BRAC UNIVERSITY
DEPT. OF COMPUTER SCIENCE AND ENGINEERING
COURSE NO.: CSE250
Circuits and Electronics Laboratory

Experiment No. 1

Name of the Experiment:

CONSTRUCTION & OPERATION OF SIMPLE ELECTRICAL CIRCUITS

OBJECTIVE:

The experiment is to acquaint the students with some simple circuits and to make them familiar with diagram reading, drawing and wiring with the help of different types of switches (SPST- Single pole single throw, SPDT- single pole double throw, DPST- Double pole single throw, DPDT- Double pole Double throw) that will be frequently encountered in different experiments.

INSTRUCTIONS:

Read the following procedure carefully and draw the circuit diagrams accordingly in the space allotted for each procedure and then implement it practically. Your report must contain neat diagrams of the circuits.
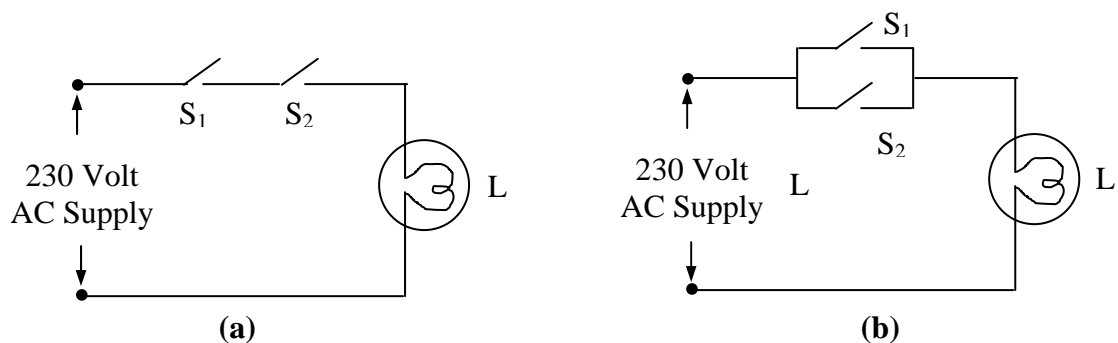
APPARATUS:

1) Two lamp boards (220v, 100w)
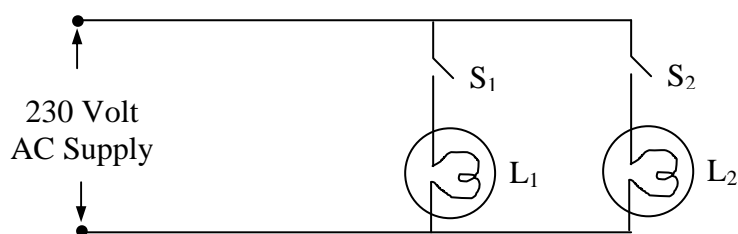2) Two SPST, two SPDT and one DPDT switch.

PROCEDURE:

1. Connect an electric lamp so that it may be operated from a 220v ac supply using an SPST switch.
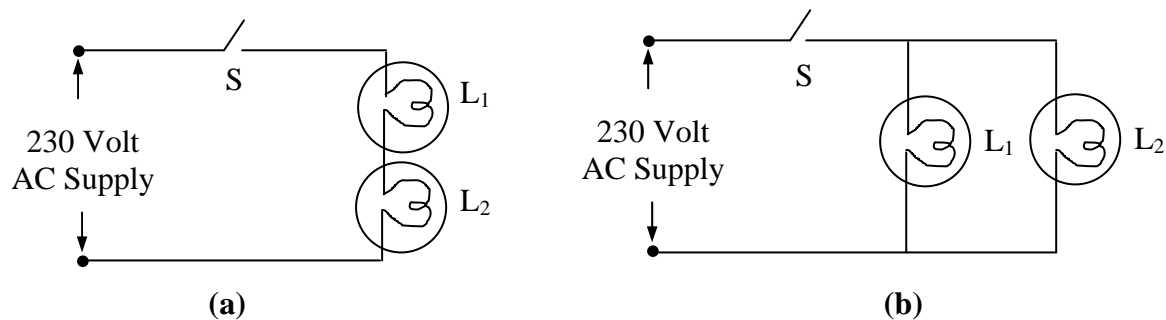

230 Volt AC Supply — S — L
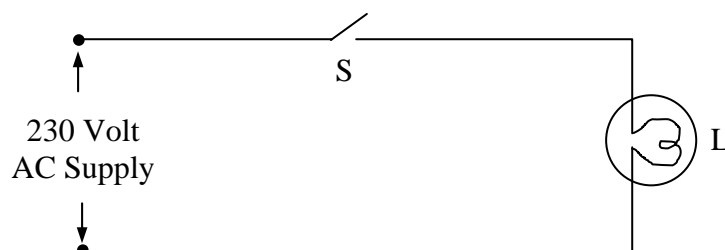
2. Connect a lamp so that it may be operated by either of two SPST switches.

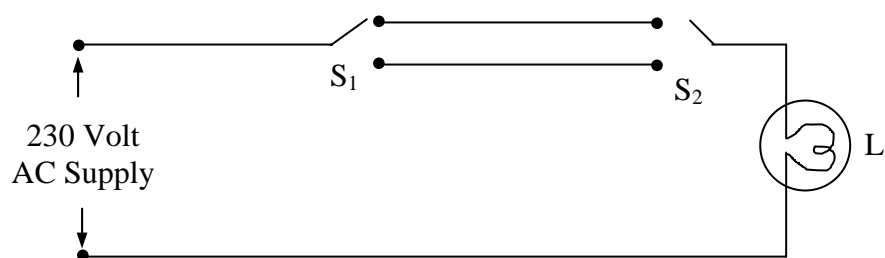**(a)**   **(b)**

3. Connect two lamps so that either may be operated from a common source by its own switch.


4. Connect two lamps so that both may be operated simultaneously from a common source by one SPST switch. Indicate the preferable one.

(a)   **(b)**

5. Connect a lamp so that it may be operated independently by either of two SPDT switches from a 220v source.

6. Connect a lamp using two SPDT and one DPDT switches to the power supply in such way so that the lamp may be turned ON/OFF by any of the three switches.
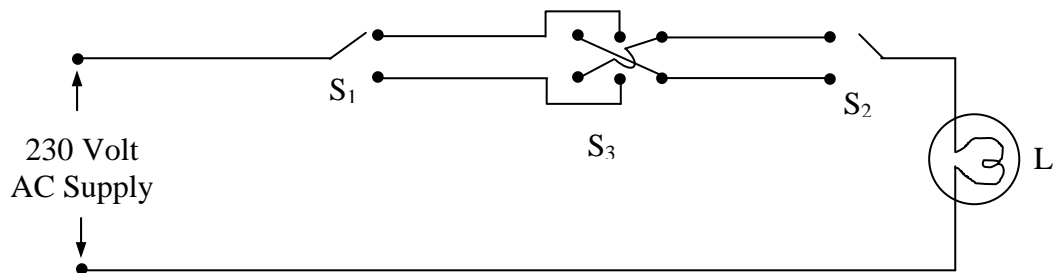

CAUTION:

1. **Don't switch on the supply until your teacher has checked the circuit.**
2. Take care of the reading of the apparatus.
3. Take care of any bare circuit element in energized condition.
4. Put on shoes with good insulation.

QUESTIONS:

1. What is an electrical circuit?
2. What is a short circuit?
3. Which method in procedure 4 is preferable? Why?
4. What is the disadvantage of the position of the switch in the following circuit?
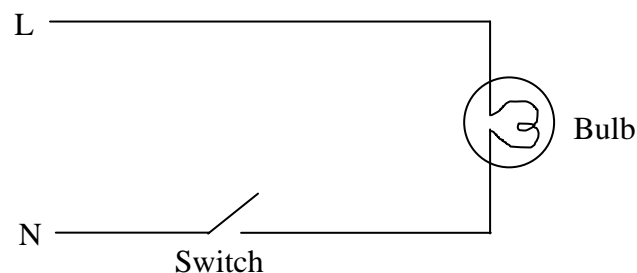
BRAC UNIVERSITY
DEPT. OF COMPUTER SCIENCE AND ENGINEERING
COURSE NO.: CSE250
Circuits and Electronics Laboratory

Experiment No. 2

Name of the Experiment:

Verification of KCL and KVL

KVL

OBJECTIVE:
This experiment is intended to verify Kirchhoff's voltage law (KVL) with the help of series circuits.

THEORY:
KVL states that around any closed circuit the algebraic sum of the voltage rises equals the algebraic sum of the voltage drops.

APPARATUS:
➢ One DC Ammeter (0 - 1A)
➢ One multimeter
➢ Three Resistors
➢ One DC power supply

PROCEDURE:

➢ Connect the resistors R_1, R_2 and R_3 in series to a DC power supply as shown in Fig 1.
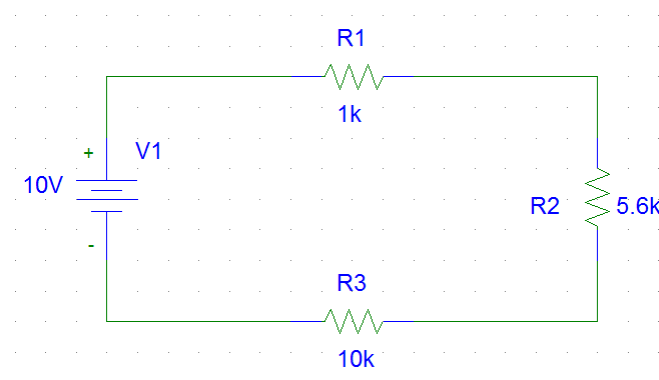

Fig. 1

➢ Take readings of V_1, V_2, V_3, V_s using multimeter. Verify KVL as $V_S = V_1 + V_2 + V_3$.

➢ Calculate the theoretical values of V_1, V_2 & V_3 & note them down in 'Theoretical Observation' row in table

Use voltage divider rule as stated below to get these values:

$V_1 = (R_1/R_e) * V;$ $V_2 = (R_2/R_e) * V;$ $V_3 = (R_3/R_e) * V$

Where, $R_e = R_1 + R_2 + R_3$

TABLE 1: Verification of KVL.

Observation	R1	R2	R3	V	V1	V2	V3
Experimental	1KΩ	5.6KΩ	10KΩ	10V			
Theoretical							

REPORT:
1. State the rules of connecting voltmeter and ammeter in the circuit.
2. Comment on the results obtained and discrepancies (if any).

KCL

OBJECTIVE:

This experiment is intended to verify Kirchhoff's current law (KCL) with the help of a simple parallel circuit.

THEORY:

KCL states that the algebraic sum of the currents entering any node equals the sum of the currents leaving the node.

APPARATUS:

➢ One DC Ammeter (0 - 1A)
➢ Three resistors
➢ One multimeter
➢ One DC supply

PROCEDURE:

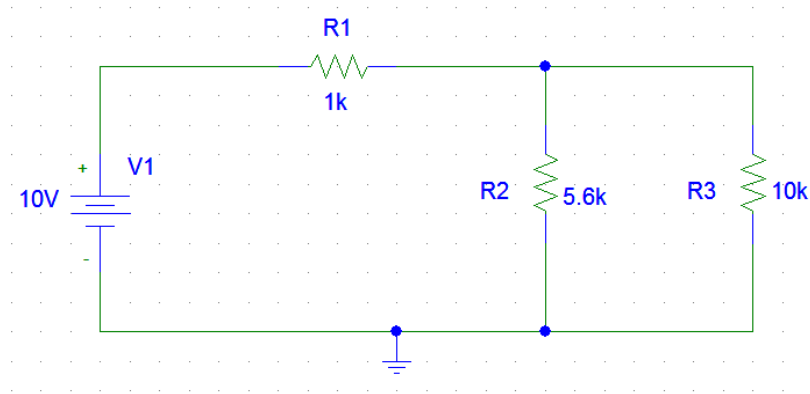➢ Connect the resistors in parallel across the power supply as shown in figure2

Fig.2

- ➢ Measure V_S, I_O, I_1, I_2, I_3.
- ➢ Verify KCL as $I_S = I_1 + I_2 + I_3$.
- ➢ Calculate the theoretical values of I, I1, I2 & I3 & note them down in 'theoretical observation' row in table

  Use the following to get these values:

  $I1 = V/R1;$ $I2 = V/R2;$ $I3 = V/R3;$ $I = I1 + I2 + I3$

TABLE 1: Verification of KCL

Observation	R1	R2	R3	V	I=V1/R1	I1=V2/R2	I2=V3/R3	I= I1+I2
Experimental	1KΩ	5.6KΩ	10KΩ	10V				
Theoretical								

REPORT:

1. Comment on the obtained results and discrepancies (if any).

Experiment No. 03

Name of the Experiment:

OBSERVATION OF APPLICATION OF A LOAD, AFFECTING THE TERMINAL VOLTAGE OF THE SUPPLY

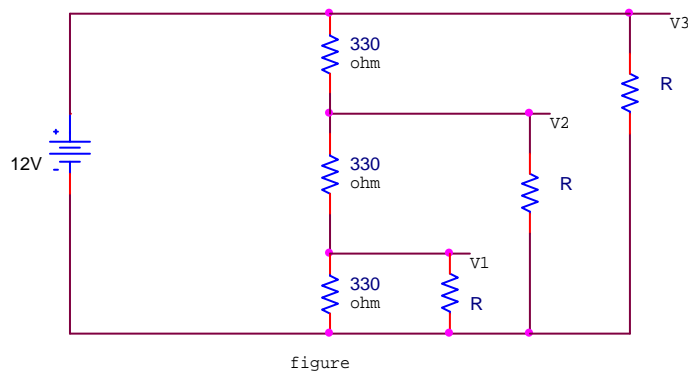## OBJECTIVE:

The objective of this experiment is to observe the change in supply voltage according to changing load.

APPARATUS:
1) DC Power Supply
2) Resistor 330Ω (3), 5.6kΩ (3), 10kΩ (3), 18kΩ (3)
3) Trainer Board
4) Digital Multimeter
5) Wires

FIGURE:

figure

PROCEDURE:

- Set up the circuit in no load condition. Use 12volt from DC power supply.
- Measure the voltage across each of the series resistances & also measure the current through the closed path.
- Connect the load using R=100Ω. Measure voltage V_1, V_2 & V_3 & note them down in tabular form. Also measure the current I_1, I_2 & I_3 through the three load branches & note them down.
- Replace the 100Ω load with 5.6kΩ, 10kΩ & 18kΩ resistance. Repeat step 3.

TABLE:

Load Resistance	V_1	V_2	V_3	$I_1 = V_1/R$ (mA)	$I_2 = V_2/R$ (mA)	$I_3 = V_3/R$ (mA)
No Load						
100Ω						
5.6kΩ						
10kΩ						
18kΩ						

REPORT:
1. Compare the different values of voltages obtained in different steps.
2. Compare the different values of current obtained in different steps.
3. What is the main observation in change in voltage or current, while changing load resistance?
4. Let us assume, for the figure given, the value of the series resistances are 220Ω each. Find out V_1, V_2, V_3 & I_1, I_2, I_3 for R=80Ω, 1kΩ, 100kΩ.

EXPT. NO. 3

Name Of The Experiment:

Verification of Superposition Principle

OBJECTIVE:

To verify experimentally the Superposition theorem which is an analytical technique of determining currents in a circuit with more than one emf source.

THEOREM:

In a linear circuit containing multiple independent sources and linear elements (e.g. resistors, inductors, capacitors) the voltage across (or the current through) any element when all the sources are acting simultaneously may be obtained by adding algebraically all the individual voltages (or the currents) caused by each independent source acting alone, with all other sources deactivated.
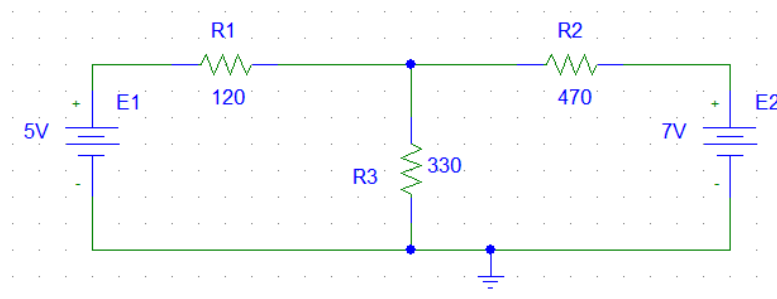
An independent voltage source is deactivated (made zero) by shorting it and an independent current source is deactivated (made zero) by open circuiting it. However, if a dependent source is present it must remain active during the superposition process.
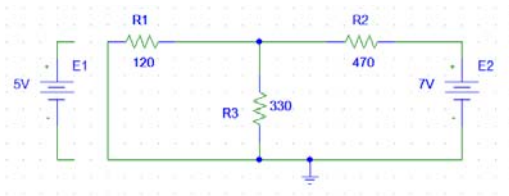
APPARATUS :

➢ Two DC power supplies.
➢ One multimeter.

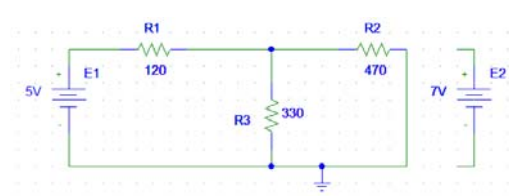### PROCEDURES :

1) Set up the circuit as in Fig. 1.

When E2 active: When E1 active:

2) Apply 5 volts from E_1 and 7 volts from E_2.
3) Measure the current I and record it in the given table.
4) Render E_2 inactive (keeping E_1 active) & measure the current $I^{'}$ in the branch R_3.
5) Render E_1 inactive (keeping E_2 active) & Measure the current $I^{''}$ in the branch R_3.
6) Verify if $I = I^{'} + I^{''}$ which would validate the superposition theorem for this particular circuit.

