KEMENTERIAN PENGAJIAN TINGGI

ELECTRONIC CIRCUITS WORKBOOK USING MULTISIM ONLINE SIMULATOR

STUDENT GUIDE EDITION

AUTHOR

Anira Binti Abdul Rashid
Siti Mariam Binti Hussain
Gauri A/p Birasamy

ELECTRICAL ENGINEERING DEPARTMENT

ELECTRONIC CIRCUITS WORKBOOK USING MULTISIM ONLINE SIMULATOR

STUDENT GUIDE EDITION

AUTHOR

Anira Binti Abdul Rashid
Siti Mariam Binti Hussain
Gauri A/p Birasamy

ISBN: 9789670855769

Politeknik Tuanku Sultanah Bahiyah (PTSB)

Kulim Hi-Tech Park,

09090 Kulim, Kedah.

https://ptsb.mypolycc.edu.my

Thanks to our family and colleagues at Electrical Engineering Department, Politeknik Tuanku Sultanah Bahiyah for being the support team to complete this e-book. Last but not least we also thank you to the "PTSB Team E- Book" and technical team for guiding us in completing this e-book.

PREFACE

Electronic Circuits Workbook using multisim online simulator is our special present for students who are studying Electronic Circuits course as well as all the lecturers involve. It provides guidance for students to do their practical lab by using multisim online simulator. There are 6 chapters covered in this e-book which is Bridge Rectifier Circuit, Oscillator, Operational Amplifier, Astable Multivibrator, Filter and Analog to Digital or Digital to Analog Converter. In this e- book will present valuable information with supporting diagrams to help students more understanding in doing their practical laboratory. Finally, we hope this book will benefit all those seeking knowledge on this subject and will help in understanding electronic circuits better.

TABLE OF CONTENTS

INTRODUCTION 1

UNIT 1 : BRIDGE RECTIFIER CIRCUIT 2
1.1 THEORY 2
1.2 PRACTICAL OF BRIDGE RECTIFIER 6
1.3 QUESTION 8

UNIT 2 : COLPITTS OSCILLATOR CIRCUIT 9
2.1 THEORY 9
2.2 PRACTICAL OF COLPITS OSCILLATOR CIRCUITS 11
2.3 QUESTION 14

UNIT 3 : OPERATIONAL AMPLIFIER 15
3.1 THEORY 15
3.2 PRACTICAL OF NON-INVERTING AMPLIFIER 16
3.3 QUESTION 22

UNIT 4 : ASTABLE MULTIVIBRATOR 23
4.1 THEORY 23
4.2 PRACTICAL OF ASTABLE MULTIVIBRATOR USING TIMER (555) IC 26
4.3 QUESTION 30

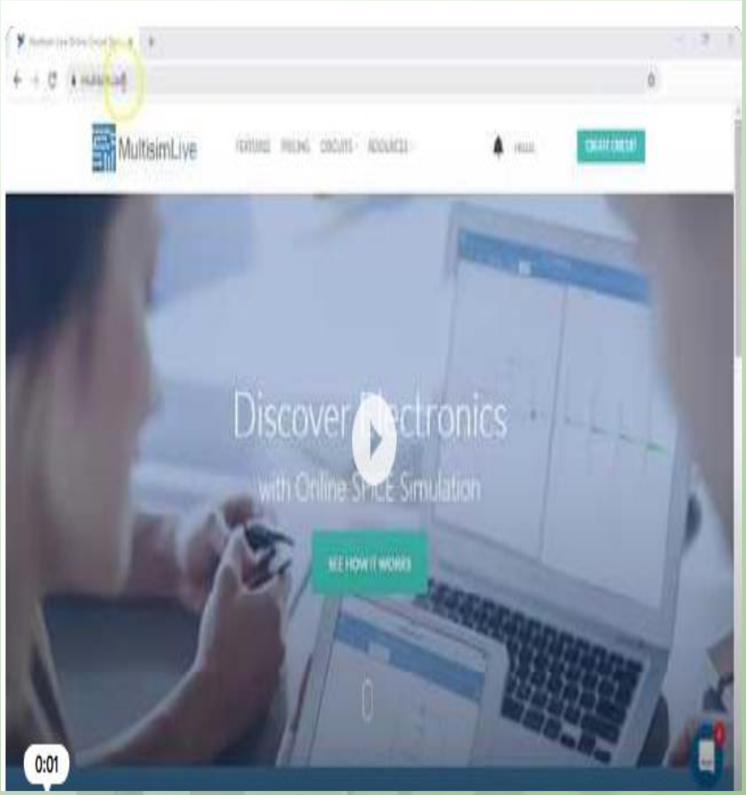
UNIT 5 : LOW PASS FILTER 31
5.1 THEORY 31
5.2 PRACTICAL LABORATORY 33
5.3 QUESTION 36
5.4 EXERCISES 36

UNIT 6 : DIGITAL TO ANALOG CONVERTER 37
6.1 THEORY 37
6.2 PRACTICAL LABORATORY 38
6.3 QUESTION 41

ANSWERS 43
REFERENCES 60

INTRODUCTION

NI Multisim (formerly MultiSIM) is an electronic schematic capture and simulation program which is part of a suite of circuit design programs, along with NI Ultiboard. Multisim is widely used in academia and industry for circuits education, electronic schematic design and SPICE simulation.



Multisim™ software provides SPICE simulation, analysis, and printed circuit board (PCB) tools to help you quickly iterate through designs and improve prototype performance. Move from schematic to layout seamlessly to save time and reduce prototype iterations.

https://www.instructables.com/How-to-Use-Multisim-Live/

1.1 INTRODUCTION

Direct Current (DC) power supply units in electronic components will be used to convert Alternating Current (AC) voltage to DC voltage. **Figure 1.1** show the block diagram and correspond output from each power supply block. The character of the output voltage from power supply is almost same as DC voltage supplied by a battery.

BLOCK DIAGRAM AND FUNCTION EACH UNIT

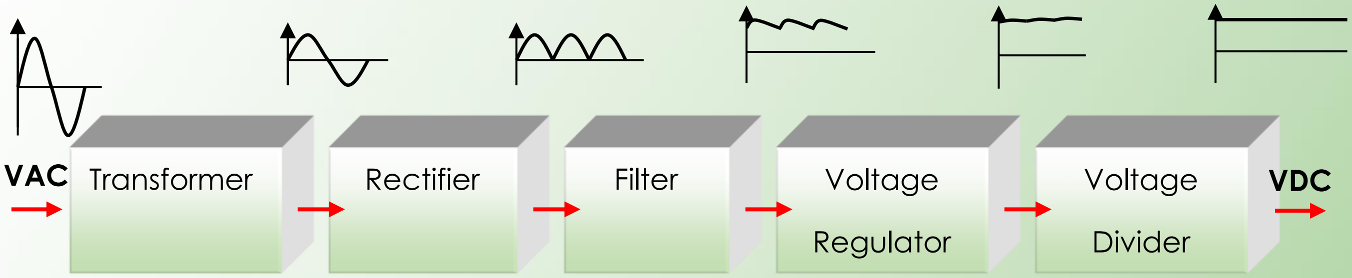
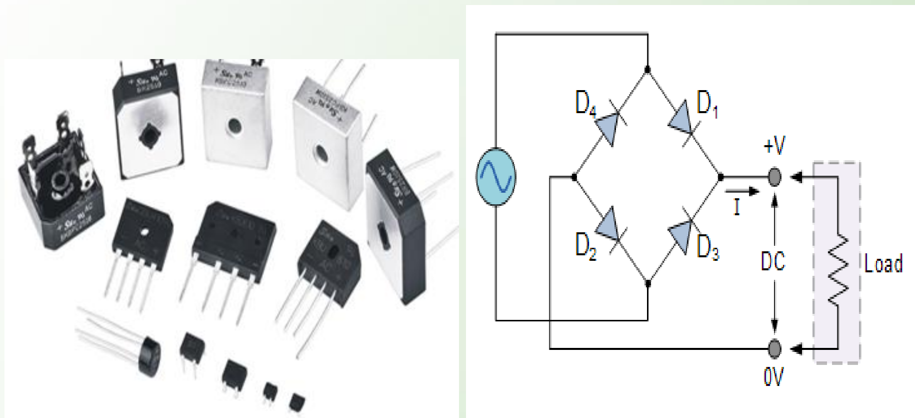


Figure 1.1: Block diagram of Power supply and output waveform for each block

- Transformer - Reduce the value of the input ac voltage.
- **Rectifier** - A circuit that converts ac voltage to pulsating dc voltage.
- **Filter** - The main task of filter is to convert the voltage at pulse to the voltage rippled.
- **Voltage Regulator** - To stabilize the pure DC voltage (at the output of the filter).
- **Voltage Divider** - Circuit is needed in power supplies for electric equipment requires.

Rectifier is the main circuit converting the AC voltage to DC voltage. It is divide into three types. Half-wave rectifier, Full-wave rectifier and Bridge rectifier. Bridge rectifier is consisting of four individual diode connected in a closed loop and produce full wave DC voltage provides the same polarity of output voltage for either polarity of input voltage.



(a) (b)

Figure 1.2: (a)Component and (b) Circuit of Bridge rectifier

Typical applications Bridge Rectifier are:

i) Welding- to supply polarized voltage for welding. In such circuits control of the output current is required and this is sometimes achieved by replacing some of the diodes in bridge rectifier with thyristor, whose voltage output can be regulated by means of phase fired controllers.

ii) Generator

iii) Battery charger

iv) AC motor drive

v) Traction markets

vi) Detection of amplitude modulated radio signals

1.1.1 Operation of Bridge Rectifier

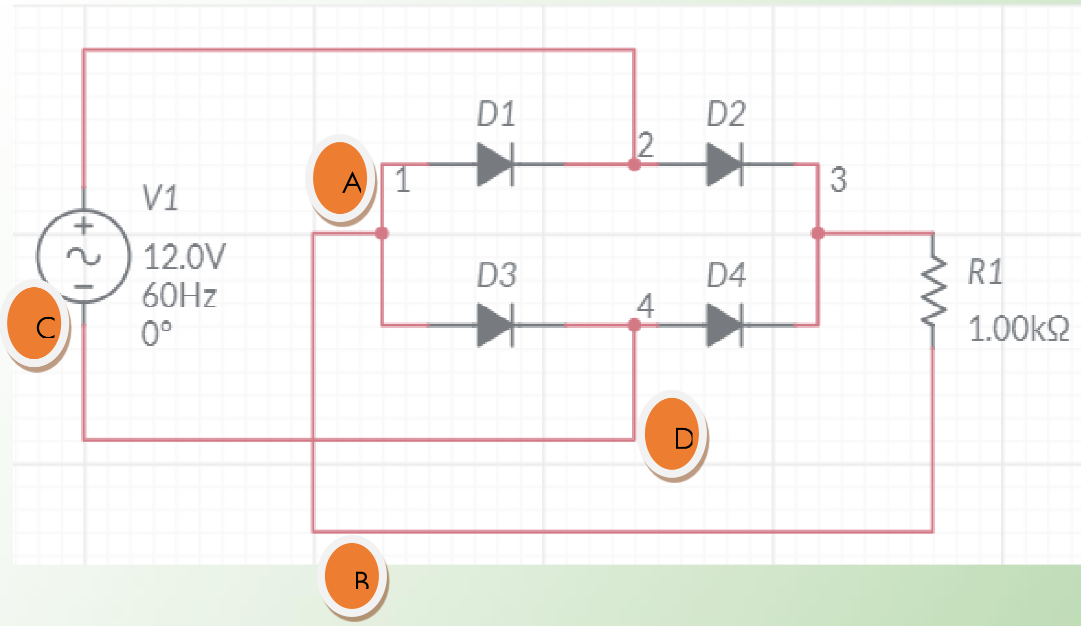
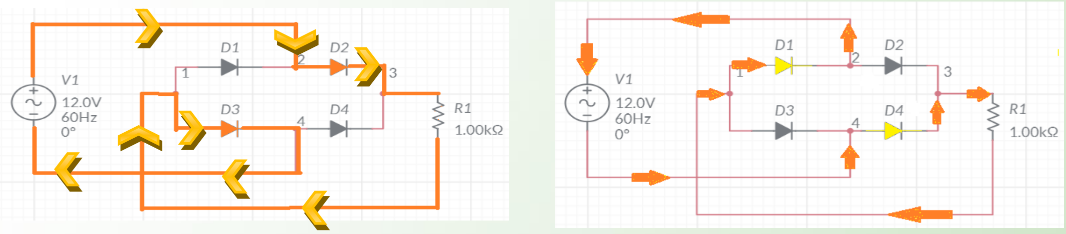


Figure 1.3: Circuit of Bridge rectifier

The four diodes are connected in close loop as figure 1.3 to perform as bridge rectifier in power supply unit. The input AC voltage is supply at terminal A and B and the output measure across terminal C and D. While, the four diodes are label as D1, D2, D3 and D4 connected in "series pairs". The operation of bridge rectifier separates in these two cycles, that is positive cycle and negative cycle.

During the positive of the input voltage, terminal A is positive cycle and terminal B is negative cycle. Diodes D2 and D3 are forward biased while, D1 and D4 are reverse biased and the current flow through load R1. Figure 1.4 show the flow of currents in bridge rectifier.



(a) (b)

Figure 1.4: Flow of the currents (a) Positive half cycle and (b) Negative half cycle

When the negative of the input voltage, terminal B is positive cycle and terminal A is negative cycle. Diodes D1 and D4 are forward biased while, D2 and D3 are reverse biased and the current flow through load R1. Since the direction current flow through R_1 is similar to the current flow through the positive cycles, so similar wave will produce.

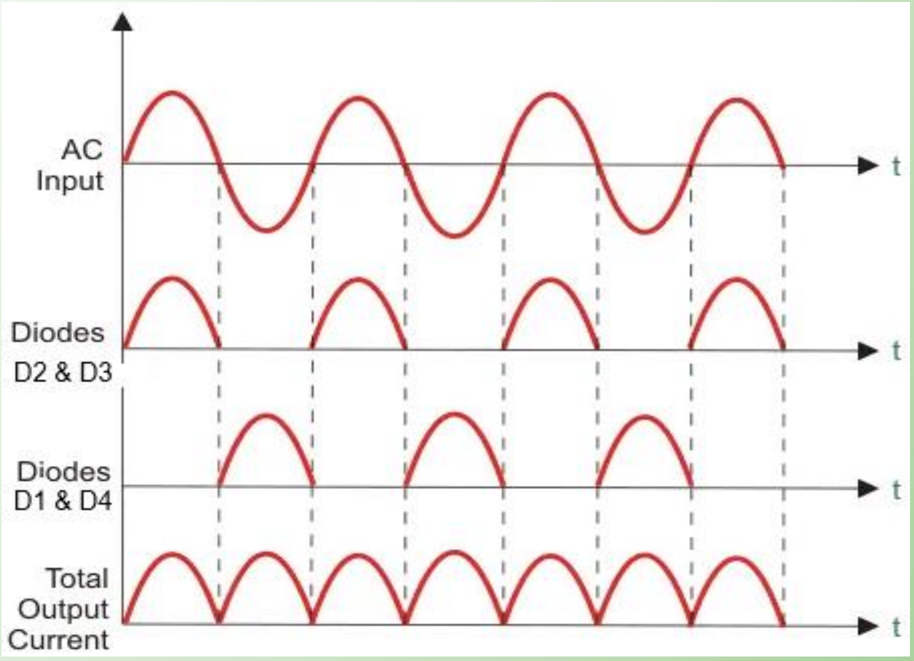


Figure 1.5: Output at R1

1.2 PRACTICAL OF BRIDGE RECTIFIER

OBJECTIVES

i. Able to construct bridge rectifier.

ii. Measure the output of a bridge rectifier

PROCEDURE :

1. Sketch the circuit diagram bridge rectifier figure 1.1 by using Multisim Online Simulator. Refer https://www.multisim.com/
2. Adjust the input voltage 12V amplitude and the frequency at 60Hz.
3. Simulate the system by using the run button to achieved the output waveforms.
4. Measure of Vout across R1.

RESULTS:

The output from practical show as in figure 1.6.

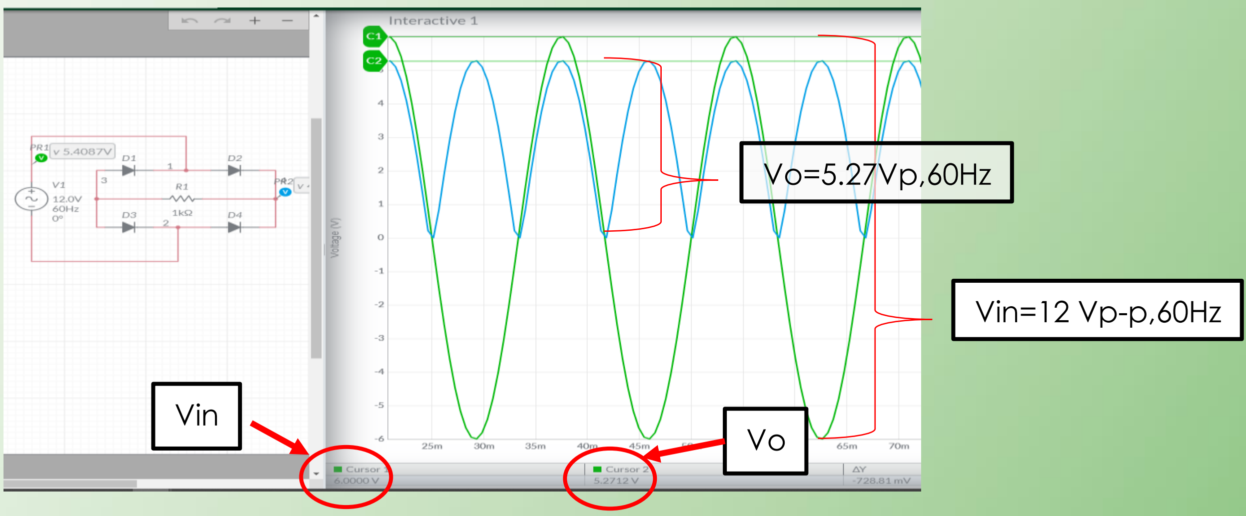


Figure 1.6: Output at R1

***Hint: There is a different between Vin and Vo cause by the types of diode use. Since current across the diode at one cycle and voltage drop at diode is 0.7V (assumed 2 germanium diode), voltage drop is $V_o = V_{in} - 0.7V$**

1.3 QUESTION

1. Determine the output(Vo) across R_L in Figure 1.7.

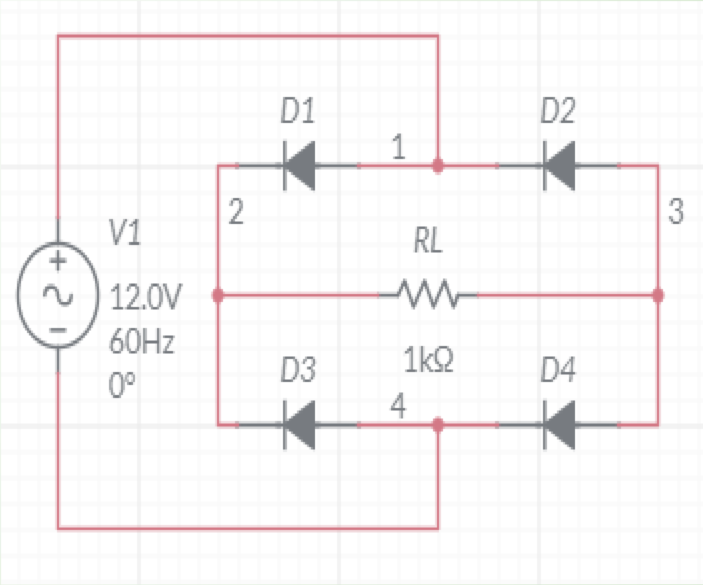


Figure 1.7: Output at RL

2. If the diode at figure 1.7 change from germanium diode to silicon diode, calculate the Vo across RL.
3. Measure the output(Vo) across R_L and calculate the Vo in Figure 1.8.

