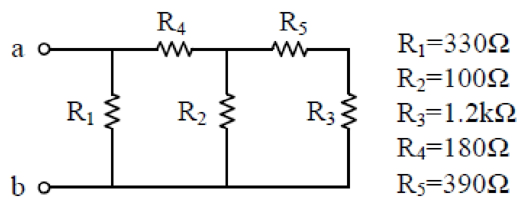


Figure 9. Ladder Circuit

Table-4	**Circuit**	**R_{ab} [*Ω*]**
		Calculated
	Figure 7	
	Figure 8	
	Figure 9	

EXPERIMENT NO 2

Objectives

The objectives of Experiment 2 are

- to learn how to build circuits on a breadboard,
- to learn how to measure voltages and currents using multimeter
- to verify Kirchhoff's Laws

A. BACKGROUND

Ohm's law states that current is directly proportional to voltage and inversely proportional to resistance. Ohm's law is given in the following formula:

$$I = \frac{V}{R}$$

where *I* is current in amperes (A), *V* is voltage in volts (V), and *R* is resistance in ohms

Kirchhoff's Voltage Law:

Kirchhoff's voltage law is a fundamental circuit law that states that the algebraic sum of all the voltages around a single closed path is zero or, in other words, the sum of the voltage drops equals the total source voltage.

Kirchhoff's voltage law applied to a series circuit is illustrated in Figure 1. For this case, Kirchhoff's voltage law can be expressed by following equation:

$V_S = V_1 + V_2 + V_3 + ... + V_n$

where the subscript *n* represents the number of voltage drops.

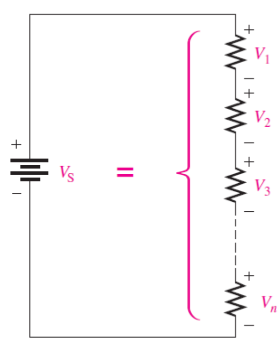


Figure 1. Sum of n voltage drops equals to the source voltages

If all the voltage drops around a closed path are added and then this total is subtracted from the source voltage, the result is zero. This result occurs because the sum of the voltage drops always equals the source voltage.

Therefore, another way of expressing Kirchhoff's voltage law in equation form is

$V_S - V_1 - V_2 - V_3 - ... - V_n = 0$

Kirchhoff's Current Law:

Kirchhoff's current law, often abbreviated KCL, can be stated as follows:

The sum of the currents into a node (total current in) is equal to the sum of the currents out of that node (total current out).

A node is any point or junction in a circuit where two or more components are connected. In a parallel circuit, a node or junction is a point where the parallel branches come together. For example, in the circuit of Figure 1, point A is one node and point B is another. Let's start at the positive terminal of the source and follow the current. The total current I_T from the source is into node A. At this point, the current splits up among the three branches as indicated. Each of the three branch currents (I_1, I_2 and I_3) is out of node A. Kirchhoff's current law says that the total current into node A is equal to the total current out of node A; that is,

$$I_T = I_1 + I_2 + I_3$$

Now, following the currents in Figure 1 through the three branches, you see that they come back together at node *B*. Currents I_1, I_2, and I_3 are into node *B*, and I_T is out of node *B*. Kirchhoff's current law formula at node *B* is therefore the same as at node *A*.

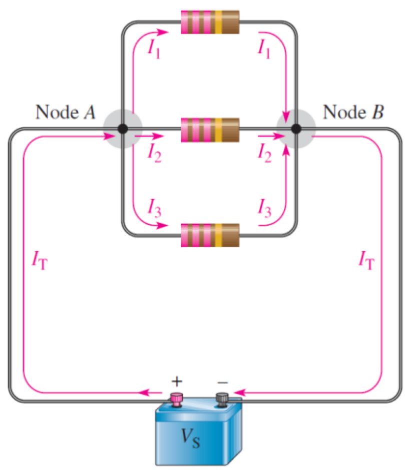


Figure 2. Kirchhoff's Current Law

Figure 2 illustrates the general case of Kirchhoff's current law and can be written as a mathematical relationship:

$$I_{IN(1)} + I_{IN(2)} + I_{IN(3)} + \dots + I_{IN(n)} = I_{OUT(1)} + I_{OUT(2)} + I_{OUT(3)} + \dots + I_{OUT(m)}$$

Moving the terms on the right side to the left side and changing the sign results in the following equivalent equation:

$$I_{IN(1)} + I_{IN(2)} + I_{IN(3)} + \dots + I_{IN(n)} - I_{OUT(1)} - I_{OUT(2)} - I_{OUT(3)} - \dots - I_{OUT(m)} = 0$$

This equation shows that all current into and out of the junction sums to zero and can be stated as "**The algebraic sum of all of the currents entering and leaving a node is equal to zero".**

An equivalent way of writing Kirchhoff's current law can be expressed using mathematical summation shorthand.

$$\sum_{i=1}^{n} I_i = 0$$

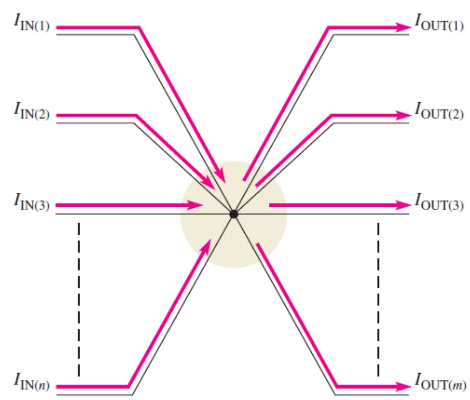


Figure 3. Generalized circuit node illustrating Kirchhoff's current law.

To express Kirchhoff's current law with this method, all currents are assigned a sequential subscript (1, 2, 3, and so on) no matter if the current is into or out of the node. Currents into the node are positive and currents leaving the node are negative.
You can verify Kirchhoff's current law by connecting a circuit and measuring each branch current and the total current from the source. When the branch currents are added together, their sum will equal the total current. This rule applies for any number of branches.

How to measure current using digital multimeter (DMM):

Step 1. Turn on the DMM. Most digital systems need a waiting period before they are ready for use. Just allow a minute for the device.

Step 2. Set the dial to the proper setting to read amps. To measure alternating current in amps, use the setting "A~" or AAC; to measure direct current in amps, use the setting "A=" or ADC.

