Lab 1
Resistors & DC Analysis

A. Background

A. 1. Resistor Color Code

The resistors that we use in electronics are made of various materials. Most abundant are carbon resistors. There is a color code for resistance values. The resistance of a resistor is expressed in terms of a sequence of colored bands on the resistor.

The coding of color is defined in the international standard 1EC60062. Also, the entire body of the resistor is colored; there are one or more colors on the ends of the resistors (called as tip colors) and lastly, there is a spot of paint on the body.

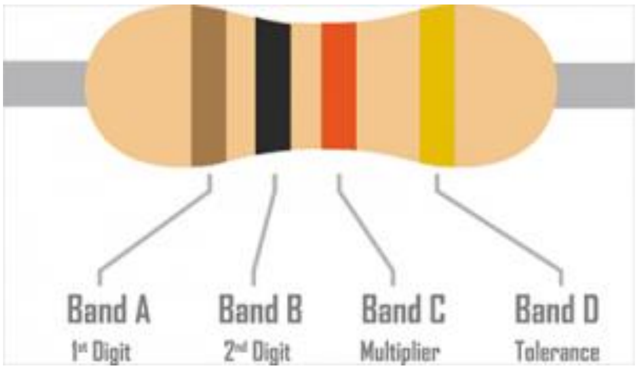
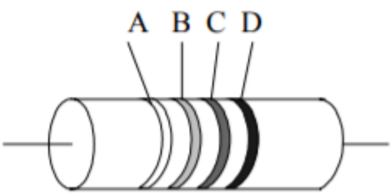


Figure 1. 4-band Resistors

Figure 2. Color Code Chart



A: First significant figure of resistance
B: Second significant figure
C: Multiplier
D: Tolerance

Color	Significant figure	Multiplier	Tolerance
Black	0	E0	
Brown	1	E1	
Red	2	E2	
Orange	3	E3	
Yellow	4	E4	
Green	5	E5	
Blue	6	E6	
Violet	7	E7	
Gray	8	E8	
White	9	E9	
Gold			%5
Silver			%10
No color			%20

The number of colored bands can be from 3 to 6. To read colored bands on the 4-band resistor, we need to follow the color code chart shown in Figure 2. Before reading the resistor color codes, we need to first figure out from which band to start with. The tolerance band is situated with some gap from the other 2-3 bands, so we get the idea from where to start with.

Most of the common resistors are available in standard values. The two significant figures of standard resistor values are: 10, 12, 15, 18, 22, 27, 33, 39, 47, 56, 68, and 82. Hence, a 100 Ω resistor is marked as brown-black-brown and a 4.7 KΩ resistor is marked as yellow-violet-red.

Tolerance is the percentage of error in the value of resistance. If the tolerance is 5%, it means you can expect 5% change in the value of resistance. If the tolerance is less than 2% in a resistor, this type of resistors is known as precision resistors. (Note: If no tolerance band is given, then the tolerance should be taken as ±20%)

A. 2. Energy Sources

All circuits consume energy in order to work. Energy sources are either in form of voltage sources or current sources in electronic circuits. For example, batteries are voltage sources.

The voltage and current source symbols are shown in Figure 3. We need the concept of ideal source in order to model the real sources mathematically. An ideal voltage source is capable of providing the defined voltage across its terminals regardless of the amount of current drawn from it. This means that even if we short circuit the terminals of a voltage source and hence draw infinite amperes of current from it, the ideal voltage source, e.g. the one in Figure 3.1(a), will keep on supplying Vo to the short circuit. This inconsistent combination obviously means that the ideal supply is capable of providing infinite amount of energy. Similarly an ideal current source can provide the set current whatever the voltage across its terminals may be.

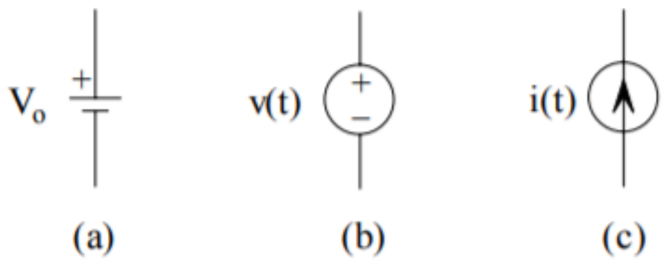


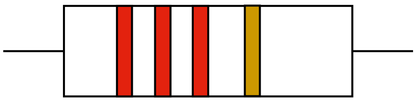
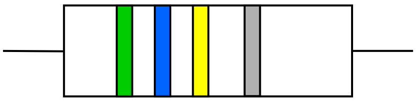
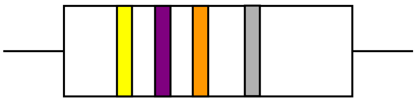
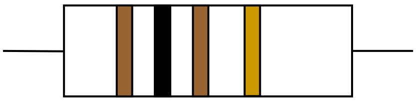
Figure 3. (a) d.c. voltage source, (b) general voltage source, (c) current source

Real sources deviate from ideal sources in only one aspect. The voltage supplied by a real source has a dependence on the amount of current drawn from it. For example, a battery has an internal resistance, and when connected to a circuit, its terminal voltage decreases by an amount proportional to the current drawn from it.

B. Experimental Work

B. 1. Resistor Color Code

Find the resistance values of the resistors given in the following table.

	Resistance Value	Tolerance
	22x100ohm=2200ohm	+-%5
	56x10000ohm=560000ohm	+-%10
	47x1000ohm=47000ohm	+-%10
	10x10ohm=100ohm	+-%5

B. 2. Wheatstone Bridge

Consider the Wheatstone bridge shown in Figure 4.

1) Simulate the circuit in OrCAD/PSpice to find Va, Vb and Vab.

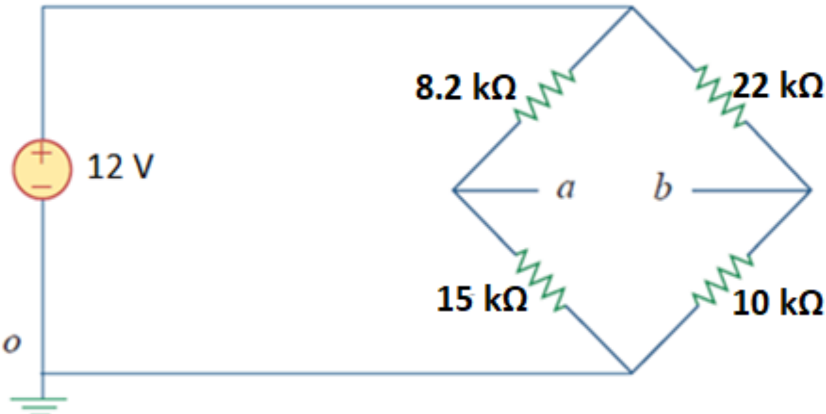
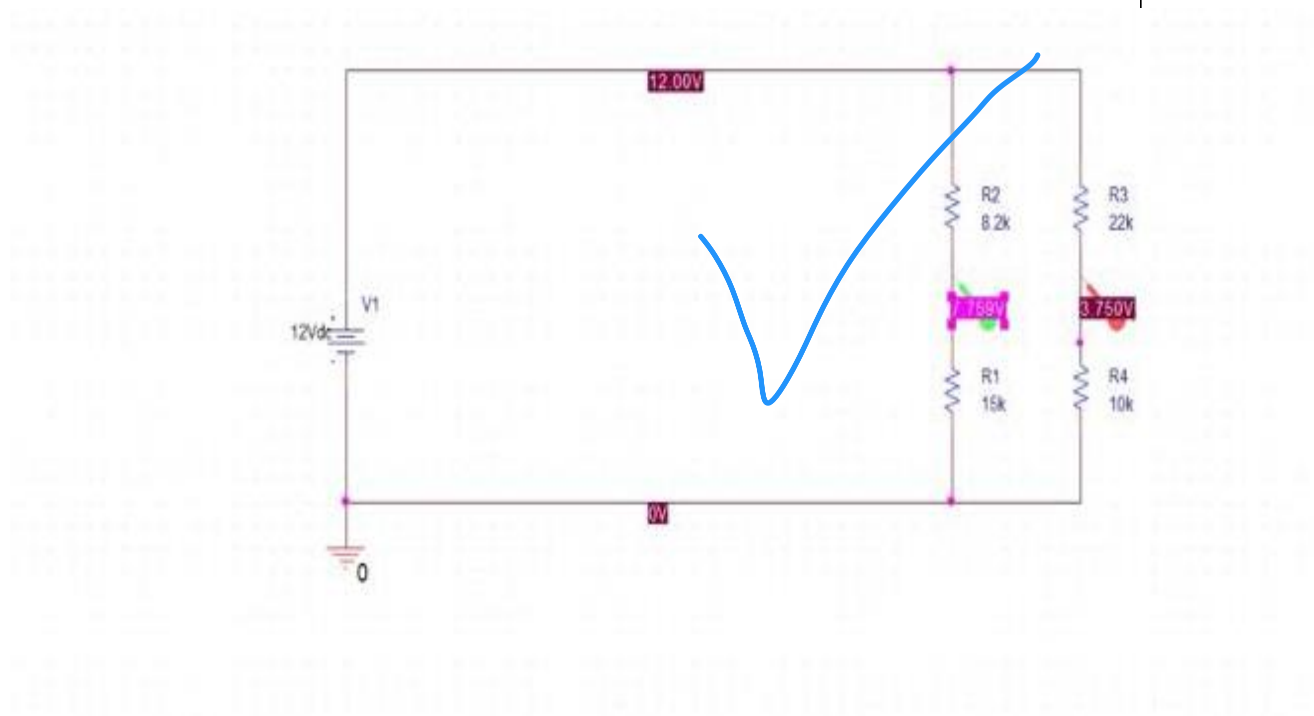
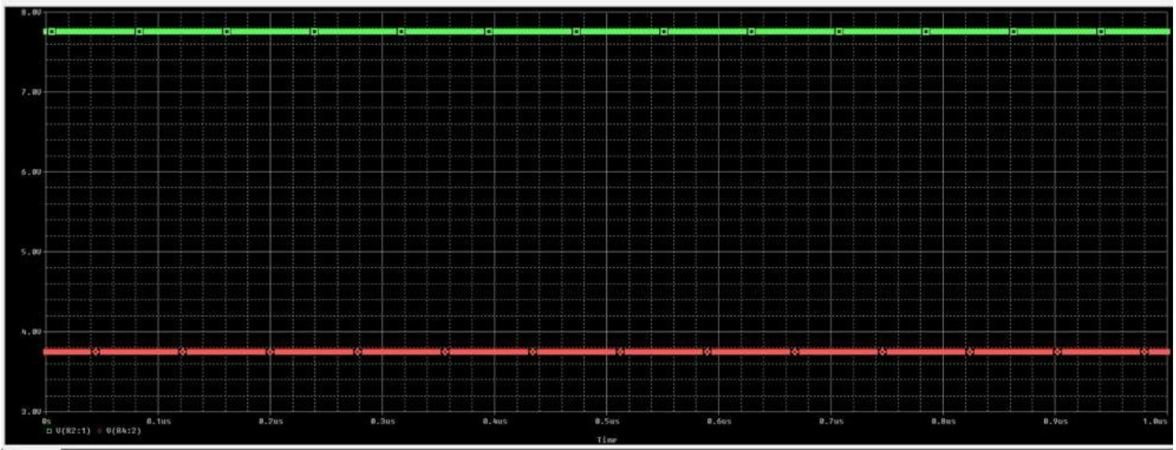


Figure 4. Wheatstone Bridge

i. Circuit Schematic



ii. Simulation Output (Va, Vb and Vab)



2) Set up the given circuit on a breadboard and measure the following DC values.

V_a = ..7.759.........V

V_b = ..3.750.........V

V_{ab} = ..4.009.......V

3) Calculate Va, Vb and Vab by hand, and compare your results.

iii. Hand Calculations (Calculate Va, Vb and Vab by hand)

$V = 12V \quad R_1 = 8{,}2k\Omega,\ R_2 = 15k\Omega,\ R_3 : 22k\Omega,\ R_4 = 10k\Omega$

$I = \frac{V}{R} \Rightarrow \frac{V}{R_1+R_2} = \frac{12V}{8{,}2k\Omega + 15k\Omega} = 0{,}517\ mA$

$I_2 = \frac{V}{R_3+R_4} = \frac{12V}{22k\Omega + 10k\Omega} = 0{,}375\ mA$

$V_1 =$

Va = V2 = 7.759V
Vb = V4 = 3.75V
Vab = Va - Vb = 4009V ✓

0,517mA · 8.2kΩ = 4.241V
V2 = 0,517mA · 15kΩ = 7,759V
V3 = 0,375mA · 22kΩ = 8.25V
V4 = 0,375mA · 10kΩ =

B. 3. Two-way power divider

Consider the two-way power divider circuit shown in

5,75V

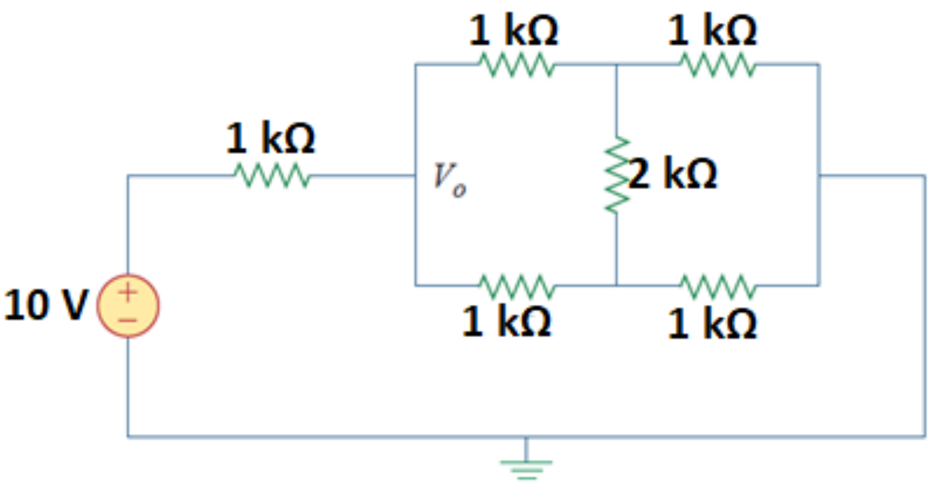
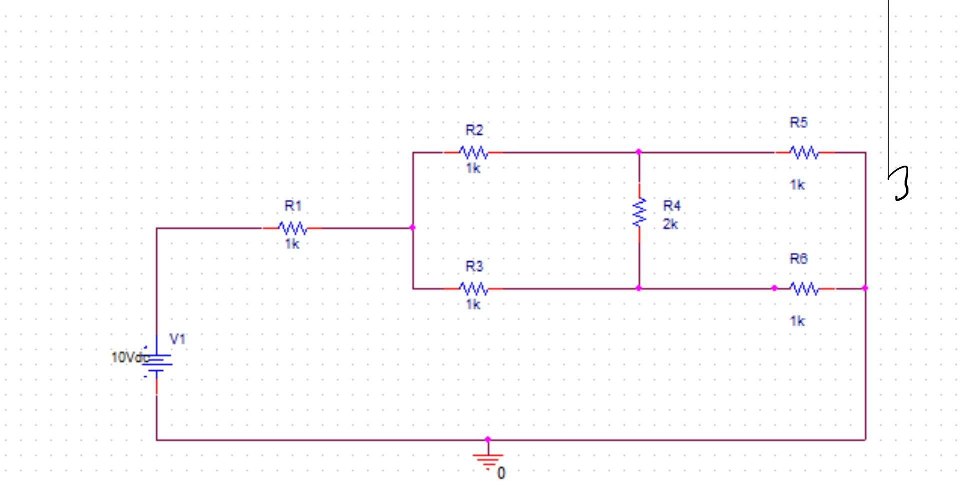


Figure 5.

1) Simulate the two-way power divider circuit in PSpice to find Vo.

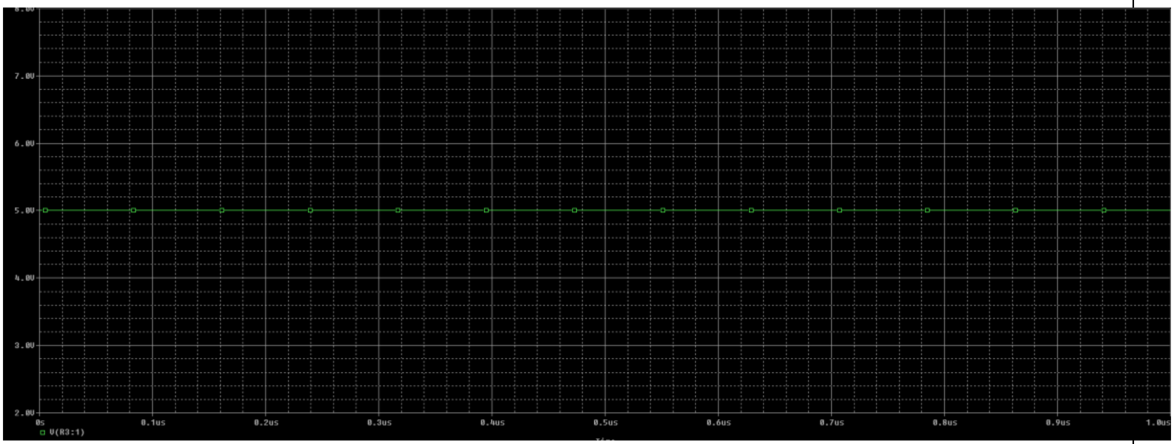
Figure 5. Two-way Power Divider

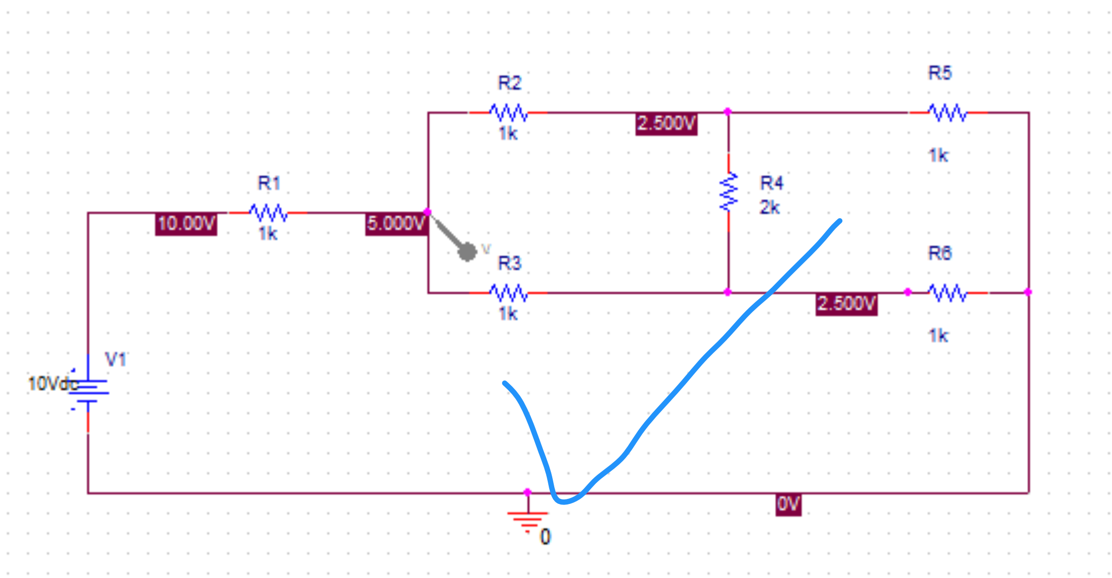
iv. Circuit Schematic



3

v. *Simulation Output (Vo)*



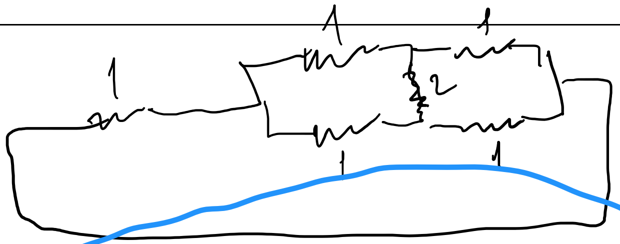


2) Set up the given circuit on a breadboard and measure Vo.

V_o = 4.8517..........V

3) Compare your results(Simulation and measurement).

vi. Hand Calculations (Comparison between Simulation&Measurement)



$$\frac{1}{2}+\frac{1}{2}=\frac{2}{2}=R=1$$

$$1+1=R_{es}=2k\Omega=2000\Omega$$

$$\frac{10}{2000\Omega}=\frac{1}{200}=i$$

$$V-V_{1k\Omega}=V_o$$

$$10-\frac{1}{200}\cdot 1000=5$$

$$\text{Error}(\%)=\frac{(\text{Simulation value})-(\text{measurement value})}{\text{m. value}}$$

$$100\times\frac{5-4{,}8517}{4.8517}=3{,}05\%\mp$$

B. 4. Using a Multimeter for Voltage Measurement

Consider the circuit shown in Figure 6.

1) Obtain Vo and Io in this circuit using PSpice

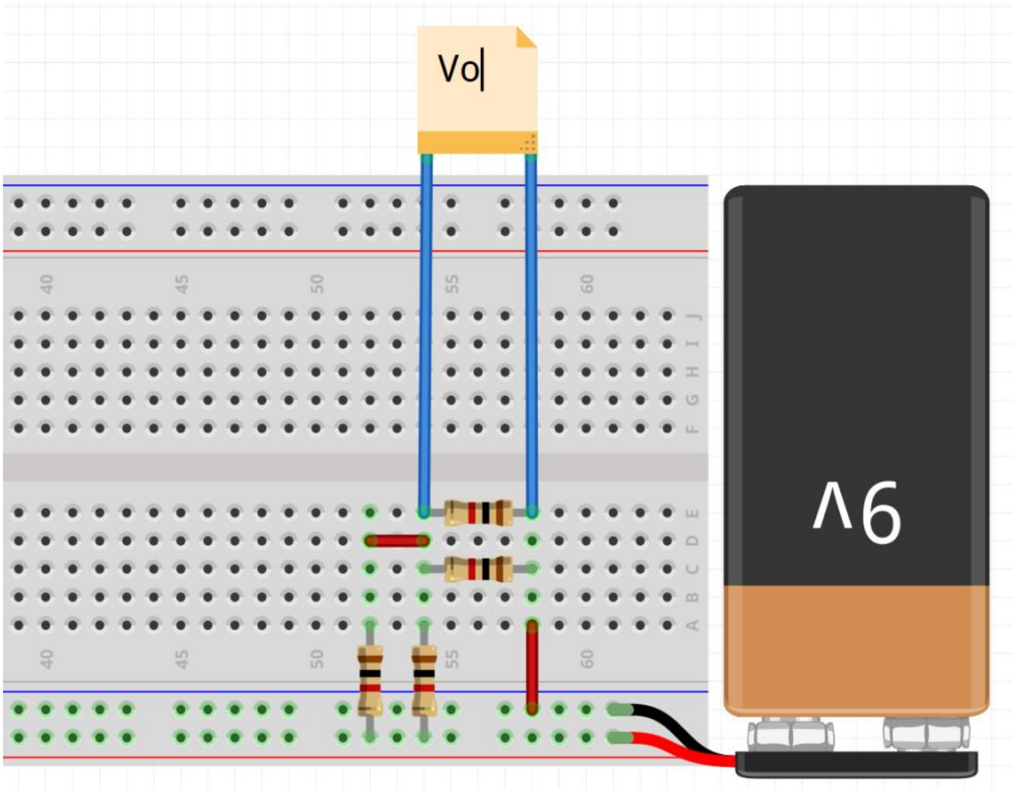
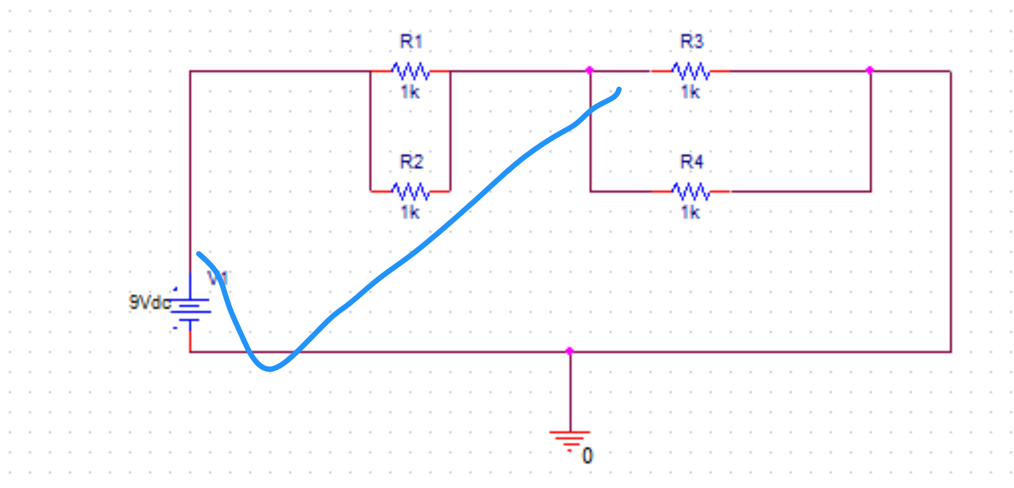
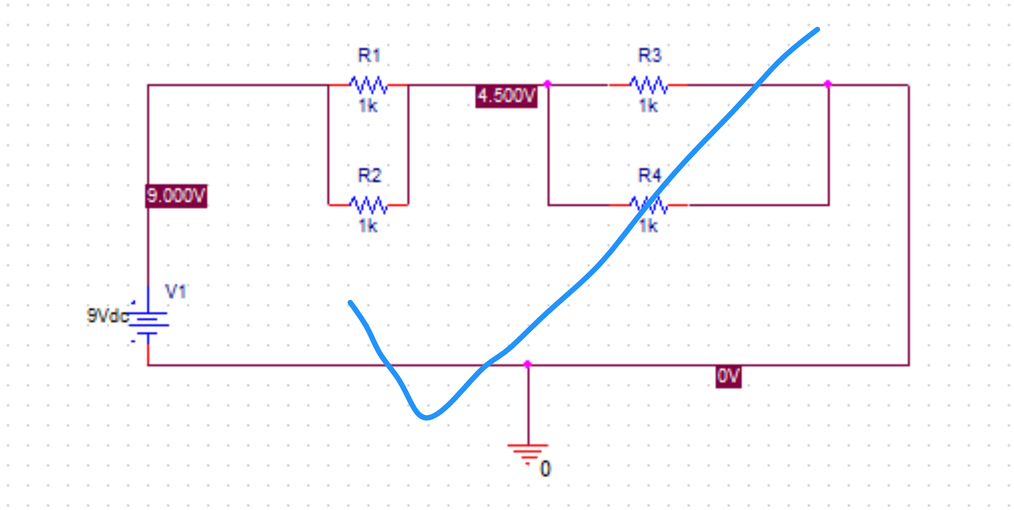


Figure 6. Voltage Measurement (Vo)

vii. *Circuit Schematic.*



viii. *Simulation Output* (Vo).



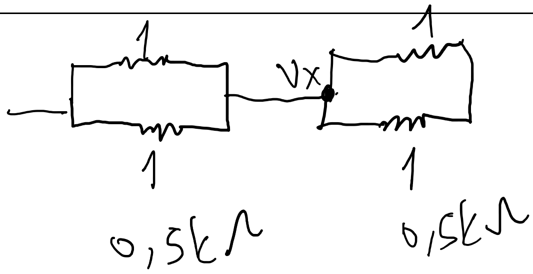
2) Set up the given circuit on a breadboard and measure Vo and Io.