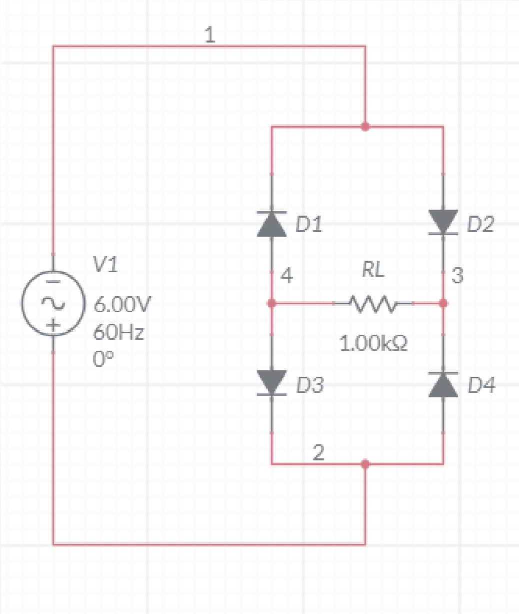


Figure 1.8: Output at RL

4. A fullwave bridge rectifier has input voltage 20 Vp-p 50 Hz. Given transformation ratio is 2:1. Assume there is no voltage drop at diode, calculate: -

i) Output voltage signal

ii) Output frequency signal

2.1 THEORY

Oscillator is the source of basic signal generator for multi-application in the electronic circuit. It will change the DC voltage into AC signal and be able to generate any frequency value needs by the circuit.

All basic oscillator circuit consist 3 parts like amplifier, feedback circuit and frequency generator circuit

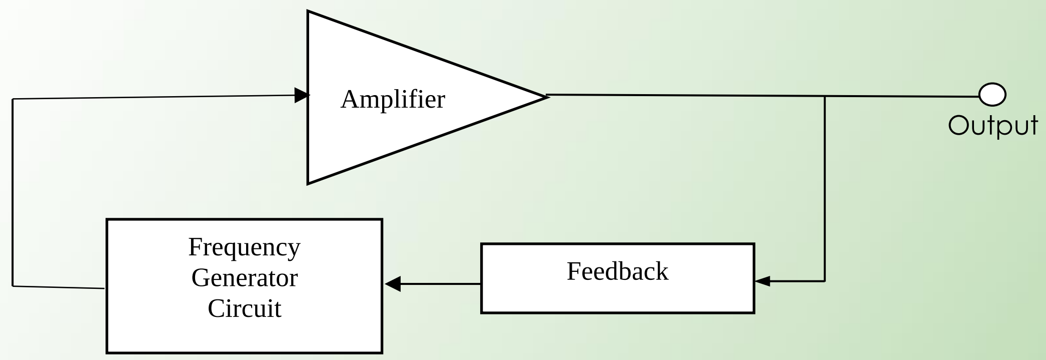


Figure 2.1: Block diagram of an Oscillator circuit

In the Colpitts circuit, two capacitors and one inductor determine the frequency of oscillation. The feedback needed for oscillation is taken from a voltage divider made by the two capacitors. The Colpitts Oscillator has fairly good frequency stability, is easy to tune, and can be used for a wide range of frequencies. The large value of split capacitance is in parallel with the junctions and minimizes the effect on frequency stability.

The frequency of oscillations for a Colpitts Oscillator is determined by the resonant frequency of the LC tank circuit and is given as:

$$f = \frac{1}{2\pi\sqrt{L C_T}}$$

where C_T is the capacitance of C1 and C2 connected in series and is given as:

$$\frac{1}{C_T} = \frac{1}{C_1} + \frac{1}{C_2} \quad \text{or} \quad C_T = \frac{C_1 \times C_2}{C_1 + C_2}$$

Advantages of Colpitts oscillator

Main advantage of Colpitts oscillator over Hartley oscilator is the better performance in the high frequency state. This is because the capacitors offer a low reactance path for the high frequency signals and thus the output signals in the high frequency domain will be more sinusoidal. Due to the excellent performance in the high state, the Colpitts oscillator can be even used in microwave uses.

2.2 PRACTICAL OF COLPITS OSCILLATOR CIRCUIT

OBJECTIVES

i) Able to construct oscillator circuit.

ii) Able to determine the oscillation frequency by using the formula and the effect of varying the values of the L and C to the oscillation frequency.

iii) Sketch the output waveform of varying the values of the L and C to the oscillation frequency.

PROCEDURE

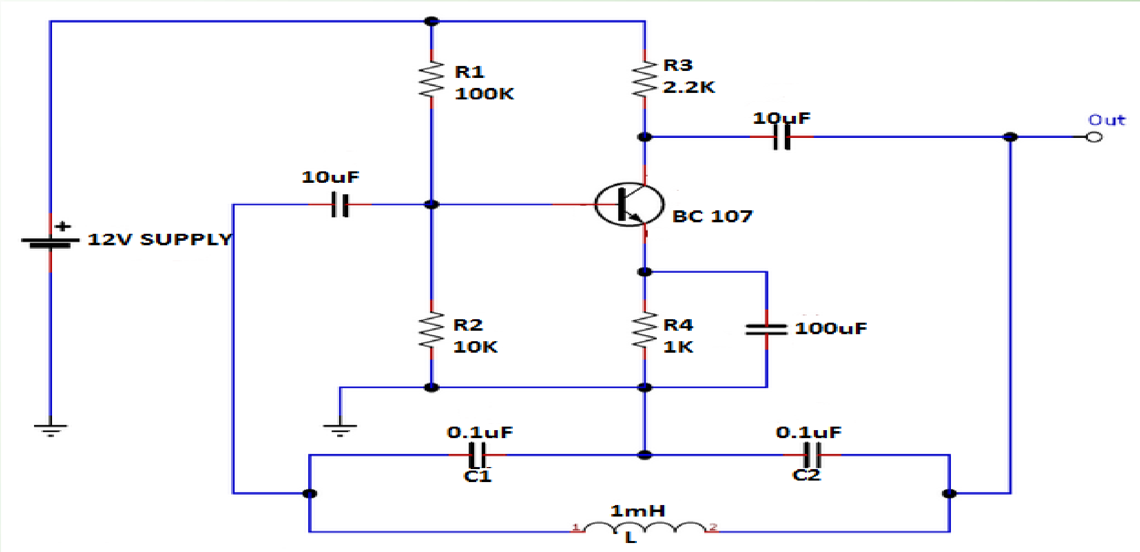
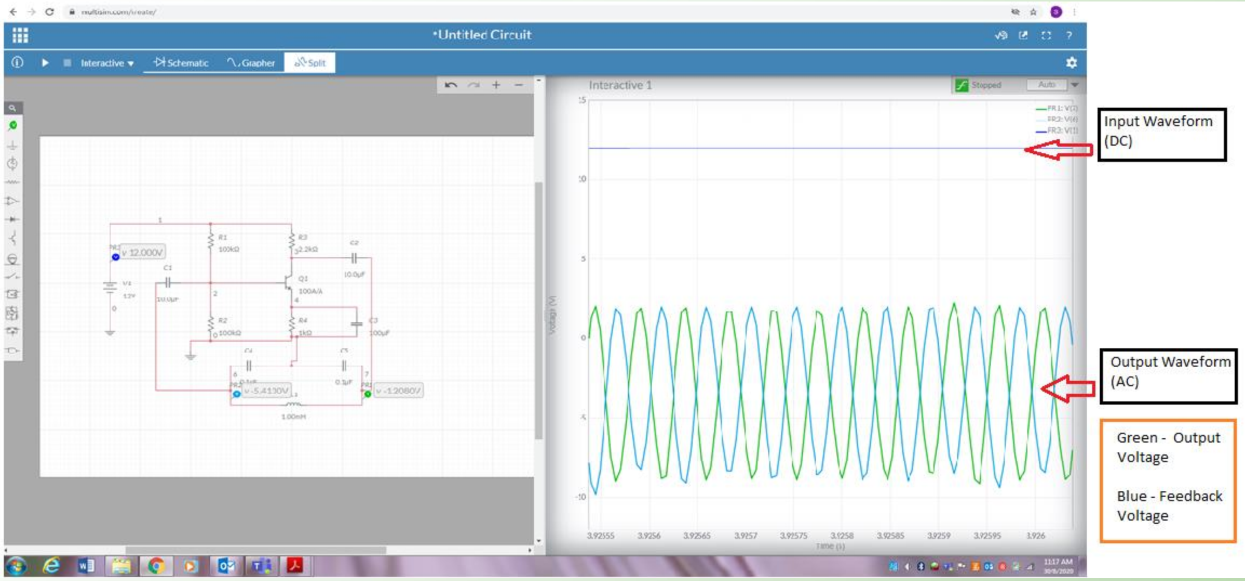


Figure 2.2: Schematic of Colpitts Oscillator Circuit

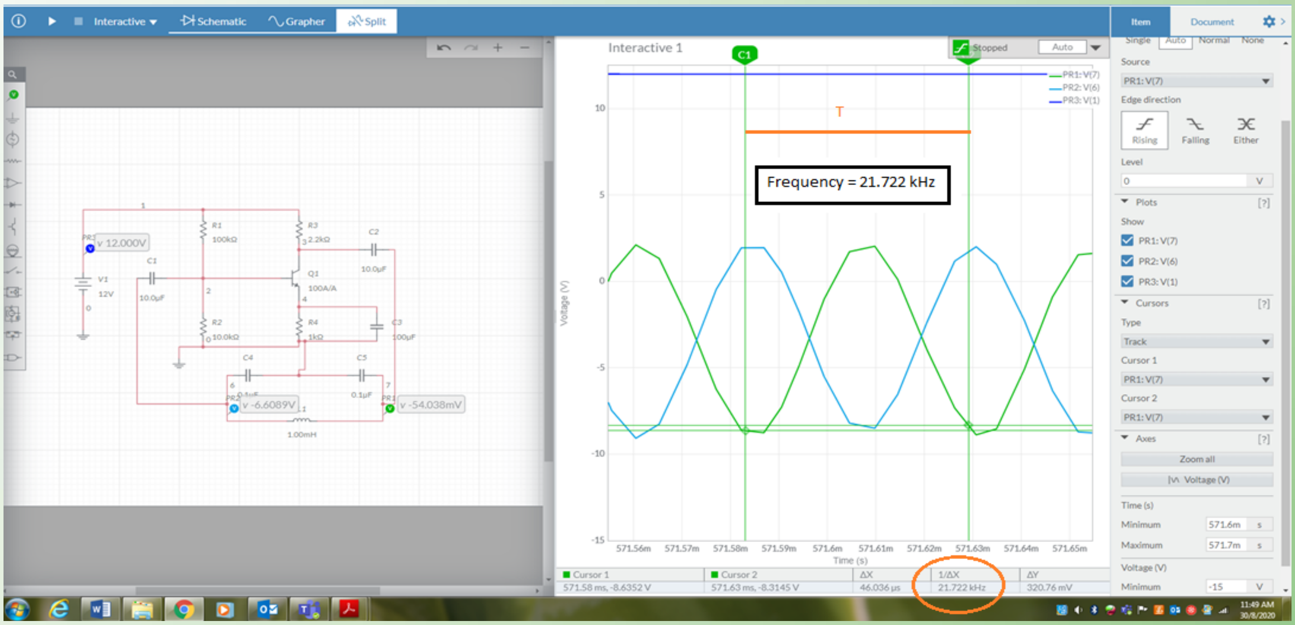
1. Sketch the circuit diagram of **figure 2.2** by using Multisim Online Simulator. Refer https://www.multisim.com/
2. Simulate, draw and measure the output waveform to get output frequency. Compare that value with the calculation value.
3. Change the value of L, and repeat procedure number 2.

OUTPUT

1. Input Waveform & Output Waveform



2. L = 1mH



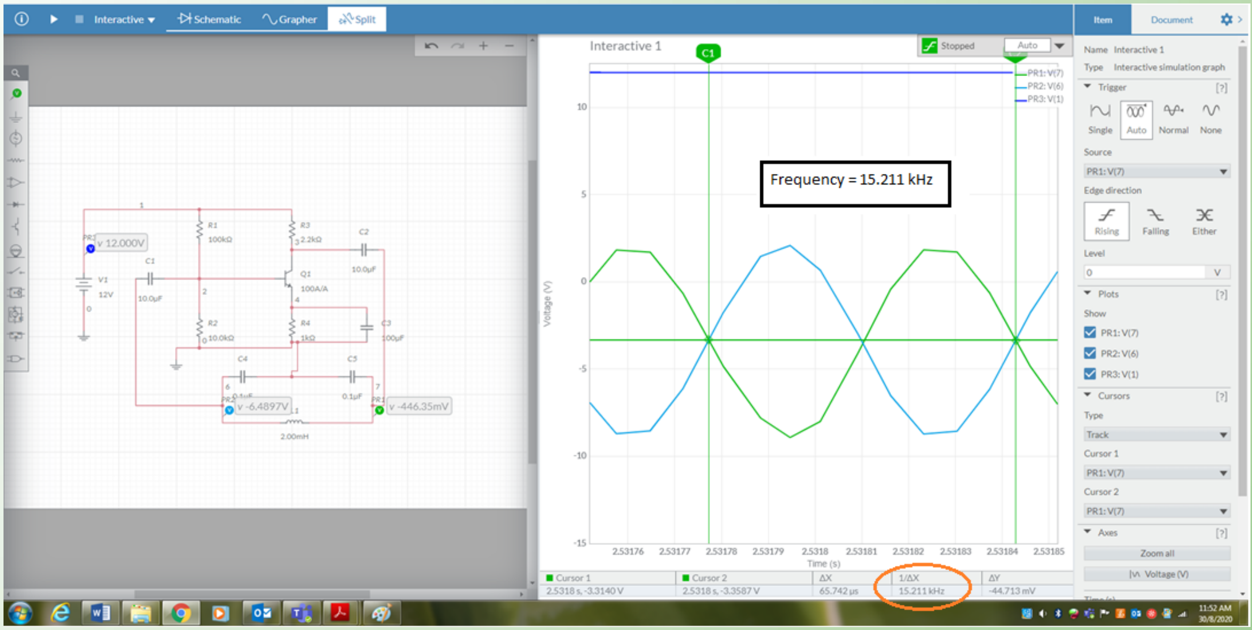
Calculation of Output Frequency using formula :

$$CT = \frac{C1XC2}{C1 + C2}$$

$$CT = \frac{0.1\mu X 0.1\mu}{0.1\mu + 0.1\mu} = 50nF$$

$$fr = \frac{1}{2\pi\sqrt{1m \; x \; 50nF}} = \textbf{22.5 kHz}$$

3. L = 2mH



Calculation of Output Frequency using formula :

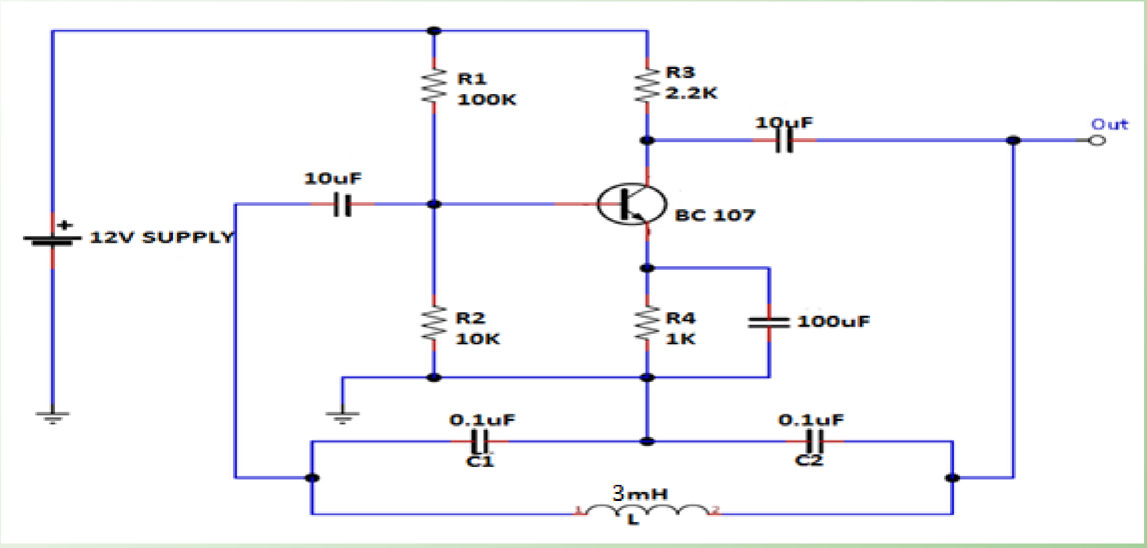
$$CT = \frac{C1XC2}{C1 + C2}$$

$$CT = \frac{0.1\mu X 0.1\mu}{0.1\mu + 0.1\mu} = 50nF$$

$$fr = \frac{1}{2\pi\sqrt{2m\ x\ 50nF}} = \mathbf{15.92\ kHz}$$

2.3 QUESTION

a. Sketch the circuit diagram below by using Multisim Online Simulator. Set value L=3mH. Simulate, draw and measure the output waveform to get the value of output frequency.



b. Compare that frequency value with the calculation value.

OPERATIONAL AMPLIFIER

3.1 THEORY

An operational amplifier is an integrated circuit that can amplify electric signals, high gain and direct coupled differential linear amplifier. **Figure 3.1** shows circuit symbol and pin configuration of an OP-AMP.

