BECE101P

Basic Electronics Lab

Winter Semester 2022

(Slot: L19-L20)

Lab Record

by

Student name: Soma Anirudh

(Reg. No.): 21BCE5537

Submitted to

Faculty Name: Dr. Idayachandran G

School of Electronics Engineering

Vellore Institute of Technology,

Chennai Campus-600127

June 2022

Table of contents

Experiment-1

Name: Soma Anirudh

Slot: L19+L20 BECE101P

Date: February 23, 2022

Register number: 21BCE5537

Experiment number: 1

Study of electronic components and electronic measurement devices

The general aim of this experiment is to study and understand various functions of various components in electric circuits.

Task-1

Aim: Finding the colour code of resistors using 4 band system.

Description:

	Colour1	Colour2	Colour3	Colour4
330kwith 5% tolerance	Orange	Orange	Yellow	Gold
470kwith 10% tolerance	Yellow	Violet	Yellow	Silver
66k with 10% tolerance	Blue	Blue	Orange	Silver
3.2with 5% tolerance	Orange	Red	Gold	Gold
540 with 10% tolerance	Green	Yellow	Brown	Silver
27 with 5% tolerance	Red	Violet	Black	Gold

Task-2

Aim: Finding the missing capacitance values with codes

Description:

Ceramic capacitors:

- 492 – *4.9 nF*
- 103 – *10 nF*
- 352 – *3.5 nF*
- 104 – *100 nF*
- 285 – *2800 nF*
- 681 – *0.68 nF*

Task-3

Aim: Difference between CRO and DSO.

Description:

CRO(Cathode Ray Oscilloscope)	**DSO**(Digital Storage Oscilloscope)
1)CRO Cannot store and analyse signals and data	1) DSO Can store and analyse signals and data
2)CRO has higher writing speed and bandwidth	2) DSO has lower writing speed and bandwidth compared to CRO.
3) It is less stable and accurate because the time base is generated by the ramp circuit.	3) It is more stable and accurate because the time base is generated by a crystal clock.
4) CRO measures but doesn't display the properties of the waveform.	4) DSO measures and displays the properties of the waveform.
5) CRO is more expensive than DSO	5) DSO is cheaper than CRO
6) Simple CRO does not have any memory.	6) DSO has storage memory.
7) Manipulations are not possible in a CRO due to unavailability of reading out the memory	7) Manipulations are possible at any given time due to availability of readout memory.

Task-4

Aim: Using a Function Generator

Description:

- To generate Sine Signal with peak-to-peak voltage of 10V:

  Waveform type must be set to sine-wave and DC offset must be altered to 10 volts to obtain a peak-to-peak voltage of 10 volts.

- To generate square signal with 50% duty cycle:

  This control on the function generator changes the ratio of high voltage to low voltage time in a square wave signal.Waveform type must be set to square wave and its duty cycle must be changed to a 1:2 ratio that is to get a 50% duty cycle.

Task-5

Aim: Using a Cathode Ray Oscillator (CRO)

Description:

- **To measure Voltage:**
  Cathode Ray Oscilloscope can be used for the measurement of voltage of any electrical specification as the deflection of the electrostatic beam is directly proportional to the deflection plate voltage.Voltage can be measured by CRO directly if it is in the range of around 35V For high voltage measurements you need to minimize the voltage below the permissible limits. To measure the voltage, you can zoom in or out the voltage waveform by selection of proper volt/division setting which is provided by a selector switch in the CRO.

- **To measure Frequency:**
  Increase the vertical sensitivity to get the clear picture of the wave , adjust the sweep rate in such a way that screen displays a more than one but less than two complete cycles of the wave. Count the number of divisions of one complete cycle on the graticule from start to end. Take horizontal sweep rate and multiply it with the number of units that was counted for a cycle. It will give you the time period of the wave. Inverse the time period to find frequency (T=1/f or f=1/T)

Experiment-2

Name: Soma Anirudh

Slot: L19+L20 BECE101P

Date: February 26, 2022

Register number: 21BCE5537

Experiment number: 2

Ohms Law using LtSpice

The general aim of this experiment is to study and understand ohms law through different circuits in LtSpice.

Software Required: LtSpice

Theory:

Ohm's Law states that the current flowing in a circuit is directly proportional to the applied potential difference and inversely proportional to the resistance in the circuit.

Ohm's law can be expressed in a mathematical form:

$V = IR$

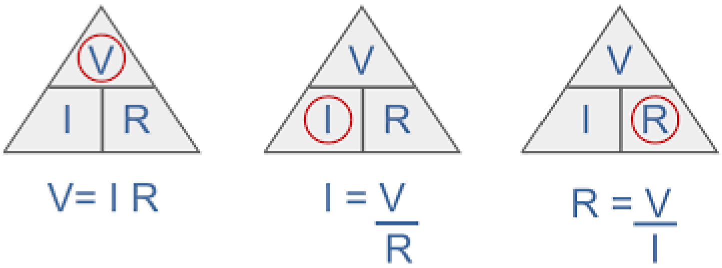
Where:

V = voltage expressed in Volts
I = current expressed in Amps
R = resistance expressed in Ohms

The formula can be manipulated so that if any two quantities are known the third can be calculated.

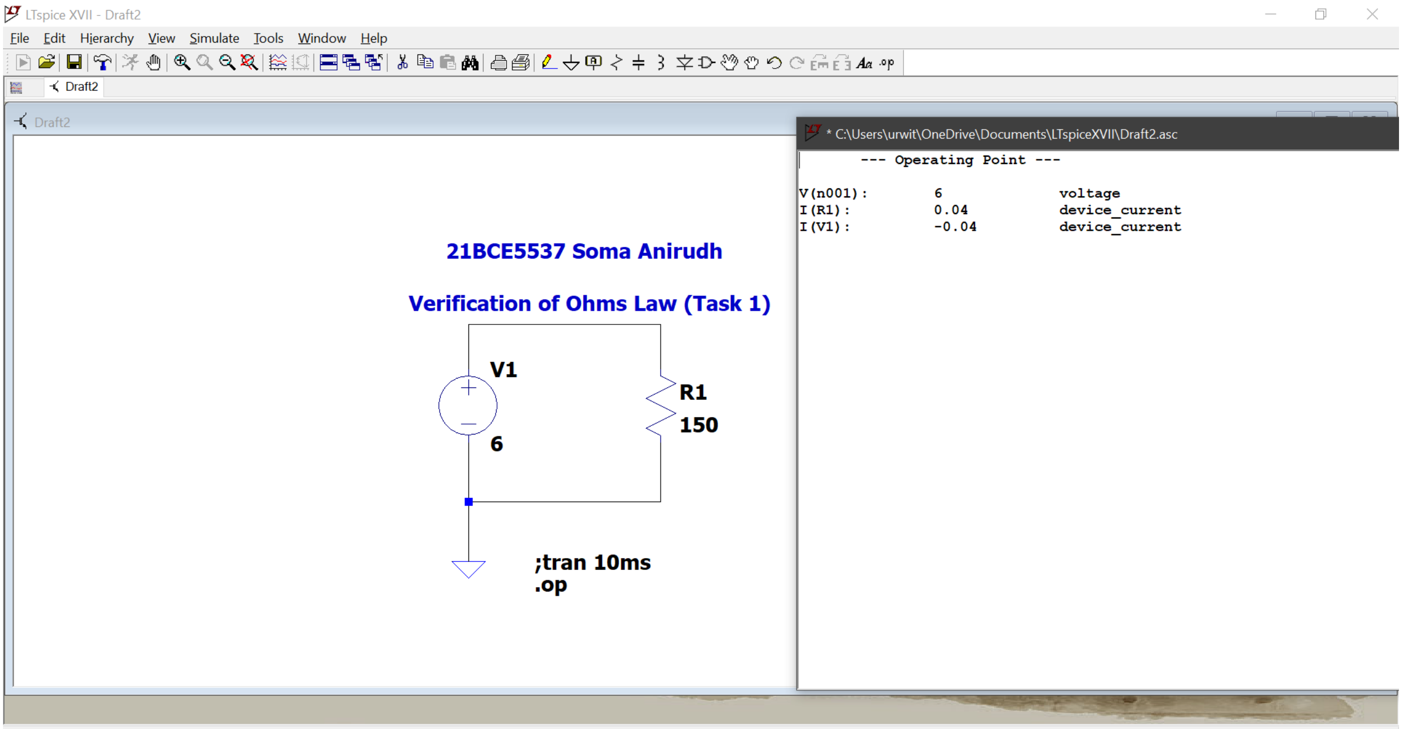
$I=V/R$ and $R=V/I$

In the top corner of the Ohms law triangle is the letter V, in the left-hand corner, the letter I, and in the right-hand bottom corner, R.

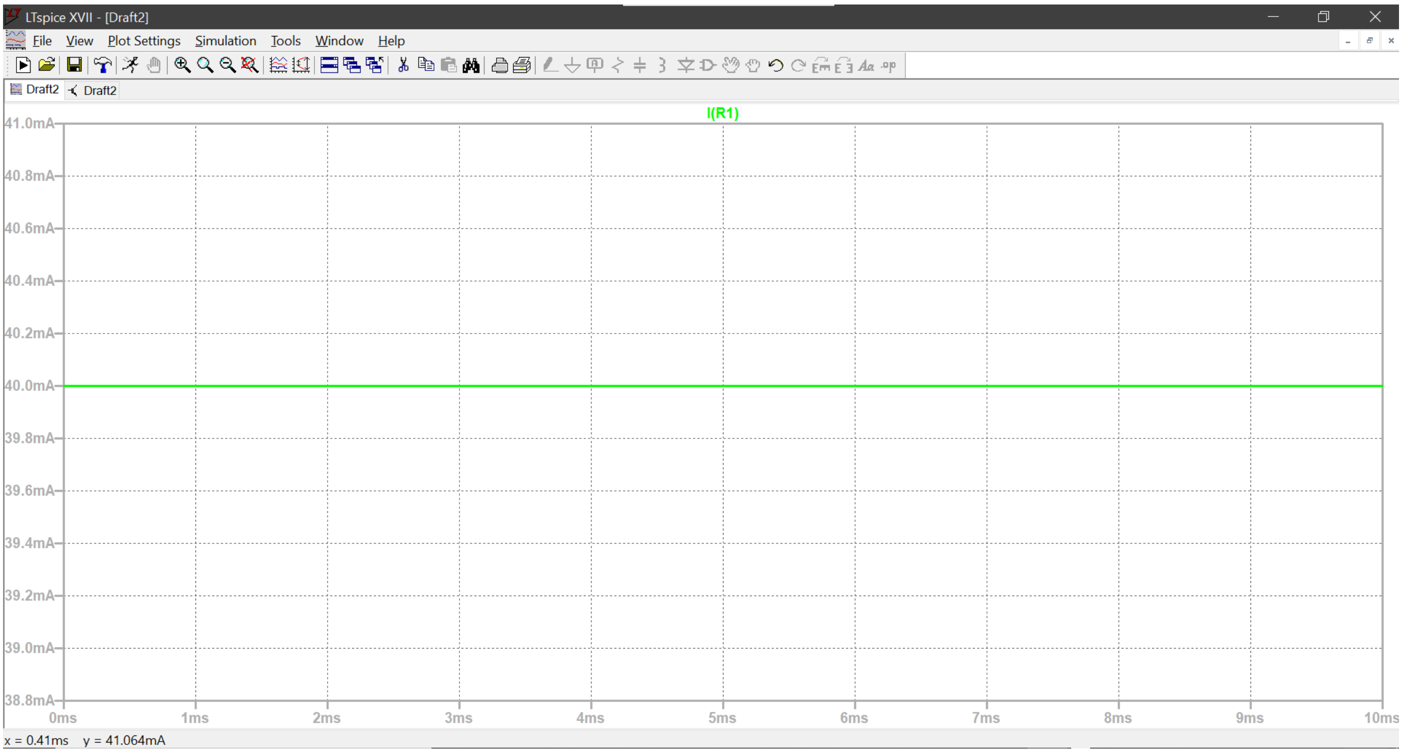


Task-1: Verifying ohms law for a simple circuit

Circuit Diagram:



Stimulated Output: (Transient Analysis)



<u>Theoretical Calculations:</u>

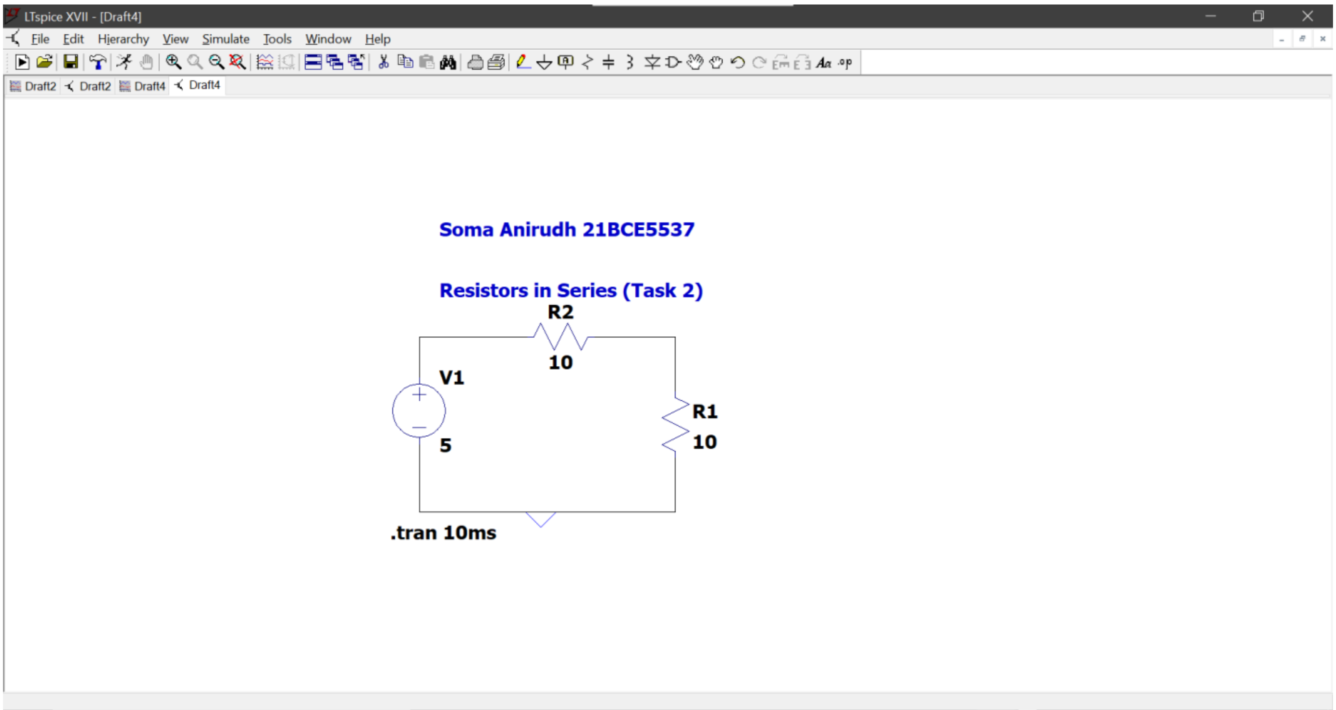
Given Voltage V1=6V and Resistance R1=150 OHM

From Ohm's law, we know that **V=IR** ,(Therefore I=V/R)

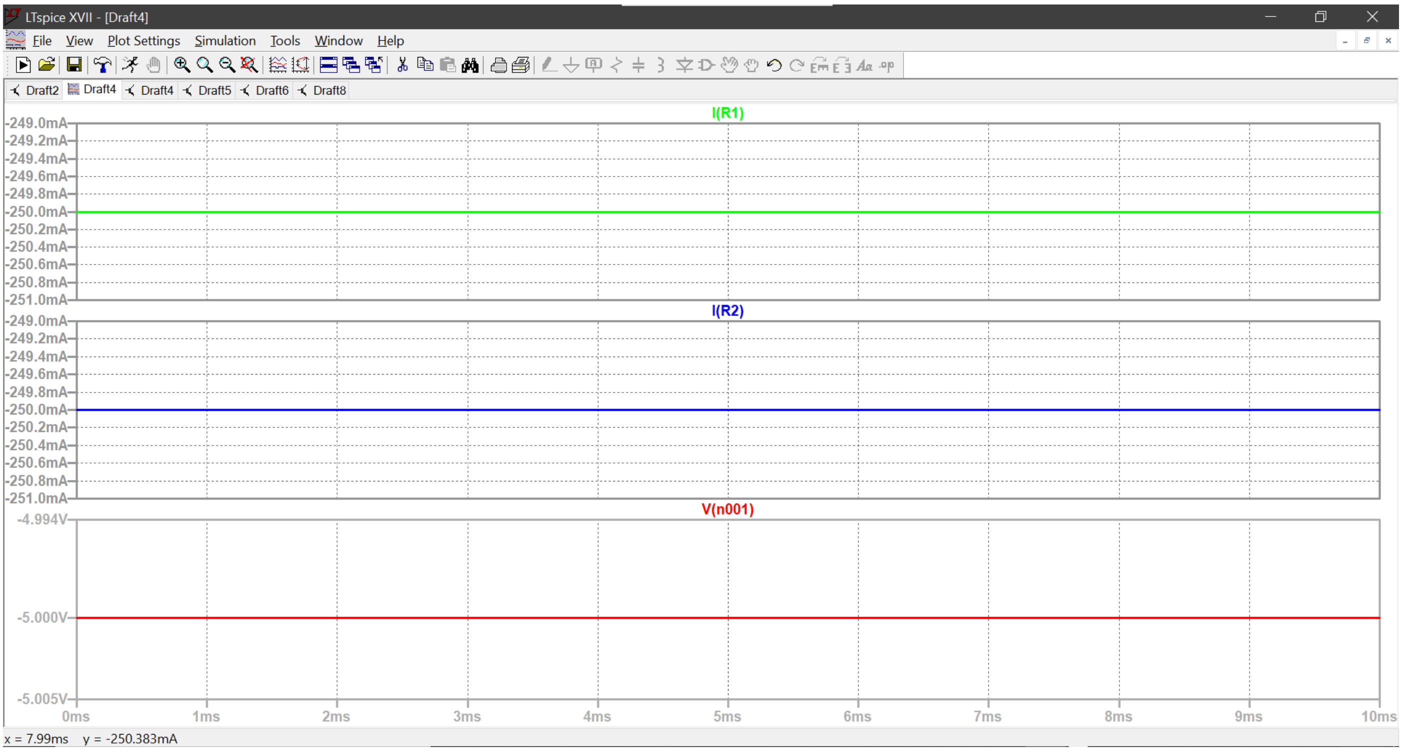
I=6V/150Ω= 0.04A =<u>40Ma</u>

<u>Task-2:</u> Ohms law for a simple series circuit.

<u>Circuit Diagram:</u>



Stimulated Output: (Transient Analysis)



* C:\Users\urwit\OneDrive\Documents\LTspiceXVII\Draft4.asc

```
       --- Operating Point ---

V(n001):        5            voltage
V(n002):        2.5          voltage
I(R2):          0.25         device_current
I(R1):          0.25         device_current
I(V1):          -0.25        device_current
```

<u>Theoretical Calculations:</u>

In Series, the current passing in same I=I1=I2

Req =R1+R2 (Equivalent Resistance for series circuit)=10+10 = 20ohm

Voltage= 5V ; Using ohms law; V=IR and I=V/R

I=V/Req = 5A/20Ω = 0.25A <u>=250Ma</u>

P=VI = 5V X 0.25A = <u>125Mw</u>

We van verify by applying KVL , which gives

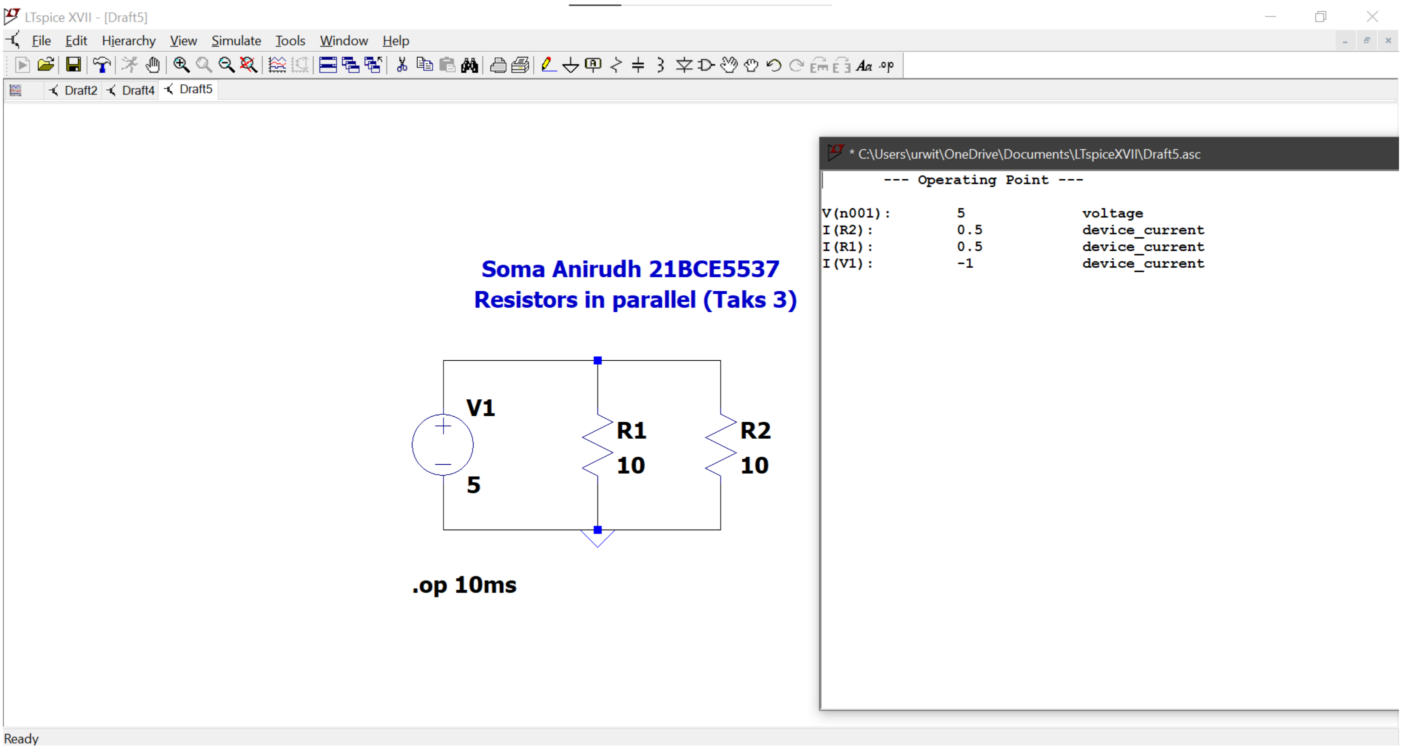
V=I1*R1+I2*R2

5 = 0.25*10+0.25*10

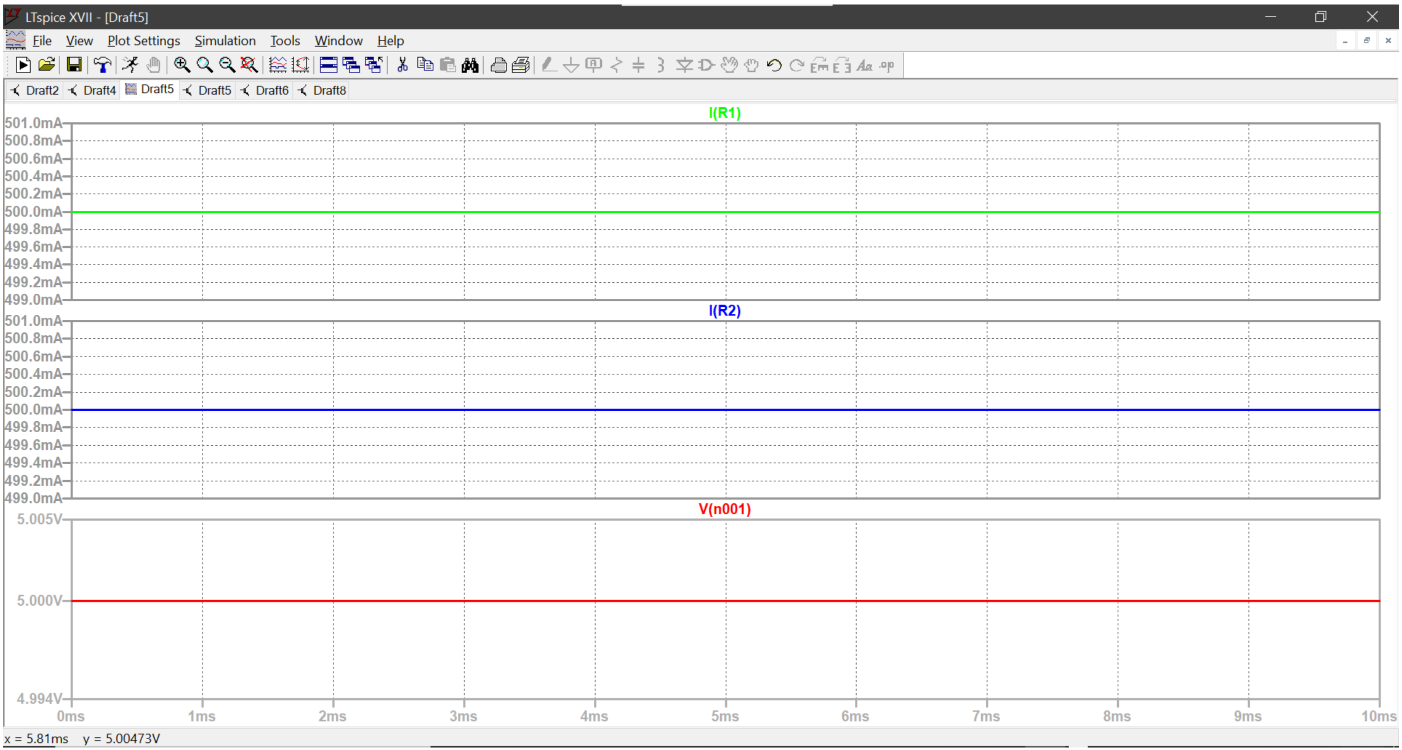
5=5

<u>Task-3:</u> Ohms law for a simple parallel circuit.

<u>Circuit Diagram:</u>



Stimulated Output: (Transient Analysis)



Theoretical Calculations: (For task 3)

In Parallel circuits , Equivalence Resistance is given by

Req=R1*R2/R1+R2

Req = 10*10/20 =5Ω

Voltage= 5V ; Using ohms law; V=IR and I=V/R

TOTAL CURRENT= V/Req= 5V/5Ω = 1A

Individual current can be calculated using ,

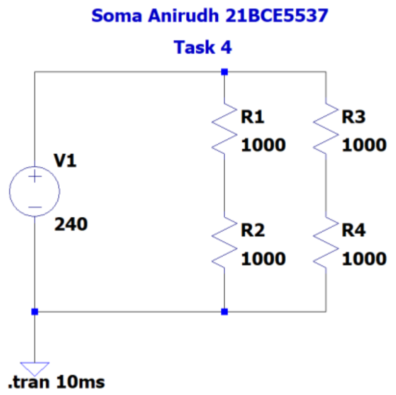
I(R1)=10X1/10+10=0.5A

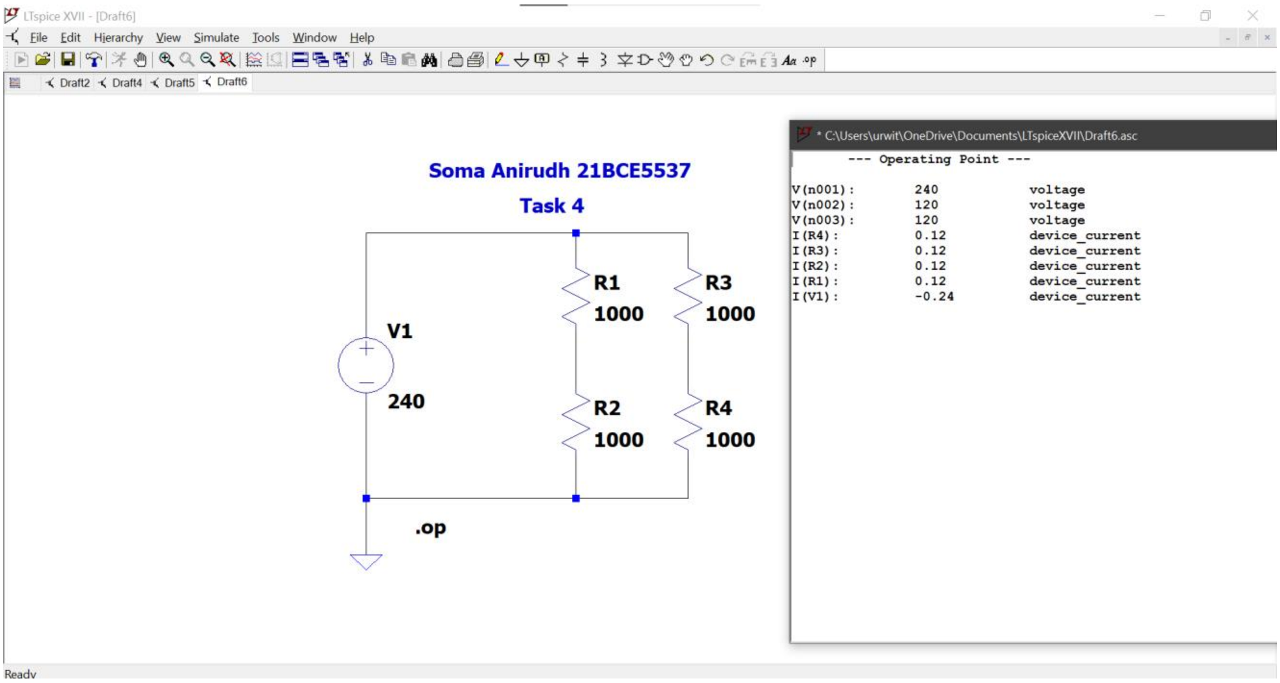
I(R2)= 10X1/10+10=0.5A

P=VI =5V*1A=<u>5W</u>

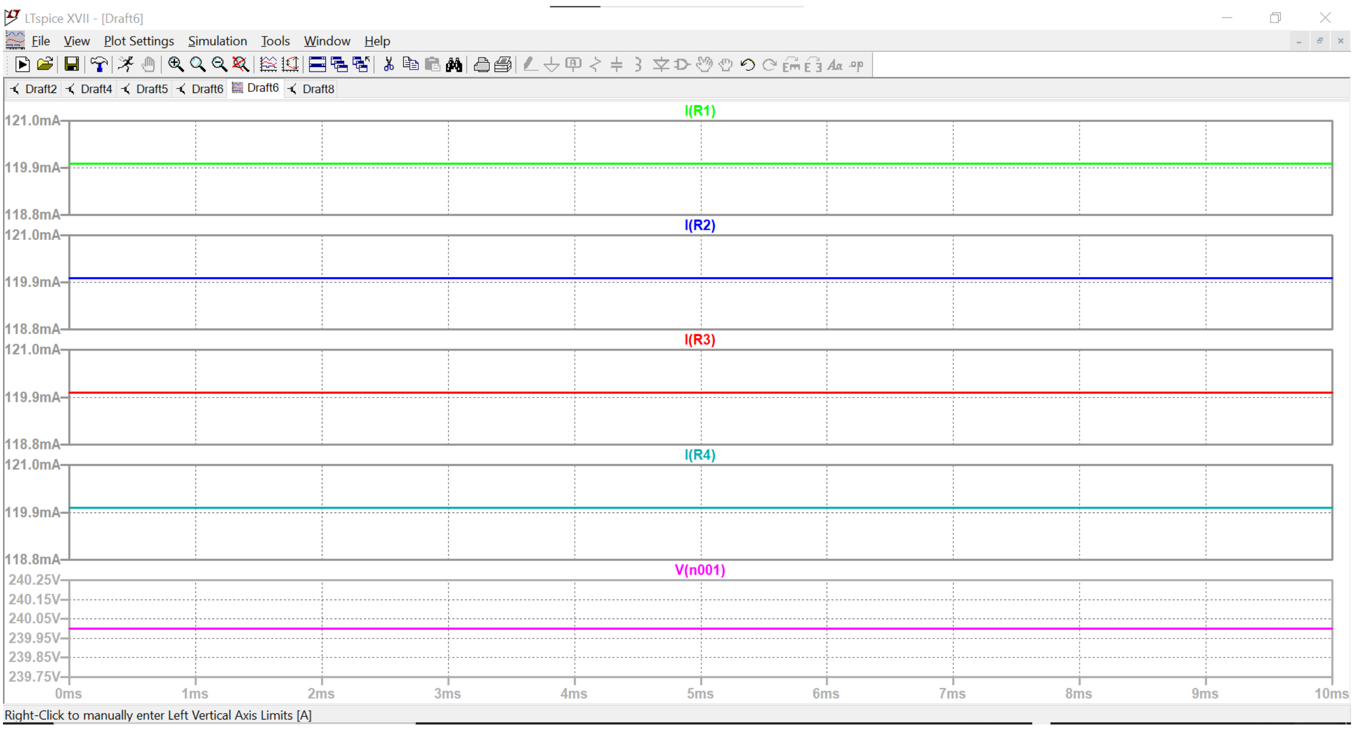
<u>Task-4:</u> Ohms law for a combined circuit.