TABLE:

No. of Obs.	R_1 (ohms)	R_2 (ohms)	R_3 (ohms)	I_2 with both E_1 and E_2 active (amps)	$I_2^{'}$ with only E_1 active (amps)	$I_2^{''}$ with only E_2 active (amps)	$I_2^{'} + I_2^{''}$
1.	100Ω	220Ω	470Ω				
2.	$1.2K\Omega$	$2.2K\Omega$	$3.3K\Omega$				

REPORT :

1. Show results in tabular form.
2. Comment on the obtained results and discrepancies (if any).

CAUTIONS:

1. **Don't switch on the supply until the circuit has been checked by your teacher.**
2. **Take care of the reading of the apparatus.**
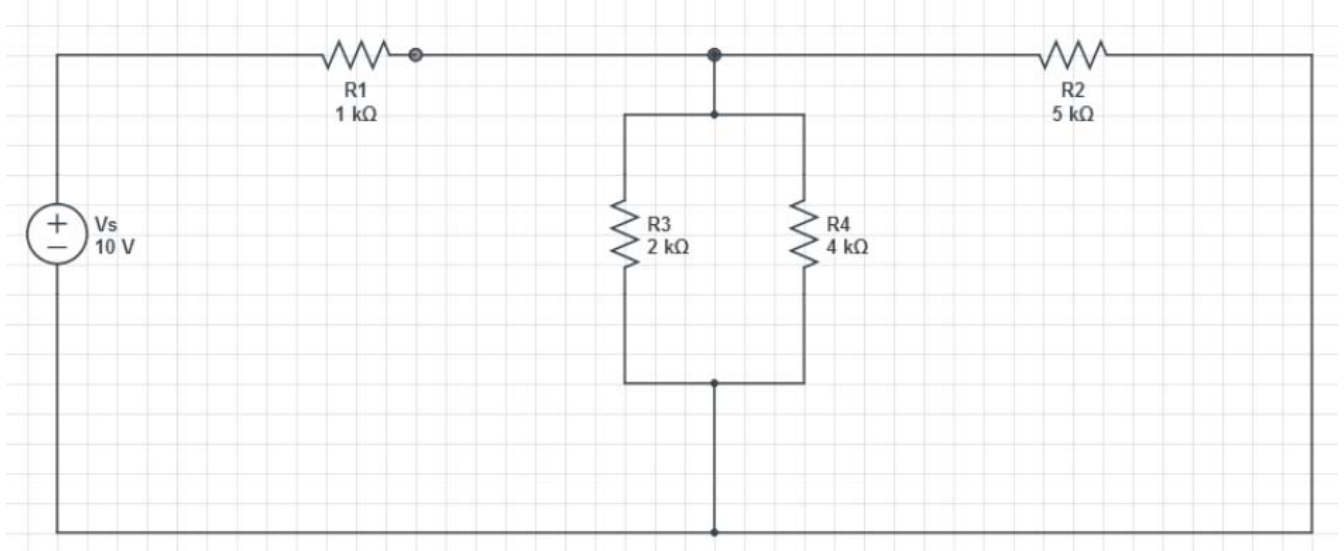3. **Take care of any bare circuit element in energized condition.**

QUESTION:

1. Define linear element, nonlinear element, linear circuit & nonlinear circuit.
2. "Although Superposition Principle can be used to determine voltage and current in a linear circuit, it cannot be used to determine power." --- Elucidate the statement.
3. Why an independent voltage source is deactivated by short circuiting it and an independent current source is deactivated by open circuiting it?
4. Find analytically the current I using
   - ➢ Superposition Principle
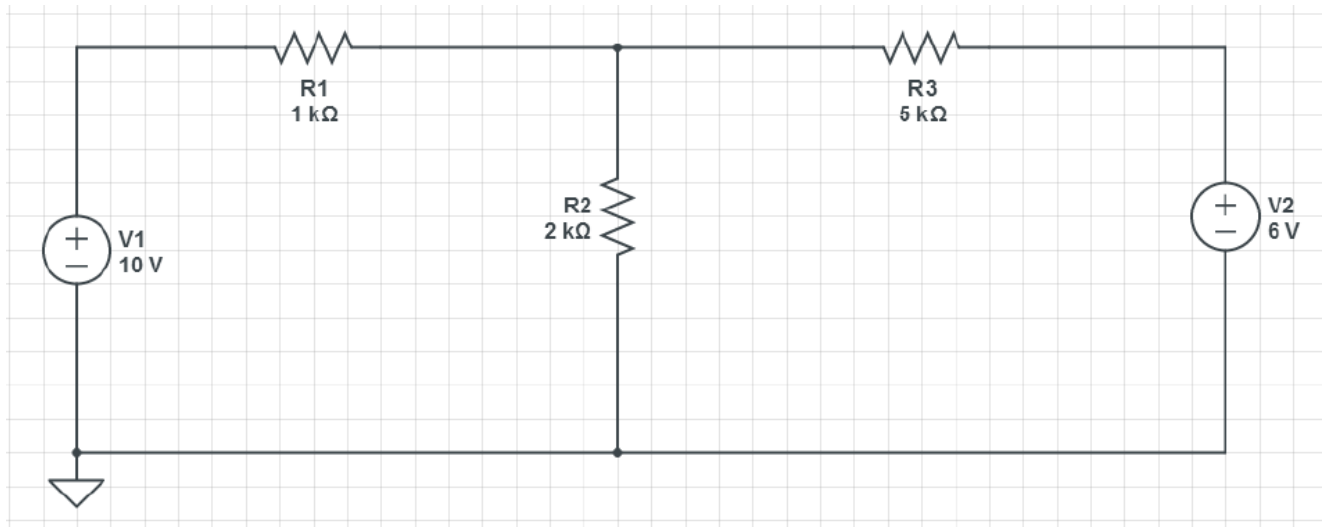   - ➢ Mesh Current Method
   - ➢ Node Voltage Method
   for E_1 = 5 volts, E_2 = 7 volts and R_1 , R_2 , R_3 at their values recorded in the observation of the Table shown.
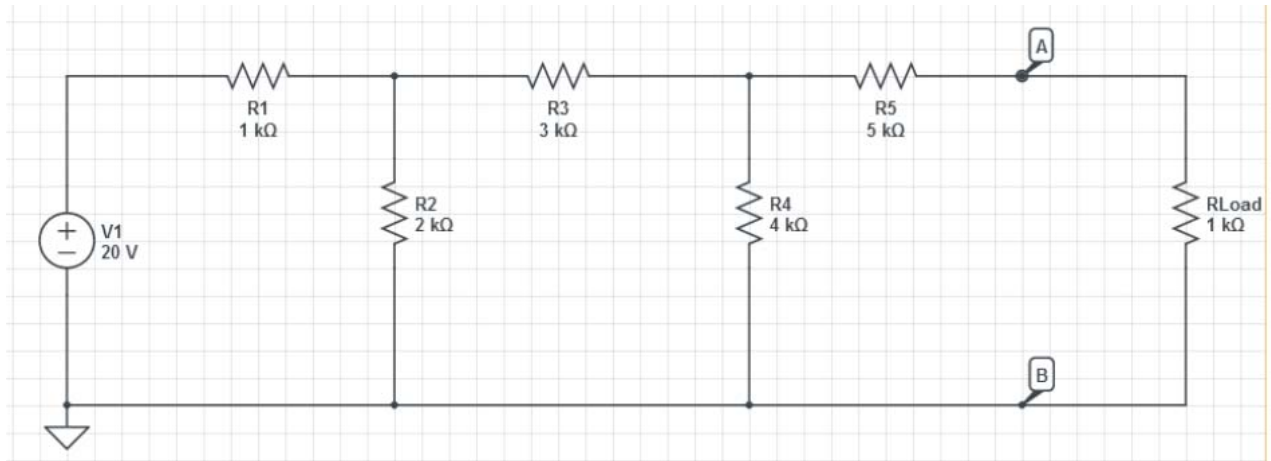
1. Implement the following circuit and verify **KVL** and **KCL**.

2. Find out the current through the **R2** resistor in the following network.
Then use appropriate steps to verify **SUPERPOSITION** theorem in the network.
(find the currents when each source is active, deactivating the other; then perform algebraic sum)

3. Find out the current through the **Rload** of the following network.
Use appropriate simulation results to find out the **THEVENIN's EQUIVALENT** circuit with respect to the A-B terminals.
Redraw and re-simulate the equivalent circuit to recover the value of **current through the load**.
Compare the results found from both the methods.

Experiment No.5

Name Of The Experiment:

Verification of Thevenin's Theorem and Maximum Power Transfer Theorem

PART 1:

OBJECTIVE :

To verify Thevenin's theorem with reference to a given circuit theoretically as well as experimentally.

INTRODUCTION :
It is often desirable in circuit analysis to study the effect of changing a particular branch element while all other branches and all the sources in the circuit remain unchanged. Thevenin's theorem is a technique to this end and it reduces greatly the amount of computations which we have to do each time a change is made. Using Thevenin's theorem the given circuit excepting the particular branch to be studied is reduced to the simplest equivalent circuit possible and then the branch to be changed is connected across the equivalent circuit.

The Thevenin's theorem states that any two terminal linear bilateral network containing sources and passive elements can be replaced by an equivalent circuit consist of a voltage source V_{th} in series a resistor R_{th} where

V_{th} = The open circuit voltage (V_{OC}) at the two terminals A & B.
R_{th} = The resistance looking into the terminals A and B of the network with all sources removed.
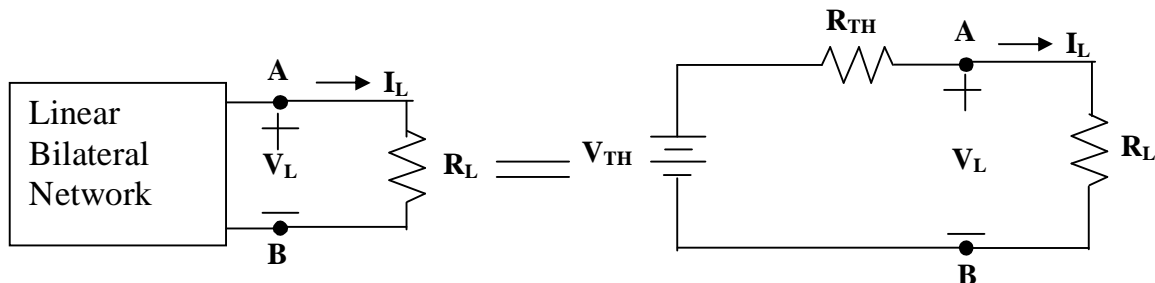
There are several methods for determining Thevenin resistance R_{TH}. An attractive method for determining R_{TH} is : (1) determine the open circuit voltage, and (2) determine the short circuit current I_{SC} as shown in the figure; then

$$R_{TH} = \frac{V_{OC}}{I_{SC}}$$
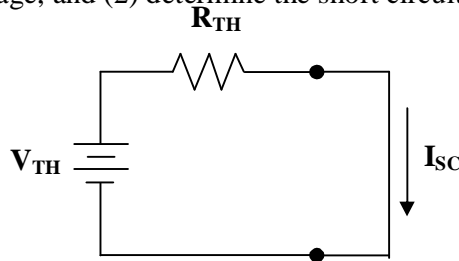
APPARATUS :

Ø Resistor: R₁:1K, R₂:3.3K , R₃:3.3k
Ø Multimeter
Ø DC Power Supply

PROCEDURE :

For Original Circuit:

1. Arrange the original circuit as shown in figure 1.Apply 10V dc from dc power supply.
2. Measure V_L, I_L for three values of R_L & record the data in the table.
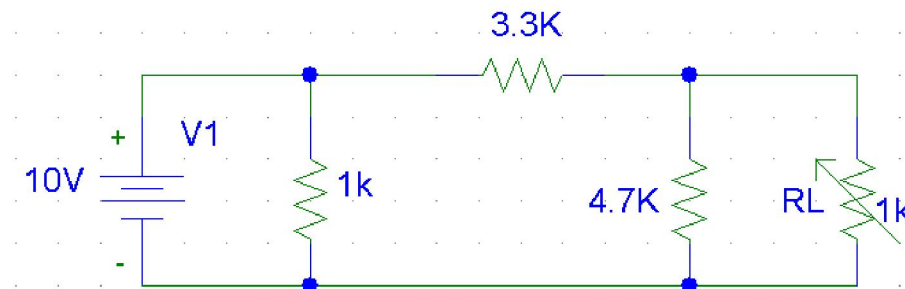

FIG.1: Original Circuit

Table1:

Data for Original circuit R_1=   , R_2=   , R_3=   , V_S=

No. of Obs.	Values of R_L	Load Voltage V_L	Load current I_L
1.			

FINDING V_{Th} & R_{TH} :

3. Remove the load resistance R_L and find the open circuit voltage between terminals A & B. This voltage is Thevenin voltage i.e. $V_{TH}=V_{OC}$.
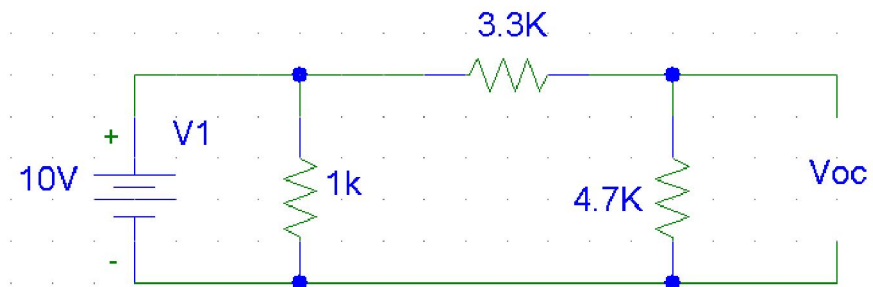

FIG.2: Circuit for finding V_{OC}

4. Place a short circuit between terminals A & B and find the short circuit current I_{SC}. Divide The open circuit voltage by the short circuit current to find the Thevenin resistance R_{TH} i.e $R_{TH} = \dfrac{V_{OC}}{I_{SC}}$
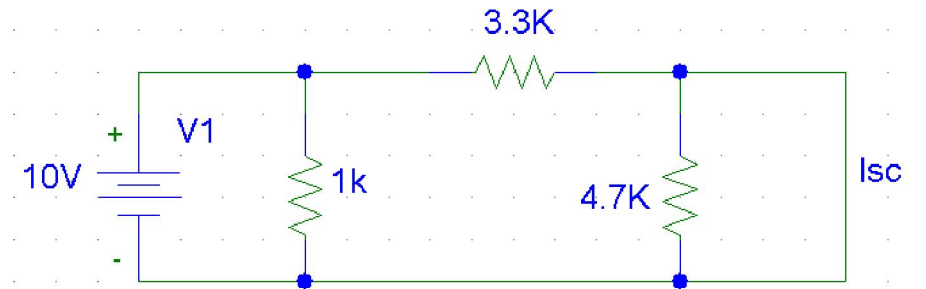

FIG.3: Circuit for finding I_{SC}

For Thevenin Equivalent Circuit:

5. Construct the Thevenin's equivalent circuit as shown in figure 4 setting the power supply at V_{TH} volts and the rheostat at R_{TH} ohms. Now measure the load current I_L and the load voltage V_L for the values of R_L determined in step 2. Compare these values with previous values.
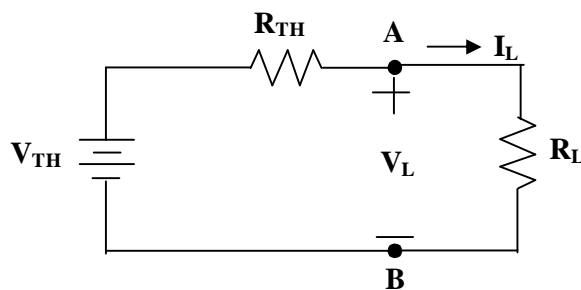

FIG.4: Thevenin Equivalent Circuit of Circuit1.

EXPERIMENTAL DATA:
$V_{TH} =$, $R_{TH} =$

Table2: Data for Thevenin equivalent circuit

No. of Obs	Values of R_L	Load Voltage V_L	Load current I_L
1.			

REPORT:

1. Find theoretically the Thevenin equivalent circuit for the values of R_0, R_2, R_3 & V_S recorded in table. Also find I_L, V_L.
2. Show the results in tabular form.
3. Comment on the results obtained and discrepancies (if any).

QUESTION:

1. Define unilateral, bilateral & equivalent circuit.
2. Describe other methods for determining Thevenin resistance.
3. Mention the advantages of using Thevenin Theorem.

REFERENCE:

♦ Introduction to Electric Circuits by R.C. Dorf & J.A. Svoboda.
♦ Introductory Circuit Analysis by R. L. Boylestad.
♦ A Text Book of Electrical Technology, Vol.1 by B. L. Theraja & A. K. Theraja

PART 2:

OBJECTIVE: The objective of this experiment is to verify maximum power transfer theorem.

THEORY: The maximum power transfer theorem states that a resistive load will receive maximum power when its total resistive value is exactly equal to the Thevenin's resistance of the network as "seen" by the load.
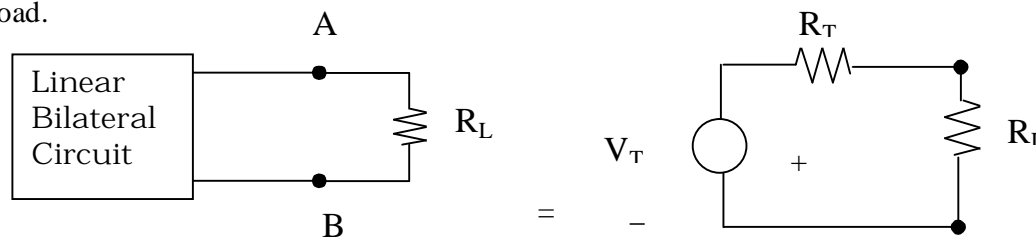
We know that any circuit A terminated with a load R_L can be reduced to its Thevenin's equivalent. Now according to this theorem the load R_L will receive maximum power when $R_L = R_{TH}$

The efficiency of power transfer is defined as the ratio of the power delivered to the load P_{OUT}, to the power supplied by the source P_{IN}.

$$\%\eta = \frac{P_{OUT}}{P_{IN}} \times 100 = \frac{V_L}{V_{TH}} \times 100 = \frac{R_L}{R_L + R_{TH}} \times 100$$

The voltage regulation is defined as

$$\%VR = \frac{\text{Load voltage at no load} \quad - \quad \text{Load voltage at full load}}{\text{Load voltage at full load}} \times 100$$

$$= \frac{R_{TH}}{R_L} \times 100$$

At maximum power transfer condition, η= 50 % & VR= 100 %.
A relatively low efficiency of 50 % can be tolerated in situations where power levels are relatively low such as in electronic & communications circuits for transmission & reception of signal where the Engineer's goal is to receive or transmit maximum amount of power.

However, when large power levels are involved, such as at generating stations, efficiencies of 50 % would not be acceptable. The goal here is high efficiency and not maximum power. Power utility systems are designed to transmit the power to the load with the greatest efficiency by reducing the losses on the power lines. Thus the effort is concentrated on reducing R_{TH}, which would represent the resistance of the source plus the line resistance.

APPARATUS:

1. Multimeter
2. DC power supply
3. Resistors: R_{th}=1.94K (1k+1k), R_L=470, 1K, 1.5K, 1.94K, 3.3K, 4.7K
4. Wires

EXPERIMENTAL SETUP:
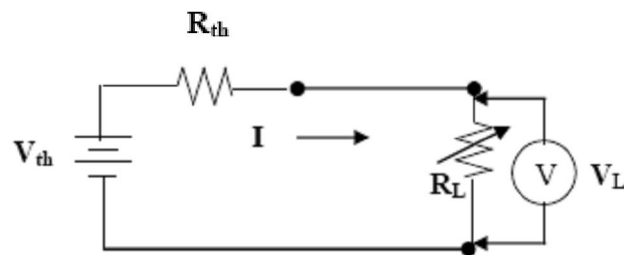
PROCEDURE:
1. Set up the circuit as shown in figure.
2. Apply V_{th} from dc power supply.
3. Vary the load resistor from 470 to 4.7K & measure the voltages VL & I.

EXPERIMENTAL DATA:

No. of Obs.	V_{TH}	V_L	I	$P_{IN}=V_{TH}I$	$P_{OUT}=V_LI$	LOSS= P_{IN} -P_{OUT}	%η	% VR	$R_L=V_L/I$
1.									
2.									
3.									
4.									
5.									
6.									
7.									

REPORT:

1. Show the results in tabular form.
2. Plot the following curves on the graph paper
i)   % η vs R_L
ii)   % VR vs R_L
iii)   loss vs R_L
iv) P_{OUT} vs R_L
v) I_L vs R_L
vi) V_L vs R_L

QUESTIONS:

1. Why high voltage transmission is used in case of transmitting electric power?
2. Where maximum power transfer is used?
3. Why instead of transmitting maximum power, power utility transmits power at maximum efficiency?
4. Deduce the condition for maximum power transfer.

REFERENCE:

- Introduction to Electric Circuits.
  By R.C. Dorf & J.A. Svoboda.
- Introductory Circuit Analysis.
  By R. L. Boylestad

EXPT. NO.7

Name of the Experiment:

STUDY OF TRANSIENT BEHAVIOR OF RC CIRCUIT

OBJECTIVE:
The objective of this experiment is to study Transient Response of RC circuit with step Input. In this experiment we shall apply a square wave input to an RC circuit separately and observe the respective wave-shapes and determine the time constants.