Step 3. Set the range higher than the maximum expected current reading. If you cannot read the current you might overshoot the range, then set the dial to lower range. Thus, you can protect the DMM from getting overloaded.

Step 4. Connect the black plug to the common terminal or "COM". Connect the red plug to a terminal marked for low (200mA) or high (10A) current measuring.

Step 5. Break into the circuit to measure its current. Connect the DMM in series with the circuit as shown in Figure 3. The red probe will be on one open end and the black probe will be on the other. Hence, the current will flow through the DMM and it will be possible to read current.

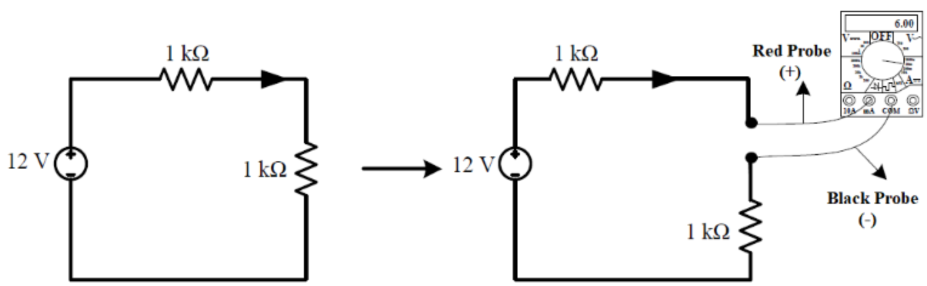


Figure 4

B. *PRELIMINARY WORK:*

a. *Theoretical Part*

i. Calculate all node voltages in Fig. 5 using node voltage method and write your results on Table-1.

ii. Determine voltages across each component in Fig. 5 using calculated node voltages in the previous question and write them into related fields on Table-2.

iii. Show that Kirchhoff's Voltage Law is satisfied for all meshes in Fig. 5 on related fields in Table-3.

iv. Calculate all mesh currents and write them in the related fields on Table-4.

v. Determine all resistor currents using calculated mesh currents in previous question and write them into related fields in Table-5.

vi. Show that Kirchhoff's Current Law is satisfied for all nodes in Fig. 5 on related fields in Table-6

b. *Simulation Part*

i. Simulate the circuit on OrCAD and fill all related fields in Tables 1-6.

Instructions for preliminary work report:

1. Clearly show each step of your theoretical work in your report. You can solve the question on paper and add pictures. Make sure your writing is legible and your steps are clear to follow.
2. Using OrCAD PSpice, build each circuit schematic and add them to your report with simulation results for all circuits in the order of experimental part. Take screenshots for your schematic and simulation results and insert them into your report. They must be neat and readable.
3. If any column is available in tables for simulation results, fill the related fields and add them to your report.

This preliminary work report is of prime importance to check your results in experiment.

C. *EXPERIMENTAL PART*

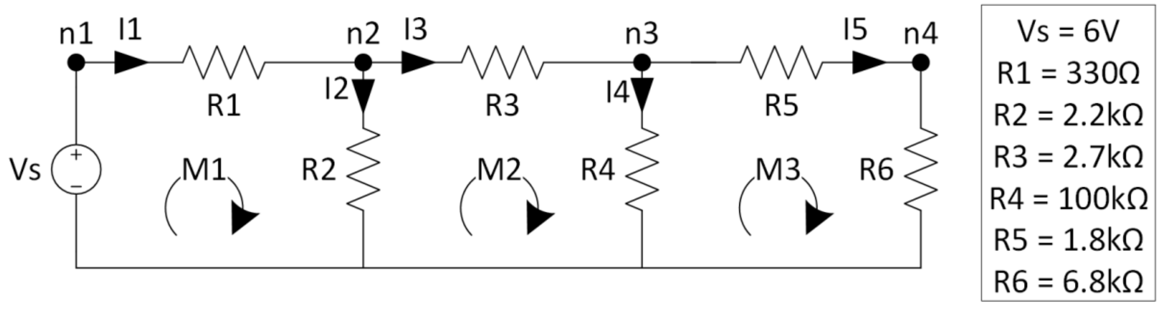


Figure 5

1. Build the circuit given in Figure 5 on the breadboard.
2. Measure the voltages at each node and across each resistor using your multimeter and write them in Table-2.
3. Show that Kirchhoff's Voltage Law is satisfied at each mesh.
4. Measure the currents across each resistor and each mesh.

5. Show that Kirchhoff's Current Law is satisfied at each node.

EXPERIMENTAL WORK 2:

Table-1

Voltages at	Calculated [V]	Simulated [V]
n_1		
n_2		
n_3		
n_4		

Table-2

Voltages across	Calculated [V]	Simulated [V]
R_1		
R_2		
R_3		
R_4		
R_5		
R_6		

Table-3

Calculated	M1	
	M2	
	M3	
Simulated	M1	
	M2	
	M3	

Table-4

Current at	Calculated [mA]	Simulated [mA]
M1		
M2		
M3		

Table-5

Current across	Calculated [mA]	Simulated [mA]
R_1		
R_2		
R_3		
R_4		
R_5		
R_6		

Table-6	Calculated	n1	
		n2	
		n3	
		n4	
	Simulated	n1	
		n2	
		n3	
		n4	

EXPERIMENT NO 3

OBJECTIVES

The objectives of Experiment 3 are

- to develop circuit construction skills,
- to develop dc circuit voltage and current measurement skills,
- to verify the Thevenin and Norton theorems,
- to apply experimentally the principle of superposition in the laboratory.

A. BACKGROUND

The Linearity Property:

A linear element or circuit satisfies the properties of additivity and homogeneity.

Additivity:
Additivity requires that the response to a sum of inputs is the sum of the responses to each input applied separately.

If $V_1 = i_1.R$ and $V_2 = i_2.R$

then applying $(i_1 + i_2)$

$V = (i_1 + i_2).R = i_1.R + i_2.R = V_1 + V_2$

Homogeneity:
If you multiply the input (i.e. current) by some constant K, then the output response (voltage) is scaled by the same constant.