<u>Circuit Diagram:</u>





Stimulated Output: (Transient Analysis)



Theoretical Calculations:

Equivalent Resistance can be given by

Req=(R1+R2)//(R3+R4) (Since R1,R2 are in series ; R3,R4 are in series; both the series resistances are now in parallel)

R_{12}=R1+R2= 1000Ω+1000Ω=2000Ω

R_{34}=R3+R4= 1000Ω+1000Ω=2000Ω

R_{12} and R_{34} are in parallel; R_{1234}=Req=(2000*2000)/(2000+2000) = 1000Ω

Voltage= 240V ; Using ohms law; V=IR and I=V/R

Total current(I) = 240V/1000Ω = 0.24A= 240mA

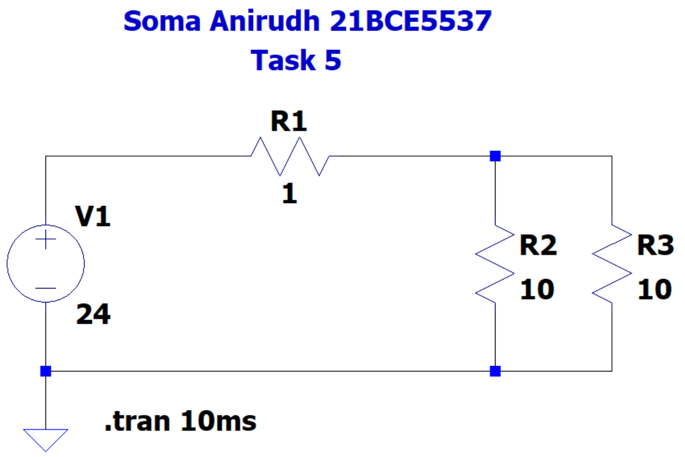
I(R1)=I(R2)=I*(R3+R4)/R1+R2+R3+R4 = 240*2000/4000 = 120mA

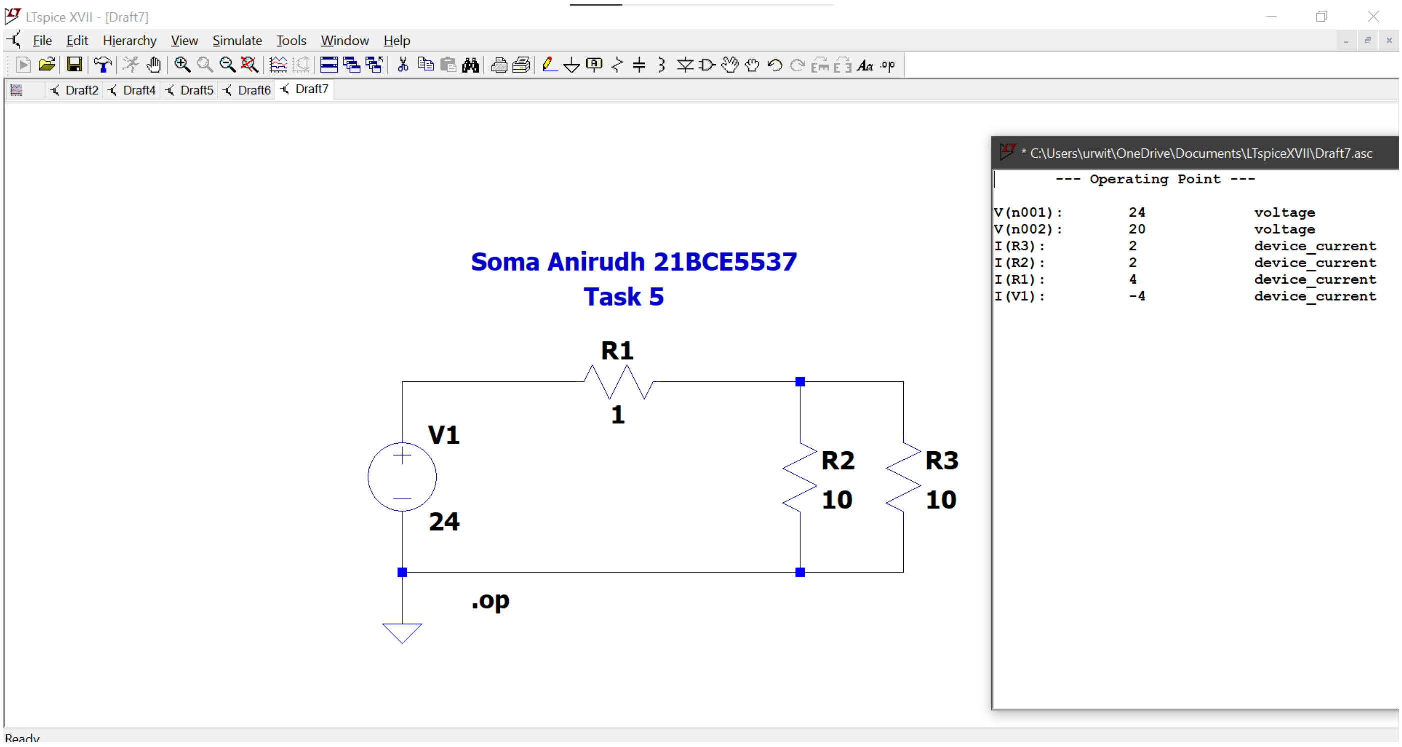
I(R3)=I(R4)= 240*2000/4000 = 120Ma

P=V^2/Req=240x240/1000=57.6W

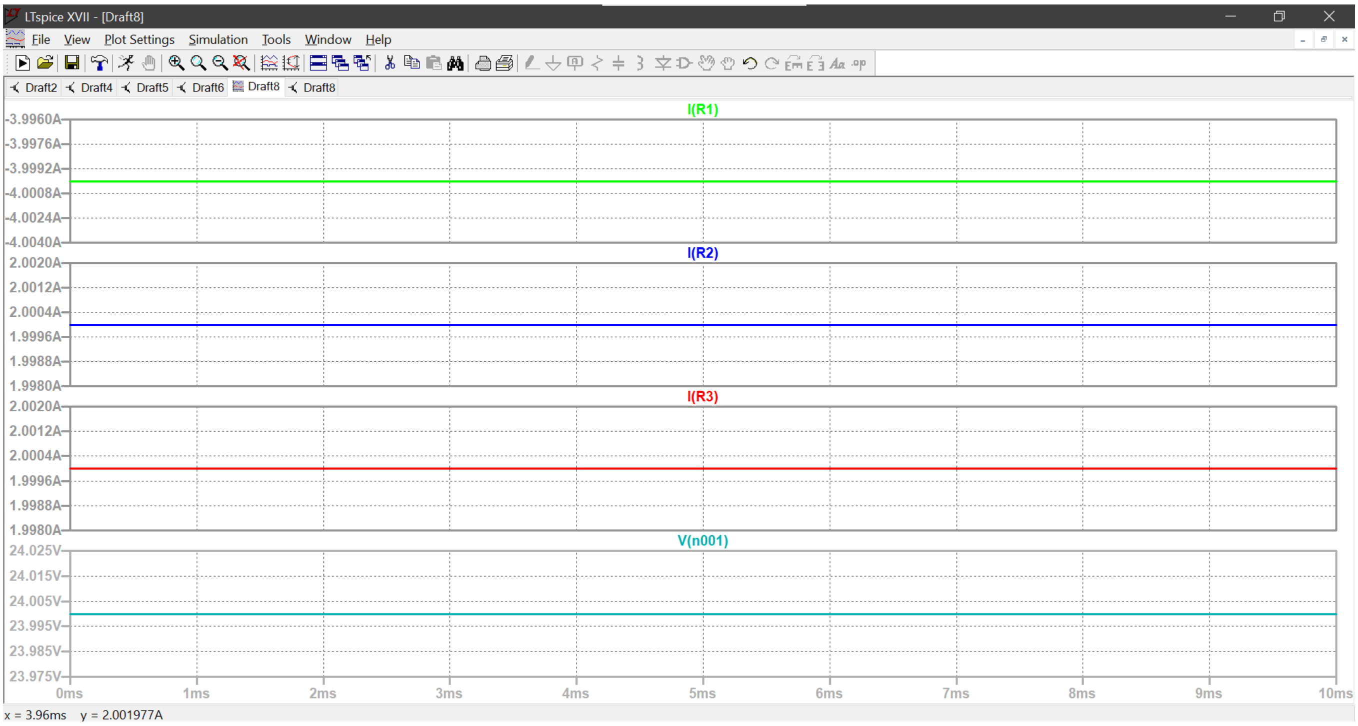
Task-5: Ohms law for a combined circuit.

Circuit Diagram:





Stimulated Output: (Using Transient Analysis)



Theoretical Calculations:

Req=R1+(R2//R3) (R2 and R3 are in parallel and their equivalent is in series with R1)

R_{23}=(R2*R3)/(R2+R3) = (10*10)/(10+10)= 5Ω

Req=R1+R_{23}=1Ω+5Ω=6Ω

Voltage= 24V ; Using ohms law; V=IR and I=V/R

Total current(I)= 24/6=4A

I(R1) = 4A

I(R2)=I(R3)= 4*10/20=2A

POWER(P) = V^2/R = 24*24/6= 96W

Experiment-3

Name: Soma Anirudh

Slot: L19+L20 BECE101P

Date: March 8, 2022

Register number: 21BCE5537

Experiment number: 3

Exp Title : VI Characteristics of PN Junction diode

The general aim of this experiment is to study and understand various characters of a PNJunction Diode through forward bias and reverse bias in LtSpice.

Software Required : LtSpice

Theory:

A diode is usually made from a piece of a semiconductor. It generally acts as a one-wayswitch for current. It allows for current to flow in one direction but not in the direction opposite to it.

Half of the diode is doped as a region and the remaining half is doped as an n half with a pndepletion region between them. The p region is called the anode and the n region is called the cathode. Anode is connected to a conductive terminal whereas the cathode is connected to another conductive terminal.

Forward Bias:

When a diode allows current flow, it is said to be of forward bias. A DC voltage is applied across a diode to bias it.

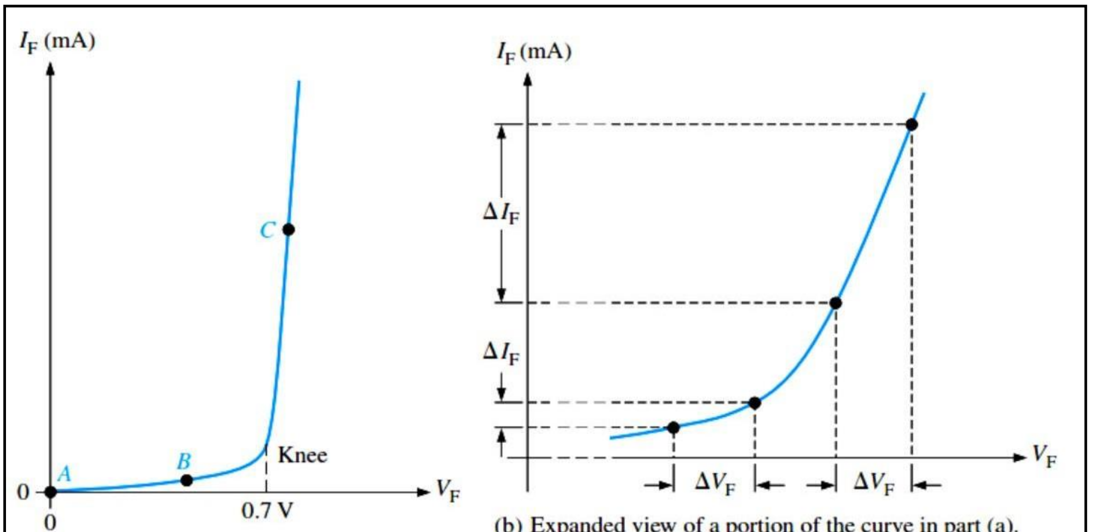
Forward bias refers to the application of a voltage across the device specifically such

that the electrical field at the junction is reduced. By applying a positive voltage to the p-type material and a negative voltage to the n-type material, an electrical field with the direction opposite to the depletion region is applied across the device.

Reverse Bias:

When a diode acts as an insulator and does not allow current to flow through it, it is said to be of reverse bias. Due to the attraction of unlike/opposite charges the positive section of the bias voltage source pulls away the free electrons from the pn junction which were major carriers in the n region. Due to this movement of electrons to the positive side of the voltage source, additional holes are created in the depletion region. This ultimately causes the widening of the depletion and decrease in the number of carriers.

Model Graph:



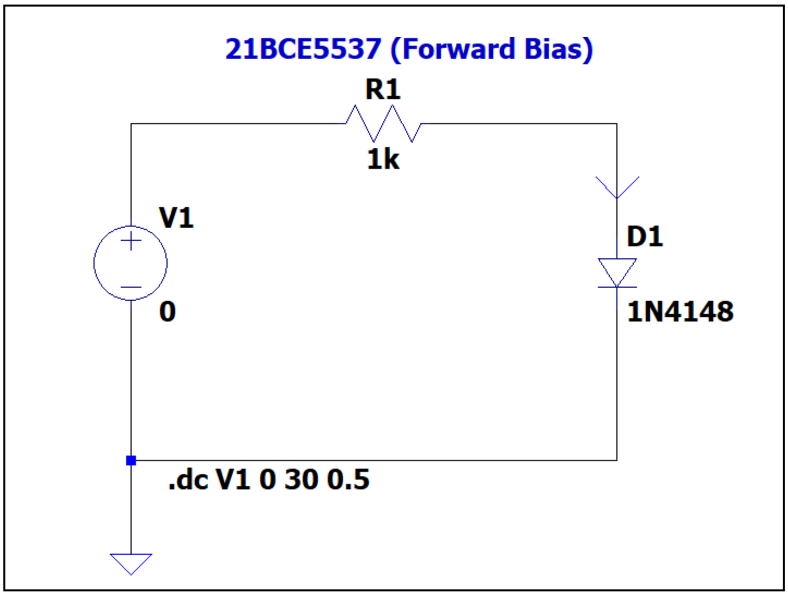
(a) *V-I* characteristic curve for forward bias.

(b) Expanded view of a portion of the curve in part (a). The dynamic resistance r'_d decreases as you move up the curve, as indicated by the decrease in the value of $\Delta V_F / \Delta I_F$.

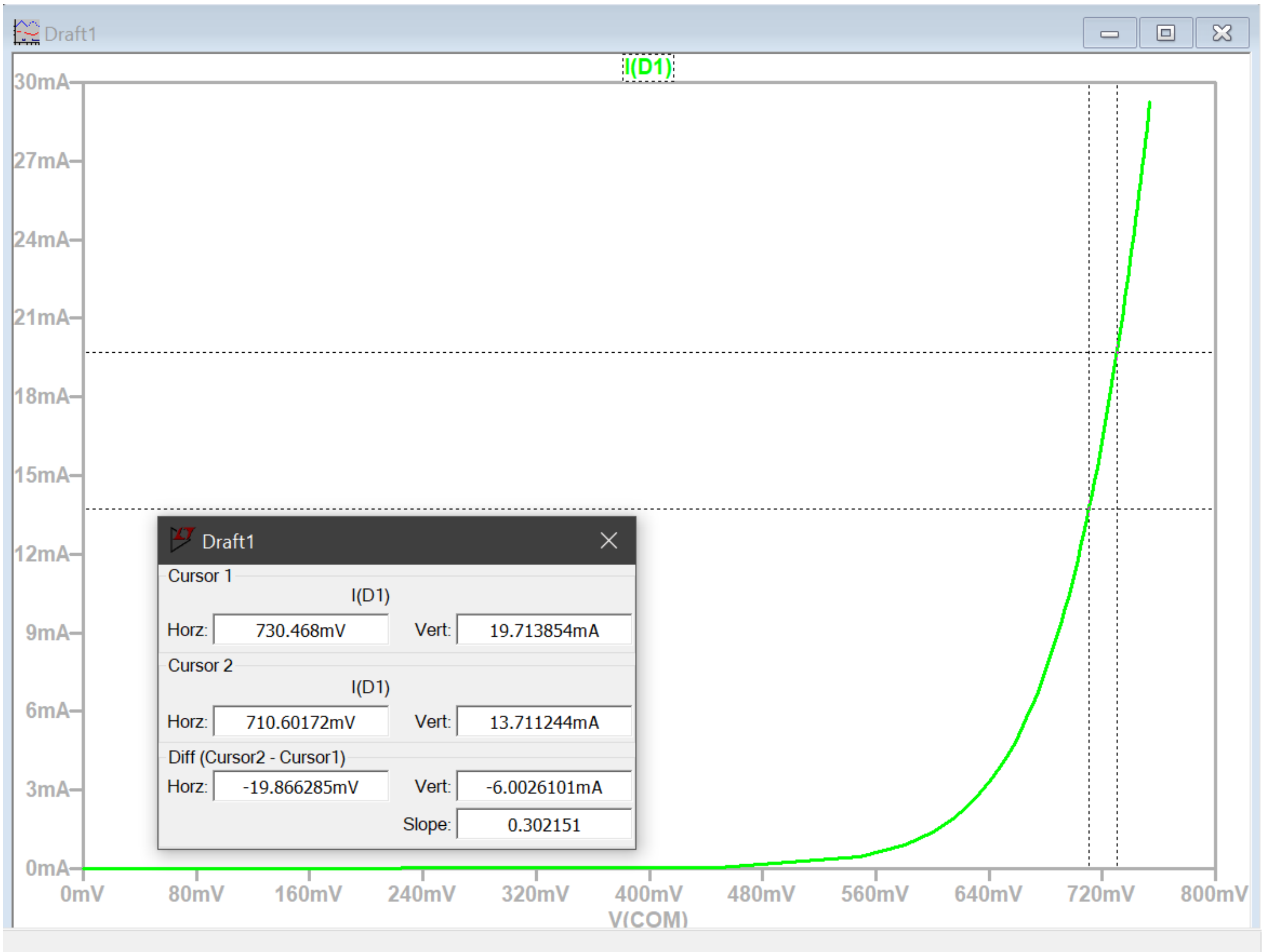
Calculations can be made using:

- Static forward resistance: R(dc)= (Vf)/(If)
- Dynamic forward resistance, R(ac)= (ΔVf)/(ΔIf)

Circuit Diagram (Forward Bias) :



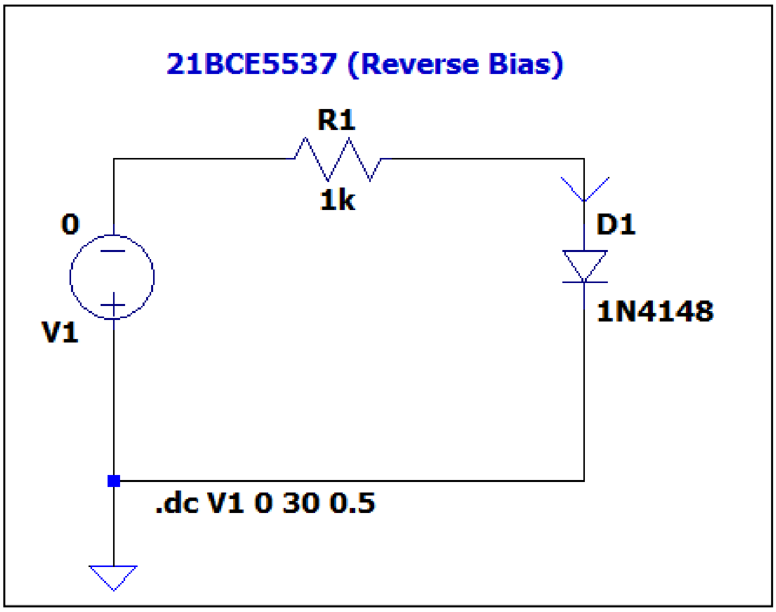
Stimulation output graph (Forward Bias):



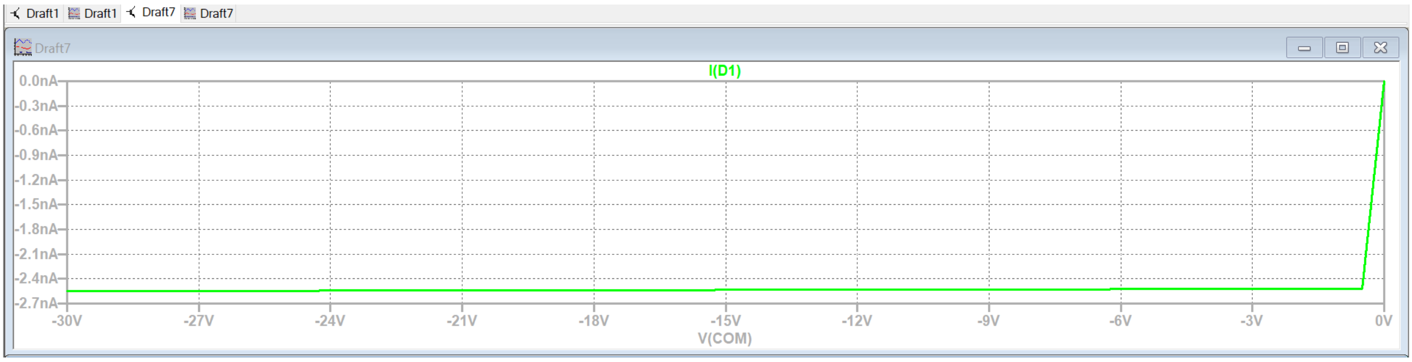
Static Resistance= 19.71385/730.468 = 0.02698797Ω = 26.9879mΩ

Dynamic Resistance= -6.0026101/-19.866285 = 0.302150608 = 302.150608mΩ

Circuit Diagram (Reverse Bias) :



Stimulation output graph (Reverse Bias):



Result:

It can be observed that the in the forward bias graph, it has a rise that increases gradually. Whereas in the graph for reverse bias, it has a sudden peak rise.

Experiment-4

Name: Soma Anirudh

Slot: L19+L20 BECE101P

Date: March 16, 2022

Register number: 21BCE5537

Experiment number: 4

Exp Title : Zener diode

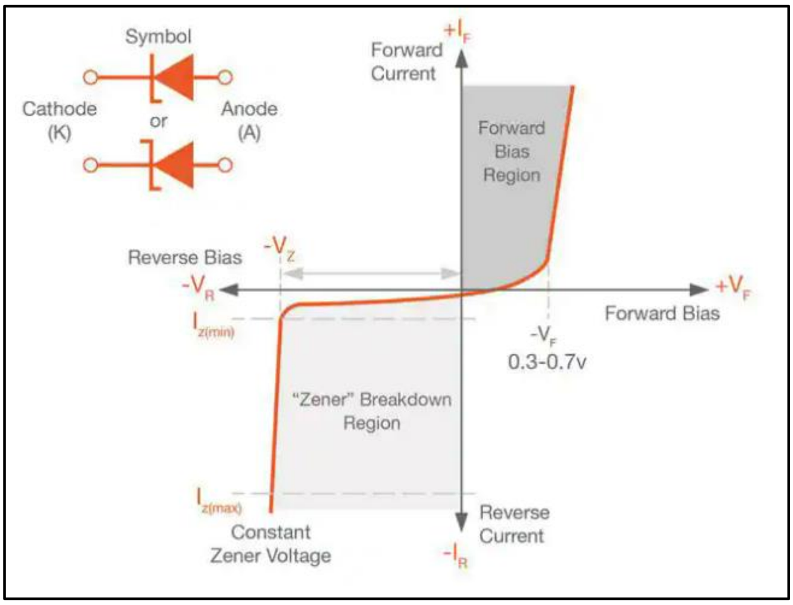
The general aim of this experiment is to study and understand various characters of a Zener Diode in LtSpice.

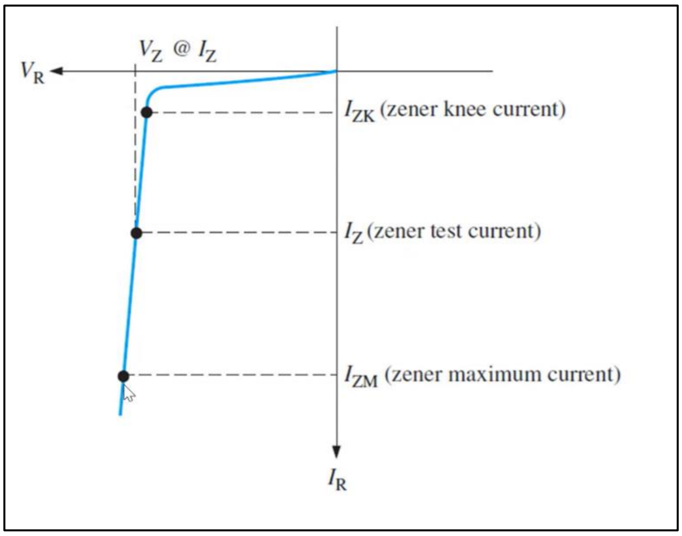
Software Required : LtSpice

Theory:

A Zener diode is a silicon semiconductor device that permits current to flow in either a forward or reverse direction. It is designed for operation in the reverse breakdown region. The diode consists of a heavily doped p-n junction, designed to conduct in the reverse direction when a certain specified voltage is reached.

The Zener diode has a well-defined reverse-breakdown voltage, at which it starts conducting current, and continues operating continuously in the reverse-bias mode without getting damaged. The breakdown voltage is set carefully controlling the doping level during its manufacture. Additionally, the voltage drop across the diode remains constant over a wide range of voltages, a feature that makes Zener diodes suitable for use in voltage regulation.