   V_o = ...4.211........V

   Io=0.00452 A DC

3) Calculate Vo and Io by hand.
4) The finite resistance of the meter introduces an error into the measurement. Find the percent error for voltage Vo.

ix. Hand Calculations (Calculate Vo by hand, and find the percent error for Vo)



$\frac{1}{1}+\frac{1}{2}=\frac{2}{2}=1=R_{eq}$

$\frac{1}{1}+\frac{1}{1}=\frac{2}{1}=\frac{1}{2}R_{eq}$

$V=I\cdot R$

$9=i\cdot 1$

$\boxed{9=i}$

$V-V_{ab}=V_x$

$9-\frac{1}{2}\cdot 9=\frac{9}{2}=V_x$

$\boxed{4{,}5=V_x}$

8mA

Error:

$4{,}5-4{,}211=0{,}289\%$

Percent error $\frac{\Delta V_o}{V_o}\times 100$

$\boxed{\frac{0{,}289}{4{,}211}\times 100}$

Lab 2
Current Measurement & Voltage Divider & Delta-Wye Conversion

A. Background
A. 1. Current Measurement

Current measurements are made in different way to voltage and other measurements. Current consists of a flow of electrons around a circuit, when using a multimeter to measure current, the only way that can be used to detect the level of current flowing is to break in to the circuit so that the current passes through the meter which can be seen in Figure 1.

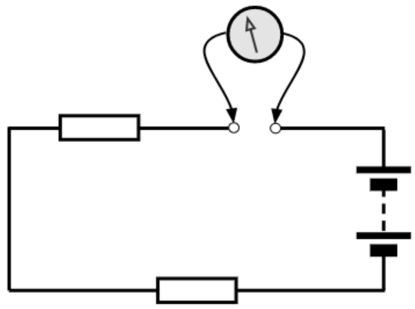


Figure 1. Simple circuit and measurement of the current

A. 2. Potentiometers

Potentiometers provide an adjustable resistance between two points as shown in Figure 2. The arrowhead represents a movable contact point. Thus, the resistance between the terminals a and b (or c and b) can be varied from 0 to 100 percent of the total resistance between a and c.

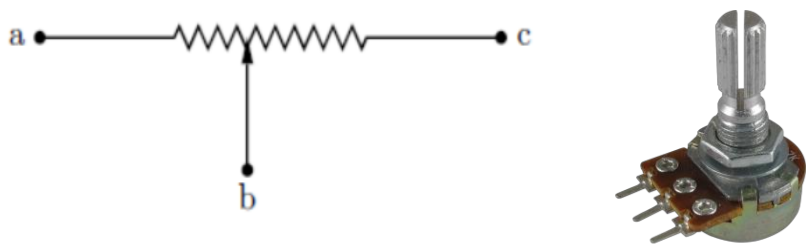


Figure 2. Potentiometer and its schematic diagram

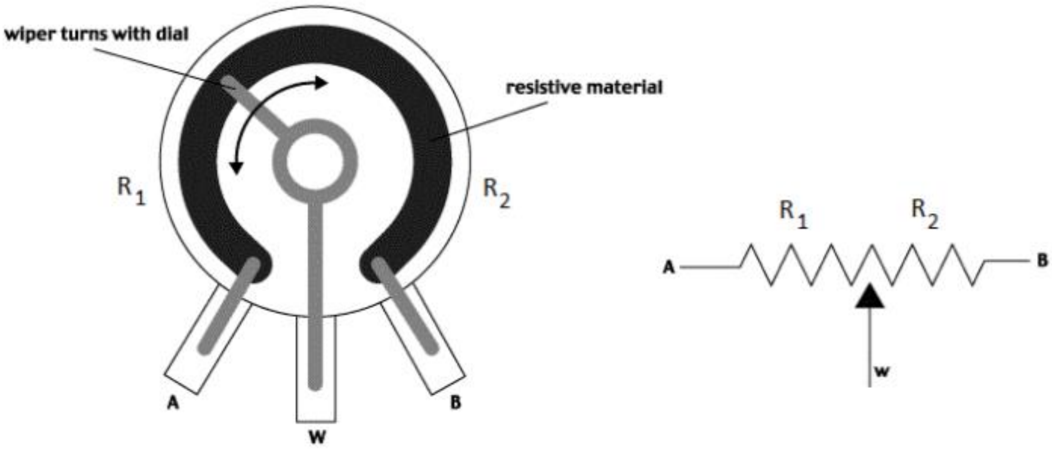


Figure 3. Circular and Straight line Potentiometers

A potentiometer can be used as a voltage control device to obtain a variable fraction of the potential between two points as shown in Figure 4. Here Vo can be varied between zero and Vs.

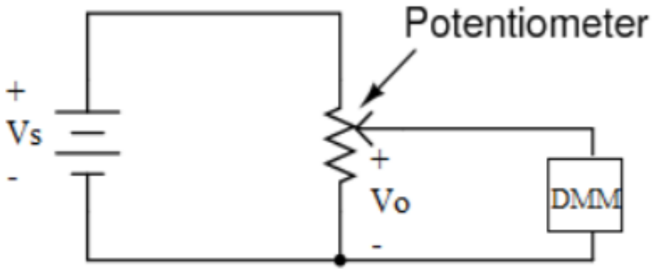


Figure 4. Potentiometer Voltage Control

A. 3. Delta-Wye Conversion

Suppose it is more convenient to work with a wye network in a place where the circuit contains a delta configuration. We superimpose a wye network on the existing delta network as shown in Figure 5 and find the equivalent resistances in the wye network. To obtain the equivalent resistances in the wye network, we compare the two networks and make sure that the resistance between each pair of nodes in the Δ network is the same as the resistance between the same pair of nodes in the Y network. The following equations can be used for the transformation:

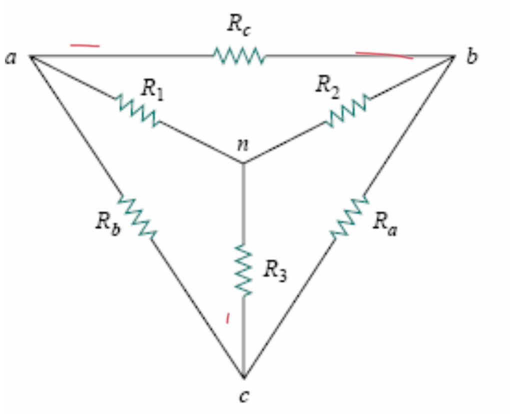


Figure 5. Superposition of Y and networks

Note that in making the transformation, we do not take anything out of the circuit or put in anything new. We are merely substituting different but mathematically equivalent three-terminal network patterns to create a circuit in which resistors are either in series or in parallel, allowing us to calculate Req if necessary.

$R_a = \dfrac{R_1R_2 + R_2R_3 + R_3R_1}{R_1}$	$R_b = \dfrac{R_1R_2 + R_2R_3 + R_3R_1}{R_2}$	$R_c = \dfrac{R_1R_2 + R_2R_3 + R_3R_1}{R_3}$
$R_1 = \dfrac{R_bR_c}{R_a + R_b + R_c}$	$R_2 = \dfrac{R_cR_a}{R_a + R_b + R_c}$	$R_3 = \dfrac{R_aR_b}{R_a + R_b + R_c}$

B. Experimental Work

B.1. Current Measurement

Consider the circuit shown in Figure 6.

1) Use OrCAD/PSpice to find all branch currents (I_1, I_2, I_3, and I_4).

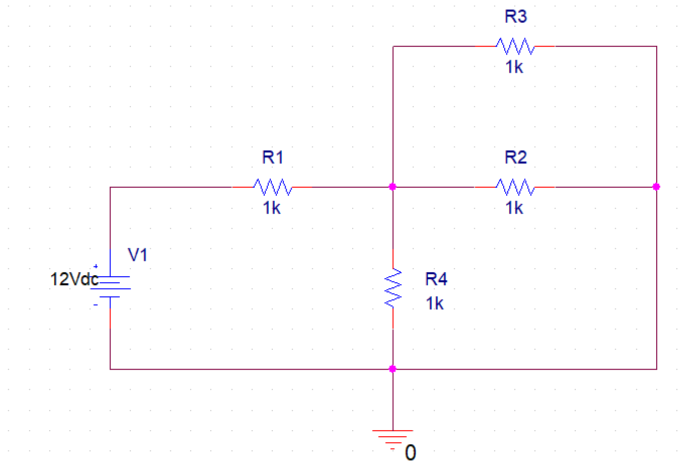
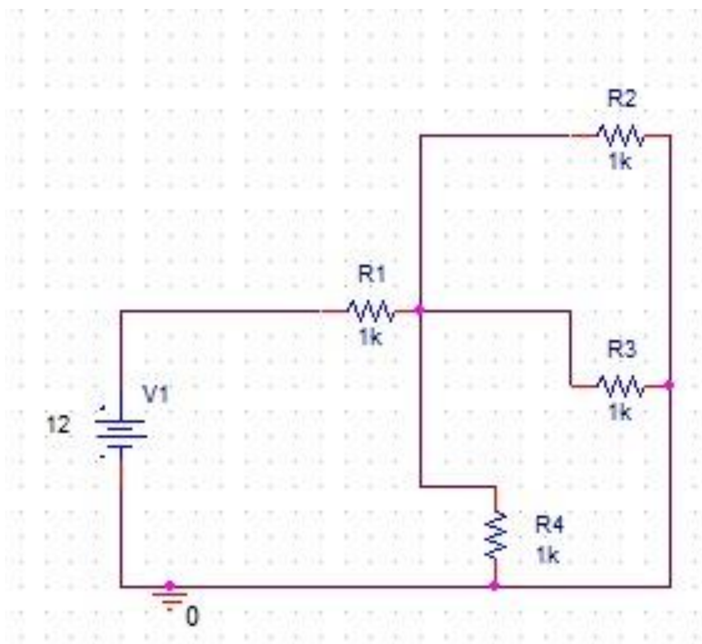
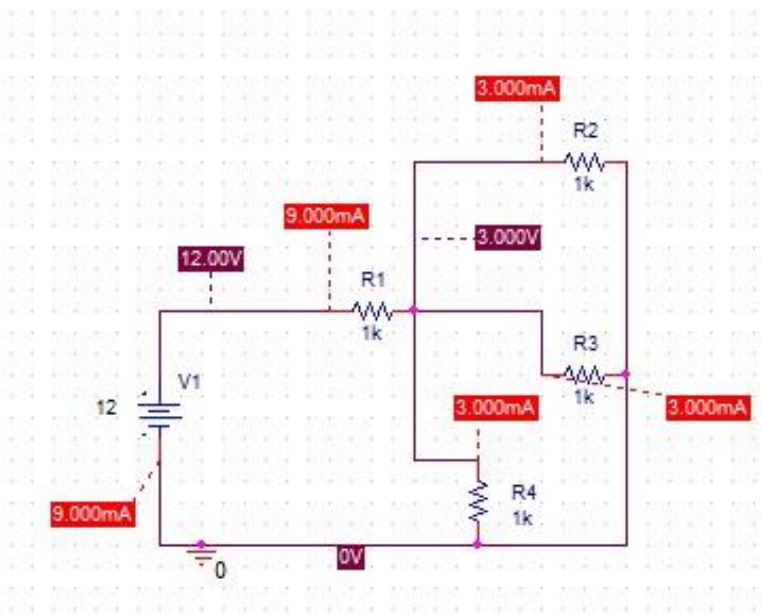


Figure 6. A simple resistive circuit

i. Circuit Schematic



ii. Simulation Output



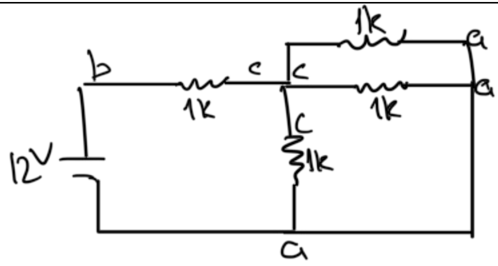
2) Set up the given circuit on a breadboard and measure all branch currents

   I1 = ...0.00906........mA **I3 = ...0.00302........mA**

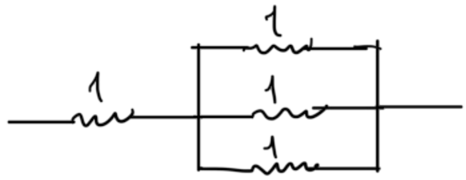
   I2 = ...0.00305........mA **I4 = ..0.00301.........mA**

3) Calculate all branch currents by hand and verify your calculations by applying KCL to the nodes.

iii. Hand Calculations (Calculate all branch currents by hand and verify by using KCL)



$\Rightarrow$



$$\frac{1}{1}+\frac{1}{1}+\frac{1}{1}=\frac{3}{1}\Rightarrow\frac{1}{3}$$

$$V = IR \Rightarrow I = \frac{V}{R} \qquad R_{eq} = \frac{1}{3}+1=\frac{4}{3}$$

$$i = \frac{12V}{\frac{4}{3}}$$

$$12\cdot\frac{3}{4} = 9A$$

B.2. Parametric Sweep for Resistance / Voltage Divider Circuit

Consider the circuit shown in Figure 7.

4) Use OrCAD/PSpice to find the voltage Vout for RL=100, 1k and 10k by performing parametric sweep analysis.

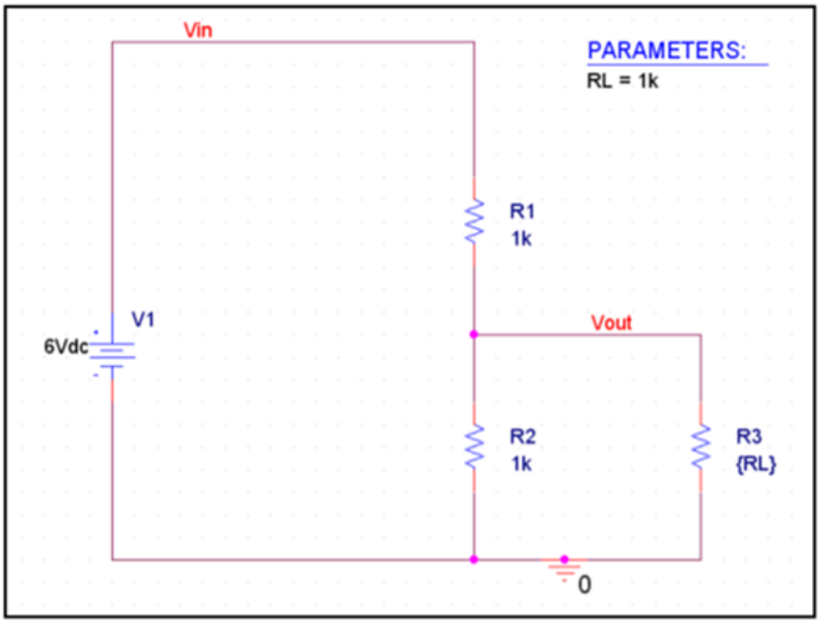
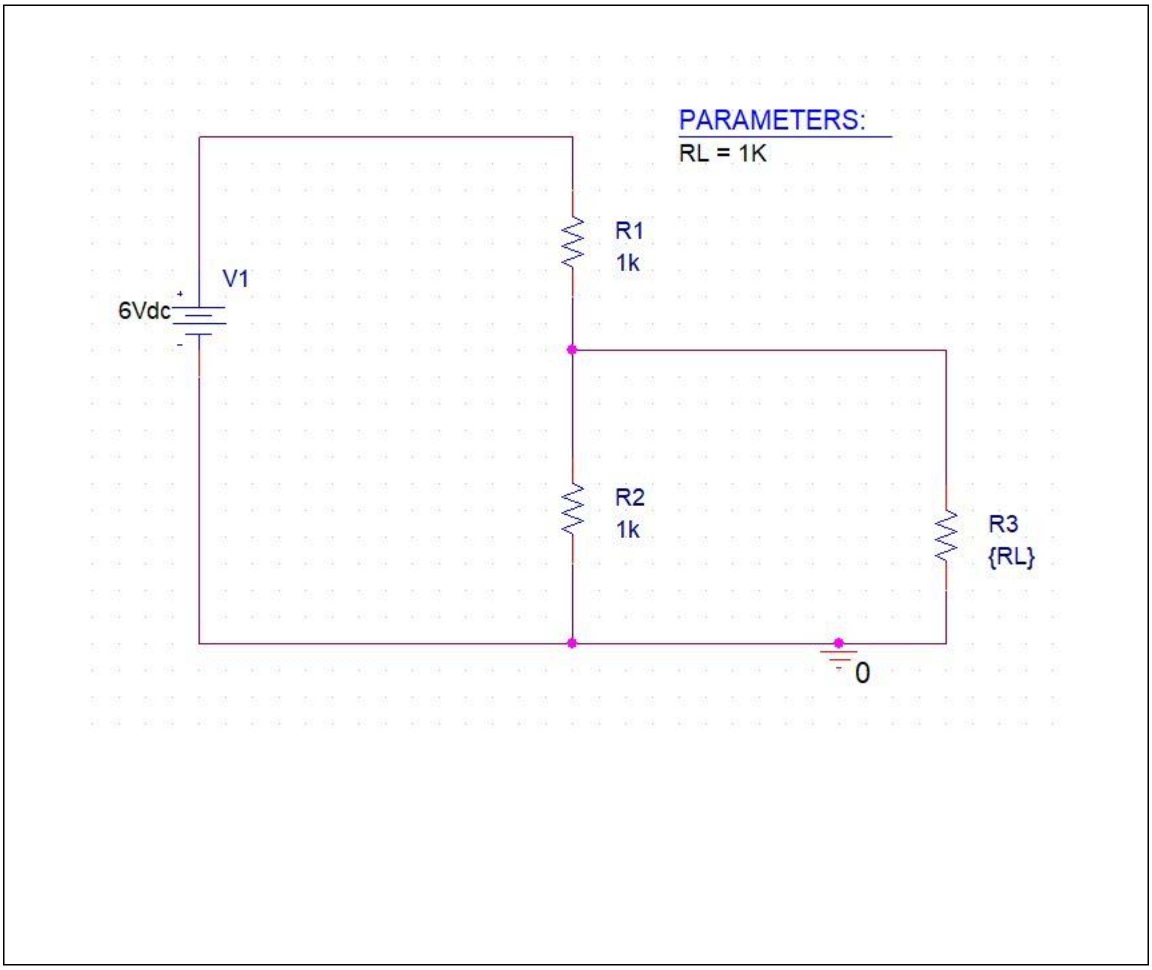


Figure 7. Voltage Divider Circuit

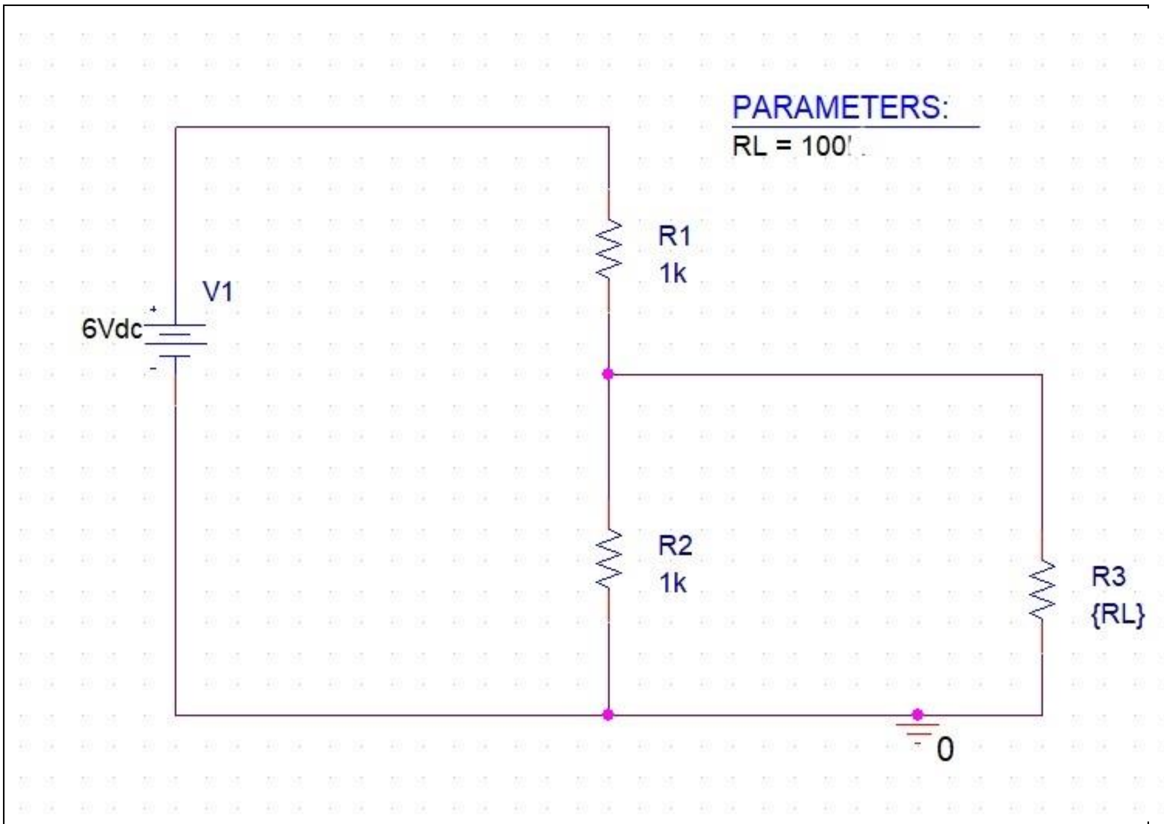
You need to make following changes to simulate your circuit in PSpice:

1) First, change the value of the part (not the name!) to {RL} (use curly braces, name is arbitrary)
2) Go to Place => Part
3) Add the part PARAM/SPECIAL to your schematic
4) Double click on the PARAM part
5) Click "New Property..."
6) Set the name to RL (same name as in “a” but with no curly braces)
7) Set the value to something, e.g., 1k (this is the value used to calculate DC bias values, choose somewhere in the range of your sweep) and then press OK.
8) Select the RL column (do not double click!) so that it is highlighted and then click Display...
9) Select "Name and Value" and press OK.
10) Go back to the schematic window.
11) An example schematic is shown above.

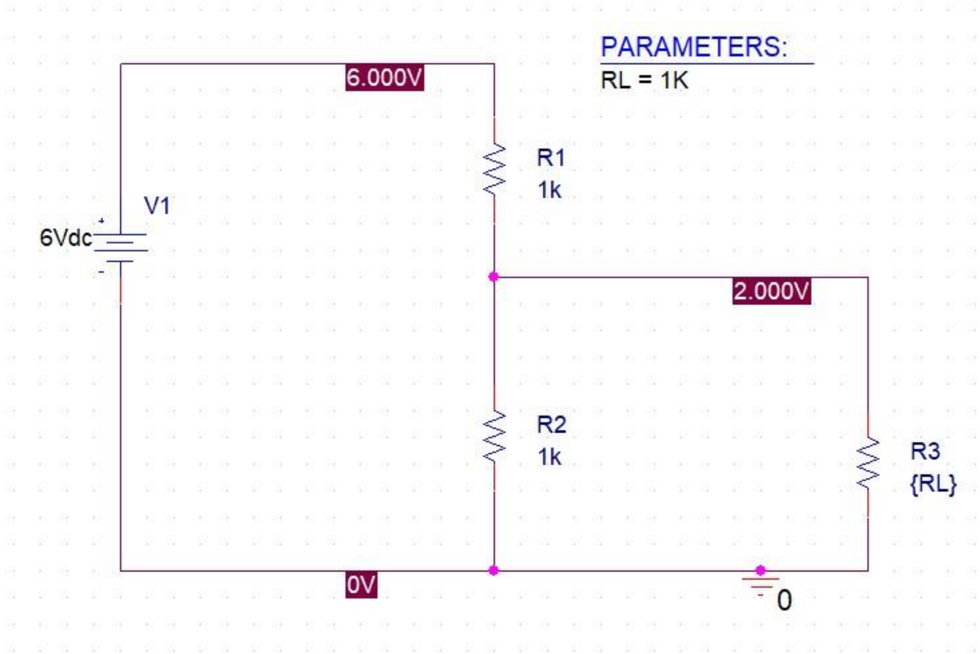
iv. Circuit Schematic



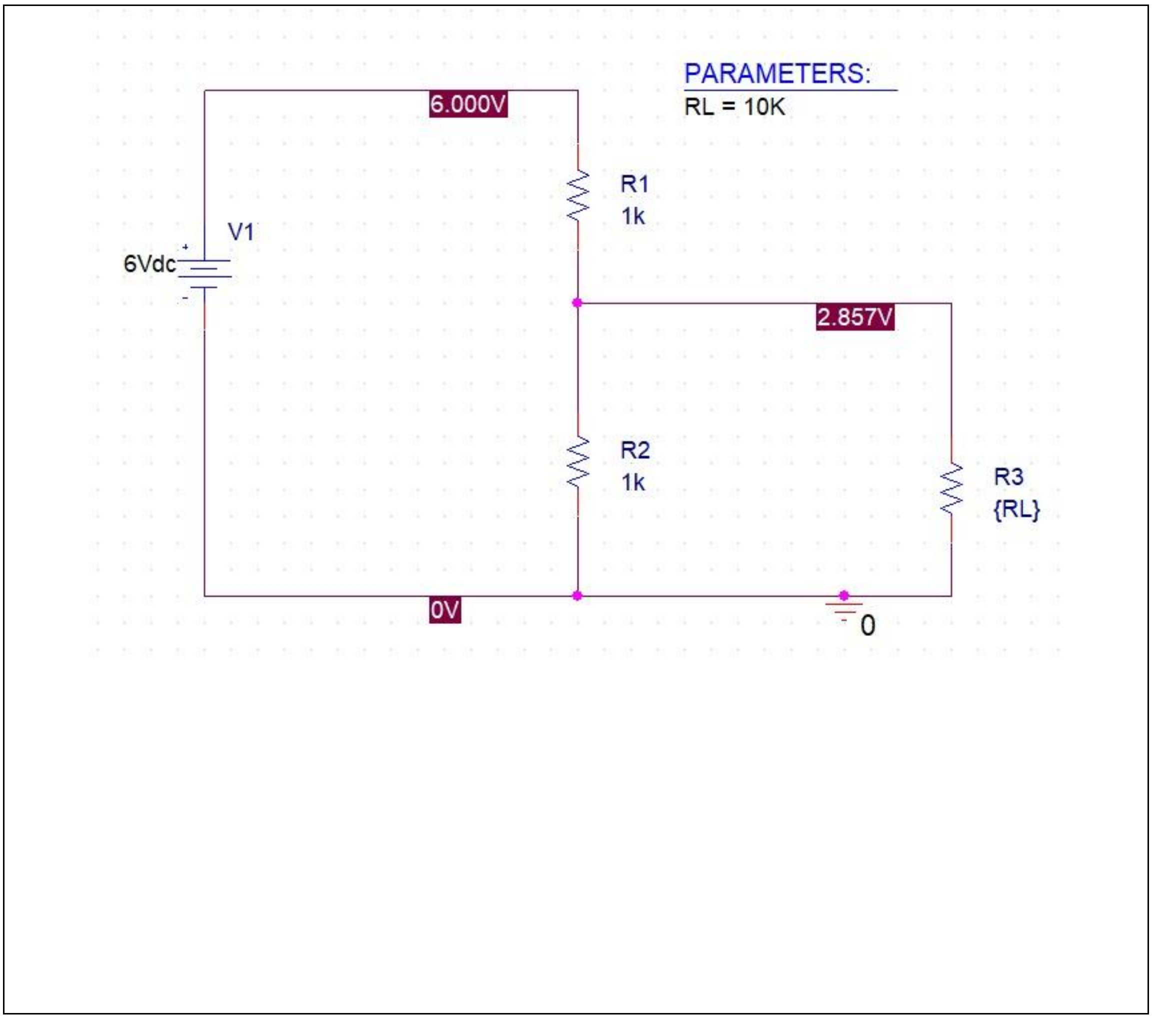
v. Simulation Output (V_{out} for RL=100)



vi. Simulation Output (V_{out} for RL=1k)



vii. *Simulation Output (V_{out} for RL= 10k)*



5) Set up the given circuit on a breadboard and measure Vout for RL=100, 1k and 10k. (Use a 10k potentiometer to adjust the resistance RL.)
Vout = 0.51210..........V for RL=100Ω
Vout = 1.9970.........V for RL=1kΩ
Vout = 2.8364..........V for RL=10kΩ
6) Calculate Vout for RL=100, 1k and 10k by hand and compare your results.

viii. Hand Calculations (Calculate V_{out} for RL=100, 1k and 10k by hand)