THEORY:
The transient response is the temporary response that results from a switching operation and disappears with time. The steady state response is that which exists after a long time following any switching operation.

Let us consider an RC circuit shown in figure.
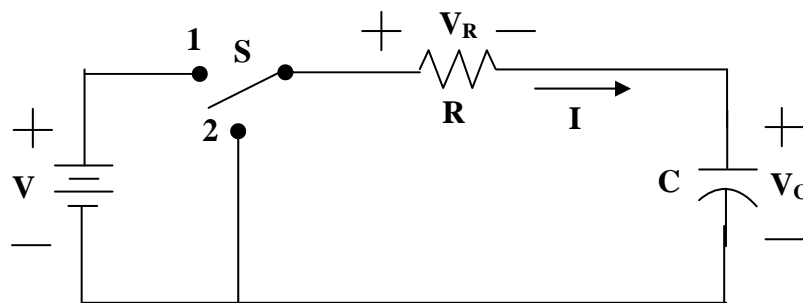
CHARGING PHASE:
When the switch is connected to position 1, applying KVL we can write

$$V = Ri + \frac{1}{C}\int idt - - - - - - - - - - -(1)$$

If the capacitor is initially uncharged, the solution of equation (1) is----

$$i = \frac{V}{R}e^{-\frac{t}{\tau}} - - - - - - - - - - - - - -(2)$$
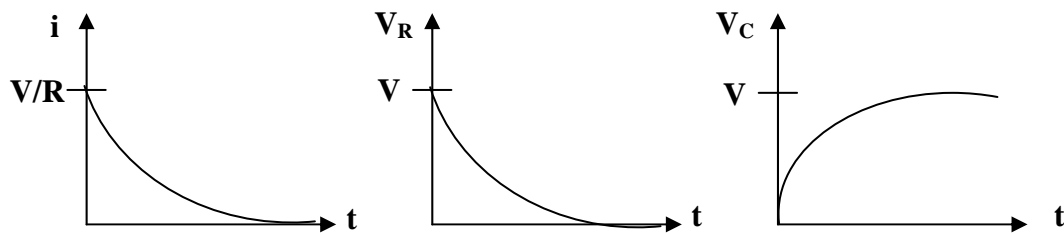
Therefore the voltage across the resistor and capacitor are given by

$$V_R = Ve^{-\frac{t}{\tau}} - - - - - - - - - - - - - -(3)$$

$$V_C = V - V_R = V(1 - e^{-\frac{t}{\tau}}) - - - - - -(4)$$

Where τ = RC and is called the time constant of the circuit.
Equation (2), (3) & (4) are plotted below:

It is seen from the curves that the voltage across the capacitor rises from zero to V volts exponentially and the charging current is maximum at the start i.e. when C is uncharged, then it decreases exponentially and finally ceases to zero when the capacitor voltage becomes V.

DISCHARGING PHASE:

When the switch is connected to position 2, applying KVL we can write

$$0 = Ri + \frac{1}{C}\int i dt ------------(5)$$

Since the voltage across the capacitor is now V, the solution of equation (5) is
Therefore the voltage across the resistor and capacitor are given by

$$i = -\frac{V}{R}e^{-\frac{t}{\tau}} ------------(6)$$
$$V_R = Ve^{-\frac{t}{\tau}} ------------(7)$$

$$V_C = Ve^{-\frac{t}{\tau}} ------------(8)$$

Equation (6), (7) & (8) are plotted below:
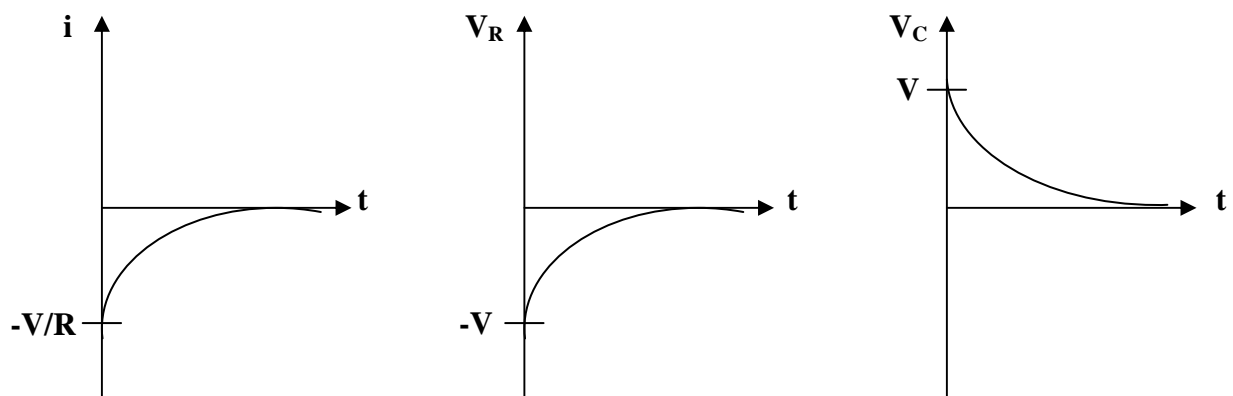
It is seen from the curves that the voltage across the capacitor falls from V to zero volts exponentially. The charging current is maximum at the start i.e. when the switch is just thrown to position 2, then it decreases exponentially and finally ceases to zero when the capacitor voltage becomes zero.

APPARATUS:

- ➢ Resistance: 1KΩ
- ➢ Capacitance: 1μF
- ➢ Oscilloscope and Chords
- ➢ Signal Generator and Chords
- ➢ Wires
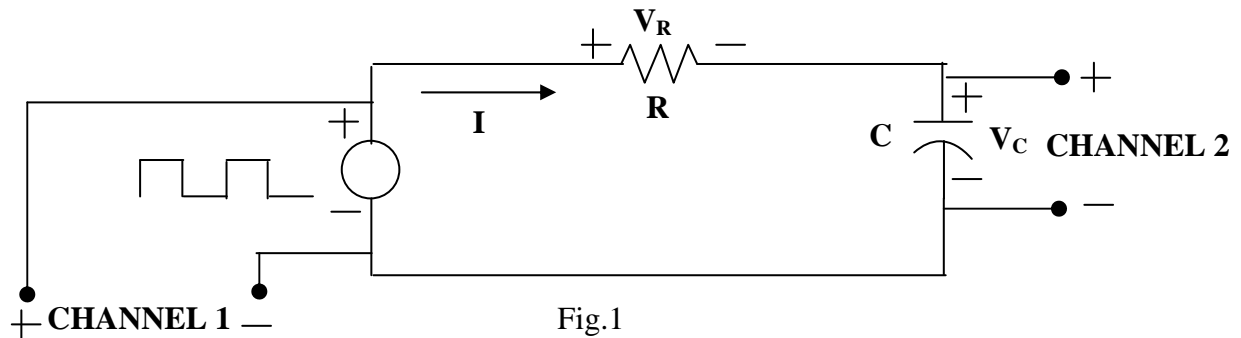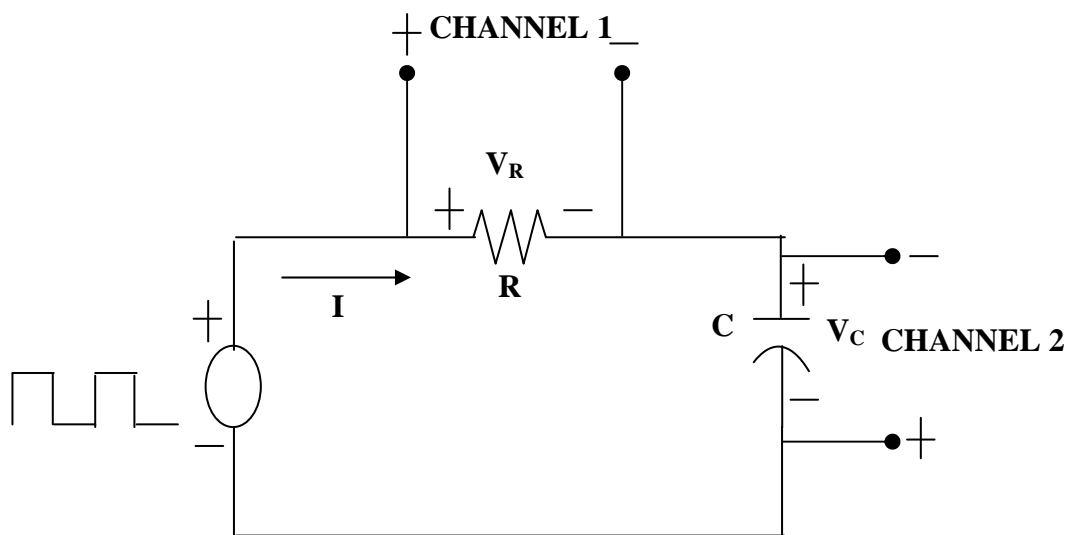- ➢ Bread board

EXPERIMENTAL SETUP:

Fig.1

Fig.2

PROCEDURE:

1. Setup the circuit as shown in figure 1.
2. Apply 100Hz square wave from signal generator.
3. Observe the wave shapes at Ch.1 and Ch.2 in DUAL mode and draw them. Find the time constant from the wave shape of V_C.
4. Disconnect Ch.1 and Ch.2 and reconnect them as shown in figure 2.
5. Observe the wave shapes at Ch.1 and Ch.2 (INV.) in DUAL mode and draw them.

REPORT:

1. Draw all the wave shapes on graph

QUESTION:

1. Define capacitor and capacitance. Write the features of a capacitor. What does capacitance measure?
2. Deduce voltage-current relationship for a capacitor. Why the voltage across a capacitor cannot change instantaneously.
3. Define time constant for an RC circuit. What is the significance of time constant? How time constant can be determined?
4. Describe the charging and discharging phase of an RC circuit both qualitatively and quantitatively.

REFERENCE BOOKS:

- Introduction to Electric Circuits.
    By R.C. Dorf & J.A. Svoboda.
- Introductory Circuit Analysis.
    By R. L. Boylestad.
- A Text book of Electrical Technology, Vol. 1
    By B.L Theraja & A.K. Theraja

BRAC UNIVERSITY
COMPUTER SCIENCE AND ENGINEERING
CSE250: CIRCUITS AND ELECTRONICS
SOFTWARE LAB ASSIGNMENT

1. Implement the following circuit and verify **KVL** and **KCL**.
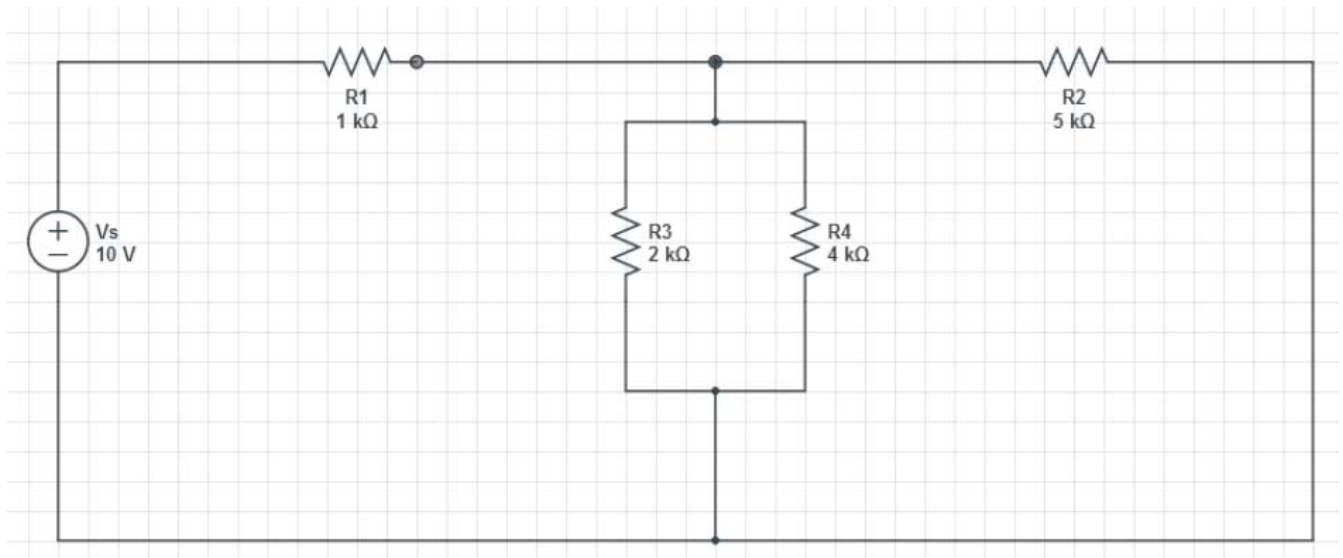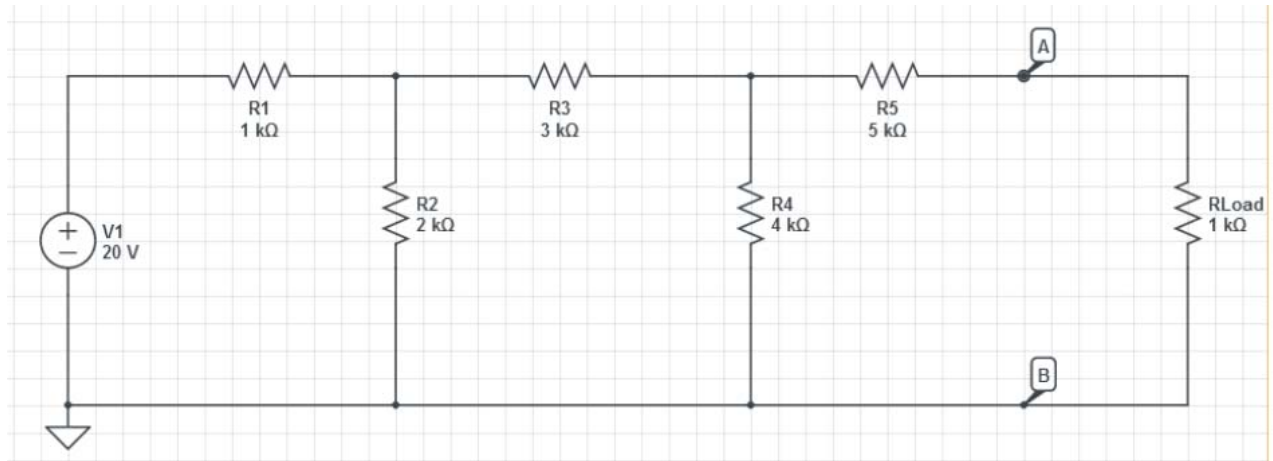

2. Find out the current through the **R2** resistor in the following network.
Then use appropriate steps to verify **SUPERPOSITION** theorem in the network.
(find the currents when each source is active, deactivating the other; then perform algebraic sum)
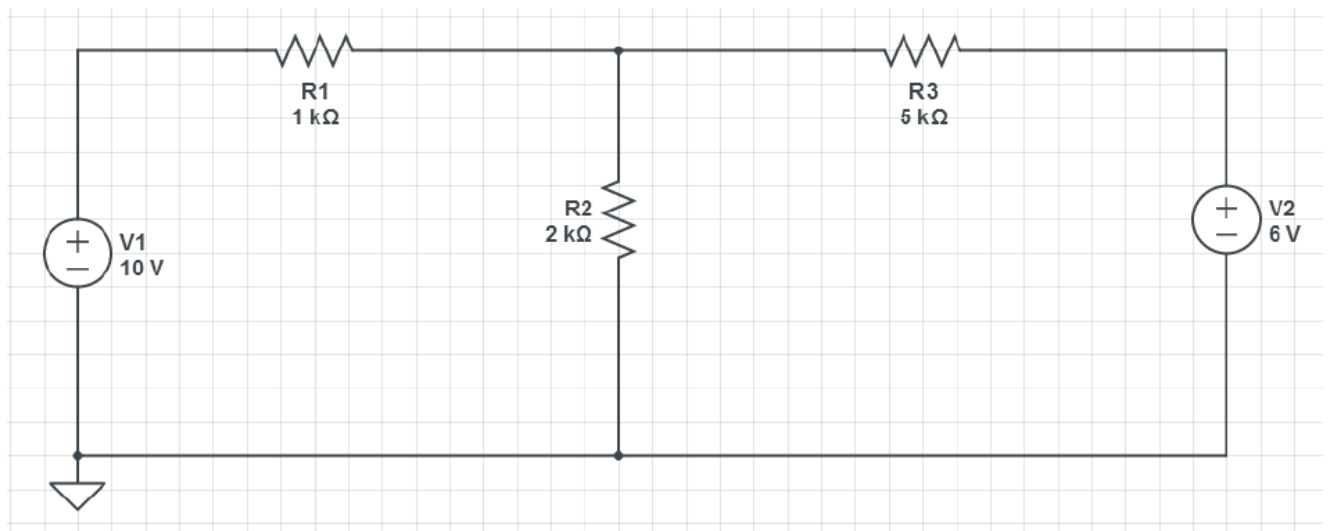
3. Find out the current through the **Rload** of the following network.
Use appropriate simulation results to find out the **THEVENIN's EQUIVALENT** circuit with respect to the A-B terminals.
Redraw and re-simulate the equivalent circuit to recover the value of **current through the load**.
Compare the results found from both the methods.

4. Let us assume, for the figure given:
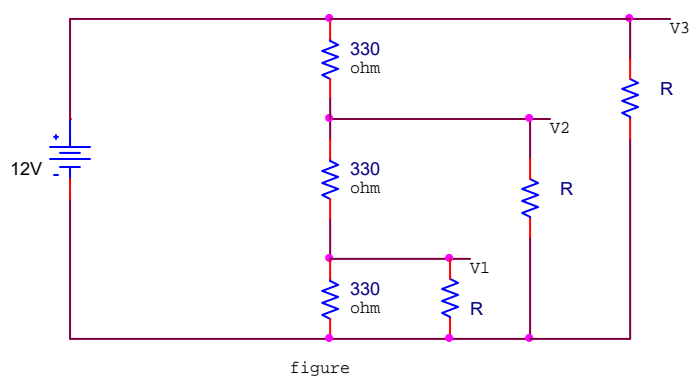
figure

The value of the series resistances are 220Ω each. Find out V_1, V_2, V_3 & I_1, I_2, I_3 for R=80Ω, 1kΩ, 100kΩ.