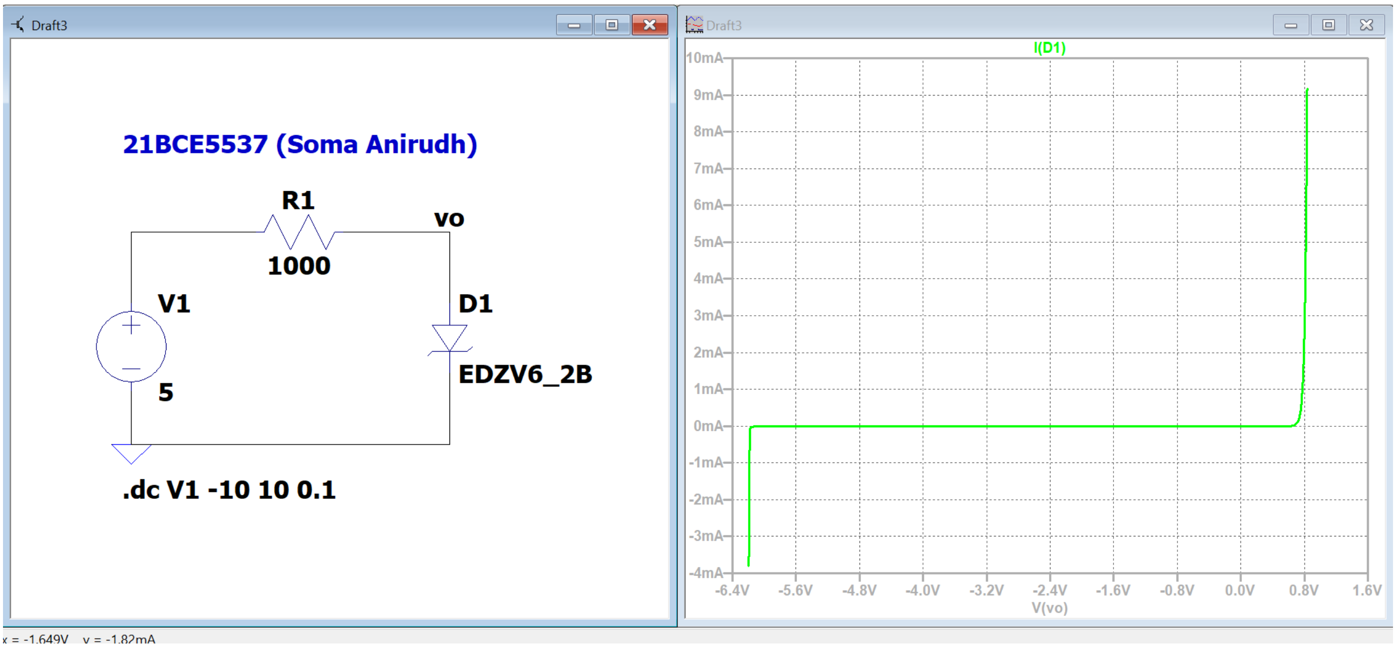


(Breakdown Characteristics)

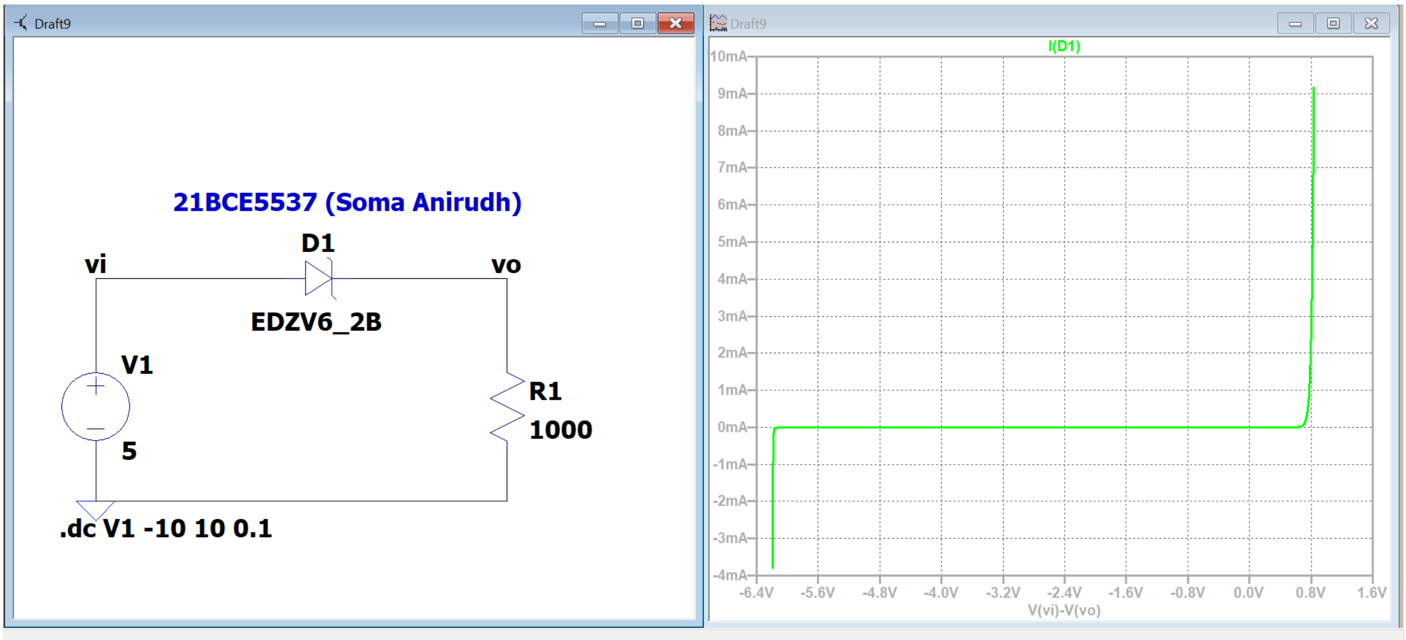
Calculations can be made using:

- **Line Regulation = $(\Delta V_{out})/(\Delta V_{in})$ *100**
  The line regulation specifies the change that occurs in the output voltage for a given change in the input voltage.
- **Load Regulation = $(V_{NL}-V_{FL})/(\Delta V_{FL})$ * 100**
  The load regulation specifies the change in the output voltage over a certain range of load current values usually from minimum current (i.e No load, NL) to maximum current (i.e Full load, FL).

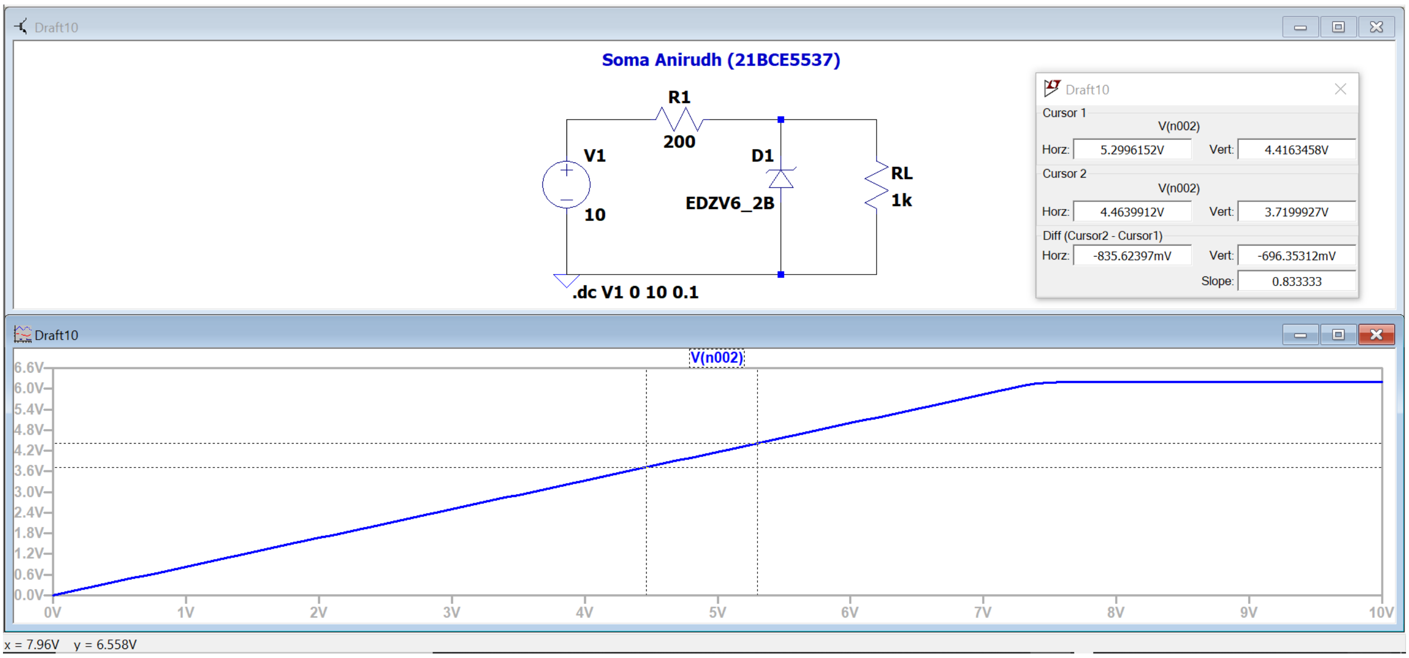
Task 1 (Circuit diagram to the left and respective graph to the right) :



Task 2(Circuit diagram to the left and respective graph to the right):

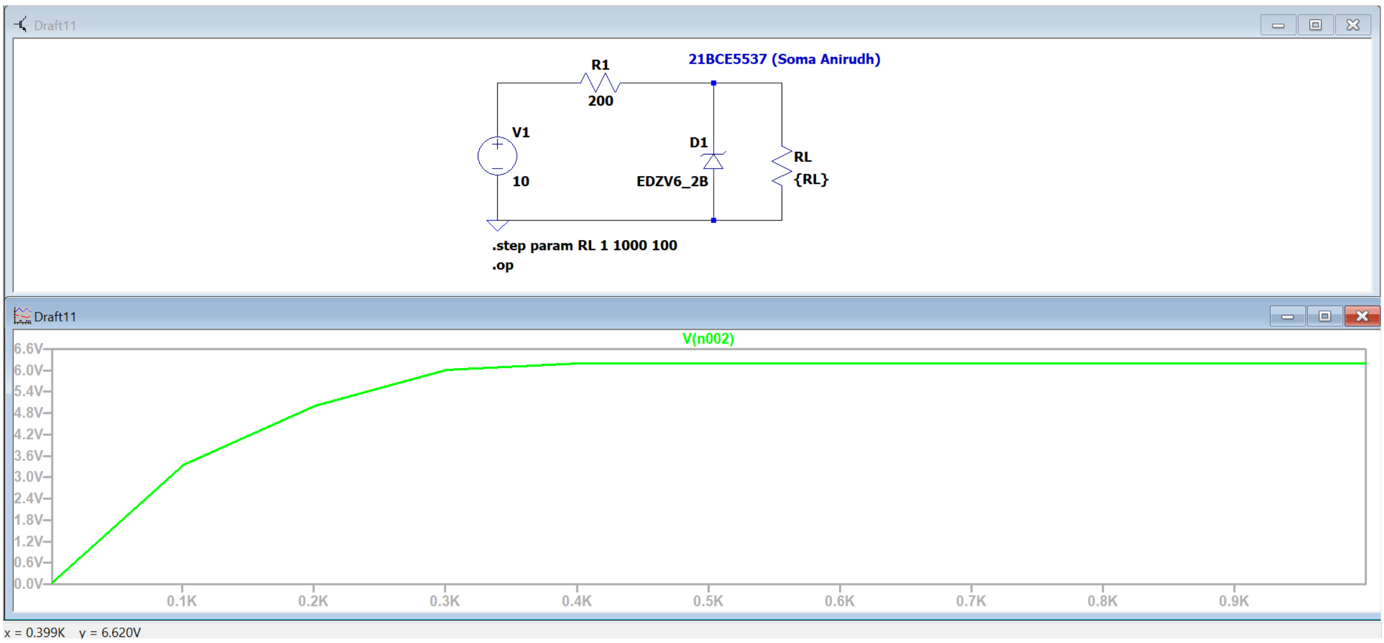


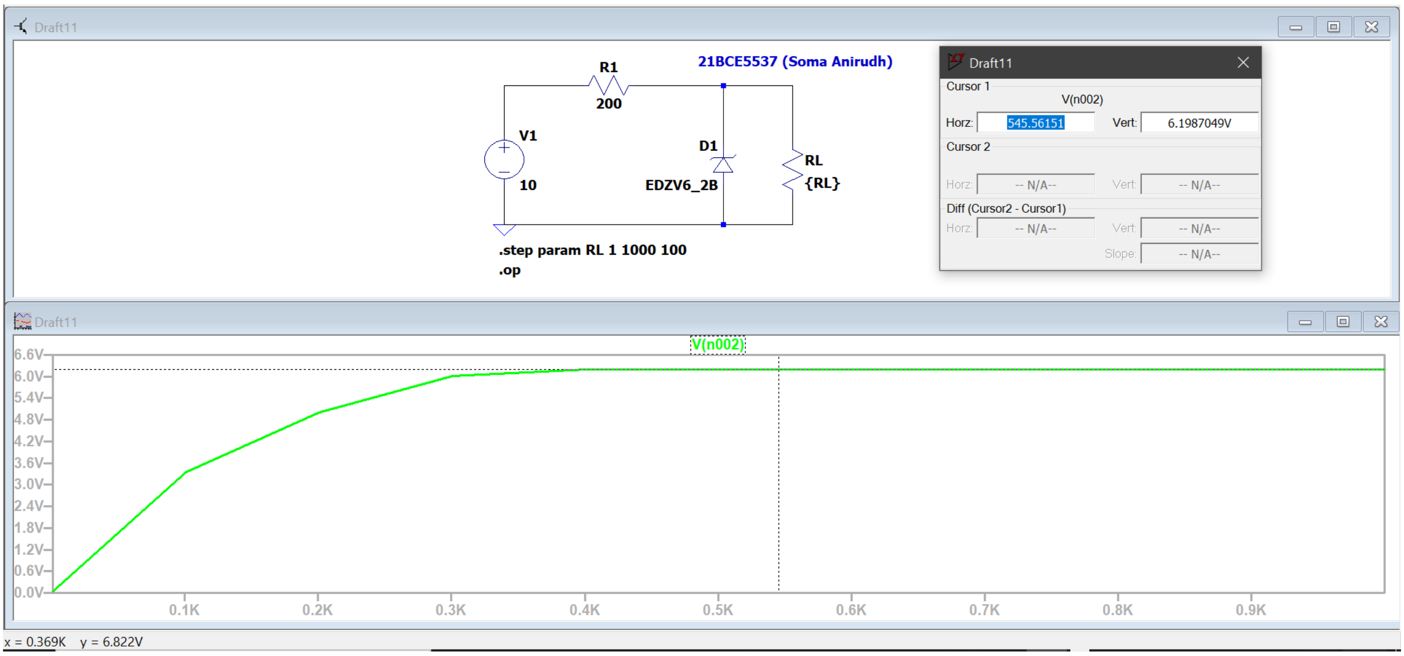
Task 3 (Circuit diagram to the left and respective graph below):

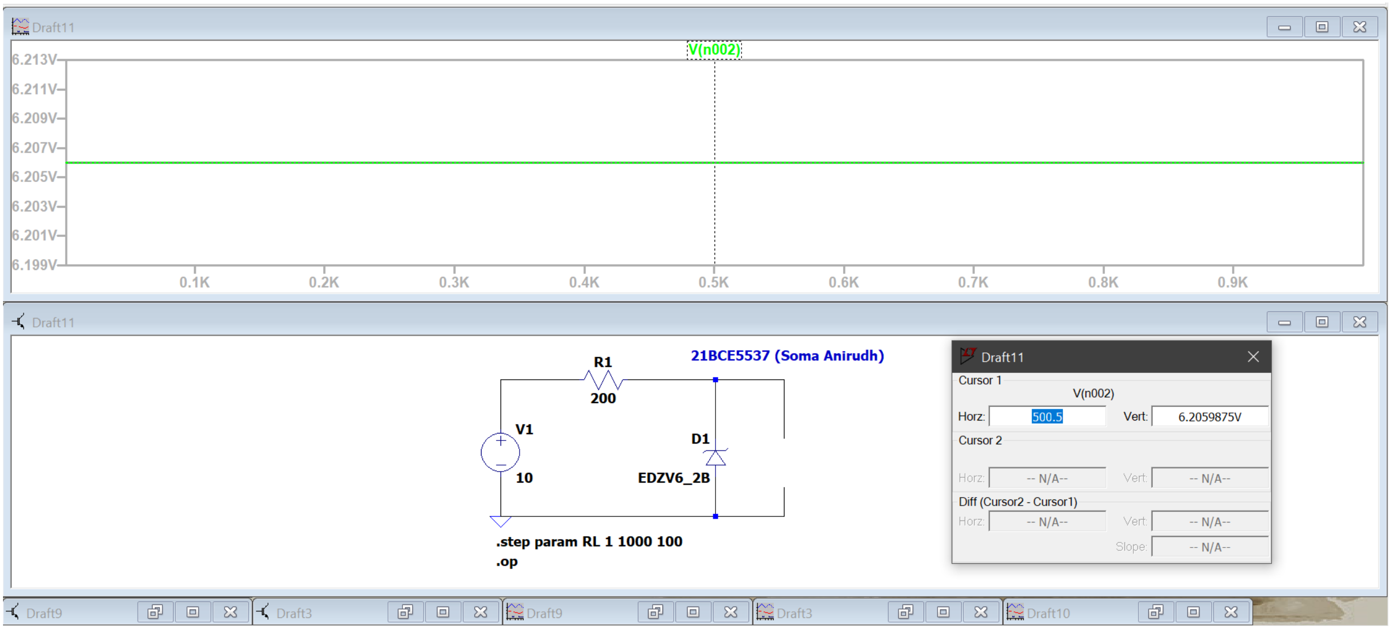


Line regulation = $(V_{f2}-V_{f1})/(V_{i2}-V_{i1}) * 100$ = $83.3*100$ = **83.3%**

Task 4 :







Load regulation= $(V_{nl} - V_{fl})/V_{nl} \times 100$

$=(6.205 - 6.198)/6.205 \times 100 =$ **0.112%**

Experiment-5

Name: Soma Anirudh

Slot: L19+L20 BECE101P

Date: March 23, 2022

Register number: 21BCE5537

Experiment number: 5

Exp Title : Half wave and Full wave rectifier

The general aim of this experiment is to study and understand various characters of a Full wave and Half wave rectifiers in LtSpice.

Software Required : LtSpice

Theory:

A half wave rectifier is a type of rectifier that allows only one half-cycle of an AC power waveform to pass through while blocking the other half. Half-wave rectifiers are used to convert AC electricity to DC voltage and are made up of only one diode.

A rectifier transforms alternating current (AC) into direct current (DC). It's done with the help of a diode or a collection of diodes. Full wave rectifiers use many diodes, whereas half wave rectifiers use only one.

The negative component of the input voltage is rectified to a positive voltage, which is then converted to DC (pulse current) using a diode bridge setup. Half-wave rectification, on the other hand, uses a single diode to remove only the negative voltage component before converting to DC. The waveform is then flattened by charging and discharging a capacitor, yielding a clean DC signal.

Full-wave rectification, on the other hand, is a more efficient way than half-wave rectification because the entire waveform is utilised. In addition, the ripple voltage that appears after smoothing varies based on the capacitor's capacitance and the load. With the same capacitance and load, full-wave rectification produces lower ripple voltage than half-wave rectification.

Stability is inversely proportional to the ripple voltage. Smaller the Ripple voltage , higher is the stability of the circuit.

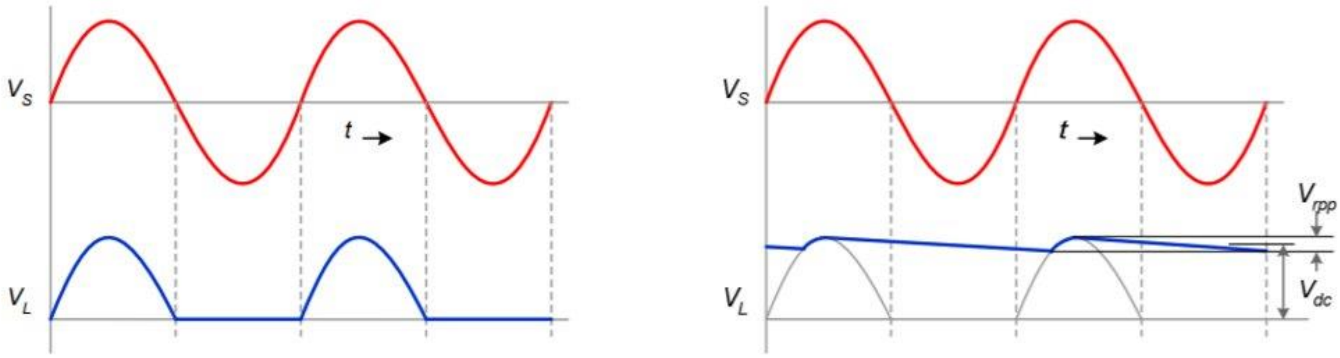
Filters are components that transform pulsing DC waveforms to continuous DC waveforms (smoothing). They accomplish this by suppressing the waveform's DC ripples.

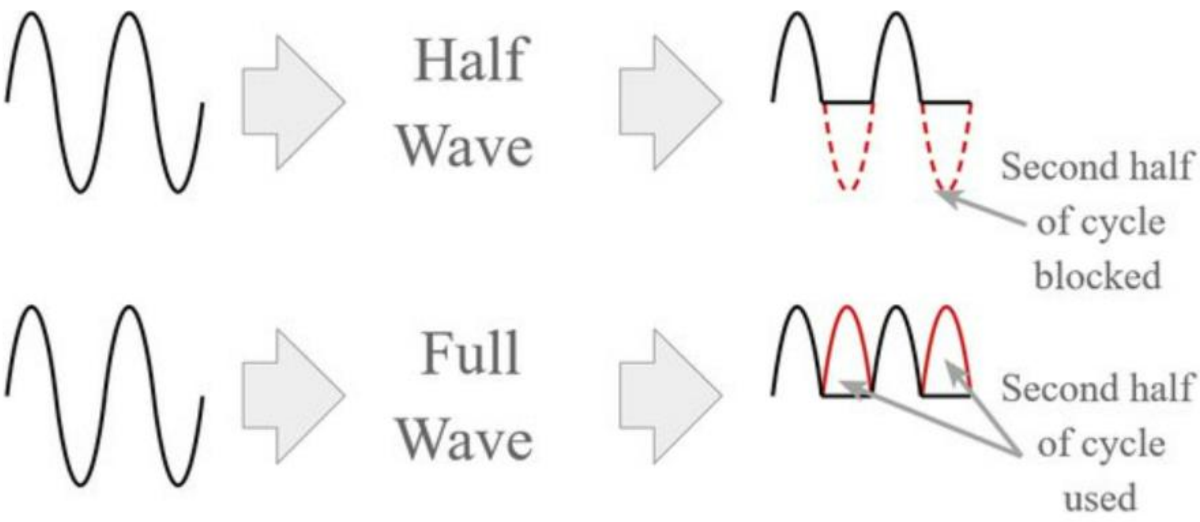
Although half-wave rectifiers without filters are theoretically possible, they are unsuitable for practical usage. Because DC equipment demands a steady waveform, this pulsating waveform must be smoothed down before it can be used in the real world. This is why, in practise, half-wave rectifiers with a filter are used. A filter can be made out of a capacitor or an inductor, but the most common configuration is a half-wave rectifier with a capacitor filter.

Model Graphs:

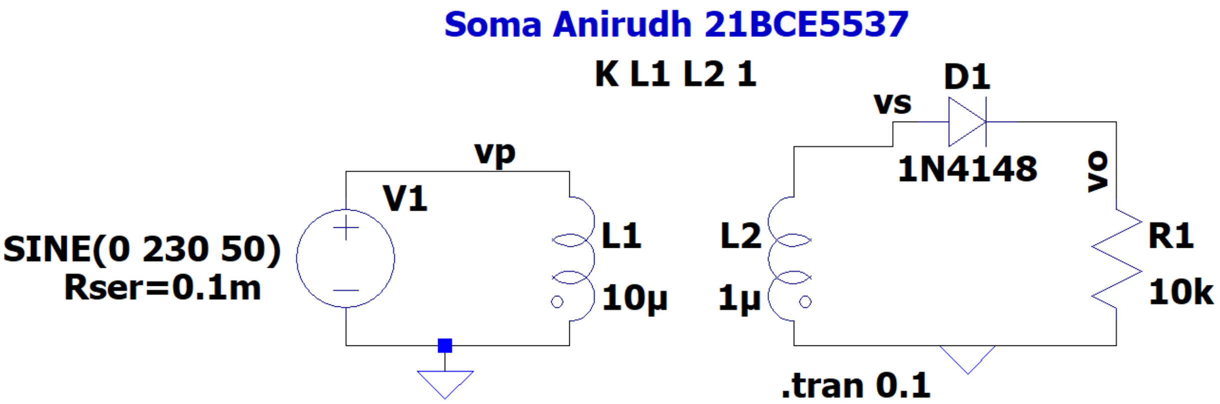
WAVEFORMS

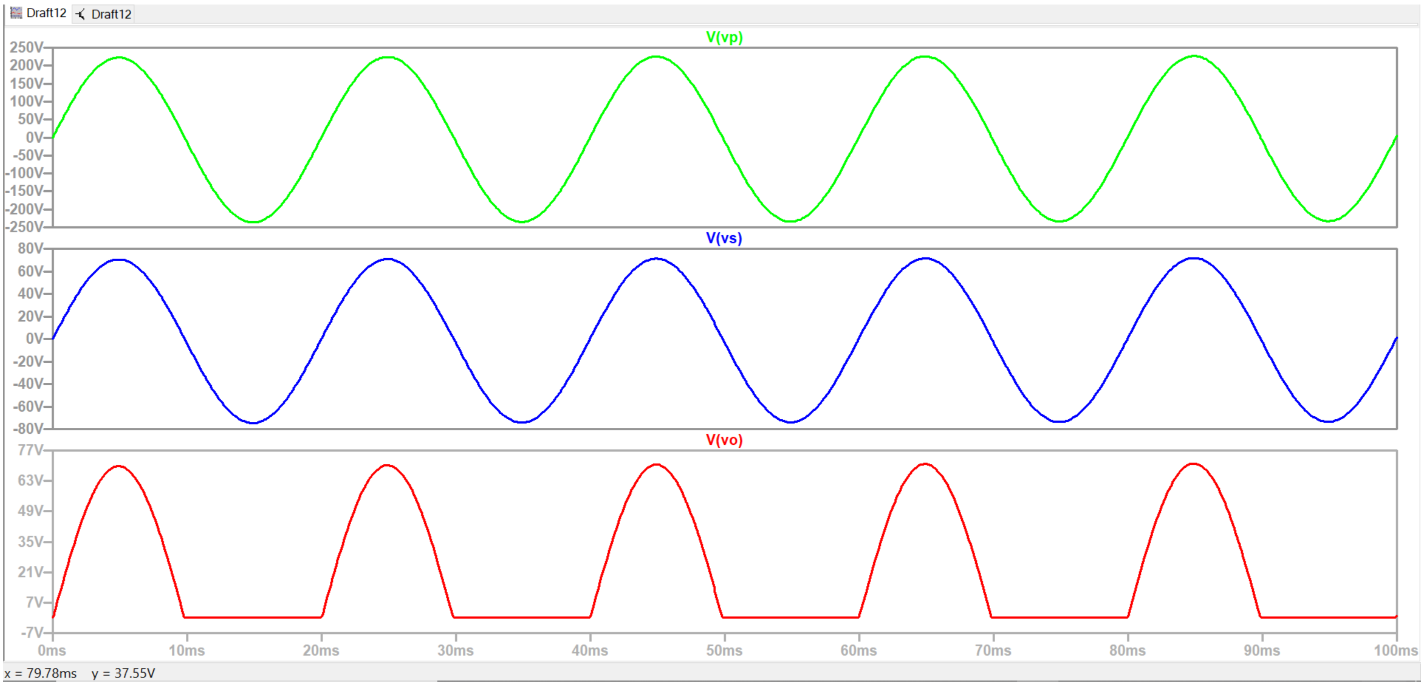
Typical waveforms of half wave rectifier without filter and with filter are shown in the figure below



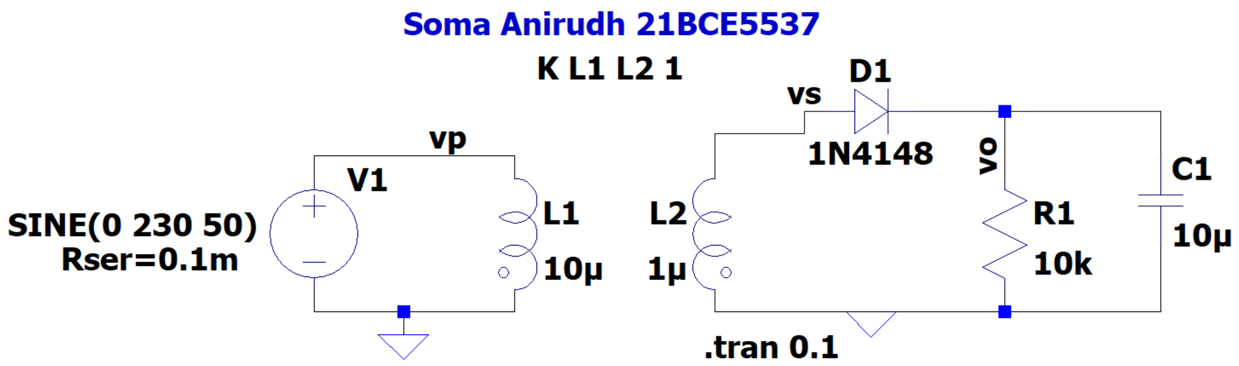


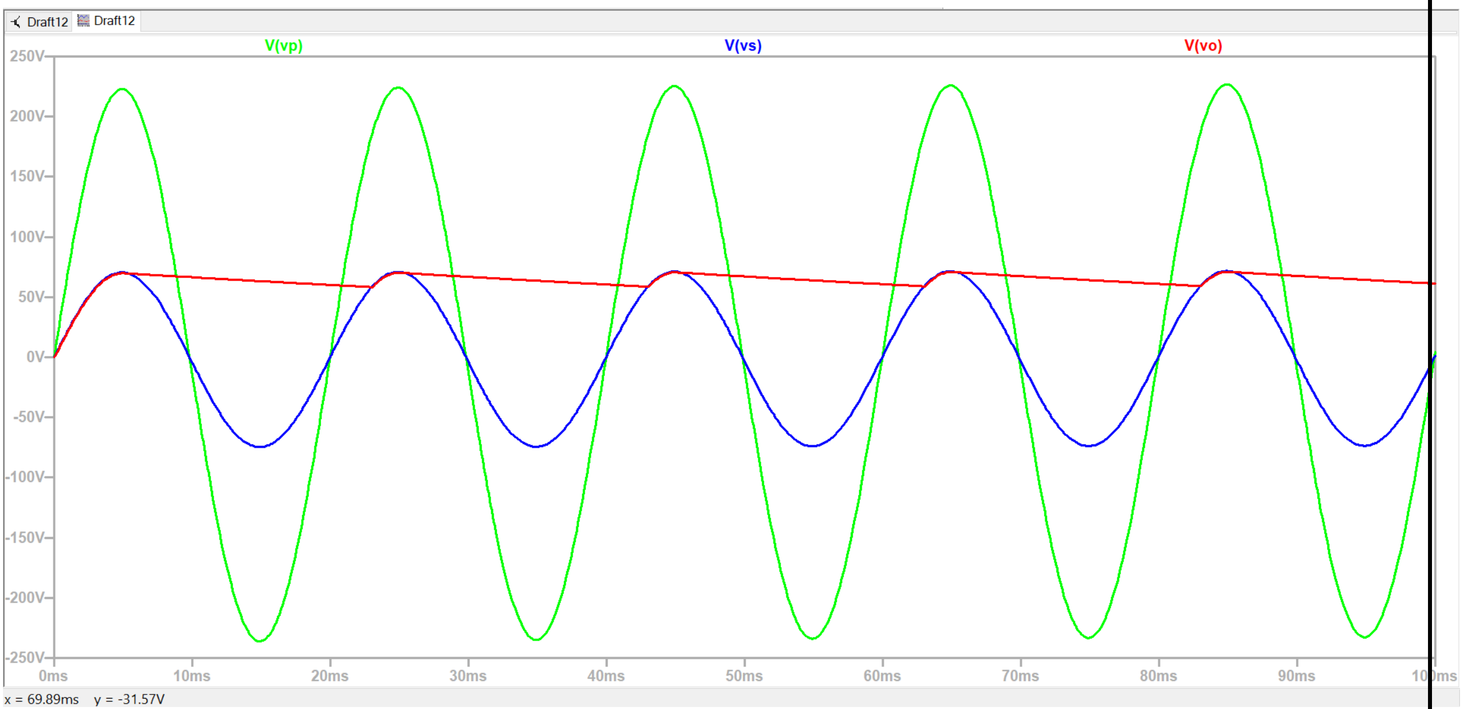
Task-1 (Half wave rectifier without capacitor):