For 1k; $\frac{1}{1}+\frac{1}{1}=2 \Rightarrow \frac{1}{2}k$

$R_{es}=\frac{1}{2}+1=\frac{3}{2}$

$I=\frac{6V}{\frac{3}{2}}=4A$

$6V-4\cdot1=2V$

For 10k;

$\frac{1}{1}+\frac{1}{10}=\frac{11}{10}\Rightarrow\frac{10}{11}$

$\frac{10}{11}+1=\frac{21}{11}$

$I=\frac{6V}{\frac{21}{11}}\Rightarrow\frac{66}{21}=3,1A$

$6-3,1\cdot1=2,9V$

For 100Ω;

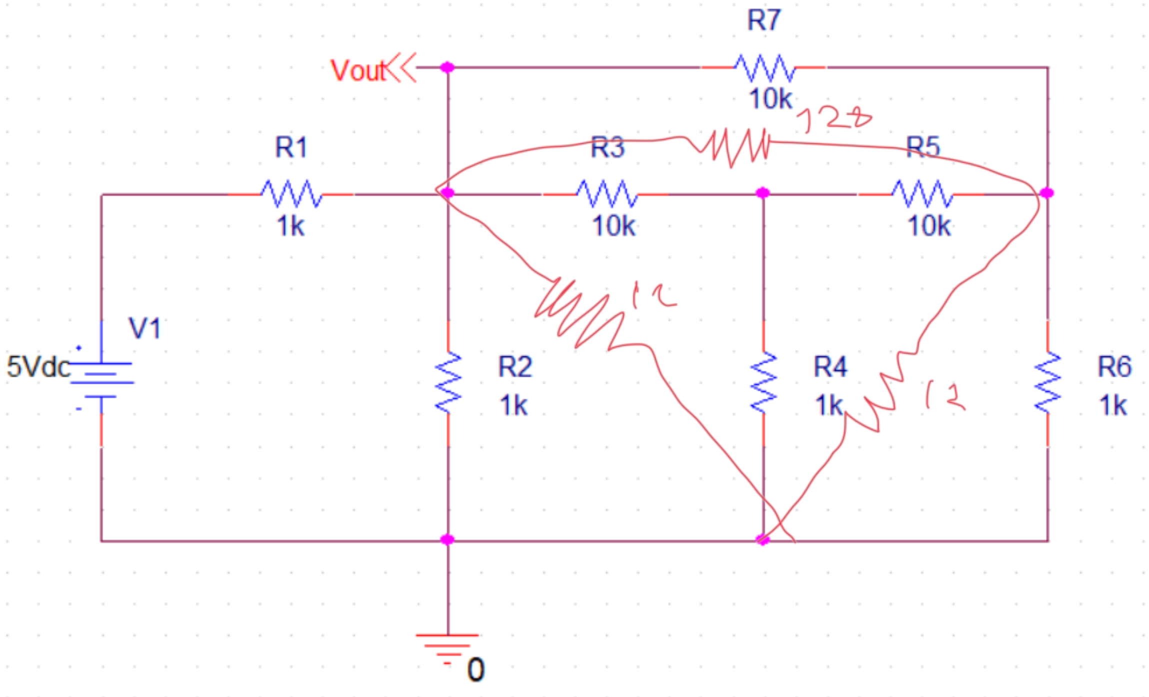
$R=\frac{1}{1000}+\frac{1}{100}=\frac{11}{100}$

$\frac{1000}{11}+1000$

$\frac{12000}{11}=R_{eq}$

$\frac{6V}{\frac{12000}{11}}=\frac{66}{12000}=I$

$V_{out}=6-\frac{66}{12000}\times1000=0,5V$



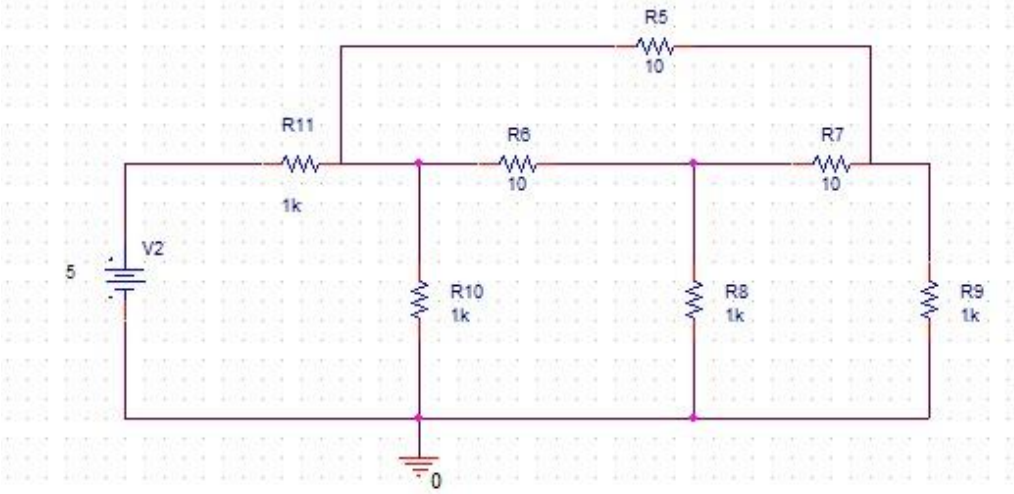
B.3. Delta-Wye Conversion

Consider the circuit shown in Figure 8.

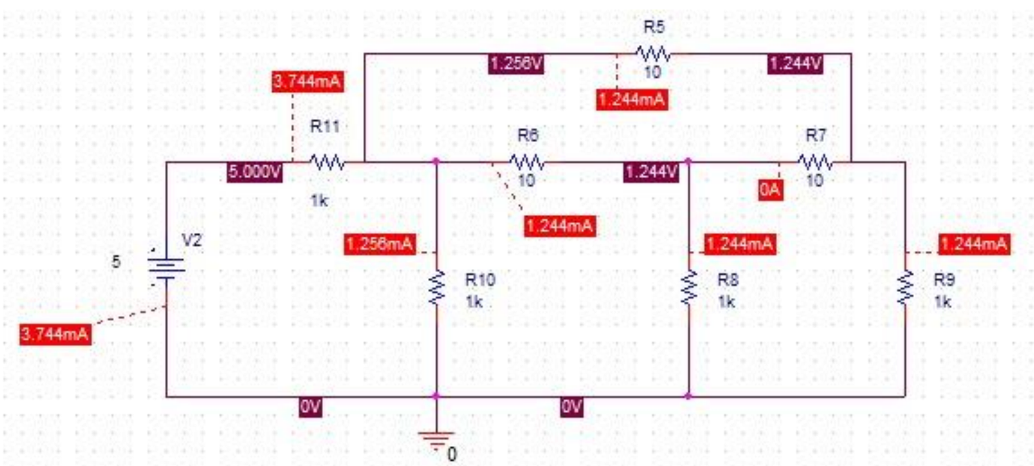
1) Simulate the circuit in OrCAD/PSpice and find the voltage Vout.

Figure 8. A simple resistive circuit

ix. Circuit Schematic & Simulation Output (Vout).



x. *Simulation Output (Vout).*



2) Set up the given circuit on a breadboard. Measure Vout, the current through the resistor R2, and the power dissipated across the resistor R2.

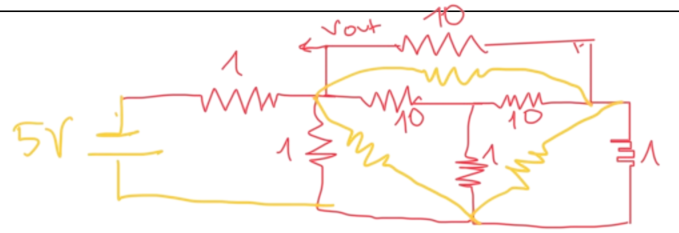
V_{OUT} = ..4.9206........V

I_{R2} = ..0.01651...A

P_{R2} = 6,125W

3) Calculate V_{OUT}, I_{R2} and P_{R2} by hand and compare your results.

xi. Hand Calculations (Calculate V_{OUT}, I_{R2} and P_{R2} *by hand)*

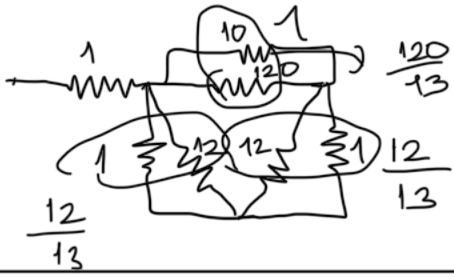


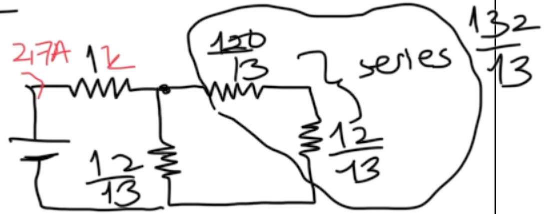
$P_{R2} = V \cdot i = i^2 \cdot R$

$(2,475)^2 \times 1 = 6,125...$

$V_{OUT} = 5V - 1 \cdot 2,7 = 2,3V$

$R_1 = 10.10 + 1.10 + 1.10 = 120$ $R_2 = 12$ $R_3 = 12$





Parallel $\Rightarrow \frac{13}{132} + \frac{13}{12} = \frac{156}{132} \Rightarrow \frac{132}{156}$

$\frac{132}{156} + 1 = \frac{288}{156} = R_{eq}$

$\frac{5V}{\frac{288}{156}} = \frac{5.156}{288} = 2,7\,mA = I$

$\boxed{I_{R2} = 2,475}$

$12k = 2,7$

$k = 0,225$

B.4. Annual energy cost of a system

Consider the three-wire system shown in Figure 9. In this system, load A consists of a motor drawing a current of 8 A, while load B is a PC drawing 2 A. Assuming 11 h/day of use for 365 days and 7 cents/kWh, calculate the annual energy cost of the system.

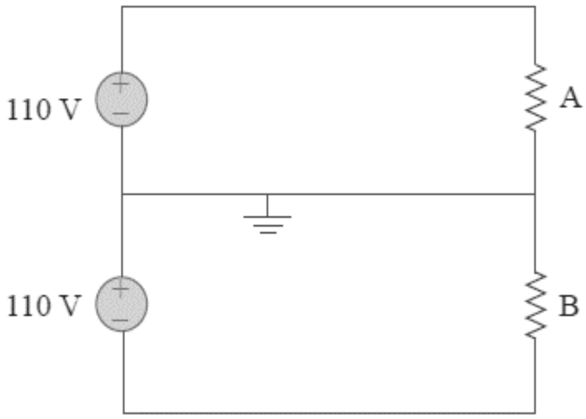


Figure 9. Schematic diagram of a three-wire system

xii. Hand Calculations *(Calculate the annual energy cost of the system.)*

$110 \times 8 = 880 W$

$110 \times 2 = 220 W$

$0,07 \times 365 \times 11 \times (880 + 220)/1000 = 309,15\$$
cost

Mehmet Arda Uçar /

Lab 3
Kirchhoff's Current Law

A. Background

Kirchhoff's Current Law (KCL), is about the currents entering and exiting a node (junction). KCL states that "sum of all currents entering and the sum of all currents leaving a node must be equal".

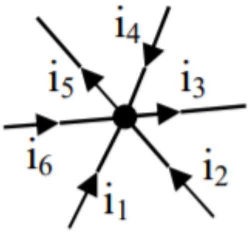


Fig. 3.1.

Consider the above node given in Fig. 3.1.

Currents entering the node = $\{i_1, i_2, i_4, i_6\}$
Currents leaving the node = $\{i_3, i_5\}$

Therefore, KCL states that:

$$i_1 + i_2 + i_4 + i_6 = i_3 + i_5$$

Current Divider:

Consider the circuitry given in Fig. 3.2.

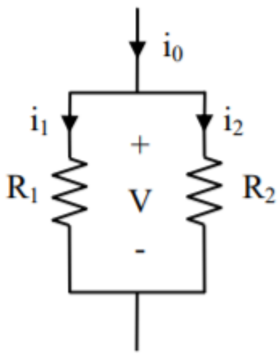


Fig. 3.2.

For the upper node, i_0 is the entering current and i_1 and i_2 are the leaving currents. Then,

$$i_0 = i_1 + i_2$$

The voltage drops over the resistors are equal, then,

$$R_1 \, i_1 = R_2 \, i_2$$

Solving the above two equations simultaneously, the branch currents i_1 and i_2 can be expressed in terms of the incoming total current i_0 as:

$$i_1 = \frac{R_2}{R_1+R_2} i_0 \quad \& \quad i_2 = \frac{R_1}{R_1+R_2} i_0$$

B. Experimental Work

B.1. Current Divider & KCL

Consider the circuit given in Fig. 3.3.

1) Use OrCAD/PSpice to find the currents i_0, i_1, i_2, i_3 and i_4 by performing bias point analysis.

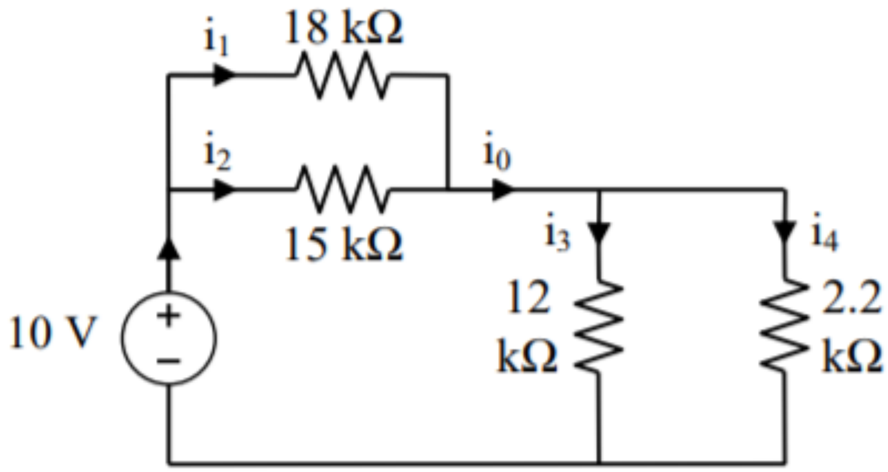
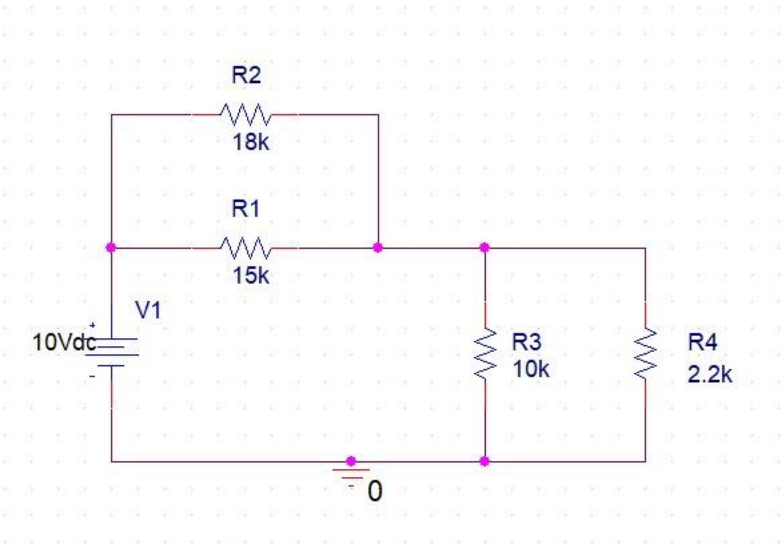
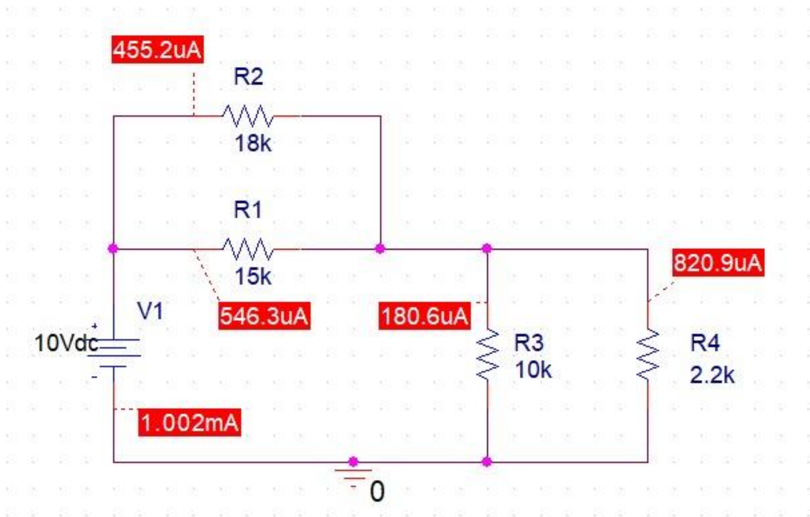


Fig. 3.3. Current Divider Circuit

i. Circuit Schematic & Simulation Output (i_0, i_1, i_2, i_3 and i_4*)*



ii. *Simulation Output (i_0, i_1, i_2, i_3 and i_4)*



2) Set up the given circuit on a breadboard and measure the currents i_0, i_1, i_2, i_3 and i_4.

i_0 = 0.00109 mA

i_1 = 0.00046 mA

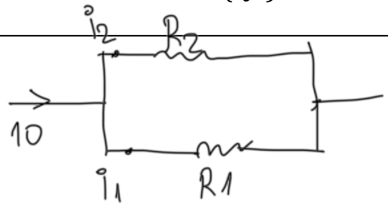
i_2 = 0.00056 mA

i_3 = 0.00018 mA

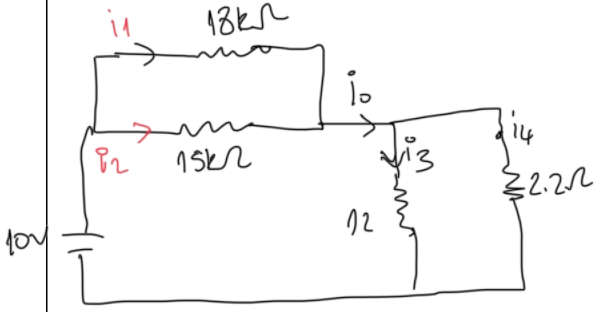
i_4 = 0.00082 mA

3) Express the currents i_1 and i_2 in terms of the current i_0, using current division rule. Similarly express the currents i_3 and i_4 in terms of the current i_0, using the current division rule. Calculate the currents i_0, i_1, i_2, i_3 and i_4, and verify KCL at all nodes.

iii. Hand Calculations (Q3)



$$\frac{R_1}{R_1+R_2} \cdot 10$$



$R_{eq}(18k // 15k) + (12k\ 2.2k)$

$\frac{1}{18} + \frac{1}{15} = \frac{11}{90} \rightarrow \frac{90}{11}$

$\frac{1}{12} + \frac{10}{22} = \frac{71}{132} \rightarrow \frac{132}{71}$

$$R_{eq} = \frac{90}{11} + \frac{132}{71} = 10{,}04\,k\Omega$$

$$i_1 = \frac{15\,k\Omega}{\underbrace{(15+18)}_{33}\,k\Omega} \cdot i_0 \qquad i_2 = \frac{18}{33} \cdot i_0$$

$$i_3 = \frac{2.2}{14.2} \cdot i_0 = \frac{11}{71} \qquad i_4 = \frac{12}{14.2} \cdot i_0 = \frac{60}{71} \cdot i_0$$

$$\frac{V}{R_{eq}} : i_0 = \frac{10V}{10{,}04} = 0{,}99 = i_0 \cong 1$$

$\boxed{\text{KCL Rule}}$

$$\boxed{\begin{aligned} i_1 &= 0{,}45\,mA \\ i_2 &= 0{,}54\,mA \\ i_3 &= 0{,}15\,mA \\ i_4 &= 0{,}85\,mA \end{aligned}}$$

$$i_1 + i_2 = i_0$$

$$0{,}45mA + 0{,}54mA = 0{,}99A$$

$$i_3 + i_4 = i_0$$

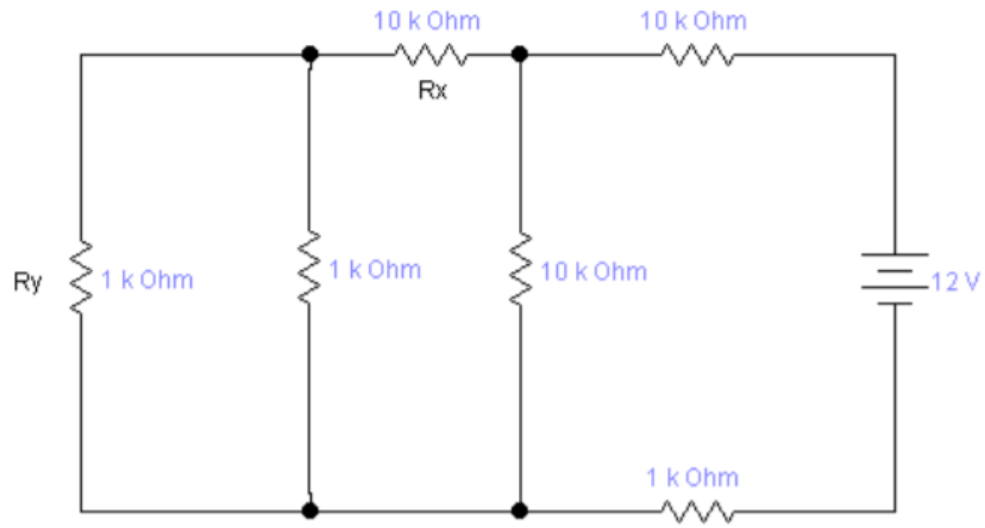
$$0{,}15mA + 0{,}85mA = 1A$$

similar

B.2. Equivalent Resistance

Consider the circuit given in Fig. 3.4.

1) Simulate the circuit in OrCAD/PSpice and find the current flowing through the 12V voltage source.
2) Calculate the equivalent resistance seen from the terminals of voltage source.
3) How can you measure the equivalent resistor experimentally? What is the



measured equivalent resistor?

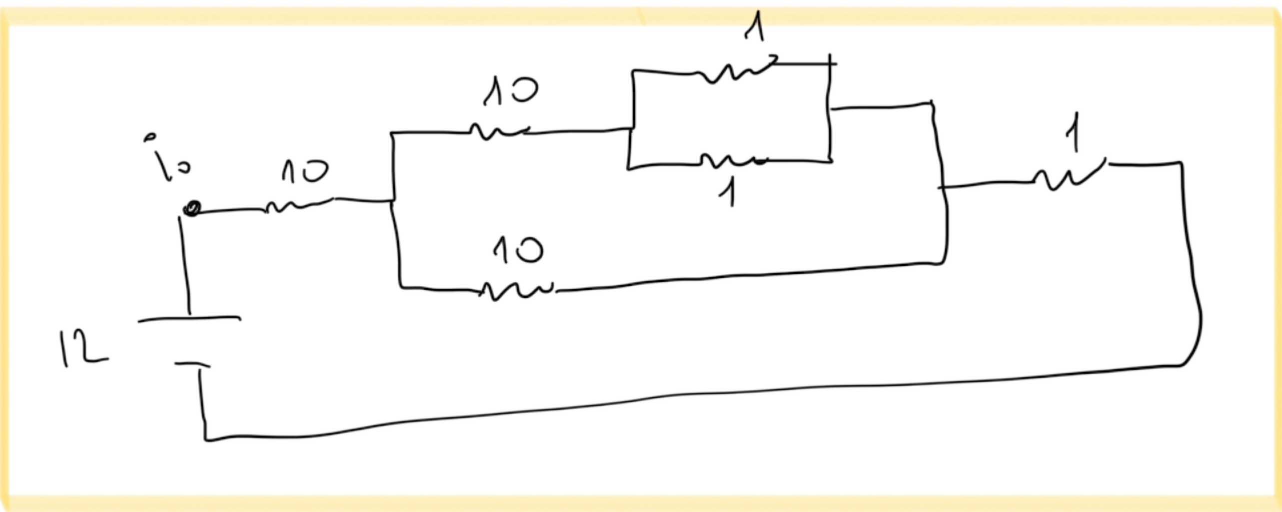


Fig. 3.4. A simple resistive circuit

R_equivalent=

$V = i \cdot R$

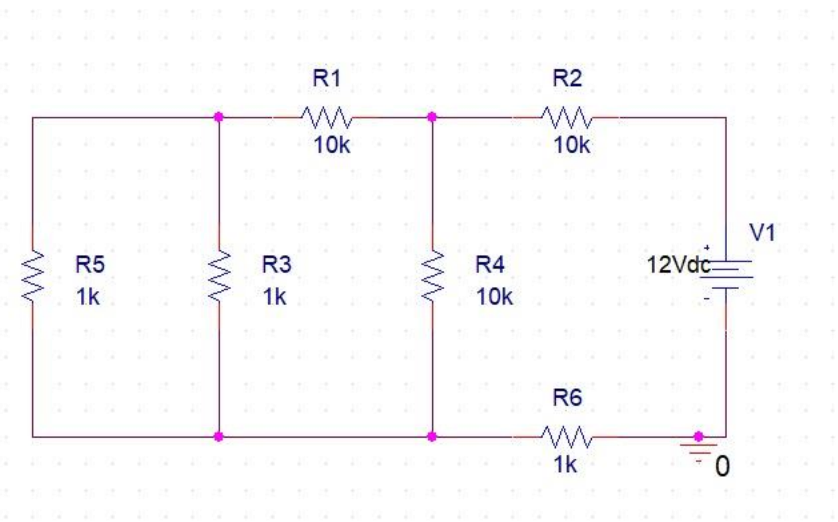
$\frac{V}{i} = R_{eq}$

$i_0 = 0.00075$

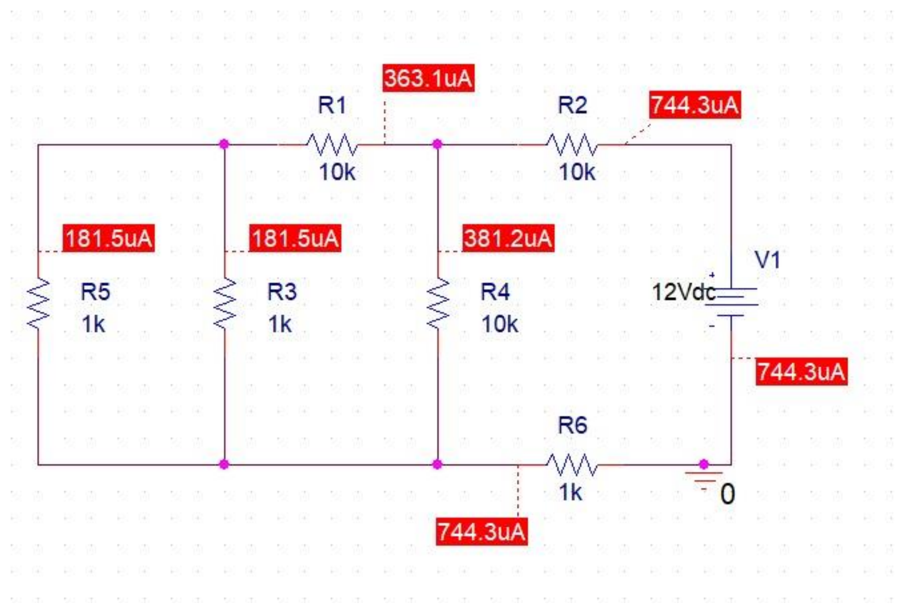
$V = 12$

$\frac{12}{0.00075} = 16.000$

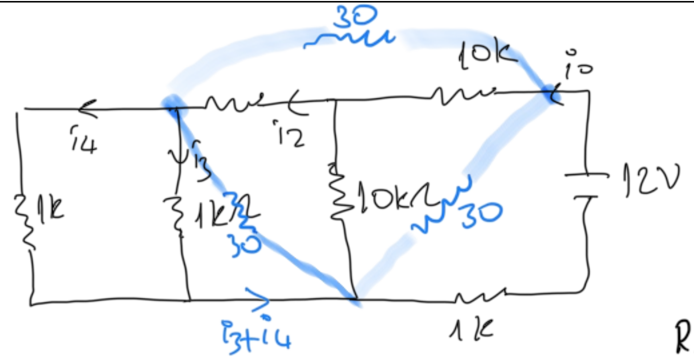
iv. *Circuit Schematic*

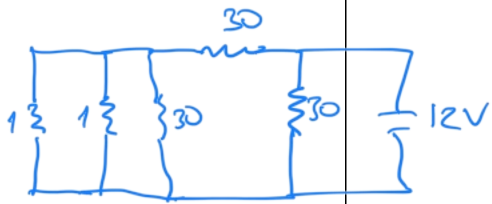


v. *Simulation Output (The current flowing through 12V voltage source)*



vi. *Hand Calculations (Calculate the equivalent resistance seen from the terminals of voltage source)*





$$R_1 = \frac{1}{1}+\frac{1}{1}+\frac{1}{30} = \frac{61}{30} ; \boxed{\frac{30}{61}}$$

$= 0,491$

$0,491 + 30 = 30,491$

$$R_{eq} = \frac{1}{30} + \frac{1}{30,491} = \frac{1}{0,067}$$

$= 15,12$

$$i = \frac{12}{15,12k} = 0,794\,mA$$

Lab 4
Circuit Analysis Techniques

A. Background

Several solution techniques may be used to solve an electrical circuit. The widely used techniques are:

i. Exhaustive Method
ii. The Node Voltage Method
iii. The Mesh Current Method
iv. Circuit Substitution using Thevenin's and/or Norton's Equivalent Circuits
v. Superposition

In this experiment, four of these techniques will be studied; namely (i) The Node Voltage Method, (ii) The Mesh Current Method, (iii) Thevenin and Norton Circuits, and (iv) Superposition. The first two methods will be demonstrated on the following circuit (Fig. 4.1).