If $V_1 = i_1.R$

then $K.V_1 = K.i_1.R$

The Superposition Theorem:

The superposition method is a way to determine currents in a circuit with multiple sources by leaving one source at a time and replacing the other sources by their internal resistances. Recall that an ideal voltage source has a zero internal resistance and an ideal current source has infinite internal resistance. All sources will be treated as ideal in order to simplify the coverage.

A general statement of *the superposition theorem* is as follows:

The current in any given branch of a multiple-source circuit can be found by determining the currents in that particular branch produced by each source acting alone, with all other sources replaced by their internal resistances. The total current in the branch is the algebraic sum of the individual currents in that branch.

The steps in applying the superposition method are as follows:

Step 1: Leave one voltage (or current) source at a time in the circuit and replace each of the other voltage (or current) sources with its internal resistance. For ideal sources a short represents zero internal resistance and an open represents infinite internal resistance.

Step 2: Determine the particular current (or voltage) that you want just as if there were only one source in the circuit.

Step 3: Take the next source in the circuit and repeat Steps 1 and 2. Do this for each source.

Step 4: To find the actual current in a given branch, algebraically sum the currents due to each individual source. Once you find the current, you can determine the voltage using Ohm's law.

Thevenin's Theorem:

A two–terminal network can be replaced by a voltage source with the value equal the open circuit voltage across its terminals, in series with a resistor with the value equal to the equivalent resistance of the network.

Norton's Theorem:

A two terminal network can be replaced by a current source with the value equal to the short–circuit current at its terminal, in parallel with a resistor with the value equal to the equivalent resistance of the network. The equivalent resistance of a two–terminal network is equal to the open circuit voltage divided by the short circuit current.

B. *PRELIMINARY WORK :*

1. Find the Thevenin and Norton equivalent circuits w.r.t terminals a and b for the circuit given in Fig.1

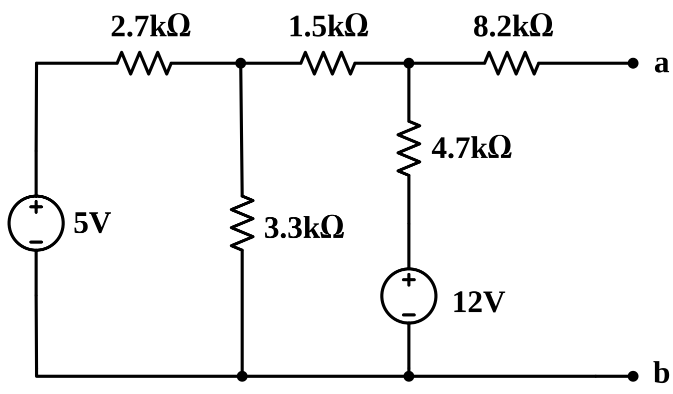


Figure 1

2. Calculate V_{ab} , I_{ab} when V_{in} = 3V, 5V, 7V, 9V, and 12V for the circuit given in Fig.2. Plot V_{in} vs. V_{ab} and I_{ab} vs. V_{ab} in MATLAB and calculate the slopes.

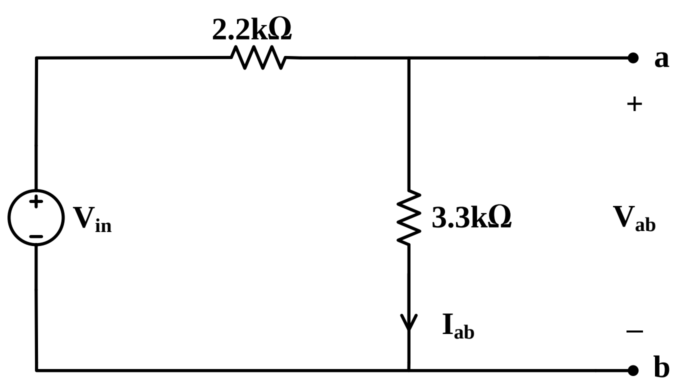


Figure 2

3. Find V_{ab} and $I_{1k\Omega}$ given in Fig.3 by using the superposition principle.

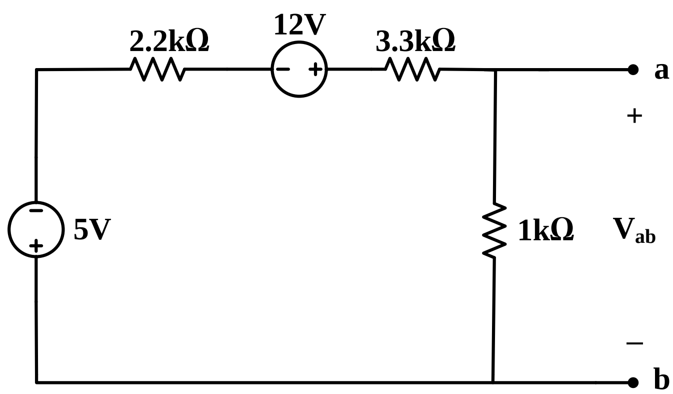


Figure 3

Instructions for preliminary work report:

1. Using OrCAD PSpice, build each circuit schematic and add them to your report with simulation results for all circuits in the order of experimental part. Give all the requested results in experimental part as graphics and/or tables.
2. If any column is available in tables for calculated results, fill the related fields and add them to your report.

This preliminary work report is of prime importance to check your results in experiment.

C. EXPERIMENTAL PART:

1. Build the circuit shown in Fig.1
   a) Connect your multimeter between a and b, and measure the voltage V_{ab}
   b) Connect your multimeter between a and b, and measure the current I_{ab}.
   c) Replace voltage sources with s/c (DO NOT SHORT ACROSS THE TERMINALS OF THE POWER SUPPLY) and then, measure the equivalent resistance between a and b.

2. Build the circuit in Fig. 2, and measure the voltage V_{ab} and the current I_{ab} for V_{in} = 3V, 5V, 7V, 9V, and 12V .

3. Build the circuit in Fig.3.
   a) Connect only voltage source of 5V and measure V_{ab} and $I_{1k\Omega}$.
   b) Connect only voltage source of 12V and measure V_{ab} and $I_{1k\Omega}$.
   c) Connect both of the sources at the same time and measure V_{ab} and $I_{1k\Omega}$.