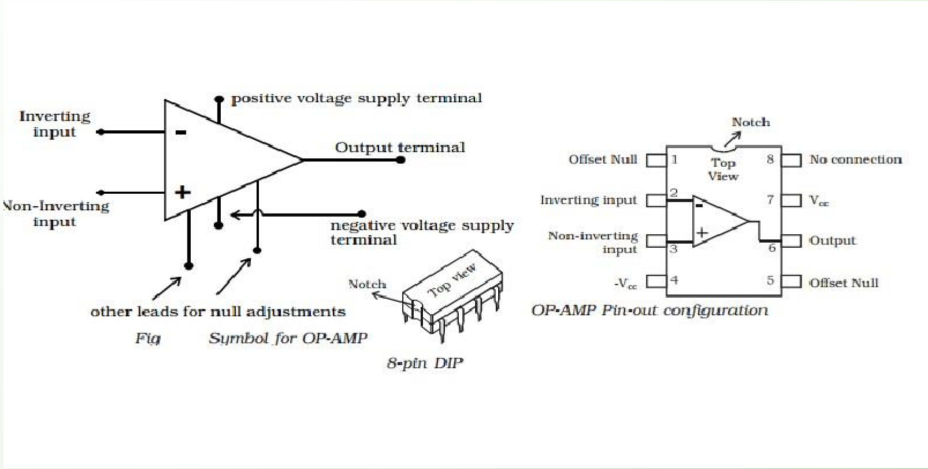


Figure 3.1: Circuit symbol and Pin configuration of an OP-AMP

Op-amp can be used to form a difference circuit configuration for various type of application. The most common op-amp circuits are inverting amplifier, non-inverting amplifier, integrator and differentiator amplifier. **Figure 3.2** represent the respective circuit configuration of Non-Inverting Amplifier.

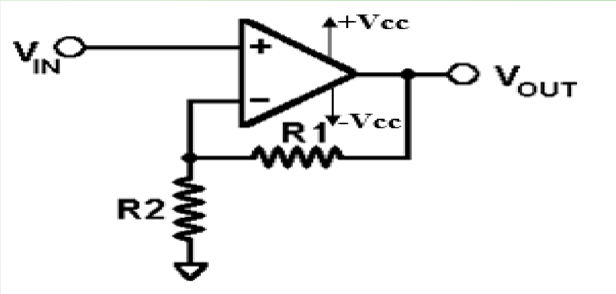


Figure 3.2: Non-Inverting Amplifier

Close Loop Gain, $Acl = 1 + \frac{Rf}{Ri}$

Phase Angle, $\theta = \frac{t}{T} X\ 360°$

The **noninverting amplifier** has much higher circuit input impedance. It does not produce an 180° voltage phase shift from input to output; i.e., the input and output voltages are in phase. A noninverting amplifier is shown in figure below

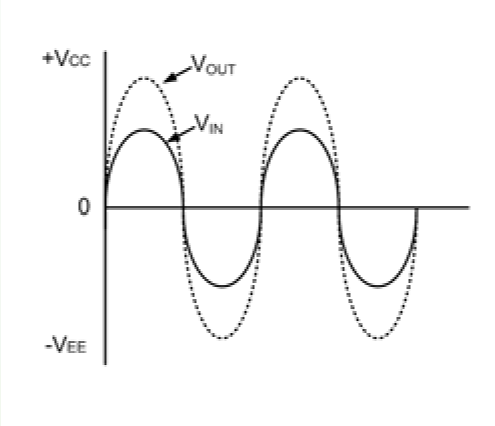


Figure 3.3: Non-Inverting Amplifier

3.2 PRACTICAL OF NON-INVERTING AMPLIFIER

OBJECTIVES

i. Able to construct op-amp circuit for Non-inverting amplifier.
ii. Determine the voltage gain (Av) or gain of closed loop (Acl).
iii. Determine the phase angle for the output waveform.

PROCEDURE

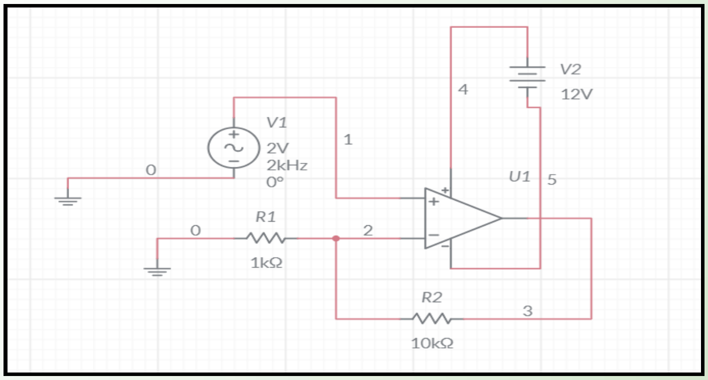
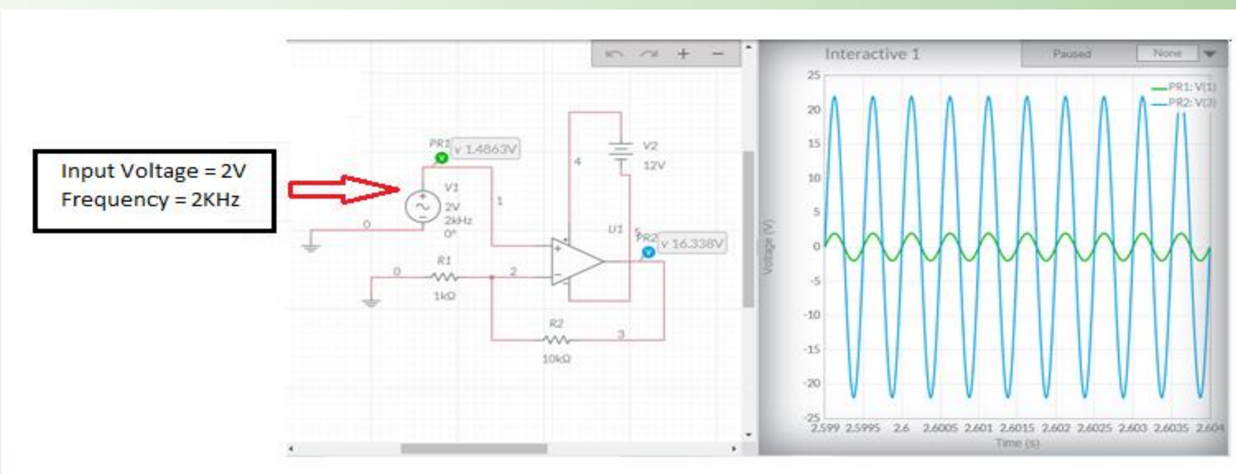


Figure 3.4 : Schematic of Non-Inverting Op-Amp

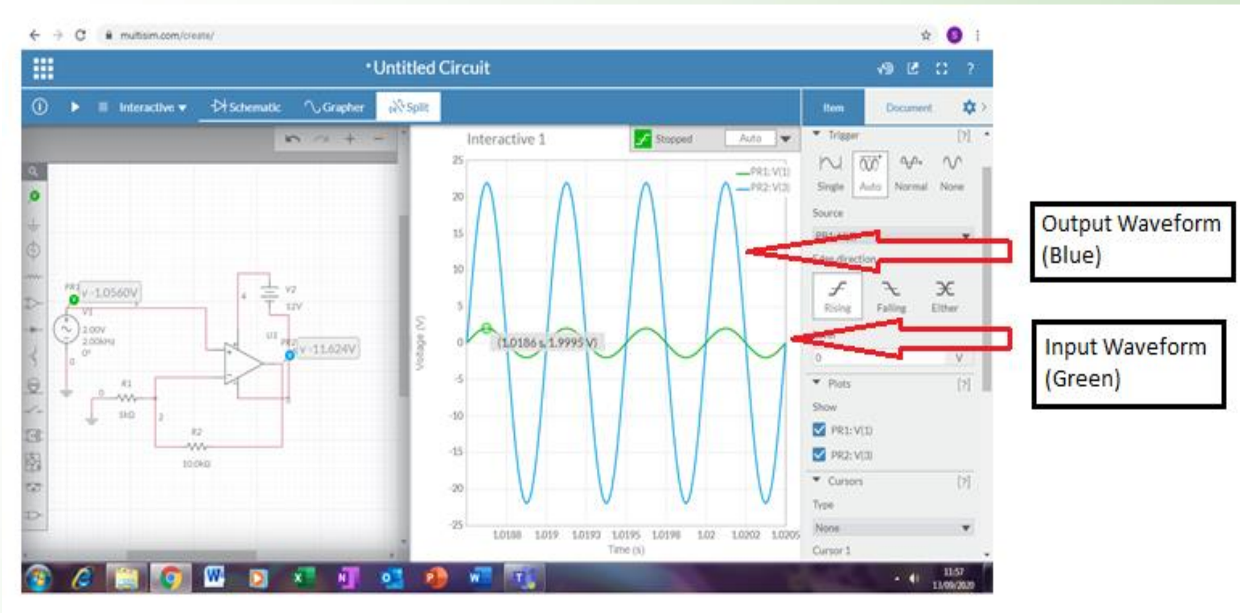
1. Sketch the circuit diagram of **figure 3.3** by using Multisim Online Simulator. Refer https://www.multisim.com/
2. Adjust the input voltage 2V amplitude and the frequency at 2kHz.
3. Simulate the system by using the run button to achieved the output waveforms.



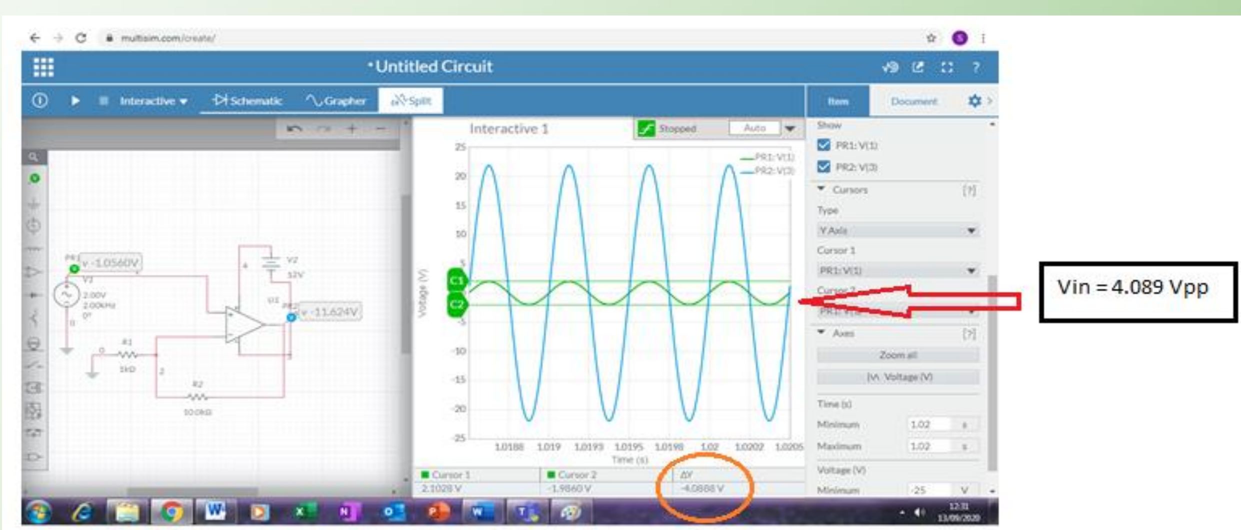
4. Using the button calculation & info, Output Voltage (Vo) and Closed Loop Gain (Acl /Av) can be calculate.

OUTPUT

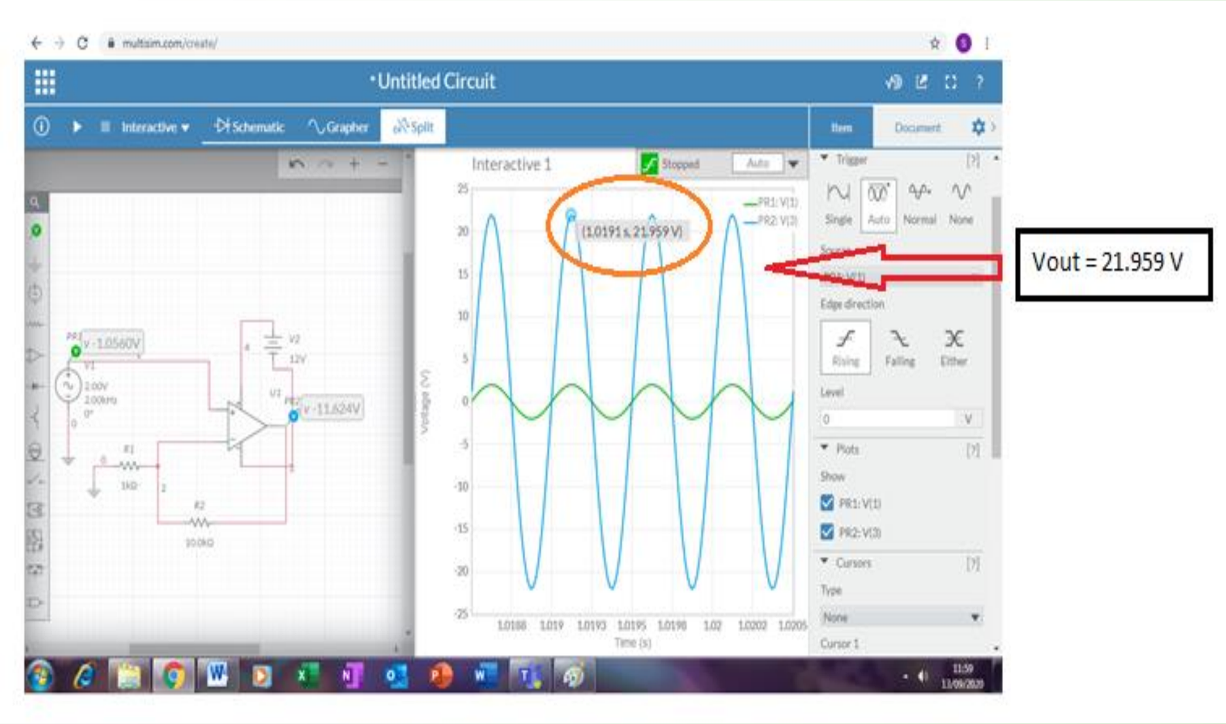
1. Input Waveform & Output Waveform



2. Vin = Vpp



3. Vout



Calculation of Output Voltage, Vout

$$Vo = 1 + \left[\frac{Rf}{Ri}\right] Vi$$

$$Vo = 1 + \left[\frac{10k}{1k}\right] 2$$

Vo = 21 V

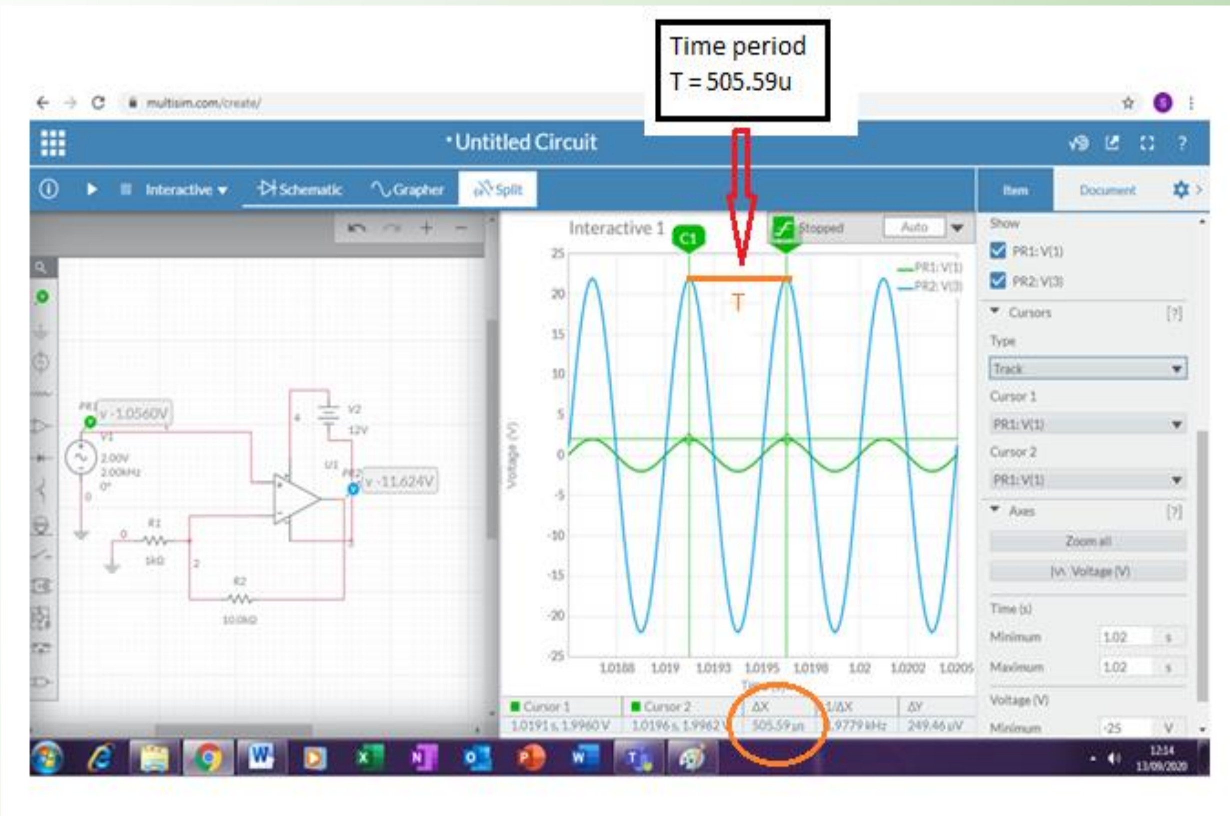
Calculation of Gain, Acl

$$Acl = \frac{Vo}{Vi}$$

$$Acl = \frac{21.959\ V}{1.9995\ V}$$

$$\boldsymbol{Acl = 10.982}$$

4. Measure of T to calculate phase angle value



Phase Angle, Ɵ

t = difference time between input and output

T = time period

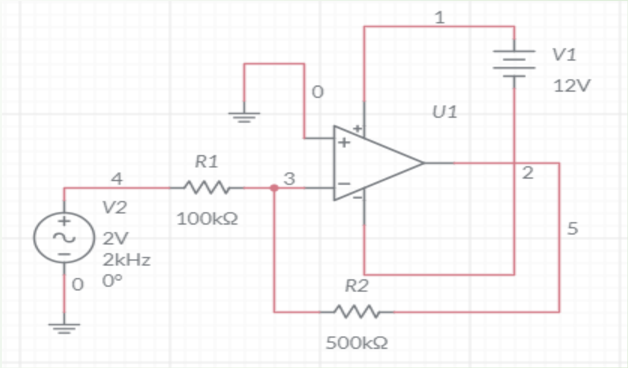
$$\theta = \frac{t}{T} \, X \, 360°$$

$$\theta = \frac{0}{505.59\mu} \, X \, 360°$$

$\theta = 0°$ **(Vo and Vi is inphase)**

3.3 QUESTION

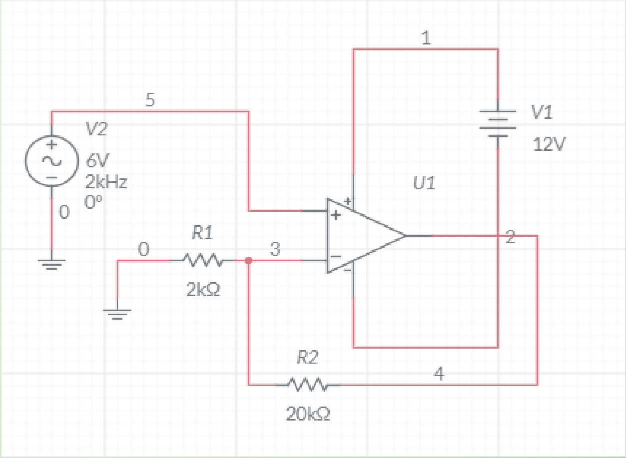
1.



a. Sketch the circuit diagram above by using Multisim Online Simulator.

b. Simulate to get the Output Voltage, Input Waveform and Output Waveform.