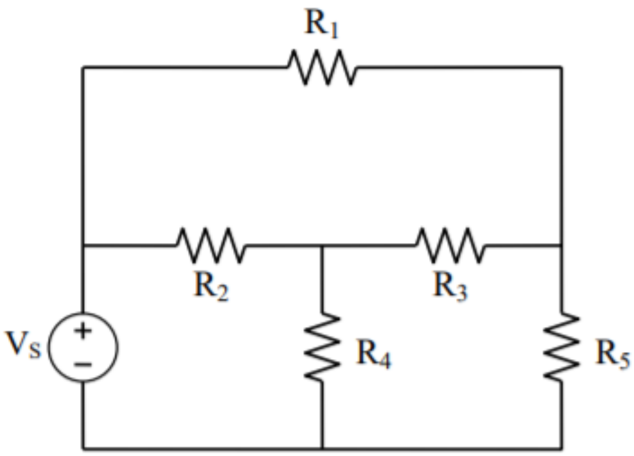


Fig. 4.1.

A.1. The Node Voltage Method

The node voltages Va and Vb at the nodes a and b (Fig. 4.2) are assumed parameters to be solved. All other circuit parameters can easily be obtained from Va and Vb.

To solve the two unknown values Va and Vb, two equations are needed. The KCL equations written for nodes a and b provide the required equations.

$$I_2 = I_3 + I_4 \tag{4.1}$$

and

$$I_5 = I_1 + I_3 \tag{4.2}$$

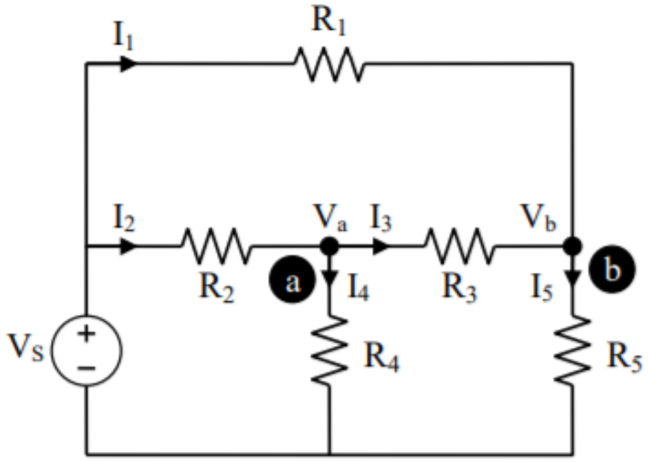


Fig. 4.2.

All these current values are required to be expressed in terms of the unknown values Va, Vb and the known source voltage Vs. Substituting

$$I_1 = \frac{V_S - V_b}{R_1}$$

$$I_2 = \frac{V_S - V_a}{R_2}$$

$$I_3 = \frac{V_a - V_b}{R_3}$$

$$I_4 = \frac{V_a}{R_4}$$

and

$$I_5 = \frac{V_b}{R_5}$$

into (4.1) and (4.2), two equations in terms of Va, Vb are obtained. The simultaneous solution of these two equations yield Va, Vb. The other circuit parameters are then obtained.

A.2. The Mesh Current Method

The meshes in a circuit are identified and mesh currents are assumed to flow through each mesh as shown in Fig. 4.3. As the circuit equations, mesh equations are written as

$R_1 I_1 - R_3 I_3 - R_2 I_2 = 0$
$R_2 I_2 + R_4 I_4 - V_s = 0$
$R_3 I_3 + R_5 I_5 - R_4 I_4 = 0$

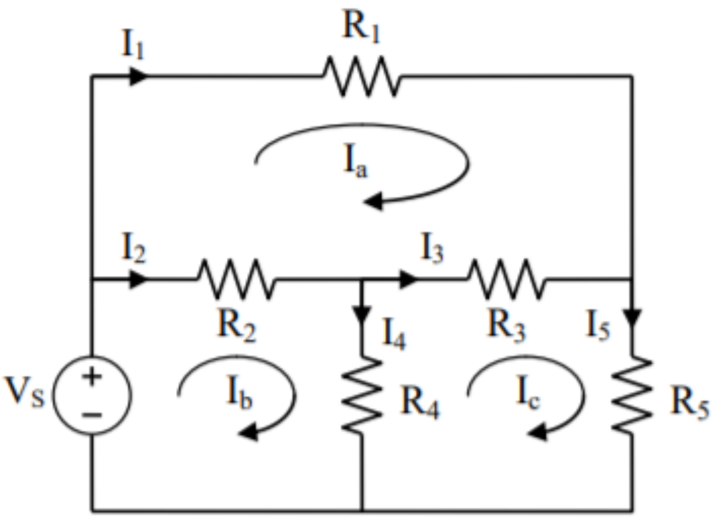


Fig. 4.3.

The branch currents I_1, I_2, I_3, I_4 and I_5 are easily expressed in terms of the unknown mesh currents Ia, Ib, and Ic as:

$$I_1 = I_a$$

$$I_2 = I_b - I_a$$

$$I_3 = I_c - I_a$$

$$I_4 = I_b - I_c$$

$$I_5 = I_c$$

Substituting these into the mesh equations written, three equations for Ia, Ib, and Ic are obtained. Solving these equations simultaneously, the unknown currents Ia, Ib, and Ic are determined. Then using these values, the other circuit parameters can be calculated.

B.3. Superposition Principle

Consider the circuit given in Fig. 4.6.

1) Use PSpice to find the Vout using Superposition Principle.

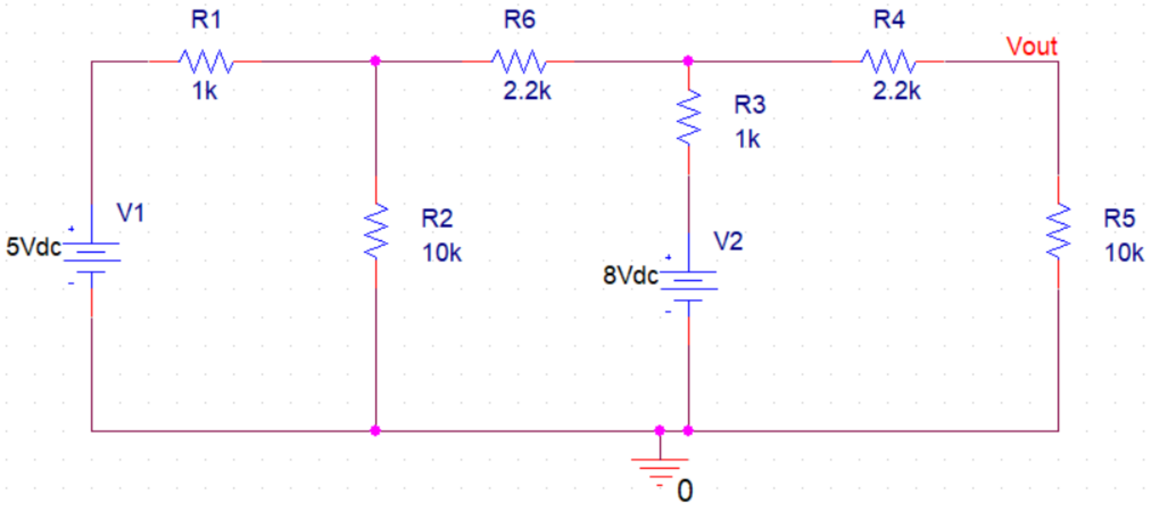
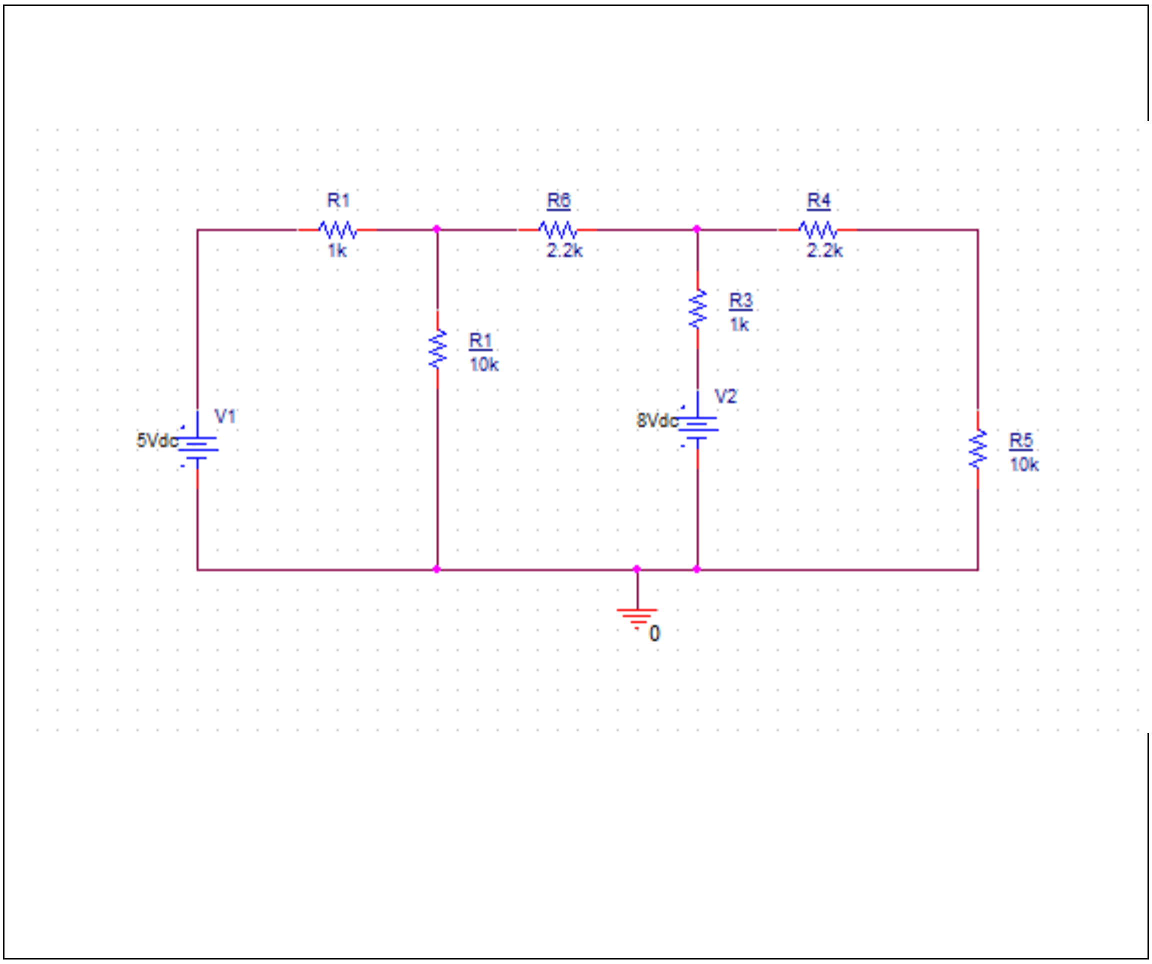
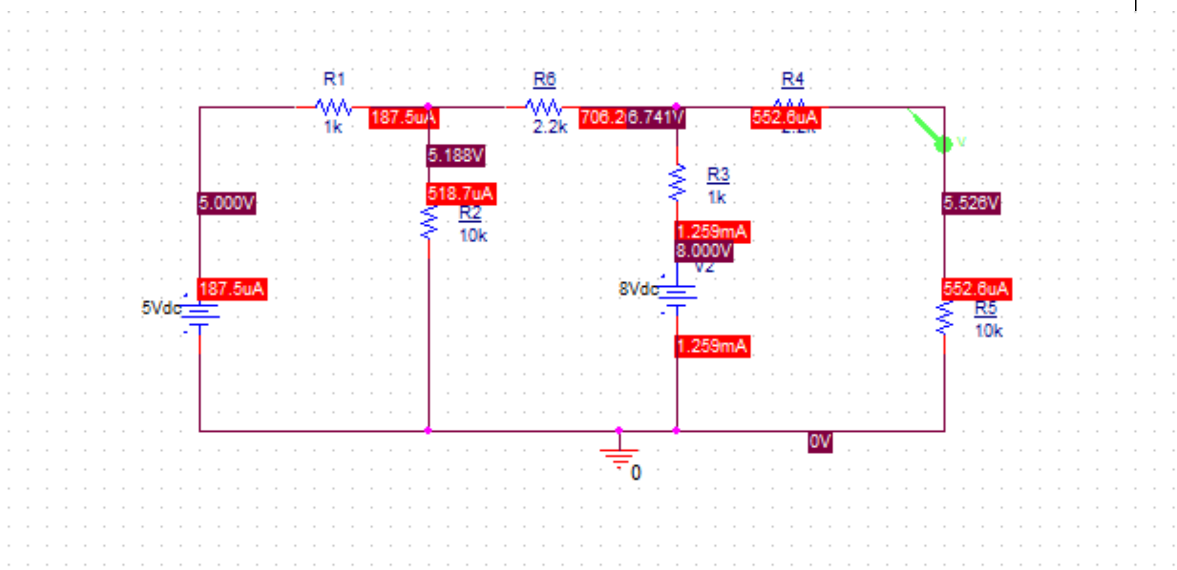


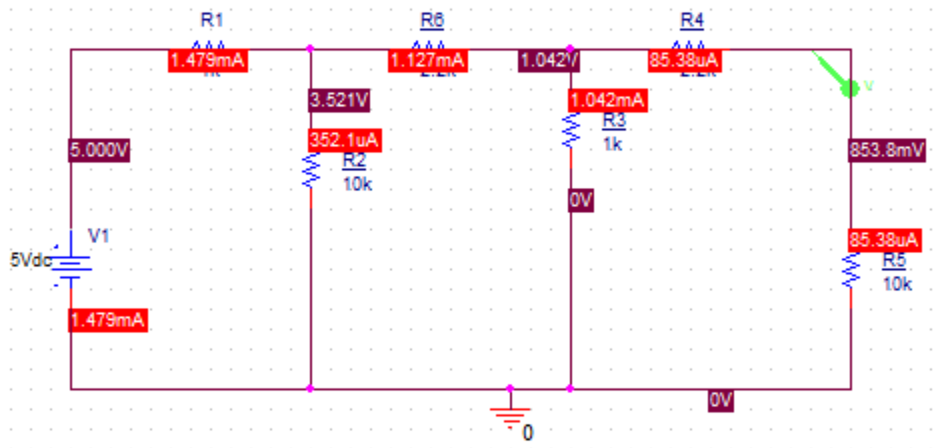
Fig. 4.6. Circuit with two independent sources

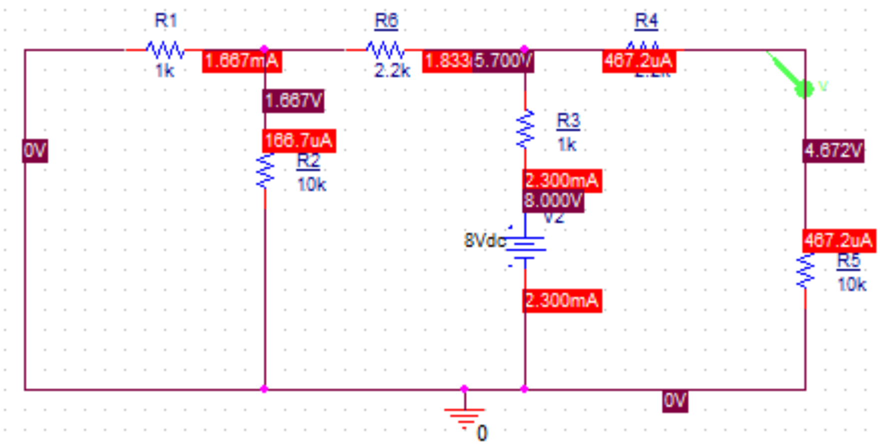
viii. Circuit Schematic.





ix. Simulation Output (V_{out} using superposition principle).

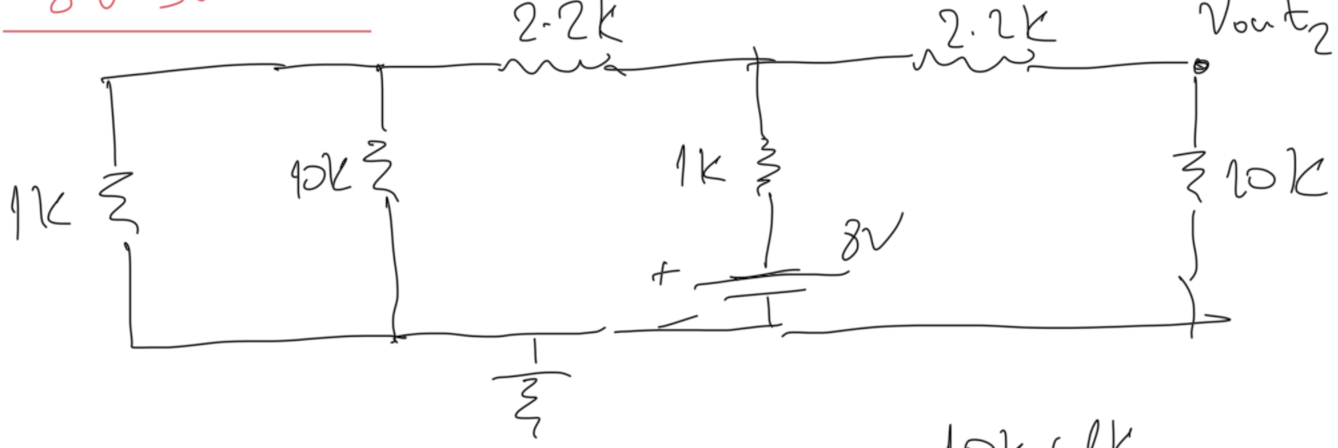




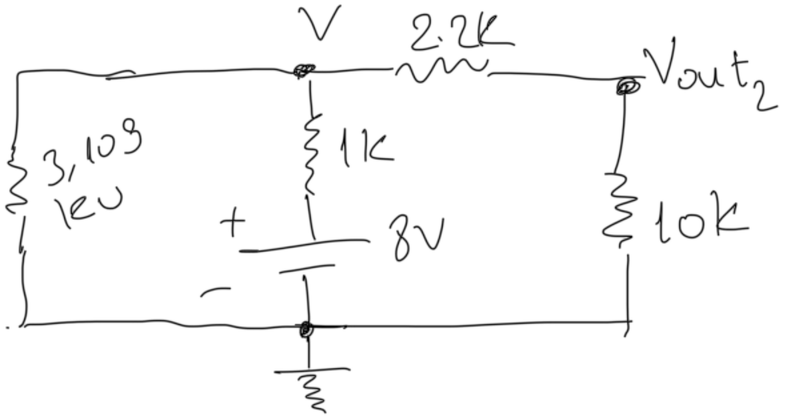
2) Set up the given circuit on a breadboard.
   a. Vout1 = …0.833………. V (5v varken)
   b. Vout2 = …4.657………. V
   c. Vout = Vout1 + Vout2 = 5.489.. V

3) Calculate Vout by hand using Superposition Principle.

8V Source



$$2.2K + \frac{10K \cdot 1K}{10K + 1K} = 3.109k\Omega$$



Apply nodal Analysis at node V

$$\frac{V}{3.109k} + \frac{V-8}{1K} + \frac{V}{12.2k} = 0$$

$$V \cdot \left[\frac{1}{3.109} + 1 + \frac{1}{12.2}\right] = 8$$

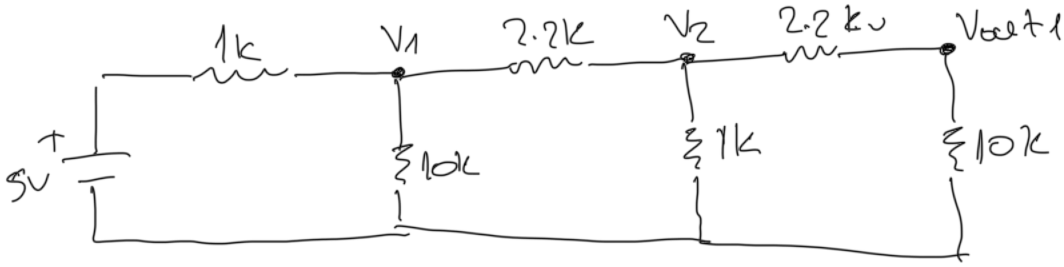
$$\boxed{V = 5.7}$$

by using voltage division rule,

$$Vout_2 = \frac{V \cdot 10K}{10K + 2.2K} = 4,67$$

$$Vout = Vout_1 + Vout_2 = 5.525$$

5V Source



Apply nodal Analysis at node V_1

$$\frac{V_1}{10K} + \frac{V_1 - 5}{1K} + \frac{V_1 - V_2}{2.2K} = 0$$

$$V_1 \cdot \left[\frac{1}{10} + 1 + \frac{1}{2.2}\right] + V_2 \left[-\frac{1}{2.2}\right] = 5 \quad (1)$$

x. *Hand Calculations (Calculate V_{out} using superposition principle by hand)*

Apply nodal Analysis at V_2

$$\frac{V_2}{1k} + \frac{V_2}{12.2k} + \frac{V_2 - V_1}{2.2k} = 0$$

$$V_1 \cdot \left[-\frac{1}{2.2}\right] + V_2\left[1 + \frac{1}{12.2} + \frac{1}{2.2}\right] = 0 \qquad (2)$$

Equation (1) & (2)

$$V_{out1} = \frac{V_2 \cdot 10k}{10k + 2.2k} = 0,852$$

B.4. Thevenin and Norton Equivalent Circuit

Consider the circuit given in Fig. 4.7.

1. Use PSpice to find the Thevenin Voltage and Norton Current.
2. Also, please calculate Thevenin Voltage and Norton Current by hand and draw the Thevenin and Norton equivalent circuits.

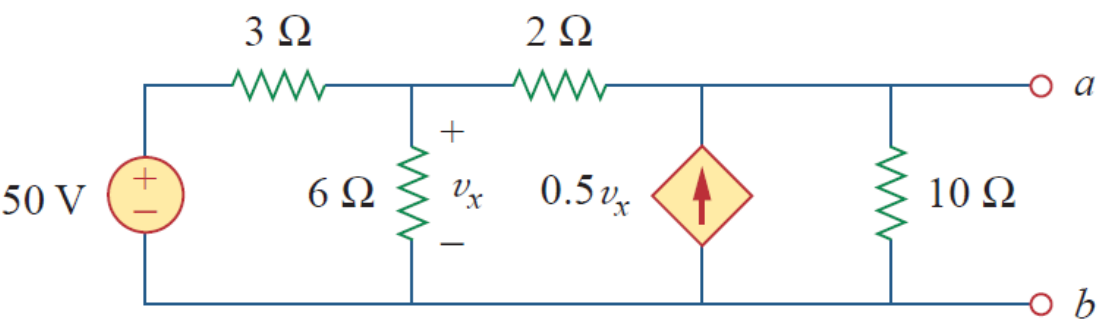
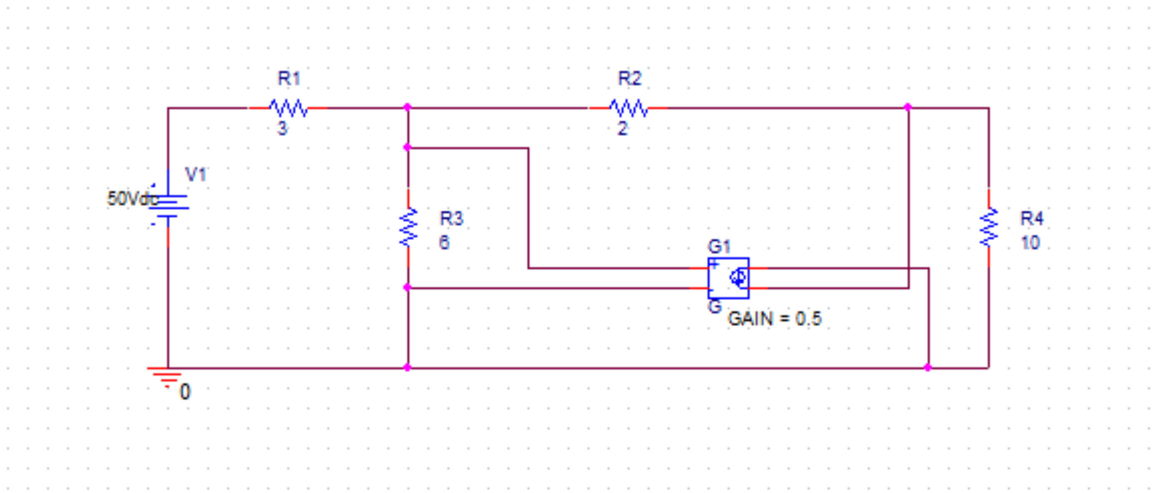
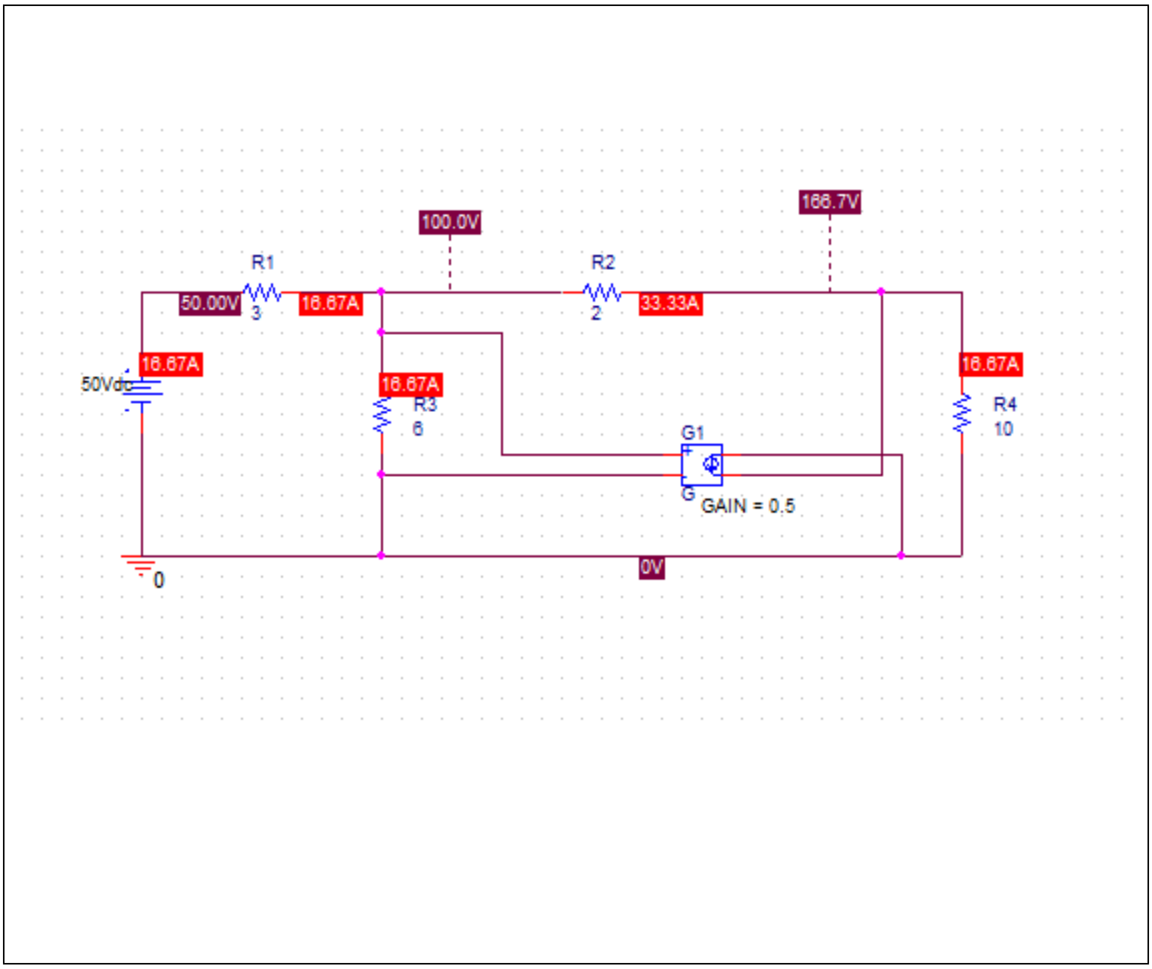