D. QUESTIONS :

1. Compare your theoretical and practical Thevenin and Norton equivalent circuits in experimental part 1 and comment on differences.
2. By using measured values in experimental part 2, sketch V_{ab} vs. V_{in} , V_{ab} vs. I_{ab} and calculate the slopes. What are the meanings of these slopes?
3. By using measured values in experimental part 3, prove the superposition principle.

EXPERIMENTAL WORK 3

Students' Names & Numbers:

Table 1

V_{ab} (V)	I_{ab} (mA)	R_{ab} (Ω)

Table 2

V_{in} (V)	V_{ab} (V)	I_{ab} (mA)
3		
5		
7		
9		
12		

Table 3

V_{ab}' (V)	$I_{1k\Omega}$' (mA)	V_{ab}'' (V)	$I_{1k\Omega}$'' (mA)	V_{ab} (V)	$I_{1k\Omega}$ (mA)

ANSWERS:

1.

2.

3.

EXPERIMENT NO: 4

Objectives

The objectives of Experiment 4 are

- to learn how to use an oscilloscope,
- to learn measuring amplitude, period, and frequency of an electrical signal using oscilloscope.
- to construct circuits using operational amplifier.

A. BACKGROUND

Signal Generator:

A signal generator is an electronic instrument that generates repeating voltage waveforms. An ideal signal generator can simply be modeled as a voltage source as shown in Fig.5.1 (a).

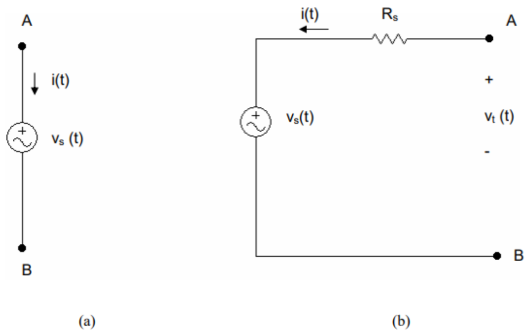


Figure 5.1

where $V_s(t)$ is a specified function of time. A practical signal generator is modeled as an ideal signal generator connected to a series source resistance (output resistance) R_s as shown in Fig.5.1 (b). The terminal voltage, v(t), is the output of the signal generator and depends on the terminal current, i(t), and R_s. $V_s(t)$ can be, in general, a sine wave, a square wave, a triangular wave or a pulse train. The first three are characterized by three parameters: frequency (or period), amplitude, and DC (Offset) value. The pulse train is associated with frequency, amplitude and pulse duration. These parameters can be set to any value in the operation range of the signal generator, using the external controls. In general, amplitude ranges of signal generators vary from 10 mV to 20 V, and frequency ranges vary from 1µHz to 40 MHz. This means signals for which the amplitude and frequency can be set to any value in these ranges can be generated using these signal generators. Signal generators usually produce more than one type of signals. Different signal types can be obtained by proper connection and/or switching. A simplified block diagram of a sine and square wave generator is given in Fig.5.2.

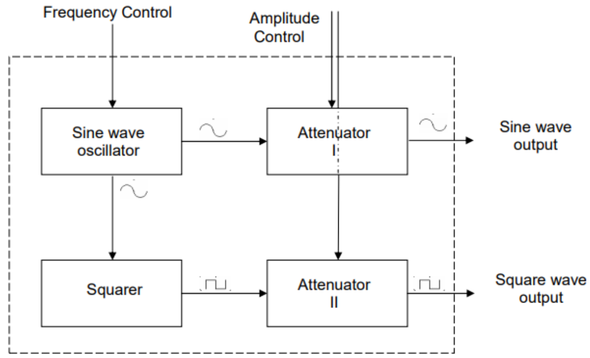


Figure 5.2

A sine wave oscillator is the heart of the signal generator. It generates a sine wave of fixed amplitude and adjustable frequency, which is set by the external frequency control. This signal is fed to both attenuator I and squarer. The signal amplitude is set to the desired value determined by external controls, by attenuator I. The output of attenuator I is a sine wave with desired amplitude and frequency. The squarer generates a square wave of fixed amplitude and at the same frequency with the sine wave. The output of the squarer is fed to attenuator II which acts similar to attenuator I. The output is the square wave with amplitude and frequency which are determined externally.

Oscilloscope:

This is one of the most important pieces of laboratory test equipment. It is basically a voltage sensing and display device; it cannot measure current directly. However, it can be used to measure a voltage proportional to a desired current, e.g., across a small sampling resistance. Most modern scopes have two input channels as shown in Fig.5.3. Two signals can thus be viewed separately, or simultaneously if they are synchronized. Calibrated gain settings enable the measurement of voltage amplitude. A horizontal time axis is provided by an internal generator. This generator produces a calibrated variable-frequency voltage the amplitude of which varies linearly with time. Thus, a voltage waveform applied to either input channel can be viewed as a function of time. And a plot of the relationship between two signals at both channels can also be performed. An important scope function is the Trigger. Circuits in this subsection enable the selection of the amplitude of the input signal at t = 0 relative to its peaks. This corresponds to having a selectable phase angle. Another important scope function is applying a mathematical operation on the signal, such as inverting, add the two signals, and subtract them.