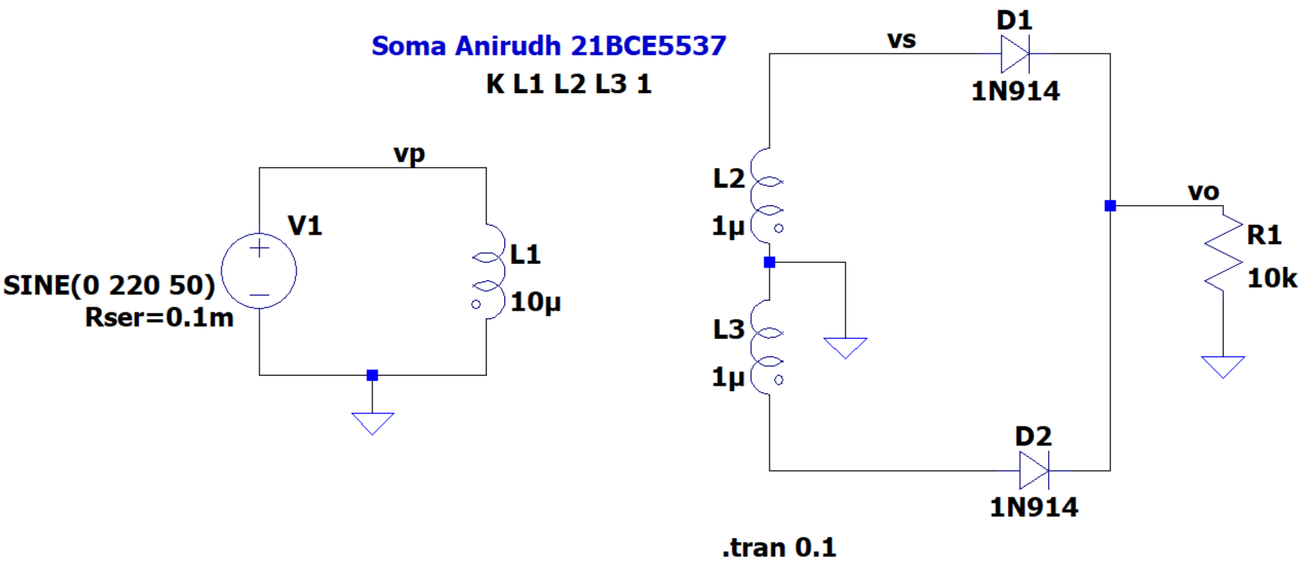


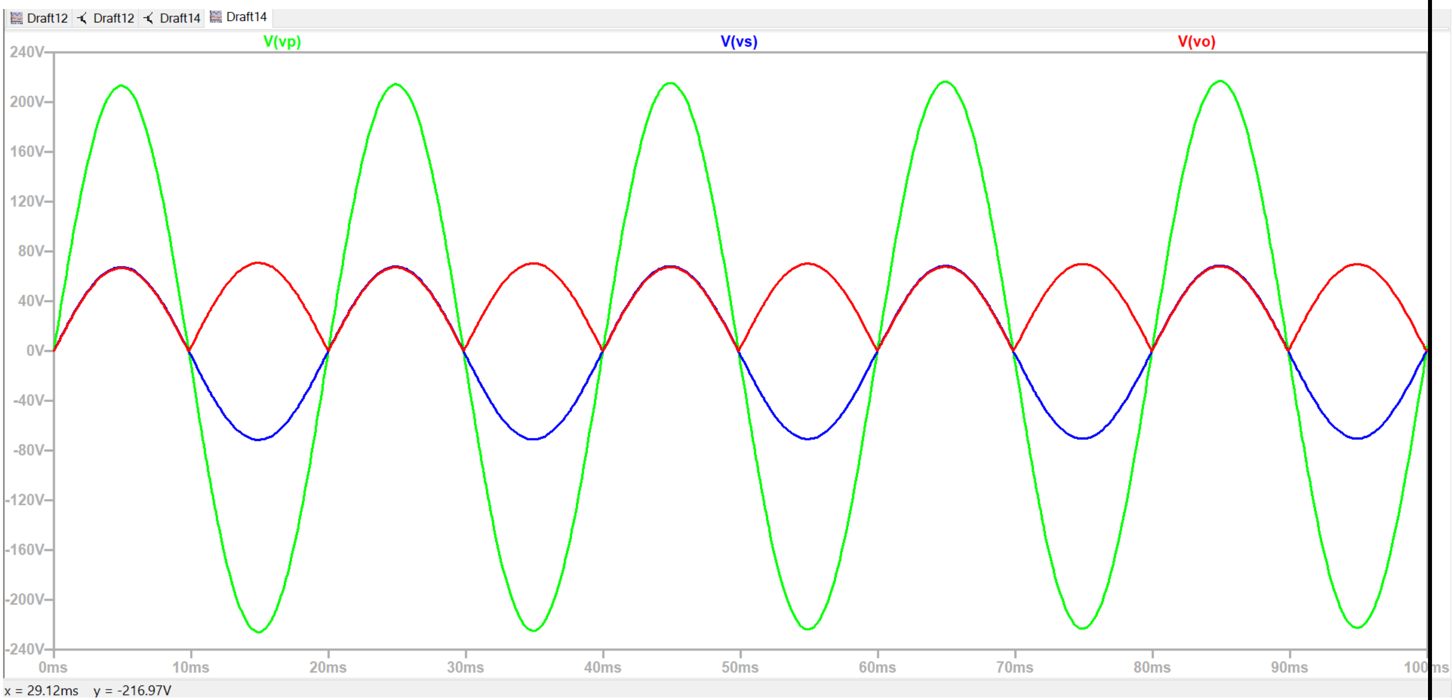
Task-2 (Half wave rectifier with capacitor):



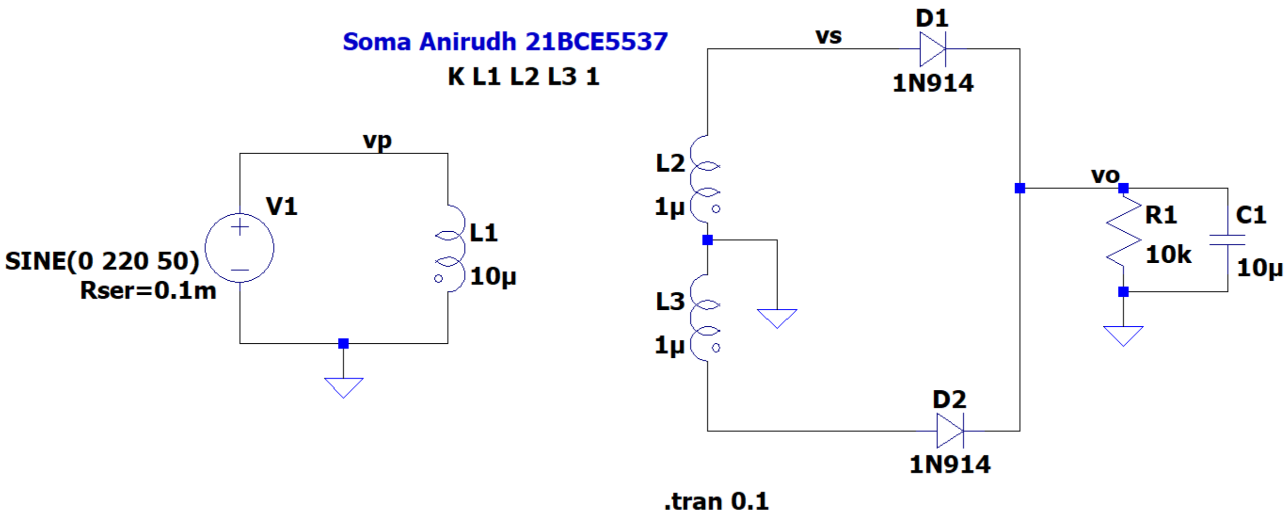


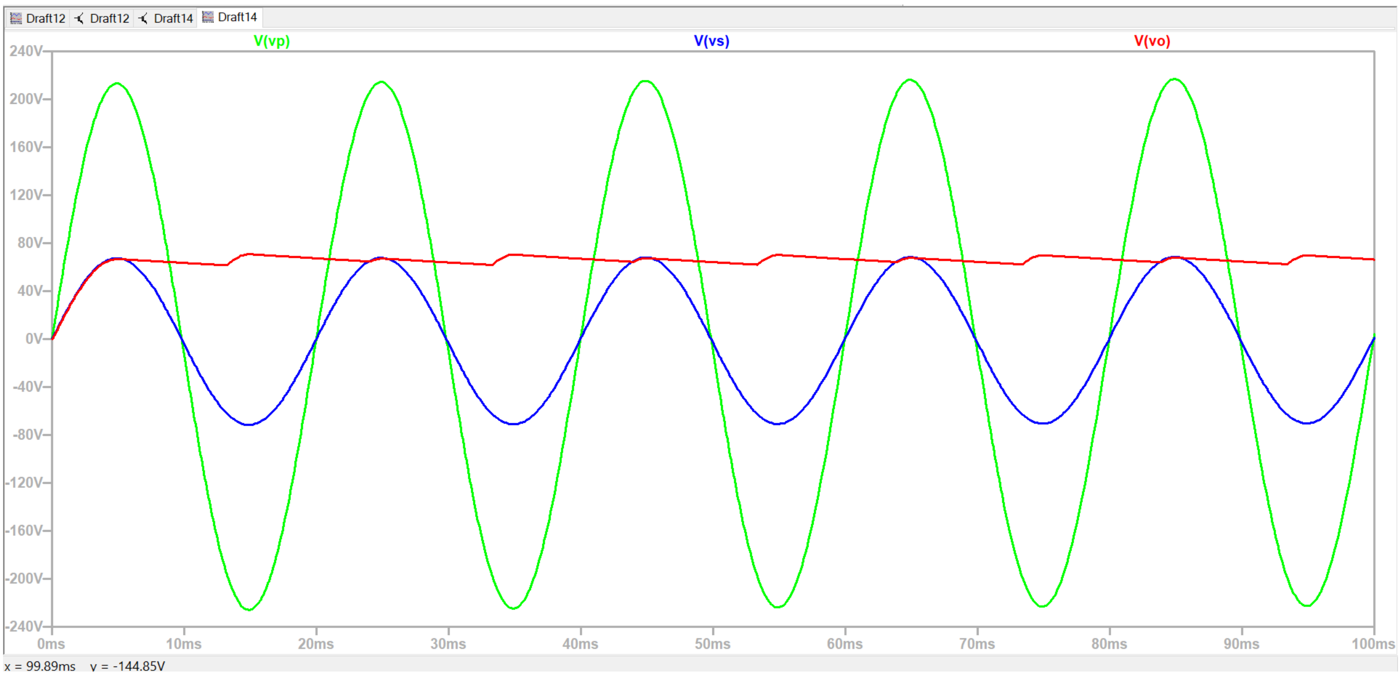
<u>Task-3</u> (Full wave rectifier without capacitor):





Task-4 (Full wave rectifier with capacitor):





Calculations:

Rectifiers without filter:

HWR	**V_m**	**$V_{RMS} = V_m/2$**	**$V_{DC} = V_m/\pi$**	**$R=\sqrt{(V_{RMS}/V_{DC})^2-1}$**
	70.777	35.385	22.55	1.213
FWR	**V_m**	**$V_{RMS} = V_m/\sqrt{2}$**	**$V_{DC} = 2V_m/\pi$**	**$R=\sqrt{(V_{RMS}/V_{DC})^2-1}$**
	70.112	35.055	22.32	1.211

Rectifiers with capacitor filter:

Type	**V_m**	**V_{rpp}**	**$V_{r,rms}= V_{rpp}/2\sqrt{3}$**	**$V_{DC} = V_m – V_{rpp}/2$**	**$r = V_{r,rms}/V_{DC}$**
HWR	70.488	11.52	3.32	64.962	0.0472
FWR	70.435	5.56	1.605	67.673	0.0237

Experiment-6

Name: Soma Anirudh

Slot: L19+L20 BECE101P

Date: March 26, 2022

Register number: 21BCE5537

Experiment number: 6

Exp Title : BJT Characteristics

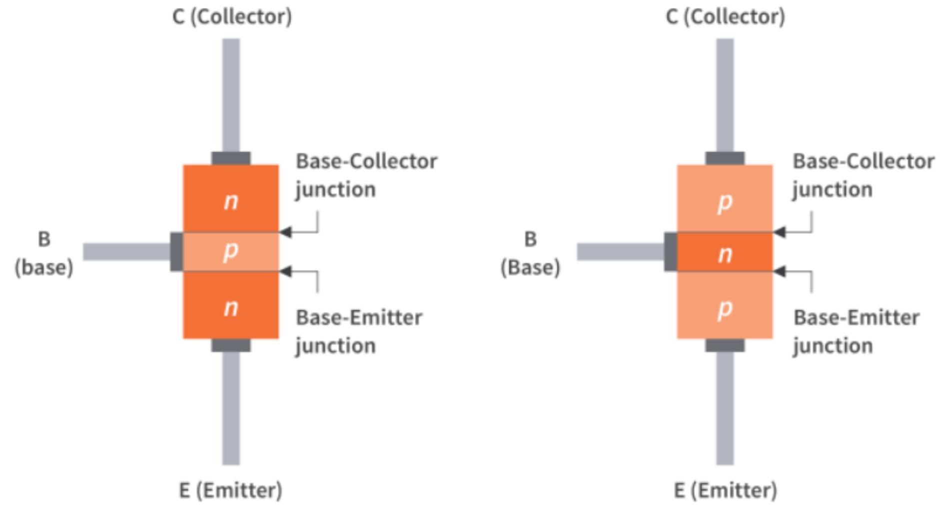
The general aim of this experiment is to study and understand various characteristics of a bipolar junction transistor.

Software Required : LtSpice

Theory:

The bipolar junction transistor (BJT) is made up of three differentially doped semiconductor regions. The base, collector, and emitter are the three regions that are doped differentially. In comparison to the collector and emitter regions, the base region is lightly doped and very thin. The emitter region is substantially doped, while the collecting zone is mildly doped.

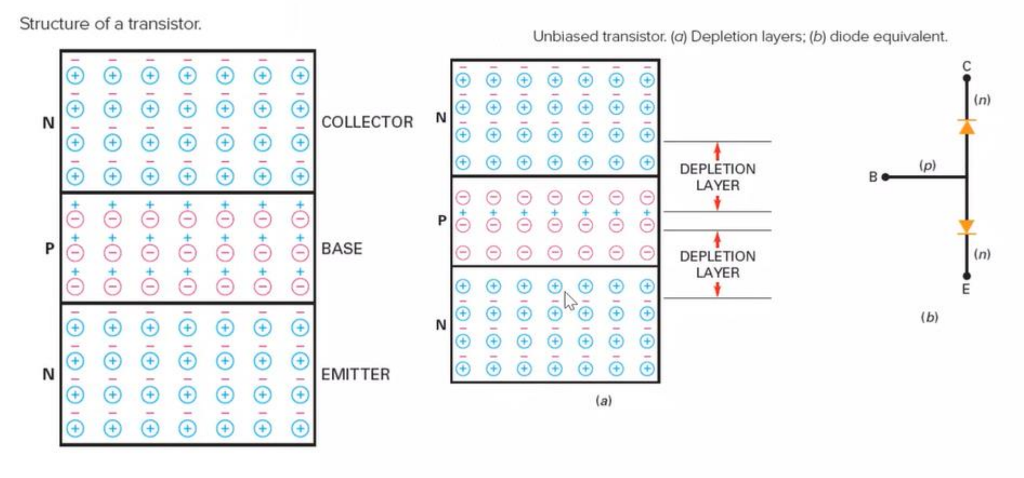
Bipolar junction transistors come in two types: **NPN** and **PNP**. Two n regions are separated by a p region of the NPN type. P-type materials are used in the base region, whereas n-type materials are used in the collector and emitter regions. The transistor is made up of two p-type areas, the collector and emitter, separated by an n-type base region of the PNP type.



NPN and PNP BJT Physical Representation

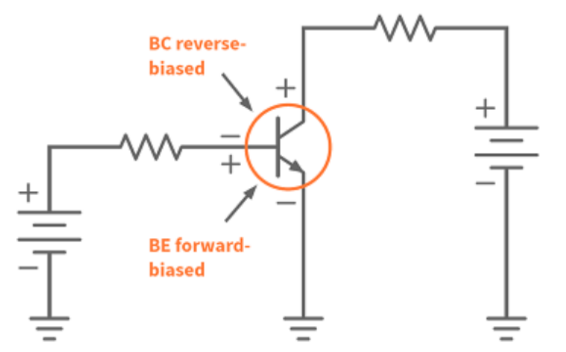
A BJT has two pn junctions that must be correctly biased by an external DC voltage to function effectively, regardless of the type. The base-emitter junction, which connects the base and emitter regions, and the base-collector junction, which connects the base and collector regions, are two of these junctions.

Unbiased transistor is a transistor with its terminals not connected to any source. Biasing a transistor is applying a suitable DC voltage across the transistor terminals to operate the transistor in the desired region.

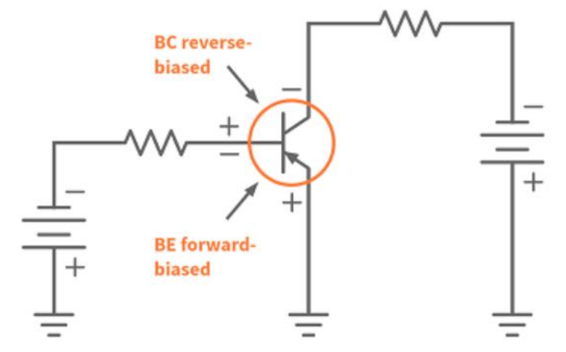


The base-emitter junction (BE) is forward biased and the base-collector junction (BC) is reverse biased.

The base-emitter junction of a bipolar junction transistor must be forward-biased while the base-collector junction must be reverse-biased in order for it to function as an amplifier - please note that this means that a npn transistor and a pnp transistor are backwards in comparison. Furthermore, as previously stated, the emitter region is extensively doped. The n-type emitter zone of a npn transistor has a very high density of free electrons, whereas the p-type emitter region of a pnp transistor has a very high density of holes.



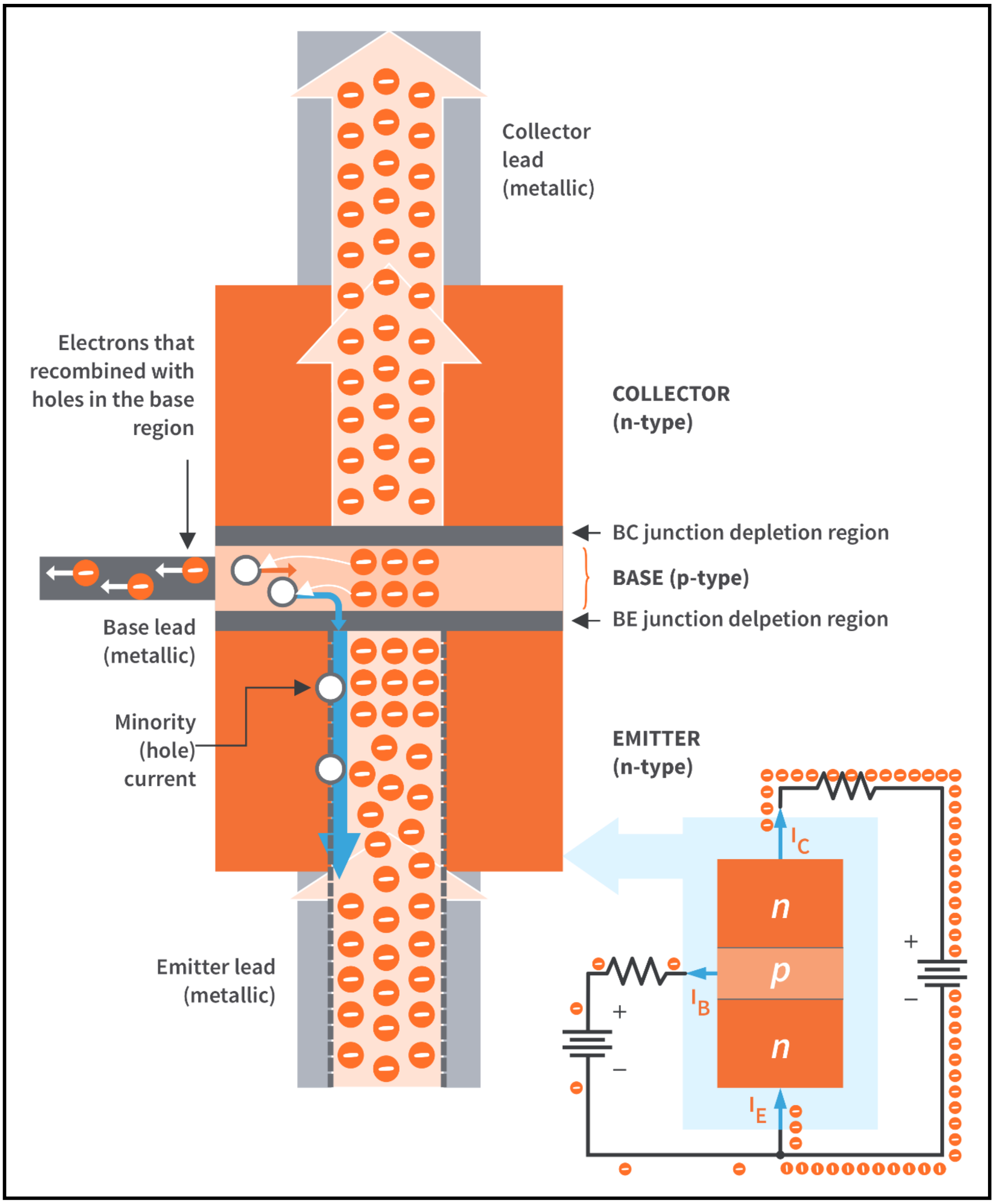
NPN BJT Bias Arrangement

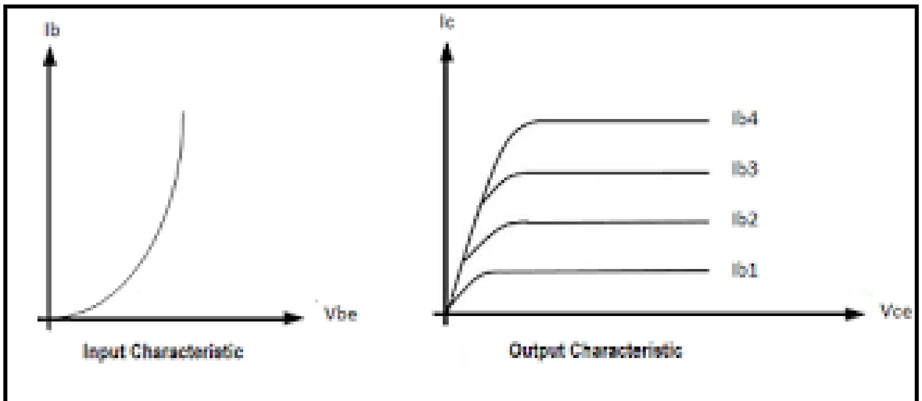


PNP BJT Bias Arrangement

The few free electrons from the emitter area that recombined with the holes in the base region became valence electrons and travel across the base region. When they leave the base region and pass through the metallic base lead, they become free electrons and form the external base current, which then passes through the metallic lead and into the external circuit before returning to the emitter region.

Free electrons that entered the base area but did not recombine with the holes are drawn to the reverse-biased base-collector junction. The free electrons are drawn to the positive side of the external bias voltage and swept over into the collector area because the collector region is connected to the positive side. They move out of the collector region, through the metallic collector lead, into the circuit, and back into the emitter region.

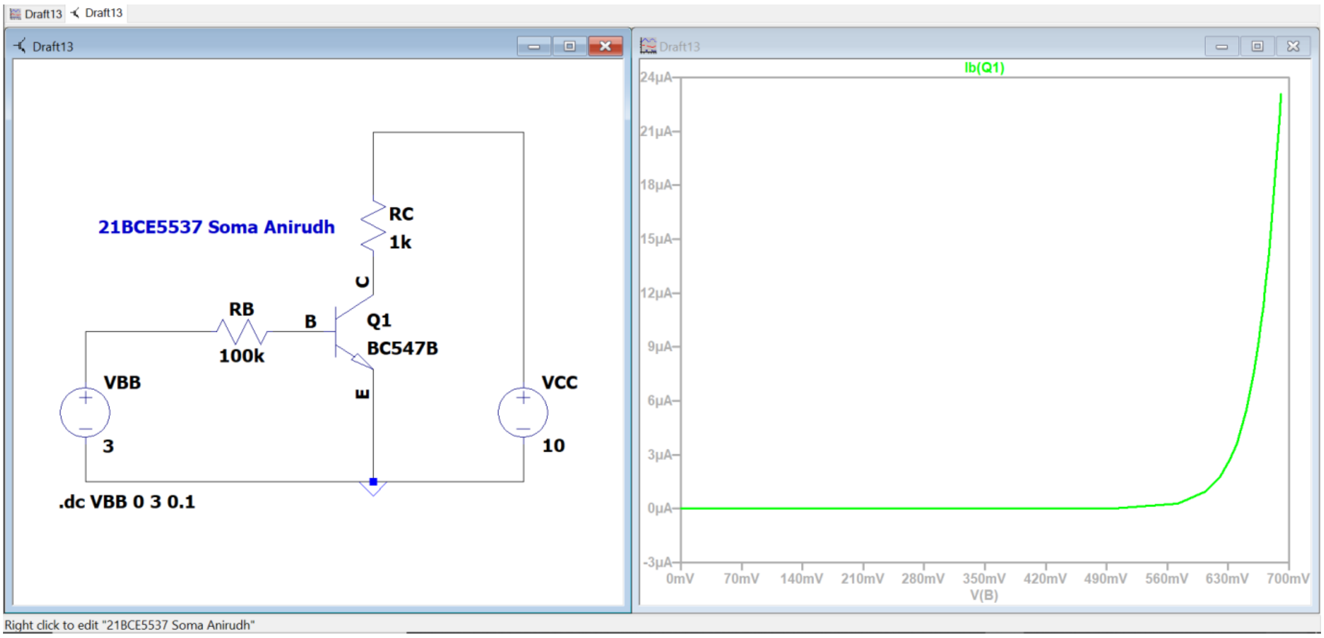


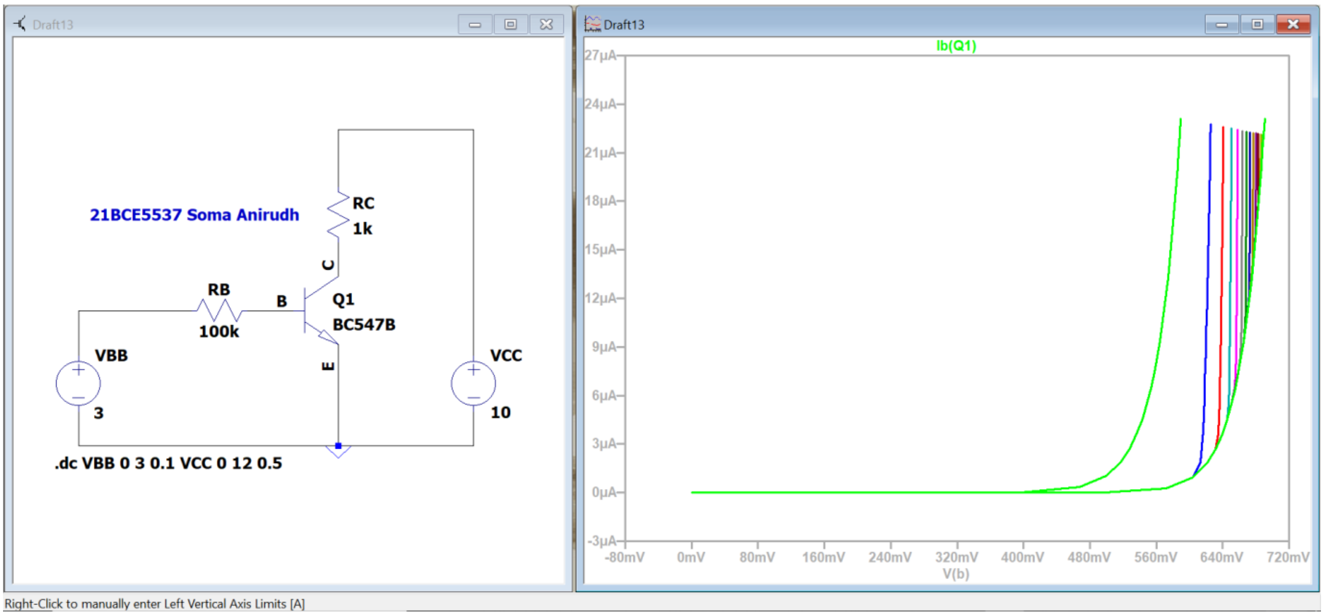


BJT characteristic curves

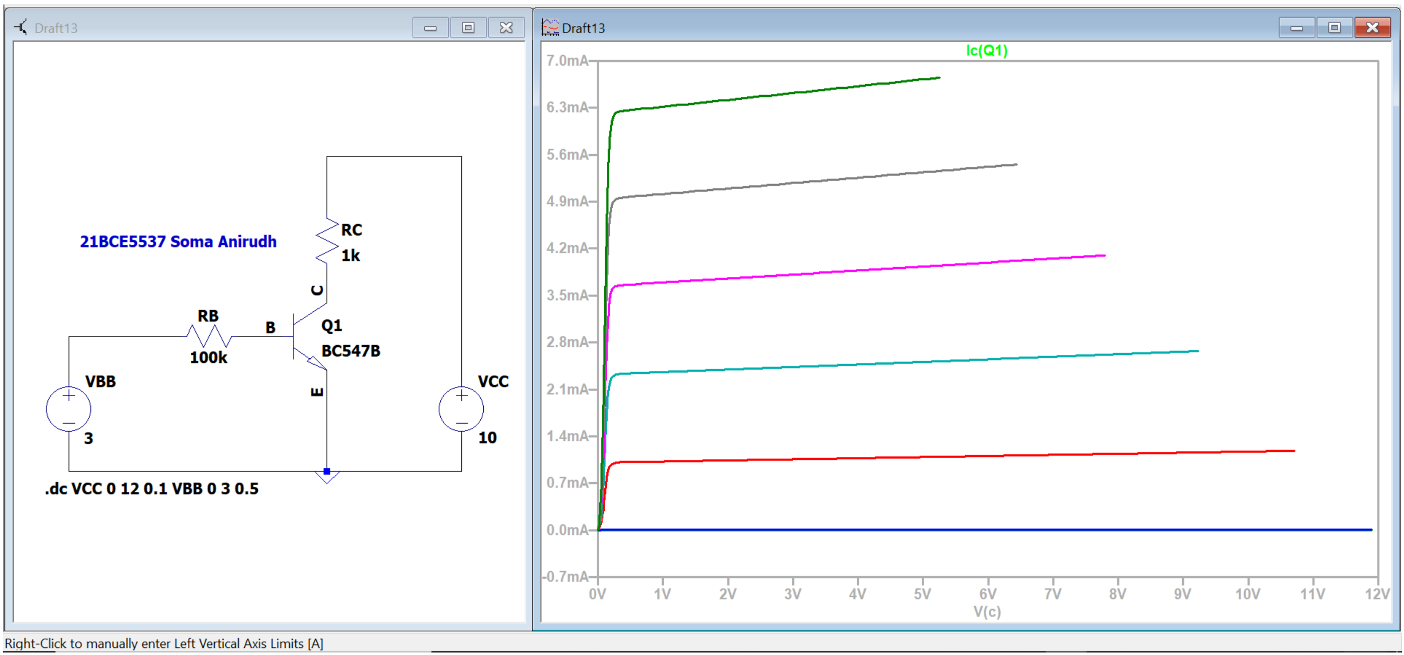
(Model Graph)