CSE250 LAB

Experiment Number : 02   **Name of the Experiment: Verification of KCL and KVL**

 Group Number: _____
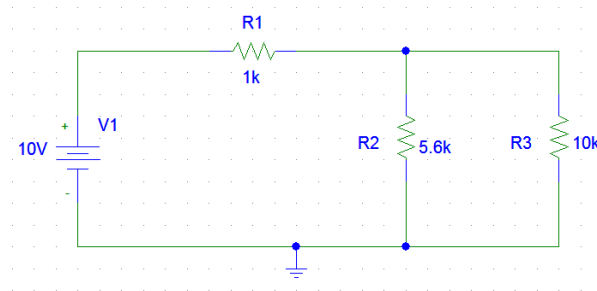
Group Members:
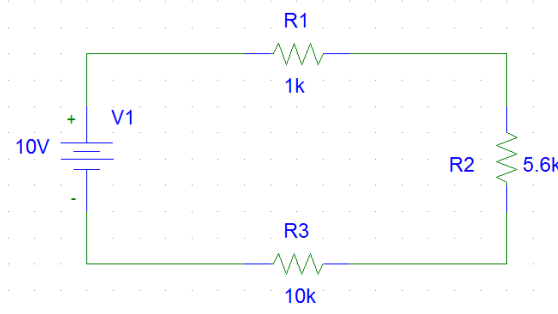
SI. NO.	Student Id	Student Name	Section

Verification of KVL:

Observation	R1	R2	R3	V	V1	V2	V3
Experimental	1KΩ	5.6KΩ	10KΩ	10V			
Theoretical							

Verification of KCL:

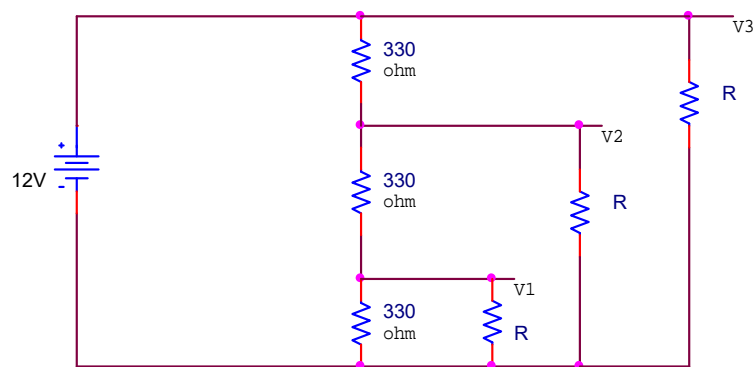| Observation	R1	R2	R3	V	I=V1/R1	I1=V2/R2	I2=V3/R3	I= I1+I2
Experimental	1KΩ	5.6KΩ	10KΩ	10V				
Theoretical								

CSE250 LAB

Experiment Number : 03

Name of the Experiment: OBSERVATION OF APPLICATION OF A LOAD, AFFECTING THE TERMINAL VOLTAGE OF THE SUPPLY

Group Number: ___

Group Members:

SI. NO.	Student Id	Student Name	Section
1			
2			
3			
4			

Circuit Diagram:

figure

Load Resistance	V_1	V_2	V_3	$I_1 = V_1/R$ (mA)	$I_2 = V_2/R$ (mA)	$I_3 = V_3/R$ (mA)
No Load						
(100Ω)						
(5.6kΩ)						
(10kΩ)						
(18kΩ)						

Experiment Number : 07

Name of the Experiment: STUDY OF TRANSIENT BEHAVIOR OF RC CIRCUIT

Group Number:

Group Members:

SI. NO.	Student Id	Student Name	Section

Original circuit: Here input 100Hz 10V p-p square wave.

CHANNEL 1

V_R

R

I

C

V_C

CHANNEL 2
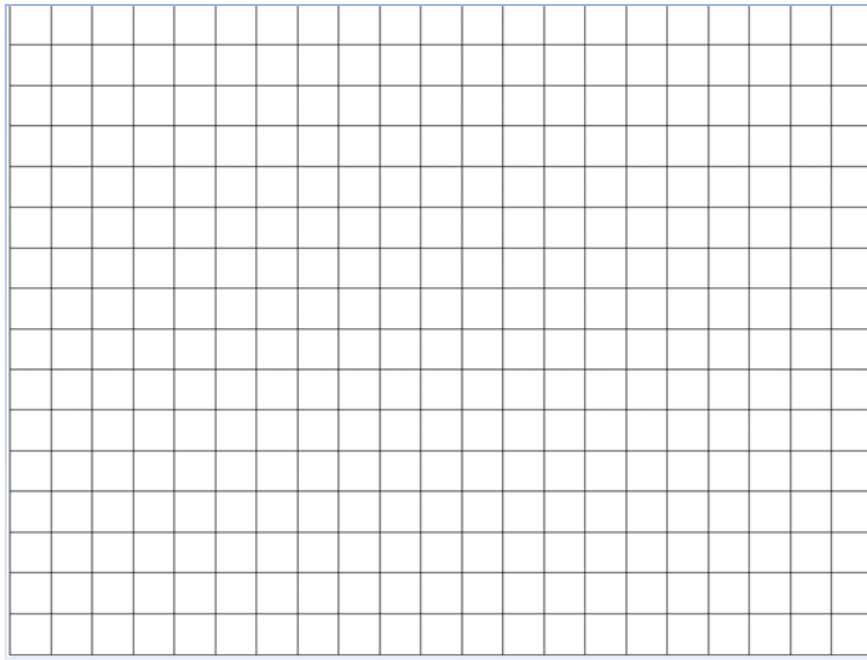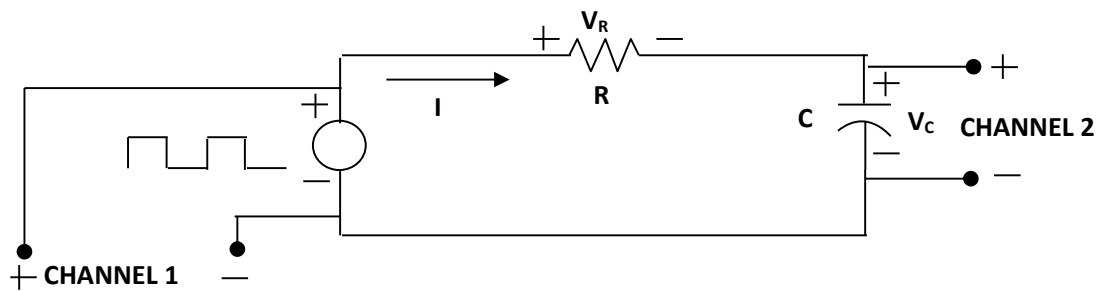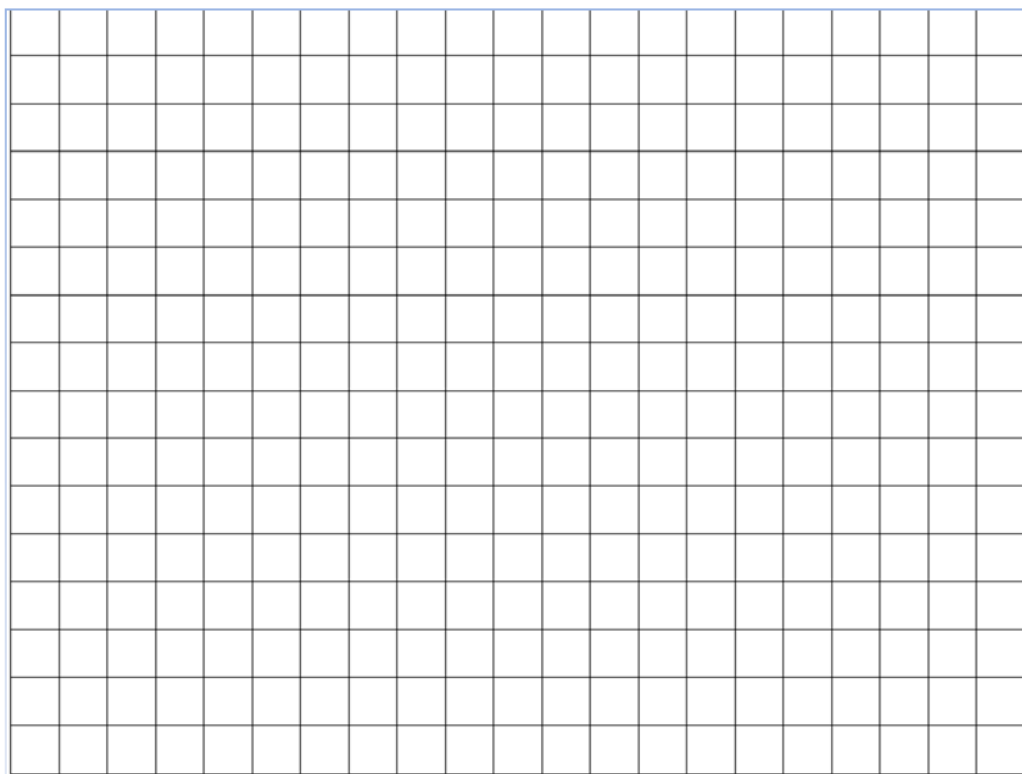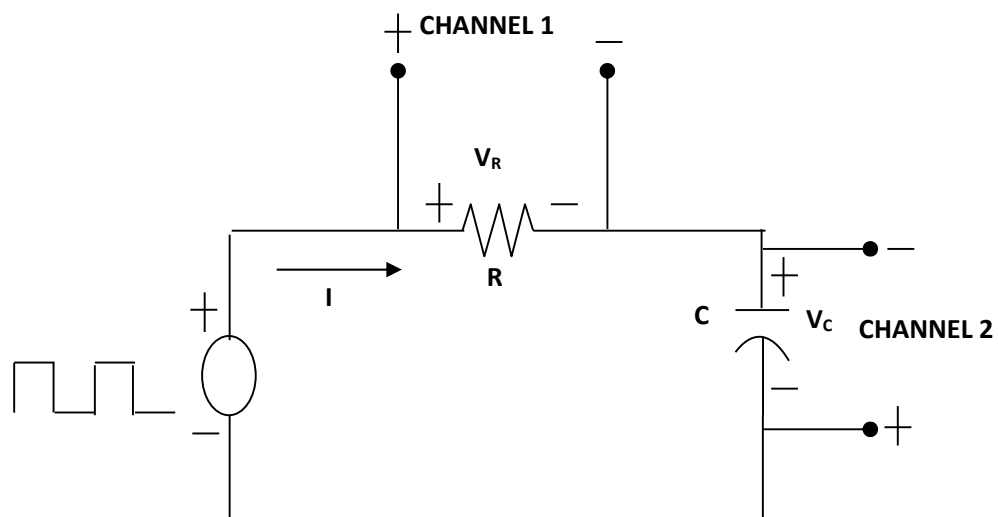
Experiment Number : 04   **Name of the Experiment: Verification of Superposition Theorem**

Group Number: _____

Group Members:

SI. NO.	Student Id	Student Name	Section

Verification of Superposition Theorem:

R3 (KΩ)	V (Volt)	I (mA)

R3 (KΩ)	V' (Volt)	I' (mA)
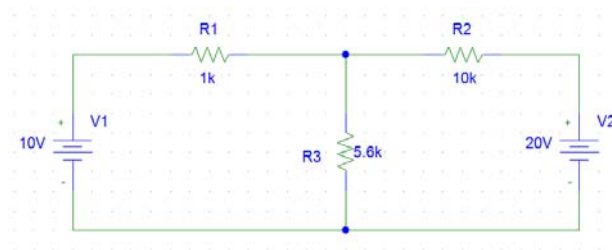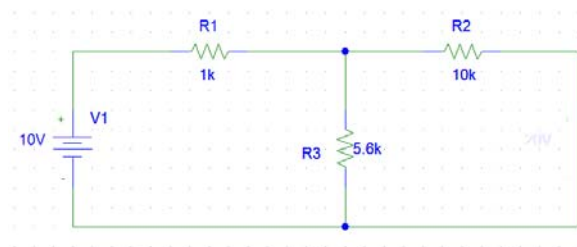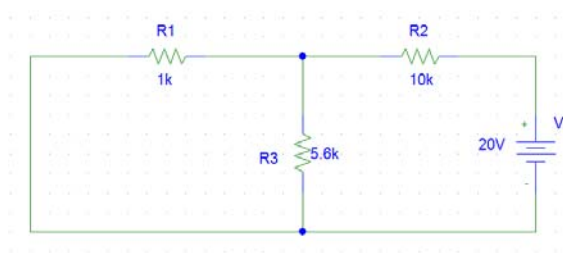
R3 (KΩ)	V'' (Volt)	I'' (mA)

V (Volt)	V'+V" (Volt)	I (mA)	I'+I" (mA)

CSE250 LAB

Experiment Number : 06

Name of the Experiment: Verification of Thevenin's Theorem and Maximum Power Transfer Theorem

Group Number:

Group Members:

SI. NO.	Student Id	Student Name	Section

Part 01: Verify Thevenin Theorem

Original circuit:
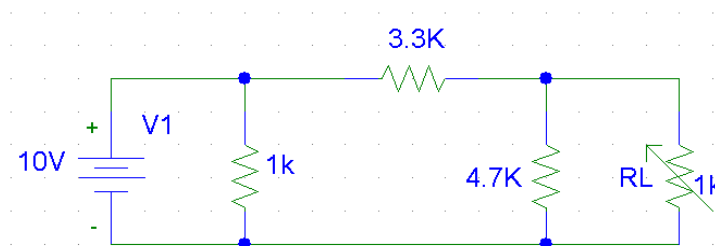
Table1: Data for Original circuit R_1= 1kΩ   , R_2 = 3.3kΩ   , R_3 = 4.7kΩ   , V_S = 10V

No. of Obs.	Values of R_L	Load Voltage V_L	Load current I_L
1.	1kΩ		
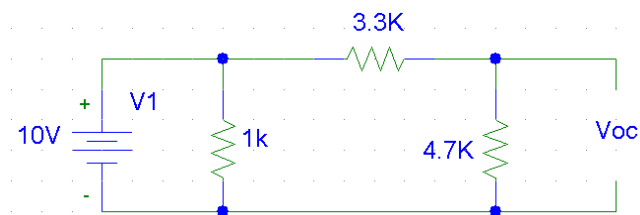
FINDING V_{Th} & R_{TH} :

Circuit for finding V_{OC}
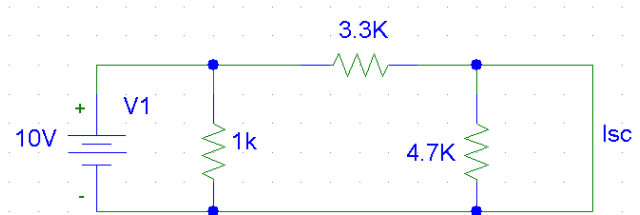
Circuit for finding I_{SC}

$$R_{TH} = \frac{V_{OC}}{I_{SC}}$$

Thevenin Equivalent Circuit:
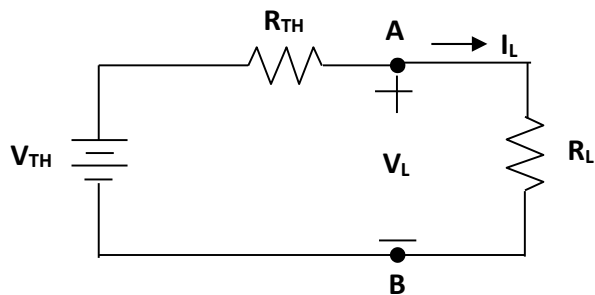
EXPERIMENTAL DATA:

$V_{TH} =$ _____ , $R_{TH} =$

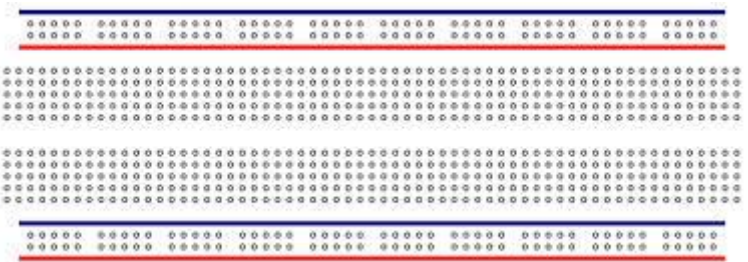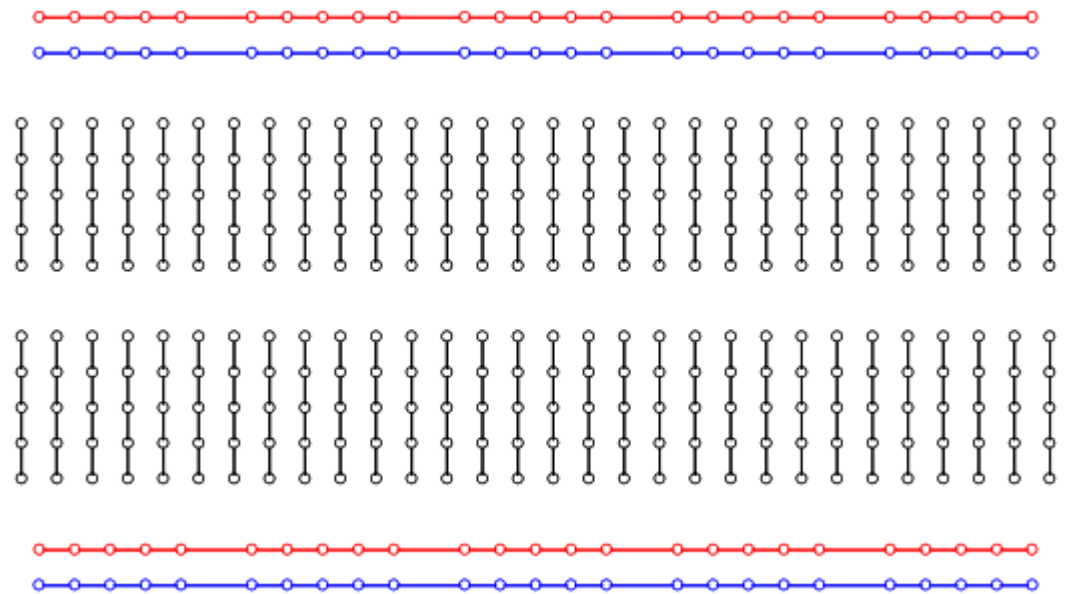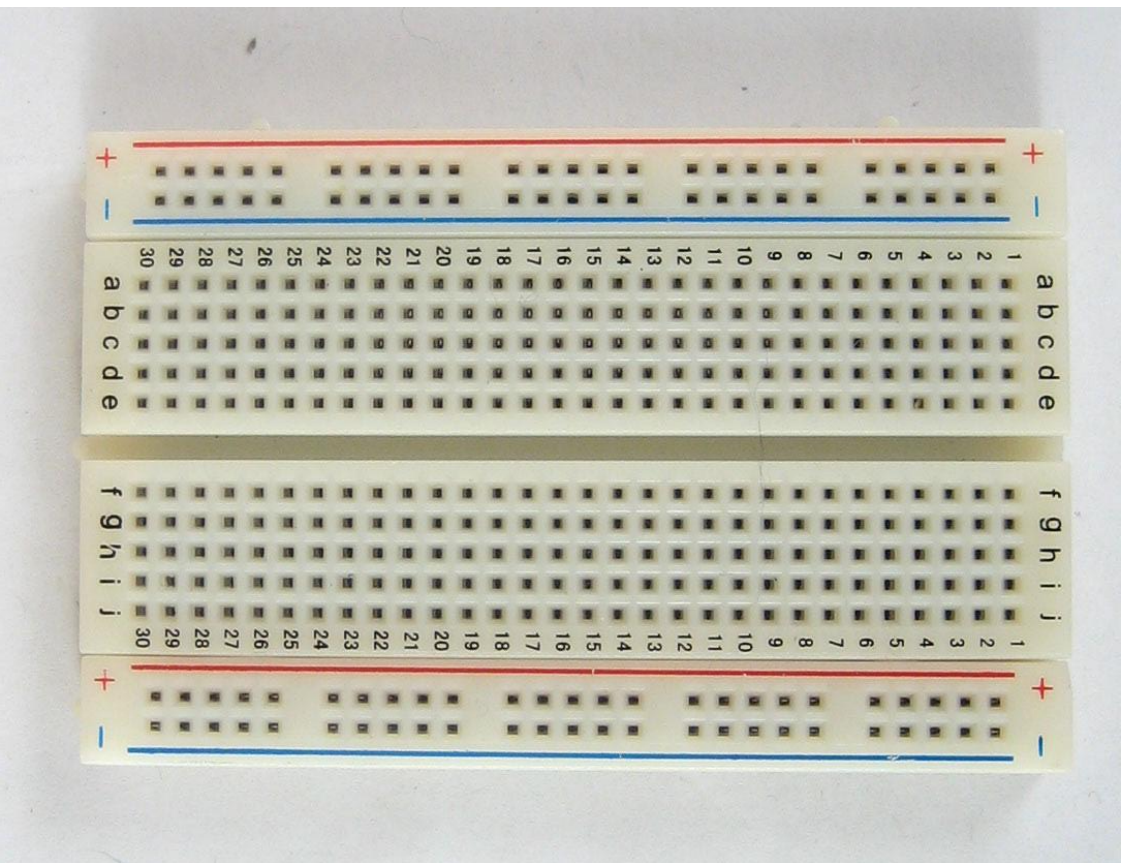**Table2: Data for Thevenin equivalent circuit**