2.



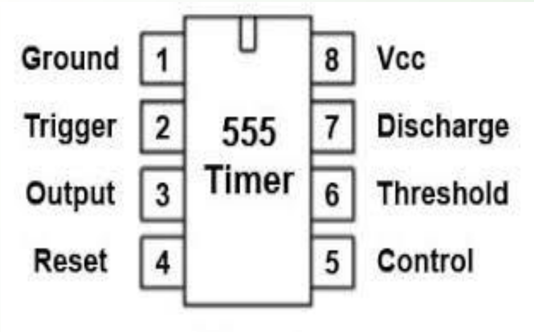
a. Sketch the circuit diagram above by using Multisim Online Simulator.

b. Simulate to get the Output Voltage, Input Waveform and Output Waveform.

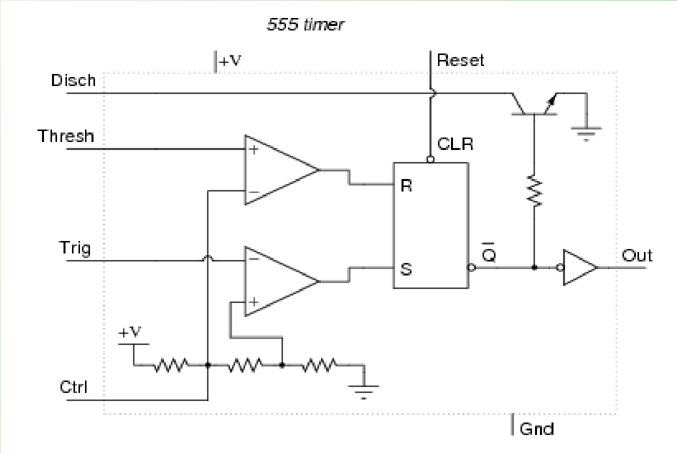
ASTABLE MULTIVIBRATOR

4.1 INTRODUCTION

The 555 timer IC is an integrated circuit (chip) used in a variety of timer, pulse generation and oscillator applications as show in figure 4.1.



(a)



(b)

Figure 4.1: (a) Timer pinout and (b)Block diagram of TIMER 555 with pins

The 555 Timer Integrated Circuit (IC) can be designed either in its Monostable mode thus generating a precision timer of a fixed time duration, or connect as Bistable mode to produce a flip-flop type switching action. Other than that, it also can connect as Astable mode to produce a stable Oscillator circuit for generating highly accurate free running waveforms whose output frequency can be adjusted by means of an externally connected RC tank circuit consisting of just two resistors and a capacitor.

In Astable Multivibrator, both output states are unstable state and output continuously changes between the two states. The main application of this astable multivibrator is in the design of relaxation oscillator.

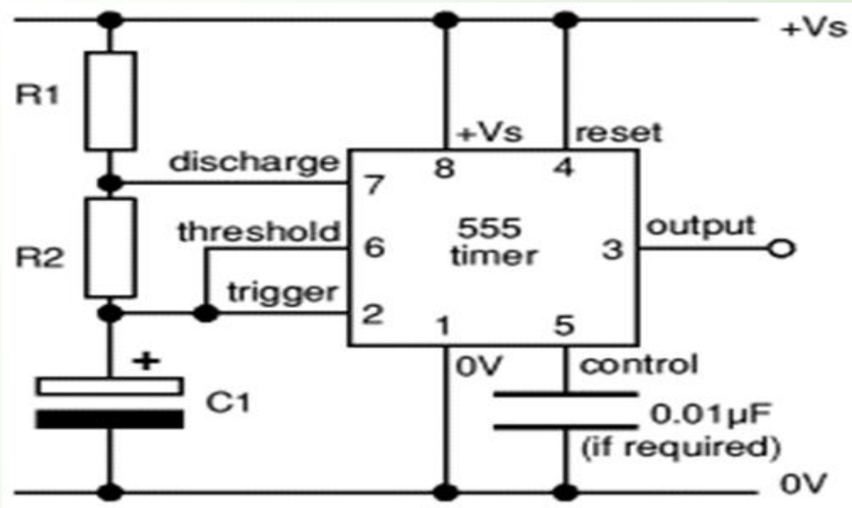


Figure 4.2: Astable using Timer 555 circuit

4.1.1 Operation of Astable using Timer 555

With the output high (+Vs) the capacitor C1 is charged by current flowing through R1 and R2. The threshold and trigger inputs monitor the capacitor voltage and when it reaches $^2/_3$Vs (threshold voltage) the output becomes low and the discharge pin is connected to 0V. The capacitor now discharges with current flowing through R2 into the discharge pin. When the voltage falls to $^1/_3$Vs (trigger voltage) the output becomes high again and the discharge pin is disconnected, allowing the capacitor to start charging again.

This cycle repeats continuously unless the reset input is connected to 0V which forces the output low while reset is 0V. An astable can be used to provide the **clock signal** for circuits such as counters.

4.1.2 Calculation of Output Voltage Frequency

An astable circuit produces a 'square wave', this is a digital waveform with sharp transitions between low (0V) and high (+Vs). Note that the durations of the low and high states may be different. The circuit is called an astable because it is not stable in any state: the output is continually changing between 'low'(TL) and 'high'(TH).

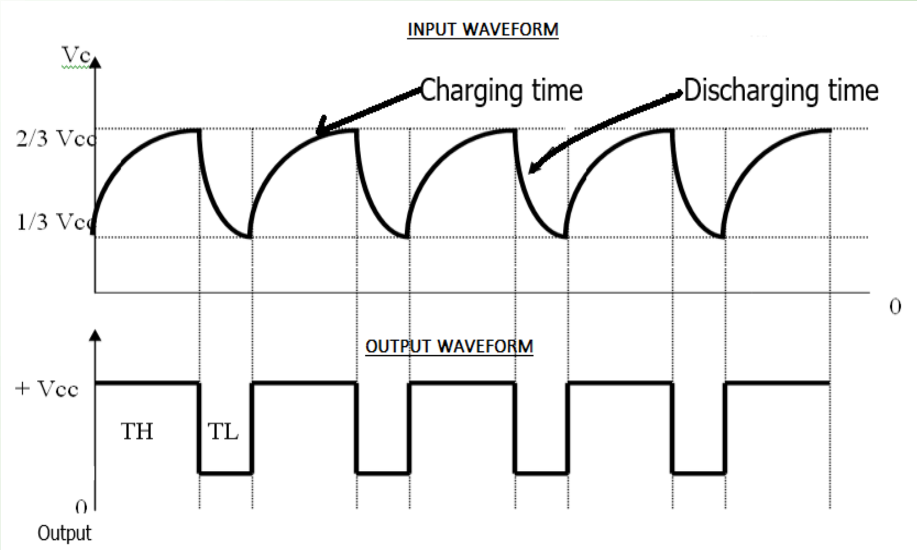


Figure 4.3: Output waveform of Astable Multivibrator

i. The time period (T) of the square wave is the time for one complete cycle:

$$T = 0.693\ X\ (R1 + 2R2) X\ C1$$

ii. Frequency (f) which is the number of cycles per second :

$$f = \frac{1.44}{(R1 + 2R2) \times C1}$$

iii. The time period can be split into two parts:

$$T = TH + TL$$

iv. Mark time (output high):

$$TH = 0.693 \times (R1 + R2) \times C1$$

v. Space time (output low):

$$TL = 0.693 \times R2 \times C1$$

vi. Percentage of duty cycle :

$$\%\ Duty\ Cycle = \frac{R_1 + R_2}{R1 + 2R_2} \times 100$$

Hint: A 50% duty cycle means the high time is equal to the low time.

4.2 PRACTICAL OF ASTABLE MULTIVIBRATOR using TIMER (555) IC

OBJECTIVES

i. Able to construct of Astable 555 timer circuit
ii. Able to draw the waveform at the output (pin 3) and capacitor (pin 6).
iii. Able to measure the specification for Astable 555 Mode (frequency (f), duty cycle (D), Time High (TH), Time Low (TL) and period (T)

PROCEDURE :

1. Sketch the circuit diagram astable multivibrator figure 4.4 by using Multisim Online Simulator. The "555" integrated circuit is connected to a capacitor and two resistors as shown, it will oscillate freely produce output on and off with a square-wave output voltage. Refer https://www.multisim.com/

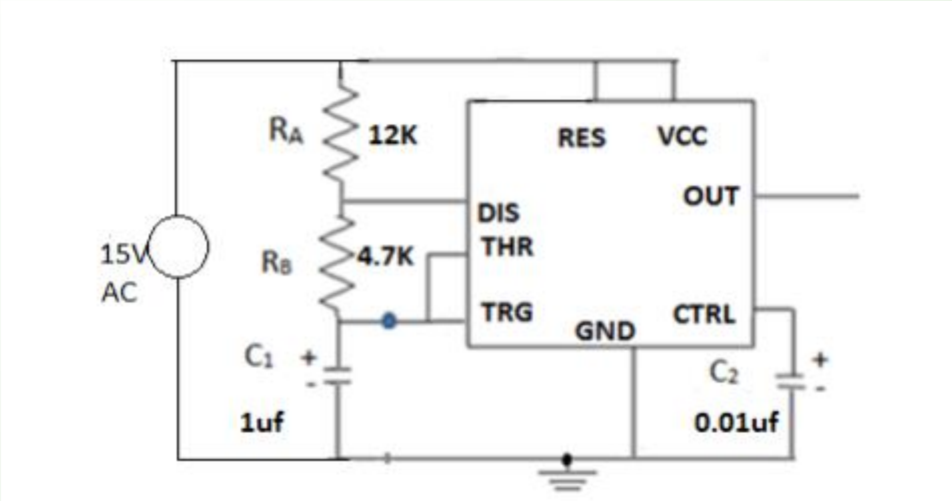


Figure 4.4: Astable Multivibrator

2. The input voltage is set to 15V amplitude, R1=12KΩ, R2=4.7KΩ, C1=1µf and C2=0.01µf.
3. Simulate the Astable 555 Timer by using the run button to achieved the output waveforms.
4. Using Formula Calculate the charge time (T_H), discharge time (T_L) and period(T). By using calculator & info calculate frequency (f) and percentage of duty cycle (%D) output waveform.
5. Sketch the output waveform pin 2 / pin 6 and pin 3.

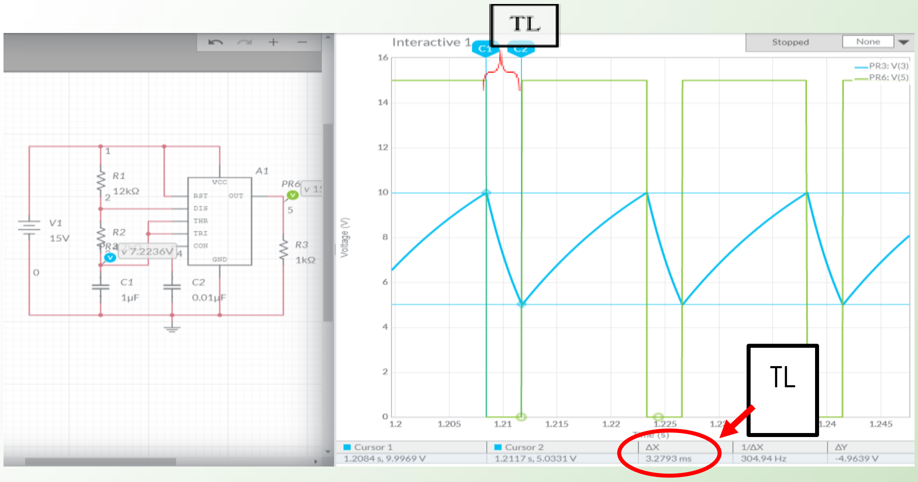
RESULTS:

The output from practical show as in figure 1.6.

i. Output wave form and Calculate TH

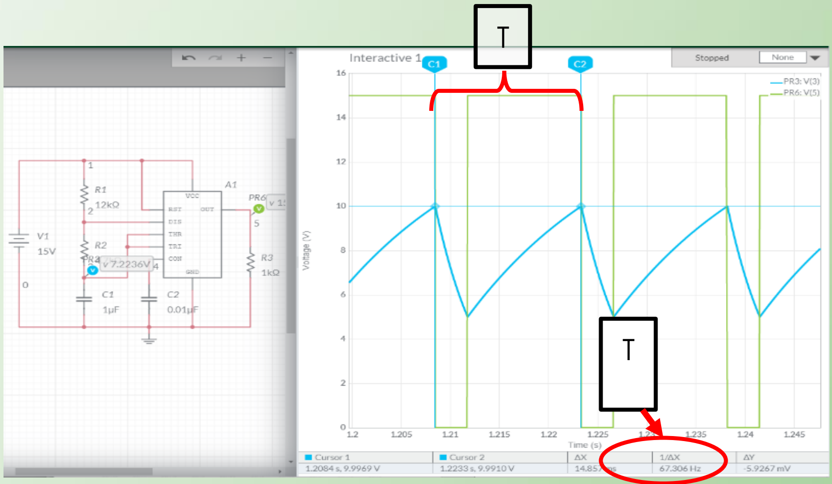
	Theory	Simulation
TH	TH = 0.693 x (R1+R2)xC1 TH = 0.693 x (12KΩ+4.7KΩ)x1µ TH = <u>0.0116s = 11.6ms</u>	TH = <u>11.591ms</u>

ii. Calculate TL



	Theory	Simulation
TL	TL = 0.693 x R2xC1 TL = 0.693 x 4.7KΩ x 1µ TL = 0.0032571s = 3.2571ms	TL = 3.2793ms

iii. Calculate T



	Theory	Simulation
T	T = TH + TL =(11.6+3.2571)ms = 14.8471ms	T = 14.857ms

iv. Calculate Duty cycle.

$$\%\ Duty\ Cycle = \frac{R_1 + R_2}{R1 + 2R_2} \times 100$$

$$\%DutyCycle = \frac{(12K\Omega + 4.7K\Omega)}{12K\Omega + (2x4.7K\Omega)} x100$$

$$\%DutyCycle = \frac{16700\Omega}{21400\Omega} x100 = 78\%$$

4.3 QUESTION

1. The basis for this circuit's timing as in figure 4.4, explain the function of resistor – capacitor(RC) network:
2. Determine the TH, TL, T for the astable multivibrator in figure 4.4, if the value of R1 is varying from 2KΩ, 10KΩ, 15KΩ and 56KΩ using multisim simulator.
3. Determine the TH, TL, T for the astable multivibrator in figure 4.4, if the value of R2 is varying from 1KΩ, 3.9KΩ, 5.6KΩ and 10KΩ using multisim simulator.
4. Determine the TH, TL, T for the astable multivibrator in figure 4.4, if the value of C1 is varying from 0.1µ, 0.2µ, 0.5µ and 10µ using multisim simulator.

5.1. THEORY

A low-pass filter passes low frequency signals, and rejects signals at frequencies above the filter's cutoff frequency. Simple RC low pass filters can be made by connecting a single resistor in series with a single capacitor. The capacitive reactance, (Xc) will be very large compared to the resistive value of the resistor, R. As a result the voltage across the capacitor, Vc will also be large while the voltage drop across the resistor, Vr will be much lower. At high frequencies the reverse is true with Vc being small and Vr being large. A filter in which the signal passes through an inductor, or in which a capacitor provides a path to ground, presents less attenuation to low-frequency signals than high-frequency signals.

The combination of resistance and capacitance gives the time constant of the filter, **τ= RC** (represented by the Greek letter). The break frequency, also called the turnover frequency or cutoff frequency (in hertz), is determined by the time constant:

$$\omega = \frac{1}{\tau} = \frac{1}{RC}$$

Or equivalently (in radians per second):

$$f_c = \frac{1}{2\pi\tau} = \frac{1}{2\pi RC}$$

Figure 5.1 and 5.2 show the Low Pass filter circuit and Frequency response curve respectively.

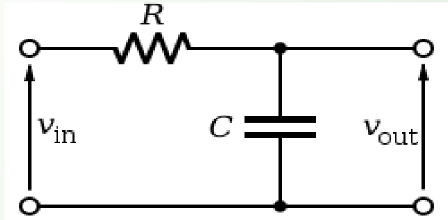


Figure 5. 1 Low Pass filter circuit

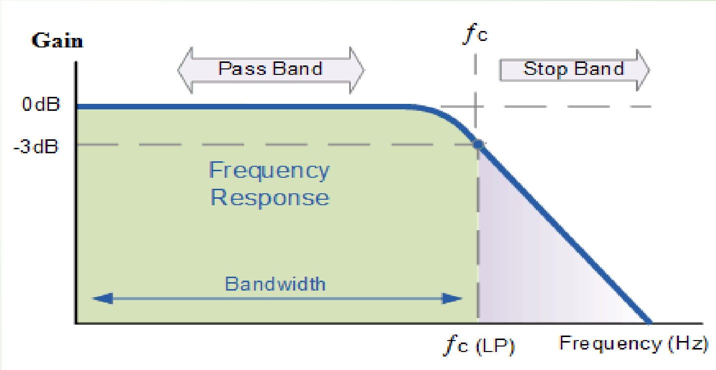


Figure 5.2 : Frequency response curve for low pass filter

The cut-off frequency or - 3dB point, can be found using the Equation 1.