Task-1:


Draft13
21BCE5537 Soma Anirudh
RC
1k
C
RB
B
Q1
BC547B
100k
VBB
E
VCC
3
10
.dc VBB 0 3 0.1
Ib(Q1)
24µA
21µA
18µA
15µA
12µA
9µA
6µA
3µA
0µA
-3µA
0mV 70mV 140mV 210mV 280mV 350mV 420mV 490mV 560mV 630mV 700mV
V(B)
Right click to edit "21BCE5537 Soma Anirudh"


Draft13
21BCE5537 Soma Anirudh
RC
1k
C
RB
B
Q1
BC547B
100k
VBB
E
VCC
3
10
.dc VBB 0 3 0.1 VCC 0 12 0.5
Ib(Q1)
27µA
24µA
21µA
18µA
15µA
12µA
9µA
6µA
3µA
0µA
-3µA
-80mV 0mV 80mV 160mV 240mV 320mV 400mV 480mV 560mV 640mV 720mV
V(b)
Right-Click to manually enter Left Vertical Axis Limits [A]

Task-2:



Calculations:

Input Dynamic Resistance:

RI= ΔVBE/ΔIB= 26.492942mV/9.0677238μA= 2921.6 ohms.

Output Dynamic Resistance:

R0=Vce/Ic=1/slope=1/0.031=32.25Ω

Experiment-7

Name: Soma Anirudh

Slot: L19+L20 BECE101P

Date: April 7, 2022

Register number: 21BCE5537

Experiment number: 7

Exp Title : MOSFET Characteristics

The general aim of this experiment is to study and understand various characteristics of a metal–oxide–semiconductor field-effect transistor (MOSFET) .

Software Required : LtSpice

Theory:

MOSFETs are tri-terminal, unipolar, voltage-controlled devices with a high input impedance that are used in a wide range of electronic circuits. Depending on whether they have a channel in their default state or not, these devices may be categorized into two types: depletion-type and enhancement-type. Furthermore, they can be either p-channel or n-channel devices, depending on whether their conduction current is caused by holes or electrons.

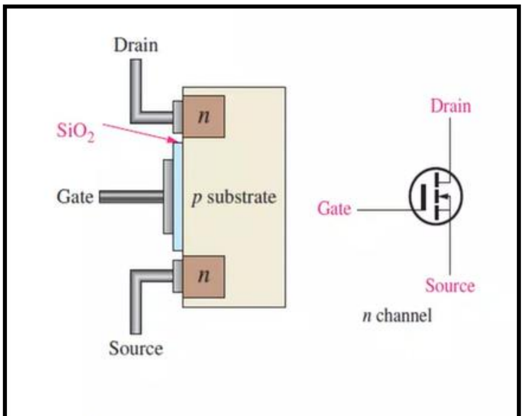
The majority of charge carriers have a significant impact on the structure of the MOSFET. As a result, in compared to the structure of the JFET, developing this sort of structure is fairly challenging. The amplification or depletion of the electric field in this MOSFET is fully reliant on the voltage provided at the terminal gate, which is in turn dependent on the channel. If it's a p-channel, the majority of the carriers will be holes, and if it's an n-channel, the most of the carriers will be electrons.

Enhancement MOSFET:

When a negative voltage is supplied to the Gate terminal, the gadget begins to conduct. When a negative voltage is given to all of the holes in the n-type, all of the minority carriers flow toward the gate terminal.

However, some of them mix with electrons that are minority carriers in the p-type drain and source along the route. However, at a specific voltage, known as the threshold voltage, the holes will be able to overcome recombination and establish a channel between the drain and the source. When a negative voltage is provided to the drain terminal in this position, the gadget begins to conduct.

The enhancement MOSFET does not have a structural conduction channel. The lightly doped p-layer substrate extends completely to the SiO_2 layer



(a) Basic construction

Depletion MOSFET:

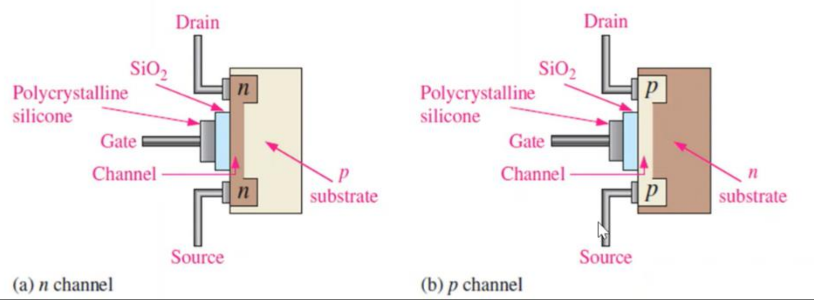
Because the bulk of the carriers in the n-channel are electrons, the positive polarity of the voltage is taken into account. The device operates similarly to a p-type MOSFET, except that when a positive voltage is supplied to the gate terminal, the device begins to conduct.

A channel is produced when the positive voltage in the gate terminal is elevated to a certain threshold voltage, and a drain and source are formed. If a positive voltage is supplied between the drain and source in this position, the device begins to conduct.

This mode of operation is identical to the P-type depletion mode, with the exception that the drain to source connection must be forward biased and a positive voltage must be provided to the Gate terminal in order for current to flow from the drain to the source.

When a negative voltage is supplied, major charge carriers are repelled towards the substrate and combine with electrons, resulting in the depletion of major charge carriers in the channel and a drop in drain current. The drain current becomes zero at a specific negative voltage. A pinch-off voltage is the name for this voltage.

The drain and the source are diffused into the substrate material which are then connected by a narrow channel adjacent to the insulated gate.

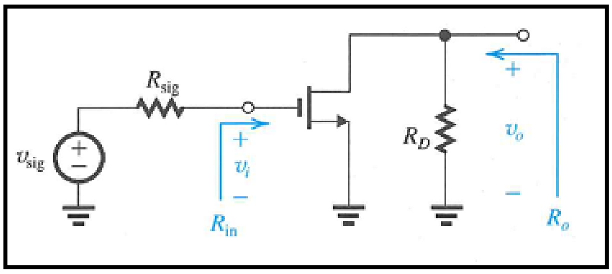


(a) n channel (b) p channel

MOSFET Configurations:

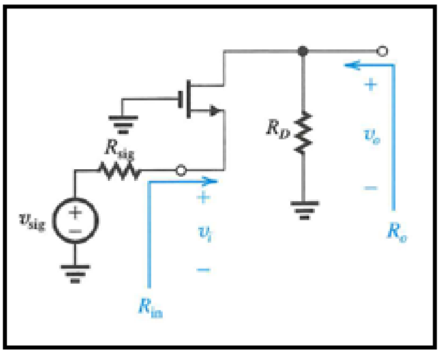
Common Source (CS) :

The MOSFET common-source (CS) amplifier is analogous to the BJT common-emitter amplifier. Its appeal stems from its high gain and the fact that it may be cascaded to generate more signal amplification.



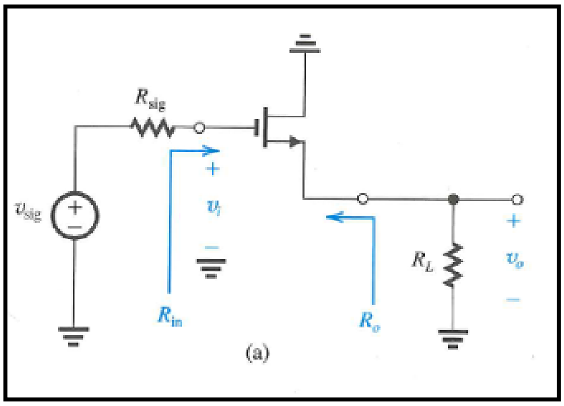
Common Gate (CG) :

Because of the maximum power theorem, it is ideal for matching sources with low input im pedance, but it pulls greater current, meaning high power consumption from the signal source.

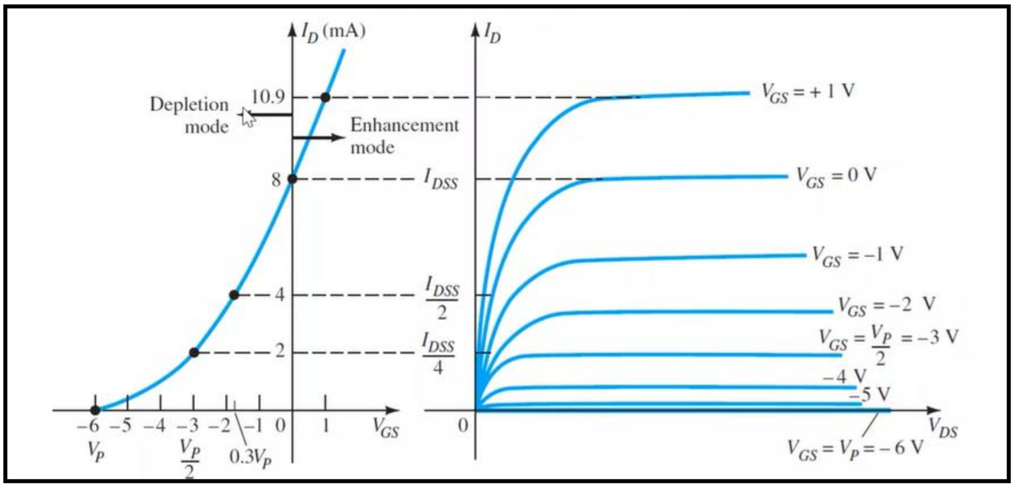


Common Drain (CD):

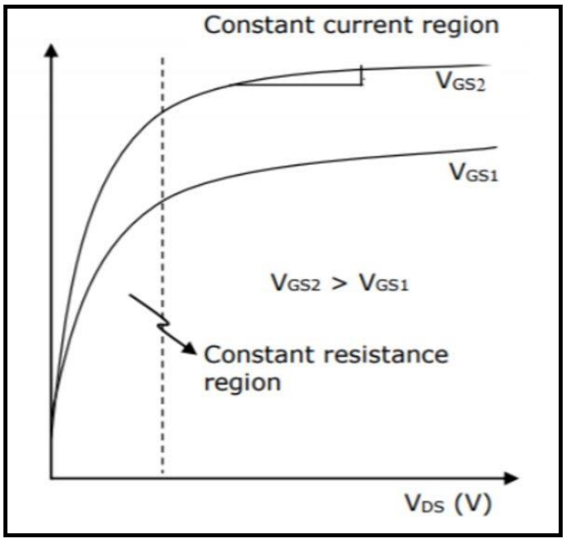
This is analogous to the BJT's emitter follower, which functions as a voltage buffer. It's a unit-gain amplifier with a high input impedance and a low output impedance. As a result, it's useful for matching a high-impedance circuit to a low-impedance circuit or a circuit that requires a higher current supply.



Model Graphs:

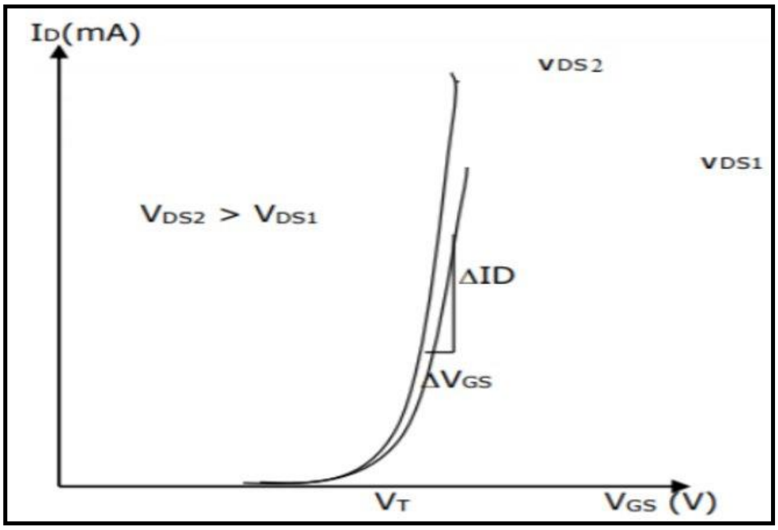


Output/Drain Characteristics:



$$r'_{ds} = \frac{\Delta V_{DS}}{\Delta I_D}$$

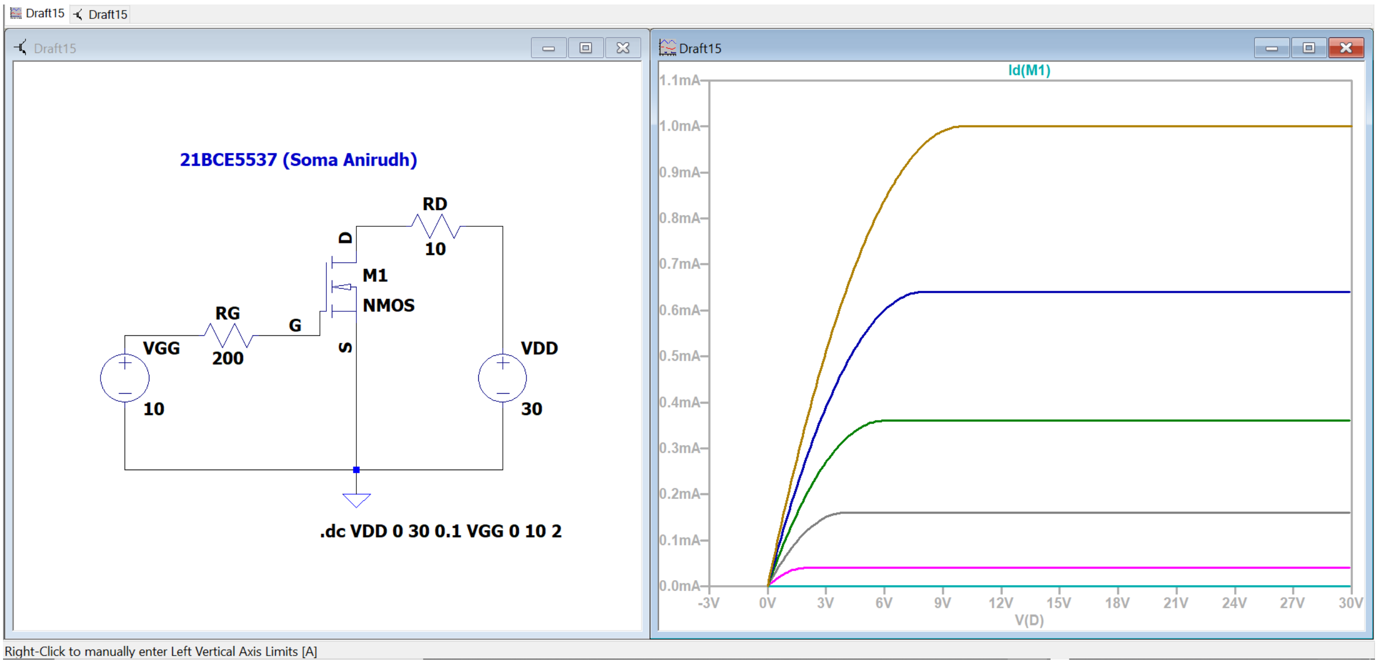
Transfer Characteristics:

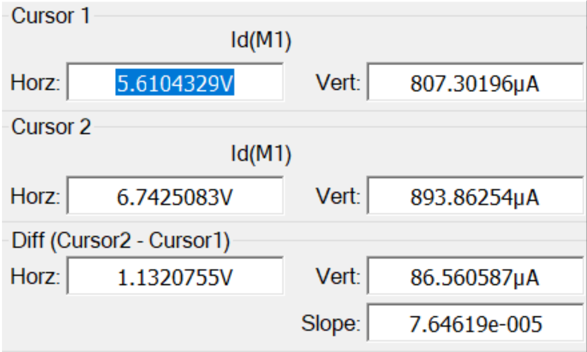


$$g_m = \frac{\Delta I_D}{\Delta V_{GS}}$$

<u>Task-1:</u>

Output/Drain Characteristics:



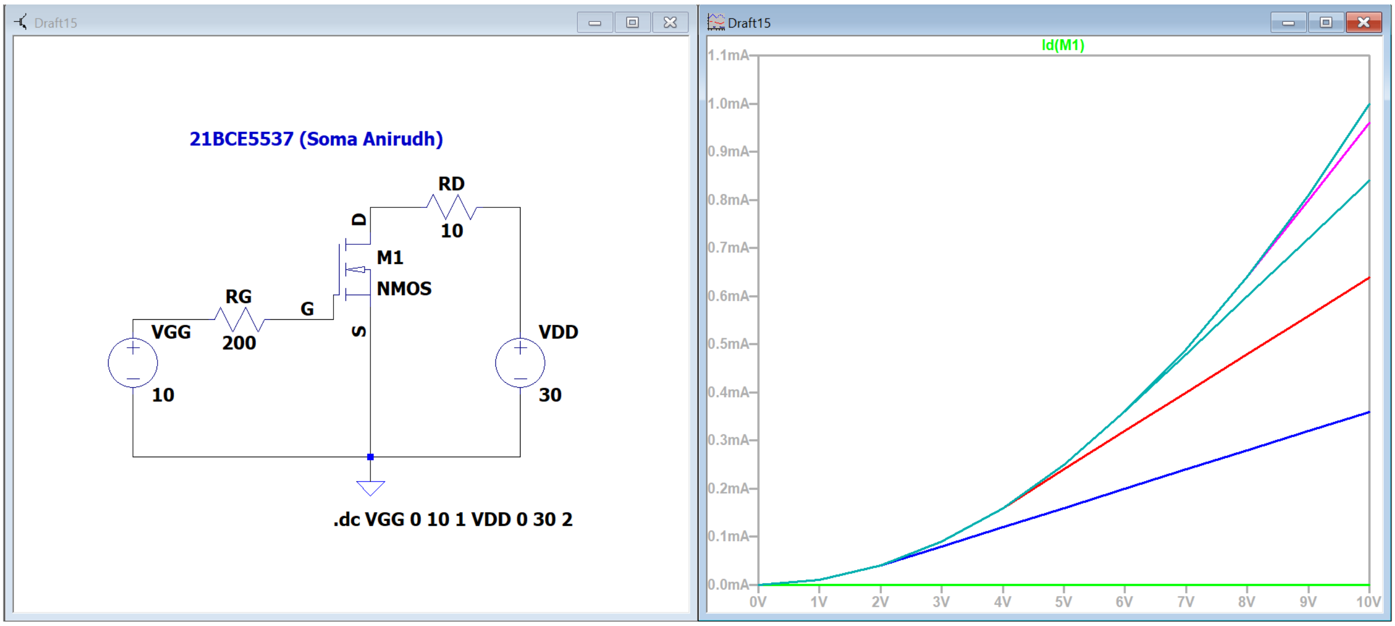


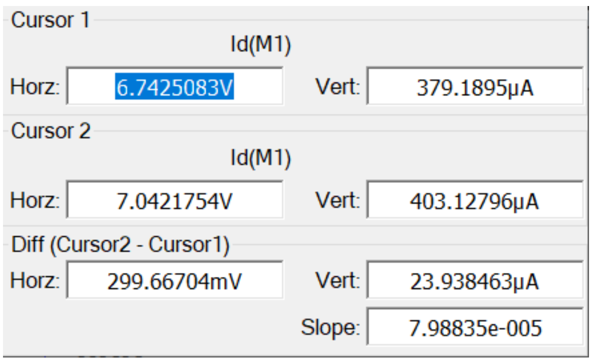
Calculation:

R(ds) = 1.1320/86.5605 * 10^-6 = 1.307 * 10^4 ohms

Task 2:

Transfer Characteristics:





Calculation:

G(m) = 23.9384 * 10^-6/ 299.667 * 10^-3 = 7.98833 * 10^-5 (ohm)^-1

Experiment-8

Name: Soma Anirudh

Slot: L19+L20 BECE101P

Date: April 22, 2022

Register number: 21BCE5537

Experiment number: 8

Exp Title : CE amplifier

The general aim of this experiment is to study and understand various characteristics of a CE Amplifier.

Software Required : LtSpice

Theory:

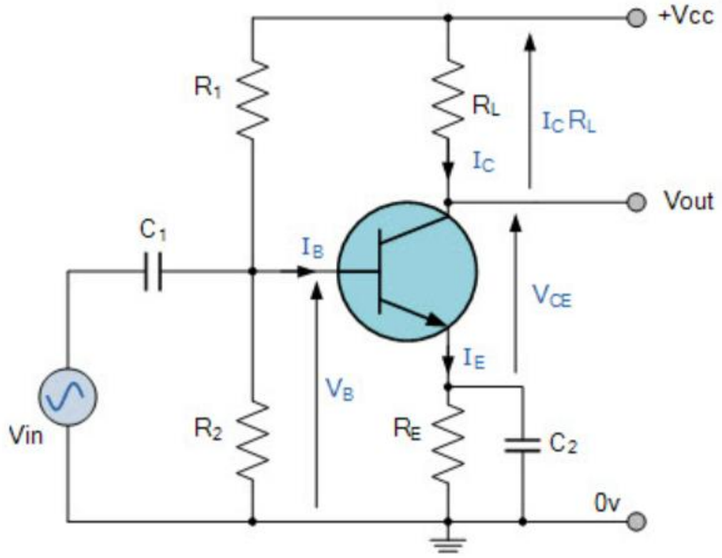
The common emitter amplifier is a voltage amplifier that consists of three basic single-stage bipolar junction transistors. The base terminal provides the amplifier's input, the collector terminal provides the output, and the emitter terminal is shared by both terminals.

The signal is sent to the base terminal of this type of amplifier, and the output is received at the collector terminal of the circuit. However, as the name implies, the basic characteristic of the emitter circuit is the same for both the input and output.

The common emitter transistor arrangement is commonly utilized in most electrical circuit designs. This design is equally good for both PNP and NPN transistors, however NPN transistors are the most commonly utilized due to their ubiquitous use.

Operation:

The forward bias across the emitter-base junction grows throughout the upper half cycle when a signal is put across it. This boosts the collector current by increasing the flow of electrons from the emitter to the collector through the base. As the collector current rises, the voltage across the collector load resistor RC lowers. The forward bias voltage across the emitter-base junction is reduced by the negative half cycle. The collector current in the whole collector resistor RC reduces as the collector-base voltage lowers. As a result, the collector resistor appears across the amplified load resistor.



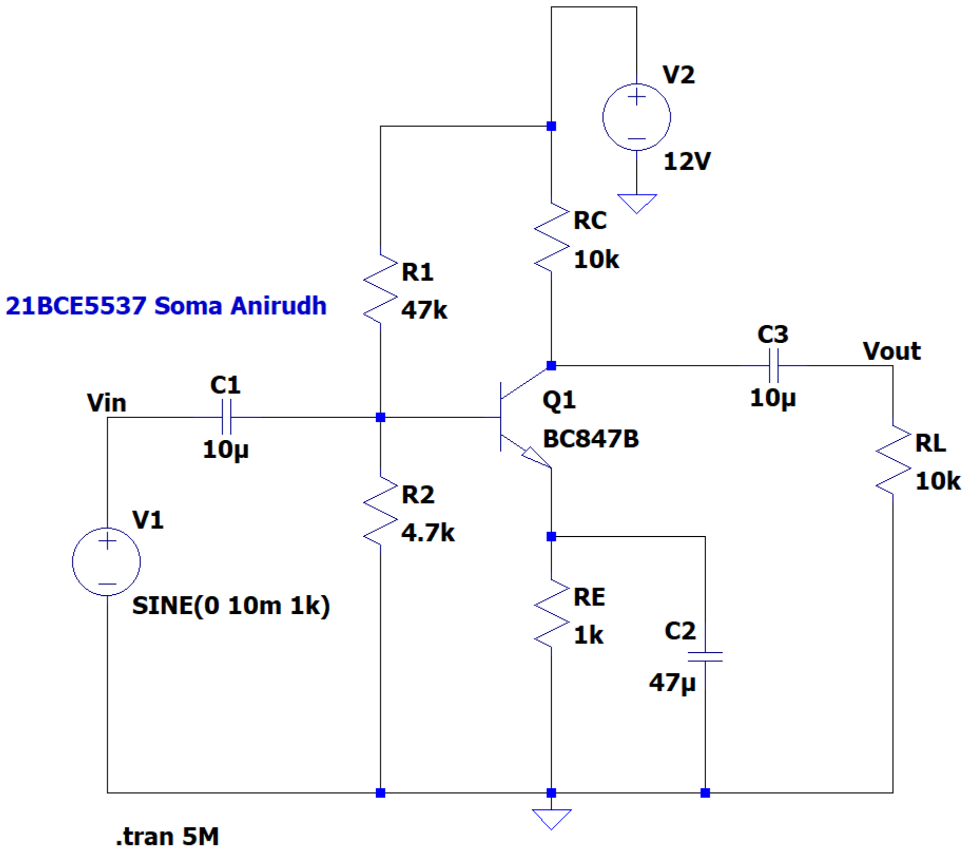
Common Emitter Amplifier Circuit

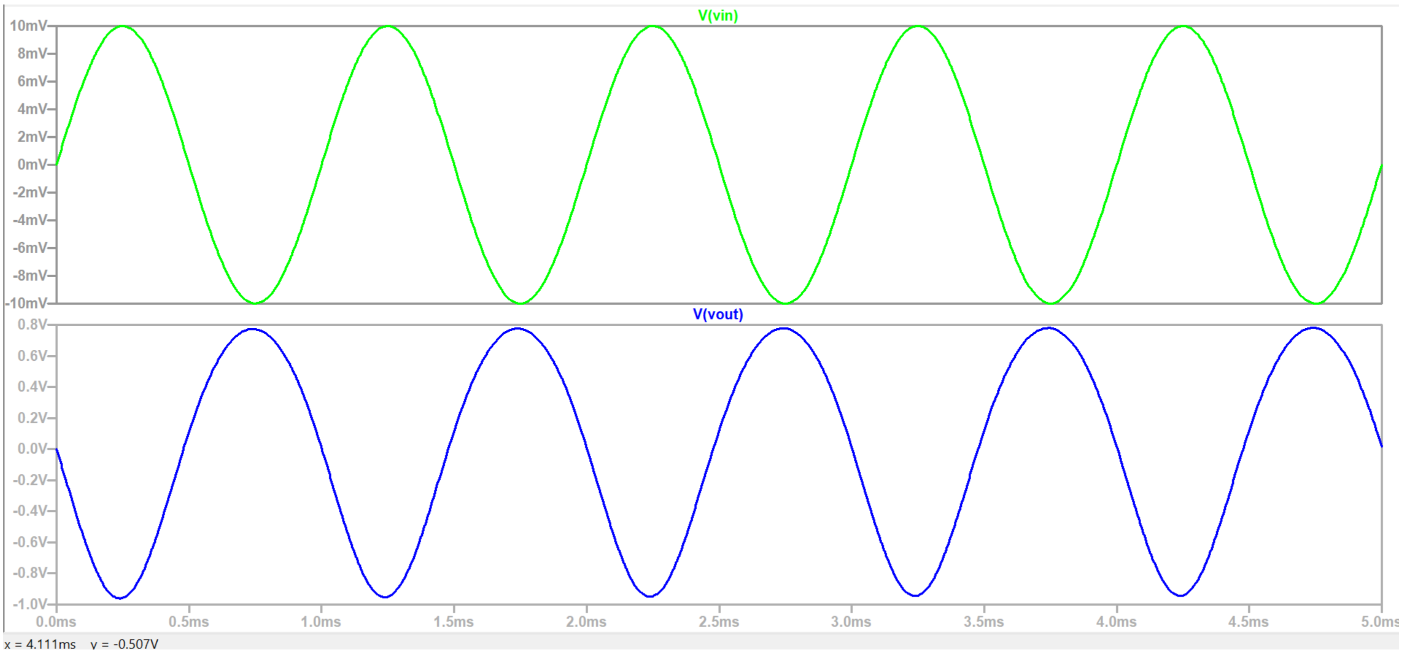
The voltage divider biassing is used to deliver the base bias voltage as needed in a common emitter amplifier circuit. A potential divider with two resistors is linked in such a way that the midway is utilized to supply base bias voltage in the voltage divider biassing.