Figure 5.3

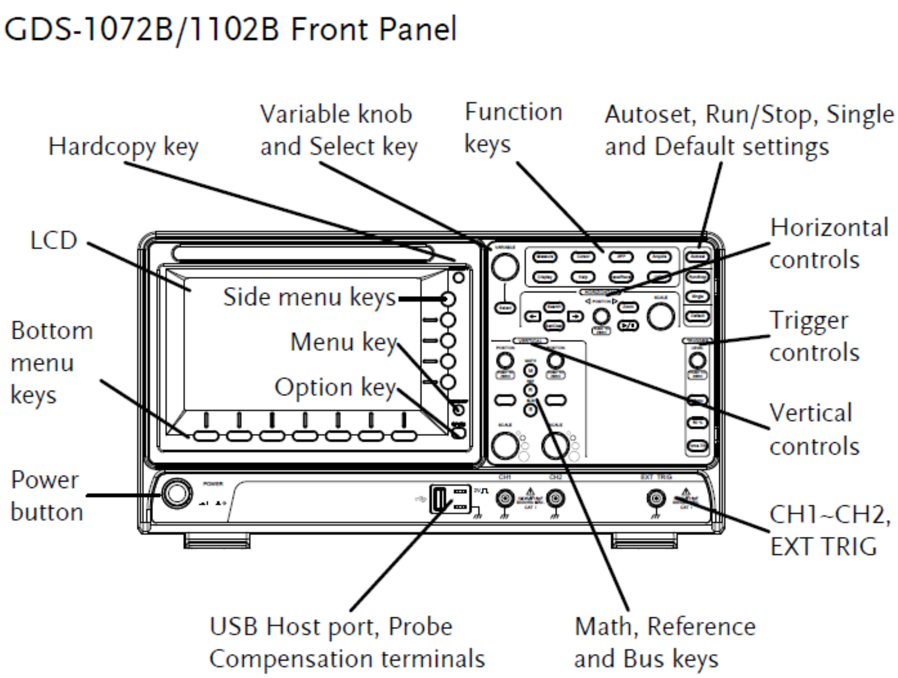


Figure 5.5

The front panel of the oscilloscope is as shown in Fig.5.5. The buttons, knobs, and important functions are shown in this figure. The details can be found in Appendix B.

Operational Amplifier Characteristics:

Operational amplifier (OPAMP) is a high gain voltage amplifier with differential inputs. The input resistance is very large. The block diagram of the OPAMP and is given on Fig. 5.5.

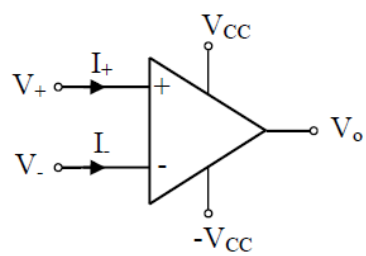


Figure 5.5

The inputs V+ and V- are the differential inputs. V_O is the output voltage and V_O is proportional to the difference voltage V+ - V-. The voltages V_{CC} and $-V_{CC}$ are the DC voltages connected to operational amplifier for proper operation. Since the input resistance is very large, the input currents I+ and I- are assumed negligibly small., i.e.,

$I+ = I- \equiv 0$ A.

The output-input voltage transfer characteristics of the OPAMP is plotted on Fig. 5.6.

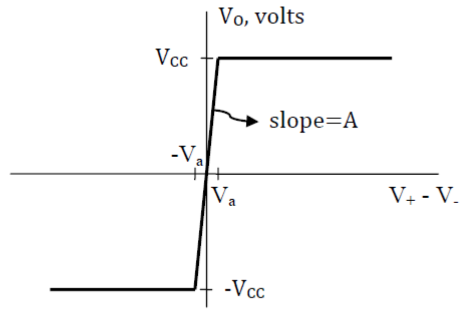


Figure 5.6

OPAMP works in linear region if $-V_{CC} \le V_O \le +V_{CC}$, otherwise it saturates.

Inverting Amplifier:

An inverting amplifier reverses the polarity of the input signal while amplifying it. The inverting amplifier circuit is as shown in Fig. 5.7.

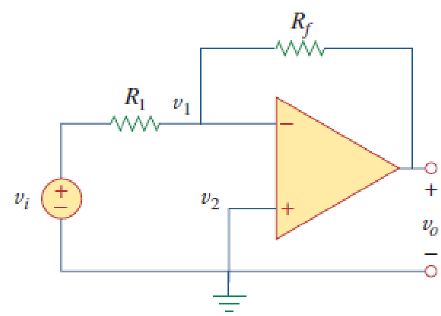


Figure 5.7

In this circuit, the noninverting input is grounded, v_i is connected to the inverting input through R_1, and the feedback resistor R_f is connected between the inverting input and output. The relationship between the input voltage v_i and the output voltage v_o is given in following equation.

$$V_O = -\frac{R_f}{R_1} V_i$$

Summing Amplifier:

A summing amplifier is an op-amp circuit that combines several inputs and produces an output that is weighted sum of the inputs. The summing amplifier, shown in Fig. 5.8, is a variation of the inverting amplifier. It takes advantage of the fact that the inverting configuration can handle many inputs at the same time.

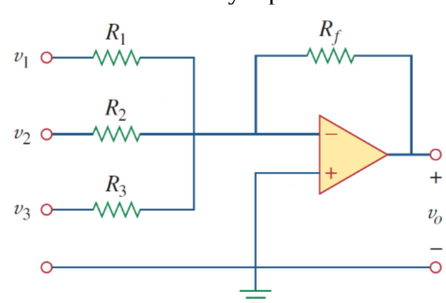


Figure 5.8

For the circuit given in Fig.5.8 the equation for v_o is as follows:

$$V_O = -(\frac{R_f}{R_1}V_1 + \frac{R_f}{R_2}V_2 + \frac{R_f}{R_3}V_3)$$

Comparator:

The Op-amp comparator compares one analogue voltage level with another analogue voltage level, or some preset reference voltage, V_{REF} and produces an output signal based on this voltage comparison. Consider an example op-amp voltage comparator circuit given in Fig.5.9.

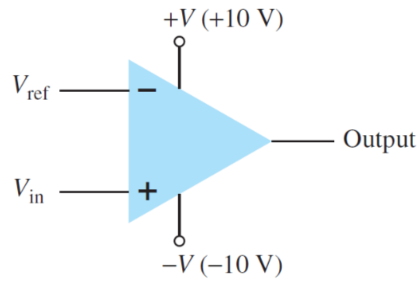


Figure 5.9

The working principle of this comparator is as follows:

If $V_{in} > V_{ref}$ then $V_{out} = +V_{CC}$

If $V_{in} < V_{ref}$ then $V_{out} = -V_{CC}$

B. PRELIMINARY WORK:

1. Study the datasheet of LM751 given in Appendix A.
2. Study the document of oscilloscope given in Appendix B.
   (No need to give summary for the above questions in the pre-lab report. Just study!)
3. Do theoretical calculations, find the relation between inputs and outputs. Run simulations for the circuits given in experimental part.
   Hint 1: Use DC sweep in the simulation for the first circuit.
   Hint 2: Use transient simulation for the other circuits. Set run time 3 times of the largest period of the signals.
   Hint 3: Use VDC, VSIN, and VPULSE for the related circuits.
   Hint 5: Set amplitude for the VSIN as half of the peak-to-peak voltage. Set VOFF = 0 and AC=0.
   Hint 5: Set lower and upper voltages as 0 and 2V for the VPULSE, respectively. Set TD=0, TR=TF=1ns, and pulse width (PW) as half of the period (PER) for the square wave.