Cutoff Frequency,

$$f_c = \frac{1}{2\pi RC} \qquad \text{.........................(1)}$$

Voltage gain is referred to the ratio of circuit output to input voltage as shown in Equation 2.

$$Av = \frac{Vout}{Vin} \qquad \text{.........................(2)}$$

The gain of the filter or any filter for that matter, is generally expressed in Decibels and is a function of the output value divided by its corresponding input value and is given as:

$$\text{Gain in decibels, } Av_{dB} = 20\log_{10}\frac{Vout}{Vin} \qquad \text{.........................(3)}$$

The circuit Vout (magnitude) is calculated as:

$$Vout = Vin\frac{Xc}{\sqrt{R^2 + Xc^2}} \qquad \text{.........................(4)}$$

5.2 Practical Laboratory

OBJECTIVES

a) Construct passive low pass filter circuit

b) Measure the Voltage Gain of Vin and Vout waveform

c) Plot the graph for the frequency response curve for each filter

PROCEDURE

PASSIVE LOW PASS FILTER

1. Sketch the circuit diagram for figure below by using multisim online simulator

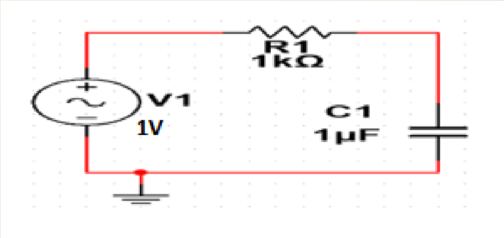


Figure 3.2.1

2. Simulate the system by using the run button to achieved the output waveforms.
3. Calculate Vout by using the given formula.

$$Vout = Vin\frac{Xc}{\sqrt{R^2 + Xc^2}}$$

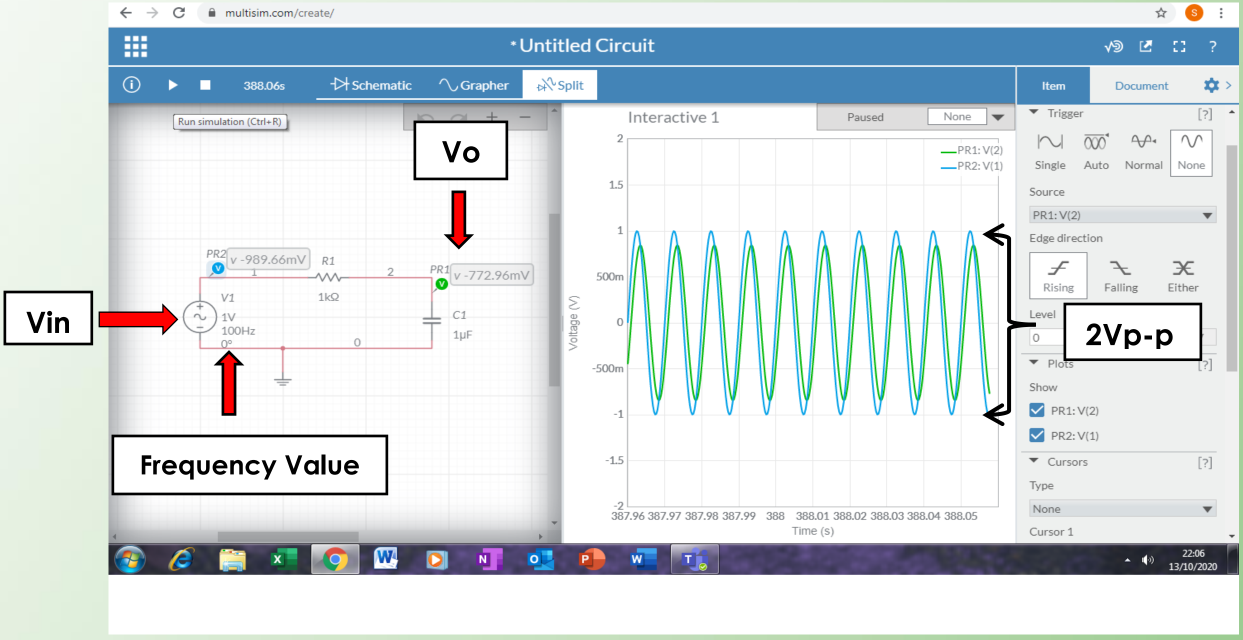
Where ; $Xc = 1/2\pi FC$ **and** $z = \sqrt{x_c^2 + R^2}$

Therefore; $v_0 = v_{in}\left(\frac{x_c}{z}\right)$

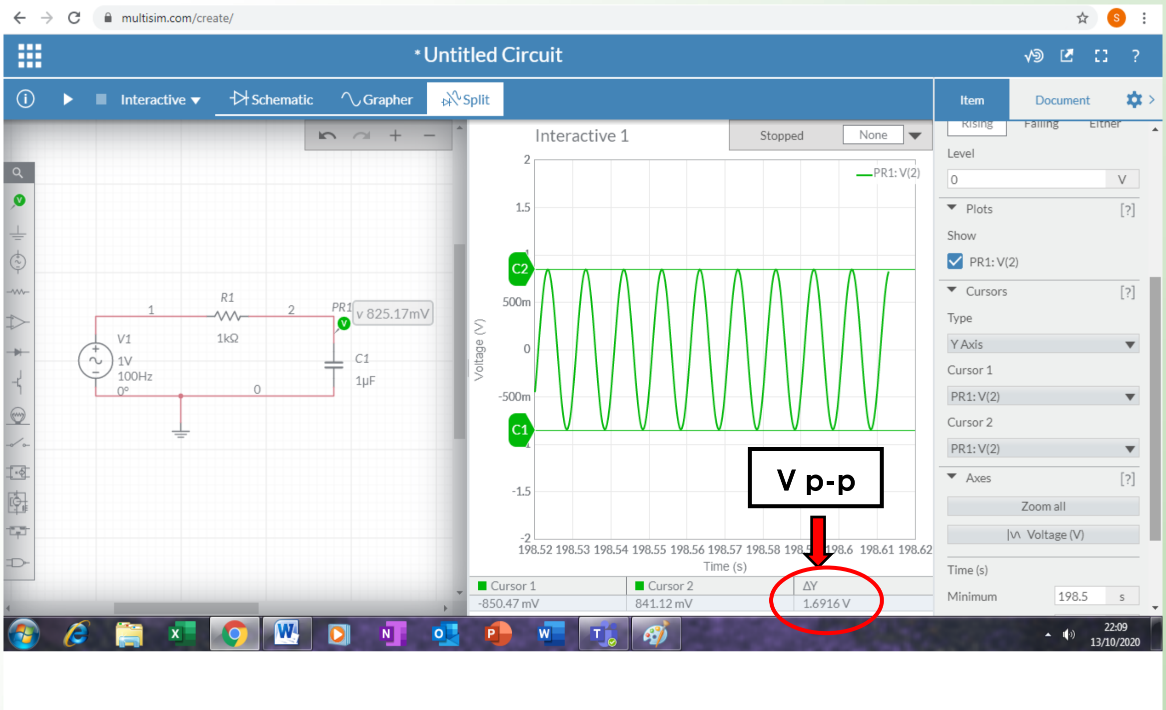
4. Set the input voltage V_{in} = 2Vp-p and frequency F = 100 Hz. Measure the input voltage V_{in} and the output voltage V_{out} .

Output

1. Input and output waveform at Vin = 2Vp-p

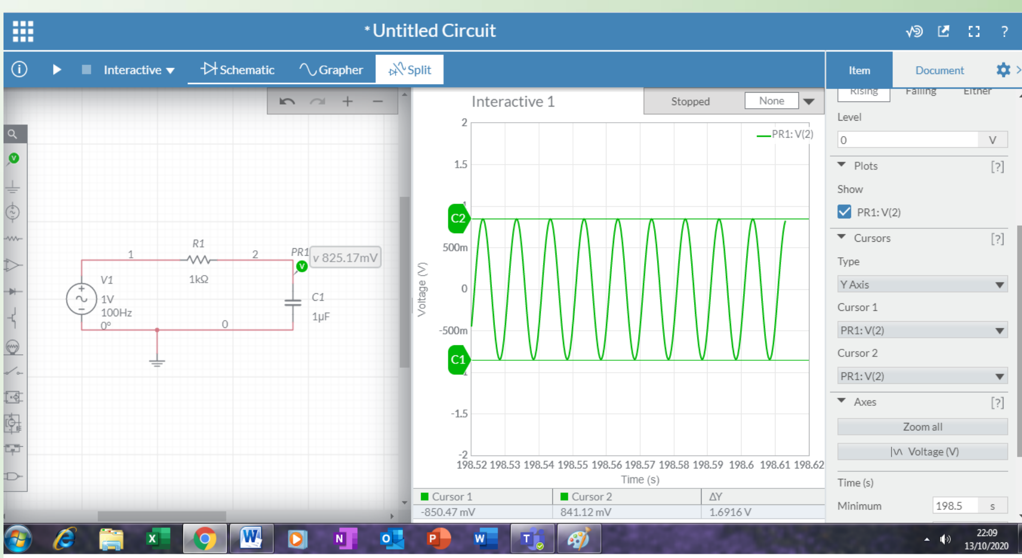


At Frequency 100 Hz

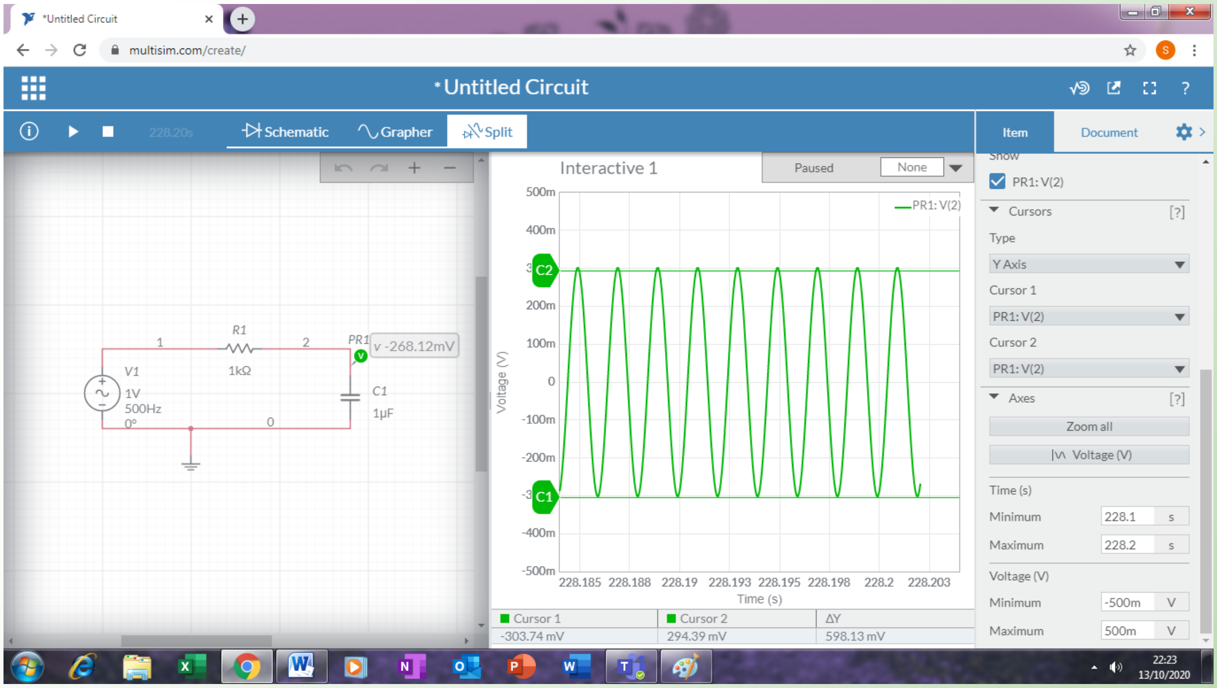


5. Repeat taking measurements for output voltage V_{out} for frequency 300 Hz and 500 Hz

At frequency 300 Hz



At frequency 500 Hz



5.3 QUESTION

1. By using multisim online simulator, sketch the same circuit as in figure 3.2.1, measure the input voltage V_{in} and the output voltage V_{out} with the frequency of 1KHz, 5KHz, 10KHz, 20KHz, 30KHz and 50KHz. Then draw the a graph of Gain (Av) Vs Frequency

5.4 Exercises

1. List SIX (6) characteristics for passive filter circuit.
2. Sketch and label the circuit and frequency response curve for Low Pass Passive filter circuit

UNIT 6

DIGITAL TO ANALOG CONVERTER

6.1 THEORY

Digital to analog converter (DAC) was used to convert digital signals to analog signals. The output signal from a computer connected to the DAC, where the DAC converts this signal is an analog voltage or current. Example: Computer manufactures digital output signal range from 00000000_2 to 11111111_2, where the DAC converts the voltage range of 0 to 10V.

DAC Block Diagram

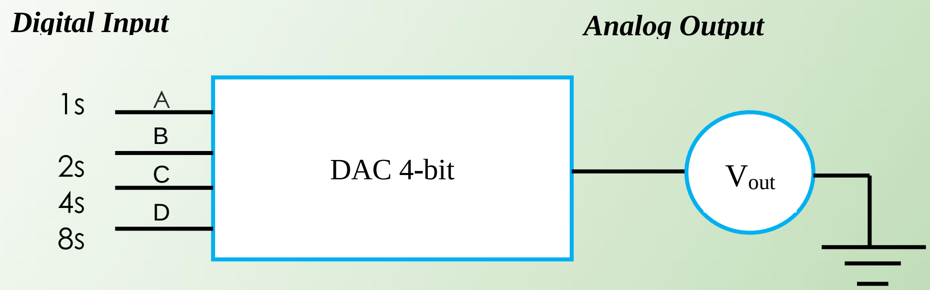


Figure 6.1: Block Diagram for DAC

Binary Weighted Resistor DAC circuit was design by using several resistor values. The output voltage of the DAC can be determined as follow:

$$\text{Gain, } Av = \frac{Vout}{Vin} = -\frac{Rf}{Rin}$$

$$V_o = -R_f\left(\frac{S1}{R} + \frac{S2}{2R} + \frac{S3}{4R} + \frac{S4}{8R}\right)Vin$$

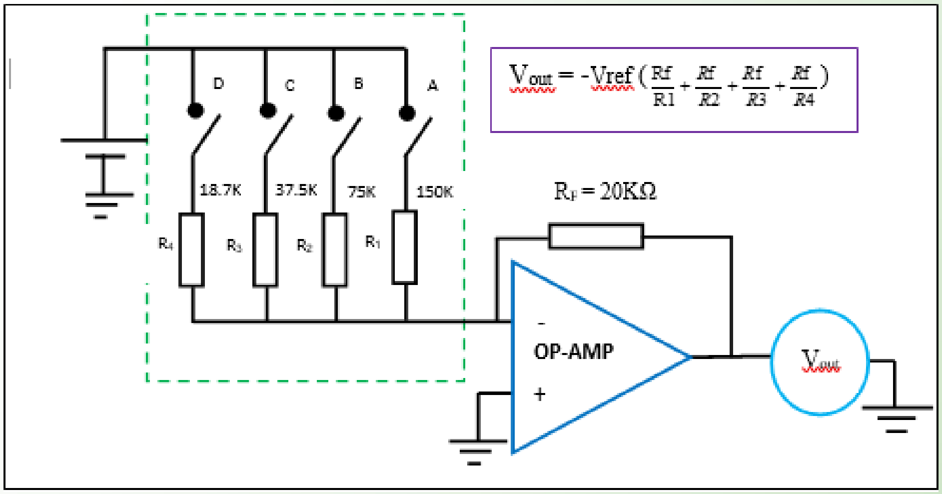


Figure 6.2: 4-Bit Binary Weighted Resistor DAC

The circuit in figure 6.2 above is a digital to analog converter circuit of resistors 4-bit binary weights. The resistor values can be calculated using the weighting of the binary number.

6.2 Practical Laboratory

OBJECTIVES

i. Construct the circuits DAC type Resistive divider circuit

ii. Measure the Resistive divider circuit's output.

PROCEDURES:

1. Draw circuit as in figure below by using multisim online simulator

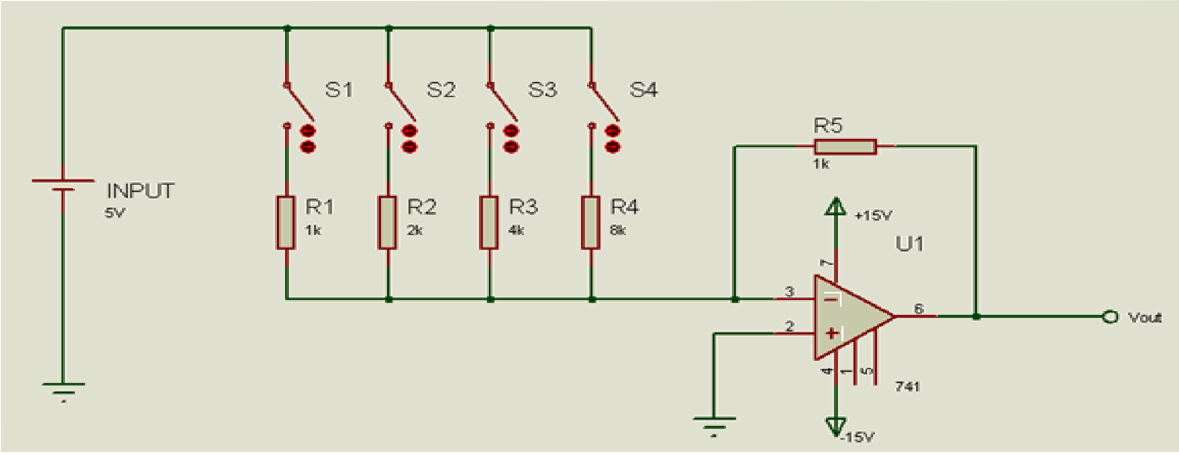
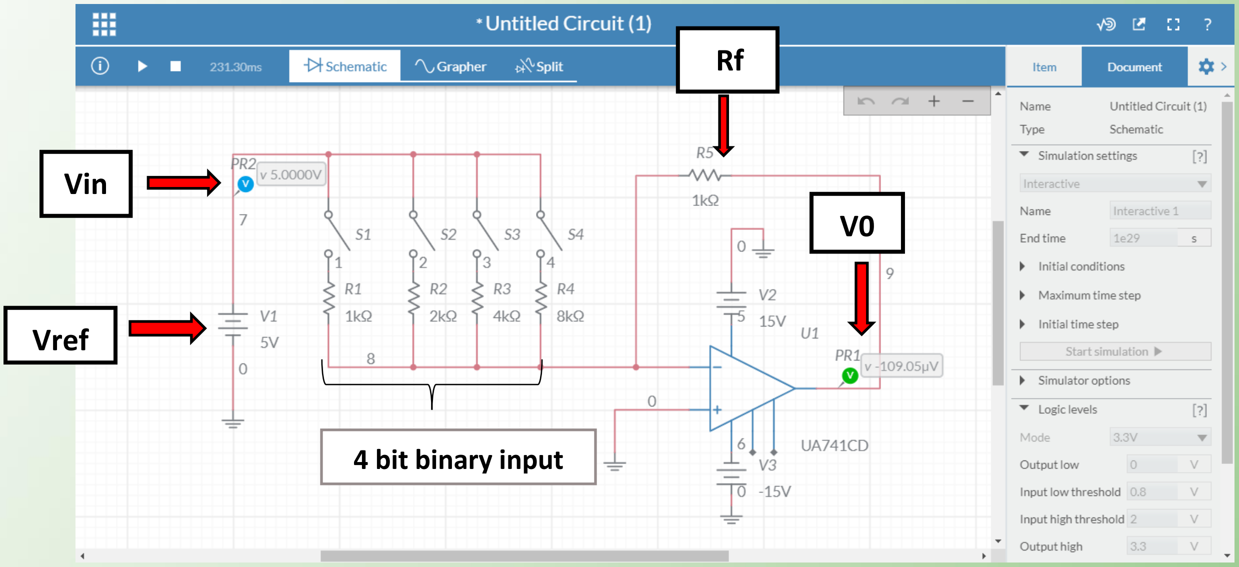