In a typical emitter amplifier, there are several sorts of electronic components. The forward bias is controlled by the R1 resistor, the development of bias is controlled by the R2 resistor, and the load resistance is controlled by the RL resistor. Thermal stability is provided by the RE resistor. The C1 capacitor, also known as the coupling capacitor, is used to isolate the AC signals from the DC biassing voltage.

Task-1:

Transient Analysis :



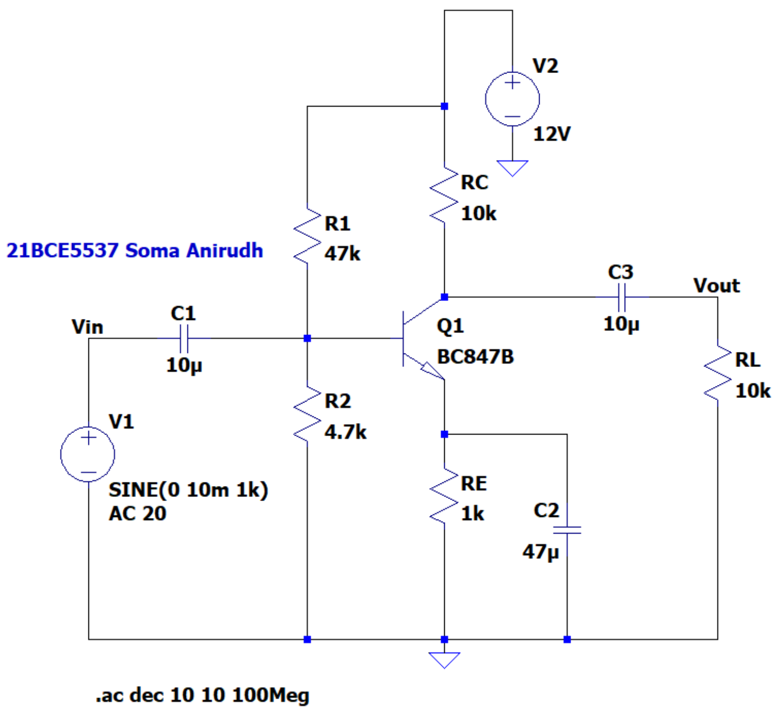


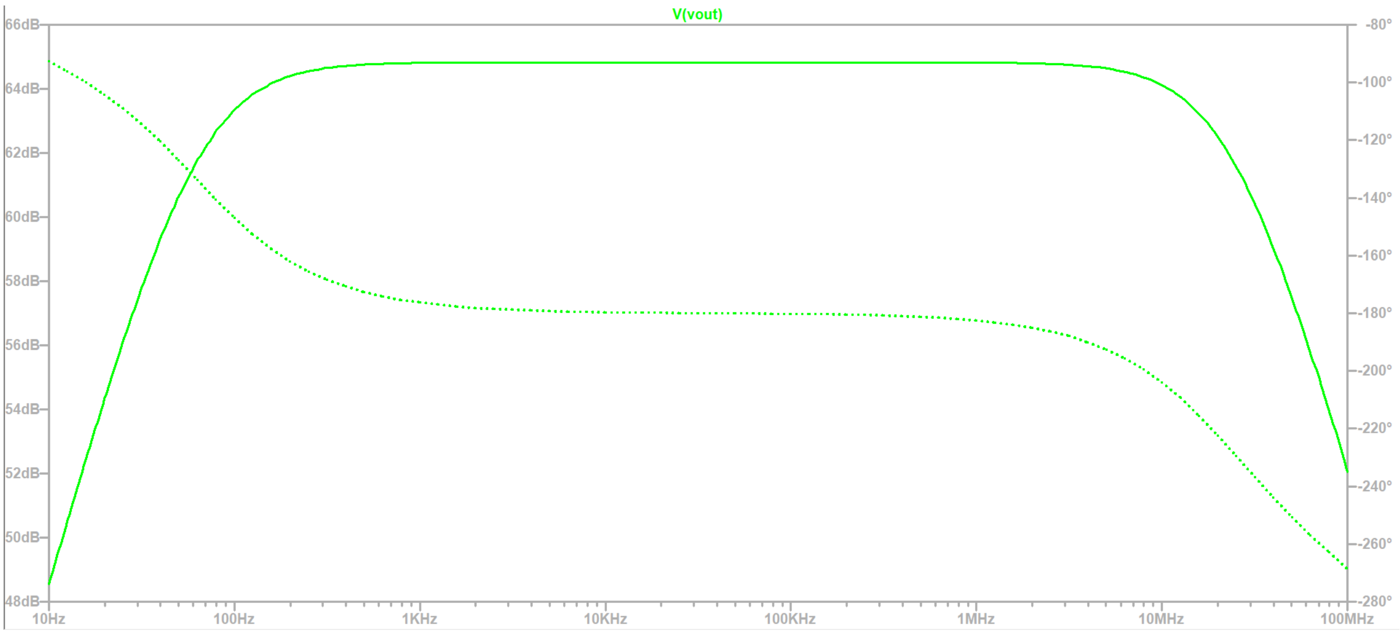
Calculations:

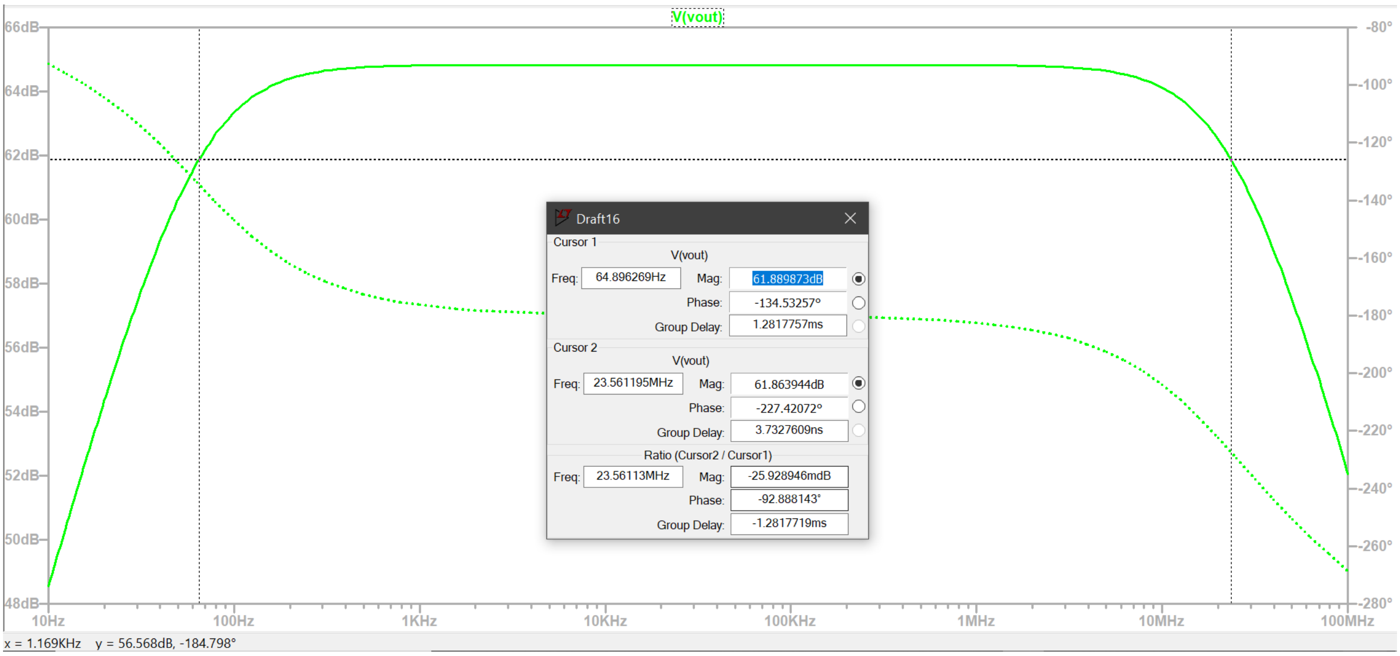
$A_v = V_{out} / V_{in} = V_C / V_B$ = 779.96mv/9.99mv = **78.07407407**

Task-2:

AC Analysis







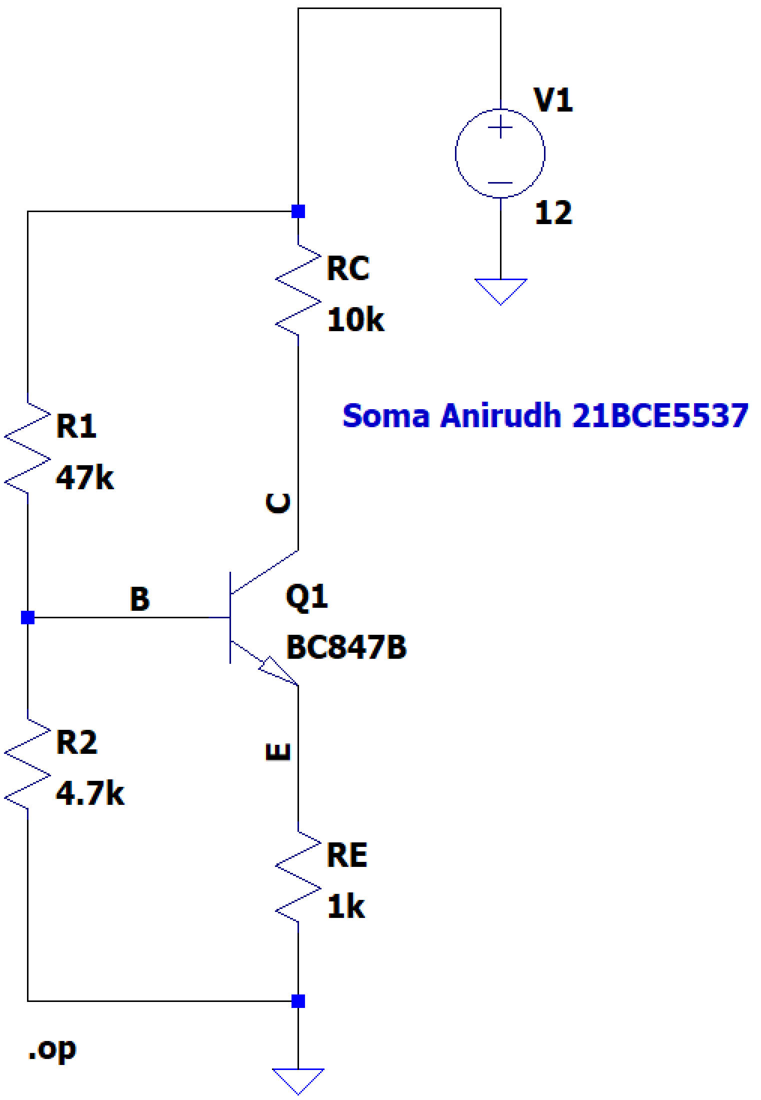
Calculations:

F_h= 23.5611MHz= 235.611 * 10^5 Hz

F_L = 64.896269 Hz

BW = F_h–F_L = 23.56113 MHz

DC Analysis



* C:\Users\urwit\OneDrive\Documents\LTspiceXVII\ece\Draft18.asc

```
       --- Operating Point ---

V(n001):        12              voltage
V(b):           1.08493         voltage
V(e):           0.46979         voltage
V(c):           7.31609         voltage
Ic(Q1):         0.000468391     device_current
Ib(Q1):         1.39864e-006    device_current
Ie(Q1):         -0.00046979     device_current
I(Rc):          0.000468391     device_current
I(Re):          0.00046979      device_current
I(R2):          0.000230837     device_current
I(R1):          0.000232235     device_current
I(V1):          -0.000700627    device_current
```

$V_{CE} = V_C - V_E = 7.31609 - 0.46979 = \mathbf{6.8463}$

Experiment-9

Name: Soma Anirudh

Slot: L19+L20 BECE101P

Date: April 22, 2022

Register number: 21BCE5537

Experiment number: 9

Exp Title : RC Phase shift oscillator

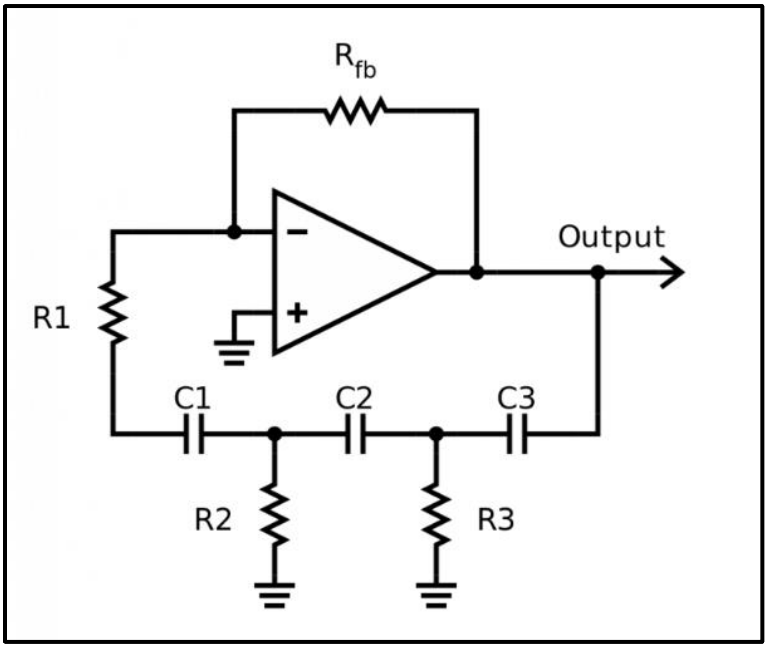
The general aim of this experiment is to study and understand various characteristics of a RC phase shift oscillator.

Software Required : LtSpice

Theory:

A phase shift oscillator is a type of linear oscillator that produces a sine wave output. It is made up of an inverting amplifier, such as an operational amplifier, or a transistor. With the aid of a phase shifting network, the output of this amplifier may be used as an input. In the form of a ladder network, this network may be constructed using resistors and capacitors. By utilizing a feedback network to give a positive response, the phase of the amplifier may be moved to 1800 at the oscillation frequency. On audio frequency, these sorts of oscillators are widely employed as audio oscillators.

A resistor as well as a capacitor can be used to construct an RC phase-shift oscillator circuit. With the feedback signal, this circuit provides the appropriate phase shift. They have exceptional frequency strength and can produce a crisp sine wave for a wide variety of loads. An simple RC network should ideally have an o/p that routes the input at a 90 degree angle.



The general equation for frequency of RC phase shift oscillator derivation can be expressed as

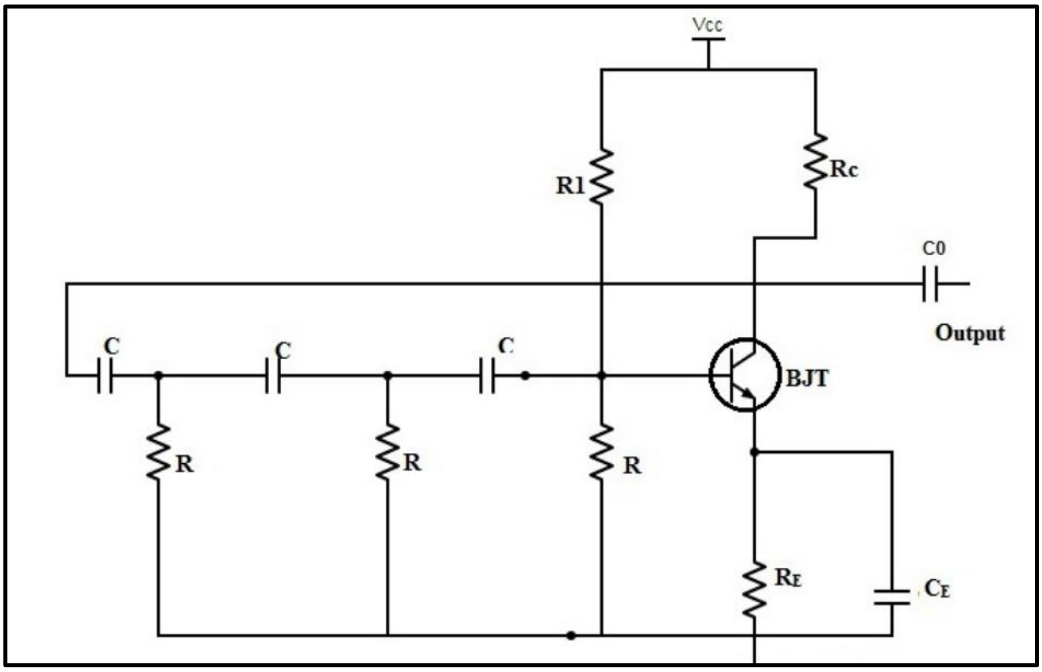
f= 1/2πRC√2N ; $\boldsymbol{F = \frac{1}{2\pi RCN\sqrt{2}}}$

Here,

R is the Resistance (Ohms)
C is the Capacitance (Farads)
N is the no. of RC networks

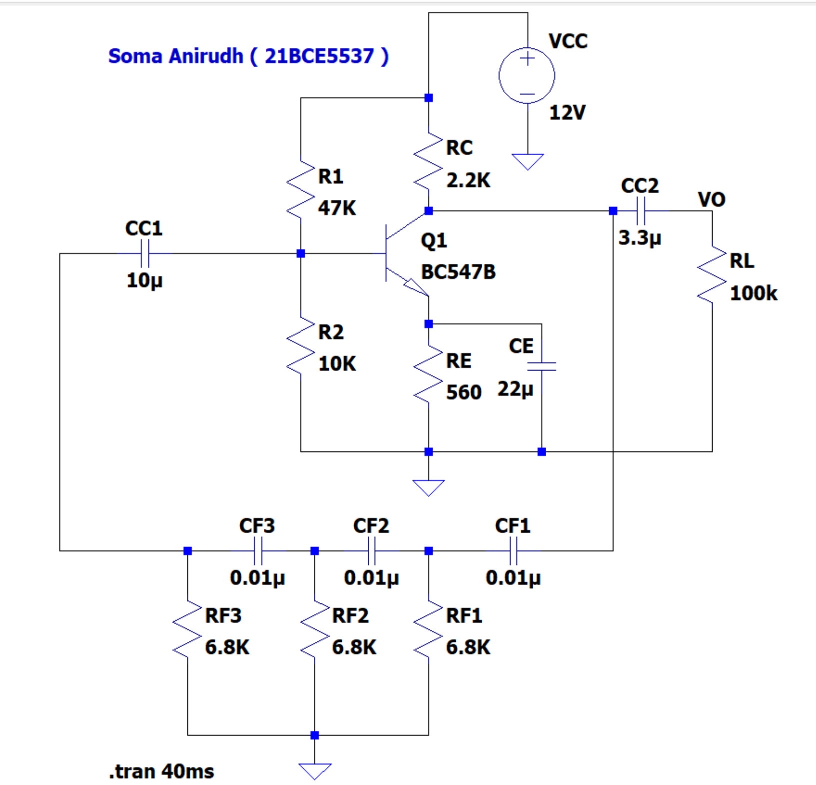
By cascading three RC phase shift networks, the following RC phase shift oscillator circuit using BJT may be created; each offers a 600 phase shift. The collector resistor, often known as the RC, in the circuit inhibits the transistor's collector current.

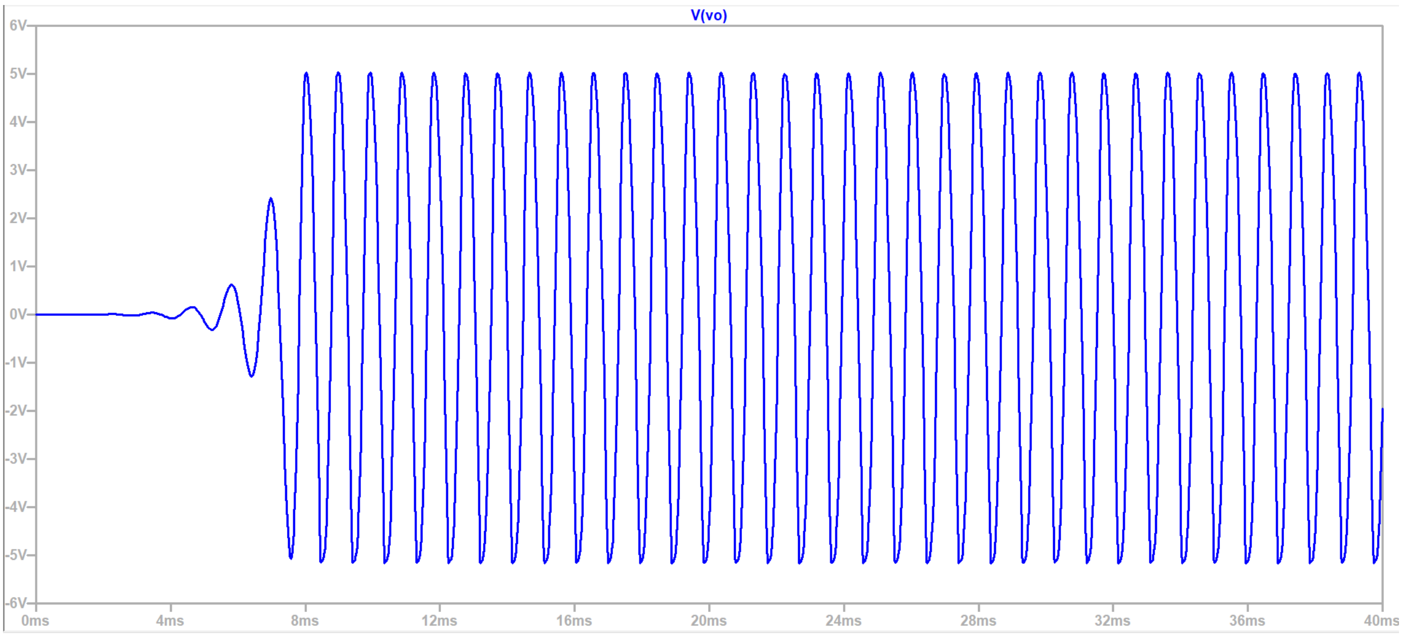
As the RE (emitter resistor) builds strength, the resistors near the transistors, such as R and R1, might create a voltage divider circuit. Following that, there are two capacitors, Co and CE, where Co is the o/p DC decoupling capacitor and CE is the emitter bypass capacitor. This circuit also shows how 3-RC networks are employed in the feedback path.



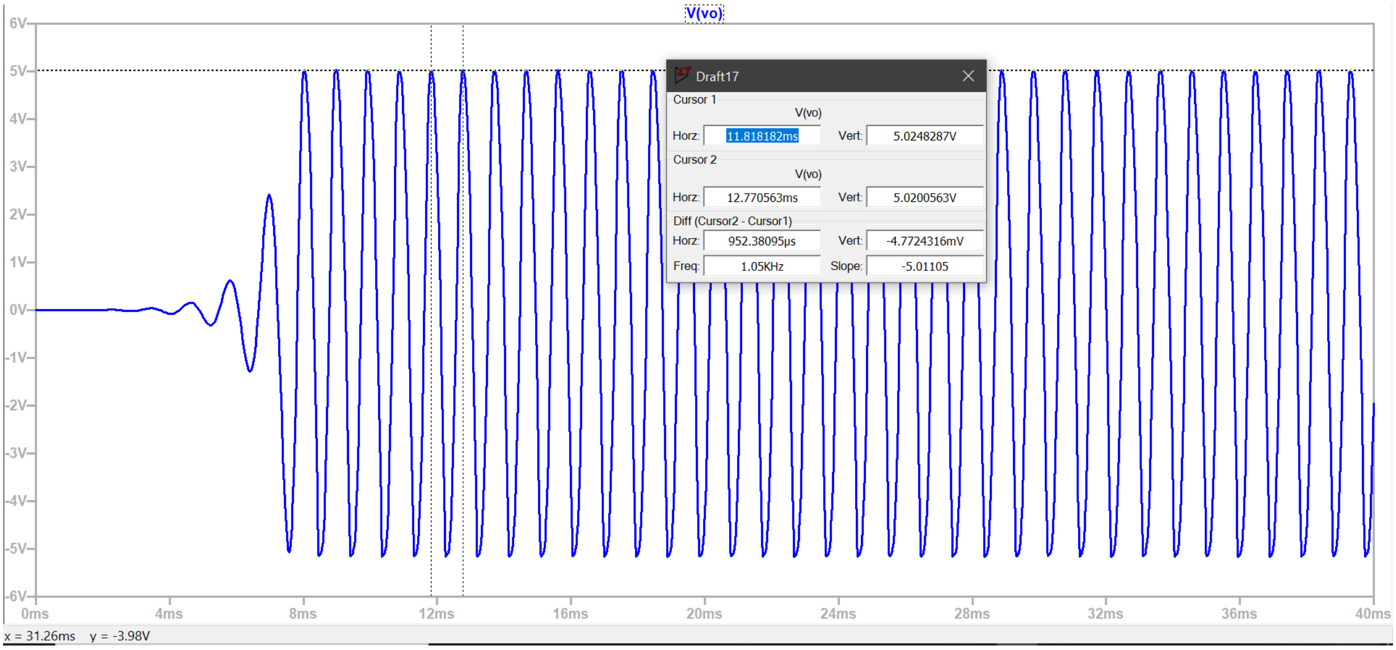
The o/p waveform will rotate 180 degrees as it travels from the o/p terminal to the transistor's base terminal as a result of this connection. After that, due to the fact that the phase discrepancy between the input and output can be 180 degrees in the common emitter (CE) arrangement, this signal can be relocated 180 degrees with the assistance of the transistor inside the network. This meets the phase disparity criteria by bringing the network phase discrepancy to 360 degrees.

LtSpice simulation circuit and graph:





Calculation of frequency and time period from the graph:



Time Period = **952.38095 µs**

Frequency = 1/T = **1.05KHz**

Experiment-10

Name: Soma Anirudh

Slot: L19+L20 BECE101P

Date: April 23rd, 2022

Register number: 21BCE5537

Experiment number: 10

Logic Gates and Implementation of Boolean Functions

Aim: To understand the working and implementation of logic gates.

Software Required : LtSpice

Theory:

A logic gate is an idealised or physical device that implements a Boolean function, which is a logical process that creates a single binary output from one or more binary inputs. The word may refer to an ideal logic gate, such as one with zero rise time and unbounded fan-out, or it may refer to a non-perfect physical device, depending on the context.

Diodes or transistors operate as electronic switches in logic gates, but they can also be built with vacuum tubes, electromagnetic relays (relay logic), fluidic logic, pneumatic logic, optics, molecules, or even mechanical parts.

There are seven basic logic gates: AND, OR, XOR, NOT, NAND, NOR, and XNOR.

AND GATE:

The AND gate gets its name from the fact that it behaves similarly to the logical "and" operator when 0 is called "false" and 1 is called "true." The circuit symbol and logic combinations for an AND gate are shown in the diagram and table below. (The input terminals are on the left, while the output terminals are on the right in the symbol.) When both inputs are "true," the outcome is "true." The output is "false" otherwise. In other words, the output is 1 only if both the first and second inputs are 1.

Input 1	Input 2	Output
	1	
1		
1	1	1

OR GATE:

The OR gate takes its name from the fact that it works in the same way as the logical inclusive "or." If one or both of the inputs are "true," the output is "true." The output is "fal se" if both inputs are "false." To put it another way, for the output to be 1, at least one OR b oth of the inputs must be 1.

Input 1	Input 2	Output
	1	1
1		1
1	1	1

XOR Gate:

The XOR (exclusive-OR) gate works similarly to the logical "either/or" gate.

If one, but not both, of the inputs are "true," the output is "true."

If both inputs are "false," the result is "false." If both inputs are "true," the output is "true."

Another approach to look at this circuit is to notice that if the inputs are different, the output is 1, but if the inputs are the same, the output is 0.

Input 1	Input 2	Output
	1	1
1		1
1	1	

NOT GATE:

A logical inverter has only one input and is frequently referred to as a NOT gate to distinguis h it from other types of electronic inverter devices.

The logic state is reversed.

If the input value is 1, the output value is 0.

If the input is zero, the output will be one.

Input	Output
1	
	1

NAND Gate:

The NAND gate is made up of an AND gate and a NOT gate.

It operates in the same way that the logical operation "and" followed by negation does.

If both inputs are "true," the outcome is "false."

The output is "true" otherwise.

Input 1	Input 2	Output
		1
	1	1
1		1
1	1	

NOR Gate:

The *NOR gate* is a combination OR gate followed by an inverter. Its output is "true" if both inputs are "false." Otherwise, the output is "false."