Fig. 4.7. Circuit with Dependent Source

xi. *Circuit Schematic.*

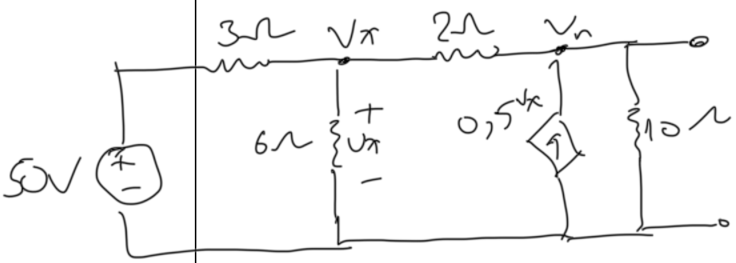


xii. *Simulation Output (Thevenin Voltage/Norton Current).*



xiii. *Hand Calculations (Calculate Thevenin Voltage/Norton Current by hand and draw the circuits)*

Thevenin Voltase



Aplying KCL at Vn

$$\frac{V_n - V_x}{2} - 0,5V_x + \frac{V_n}{10} = 0$$

$$5V_n - 5V_x - 5V_x + V_n = 0$$

$$6V_n = 10V_x \qquad V_x = 0,6V_n$$

Applying KCL at V_x

$$\frac{V_x - 50}{3} + \frac{V_x}{6} + \frac{V_x - V_n}{2} = 0$$

$$2V_x - 100 + V_x + 3V_x - 3V_n = 0$$

$$6V_x - 3V_n = 100$$

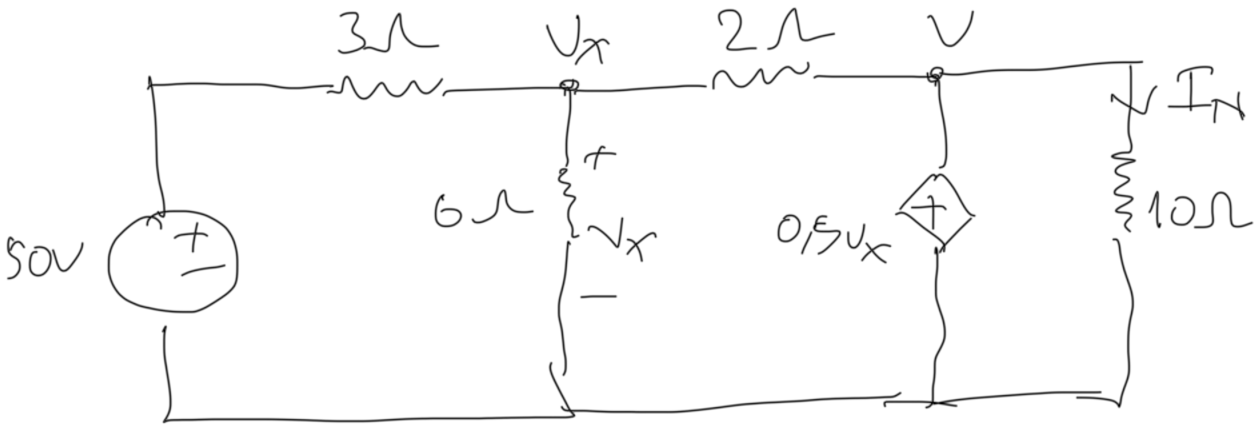
Put Vx in equation

$$6(0,6V_n) - 3V_n = 100$$

$$3,6V_n - 3V_n = 100$$

$$\boxed{V_n = 166,67V}$$

Norton Current



Applying KCL at node Vx;

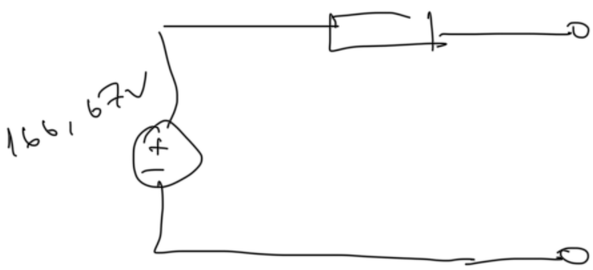
$$\frac{V_x - 50}{3} + \frac{V_x}{6} + \frac{V_x - V}{7} = 0$$

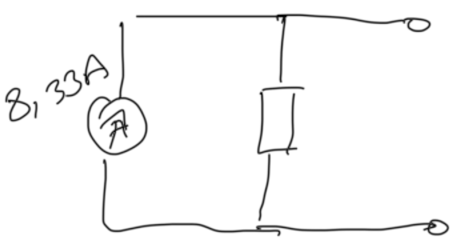
$$2V_x - 1\ 0 + V_x + 3V_x = 0$$

$$6V_x = 100 \qquad \boxed{V_x = 16,67}$$

Now; $I_N = 0,5V_x = 0,5 \times 16,67 = 8,33A$

$$\boxed{I_N = 8,33A}$$





Lab 4
Circuit Analysis Techniques

A. Background

Several solution techniques may be used to solve an electrical circuit. The widely used techniques are:

i. Exhaustive Method
ii. The Node Voltage Method
iii. The Mesh Current Method
iv. Circuit Substitution using Thevenin's and/or Norton's Equivalent Circuits
v. Superposition

In this experiment, four of these techniques will be studied; namely (i) The Node Voltage Method, (ii) The Mesh Current Method, (iii) Thevenin and Norton Circuits, and (iv) Superposition. The first two methods will be demonstrated on the following circuit (Fig. 4.1).

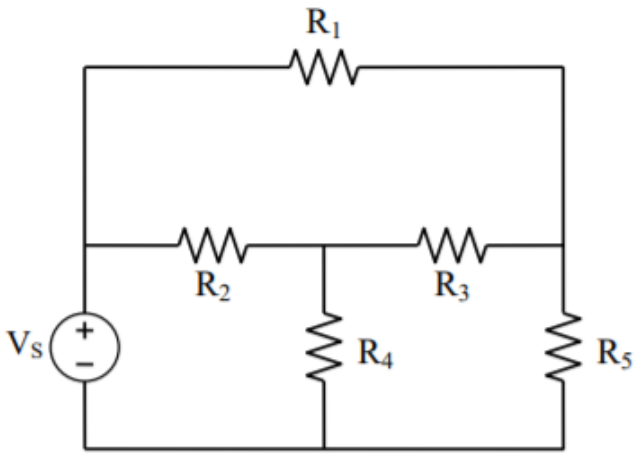


Fig. 4.1.

A.1. The Node Voltage Method

The node voltages Va and Vb at the nodes a and b (Fig. 4.2) are assumed parameters to be solved. All other circuit parameters can easily be obtained from Va and Vb.

To solve the two unknown values Va and Vb, two equations are needed. The KCL equations written for nodes a and b provide the required equations.

$$I_2 = I_3 + I_4 \tag{4.1}$$

and

$$I_5 = I_1 + I_3 \tag{4.2}$$

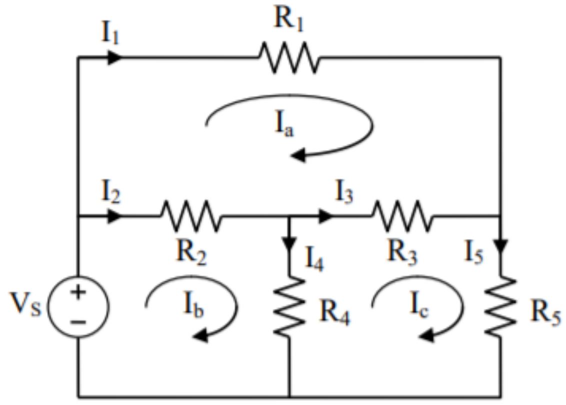
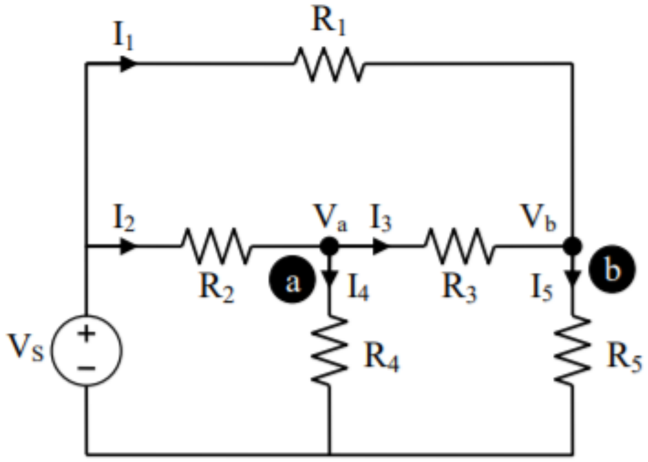


Fig. 4.2.



All these current values are required to be expressed in terms of the unknown values Va, Vb and the known source voltage Vs. Substituting

$$I_1 = \frac{V_S - V_b}{R_1}$$

$$I_2 = \frac{V_S - V_a}{R_2}$$

$$I_3 = \frac{V_a - V_b}{R_3}$$

$$I_4 = \frac{V_a}{R_4}$$

and

$$I_5 = \frac{V_b}{R_5}$$

into (4.1) and (4.2), two equations in terms of Va, Vb are obtained. The simultaneous solution of these two equations yield Va, Vb. The other circuit parameters are then obtained.

A.2. The Mesh Current Method

The meshes in a circuit are identified and mesh currents are assumed to flow through each mesh as shown in Fig. 4.3. As the circuit equations, mesh equations are written as

$R_1 I_1 - R_3 I_3 - R_2 I_2 = 0$
$R_2 I_2 + R_4 I_4 - V_s = 0$
$R_3 I_3 + R_5 I_5 - R_4 I_4 = 0$

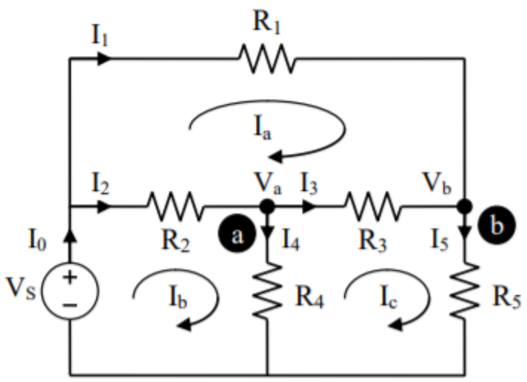


Fig. 4.3.

The branch currents I_1, I_2, I_3, I_4 and I_5 are easily expressed in terms of the unknown mesh currents Ia, Ib, and Ic as:

$$I_1 = I_a$$

$$I_2 = I_b - I_a$$

$$I_3 = I_c - I_a$$

$$I_4 = I_b - I_c$$

$$I_5 = I_c$$

Substituting these into the mesh equations written, three equations for Ia, Ib, and Ic are obtained. Solving these equations simultaneously, the unknown currents Ia, Ib, and Ic are determined. Then using these values, the other circuit parameters can be calculated.

B. Experimental Work

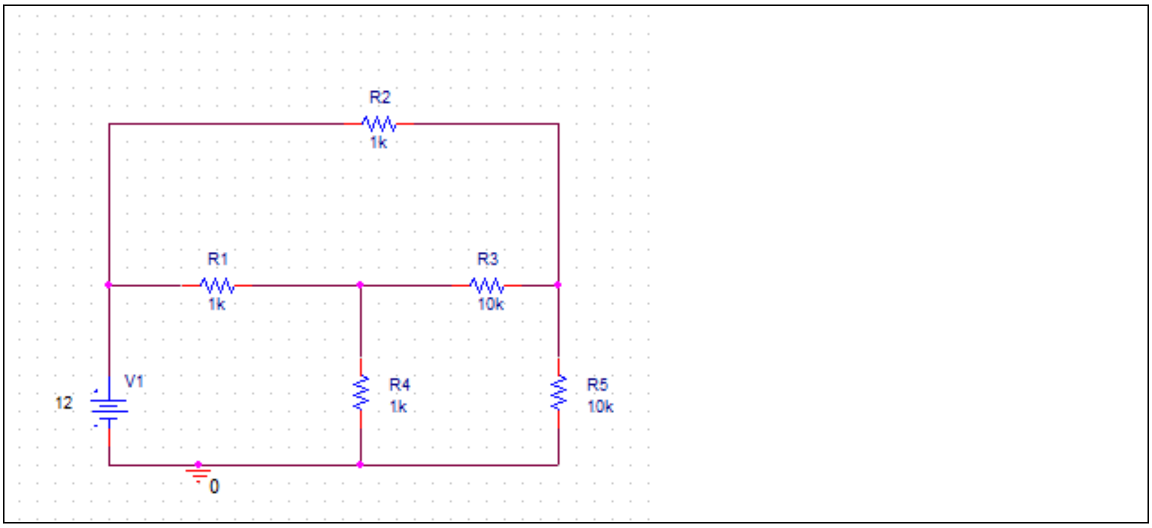
B.1. Node Voltage & Mesh Current Methods

Consider the circuit given in Fig. 4.4. (Vs=12V, $R_1=R_2=R_4=1k\Omega$ and $R_3=R_5=10k\Omega$)

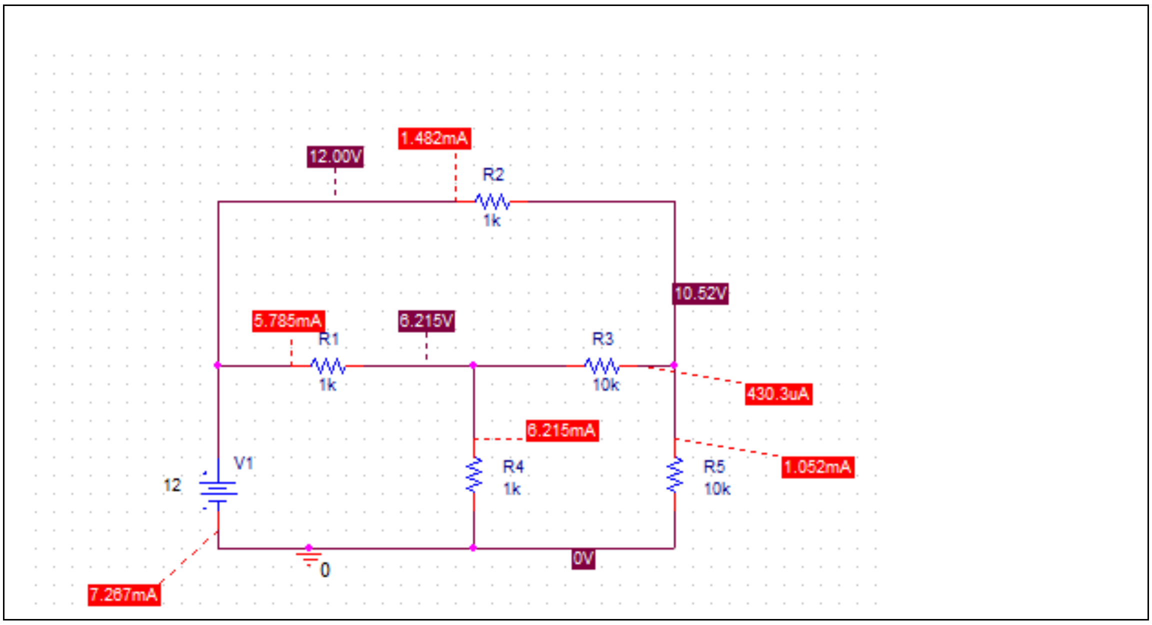
1) Use OrCAD/PSpice to find the currents I_0, I_1, I_2, I_3, I_4 and I_5 by performing bias point analysis

Fig. 4.4.

i. Circuit Schematic



ii. Simulation Output (I_0, I_1, I_2, I_3, I_4 and I_5)



2) Set up the given circuit on a breadboard.

2.1. Measure the node voltages Va and Vb. Calculate the branch currents I_1, I_2, I_3, I_4 and I_5 using Va, Vb and Vs.

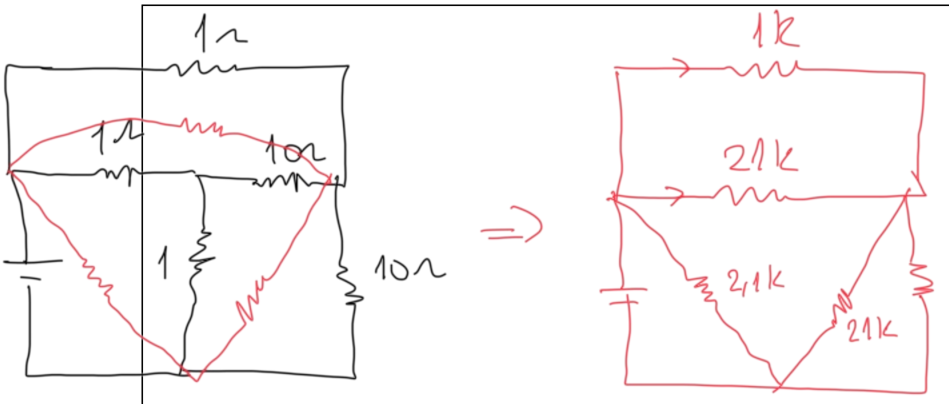
Vs (volts) (Measured)	Va (volts) (Measured)	Vb (volts) (Measured)	I_1 (mA)	I_2 (mA)	I_3 (mA)	I_4 (mA)	I_5 (mA)
11.9	6.25	10.5	1,5	5,76	0,448	6,2	1,052

2.2. Measure the branch currents I_0, I_1, I_2, I_3, I_4 and I_5

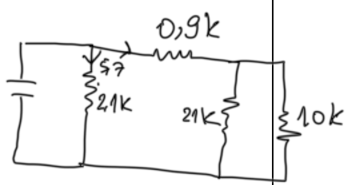
I_0 (mA)	I_1 (mA)	I_2 (mA)	I_3 (mA)	I_4 (mA)	I_5 (mA)
7.38	1.51	5.85	0.41	6.28	1.1

2.2. Compare the results you obtained in Part 2.1. and 2.2.

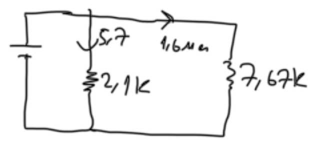
iii. Hand Calculations (Q2.2)



$$\frac{1}{1} + \frac{1}{21} = \frac{22}{21}$$



$$\frac{1}{10} + \frac{1}{21} = 6,77k$$



$V = IR \quad \frac{V}{R} = i \quad \frac{12V}{1,64} = 7,31\,mA = I_0$

$I_1 = 1,5\,mA$

$I_2 = 5,7\,mA$

$I_3 = 0,40\,mA$

$I_4 = 6,2\,mA$

$I_5 = 1,1\,mA$

3) Using node-voltage method, calculate;

3.1. The node voltages Va and Vb.

3.2. The currents I_1, I_2, I_3, I_4 and I_5 using Va, Vb and Vs.

Using mesh current method, calculate;

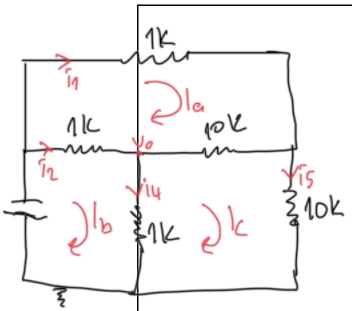
3.3. The mesh currents I_a, I_b, and I_c.

3.4. The currents I_1, I_2, I_3, I_4 and I_5 using Ia, Ib, and Ic.

3.5. Determine the current I_0 drawn from the voltage source.

3.6. Verify the power conservation principle.

iv. Hand Calculations (Q3)



$i_1 = \frac{12 - V_b}{1000}$ $\quad i_2 = \frac{12 - V_a}{1000}$

$i_3 = \frac{V_a - V_b}{1000}$ $\quad i_4 = \frac{V_a}{1000}$

$i_5 = \frac{V_b}{1000}$

$i_2 = i_3 + i_4$

$\frac{12 - V_a}{1000} = \frac{V_a - V_b}{1000} + \frac{V_a}{1000}$

$11V_a - V_b = 120 - 10V_a$

$21V_a - V_b = 120$

$V_a + 12V_b = 120$

$251 V_b = 2640$

$V_b = 10{,}52V$

$V_a = 6{,}22V$

$i_5 = i_3 + i_1$

$\frac{V_b}{10000} = \frac{V_a - V_b}{10000} + \frac{12 - V_b}{1000}$

$V_b = V_a - V_b + 120 - 10V_b$

$12V_b - V_a = 120$

$i_1 = \frac{12 - 10{,}52}{1000} = 1{,}48\,mA$

$i_2 = \frac{12 - 6{,}22}{1} = 5{,}78\,mA$

$i_3 = \frac{10{,}52 - 6{,}22}{10} = 0{,}43\,mA$

$i_4 = \frac{6{,}22}{1} = 6{,}22\,mA$

$i_5 = \frac{10{,}52}{10} = 1{,}052\,mA$

$\boxed{\text{mesh current method}}$

$i_1 = I_a$

$I_2 = I_b - I_a$

$I_3 = I_c - I_a$

$I_4 = I_b - I_c$

$I_5 = I_c$

$I_1 - 10I_3 - I_2 = 0$

$I_2 + I_4 - 12 = 0$

$10I_3 + 10I_5 - I_4 = 0$

$10I_c - 10I_a + 10I_c - I_b + I_c$

$21I_c - 10I_a - I_b = 0$

$I_b - I_a + I_b - I_c = 12$

$2I_b - I_a - I_c = 12$

$I_a - 10I_c + 10I_a - I_b + I_a = 0$

$12I_a - I_b - 10I_c = 0$

$2/ \; -I_a + 2I_b - I_c = 12$

$+ \; -10I_a + I_b + 21I_c = 0$

$I_a = 1{,}482\,mA \quad I_b = 7{,}26\,mA$

$I_c = 1{,}052\,mA$

$I_1 = 1{,}5\,mA$

$I_2 = 5{,}76\,mA$

$I_3 = -0{,}448\,mA$

$I_4 = 6{,}2\,mA$

$I_5 = 1{,}052\,mA$

B.2. Nodal Analysis with a Dependent Source

Consider the circuit given in Fig. 4.5.

1. Use PSpice to find the node voltages V_1 and V_2.
2. Also, please calculate all node voltages by hand to compare your results.

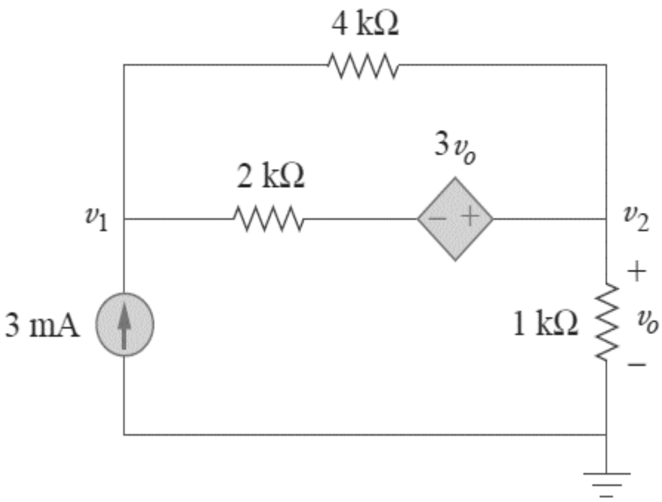
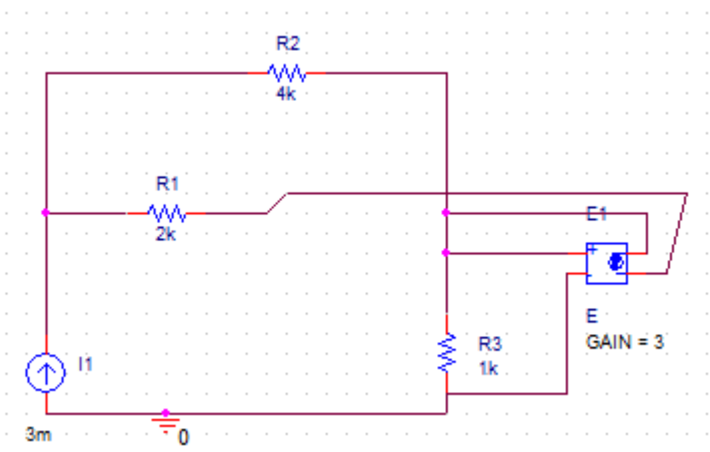
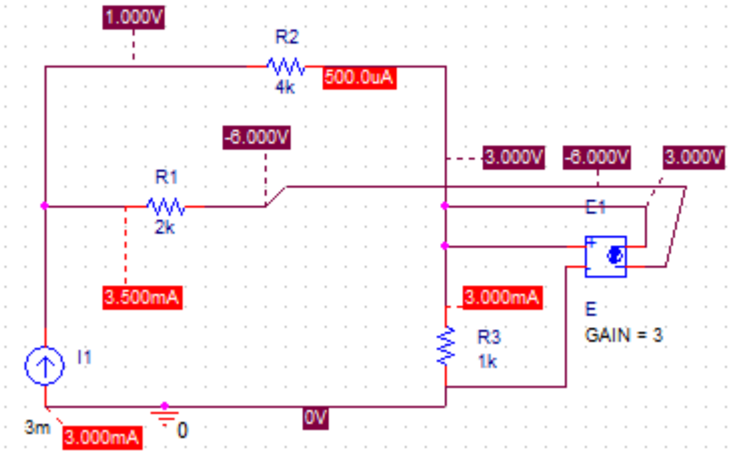


Fig. 4.5. Circuit with Dependent Source

v. Circuit Schematic .