Figure 6.3: 4 Bit Binary Weighted Resistor DAC

2. Calculate the expected value for Vout and compare with the experimental values for binary 0000 and 0001 by using the circuit in figure 6.3 above. You can (Refer video in: https://www.youtube.com/watch?v=3Gb8c1qsubQ&vl=en

Output

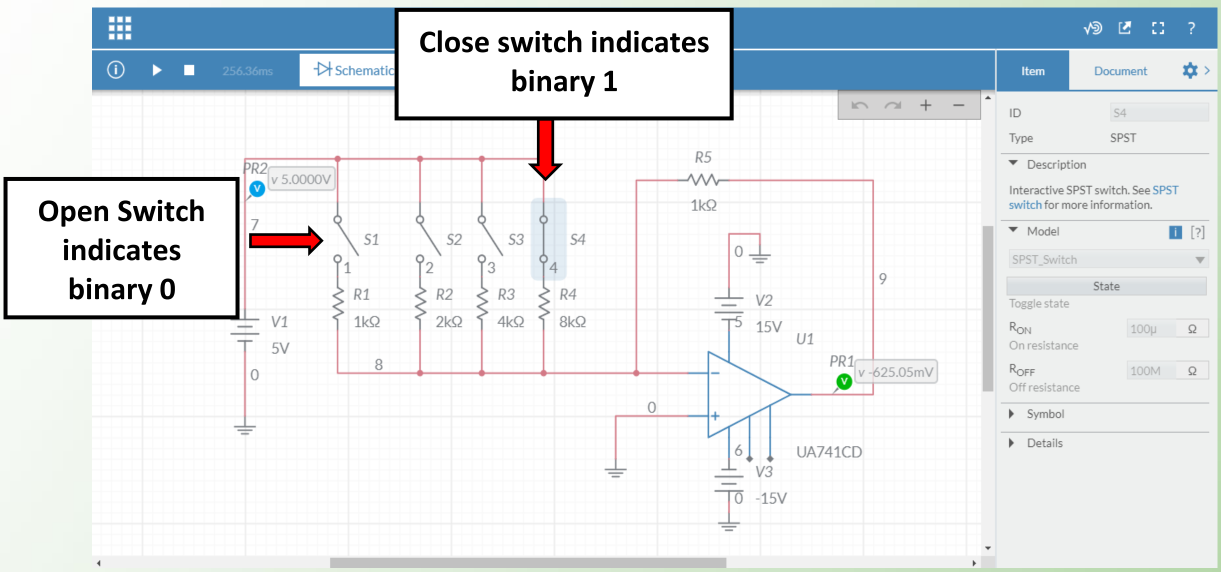
Binary Input 0000



Calculated Value

$$V_{out} = -Vref\left(\frac{Rf}{R1}+\frac{Rf}{R2}+\frac{Rf}{R3}+\frac{Rf}{R4}\right)$$
$$= -5V\,(0+0+0+0)$$
$$= 0\ V$$

Binary Input 0001



Calculated Value

$$V_{out} = -Vref\left(\frac{Rf}{R1}+\frac{Rf}{R2}+\frac{Rf}{R3}+\frac{Rf}{R4}\right)$$
$$= -5V\,(0+0+0+1k/8k)$$
$$= 0.125\ V$$

6.3 QUESTION

1. Repeat the procedure 1 and 2. Then calculate the calculated value of V0 by using the formulae in figure 6.2. Compare the both values of calculated and measured values. Record all the readings in the table 6.1

2. State the observation can be made from the value of Vout in the table.

Decimal	Binary input				Analog output (Vout)	Analog output (Vout)
	S1	S2	S3	S4	Calculated value	Measured value
0	0	0	0	0		
1	0	0	0	1		
2	0	0	1	0		
3	0	0	1	1		
4	0	1	0	0		
5	0	1	0	1		
6	0	1	1	0		
7	0	1	1	1		
8	1	0	0	0		
9	1	0	0	1		
10	1	0	1	0		
11	1	0	1	1		
12	1	1	0	0		
13	1	1	0	1		
14	1	1	1	0		
15	1	1	1	1		

Table 6.1

Exercises

1.

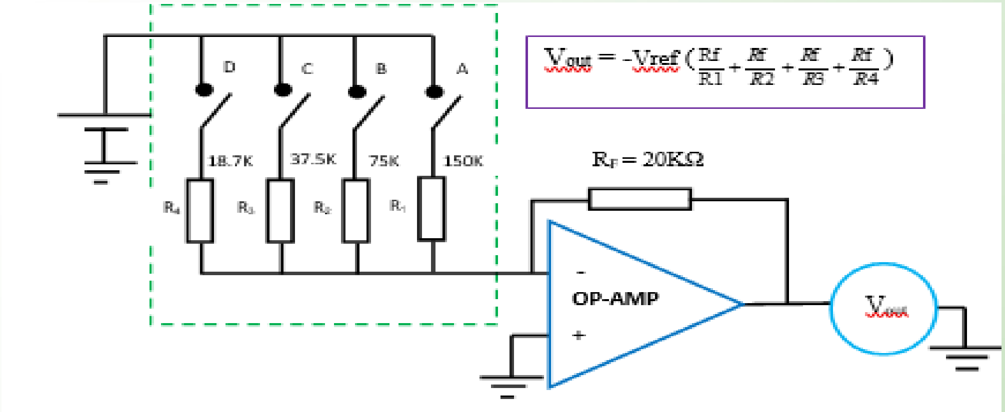


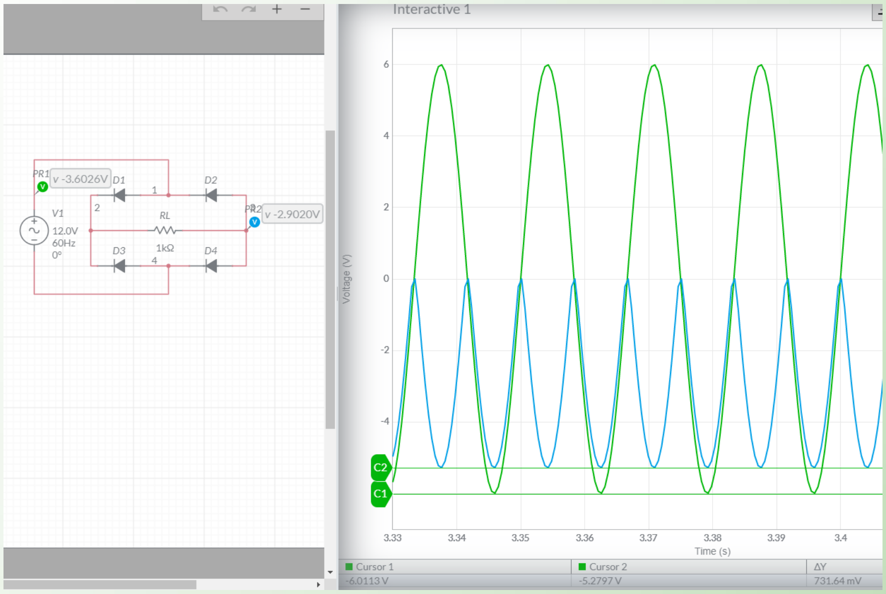
Figure 6.4

The circuit in figure 6.4 is a digital to analog converter circuit of resistors 4-bit binary weights. By using the weighting of the binary number, calculate the value for resistor R2, R3 and R4 and what are the analogue output voltage will be generated if a digital input 0001 and 0110 were applied?

ANSWERS

Unit 1

1. Vo at RL.



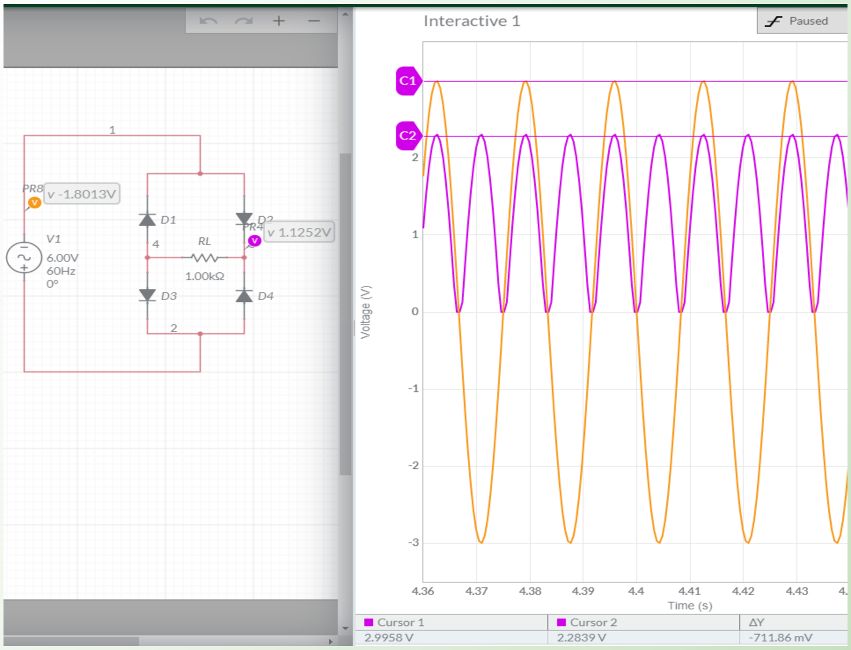
2. Silicon diode = 0.7V

Vo= Vin – (2 x 0.7V)

= (-6.0) - (-1.4)

= -4.6V

3. Vo at RL.



4.

$$Vin = 20\ Vp\text{-}p$$

$$= 10\ Vp$$

$$\frac{V_{MN}}{Vin} = \frac{Ns}{Np}$$

$$V_{MN} = \frac{Ns}{Np} x Vm$$

$$= \frac{1}{2} x 10Vp$$

$$= 5\ Vp$$

then $V_o = V_{MN}$

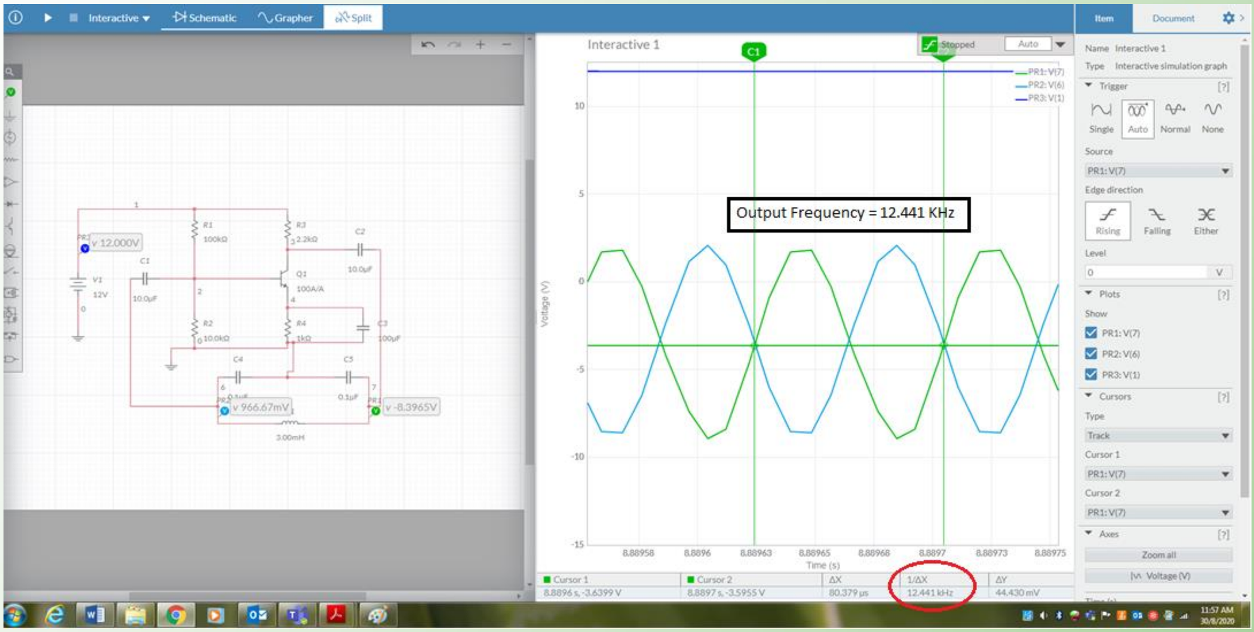
$= 5\ V_p$

Output frequency signal = 2 x frequency input signal

= 100 Hz

Unit 2

a.



b. Calculation of Output Frequency using formula :

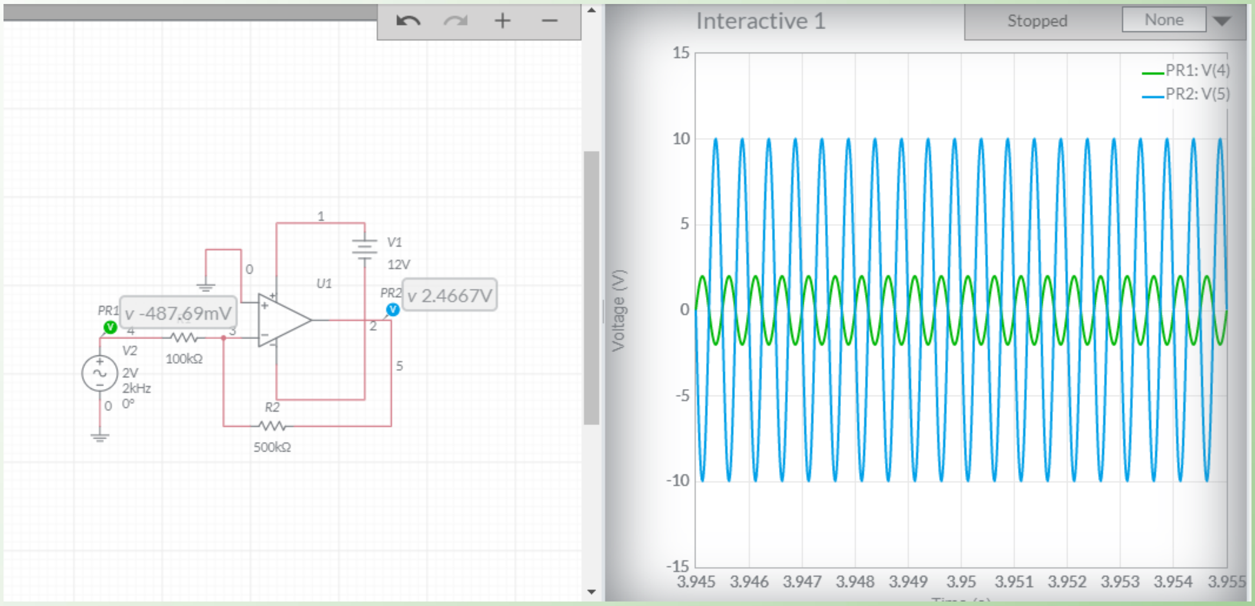
$$CT = \frac{C1XC2}{C1 + C2}$$

$$CT = \frac{0.1\mu X 0.1\mu}{0.1\mu + 0.1\mu} = 50nF$$

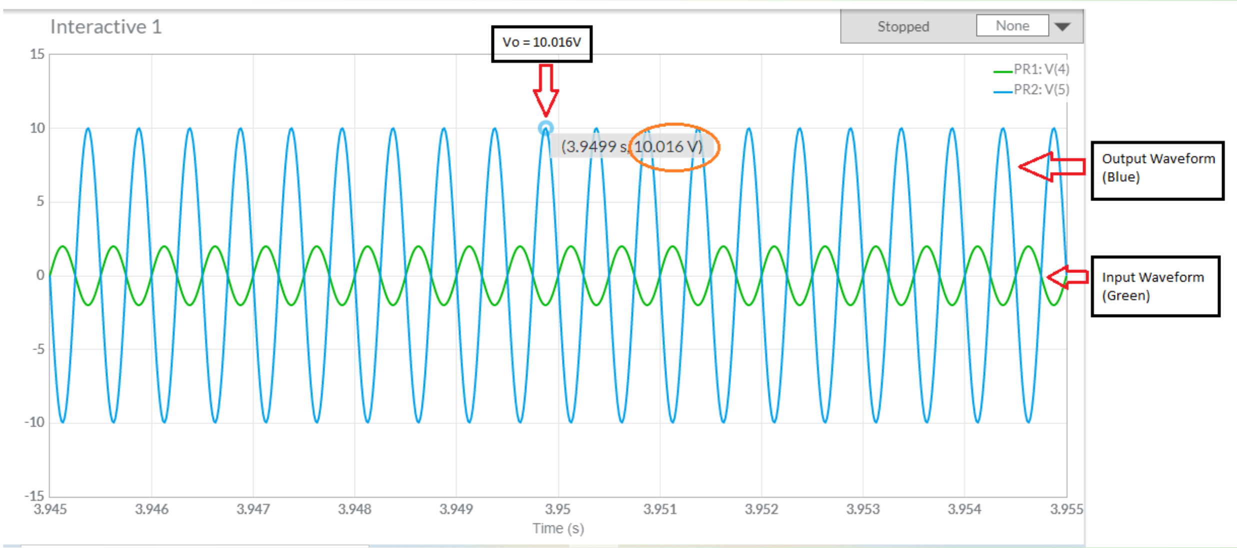
$$fr = \frac{1}{2\pi\sqrt{3m\ x\ 50nF}} = \mathbf{12.995\ kHz}$$