Input 1	Input 2	Output
		1
	1	
1		
1	1	

XNOR Gate:

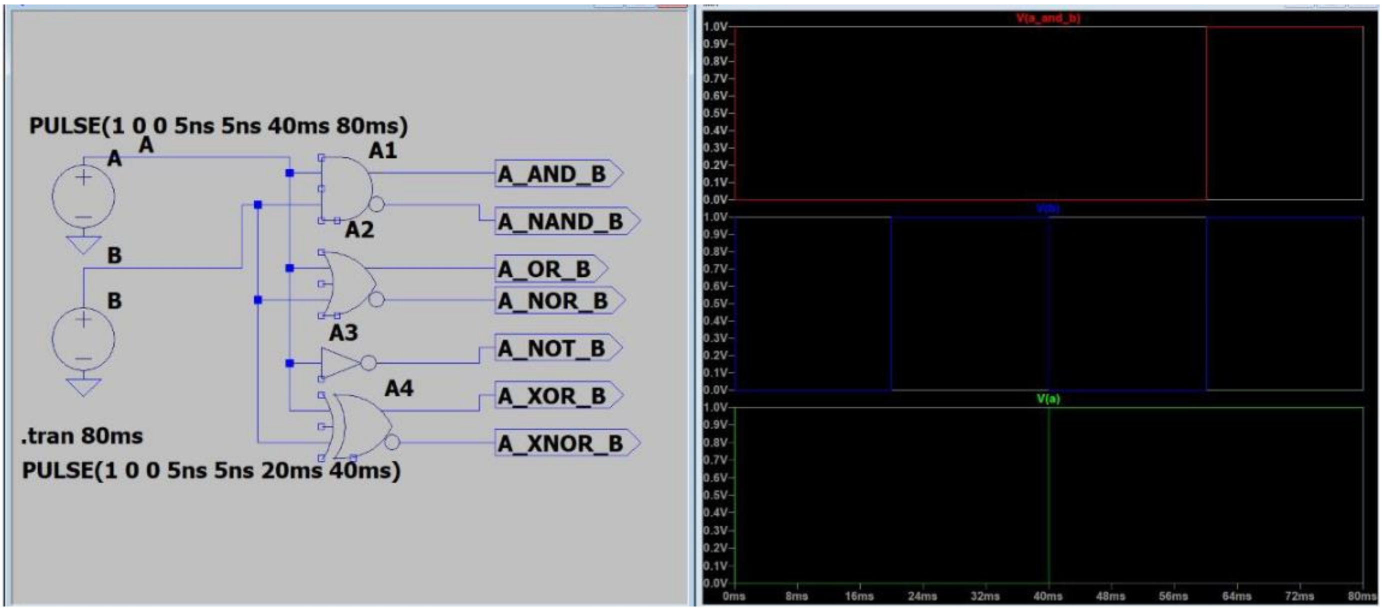
The XNOR (exclusive-NOR) gate is made up of two XOR gates and an inverter.

If the inputs are the same, the output is "true," but if the inputs are different, the output is "false."

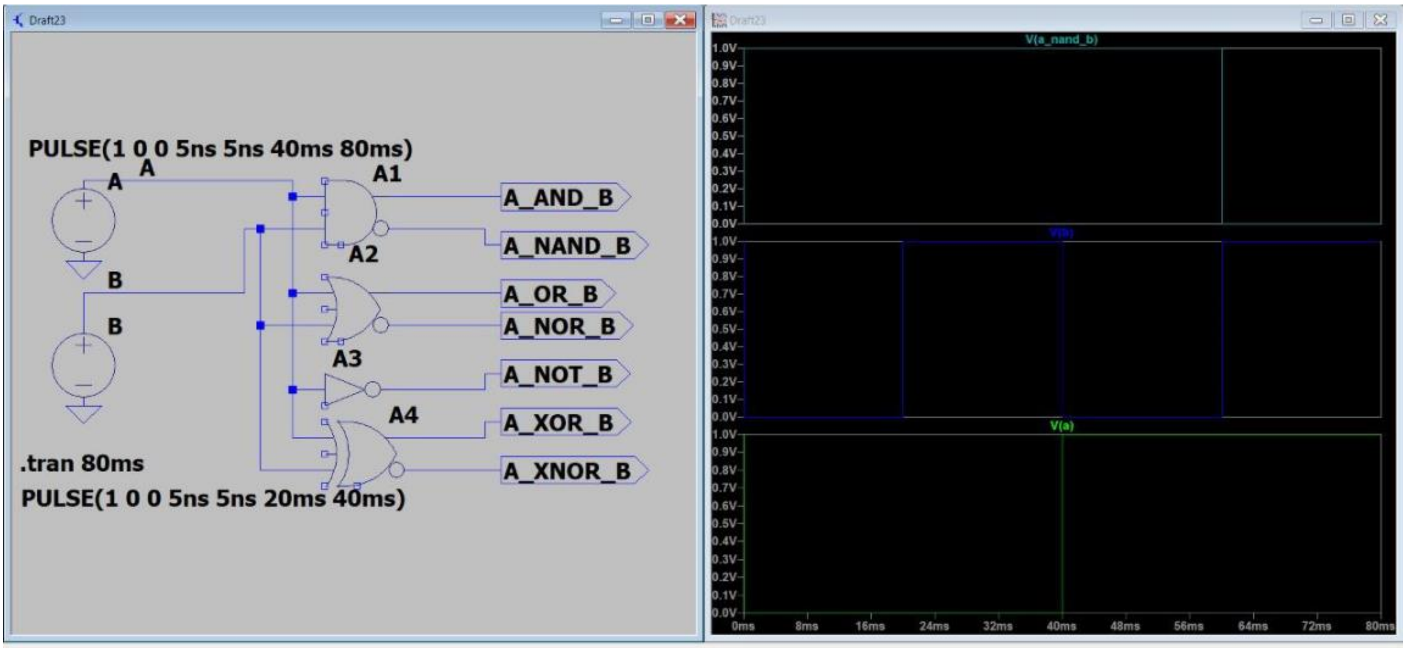
Input 1	Input 2	Output
		1
	1	
1		
1	1	1

Task-1 (Verifying all the gates):

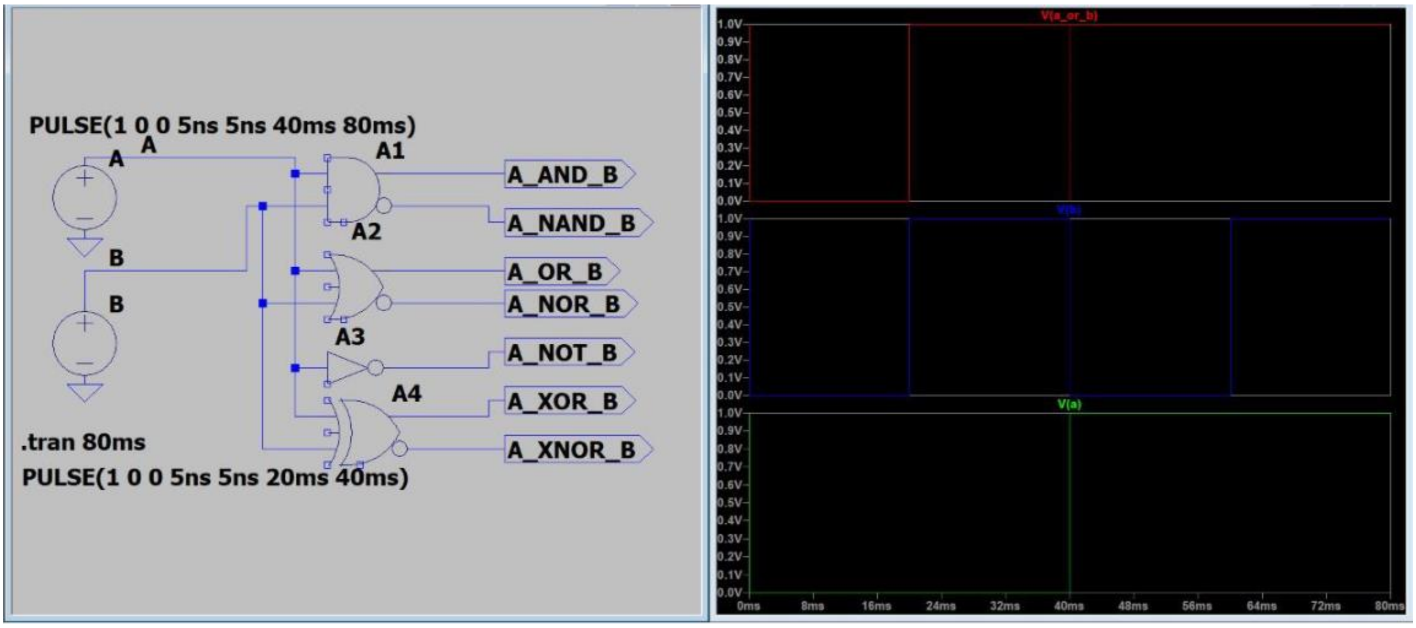
AND GATE:



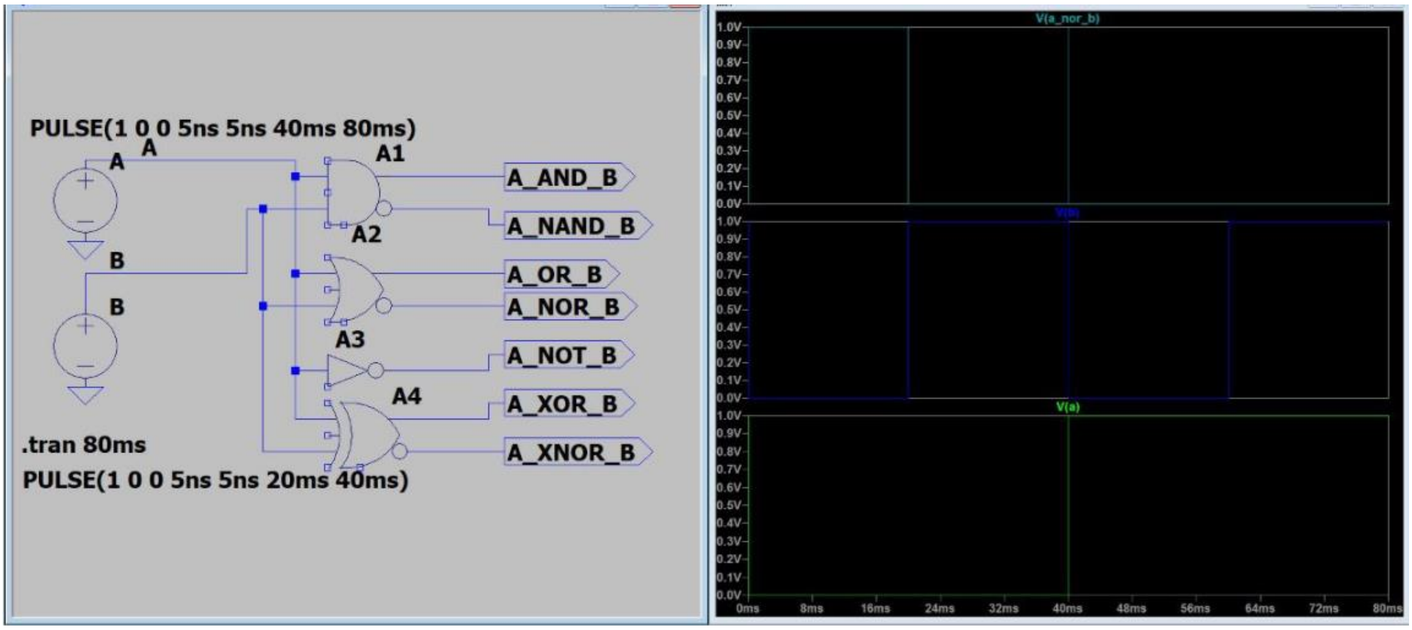
NAND Gate:



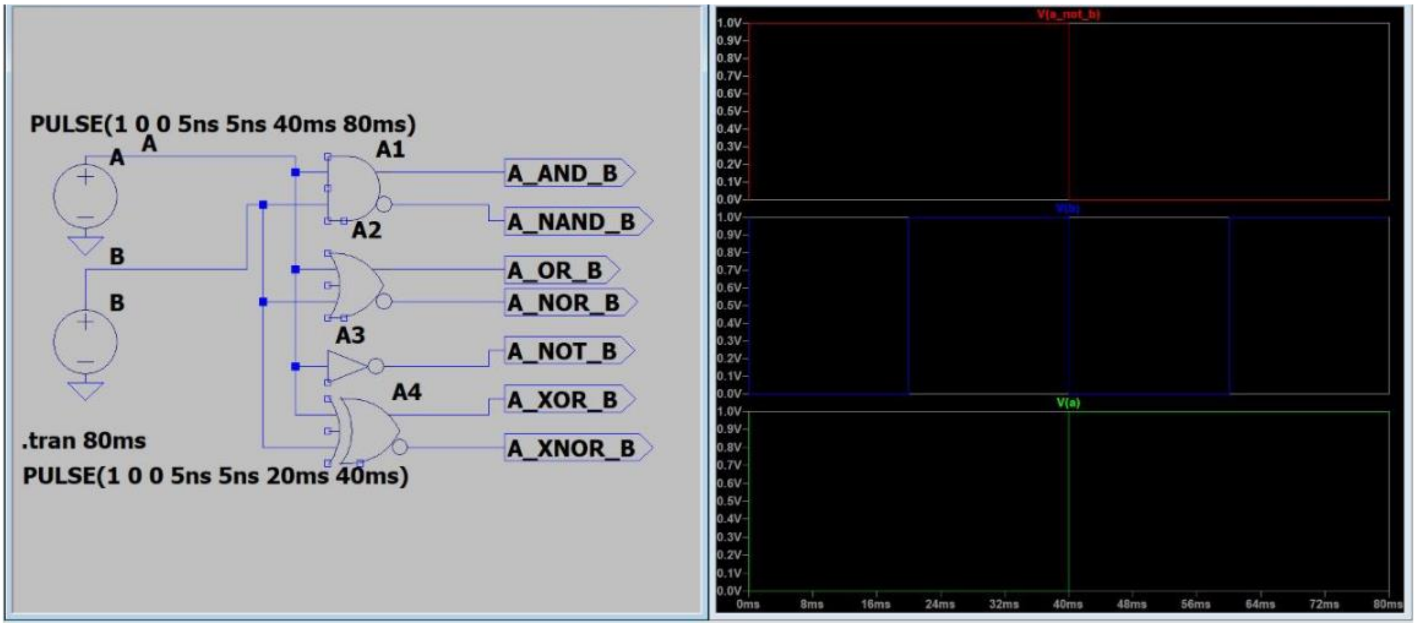
OR GATE:



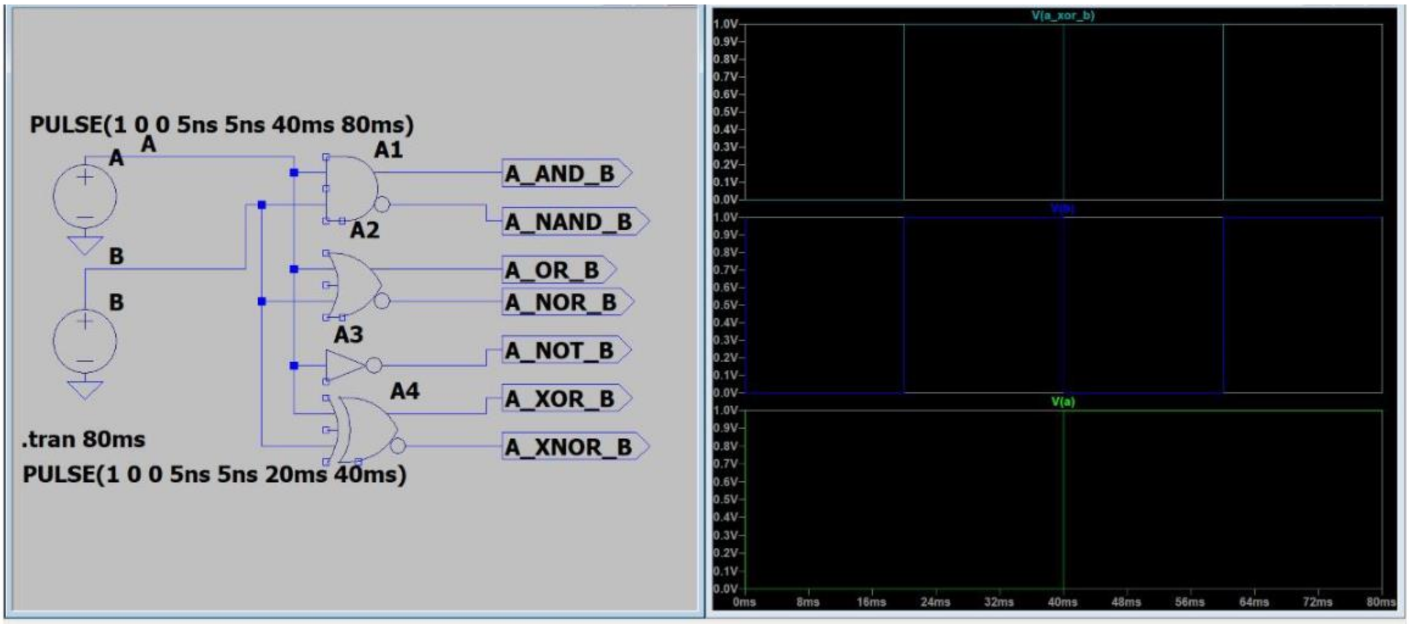
NOR GATE:



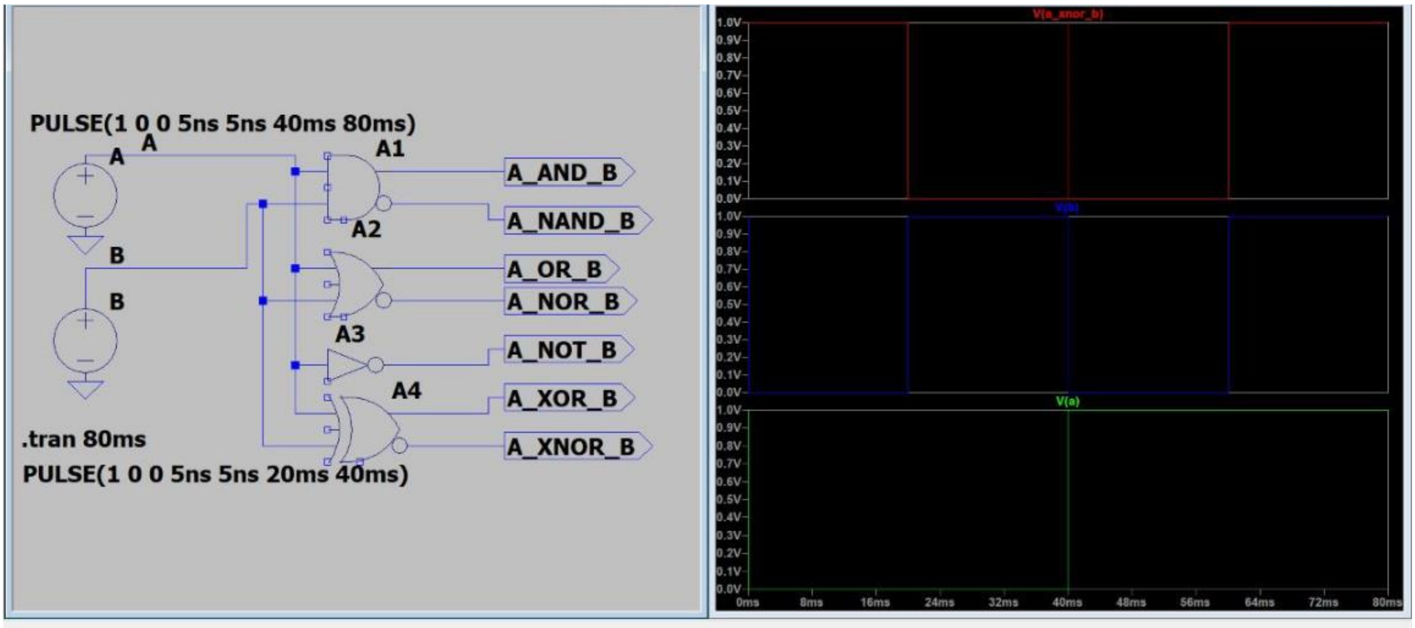
NOT GATE:



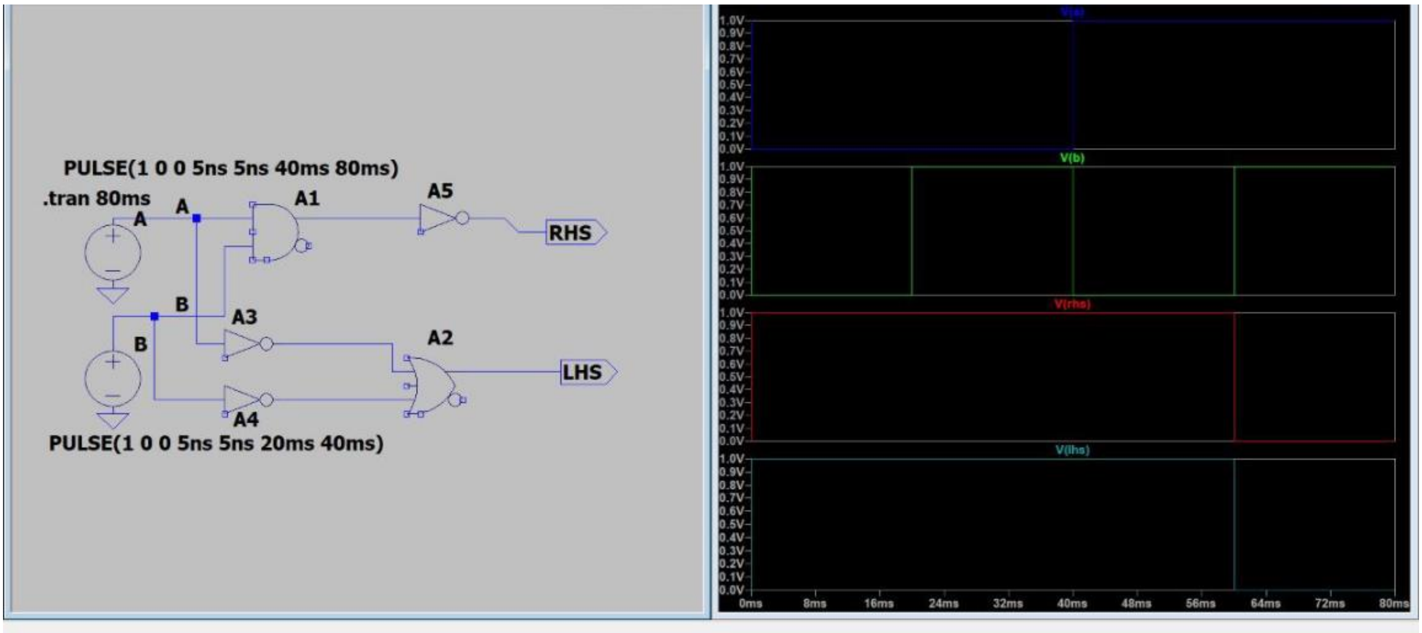
XOR GATE:

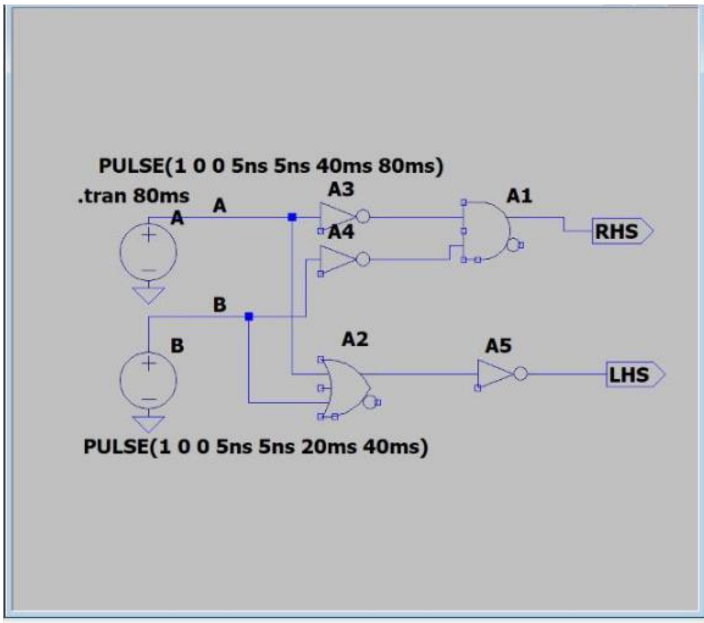


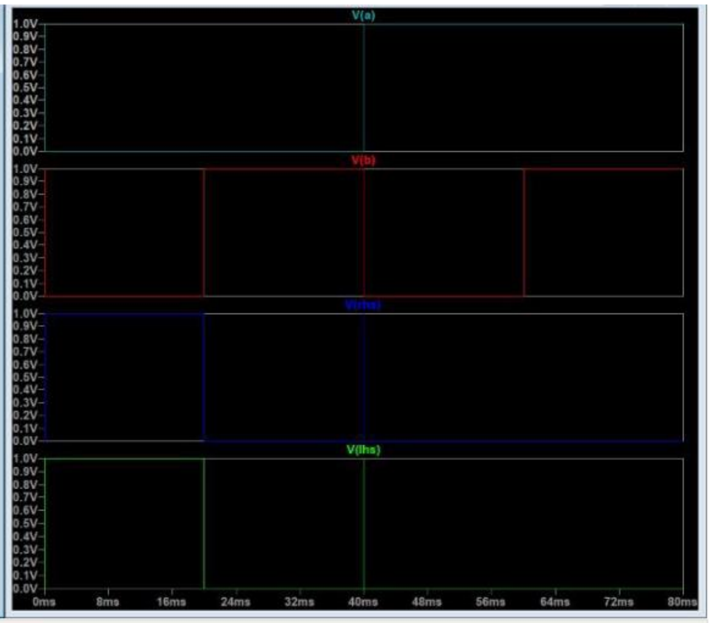
XNOR GATE:



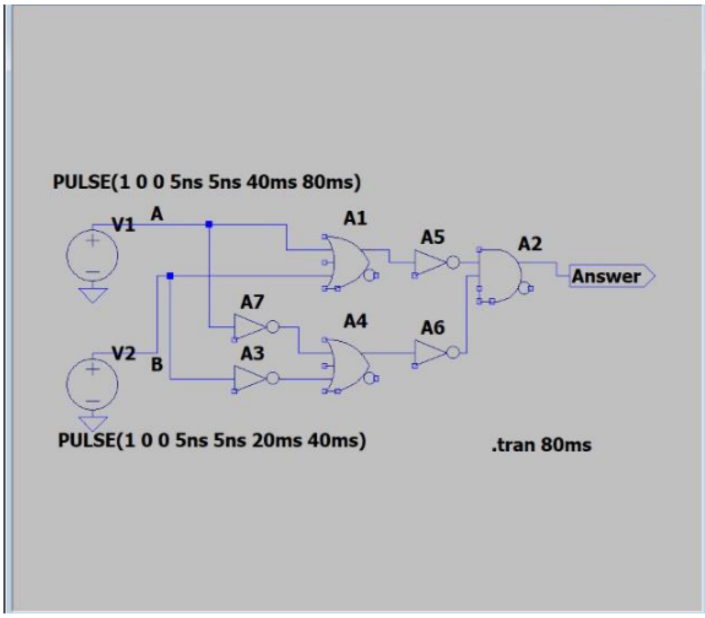
Task-2 (Verifying De-Morgan's theorem)\

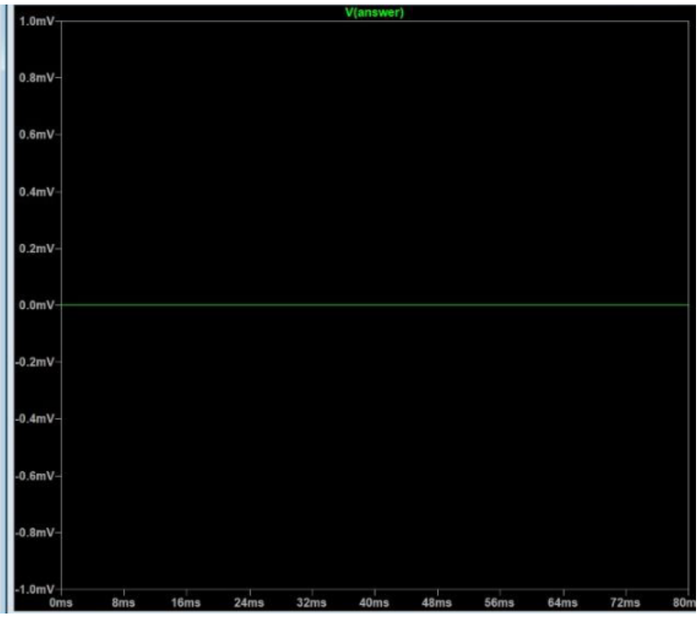




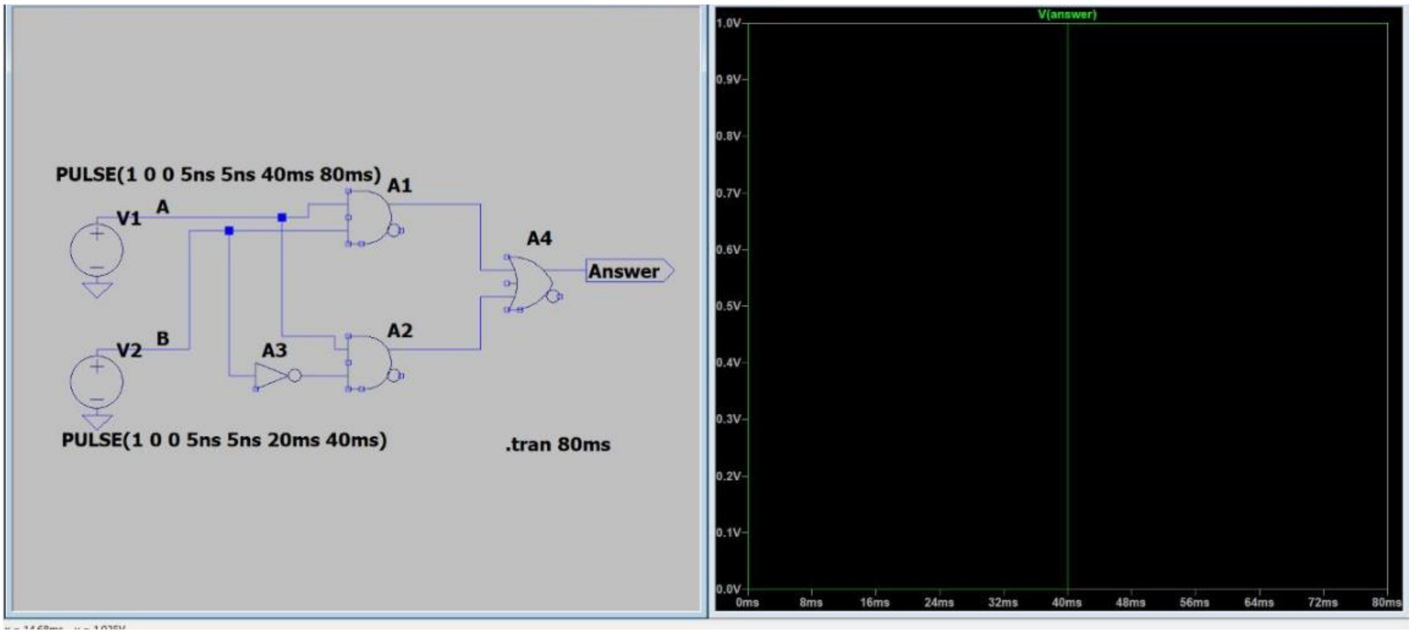


Task 3 (1): F = xy+xy′





Task 3(2): F = (A+B)′. (A′+B′)′


PULSE(1 0 0 5ns 5ns 40ms 80ms)
A1
V1
A
A4
Answer
V2
B
A3
A2
PULSE(1 0 0 5ns 5ns 20ms 40ms)
.tran 80ms
V(answer)
1.0V
0.9V
0.8V
0.7V
0.6V
0.5V
0.4V
0.3V
0.2V
0.1V
0.0V
0ms
8ms
16ms
24ms
32ms
40ms
48ms
56ms
64ms
72ms
80ms

Experiment-11

Name: Soma Anirudh

Slot: L19+L20 BECE101P

Date: April 30th, 2022

Register number: 21BCE5537

Experiment number: 11

Exp Title : Resistance temperature detector

1. The general aim of this experiment is to study and understand various characteristics of a Resistance temperature detector (static and dynamic characteristics of RTD)
2. Study effect of various parameters on RTD performance

Software Required : Virtual Lab (https://sl-coep.vlabs.ac.in/)

Theory of the Experiment:

As the name indicates, a Resistance Temperature Detector (RTD) is a sensor that measures temperature by correlating resistance to temperature. A length of tiny coiled wire is wrapped around a ceramic or glass core in most RTD components. Because the element is generally relatively delicate, it is normally protected with a sheathed probe. Pure metals like as platinum, nickel, and copper are used to make the RTD element. Temperature is determined by the material's ability to modify resistance predictably as temperature varies.