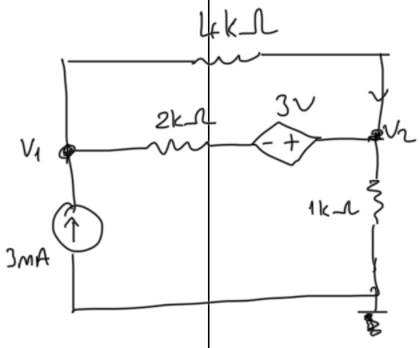


vi. Simulation Output (V_1 and V_2).



vii. *Hand Calculations (Calculate V_1 and V_2 by hand)*

$\boxed{\text{node voltage}}$



KCL at 1:

$i_2 + i_1 = 3mA$

$$\frac{V_1+3V_2-V_2}{2} + \frac{V_1-V_2}{4} = 3$$

KCL at 2:

$I_1 + I_2 = I_3$

$$\left(\frac{V_1+3V_2-V_2}{2}\right) + \left(\frac{V_1-V_2}{4}\right) = \frac{V_2}{1}$$

$2V_1 + 6V_2 - 2V_2 + V_1 - V_2 = 12$

$3V_1 + 3V_2 = 12$

$V_1 + V_2 = 4$

$V_1 = V$

$3V_1 + 3V_2 = 4V_2$

$3V_1 = V_2$

$V_2 = 3V$

$i_1 = \frac{V_1-V_2}{4}$

$i_2 = \frac{V_1+3V_2-V_2}{2}$

$i_3 = \frac{V_2}{1} = V_2 = 3V$

Mehmet Arda Uçar

Lab 5
Operational Amplifiers

A. Background

A.1. 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. 5.1.

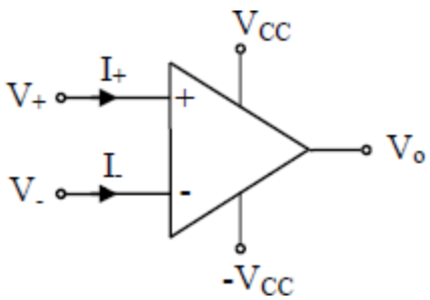


Fig. 5.1

The inputs V_+ and V_- are the differential inputs. V_0 is the output voltage and V_0 is proportional to the difference voltage V_+ and V_-. The voltages Vcc and –Vcc 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.,

$$I_+ = I_- = 0 \text{ A}$$

The output-input voltage transfer characteristics of the OPAMP is plotted on Fig. 5.2.

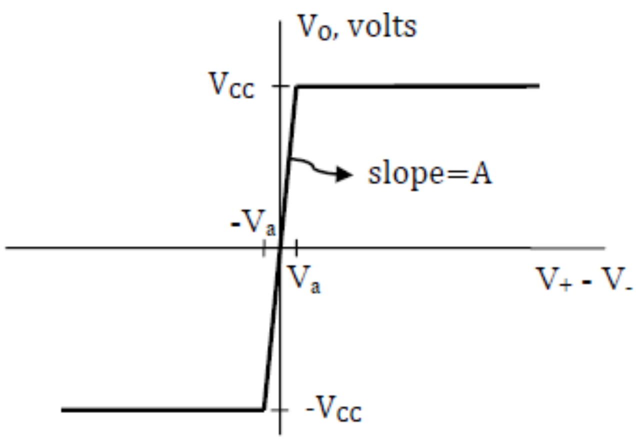


Fig. 5.2

The output Vo is proportional to the difference signal $V_+ - V_-$ in a very narrow range (-

Va < Vo < Va) and the gain there is A. The output voltage Vo is always between –Vcc and Vcc, i.e., -Vcc < Vo < Vcc.

If Vcc = 10 V, and if A = 10^6, then Va = 1μV. Since Va is in general very small, in the linear region where –Vcc < Vo < Vcc,

$$V_+ - V_- = 0\ V \qquad\qquad -Vcc < Vo < Vcc$$

or

$$V_+ = V_- \qquad\qquad -Vcc < Vo < Vcc$$

A.2. Inverting Amplifier

Consider the operational amplifier circuit given in Fig. 5.3.

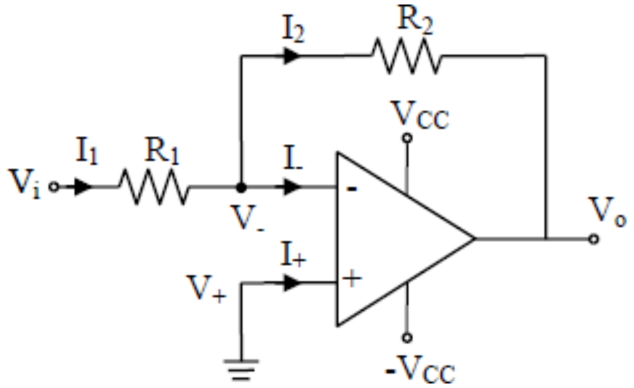


Fig. 5.3

Since $V_+ = V_-$ and since $V_- = 0$ V (grounded), then $V_- = 0$ V (not actual ground, but called as virtual ground). The current I_1 then can be calculated as,

$$I_1 = \frac{V_i - V_-}{R_1} = \frac{V_i}{R_1}$$

Similarly,

$$I_2 = \frac{V_- - V_0}{R_2} = \frac{-V_0}{R_2}$$

Since $I_- = 0$ A, then $I_1 = I_2$. Hence

$$\frac{V_i}{R_1} = -\frac{V_0}{R_2}$$

or,

$$\frac{V_0}{V_i} = -\frac{R_2}{R_1}$$

Since the gain is negative, the above amplifier is called inverting amplifier.

A.3. Noninverting Amplifier

Consider the operational amplifier circuit given in Fig. 5.4.

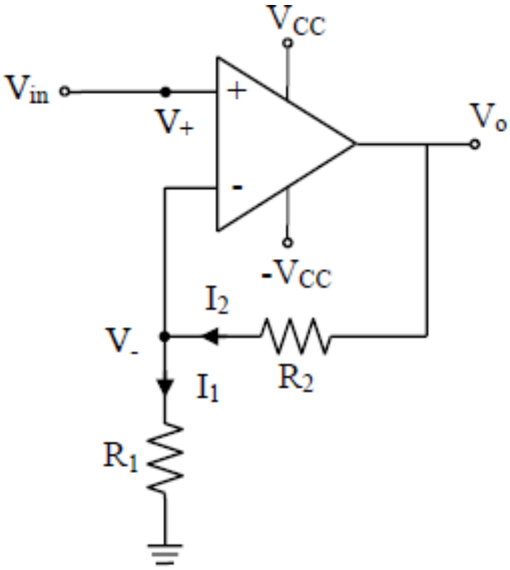


Fig. 5.4.

The currents I_1 and I_2 are calculated as

$$I_1 = \frac{V_-}{R_1}$$

and,

$$I_2 = \frac{V_0 - V_-}{R_2}$$

Since $I_+ = 0$ A, then $I_1 = I_2$. Hence

$$\frac{V_-}{R_1} = \frac{V_0 - V_-}{R_2}$$

or,

$$\left(\frac{1}{R_1} + \frac{1}{R_2}\right) V_- = \frac{1}{R_2} V_0$$

Since $V_- = V_+ = V_i$, the final gain expression is obtained as

$$\frac{V_0}{V_i} = \left(1 + \frac{R_2}{R_1}\right)$$

Since the gain is positive, above amplifier is called as noninverting amplifier.

A.3. A Practical OPAMP

In the market there are several different operational amplifier having different characteristics. The most commonly used operational amplifier is uA741. The data sheet of the operational amplifier uA741 is given at the web page

http://homes.ieu.edu.tr/maskar/EEE205/General/ua741-OPAMP.pdf

The top view of uA741 is given in Fig. 5.5.

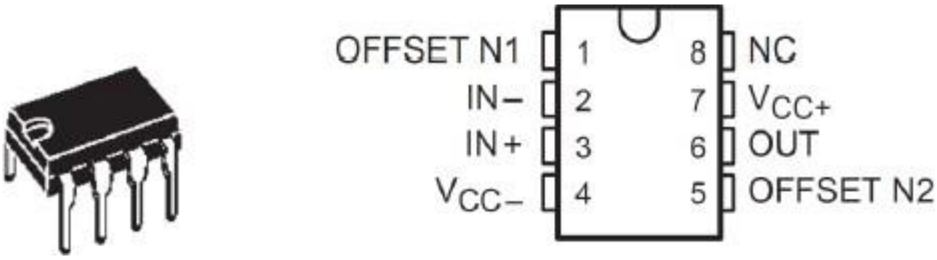


Fig. 5.5.

As seen from the voltage transfer characteristics in Fig. 5.2., the output voltage Vo may not be zero when $V_+ - V_- = 0$ V, i.e., the voltage transfer characteristics may not pass through origin. It is said that there is an offset at the input. To satisfy this condition OFFSET N1 and OFFSET N2 terminals are used. This is known as cancellation.

In the data sheet, the following circuit (Fig. 5.6) with a potentiometer is suggested for offset cancellation.

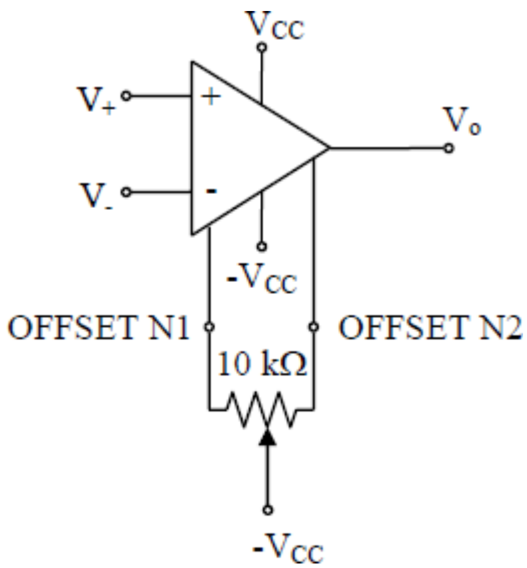
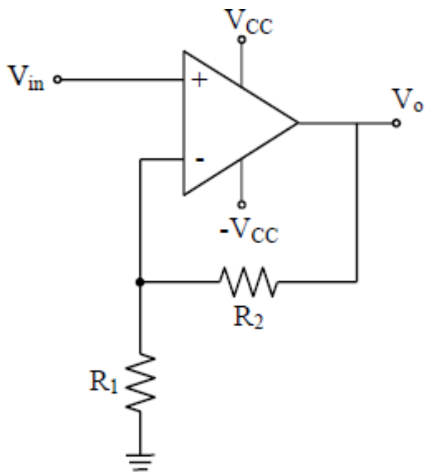


Fig. 5.6

B. Experimental Work
B.1. Noninverting Amplifier with DC Input

Consider the circuit given in Fig. 5.7.

1) Use OrCAD/PSpice to find output voltages when the different DC input voltages are applied by using DC sweep analysis.



Circuit Parameters

$R_1 = 1\ k\Omega$

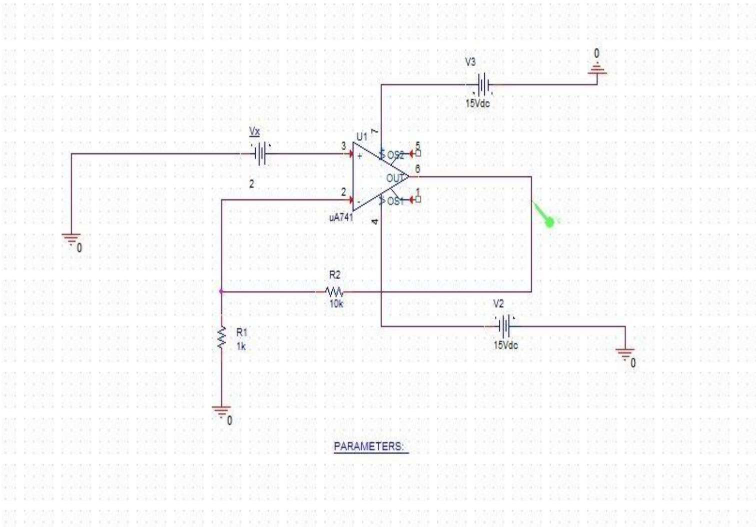
$R_2 = 10\ k\Omega$

$V_{CC} = 15\ V$

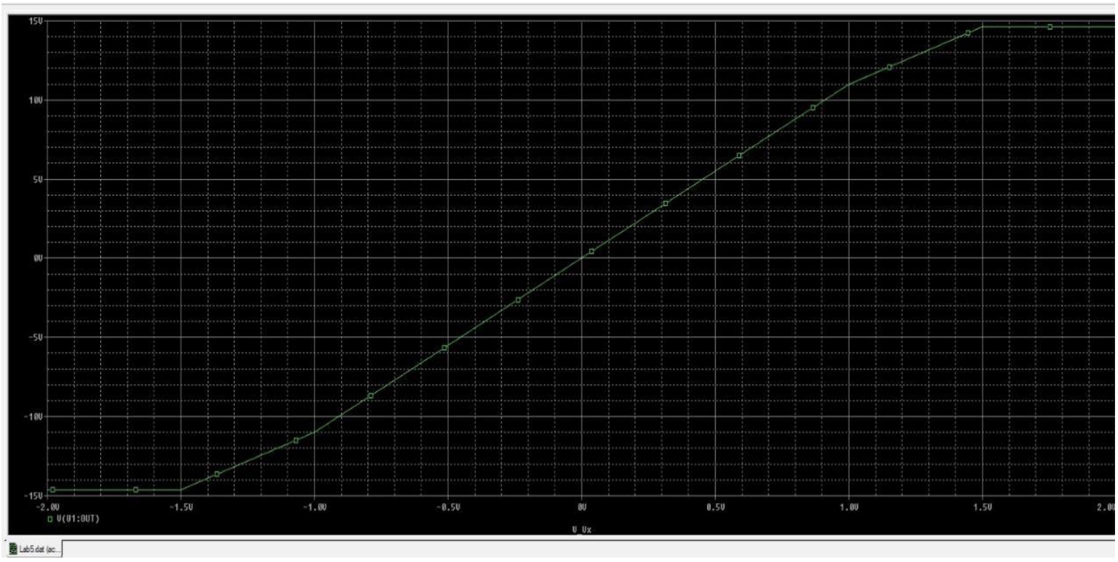
$-V_{CC} = -15\ V$

Fig. 5.7.

i. Circuit Schematic

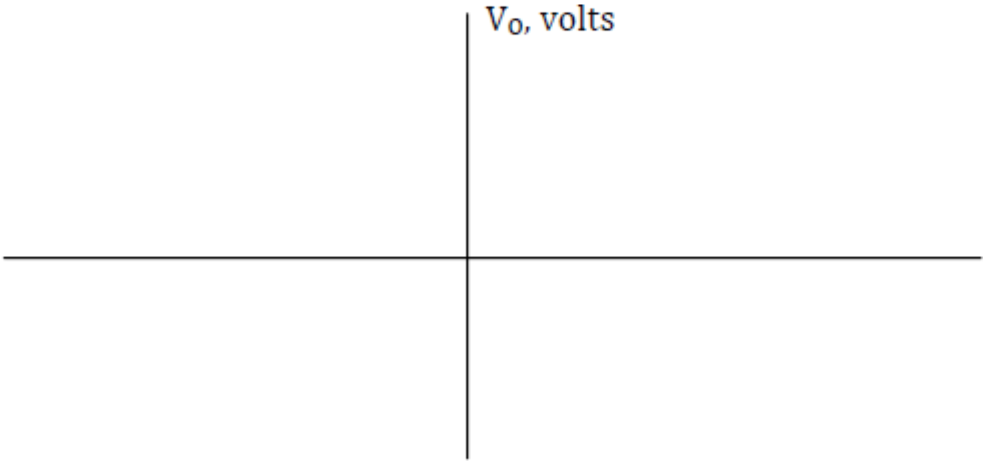


ii. Circuit Schematic Output



2) Set up the given circuit on a breadboard. Measure the output voltages and fill in the following table and calculate the gain. By using the table, plot the voltage transfer characteristics

iii. Plot Voltage Transfer Characteristics

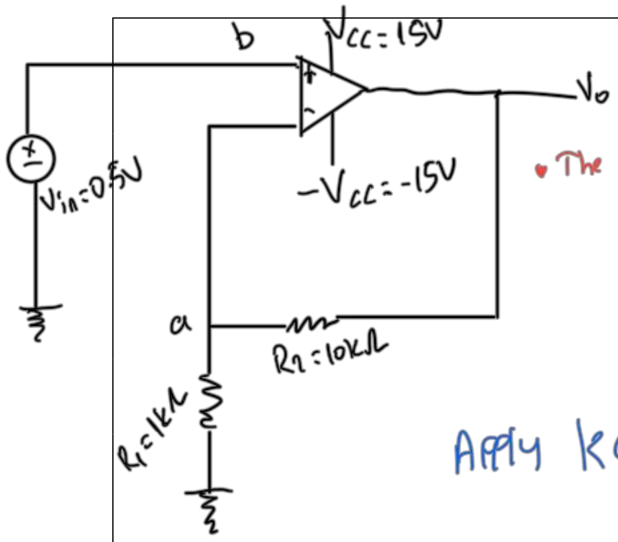


Vi (V)	Vo (V)	Gain (Vo/Vi)
-2	-12.775	6.3875
-1.5	-12.775	8.5166
-1	-10.823	10.8230
-0.75	-8.227	10.9693
-0.5	-5.457	10.9140
-0.25	-2.734	10.9360
0	-0.777	0
0.25	3.699	14.796
0.5	5.204	10.408
0.75	8.089	10.7853
1	10.829	10.8290
1.5	14.749	9.8326
2	14.747	7.3735

Table 5.1

3) Also, please calculate Vo when Vi = 0.5 V.

iv. Hand Calculation (Find Vo when Vi = 0.5 V)



• From the circuit the voltage at the non-inverting terminal of operational amplifier is $= V_b = V_{in} = 0.5V$

• The voltage at a is $= V_a = V_b = 0.5V$

Apply KCL at node a;

$$\frac{V_a - V_o}{R_2} + \frac{V_a}{R_1} = 0$$

$$\frac{0.5 - V_o}{10k} + \frac{0.5}{1k} = 0$$

$$0.5\left(\frac{1}{10k} + \frac{1}{1k}\right) = \frac{V_o}{10k}$$

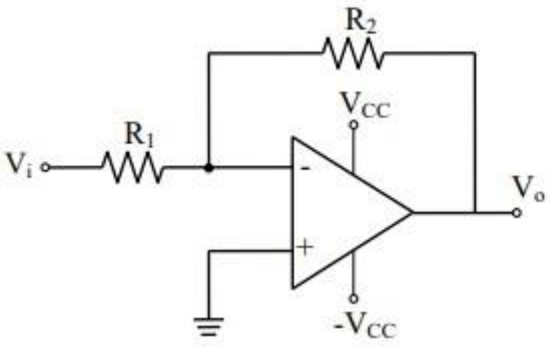
$$0,5(10+1) = V_o$$

The output voltage of the non-inverting Amlifier is $= V_o = 5.5V$

B.2. Inverting Amplifier with Sinusoidal Input

Consider the circuit given in Fig. 5.8.

1) Use PSpice to find the output voltage when the input voltage is 1 Vp-p and the frequency is 1 kHz. (Hint : Use Transient Response)



Circuit Parameters

$R_1 = 1\ k\Omega$

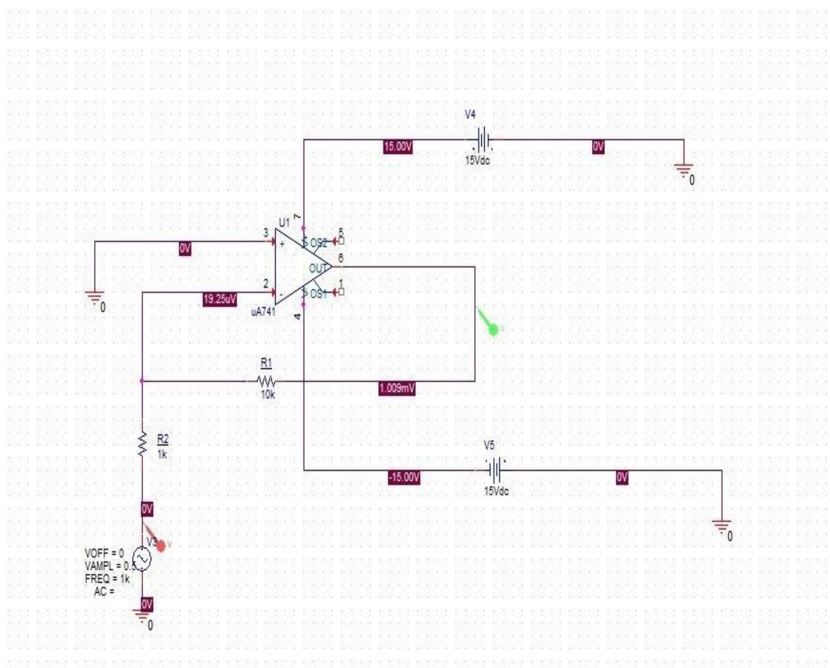
$R_2 = 10\ k\Omega$

$V_{CC} = 15\ V$

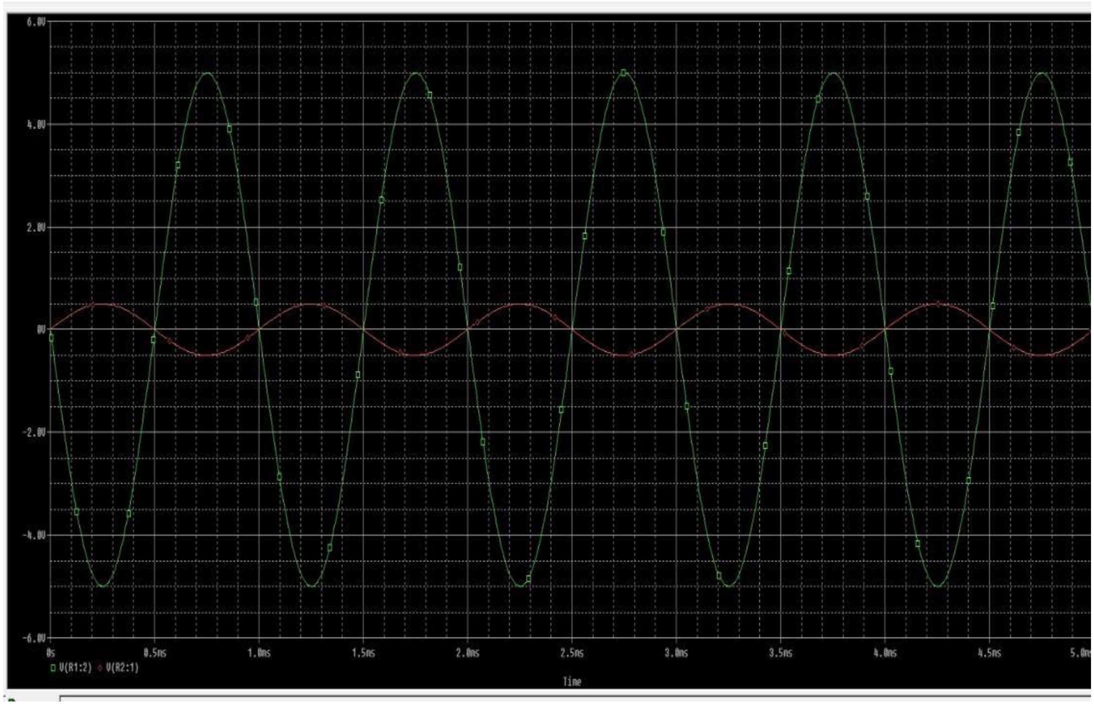
$-V_{CC} = -15\ V$

Fig. 5.8.

v. Circuit Schematic

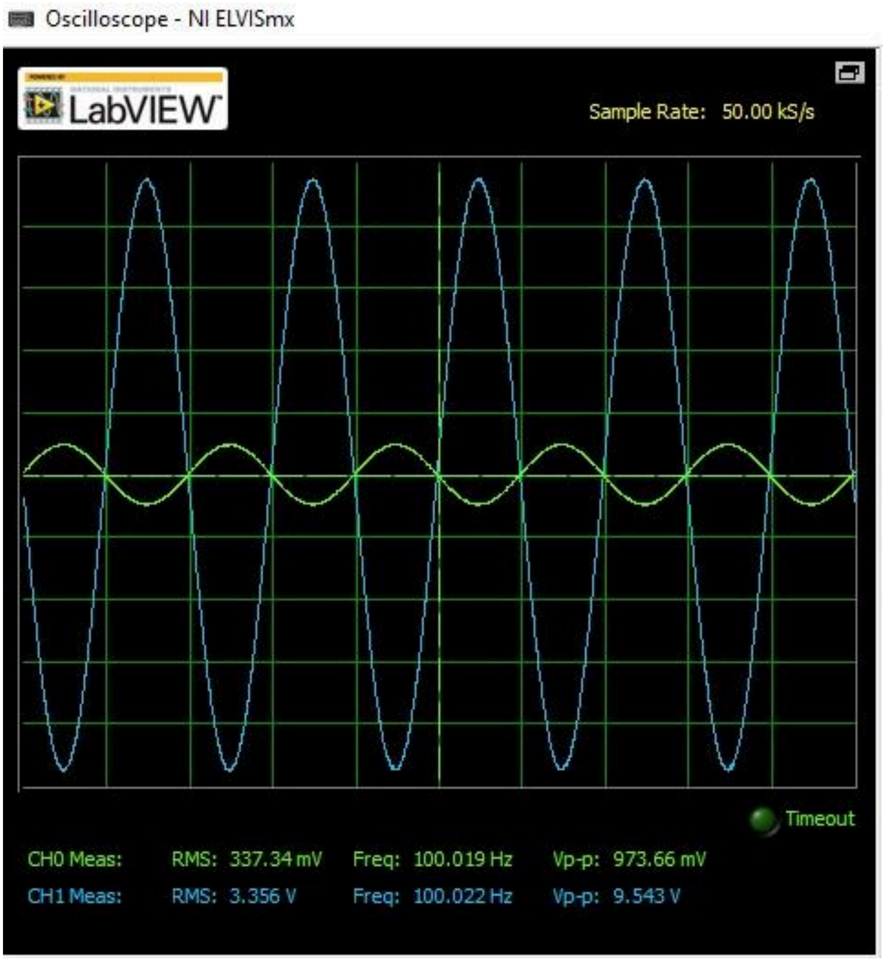


vi. Simulation Output (Both Vi & Vo)



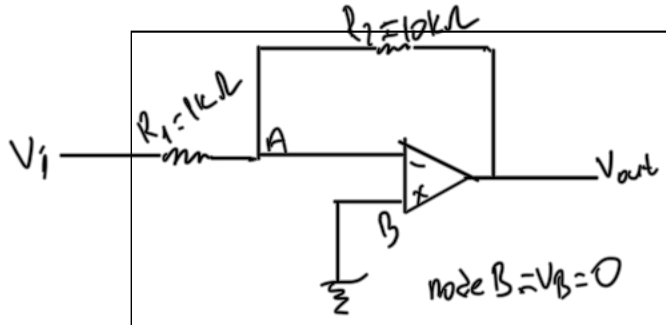
2) Set up the given circuit on a breadboard. Measure the output voltage when the input voltage is 1 Vp-p and the frequency is 1 kHz.

vii. Mesurement Output (Both Vi & Vo)



3) Explain what happens when you increase the input voltage.

viii. Hand Calculation (Find Vo)



Apply KCL at node A

$$\frac{V_A - V_i}{R_1} + \frac{V_A - V_o}{R_2} = 0$$

$$\frac{0 - V_i}{10k} + \frac{0 - V_o}{1k} = 0$$

$$V_o = V_i \cdot \left(-\frac{10}{1}\right)$$

The output voltage = $V_o = -10 V_i$

By increasing input voltage Vi, the output voltege incerases Linearly by 10 times and the phase Shift Between input and output is 180°

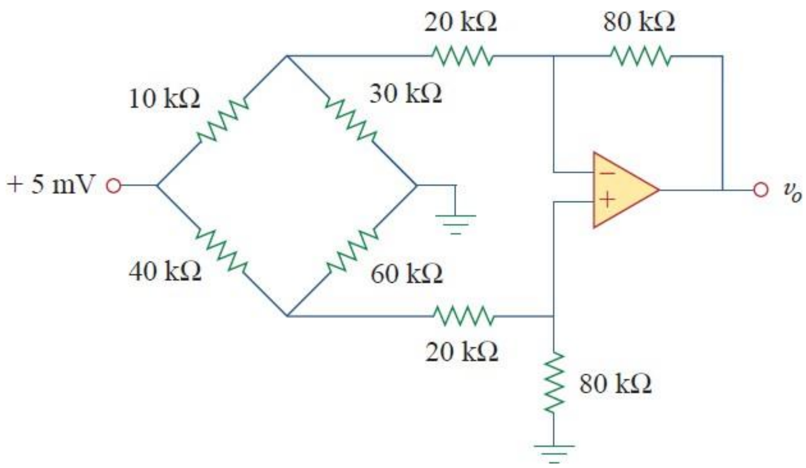
B.3. Differential Amplifier

The figure 5.9. shows a differential amplifier driven by a bridge.