No. of Obs	Values of R_L	Load Voltage V_L	Load current I_L
1.	1kΩ		

PART 2: Verify maximum power Transfer theorem
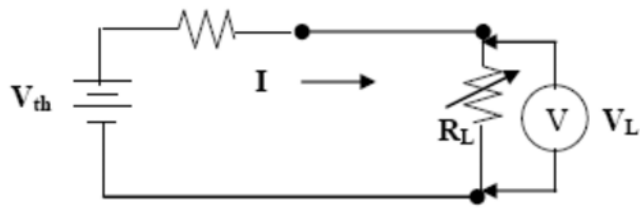
EXPERIMENTAL DATA:

No. of Obs.	V_{TH} (V)	V_L (V)	I (mA)	$P_{IN}=V_{TH}I$ (mW)	$P_{OUT}=V_LI$ (mW)	LOSS= P_{IN} -P_{OUT}	%η	%VR	$R_L=V_L/I$
1.									470Ω
2.									1kΩ
3.									1.2kΩ
4.									2kΩ
5.									2.2kΩ
6.									3.3kΩ
7.									4.7kΩ

BRAC UNIVERSITY
Department of Electrical and Electronic Engineering
Course No.: CSE250

Experiment 7

Name of the Experiment:

Familiarization with the alternating current (AC) waves

Objective:
In this experiment, we shall study some aspects of sinusoidal waveform, and correlate these with practically measurable values such as- rms. value (also called effective value), phase angle and time period. Also an exposure to simple ac circuit and some circuit elements are made. Try to familiarize yourself with

- Oscilloscope
- How to measure peak value, phase angle and time period (or frequency) using oscilloscope
- The methods of measuring rms. value both using oscilloscope and multimeter
- Difference between AC & DC setting of multimeter & oscilloscope
- Capacitor, resistor and breadboard

Introduction:
Any periodic variation of current or voltage where the current (or voltage), when measured along any particular direction goes positive as well as negative, is defined to be an AC quantity. Sinusoidal AC wave shapes are the ones where the variation (current or voltage) is a sine function of time.
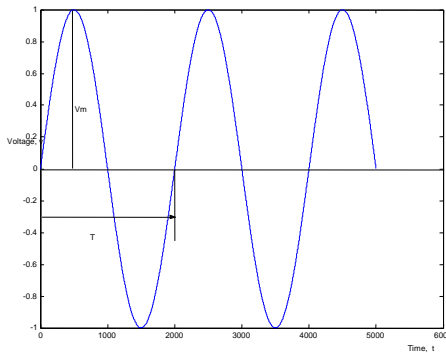

Figure 1(a): AC Voltage waveform

Here, Time period = T, Frequency, $f = \dfrac{1}{T}$.

$v(t) = V_m \sin(2\pi ft)$

Effective value:

The general equation of rms. value of any function (voltage, current or any other physical quantity for which rms. calculation is meaningful) is given by the equation,

$$V = \sqrt{\frac{1}{T}\int_0^T v^2 \, dt}$$

Now, for sinusoidal functions, using the above equation we get the rms. value by dividing the peak value (V_m) by square root of 2. That is,

$$V = \sqrt{\frac{1}{T}\int_0^T (V_m \sin(2\pi ft))^2 \, dt}$$

$$= \sqrt{\frac{1}{2\pi}\int_0^{2\pi} (V_m \sin(\theta))^2 \, d\theta} \; = \; \frac{V_m}{\sqrt{2}}$$

Similarly, for currents, $I = \dfrac{I_m}{\sqrt{2}}$. These rms. values can be used directly for power calculation. The formula for average power is given by $P_{avg} = \dfrac{1}{T}\int_0^T (vi)\,dt$. And for sinusoids this leads to $P_{avg} = VI \cos(\theta)$. Here, V and I are rms values and θ is the phase angle between voltage wave and current wave. The phase angle is explained in the next section.

Phase Angle:

Phase difference between two ac sinusoidal waveforms is the difference in the electrical angle between two identical points of the two waves. In figure 2, the voltage and current equations are given as:

V = V_m sin(2 π ft)
I = I_m sin (2 π f.t - θ)

Impedance:

Fig 1(b): Two sinusoidal waves with phase difference
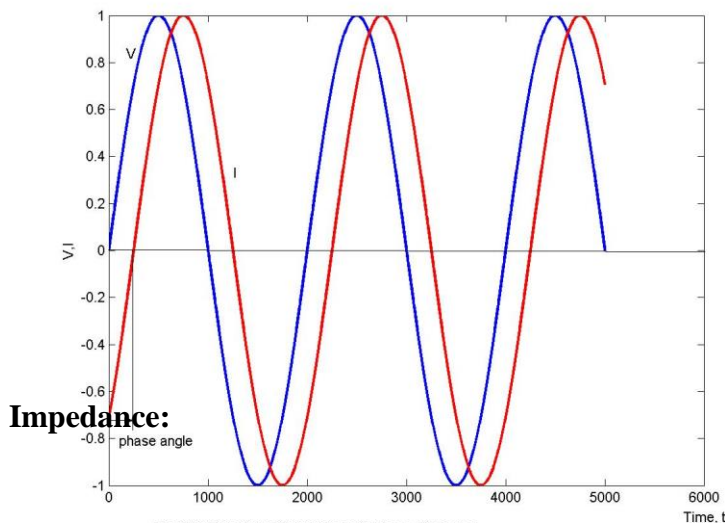
For, ac circuit analysis, impedance plays the same role as resistance plays in dc circuit analysis. It can be stated fairly safely that, the concept of impedance is the most important thing that makes the ac analysis so much popular to the engineers. As you will see in your later courses, any other periodic forms of time varying voltages or currents, are converted into an equivalent series consisting of sines and cosines (much like any function can be expanded by the power series of the independent variable using the Taylor series), only because the analysis of sinusoidal voltages are very much simple due to the impedance technique.

What is impedance anyway? Putting it simply, it is just the ratio of rms voltage across the device to the rms current through it. That is:

$$Z = \frac{V}{I\angle\theta} = \frac{V_m}{I_m\angle\theta}$$

Its unit is ohms.

Equipments:

1. Oscilloscope
2. Function generator (used as ac source)
3. Resistors : 1k, 220 Ω
4. Capacitor : 1 μf
5. Multimeter
6. AC ammeter
7. Switch : SPST
8. Breadboard

Circuit Diagram:
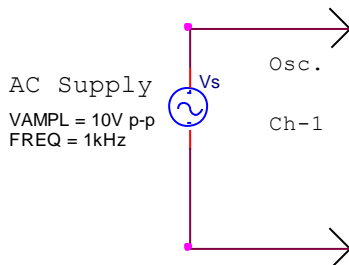
Figure-2
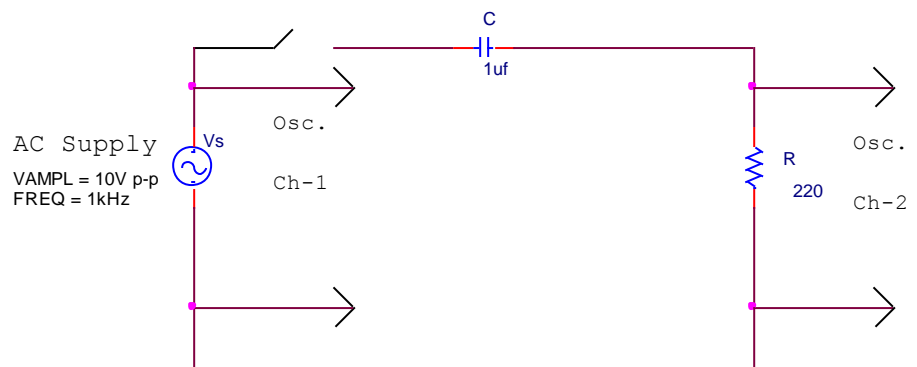
Figure 3

Procedure:

1. Connect the output of the function generator directly to channel 1 of the oscilloscope as shown in figure 2. Set the amplitude of the wave at 10v peak to peak and the frequency at 1 kHz, Select sinusoidal wave shape.

2. Sketch the wave shape observed on the oscilloscope. Determine the time period of

the wave and calculate the frequency.

3. Measure the voltage with an ac voltmeter.

4. Change the frequency to 500 Hz and note what happens to the display of the wave. Repeat when the frequency is increased 2 kHz.

5. Construct the circuit as shown in figure 3. Measure the input voltage with multimeter with ac voltage mode and the input current, with an ac ammeter. The ratio between the voltage to the current gives the magnitude of the impedance Z.

6. Observe the wave shapes in channel 1 and 2 **simultaneously**. Find the frequency of both the waves (are they equal to the supply frequency) and their amplitude from the display. The phase difference is given by **360f.t degree**, where 't' is the **time delay** between the two waves. Note that the voltage in channel 2 is the voltage across a resistance and hence this is in phase with the current flowing in the circuit.

Report:

1. Compare the frequency of the wave determined from the oscilloscope in step 2 of the procedure with the mentioned value on the function generator.

2. Calculate the rms value of the voltage observed in step 2 of the procedure and compare with that measured in step 3.

3. How does the time period vary when the frequency of the wave is changed in step 4?

4. Calculate the magnitude of the impedance from the readings taken in step 5.

5. Find the magnitude and phase angle of the impedance from the readings taken in step 5 and 6.

BRAC University
Department of Electrical and Electronic Enginering
Introduction to PSpice

Objective

The objective of this module is to learn the fundamentals of computer aided circuit simulation using PSpice.

Introduction

SPICE is a powerful general purpose analog circuit simulator that is used to verify circuit designs and to predict the circuit behavior. This is of particular importance for integrated circuits. It was for this reason that SPICE was originally developed at the Electronics Research Laboratory of the University of California, Berkeley (1975), as its name implies

<p align="center"><u>S</u>imulation <u>P</u>rogram for <u>I</u>ntegrated <u>C</u>ircuits <u>E</u>mphasis</p>

PSpice program is extensively used to simulate and predict the behavior of experimental circuits. Simulation is very important as it allows a potential circuit to be tested for errors before it is actually built, thus saving time and cost.

Some Facts and Rules about PSpice

- PSpice is not case sensitive. This means that names such as *Vbus*, *VBUS are vbus* are equivalent.
- All element names must be unique.
- In PSpice circuits can be analyzed in two ways
  o Text based: using "Netlist" to code up circuits
  o Graphics based: using "Schematics" to draw circuits and simulate

Large and Small Numbers in PSpice

Unfortunately, PSpice cannot recognize Greek fonts or even upper vs. lower case. Thus our usual understanding and use of the standard metric prefixes has to be modified. The metric prefix designations used in PSpice are:

Number	Prefix	Common Name
10^{12}	T or t	tera
10^9	G or g	giga
10^6	MEG or meg	mega
10^3	K or k	kilo
10^{-3}	M or m	mili
10^{-6}	U or u	micro
10^{-9}	N or n	nana
10^{-12}	P or p	pico
10^{-15}	F or f	femto

An alternative to this type of notation, which is in fact, the default for PSpice output data, is "textual scientific notation." This notation is written by typing an "E" followed by a signed or unsigned integer indicating the power of ten. Some examples of this notation are shown below:

<p align="center">656,000 = 6.56E5</p>

<p align="center">-0.0000135 = -1.35E-5</p>

# Part 2: Introduction to Schematic

In this part you will learn to use the PSpice circuit simulation with the schematic capture front end, Schematics. Click **Schematic** button in the MicroSim Design Manager window to get the following schematic window.
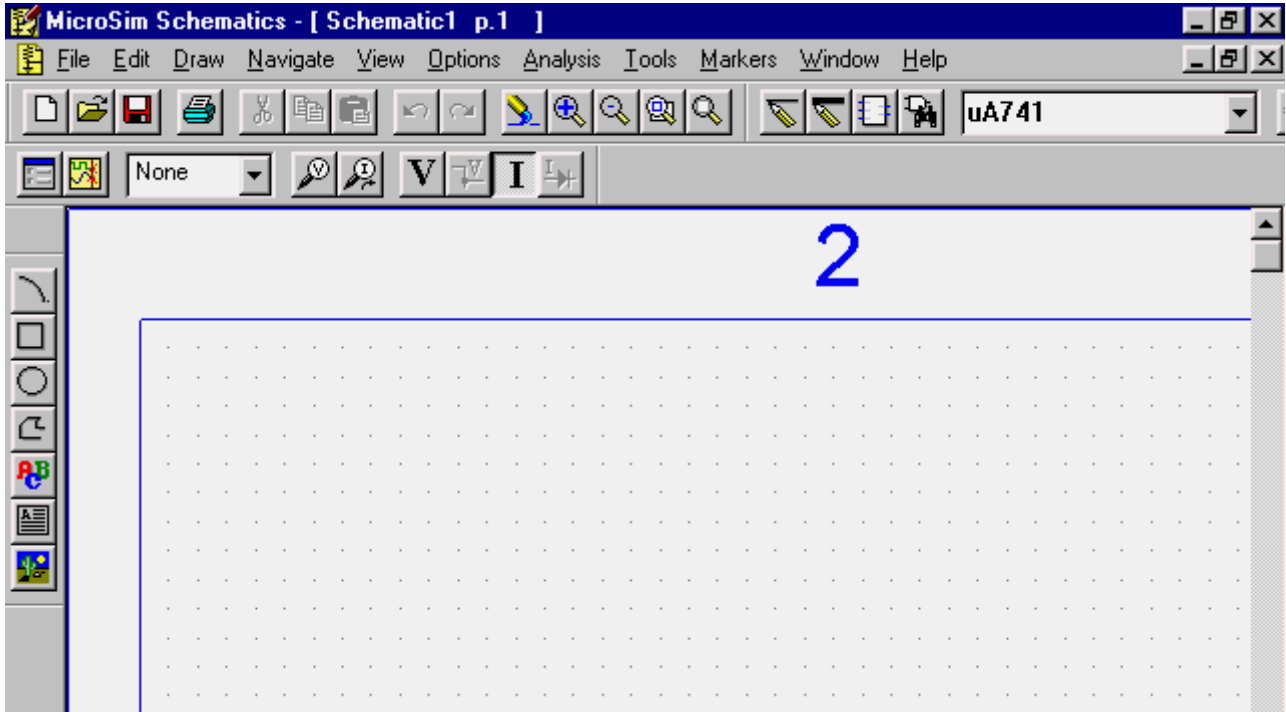
A. Drawing the Circuit

1. Getting the Parts

- The first thing that you have to do is get some or all of the parts you need to simulate your circuit.
- This can be done by
  - Clicking on the 'get new parts' button ⬛, or
  - Pressing "Control+G", or
  - Going to "Draw" and selecting "Get New Part..."
- Once this box is open, select a part that you want in your circuit. This can be done by typing in the name or scrolling down the list until you find it
- Some common parts are:
  - r - resistor
  - GND_ANALOG or GND_EARTH -- this is very important, **you MUST have a ground in your circuit**
  - VAC and VDC – voltage sources
  - IAC and IDC – current sources
- Upon selecting your parts, click on the place button then click where you want it to be placed.
- Once you have all the parts you think you need, close that box. You can always open it again if you need more or different parts.
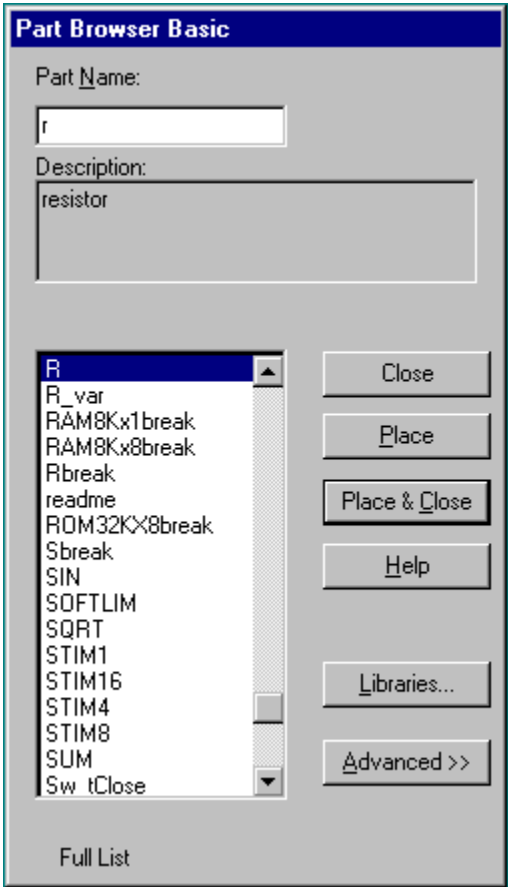
2. Placing the Parts

- You should have most of the parts that you need at this point.
- Now put them in the places that make the most sense (usually a rectangle works well for simple circuits). Just select the part and drag it where you want it.
- To rotate parts so that they will fit in your circuit nicely, click on the part and press "Ctrl+R" (or Edit "Rotate"). To flip them, press "Ctrl+F" (or Edit "Flip").
- If you have any parts left over, just select them and press "Delete".

3. Connecting the Circuit

- Now you'll have to connect the parts with wires.
- Go up to the tool bar and
  - select "Draw Wire" ![icon] or
  - "Ctrl+W" or
  - go to "Draw" and select "Wire".
- With the pencil looking pointer, click on one end of a part, when you move your mouse around, you should see dotted lines appear. Attach the other end of your wire to the next part in the circuit.
- Repeat this until your circuit is completely wired.
- If you want to make a node (to make a wire go more then one place), click somewhere on the wire and then click to the part (or the other wire). Or you can go from the part to the wire.
- To get rid of the pencil, right click or press "Esc".
- If you end up with extra dots near your parts, you probably have an extra wire, select this short wire (it will turn red), then press "Delete".
- If the wire doesn't go the way you want (it doesn't look the way you want), you can make extra bends in it by clicking in different places on the way (each click will form a corner).

4. Changing the Name of the Part

- To change the name, double click on the present name (C1, or R1 or whatever your part is), then a box will pop up (Edit Reference Designator). In the top window, you can type in the name you want the part to have.
- Please note that if you double click on the part or its value, a different box will appear.

```
Edit Reference Designator                    [X]

Package
Reference Designator:    [R1              ]

Gate:                    [          ▼]

Package Type:            [RC05          ▼]
(Footprint)

         [ OK ]      [ Cancel ]
```
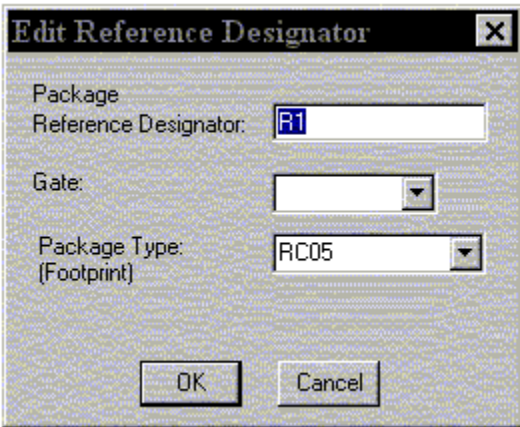
5. Changing the Value of the Part

```
Set Attribute Value          [X]
          VALUE
[1k                            ]
      [ OK ]    [ Cancel ]
```
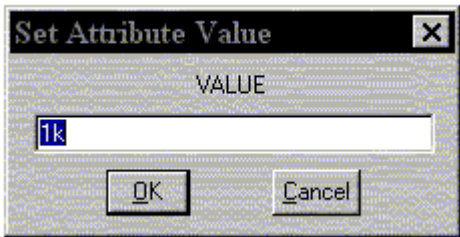
- If you only want to change the value of the part, you can double click on the present value and a box called "Set Attribute Value" will appear. Type in the new value and press OK.