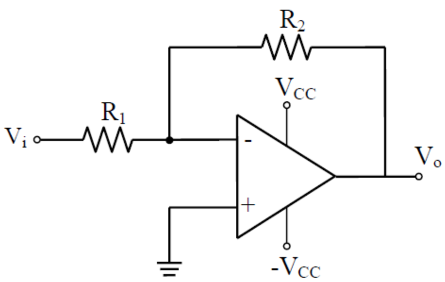
Instructions for preliminary work report:

1. Using OrCAD PSpice, build each circuit schematic and add them to your report with simulation results for all circuits in the order of experimental part. Give all the requested results in experimental part as graphics and/or tables.
2. If any column is available in tables for calculated results, fill the related fields and add them to your report.

   This preliminary work report is of prime importance to check your results in experiment.

C. EXPERIMENTAL PART:

1. Build the circuit given in Fig.5.10.



Circuit Parameters
$R_1 = 1\ k\Omega$
$R_2 = 10\ k\Omega$
$V_{CC} = 12\ V$
$-V_{CC} = -12\ V$

Figure 5.10

Measure output voltages for the input voltages given in Table 1 and calculate the gain for each one. Use a 1kHz square wave for the input. Then sketch the voltage transfer characteristic of the circuit.

2. Build the circuit given in Fig.5.11.
   a. Apply ±12V for $\pm V_{CC}$.
   b. Set V_1 (t) as $2V_{pp}$ (peak-to-peak voltage) square wave with frequency of 1kHz
   c. Set V_2 (t) as $2V_{pp}$ sine wave with frequency of 5kHz.
   d. Observe V_O (t) and comment.

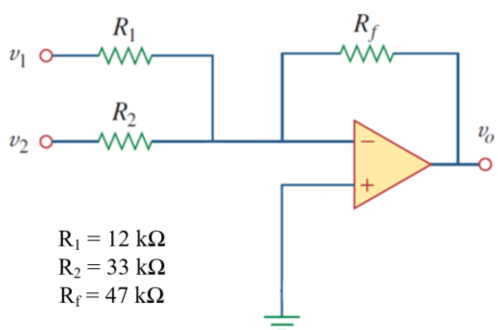


Figure 5.11

3. Build the circuit given in Fig.5.12.
   a. Apply ±10V for $\pm V_{CC}$.
   b. Apply 5V (V_{ref}) to inverting input of the opamp.
   c. Set V_{in} as $12V_{pp}$ (peak-to-peak voltage) sine wave with frequency of 1kHz.
   d. Observe the output and comment.

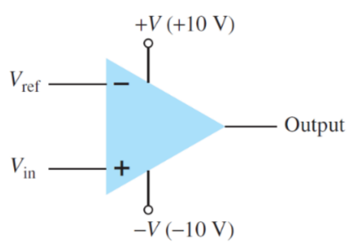


Figure 5.12

EXPERIMENTAL WORK 5

Students' Names & Numbers:

Table 1. Simulated and measured results for the circuit given in Fig. 5.10

$V_{i(p-p)}$ (V)	Simulation		Measurement	
	V_o (V)	Gain (V/V)	V_o (V)	Gain (V/V)
-2.5				
-2.0				
-1.5				
-1.0				
-0.5				
0.0				
0.5				
1.0				
1.5				
2.0				
2.5				

<u>Instructions:</u>

- **Sketch the voltage transfer characteristic of the circuit given in Fig. 5.10.**
- **Sketch the waveforms which you observed on the scope for the circuits given in Fig. 5.11 and Fig. 5.12.**
- **Write your comments below your sketches.**

EXPERIMENT NO: 5

Objectives

The objective of Experiment 5 is experimental verification of the time responses of RC, RL, and RLC circuits.

A. BACKGROUND

The details are given in the BME2301 Circuit Theory course notes.

B. PRELIMINARY WORK:

1. For Fig. 6.1, do your theoretical calculations and sketch the waveforms of $V_C(t)$ and $V_R(t)$ roughly. Compare your theoretical and simulated voltage waveforms, and comment on differences.
2. For Fig. 6.2, do your theoretical calculations and sketch the waveforms of $V_L(t)$ and $V_R(t)$ roughly. Compare your theoretical and simulated voltage waveforms, and comment on differences.
3. For Fig. 6.3, do your theoretical calculations and sketch the waveforms of $V_C(t)$ and $V_R(t)$ roughly. Compare your theoretical and simulated voltage waveforms, and comment on differences.

   (Note that in PSpice simulations, you will use voltage probe instead of oscilloscope and VPULSE instead of the box named as “Square Wave” for the given circuits in experimental part.)

Instructions for preliminary work report:

1. Using OrCAD PSpice, build each circuit schematic and add them to your report with simulation results for all circuits in the order of experimental part. Give all the requested results in experimental part as graphics and/or tables.
2. If any column is available in tables for calculated results, fill the related fields and add them to your report.

   This preliminary work report is of prime importance to check your results in experiment.

C. EXPERIMENTAL PART:

1. Build the circuit given in Fig.6.1. (R=1kΩ, C=100nF)
   a) Set period and peak-to-peak voltage of the square wave as $T = 10RC$ and $V_{p\text{-}p}$=10V. Obtain and sketch the waveforms.
   b) Set period and peak-to-peak voltage of the square wave as $T = RC$ and $V_{p\text{-}p}$=10V. Obtain and sketch the waveforms.
   c) Set period and peak-to-peak voltage of the square wave as $T = RC/10$ and $V_{p\text{-}p}$=10V. Obtain and sketch the waveforms.
   d) Comment on differences.