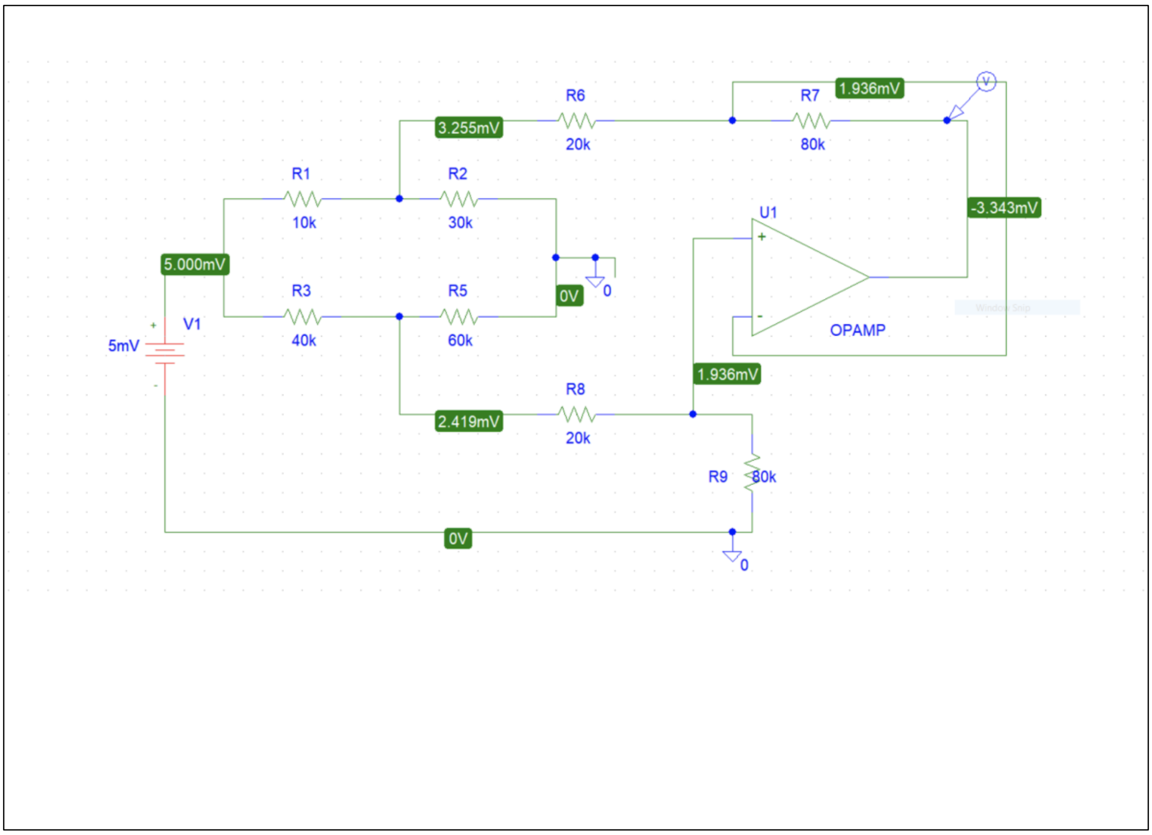
1) Use OrCAD/PSpice to find the V_0.

2) Also, please calculate Vo voltage by hand to compare your results.

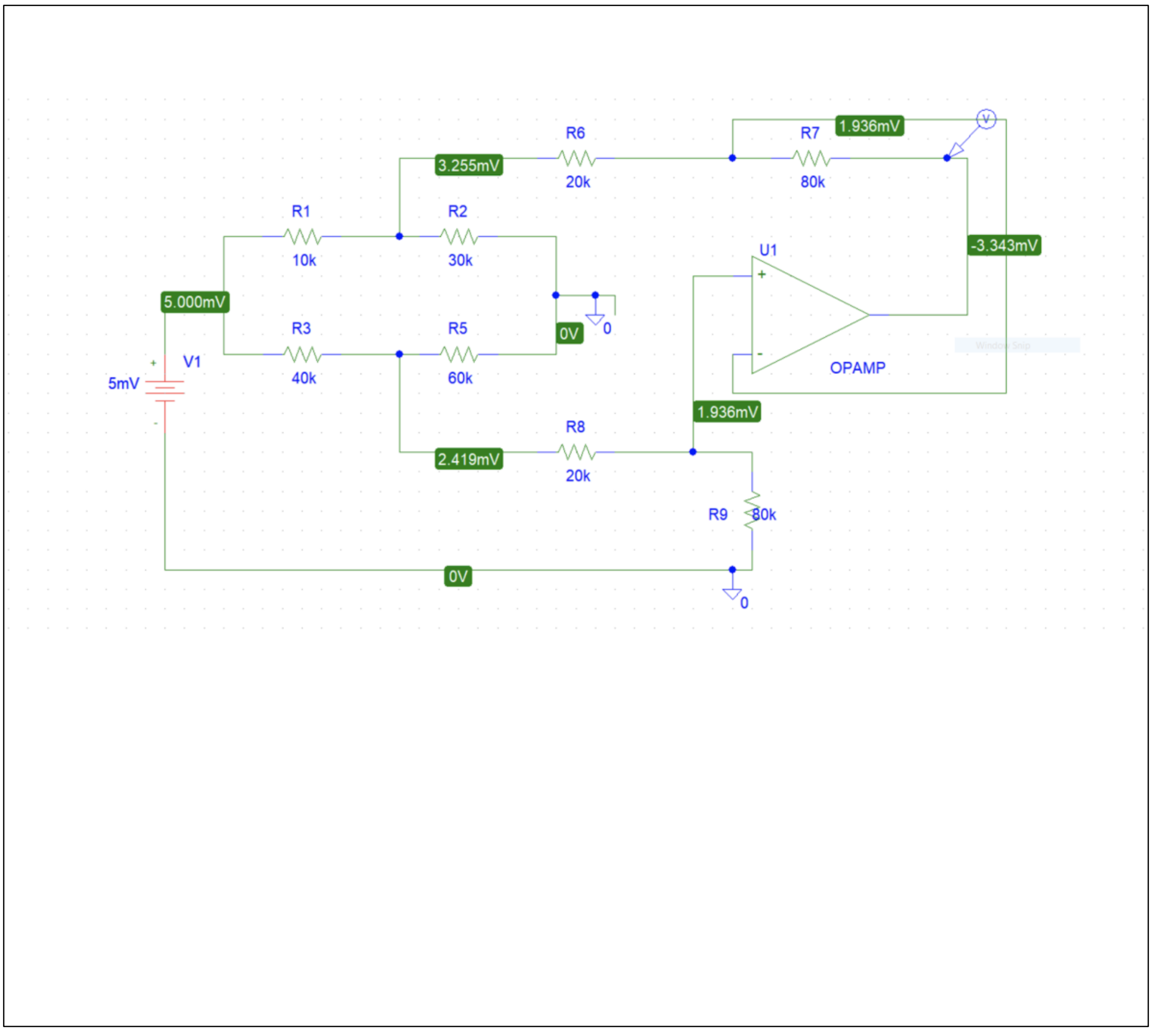
Fig. 5.9.

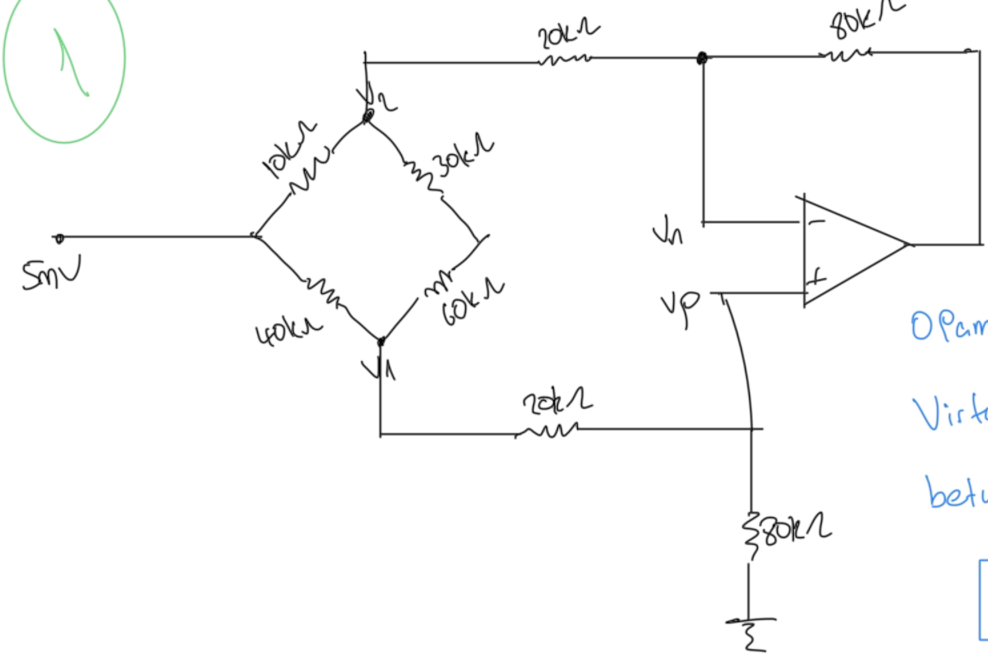


i. Circuit Schematic



ii. *Simulation Output (V_0).*





OPamp is in negative feedback

Virtual short exists

between Vp and Vn

$\boxed{V_p = V_n}$

2

iii. *Hand Calculations (Calculate the gain V_0 by hand)*

KCL at V_1

$$\frac{V_1 - 5mV}{40k\Omega} + \frac{V_1}{60k\Omega} + \frac{V_1}{100k\Omega} = 0$$

$$V_1\left[\frac{1}{4} + \frac{1}{6} + \frac{1}{10}\right] = \frac{5mV}{4}$$

$$\boxed{V_1 = 2.419mV}$$

$$V_P = V_1 \times \frac{80k\Omega}{80k\Omega + 20k\Omega}$$

$$\boxed{V_P = 1.935\,mV}$$

3

Since, $V_P = V_n$

KCL at node V_2

$$\frac{V_2 - 5mV}{10k} + \frac{V_2 - V_n}{20k} + \frac{V_2}{30k} = 0$$

$$V_2\left[\frac{1}{10} + \frac{1}{20} + \frac{1}{30}\right] = \frac{5mV}{10} + \frac{1.935mV}{20}$$

$$V_2\left[\frac{1}{1} + \frac{1}{2} + \frac{1}{3}\right] = \frac{5mV}{1} + \frac{1.935\,mV}{2}$$

$V_2 = 3.2547$
$\cong 3.255mV$

4

KCL at node V_n;

$$\frac{V_n - V_2}{20k} + \frac{V_n - V_0}{80k} = 0$$

$$\frac{V_n}{20} - \frac{V_2}{20} + \frac{V_n}{80} = \frac{V_0}{80}$$

$V_o = 4V_n - 4V_2 + V_n = 5V_n - 4V_2$

5

$V_o = -3.345mV$ (Calculated V_o)

$V_o = -3.343mV$ (measured V_o)

A.4. Cascaded Op Amp Circuits

Consider the Cascaded Op amp circuit below.

1) Use PSpice to find the voltage V_o shown in the circuit below.
2) Also, please calculate V_o by hand assuming opamp is ideal.

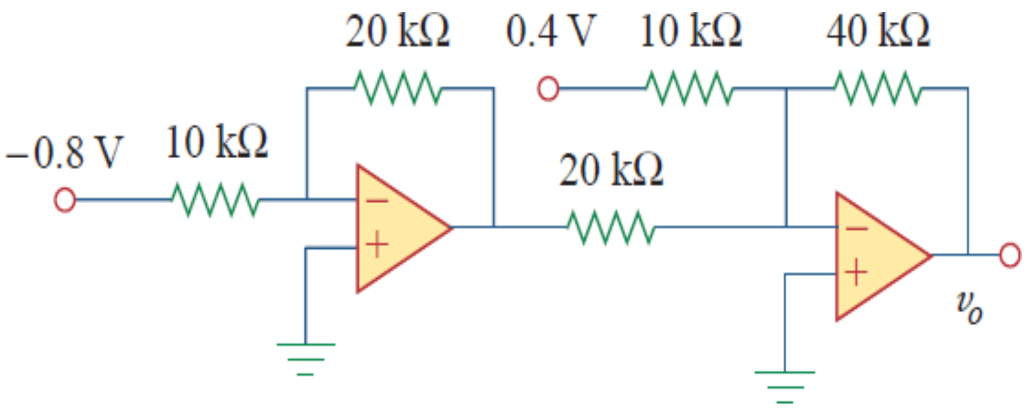
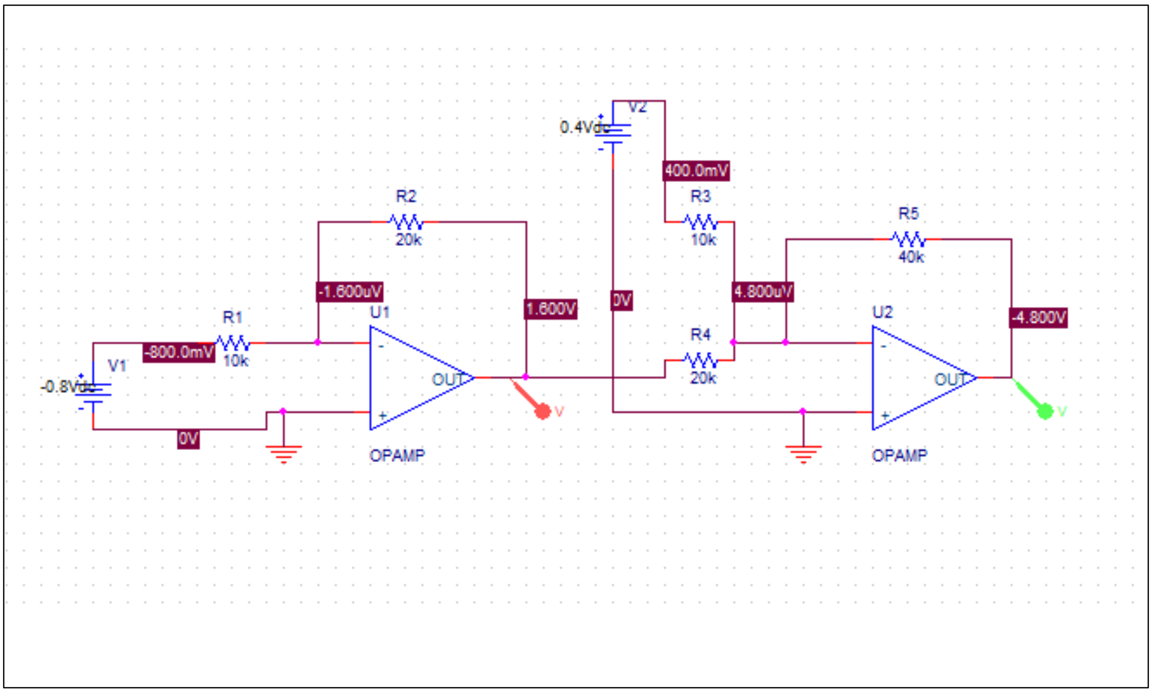
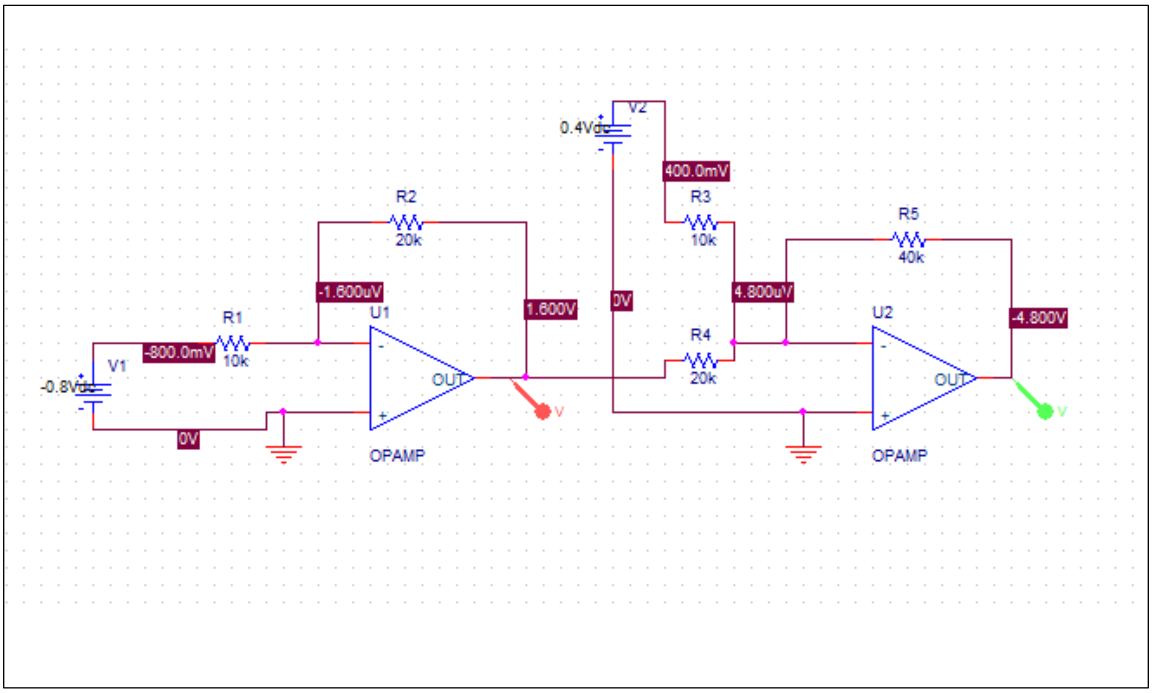


Fig. 5.10.

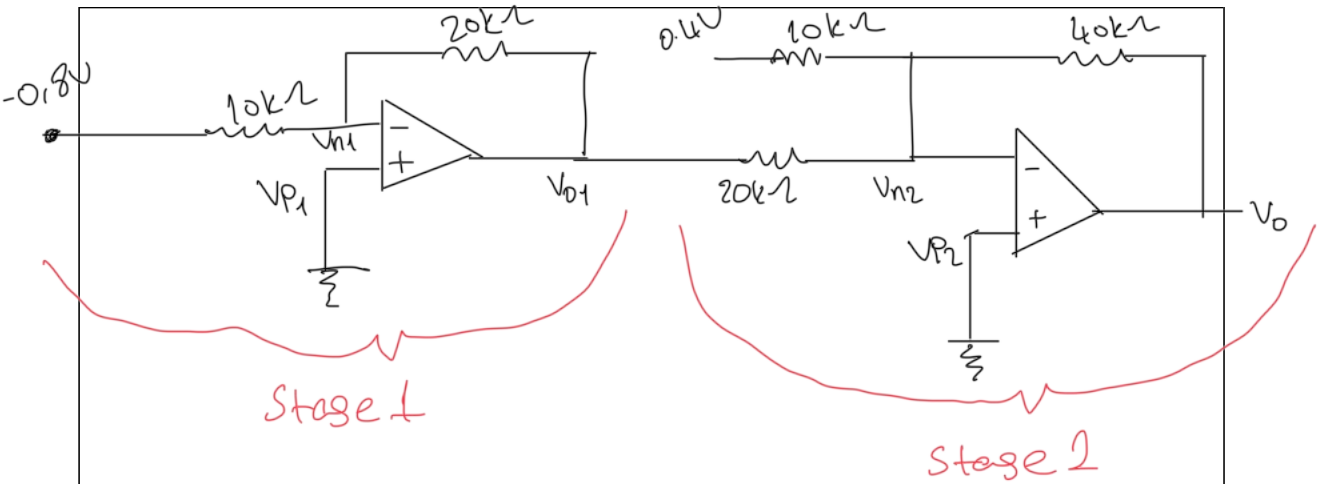


iv. Circuit Schematic

v. *Simulation Output (V_o)*



vi. *Hand Calculations (calculate V_o)*



* Both opamp is in negative feedbacks, virtual short exists;

$V_{n1} = V_{p1}$ and $V_{n2} = V_{p2}$

Stage 1

Output V_{o1} of stage 1 is

KCL at node V_{n1};

$$\frac{V_{n1}-(-0.8)}{10k\Omega}+\frac{V_{n1}-V_{o1}}{20k\Omega}=0$$

$$V_{n1}=V_{p1}=0$$

$$\frac{V_{o1}}{20k\Omega}=\frac{0.8}{10k\Omega} \qquad \boxed{V_{o1}=1.6V}$$

Stage 2

KCL at node V_{n2};

$$\frac{V_{n2}-V_{o1}}{20k\Omega}+\frac{V_{n2}-0.4}{10k\Omega}+\frac{V_{n2}-V_o}{40k\Omega}=0$$

$$V_{n2}=V_{p2}=0 \qquad V_{o1}=1.6V$$

$$\frac{0-1.6}{20k\Omega}+\frac{0-0.4}{10k\Omega}=\frac{V_o}{40k}$$

$$V_o=40k\left[\frac{-1.6}{20k\Omega}+\frac{-0.4}{10k}\right]$$

$$\boxed{V_o=-4.8V}$$

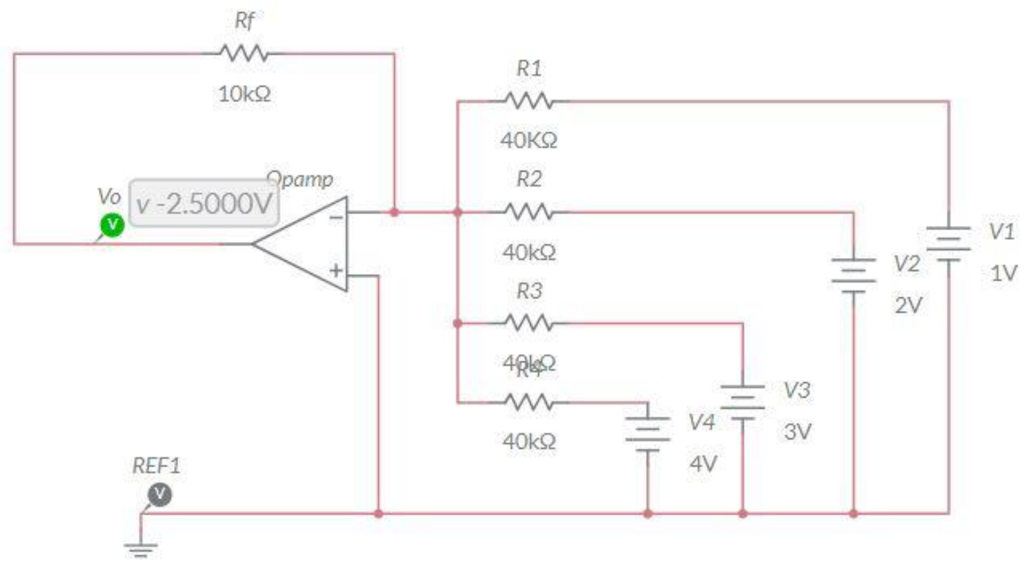
B.5. Design Problem

An averaging amplifier is a summer that provides an output equal to the average of the inputs. By using proper input and feedback resistor values one can get

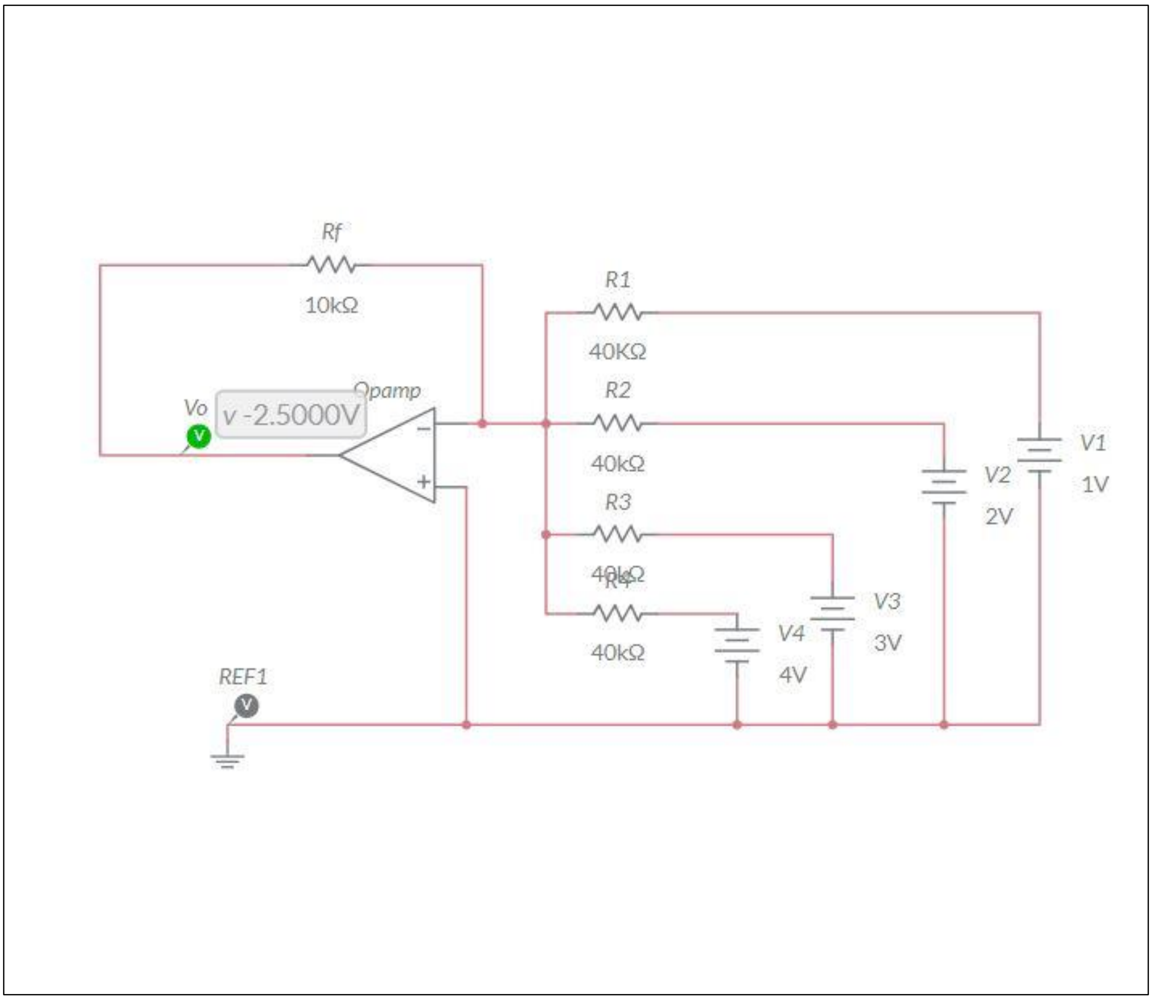
$$-Vout = \frac{1}{4}(v1 + v2 + v3 + v4)$$

Using a feedback resistor of 10kΩ design an averaging amplifier with four inputs.