- If you double click on the part itself, you can select VALUE and change it in this box.

```
V1  PartName: vdc                              [X]

Name                 Value
[REFDES        ]  = [V1                    ]      [ Save Attr ]

 * REFDES=V1                                      [ Change Display ]
 * TEMPLATE=V^@REFDES %+ %- @DC
   DC=0V                                          [ Delete ]
 * SIMULATIONONLY=
 * PART=vdc
 * MODEL=
   PKGREF=V1

 ☑ Include Non-changeable Attributes              [ OK ]
 ☑ Include System-defined Attributes              [ Cancel ]
```
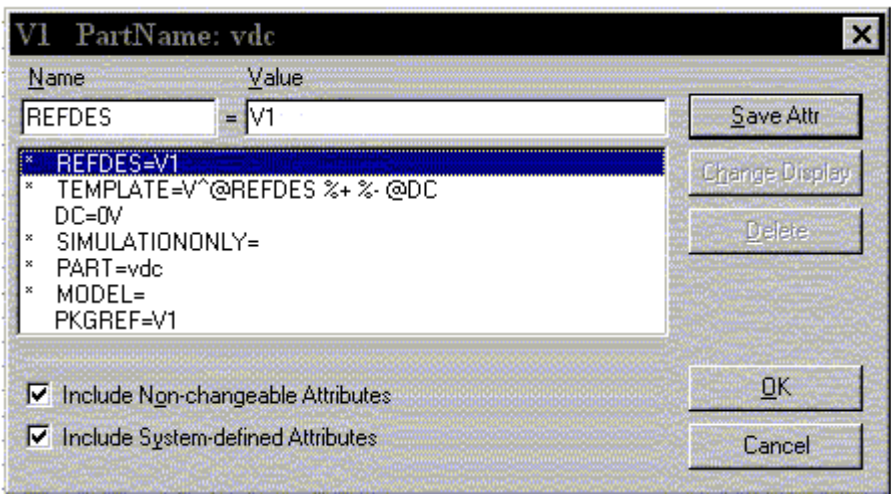
6. Making Sure You Have a GND

This is very important. You cannot do any simulation on the circuit if you don't have a ground. If you aren't sure where to put it, place it near the negative side of your voltage source.

7. **Voltage and Current Bubbles**

- These are important if you want to measure the voltage at a point or the current going through that point.
- To add voltage or current bubbles, go to the right side of the top tool bar and select "Voltage/Level Marker" (Ctrl+M) ⊘ or "Current Marker" ⊘. To get either of these, go to "Markers" and either "Voltage/Level Marker" or "Current Marker".

B. Analysis

- Open the analysis menu by clicking the ▣ button. Enable the appropriate analysis options and the press close.
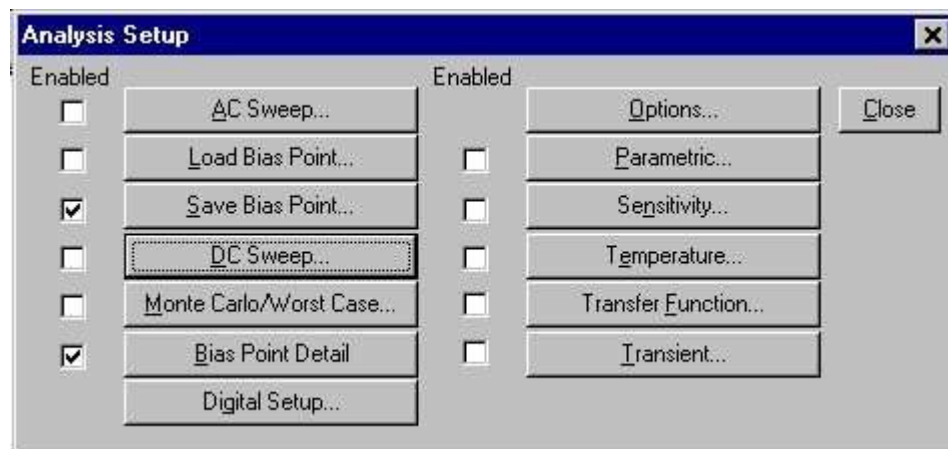
- Click on the Simulate button on the tool bar ▨ (or Analysis, Simulate, or F11).

C. Probe

1. **Before you do the Probe**

- You have to have your circuit properly drawn and saved.
- There must not be any floating parts on your page (i.e. unattached devices).
- You should make sure that all parts have the values that you want.
- There are no extra wires.
- It is very important that you have a ground on your circuit.
- Make sure that you have done the Analysis Setup and that only the things you want are enabled.

2. **To Start the Probe**

- Click on the Simulate button on the tool bar ▨ (or Analysis, Simulate, or F11).
- It will check to make sure you don't have any errors. If you do have errors, correct them.
- Then a new window will pop up. Here is where you can do your graphs.

3. **Graphing**

- If you don't have any errors, you should get a window with a black background to pop up
- If you did have errors, go To "View - Output File" to check the errors.

4. **Adding/Deleting Traces**
- PSpice will automatically put some traces in.
- To change them go to Trace - Add Trace or ⩘ on the toolbar. Then select all the traces you want.
- To delete traces, select them on the bottom of the graph and press "Delete".

5. **Doing Math**

- In Add Traces, there are functions that can be performed, these will add/subtract (or whatever you chose) the lines together.
- Select the first output then either on your keyboard or on the right side, click the function that you wish to perform.
- It is interesting to note that you can plot the phase of a value by using IP(xx), where xx is the name of the source you wish to see the phase for.
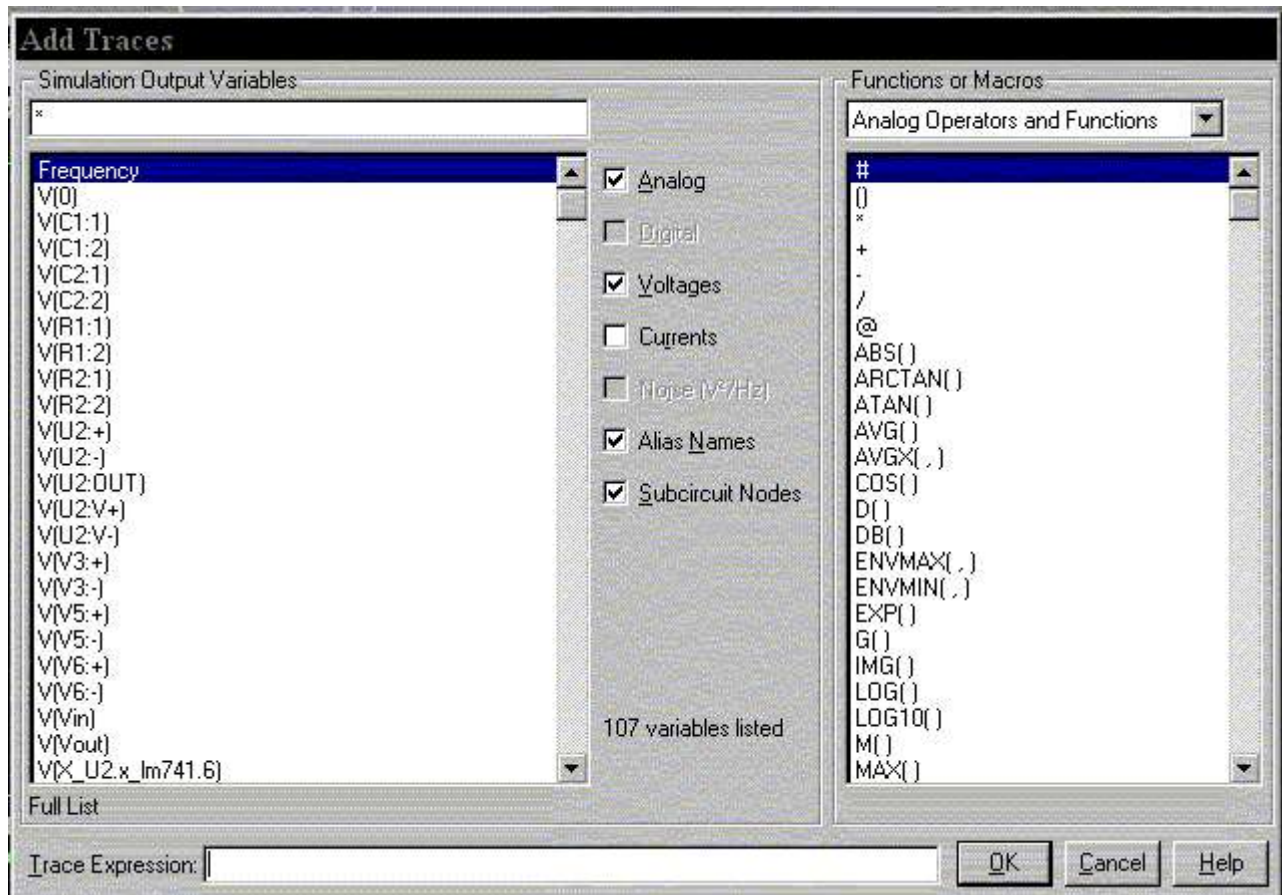
6. Finding Points

- There are Cursor buttons that allow you to find the maximum or minimum or just a point on the line. These are located on the toolbar (to the right).

- Select which curve you want to look at and then select "Toggle Cursor" .

- Then you can find the max, min, the slope, or the relative max or min (is find relative max).

7. Saving

- To save your probe you need to go into the tools menu and click display, this will open up a menu which will allow you to name the probe file and choose where to save it. You can also open previously saved plots from here as well.

Circuit Example

- Using the steps explained above draw and simulate the following circuit. This is the same circuit that we used for our example 1.

- In Analysis Setup window only enable "**Bias Point Detail"** option.
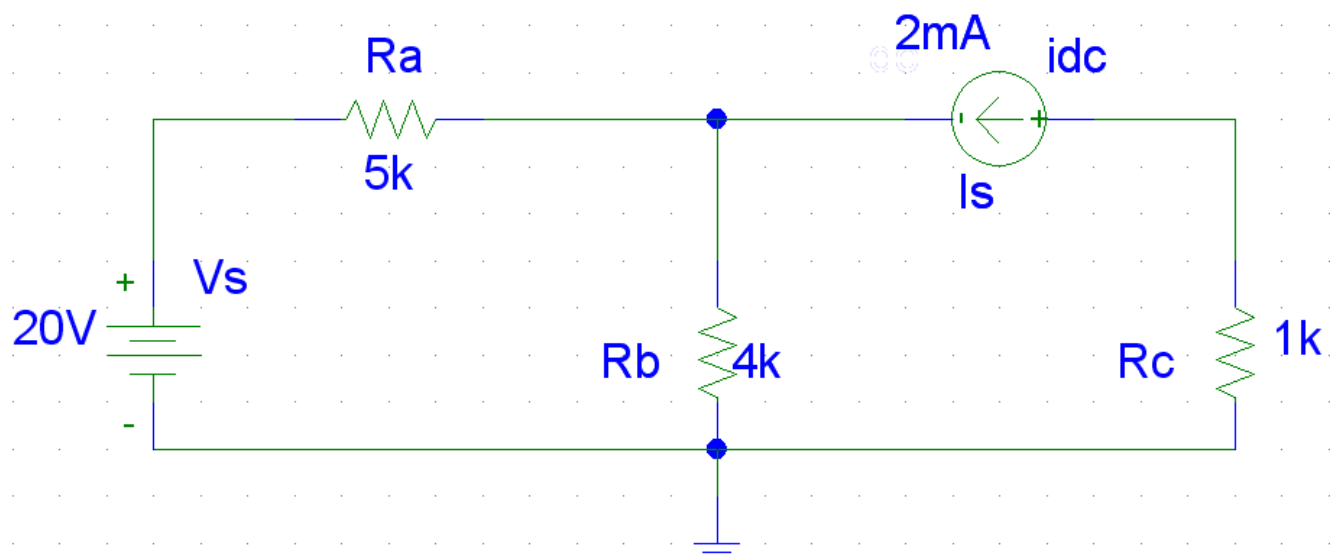
- To examine the node voltages and current through each part go to Analysis menu in the Schematic window and then Display results on Schematic. Put the check marks beside Enable, Enable Voltage Display and Enable Current Display.
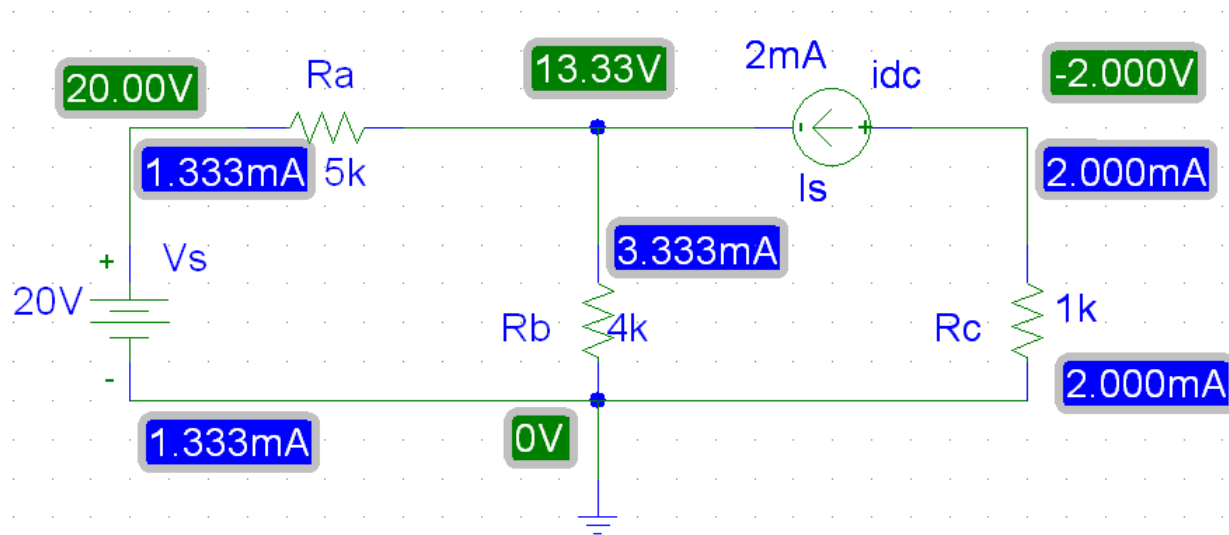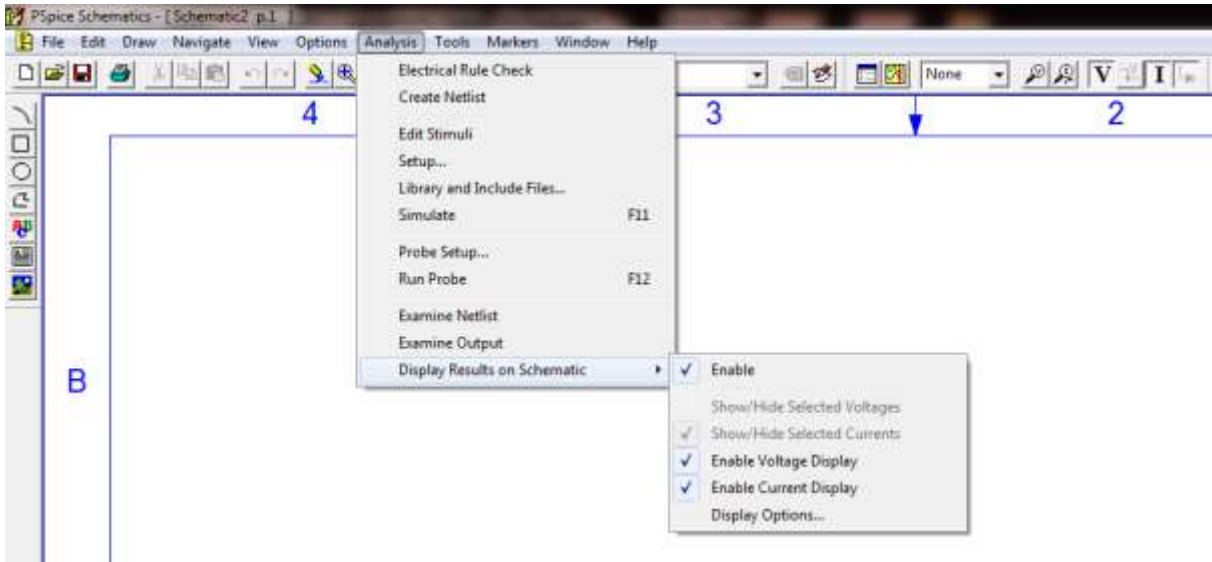
Varying Resistance in Schematics:
1. Place a 1k ohm resistance R_L and then draw the circuit diagram in **Microsim Schematic window**. Save the file.
2. First, double-click the value label of the resistor, R_L that is to be varied. This will open a "Set Attribute Value" dialog box. Enter the name **{RVAR}** (including the curly braces) in place of the component value. Choose "**Get New Part**" from the menu and select the part named **param.**
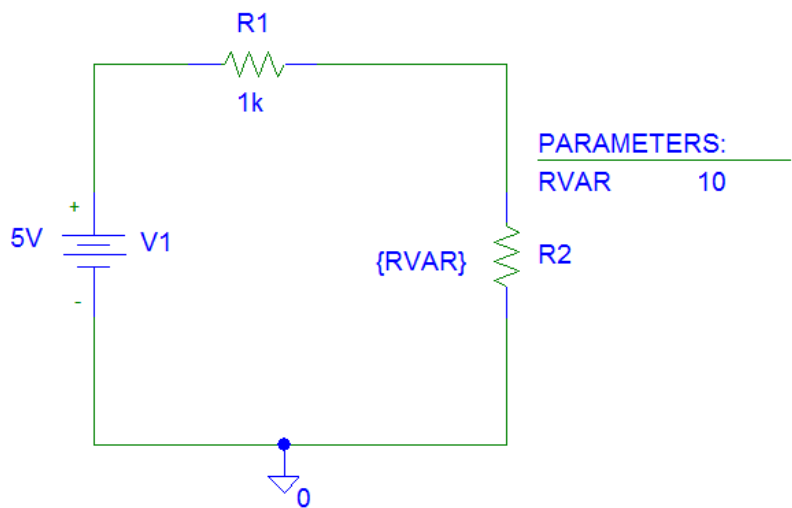
3. Place the box anywhere on the schematic page. Now double-click on the word **PARAMETERS** in the box title to bring up the parameter dialog box. Set the NAME1= **RVAR** (no curly braces) and the VALUE1= 10 (any value) to the nominal resistance value.
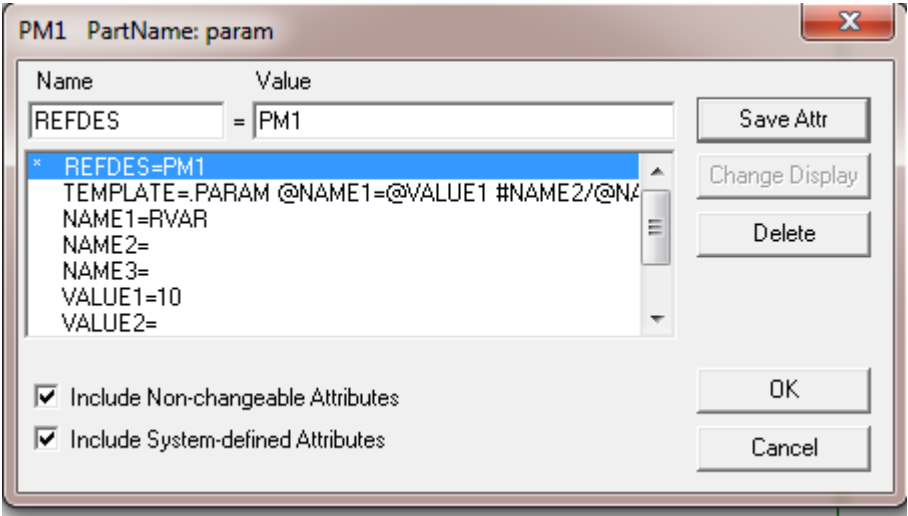
4. In the Analysis Setup dialog box, click the 'DC Sweep' button and Select 'Linear' type and 'Global Parameter' as a sweep variable. Type **RVAR** as a sweep variable 'Name' with Start value = 100, End value = 2000 and Increment = 100.
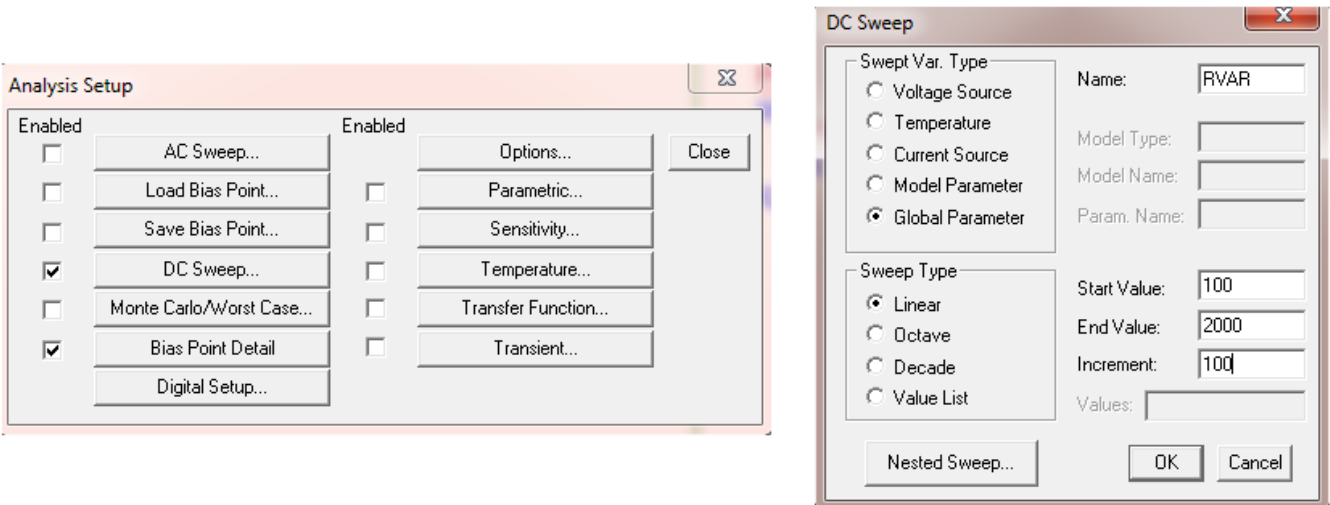
Transient Analysis Basics:

Transient Analysis is used to observe the behavior of a circuit parameter in time domain.

In the Analysis Setup dialog box, click the 'Transient' button and hence set the timing parameters.
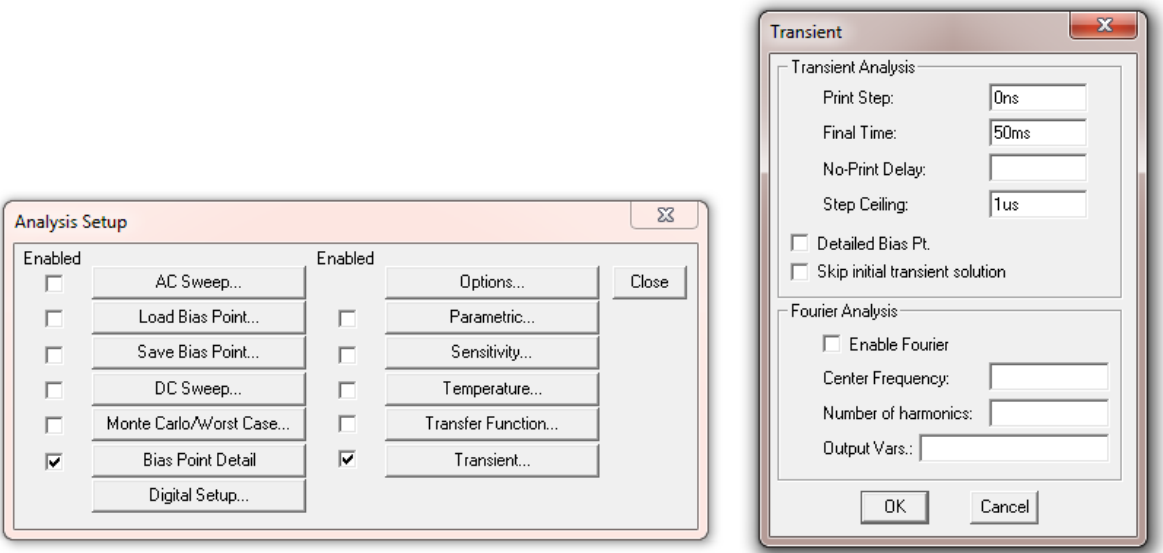
The first parameter, *print step* is the frequency with which data is saved. The actual time steps used by PSpice may be different from this.

The second parameter, *final time* is the value of time at which the simulation will be ended. Since PSpice starts at t = 0, there will be a total of 50ms time span of simulation for the circuit.

The third parameter, *print delay* is the print delay time. In some cases, we do not want to store the data for the entire time span of the simulation. Most of the time, this parameter is set to zero or not used.

The fourth parameter, *step ceiling* is the maximum time step size PSpice is allowed to take during the simulation. Since PSpice automatically adjusts its time step size during the simulation, it may increase the step size to a value greater than desirable for displaying the data. When the variables are changing rapidly, PSpice shortens the step size, and when the variables change more slowly, it increases the step size. Use of this parameter is optional.

Creating a time-varying source using VPULSE:

To create a time-varying periodic source of arbitrary waveshape **VPULSE** is used. Choose "**Get New Part**" from the menu and select the part named **VPULSE.** Now double-click on its symbol to bring up the following dialog box. Set the parameter values appropriately.
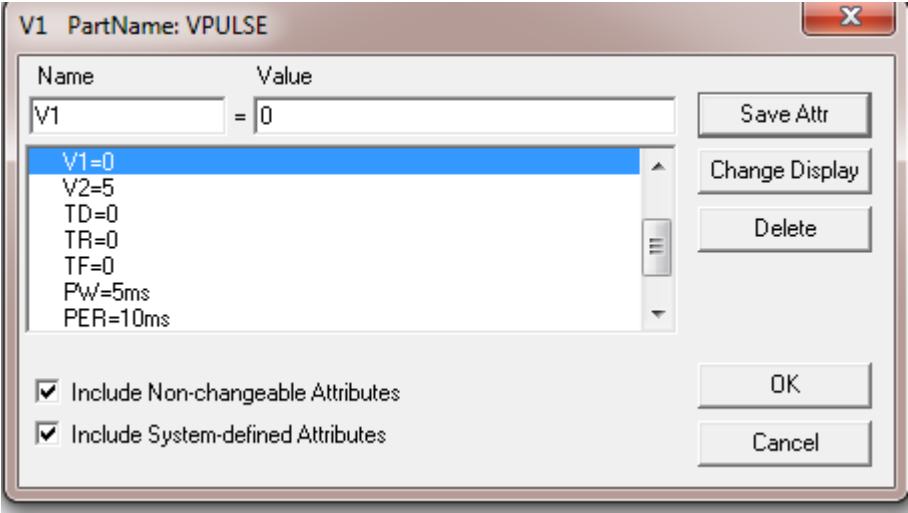
For a 100Hz square wave signal ranging from 0 to 5V with 50% duty cycle
set DC=0, AC=0, V1=0, V2=5, TD= 0, TR=0, TF=0, PW=5ms, PER=10ms.

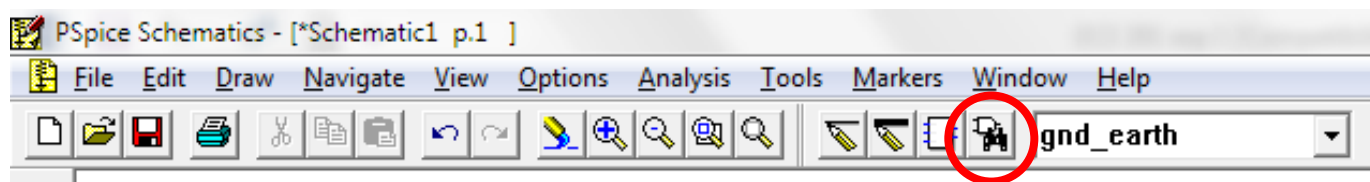The significance of the parameters are explained in the figure below.
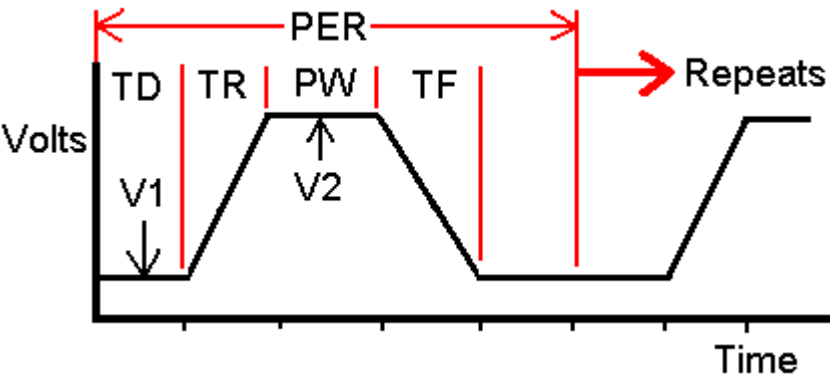
1.   Go to the Start menu and locate the application **"Schematics"**
2.   Now, set up the following circuit following the instructions given below-
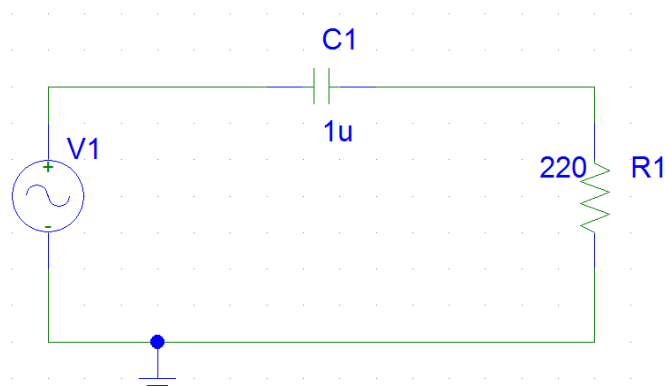

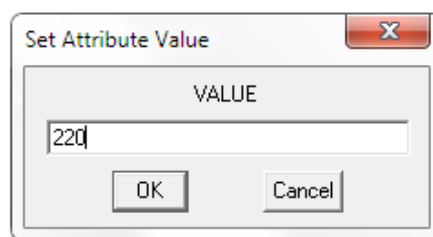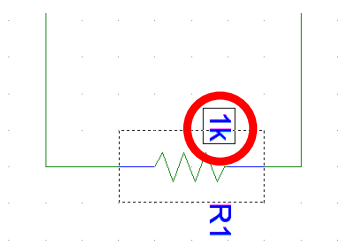3.   Get the following parts by clicking the **"Get new Part"** button from the toolbar above, OR press Ctrl+G.

In the box "Part Name" that appears, type in-
a)   **Vsin** (for AC Voltage source)
b)   **R** (Resistor)
c)   **C** (Capacitor)
d)   **Gnd_earth-** (for ground- i.e. a reference potential from which all voltages are measured)

4.   Place the parts as shown in the first figure. To rotate a part, select the part and press Ctrl+R. After that, connect each part using wires. To select wire, use the button from the toolbar above.
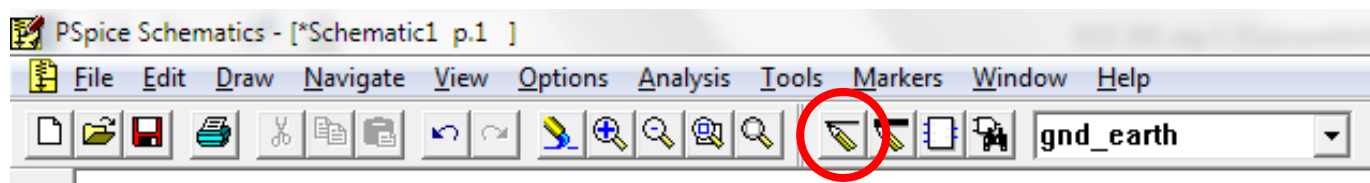
5.   To change the parameter values (resistance for resistors, capacitance for capacitors, etc) simply double click on the default value that is shown alongside the part, and enter the desired value in the window that appears.
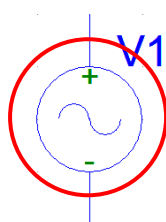
Pspice Schematics understands-

n for nano
u for micro
k for kilo

and so on.
So, to enter the capacitance value of 1 micro farad, enter "**1u**".

6.   Set up the AC voltage source by double clicking on it. In the window that appears, enter the values as shown-
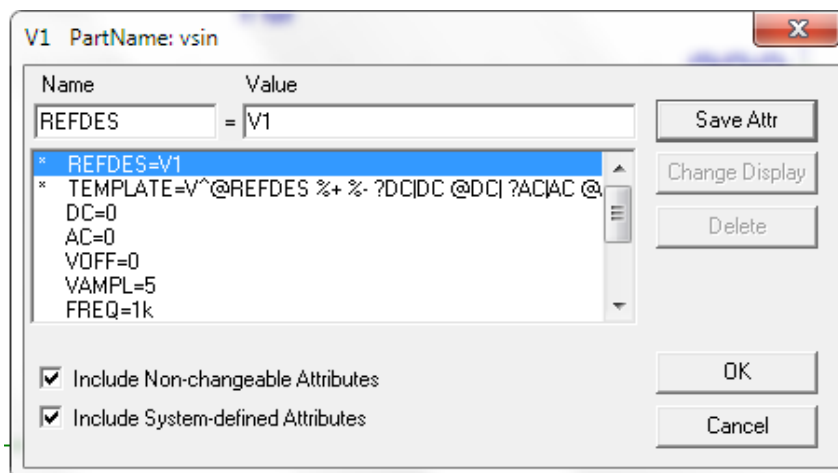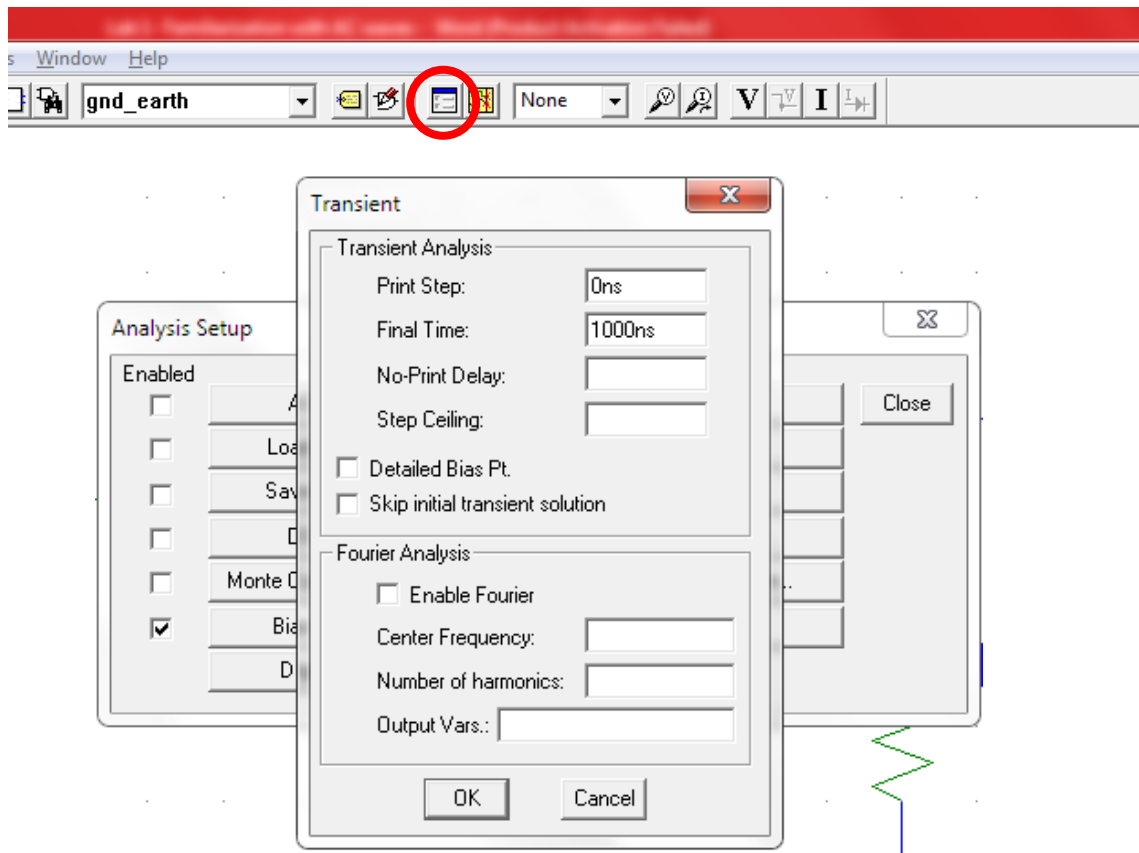

DC=0
AC=0
VOFF=0
VAMPL=5 (if you want 10V peak to peak)
FREQ=1k

There is no need to change any other values.

7.   Get the voltage markers using the button from the toolbar above, and place the markers above the resistor (load) and the voltage source (Vsin).
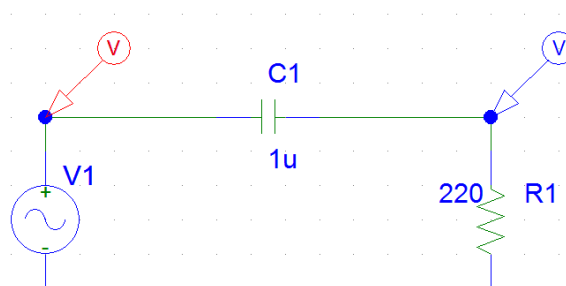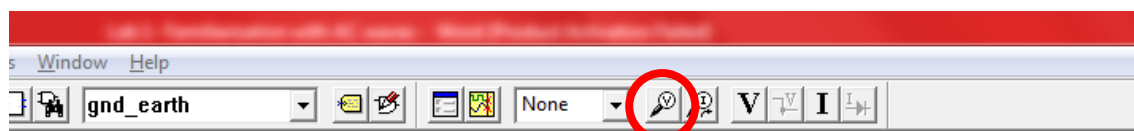
8.   Next, from the toolbar again, press on the 'Setup Analysis' button. In the window that appears, click on the button 'Transient'.