Unit 3

1.a

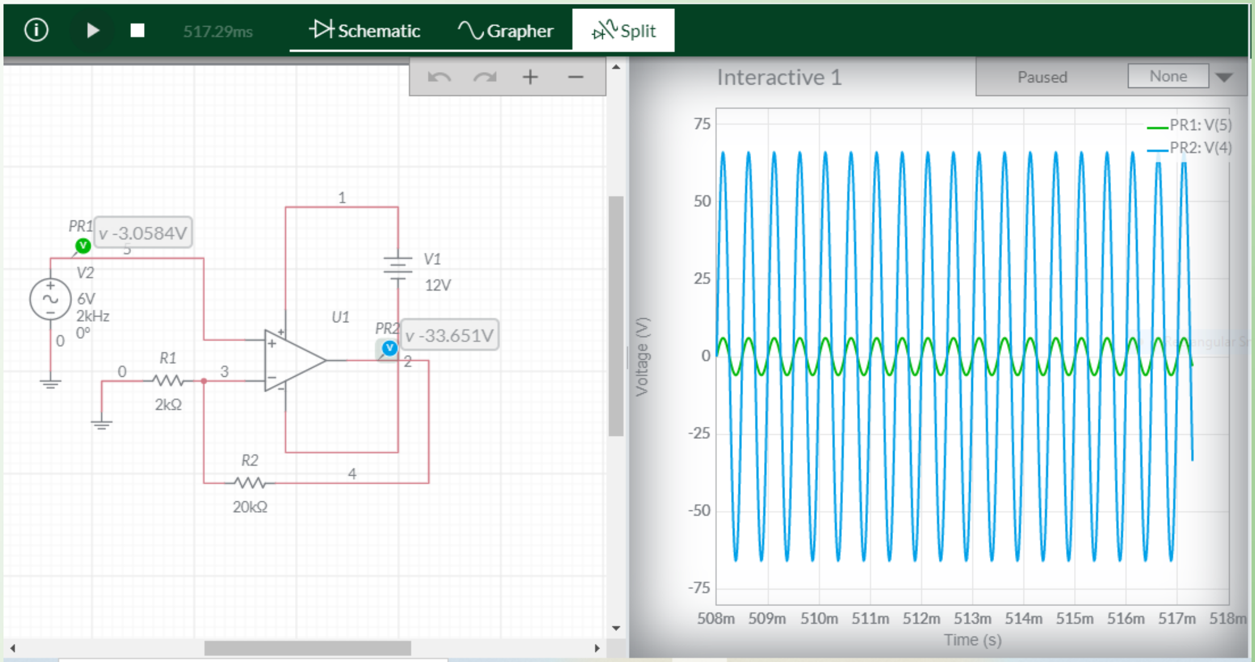


1.b

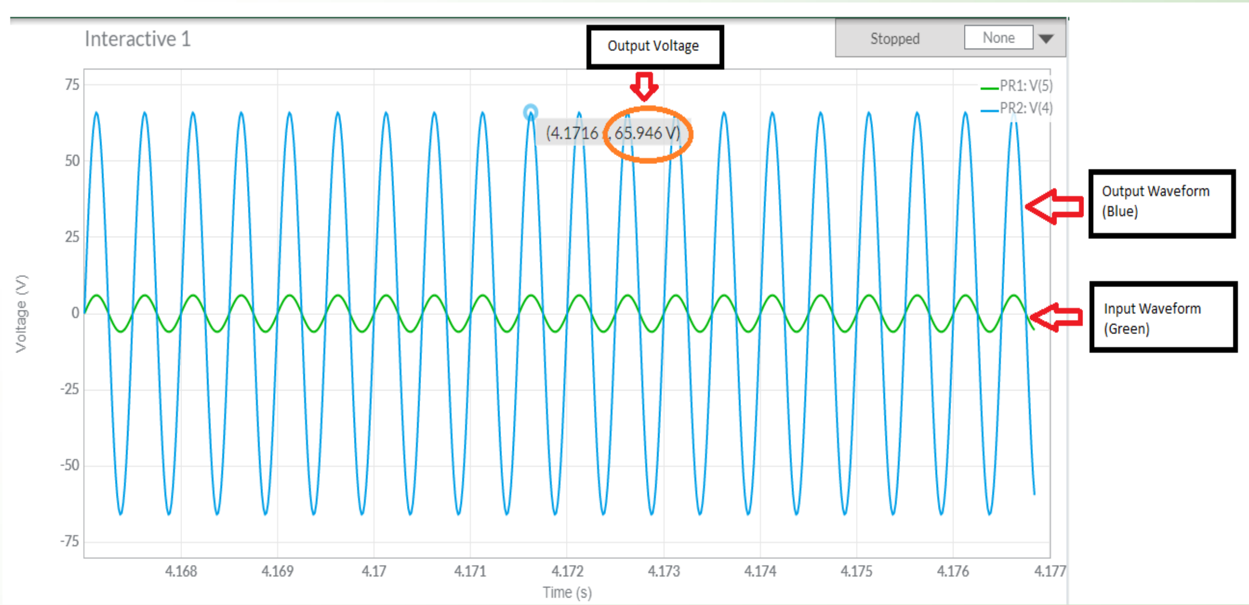


The Phase angle of input and output waveform in invert because the circuit is Inverting Op-Amp.

2. a)



b)



The Phase angle of input and output waveform is in phase because the circuit is Non-Inverting Op-Amp.

Unit 4

1. Basically, astable multivibrator acts as an oscillator, so the switching back and forth between “high” (full voltage) and “low” (no voltage) output states. The time duration of one of these states is set by the charging action of the capacitor, through both resistors (R1 and R2). The other state’s time duration is set by the capacitor discharging through one resistor (R2). Obviously, the charging time constant must be τcharge = (R1 + R2)C1, while the discharging time constant is τdischarge = R2C.

1. If value of R_1 is varying from 2KΩ, 10KΩ, 15KΩ and 56KΩ

	R_1=2KΩ	R_1=10KΩ	R_1=15KΩ	R_1=56KΩ
TH	4.5701ms	10.186ms	13.516ms	42.018ms
TL	3.3308ms	3.4280ms	3.5260ms	3.3301ms
T	7.9783ms	13.712ms	16.944ms	45.348ms

***Hint: When R1 value increase, to increase High Time (TH), without affecting the Low Time (TL)**

2. If value of R_2 is varying from 1KΩ, 3.9KΩ, 5.6KΩ and 10KΩ.

	R_2=1KΩ	R_2=3.9KΩ	R_2=5.6KΩ	R_2=10KΩ
TH	8.9128ms	10.970ms	12.047ms	15.279ms
TL	783.55µs	2.8404ms	4.1136ms	7.0519ms
T	9.7943ms	13.908ms	16.161ms	22.429ms

***Hint: When R1 value increase, to increase High Time (TH), increase the Low Time (TL) and decrease the duty cycle.**

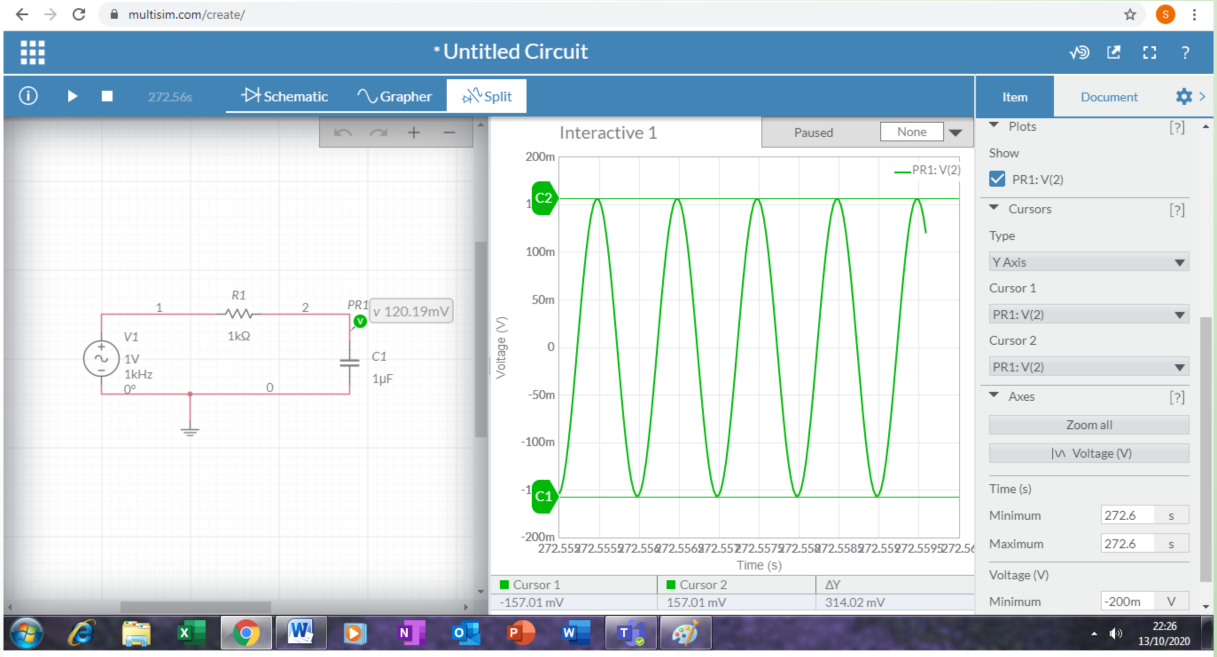
3. If the value of C1 is varying from 0.1µ, 0.2µ, 0.5µ and 10µ using multisim simulator.

	C_1=0.1µ	C_1=0.2µ	C_1=0.5µ	C_1=10µ
TH	1.1666ms	2.3182ms	5.7971ms	115.84ms
TL	326.46µs	673.44 µs	1.6162ms	33.178ms
T	1.4883ms	2.9787ms	7.4133ms	148.46ms

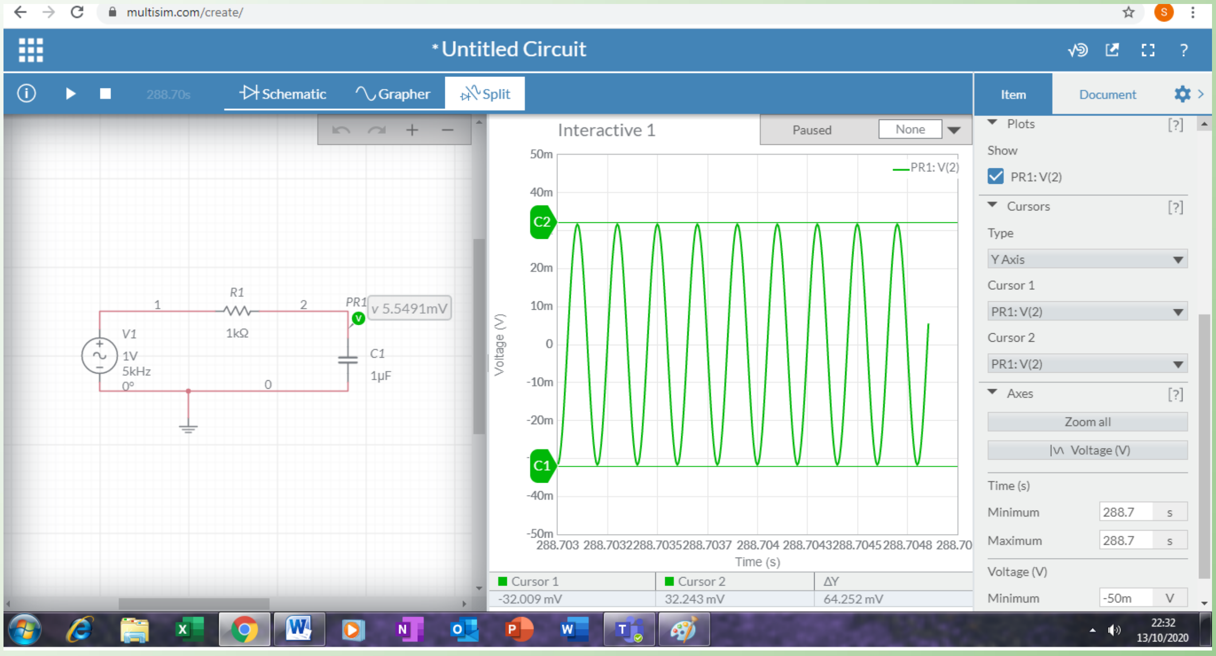
***Hint: When C value increase, it will increase period (reduce the frequency).**