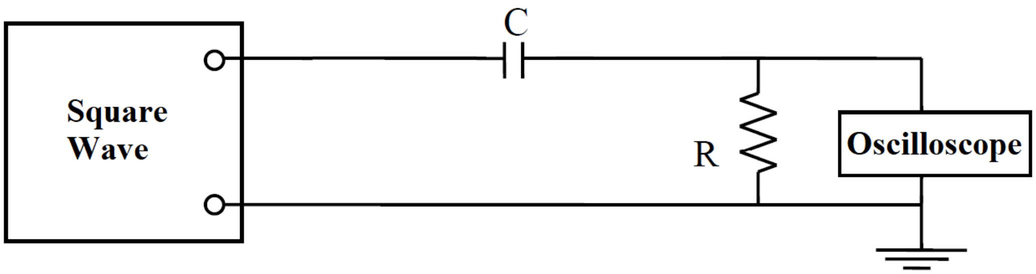


Figure 6.1

2. Build the circuit given in Fig.6.2. ($R=68\Omega$, $L=1mH$)

   a) Set period and peak-to-peak voltage of the square wave as $T = 10L/R$ and $V_{p-p}=10V$. Obtain and sketch the waveforms.
   b) Set period and peak-to-peak voltage of the square wave as $T = L/R$ and $V_{p-p}=10V$. Obtain and sketch the waveforms.
   c) Set period and peak-to-peak voltage of the square wave as $T = L/(10R)$,and $V_{p-p}=10V$. Obtain and sketch the waveforms.
   d) Comment on differences.

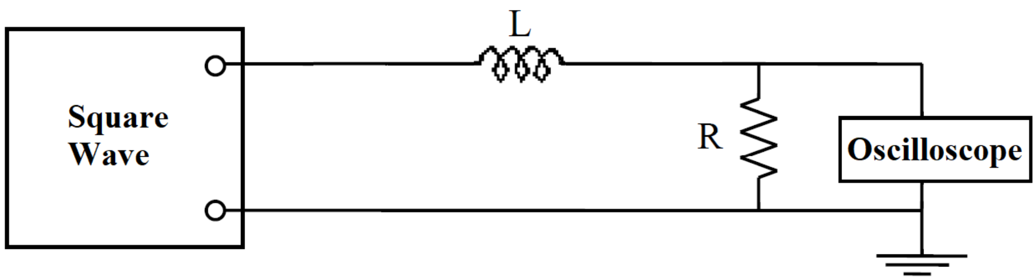


Figure 6.2

3. Build the circuit given in Fig.6.3. Set period and peak-to-peak voltage of the square wave as T=1ms, and $V_{p-p}=10V$.
   a. Obtain and sketch the waveforms for $R=2k\Omega$, C=1nF, and L=1mH.
   b. Obtain and sketch the waveforms for $R=68\Omega$, C=1nF, and L=1mH.
   c. Obtain and sketch the waveforms for $R=100k\Omega$, C=1nF, and L=1mH.
   d. Comment on differences.

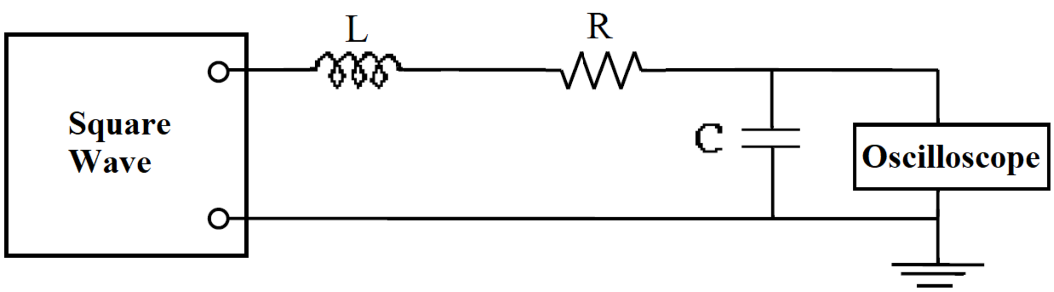


Figure 6.3

EXPERIMENTAL WORK 6

Students' Names & Numbers:

Instructions:

- **Sketch the waveforms which you observed on the scope for the circuits given in experimental part.**
- **Write your comments below your sketches.**

EXPERIMENTAL WORK 6

EXPERIMENT NO: 6

Objectives

The objective of Experiment 6 is experimental verification of the frequency responses of RC filters.

A. BACKGROUND

The details are given in the BME2301 Circuit Theory course notes.

B. PRELIMINARY WORK:

1. For Fig. 7.1, do your theoretical calculations and roughly sketch the frequency response (|H(jω)|) and phase ($\measuredangle(j\omega)$). Compare your theoretical and simulated results, and comment on differences.
2. For Fig. 7.2, do your theoretical calculations and roughly sketch the frequency response (|H(jω)|) and phase ($\measuredangle(j\omega)$). Compare your theoretical and simulated results, and comment on differences.
3. For Fig. 7.3, do your theoretical calculations and roughly sketch the frequency response (|H(jω)|) and phase ($\measuredangle(j\omega)$). Compare your theoretical and simulated results, and comment on differences.
4. Using MATLAB, plot the frequency response (|H(jω)|) and the phase ($\measuredangle(j\omega)$) for each circuit.

Instructions for preliminary work report:

1. Using OrCAD PSpice, build each circuit schematic and add them to your report with simulation results and MATLAB plots for all circuits in the order of experimental part. Give all the requested results in experimental part as graphics and/or tables.
2. If any column is available in tables for calculated results, fill the related fields and add them to your report.

This preliminary work report is of prime importance to check your results in experiment.

C. EXPERIMENTAL PART:

1. Build the circuit given in Fig.7.1. (R=1k*Ω*, C=10nF) Set the peak-to-peak voltage of the input as $V_{p\text{-}p}$=1V and measure Vo and phase at frequencies given in Table 1. Find the cut-off frequency for the filter. Fill the table and sketch the frequency response (in dB) and phase.