For '**Print Step**', type in a very small value, e.g.- **1ns**. (ns= nano seconds)

For '**Final Time**', type in 4 ms (ms=millisecond). The final time tells the software when to stop the simulation. We want to see the input and output waveshapes for 4 complete cycles of the input sinusoid; the frequency of our sine wave source was kept at 1 khz, which gives the corresponding time period to be= 1ms. Thus, to see 4 complete cycles, we need to enter (4x1=) 4 ms.

Press 'OK', and then press 'Close' to get back to the schematic design.
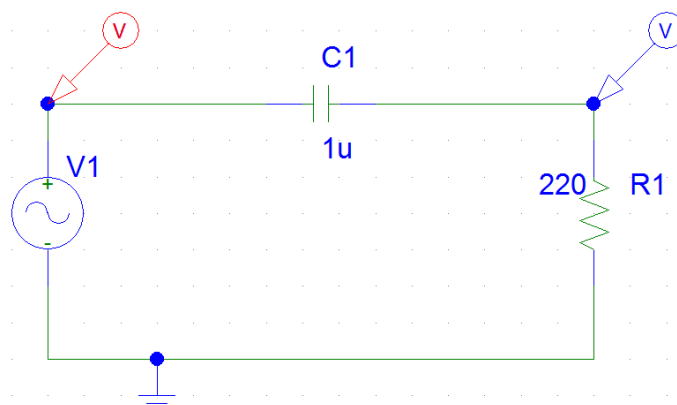
9.   Go ahead and save the schematic in your desired directory, and press the button 'Simulate' from the toolbar above-
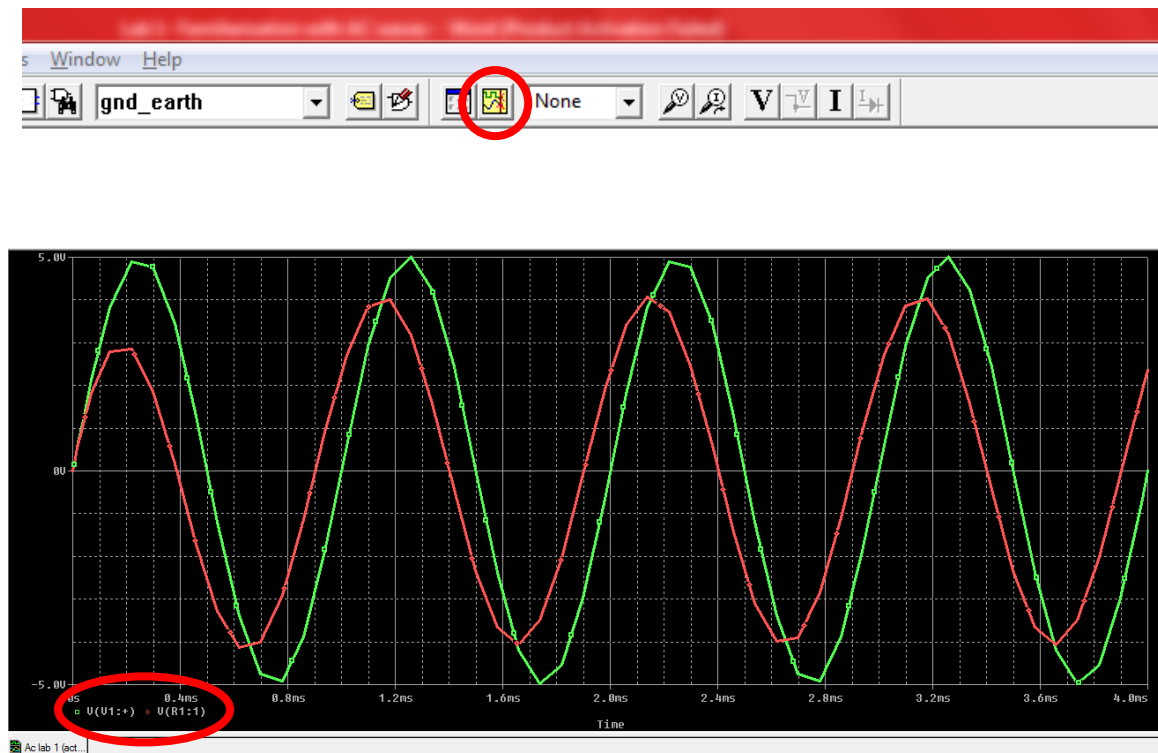

In a new window, you will be shown the simulation results as seen in the figure above. The red circle at the bottom shows where you can find which waveshape is which. In this case, the green one is the input waveshape, and the red one is appearing across the load, i.e. the resistor.

10.   In the same window on the top ribbon, find and press the 'Toggle cursor' button.
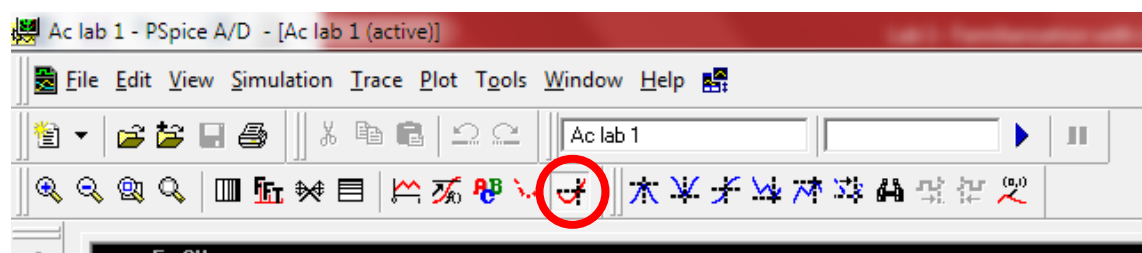
Notice that as you click on the waveshapes, a cursor runs along the curve. Left clicking selects one waveshape and its corresponding cursor, and right clicking selects the other. A small window with the values of the cursors appear on the window which can be dragged around.
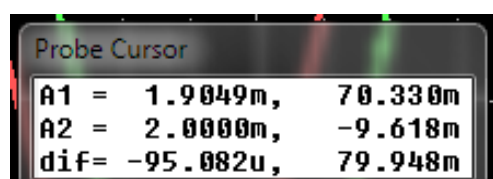
Position the two cursors, such that they both are at the adjacent zero crossings of the two waveshapes, so as to be able to measure the time difference between them. The difference in the x axis appears on the left hand column of the "Probe Cursor" window, beside 'dif'.

Using this information and the formula *2*pi*f*t*, find the phase difference between the two waveshapes, where, 't' is the time difference, 'f' is the source frequency, and 'pi' is equal to 3.142.

Change the source (from Vsin) frequency to 2 kHz, and then to 500 Hz, and for both the frequencies obtain the output and input waveshapes together. Using the cursors, determine the phase difference as well.

Include the schematic circuit design, the waveshapes for all the three frequencies and the calculated phase differences in a new document and submit the hard copy. (See the file- "How to submit Pspice Assignment" for submission guidelines).

Steps to analyze with AC voltage source:

1. Choose VSIN from get new parts and place it on the schematic window.
2. To change attribute double click on the VSIN symbol and a popup window will appear (see Figure)
3. Put values of VOFF (voltage offset), VAMPL(voltage amplitude) and FREQ (frequency) in the space provided under the name "Value". First select the parameter you want to put/change the value of (say, VOFF), the type the value (say, "0"), then press "Enter". Once you are done placing all the value to define the VSIN, click "OK" to get out of the window. As in the figure the voltage source is characterized with Amplitude of 10 V, frequency 100Hz and the offset is 0.
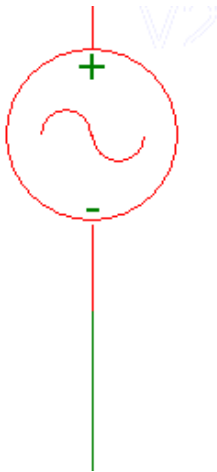
Steps to use variable resistor and how to simulate circuit using variable resistor

1. "get new part" choose "R_var" and place it on the schematic.
2. Double click on "R_var" symbol and Change the **value** of the part (not the name!) to {RL} (use curly braces, name is arbitrary) and then press "Enter". Click "OK".
3. Go to "get new part" select "PARAM" and place it on the schematic.
4. Double click on "PARAMETERS"
5. Set the name to RL (same name as in "a" but with no curly braces) to NAME1
6. Set VALUE1=1k and then "OK" to exit the window.

Simulation Settings:

1) Click on "Set-up Analysis"
2) Analysis type: DC Sweep
3) Sweep variable: Global parameter
4) Parameter name: RL (or name of the parameter you used without curly braces)
5) Sweep Type: Linear
6) Fill in the Start, End, and Increment values. Note for resistance, the start value cannot be 0! Use 0.1 instead.

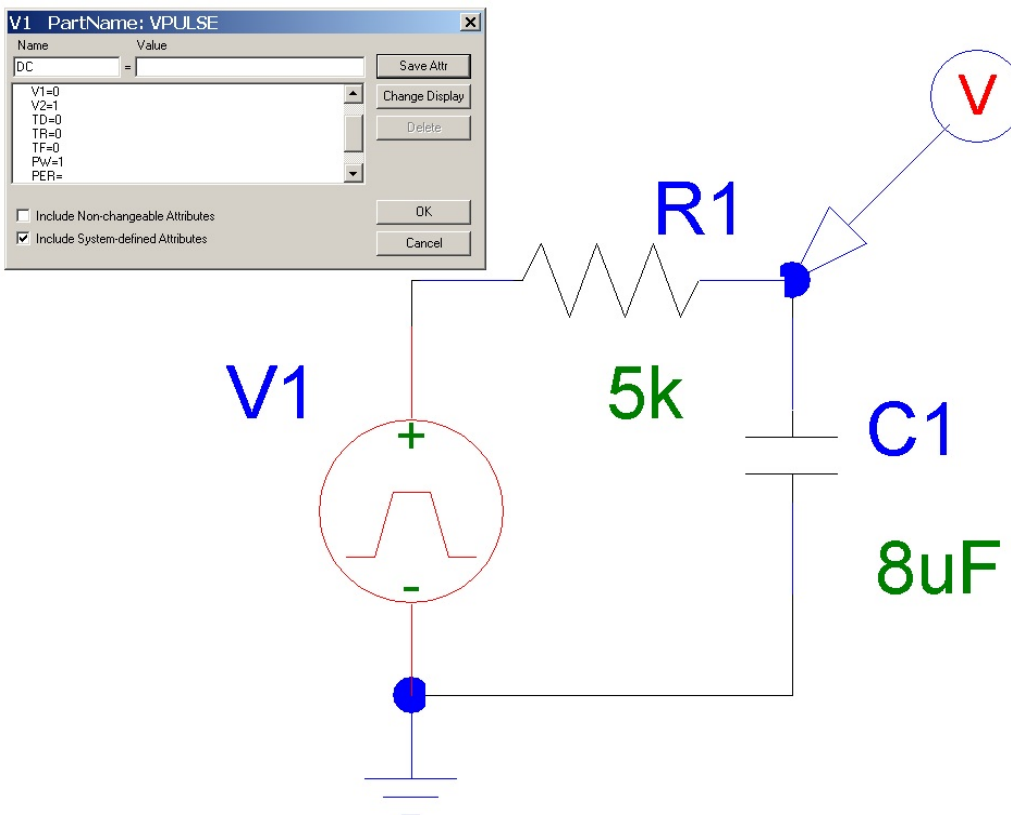Press OK and simulate

PARAMETERS:

RL   1k

R1

{RL}
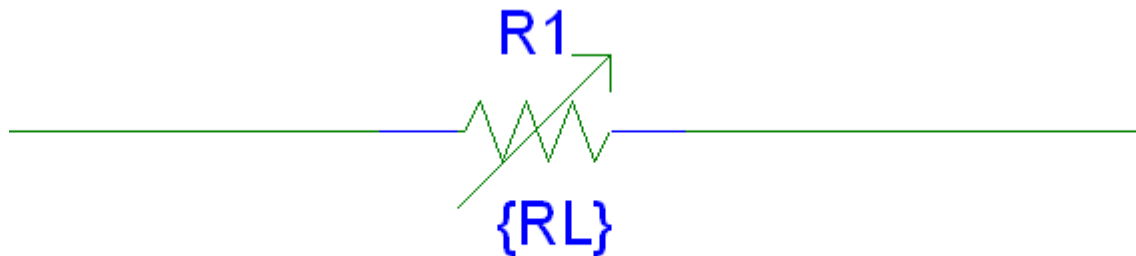
Transient Analysis on an RC circuit with PSPICE

Build the circuit as shown above in PSPICE. The voltage source is called VPULSE and is a pulse that starts at TD, has a rise time of TR, has a bottom voltage of V1, and a top voltage of V2, has a fall time of TF, and a pulse width of PW. The ground of the voltage source is obtained with EGND. The time constant for the above circuit is 40 ms. To do the analysis select transient from the setup menu. See the figure below:
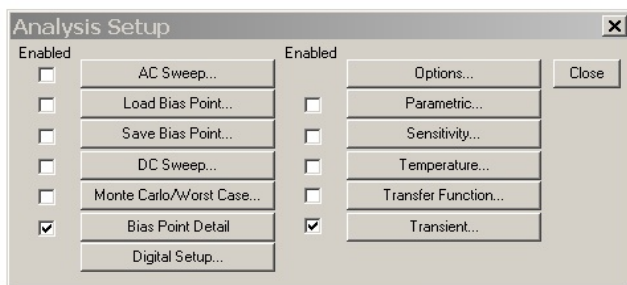

To observe the results of the simulation, just choose the simulation button. The results should look like the graph on the following page.
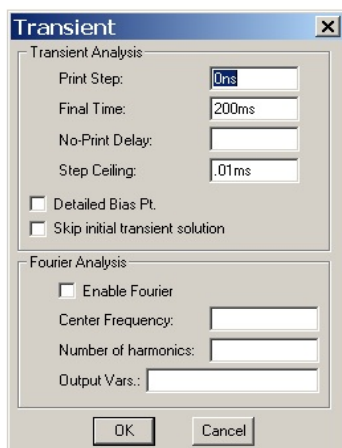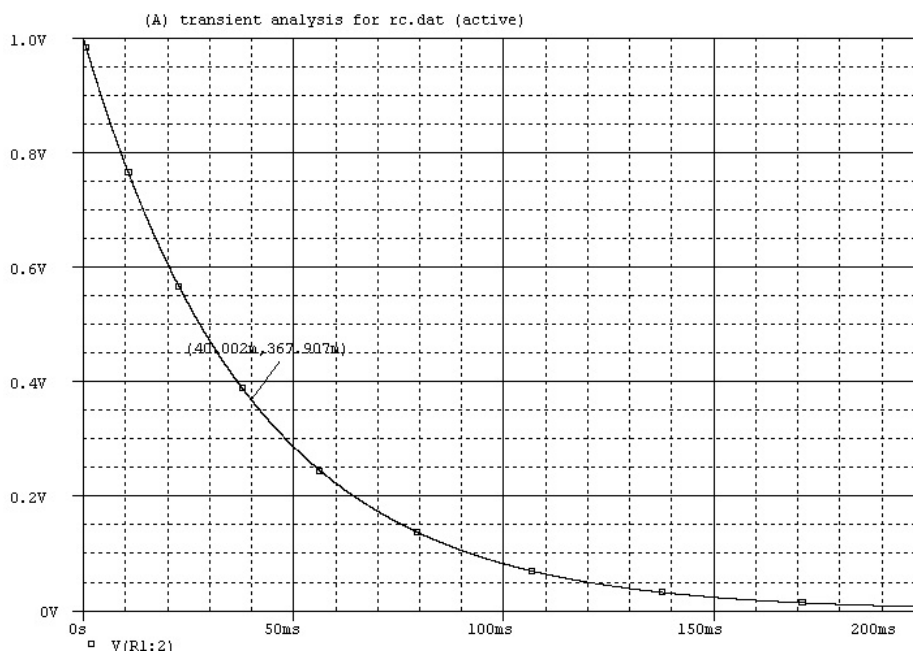

The parameters on the transient setup are:
Print step 0ns
Final Time 200 ms
Step Ceiling 0.01ms

Notice that the voltage at 40 ms is 0.63 volts as predicted from the charging equation for a capacitor.

(A) transient analysis for rc.dat (active)
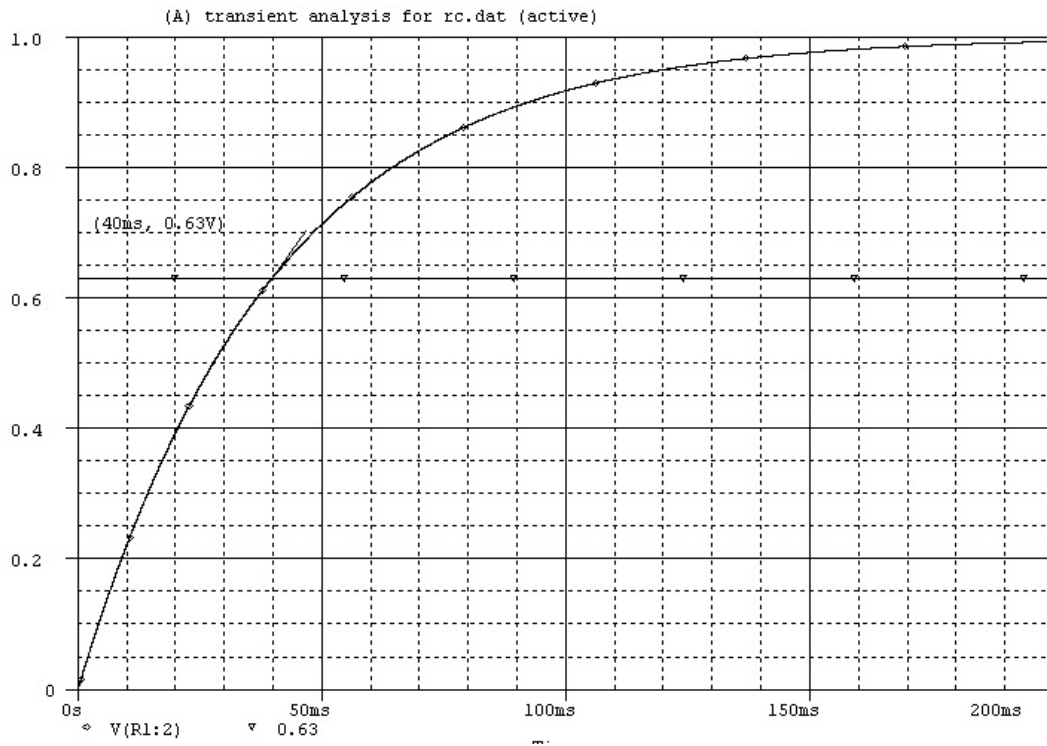
To get the discharge curve, simply exchange the values of V1 and V2. The output is as follows:

(A) transient analysis for rc.dat (active)

Changing the pulse width to 0.5 s, the final time to .8s, and displaying the pulse as well, one can see the charge and discharge curves for the capacitor. The pulse is in green and the voltage across the capacitor is red.

CSE250

Circuits & Electronics Laboratory

Lab – Report

Section: 00X

Experiment no.

Name of the experiment:

Type here.

Prepared by:

Name: _____ID: _____

Other Group members:

Sl.	ID	Name
1.	Type here.	
2.		
3.		
4.		

Additional comments (if any):

Report Format

Cover Page
Objectives
Apparatus
Circuit Diagram
Data Table
Graph
Data Sheet
Question/Answer
Discussion