Unit 5

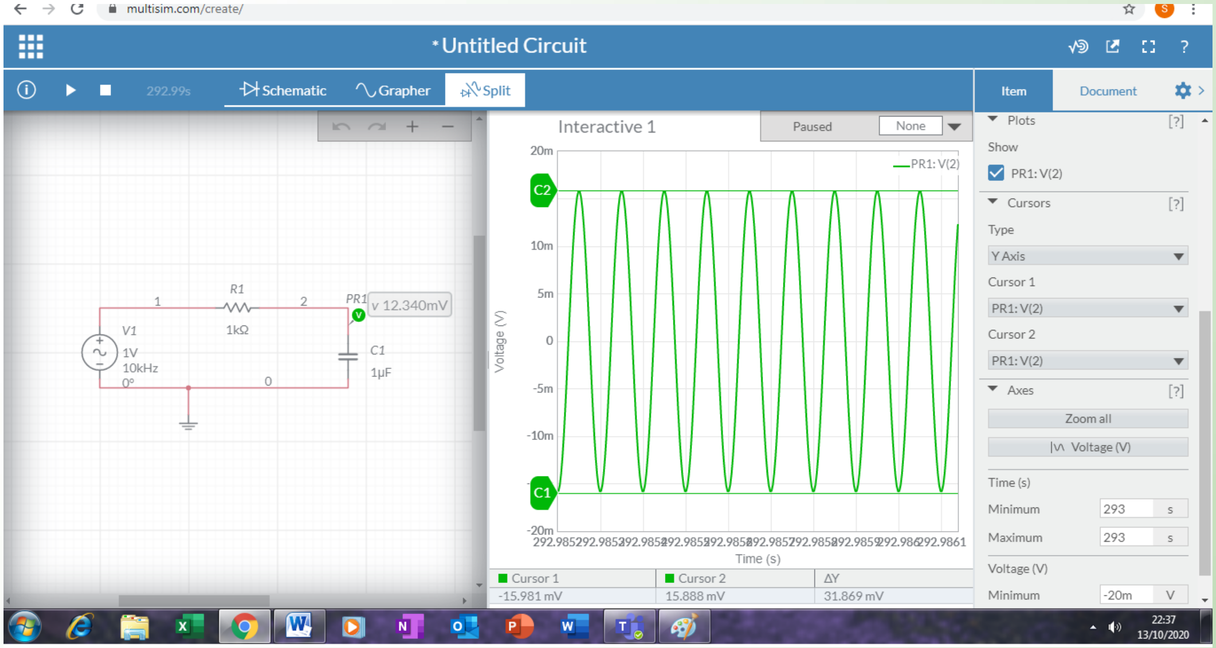
F = 1 KHz



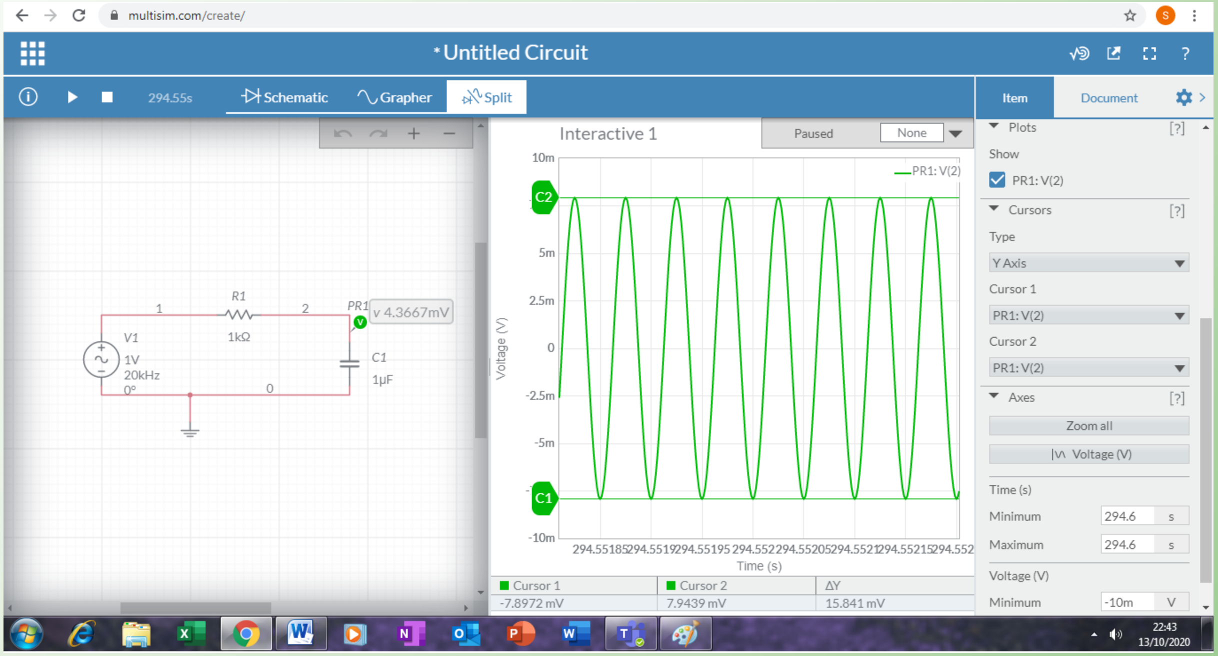
F = 5 KHz



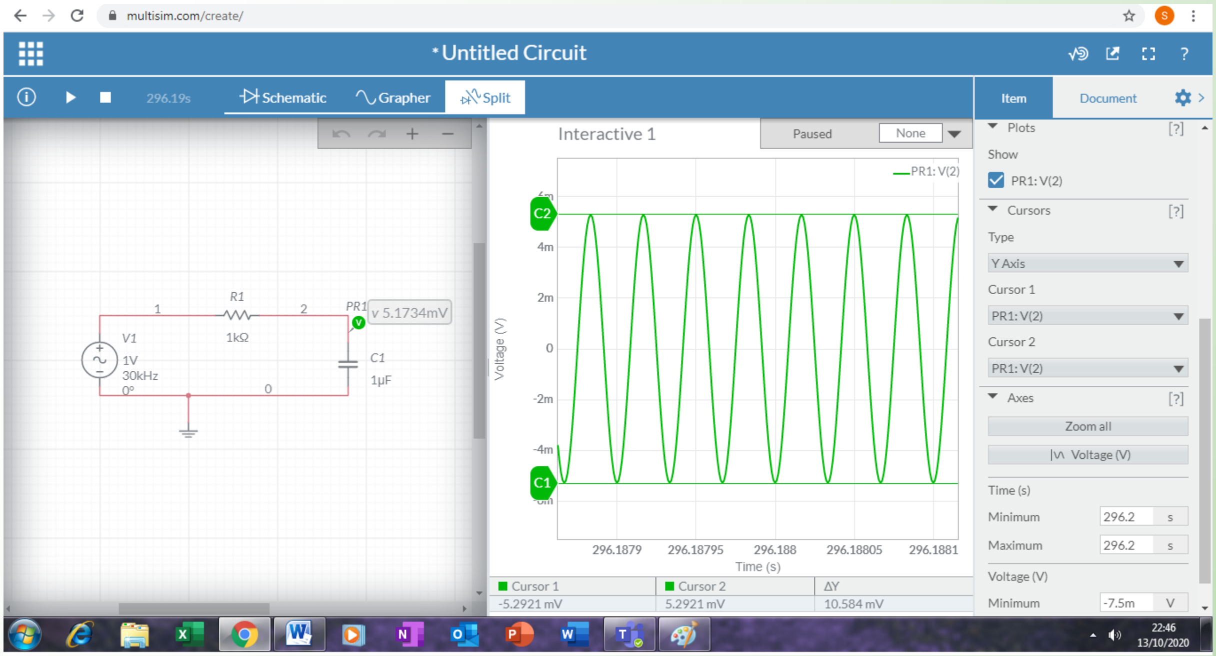
F = 10 KHz



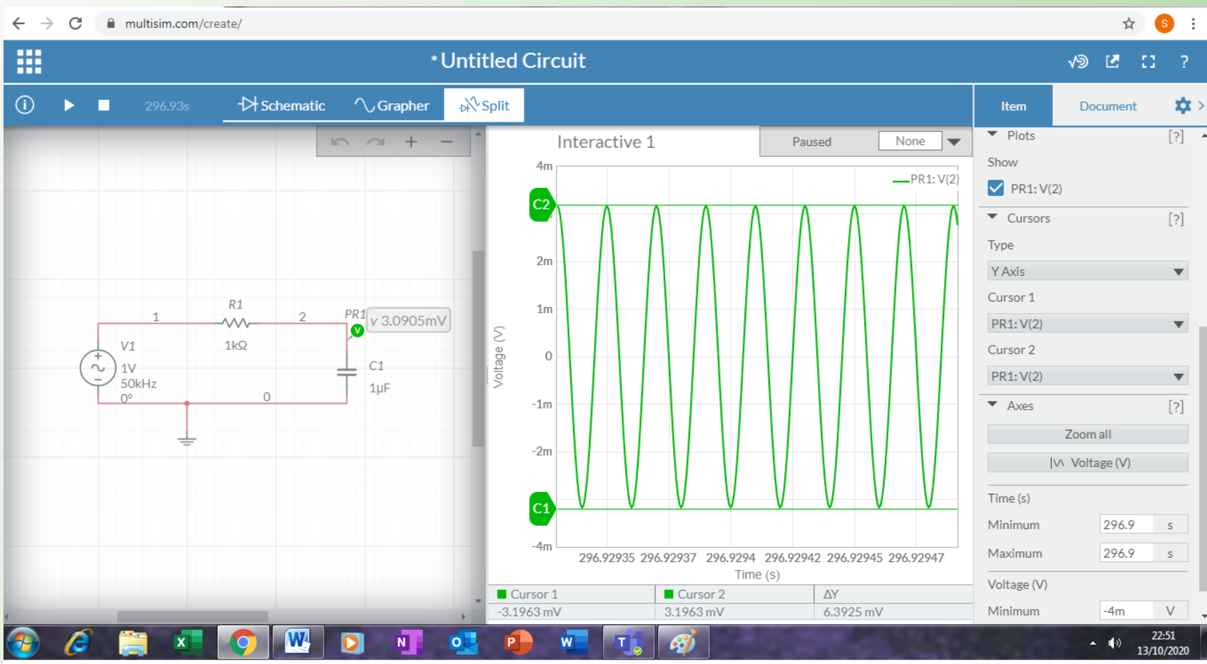
F = 20 KHz



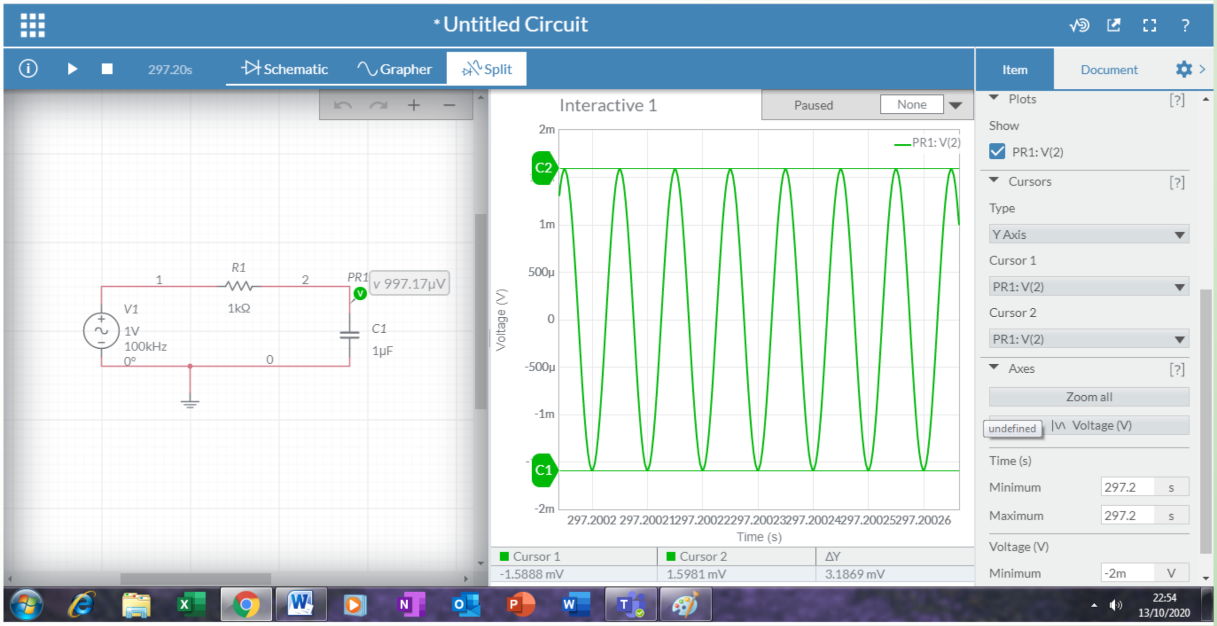
F = 30 KHz



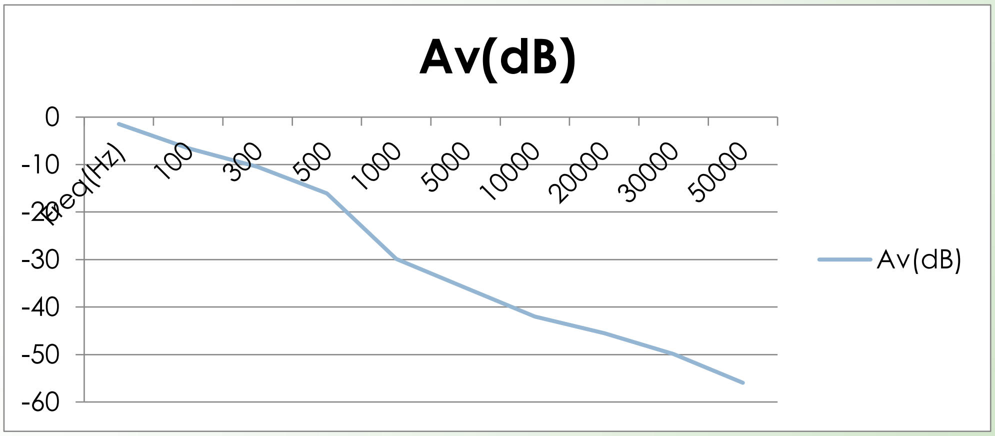
F = 50 KHz



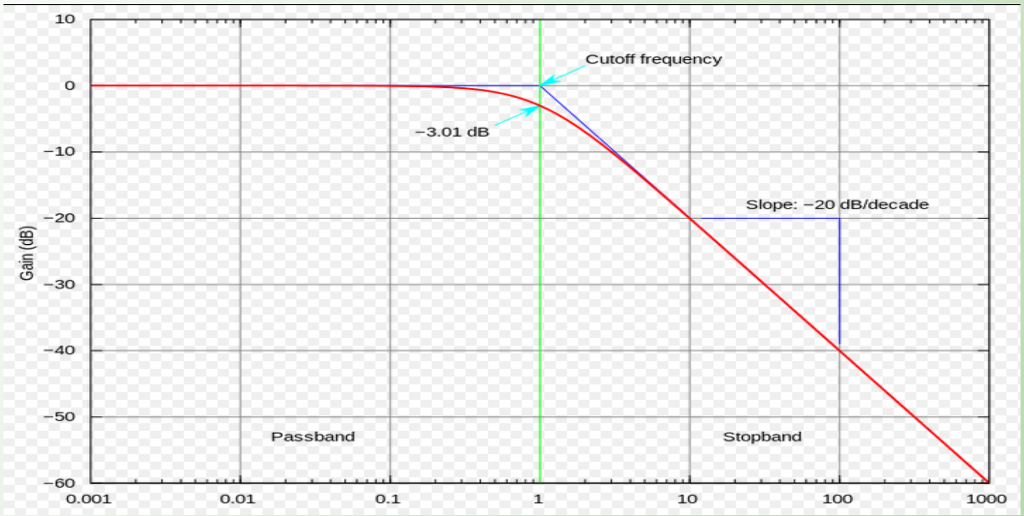
F = 100 KHz



Freq(Hz)	Av(dB)
100	-1.455
300	-6.564
500	-10.484
1000	-16.082
5000	-29.87
10000	-35.918
20000	-42.025
30000	-45.528
50000	-49.908
100000	-55.956



Graph of Gain (Av) VS Frequency



Graph of Gain (Av) VS Frequency

3.4 Exercises

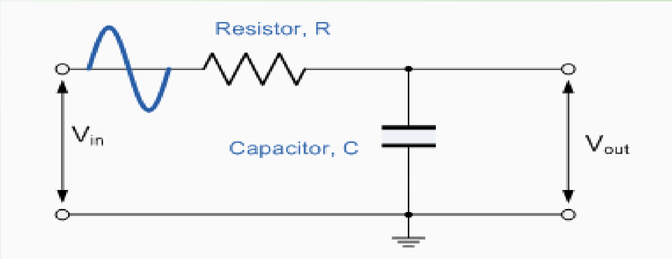
1. List SIX (6) characteristics for passive filter circuit.

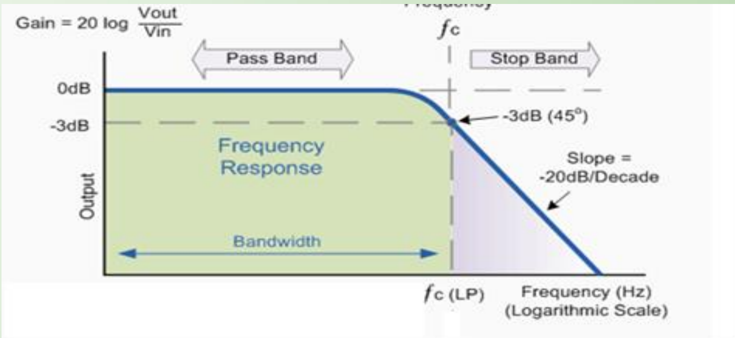
Answer

1. Use the passive component like resistor, inductor and capacitor
2. It does not need power supply
3. Input impedance is less which loads the source
4. Output impedance is more so it cannot drive the low impedance load
5. It is not possible to increase gain
6. It is not possible to adjust parameter

2. Sketch and label the circuit and frequency response curve for Low Pass Passive filter circuit

Answer





Unit 6

RESULT/ANALYSIS:

1.

Decimal	Binary input				Analog output (Vout)	Analog output (Vout)
	S1	S2	S3	S4	Calculated value	Measured value
0	0	0	0	0	$V_{out} = -Vref\,(\frac{Rf}{R1}+\frac{Rf}{R2}+\frac{Rf}{R3}+\frac{Rf}{R4})$ = - 5V (0 + 0 + 0 + 0) = 0 V	**Vo from the experiment (suppose the value will be same as calculated value or with minimum error)**
1	0	0	0	1	$V_{out} = -Vref\,(\frac{Rf}{R1}+\frac{Rf}{R2}+\frac{Rf}{R3}+\frac{Rf}{R4})$ = - 5V (0 + 0 + 0 + 1k/8k) = - 0.625 V	
2	0	0	1	0	$V_{out} = -Vref\,(\frac{Rf}{R1}+\frac{Rf}{R2}+\frac{Rf}{R3}+\frac{Rf}{R4})$ = - 5V (0 + 0 + 1k/4k + 0) = - 1. 25 V	
3	0	0	1	1	$V_{out} = -Vref\,(\frac{Rf}{R1}+\frac{Rf}{R2}+\frac{Rf}{R3}+\frac{Rf}{R4})$ = -5V (0 + 0 + 1k/4k +1k/8k) = - 1. 875 V	
4	0	1	0	0	$V_{out} = -Vref\,(\frac{Rf}{R1}+\frac{Rf}{R2}+\frac{Rf}{R3}+\frac{Rf}{R4})$ = - 5V (0 +1k/2k + 0+ 0) = - 2.5 V	

5	0	1	0	1	$V_{out} = -Vref\,(\frac{Rf}{R1}+\frac{Rf}{R2}+\frac{Rf}{R3}+\frac{Rf}{R4})$ =-5V (0 + 1k/2k +0 +1k/8k) = - 3.125 V	
6	0	1	1	0	$V_{out} = -Vref\,(\frac{Rf}{R1}+\frac{Rf}{R2}+\frac{Rf}{R3}+\frac{Rf}{R4})$ = - 5V (0 + 1k/2k + 1k/4k + 0) = -3.75 V	
7	0	1	1	1	$V_{out} = -Vref\,(\frac{Rf}{R1}+\frac{Rf}{R2}+\frac{Rf}{R3}+\frac{Rf}{R4})$ = - 5V (0 + 1k/2k + 1k/4k +1k/8k) = - 4. 375 V	
8	1	0	0	0	$V_{out} = -Vref\,(\frac{Rf}{R1}+\frac{Rf}{R2}+\frac{Rf}{R3}+\frac{Rf}{R4})$ = - 5V (1k/2k + 0 + 0 +0) = - 5.000 V	
9	1	0	0	1	$V_{out} = -Vref\,(\frac{Rf}{R1}+\frac{Rf}{R2}+\frac{Rf}{R3}+\frac{Rf}{R4})$ = - 5V (1k/1k + 0 +0 +1k/8k) = - 5.625 V	
10	1	0	1	0	$V_{out} = -Vref\,(\frac{Rf}{R1}+\frac{Rf}{R2}+\frac{Rf}{R3}+\frac{Rf}{R4})$ = - 5V (1k/1k + 0 + 1k/4k +0) = - 6.250 V	
11	1	0	1	1	$V_{out} = -Vref\,(\frac{Rf}{R1}+\frac{Rf}{R2}+\frac{Rf}{R3}+\frac{Rf}{R4})$ = - 5V (1k/1k + 0 + 1k/4k +1k/8k) = - 6.875 V	
12	1	1	0	0	$V_{out} = -Vref\,(\frac{Rf}{R1}+\frac{Rf}{R2}+\frac{Rf}{R3}+\frac{Rf}{R4})$ = - 5V (1k/1k + 1k/2k + 0 +0) = -7.500 V	

13	1	1	0	1	$V_{out} = -Vref\left(\frac{Rf}{R1}+\frac{Rf}{R2}+\frac{Rf}{R3}+\frac{Rf}{R4}\right)$ $= -5V\ (1k/1k + 1k/2k\ + 0 + 1k/8k)$ $= -8.125\ V$	
14	1	1	1	0	$V_{out} = -Vref\left(\frac{Rf}{R1}+\frac{Rf}{R2}+\frac{Rf}{R3}+\frac{Rf}{R4}\right)$ $= -5V\ (1k/1k + 1k/2k\ + 1k/4k + 0)$ $= -8.750\ V$	
15	1	1	1	1	$V_{out} = -Vref\left(\frac{Rf}{R1}+\frac{Rf}{R2}+\frac{Rf}{R3}+\frac{Rf}{R4}\right)$ $= -5V(1k/1k+1k/2k+1k/4k+1k/8k)$ $= -9.375\ V$	

2. From the table, we can say that the value for every step size increasing by 0.625 (starting from 0000 until binary 1111)

Exercises

1. Referring to figure 6.4, the highest value resistor (R1= 150KΩ) is a digital input resistor, then the values of the other resistor is;

$$R_2 = \frac{R_1}{2^1} = \frac{150K}{2^1} = 75k\Omega$$

$$R_3 = \frac{R_1}{2^2} = \frac{150K}{2^2} = \frac{150K}{4} = 37.5k\Omega,$$

$$R_4 = \frac{R_1}{2^3} = \frac{150K}{8} = 18.75k\Omega$$

What is analog output voltage will be generated if a digital input 0001 and 0110 were applied? Given Vref is 3V.

i. Binary Input = 0001

R_1 = 150K, R_F = 20K

$$V_{out} = -Vref\left(\frac{Rf}{R1}+\frac{Rf}{R2}+\frac{Rf}{R3}+\frac{Rf}{R4}\right)$$

$$V_{out} = -(-3V)\left(\frac{20K}{150K}+0+0+0\right)$$

$$\mathbf{V_{out} = 0.4V}$$

OR

Multiplier Voltage (A_V) = $\frac{R_F}{R^*} = \frac{20K\,\Omega}{150K\Omega} = 0.133$, R*= R1

$\mathbf{V_{out} = V_{ref}\ X\ A_V}$ = 3 X 0.133

$\mathbf{V_{out} = 0.4V}$

ii. Binary Input = 0110

$$V_{out} = -Vref\left(\frac{Rf}{R1}+\frac{Rf}{R2}+\frac{Rf}{R3}+\frac{Rf}{R4}\right)$$

$$V_{out} = -(-3V)\left(0+\frac{20K}{75K}+\frac{20K}{37.5K}+0\right)$$

V_{out}= 3V (0+0.27+0.53+0)

V_{out}= **2.4V**

OR

$A_V = \frac{R_F}{R^*} = \frac{20K\,\Omega}{25K\Omega} = 0.8$

R* = (R2 parallel with R3) = (75KΩ // 37.5KΩ)= 25KΩ

$\mathbf{V_{out} = V_{ref}\ X\ A_V}$ = 3 X 0.8 = **2.4V**

REFERENCES

DEE30043 ELECTRONICS CIRCUIT LABSHEET

Electronic Devices and Circuit Theory, Eight Edition, Robert L.Boylested, Louis Nashelsky, Prentice Hall, 2002

https://www.electronics-tutorials.ws/diode/diode_6.html

https://www.electrical4u.com/bridge-rectifiers/

https://www.electrical4u.com/full-wave-rectifiers/

https://www.google.com/search?q=rectifier+diode&tbm=isch&hl=ms&chips=q:rectifier+diode,online
chips:bridge+rectifier+circuit+diagram:i798ByxtWlo%3D,online_chips:output+voltage:EJjByKUJqT4%3
D&rlz=1C1GCEU_enMY865MY865&sa=X&ved=2ahUKEwiG74yJpufyAhVUcSsKHUB5ABwQ4lYoAnoECA
EQGA&biw=979&bih=685#imgrc=gnLjrZCBenKKLM

https://www.ablic.com/en/semicon/products/analog/opamp/intro/

https://www.brainkart.com/media/article/articleunp3kaabpicture11.jpg

https://www.brainkart.com/article/Circuit-symbol-and-Pin-out-configuration-of-an-OP-AMP_3003/

https://www.multisim.com/

https://www.youtube.com/watch?v=iJYm_BGqa1A

https://www.allaboutcircuits.com/tools/555-timer-astable-circuit/

https://www.allaboutcircuits.com/worksheets/timer-circuits/

https://www.allaboutcircuits.com/textbook/experiments/chpt-6/555-audio-oscillator/

https://www.electronics-tutorials.ws/waveforms/555_oscillator.html

https://www.electronicshub.org/astable-multivibrator-using-555-timer/#Astable_Multivibrator_Mode_of_555_Timer_IC https://www.electronics-tutorials.ws/filter/filter_2.html

https://www.youtube.com/watch?v=lagfhNjMuQM

https://nanohub.org/resources/14905/download/LOW_PASS_FILTERS.pdf

https://en.wikipedia.org/wiki/Low-pass_filter#/media/File:Butterworth_response.svg

Terbitan

POLITEKNIK MALAYSIA
TUANKU SULTANAH BAHIYAH

e ISBN 978-967-0855-76-9

9 7 8 9 6 7 0 8 5 5 7 6 9