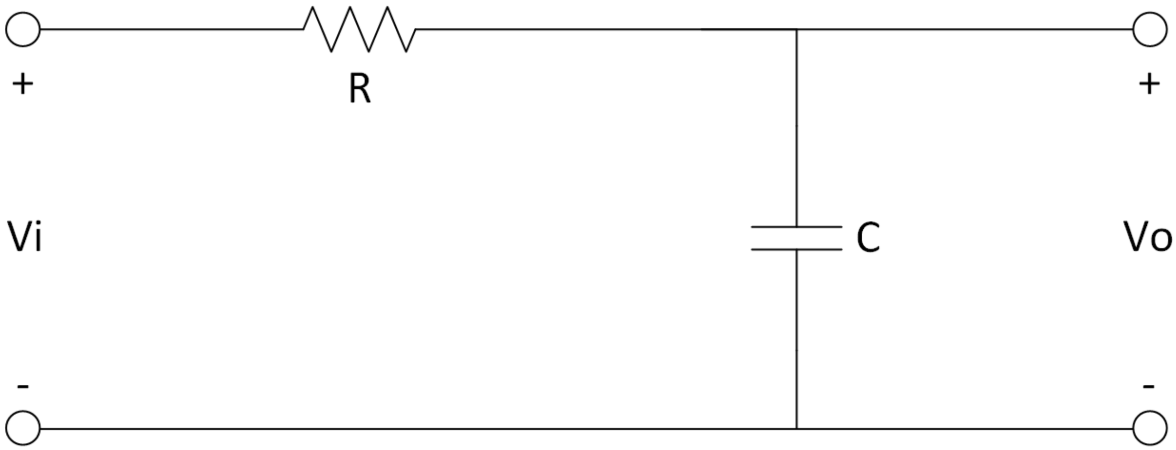


Figure 7.1

2. Build the circuit given in Fig.7.2. (R=47k*Ω*, C=10nF) Set the peak-to-peak voltage of the input as $V_{p\text{-}p}$=1V and measure Vo and phase at frequencies given in Table 2. Find the cut-off frequency for the filter. Fill the table and sketch the frequency response (in dB) and phase.

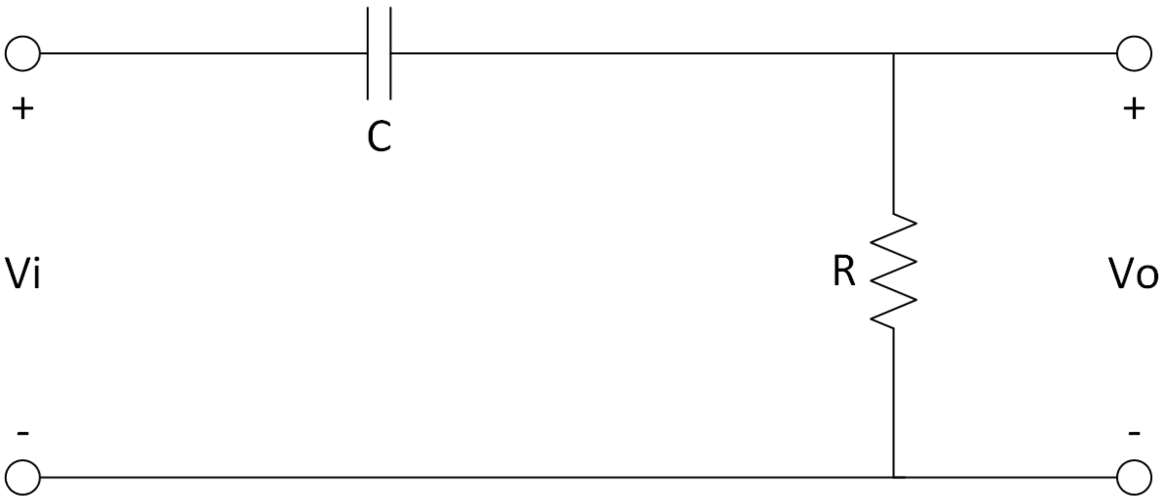


Figure 7.2

3. Build the circuit given in Fig.7.3. (R1=47kΩ, R2=1kΩ, C1=10nF, C2=10nF) Set the peak-to-peak voltage of the input as V_{p-p}=1V and measure Vo and phase at frequencies given in Table 3. Find the upper and lower cut-off frequencies for the filter. Fill the table and sketch the frequency response (in dB) and phase.

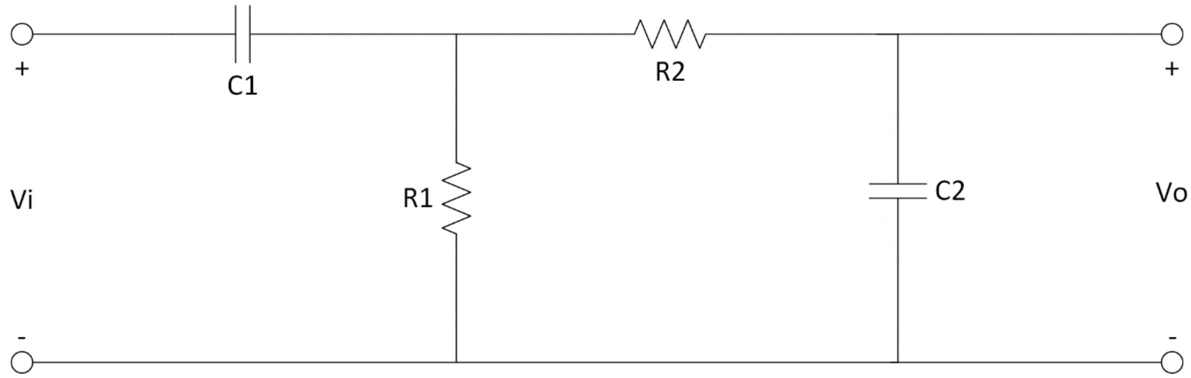


Figure 7.3

EXPERIMENTAL WORK 7

Students' Names & Numbers:

Table 1

f(Hz)	Vi (V_{p-p})	Vo (V_{p-p})	Δt	Phase (rad)	Vo / Vi	dB(Vo/Vi)
300						
1k						
3k						
10k						
30k						
100k						
300k						

Table 2

f(Hz)	Vi (V_{p-p})	Vo (V_{p-p})	Phase (rad)	Vo / Vi	dB(Vo/Vi)
10					
30					
100					
300					
1k					
3k					
10k					

Table 3

f(Hz)	Vi (V_{p-p})	Vo (V_{p-p})	Phase (rad)	Vo / Vi	dB(Vo/Vi)
10					
30					
100					
300					
1k					
3k					
10k					
30k					
100k					
300k					

- **Sketch the voltage transfer characteristics and phases of the circuits below.**