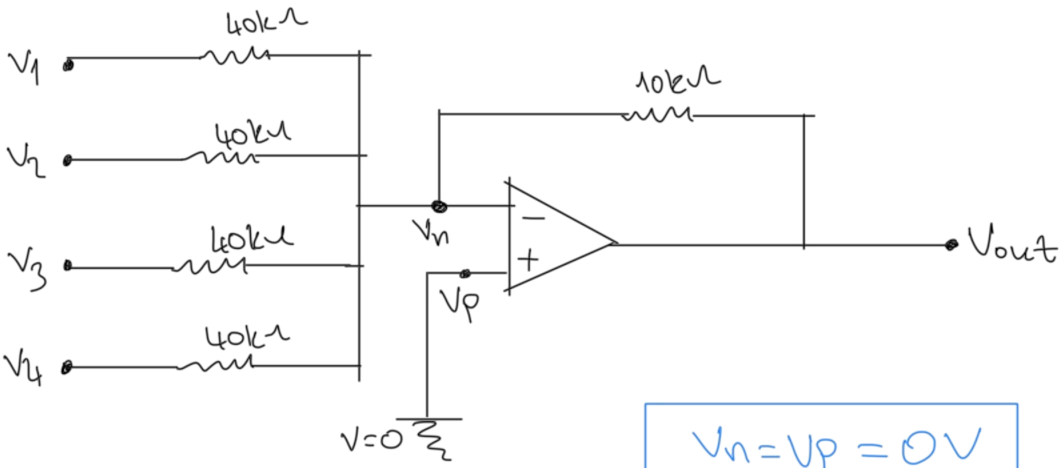
vii. Circuit Schematic



viii. Simulation Output (V_o)



ix. *Hand Calculations*



$$\boxed{V_n = V_p = 0V}$$

Applying Nodal Analysis;

$$\frac{V_4 - V_n}{40k\Omega} + \frac{V_1 - V_n}{40k\Omega} + \frac{V_2 - V_n}{40k\Omega} + \frac{V_3 - V_n}{40k\Omega} = \frac{V_n - V_o}{10k\Omega}$$

$$\frac{V_4 - 0}{40k\Omega} + \frac{V_1 - 0}{40k\Omega} + \frac{V_2 - 0}{40k\Omega} + \frac{V_3 - 0}{40k\Omega} = \frac{0 - V_o}{10k\Omega}$$

$$\frac{V_4 + V_3 + V_2 + V_1}{4} = -V_{out}$$

$$-V_{out} = \frac{1}{4}(V_1 + V_2 + V_3 + V_4)$$

$$-V_{out} = \frac{1}{4}(1+2+3+4)$$

$$-V_{out} = \frac{1}{4} \cdot 10$$

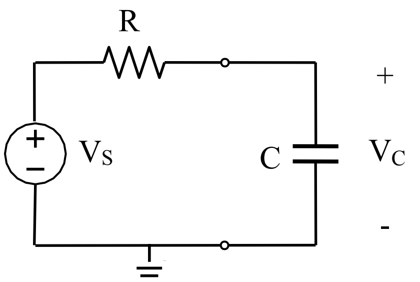
$$\boxed{V_{out} = -2.5V}$$

Lab 6
Step and Sinusoidal Responses of RC Circuits

A. Background

A.1. Step Response of RC Circuits

Consider the circuit given below in Fig. 6.1. The initial condition of the capacitor voltage is given on the right.



$V_S(t) = V_P$

$V_C(0) = V_{C0}$

Fig. 6.1. A First Order RC Circuit

The differential equation that describes the behavior of the capacitor voltage is as follows:

$$RC\frac{dV_c}{dt} + V_c = V_s$$

The input is assumed a step function of amplitude V_P as shown below.

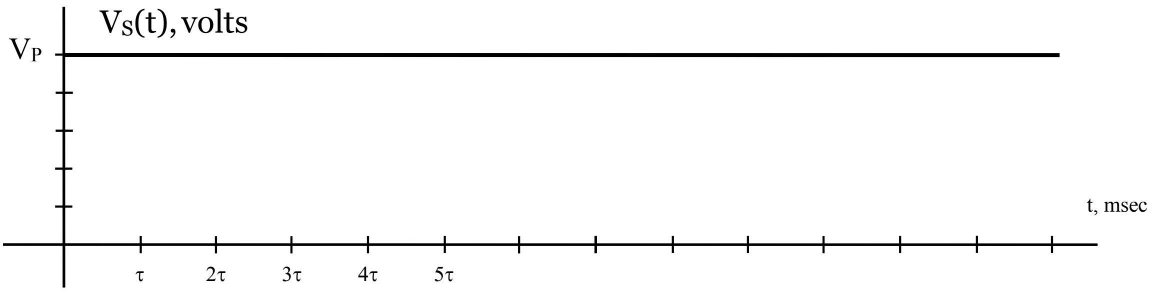


Fig. 6.2. A First Order RC Circuit

Using several different approaches, the solution to the capacitor voltage may be as obtained as

$$V_C(t) = V_P + (V_{C0} - V_P)e^{-t/\tau}\ volts$$

where $\tau = RC$

For the initial condition $V_C(0) = V_{C0} = 0$ V, the input and output voltage waveforms are plotted in Fig. 6.3.

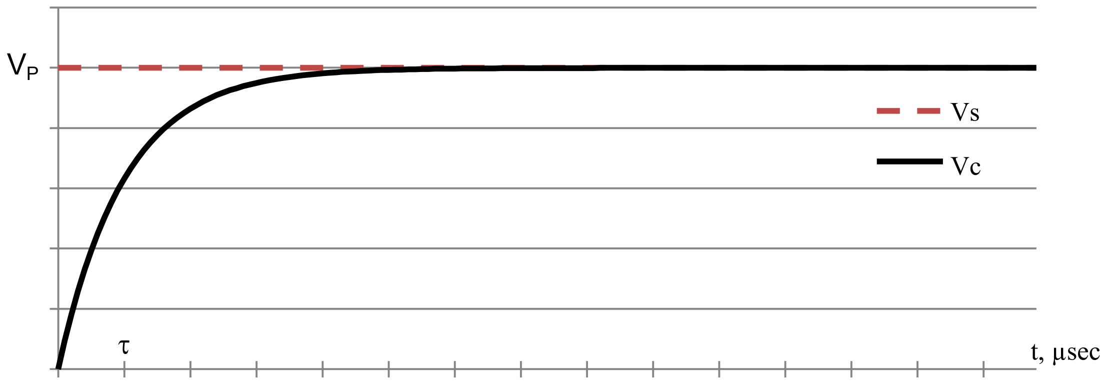


Fig. 6.3. Input and Output Waveforms in a First Order RC Circuit

Now assume a periodic waveform with $\frac{T}{2} > 5\tau$, the output will also be periodic since charging and discharging up to the final value is achieved at the end of the high and low durations.

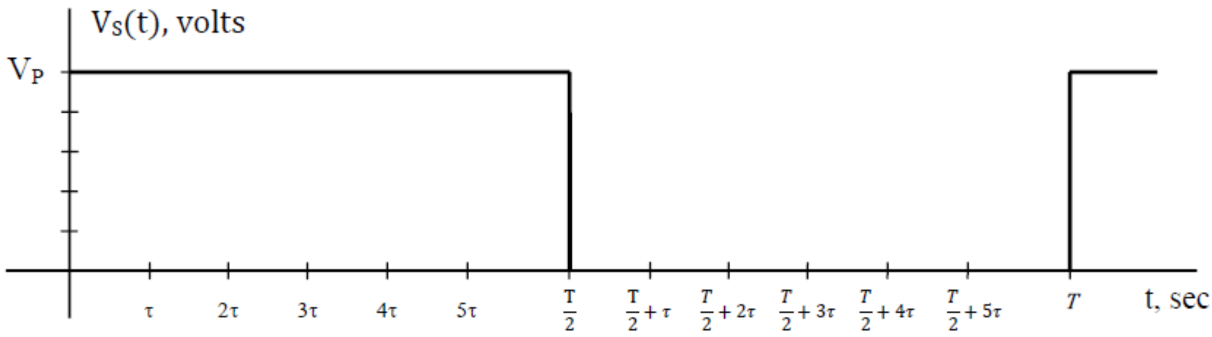


Fig. 6.4. Periodic Input Waveforms

The charging period, initial value $V_{C0} = 0V$ and $V_C(\infty) = V_P$, then

$$V_C(t) = V_P(1 - e^{-t/\tau}) \text{ volts}$$

and during discharging, $V_{C0} = V_P$ V and $V_C(\infty) = 0$ V, then

$$V_C(t') = V_P\, e^{-t/\tau} \text{ volts}$$

Substituting t' = t –T/2, it is obtained that

$$V_C(t) = \begin{cases} V_P(1 - e^{-t/\tau})\ V & \text{when } 0 \le t < \frac{T}{2} \\ V_P e^{-(t-\frac{T}{2})/\tau}\ V & \text{when } \frac{T}{2} \le t < T \end{cases}$$

The plot of Vs and Vc are given in Fig. 6.5.

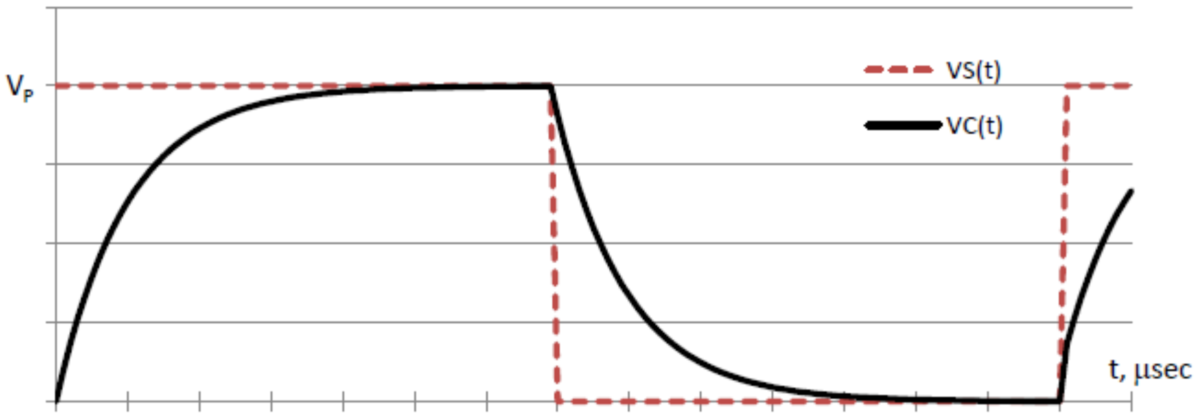


Fig. 6.5. Periodic Input and Output Waveforms

A.2. Sinusoidal Response of RC Circuits

Assume a sinusoidal voltage is applied to the input of an RC circuit (Fig. 6.6).

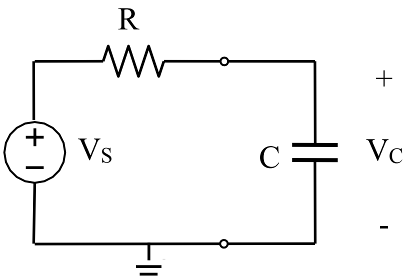


Fig. 6.6. A First Order RC Circuit with Sinusoidal Input

The capacitor voltage is obtained as (see Class Notes)

$$V_C(t) = Ae^{-t/\tau} + V_{CP} \cos(\omega_0 t + \theta) \quad \text{volts}$$

where

$$V_{CP} = \frac{V_{SP}}{\sqrt{1+(\omega_0 RC)^2}} \quad \text{and} \quad \theta = \tan^{-1}(\omega_0 RC)$$

For very large t, the first exponential term (called transient part) goes to zero. Therefore for very large t, the output observed is

$$V_C(t) = V_{CP} \cos(\omega_0 t - \theta) \quad \text{volts}$$

So for a sinusoidal input, the output is also a sinusoidal. However its amplitude and phase has been changed. The amplitude is decreased by the factor

$$\frac{1}{\sqrt{1+(\omega_0 RC)^2}}$$

The phase of the output waveform is shifted by an angle

$$\theta = \tan^{-1}(\omega_0 RC)$$

For $V_{SP} = 10$ V, $\omega_0 RC = \sqrt{3}$

$$V_{CP} = \frac{V_{SP}}{\sqrt{1+(\omega_0 RC)^2}} = \frac{10}{\sqrt{1+3}} = 5 \text{ V} \quad \text{and} \quad \theta = \tan^{-1}(\omega_0 RC) = tan^{-1}\sqrt{3} = \frac{\pi}{3} = 60^0$$

The input and output voltage waveforms are plotted in Fig. 6.7.

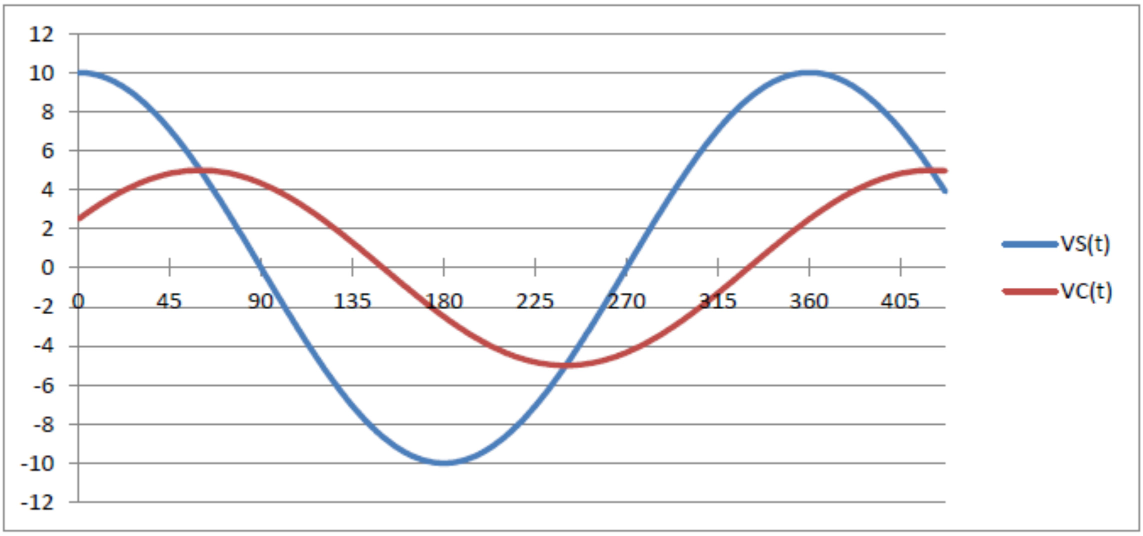


Fig. 6.7. Input and Output Waveforms at a First Order RC Circuit with Sinusoidal Input

B. Experimental Work
B.1. Step Response of RC Circuits

Consider the circuit given in Fig. 6.8.

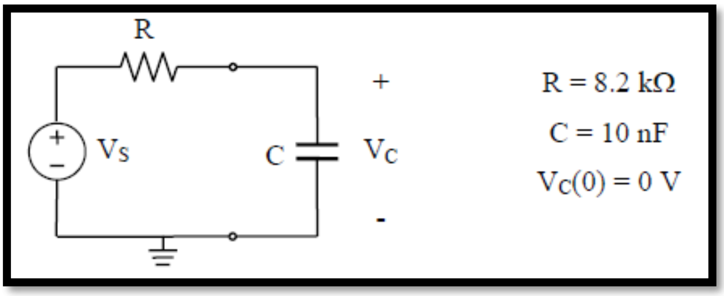
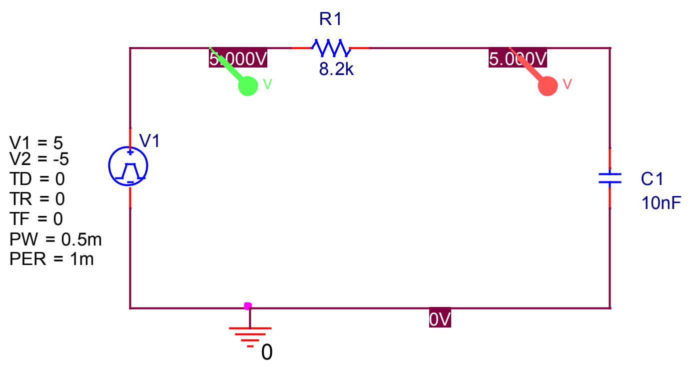


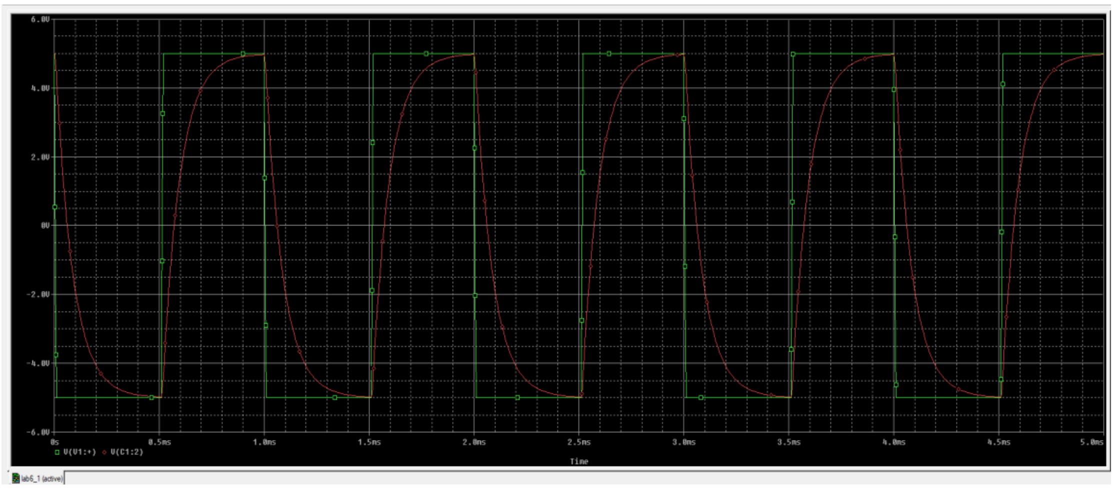
Fig. 6.8

1) Use OrCAD/PSpice to find the output voltage (Vc) when the input is a periodic 1 kHz square wave with 5 V amplitude.

i. Circuit Schematic



ii. Simulation Output



2) Set up the given circuit on a breadboard. Apply a square wave to the input. Adjust the amplitude to 5 V and frequency to 1 kHz. Measure both the input and the output voltage (Vc) and determine the time constant τ.

Time Constant : R·C

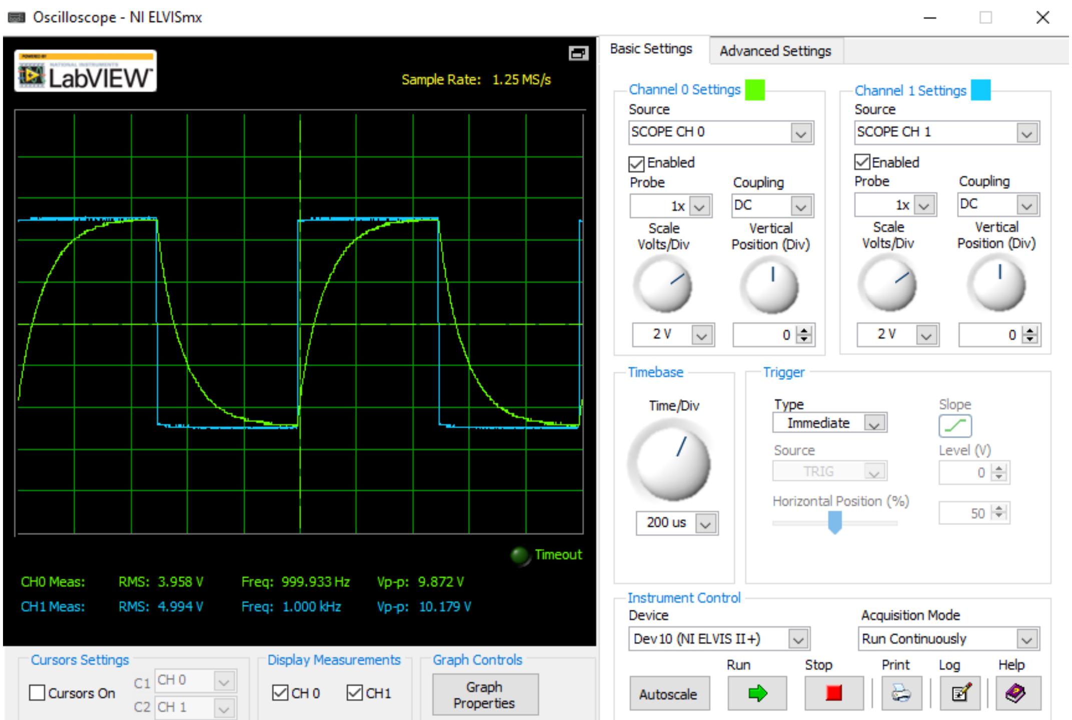
τ (time constant) = ...82μ... second.

$\tau = R \cdot C$

$\tau = 8.2 \times 10^{3} \times 10 \times 10^{-9}$

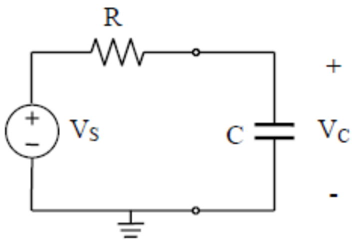
$\tau = 0.000082 \Rightarrow 82\mu$ sec

iii. Measurement Outputs from the Oscilloscope



B.2. Sinusoidal Response of RC Circuits

Consider the circuit given in Fig. 6.9.



$R = 15\ k\Omega$

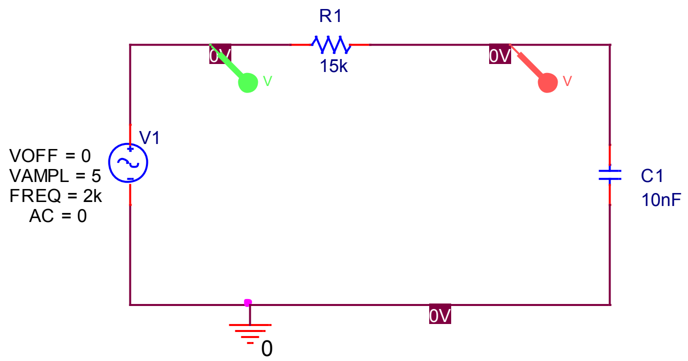
$C = 10\ nF$

$V_S(t) = 10 \cos w_0 t$ volts

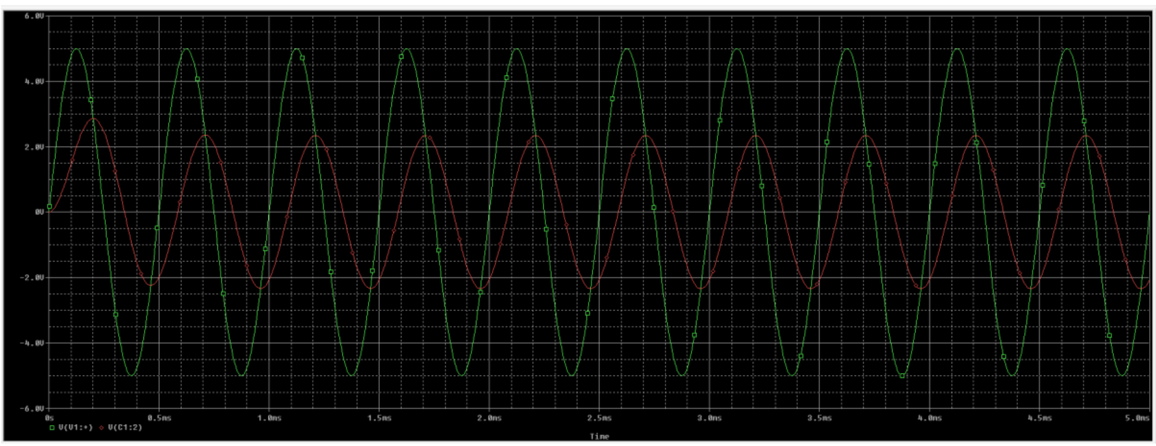
Fig. 6.9

3) Use OrCAD/PSpice to find the output voltage (Vc) when the input is a periodic 2 kHz sinusoidal wave with 5 V amplitude. Measure the phase difference.

iv. Circuit Schematic

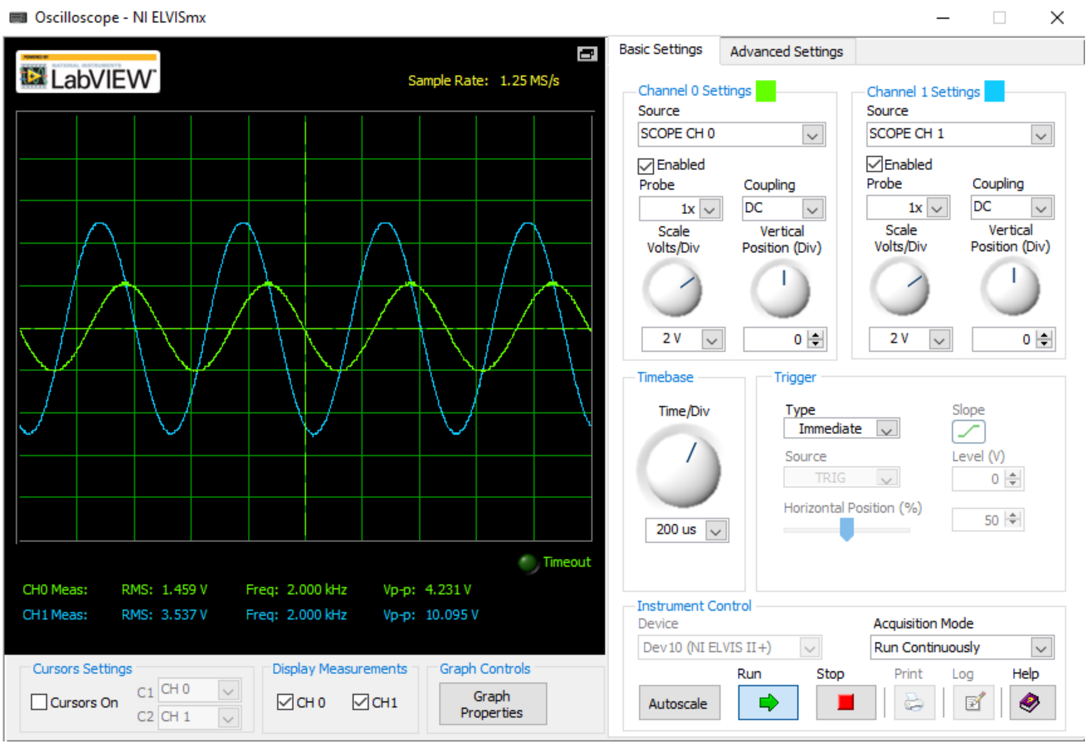


v. Simulation Output

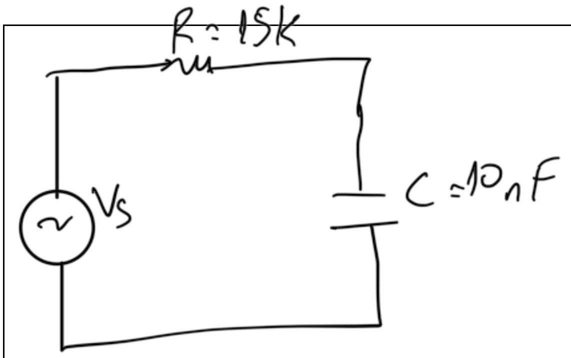


4) Set up the given circuit on a breadboard. Apply a sinusoidal input. Adjust the amplitude to 10 Vpp and frequency to 2 kHz. Measure both the input and the output voltage (Vc). Measure the phase difference.

vi. Measurement Outputs from the Oscilloscope



vii. Calculate the Phase Difference (btw input&output)



Using voltage divider rules

$$V_c = \frac{-j \cdot X_C}{R \cdot -j \cdot X_C}$$

$$= \frac{-j7,961.78}{15,000 \times -j7,961.78} \times 10\angle 0^\circ$$

$$= \frac{7,961.78 \angle -90^\circ}{16,982.04 \angle -27.96} \times 10\angle 0^\circ$$

$$= 4.69 \angle -62.04^\circ$$

$V_S = 10\cos w_0 t$

$\bar{V}_S = 10\angle 0^\circ$

$$Z_C = \frac{1}{jwC}$$

$$= \frac{1}{j \times 2\pi f \cdot C}$$

$$= \frac{1}{j \cdot 2\pi \cdot 2 \cdot 10^3 \cdot 10 \cdot 10^{-9}}$$

$$= -j7,961.78 = -jX_C$$

Rms Values;

$$V_{srms} = \frac{V_m}{\sqrt{2}} = \frac{10}{\sqrt{2}} = 7.07V$$

$$V_{crms} = \frac{4.69}{\sqrt{2}} = 3.316V$$

The phase difference is 62.04°