A variety of sensors are available for temperature measurement. The RTD is one of the most linear, stable, and repeatable temperature sensors. The output resistance of an RTD varies with temperature. A positive temperature coefficient device is an RTD. The metal's resistance increases as the temperature rises. As the name indicates, a Resistance Temperature Detector (RTD) is a sensor that measures temperature by correlating resistance to temperature.

RTDs are made of metals whose resistance rises as the temperature rises. Its resistance increases linearly with temperature within a restricted temperature range:

$R_t = R_0[1+ \alpha(t-t_0)]$

R_t is the resistance at temperature 't'.

R_0 is the resistance at a constant temperature (Generally 0^oC)

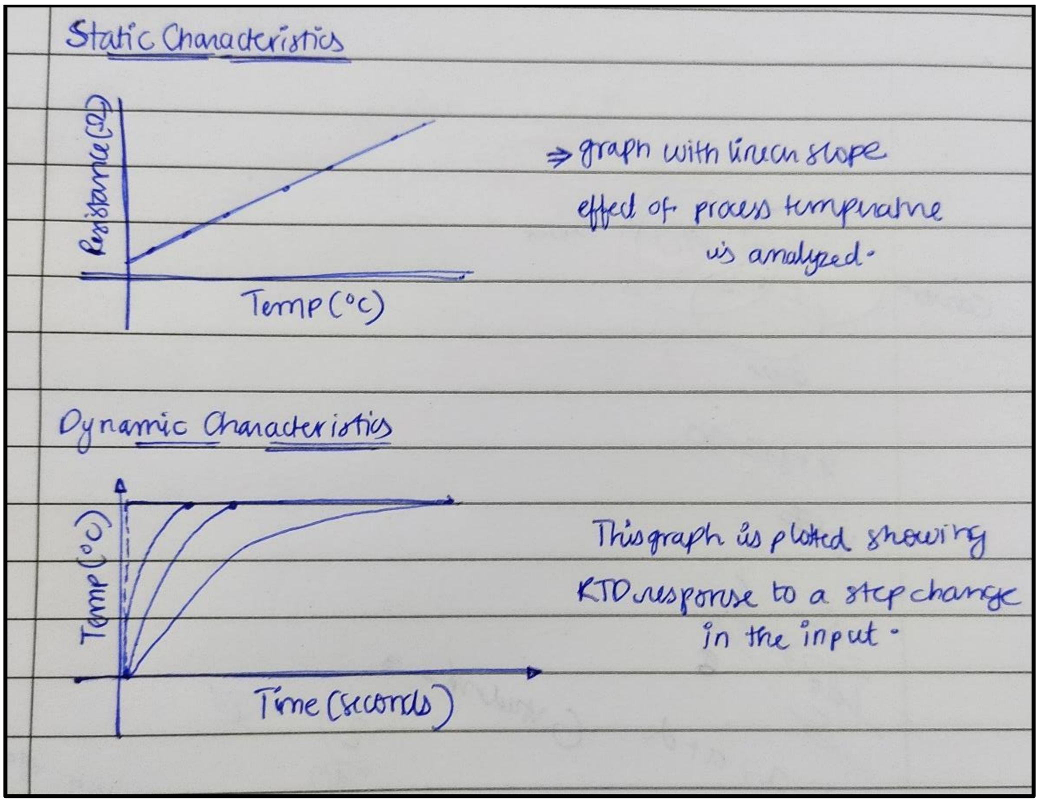
alpha= temperature coefficient of resistance (°C^{-1})

It is evident that resistance vs. temperature is linear with a slope equal to

R = $R_0(\alpha t+1)$

when t0 is set to 0°C and rearranged to the conventional linear y = mx + b form.

Model Graphs:



Calculations:

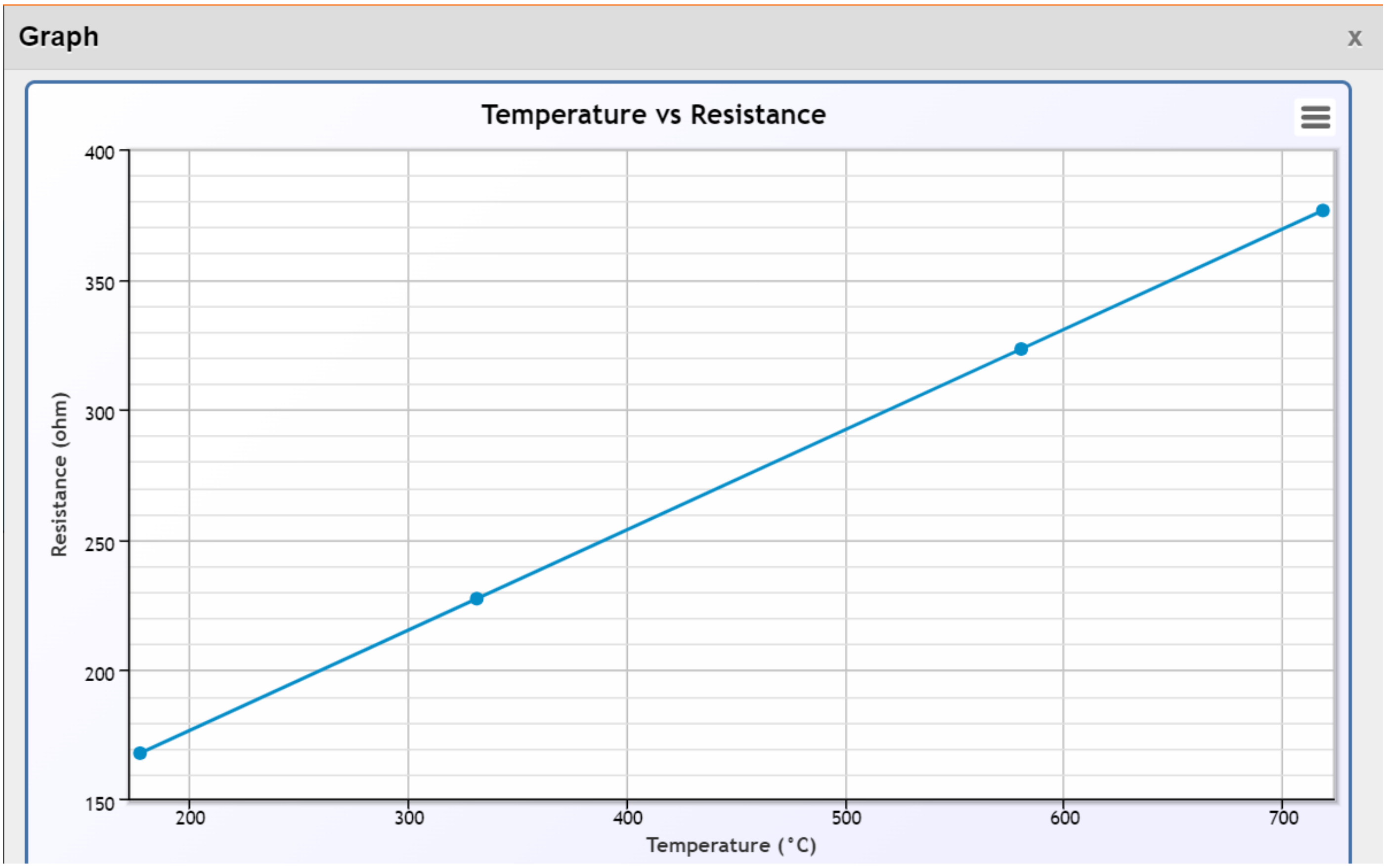
Calculations for the plot points are given by $R_t = R_0[1+ \alpha(t-t_0)]$

Where α ; Δt and R_0 are already calculated and outputted by the virtual software

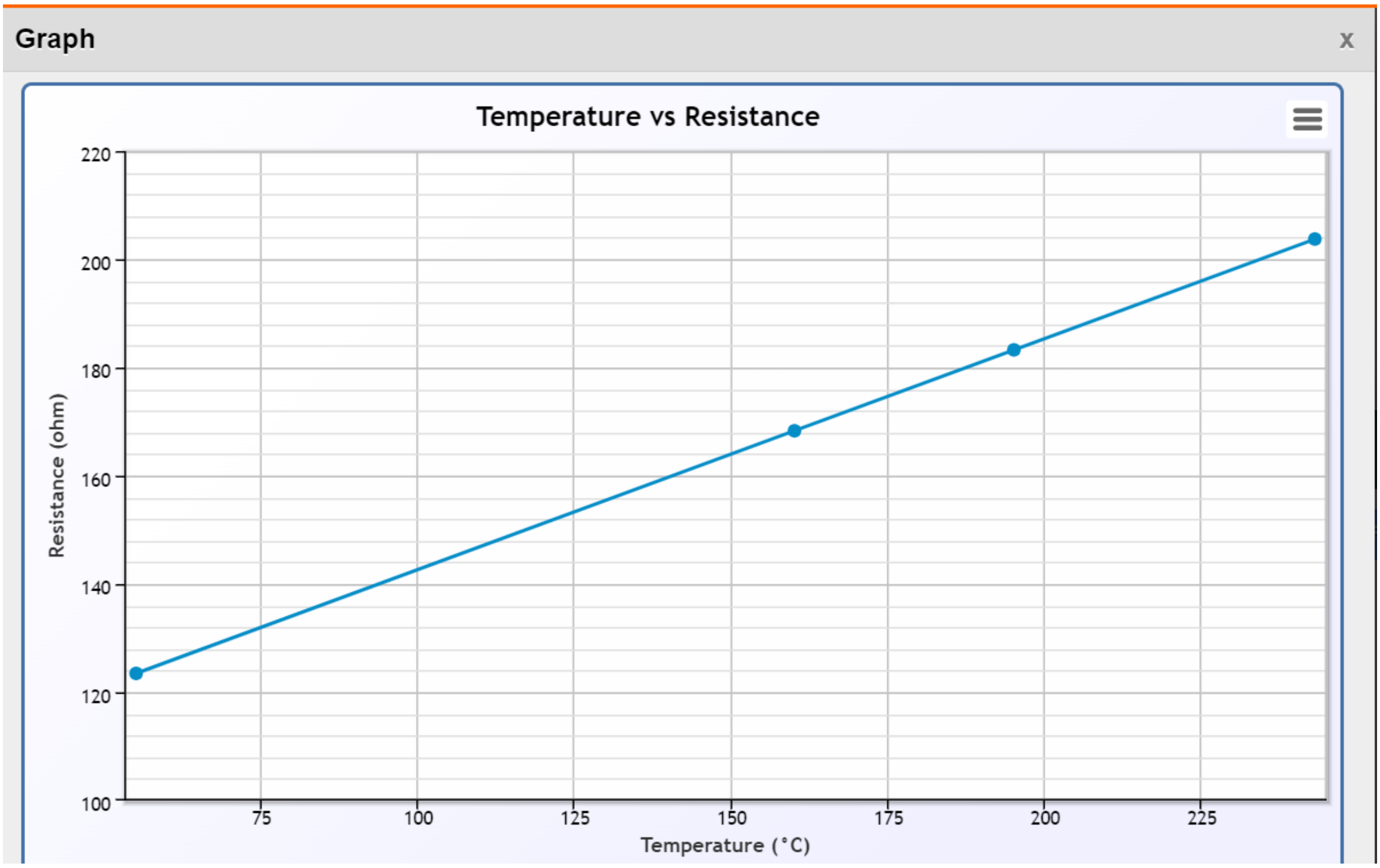
Static Characteristics:

The effect of process temperature on RTD probe resistance is analysed.

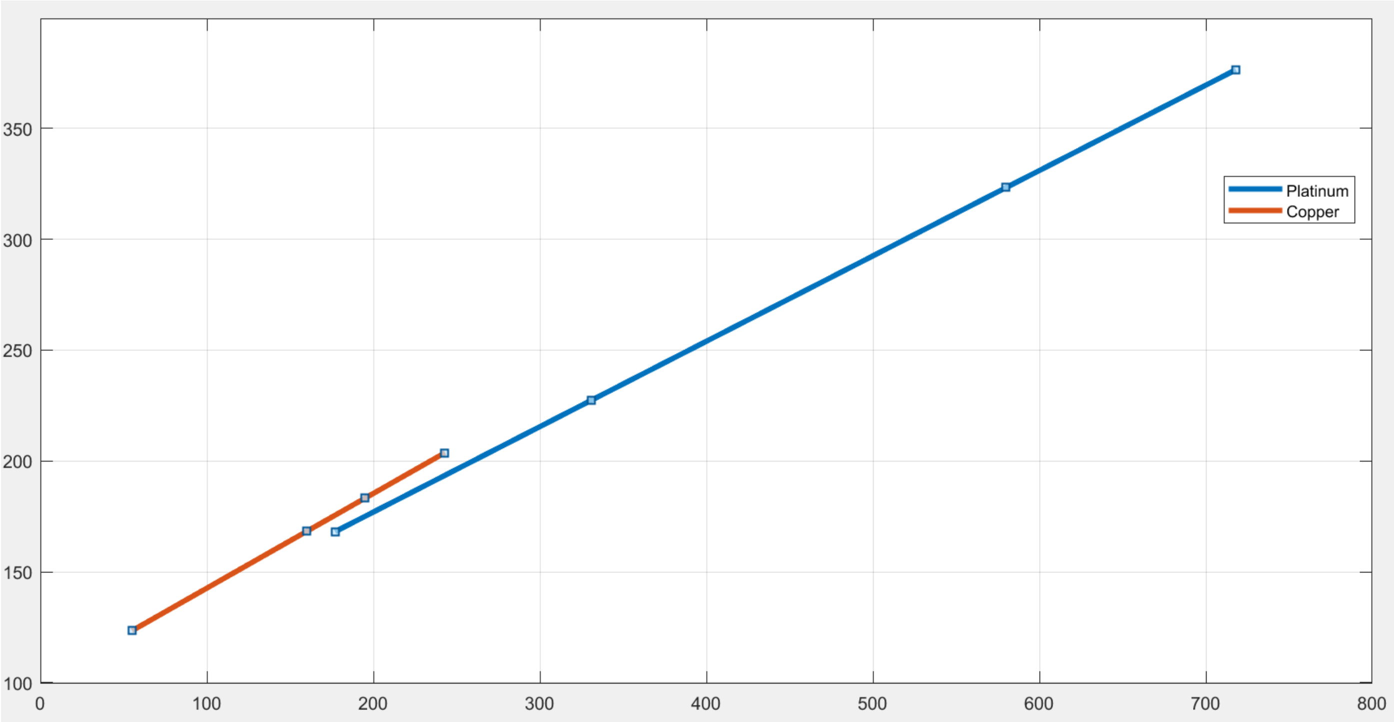
Platinum:



Copper:

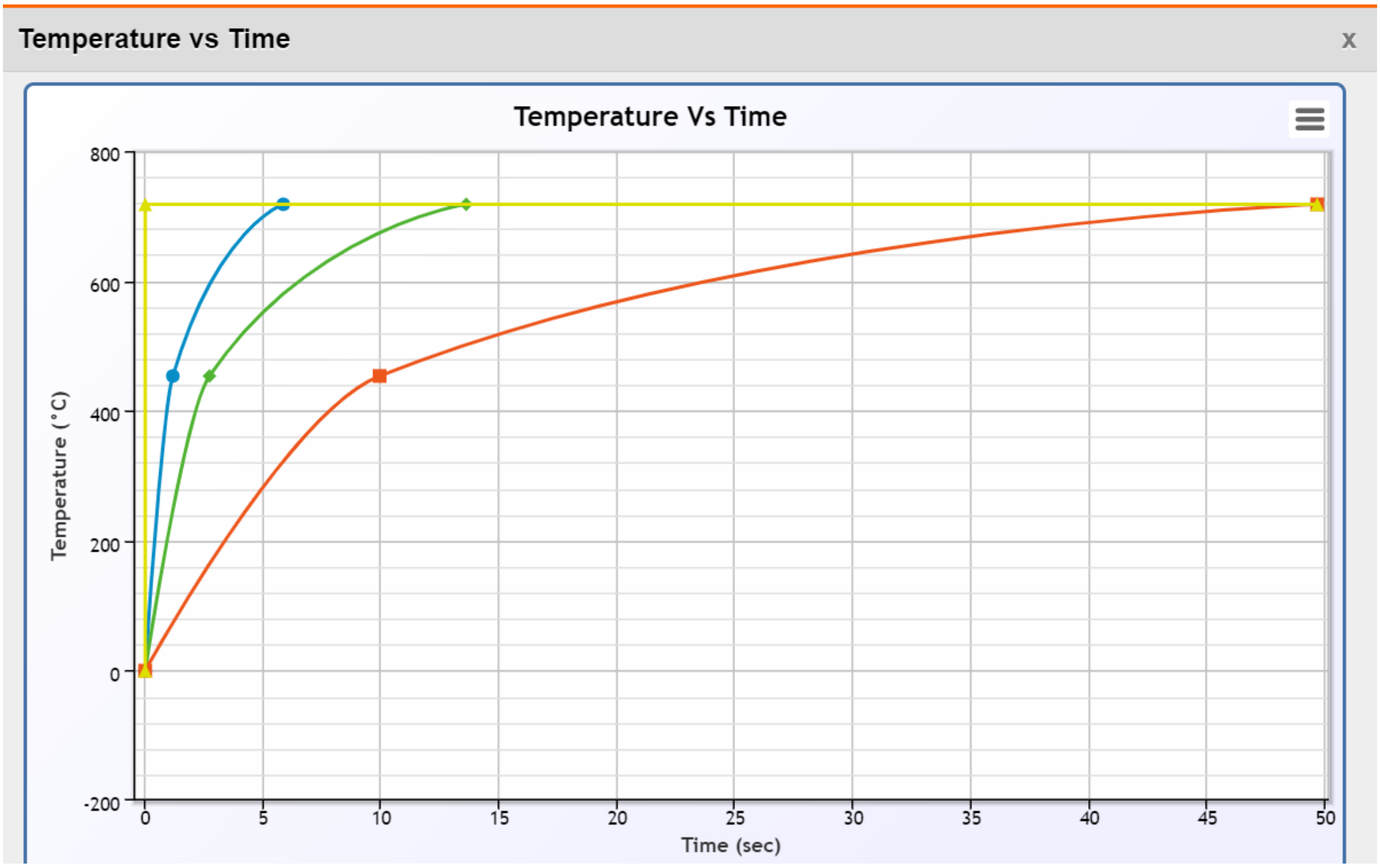


<u>Comparing copper and platinum:</u>



Dynamic Characteristics:

Platinum RTD is utilised to calculate the time constant values in this experiment. A graph is plotted showing RTD's response to a step change in the input.



Experiment-12

Name: Soma Anirudh

Slot: L19+L20 BECE101P

Date: May 19th, 2022

Register number: 21BCE5537

Experiment number: 12

Strain Gauge Sensor

Aim: To understand the working principle of Strain gauge

Objectives :

1. Plot the characteristics of Strain gauge.
2. Understand the effect of various parameters on the strain gauge performance.

Software Required : Virtual Lab (https://sl-coep.vlabs.ac.in/)

Theory:

A strain gauge is a sensor whose resistance varies with applied force; it transforms force, pressure, tension, weight, and other variables into an electrical resistance change that can be measured. Stress and strain are produced when external forces are applied to a stationary object.

The force created within an item in reaction to an applied external force is known as stress. Stress is defined as the internal force divided by the object's cross-sectional area, and it is measured in Pa (Pascal) or N/m2. Vertical stress occurs when the direction of an external force is vertical to the cross-sectional area.

When a bar is pulled, it changes length by L, resulting in a new length of L (original length) + ΔL (change in length). Strain is defined as the ratio of this change in length, ΔL to the initial length, L. The strain is measured in (epsilon) units: $\varepsilon = \Delta L / L$ .Longitudinal strain is strain that runs in the same direction as the external force.

Strain is an absolute quantity with no unit because it is a ratio. Lateral strain is defined as strain in the direction perpendicular to the external force. Each material has a certain lateral to longitudinal strain ratio. Poisson's ratio is the title given to this ratio.

$V = -(d\varepsilon(trans)/ d\varepsilon(axial))$, where V is the Poisson's ratio, (trans) is transverse strain (negative for axial tension (stretching), positive for axial compression), and (axial) is axial strain (positive for axial tension, negative for axial compression). The strain is exactly proportional to the value of stress. Thus, if we can discover the strain caused by an external force, we can find the stress in a material.

Based on the principle of operation, there are several different types of strain gauges:

Mechanical: It is made up of two layers of plastic. A ruled scale is on the bottom layer, while a red arrow or pointer is on the top layer. On one side of the crack, one layer is glued, and on the other, one layer is glued.

The layers glide past one another gently as the crack opens, and the pointer goes over the scale. As the fracture grows, the red crosshairs on the scale shift.

Mechanical strain gauges can be much more primitive. A piece of plastic or glass is stuck over a fracture and the nature of the crack is noted.

Electrical: Electrical strain gauges are tiny, rectangular-shaped strips of foil with maze-like wiring patterns on them that connect to a pair of electrical lines.

The foil strip is slightly twisted out of shape as the material is strained, and the maze-like wires are either pulled apart (so their wires are stretched somewhat thinner) or pushed together (so the wires are pushed together and become slightly thicker).

The electrical resistance of a metal wire changes as its width changes. The stress exerted is proportional to the change in resistance. The deformation is elastic if the forces are minimal, and the strain gauge gradually returns to its original shape.

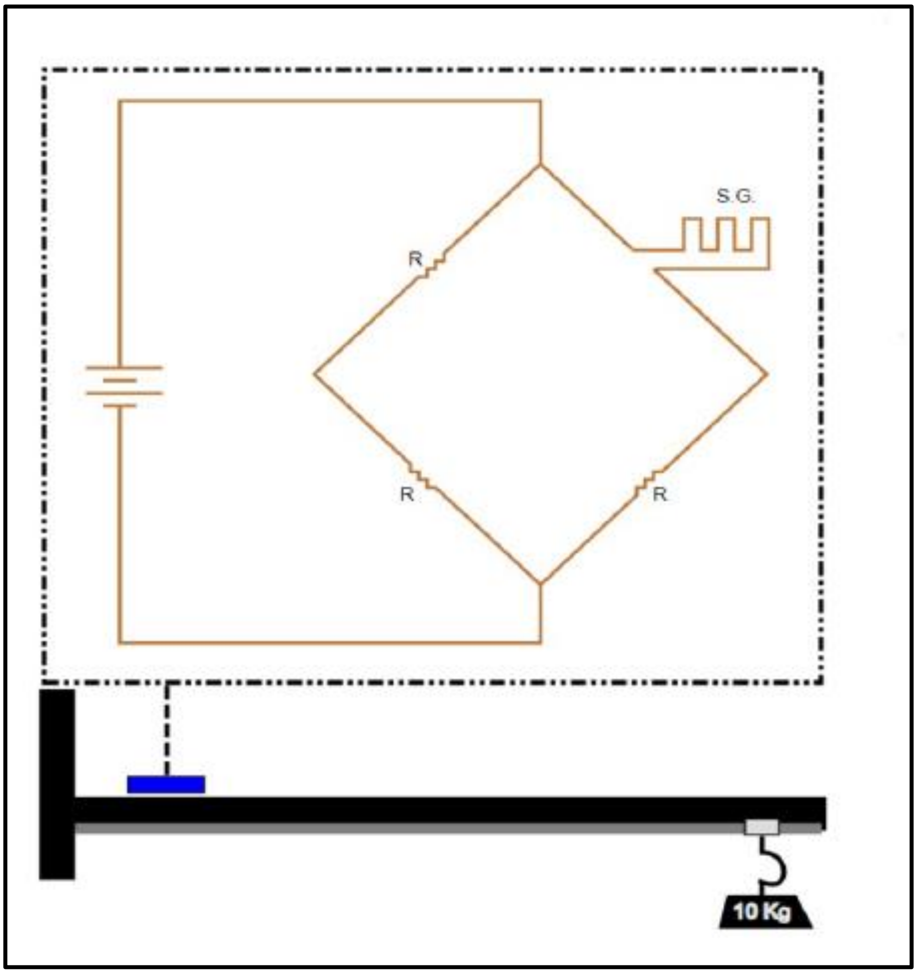
Piezoelectric: Some materials, such as quartz crystals and several types of ceramics, act as "natural" strain gauges. They create modest electrical voltages between their opposing faces when pressed and pulled. The term for this phenomena is piezoelectricity. We can simply determine the strain by monitoring the voltage from a piezoelectric sensor. The most sensitive and dependable strain gauges are piezoelectric strain gauges.

Gauge Factor:

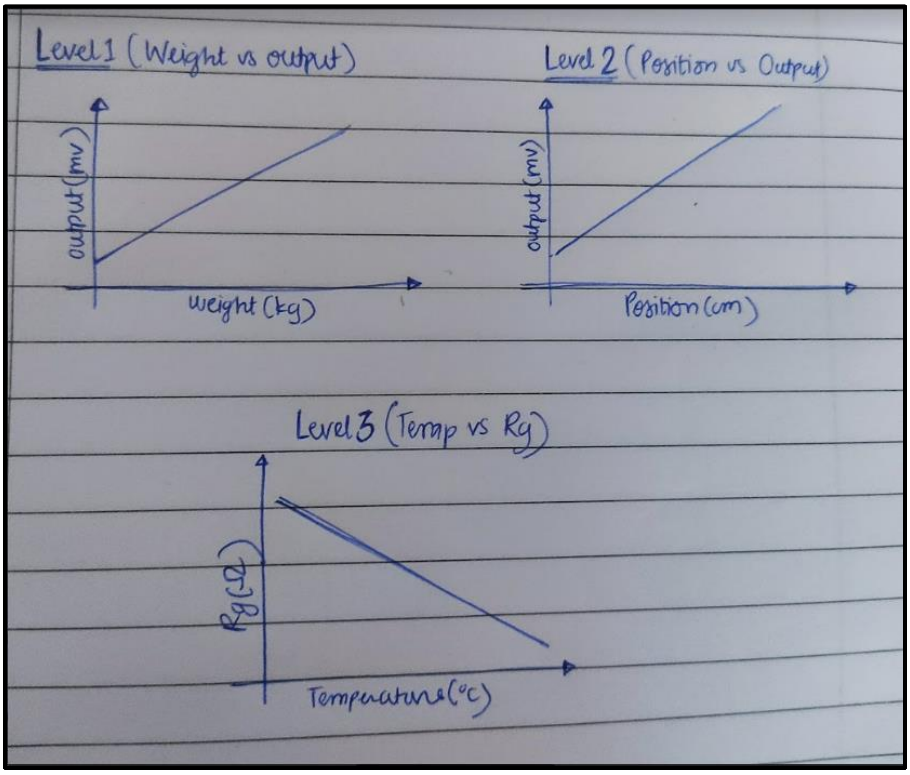
The sensitivity of strain gauges is characterised in terms of the gauge factor (gauge factor). The gauge factor is defined as the change in resistance per unit change in strain gauge wire length, expressed as

G.F. = (R/RG) / ε, where R represents the change in resistance due to strain, RG represents the resistance of the unreformed gauge, and ε represents strain.

Block-Diagram:

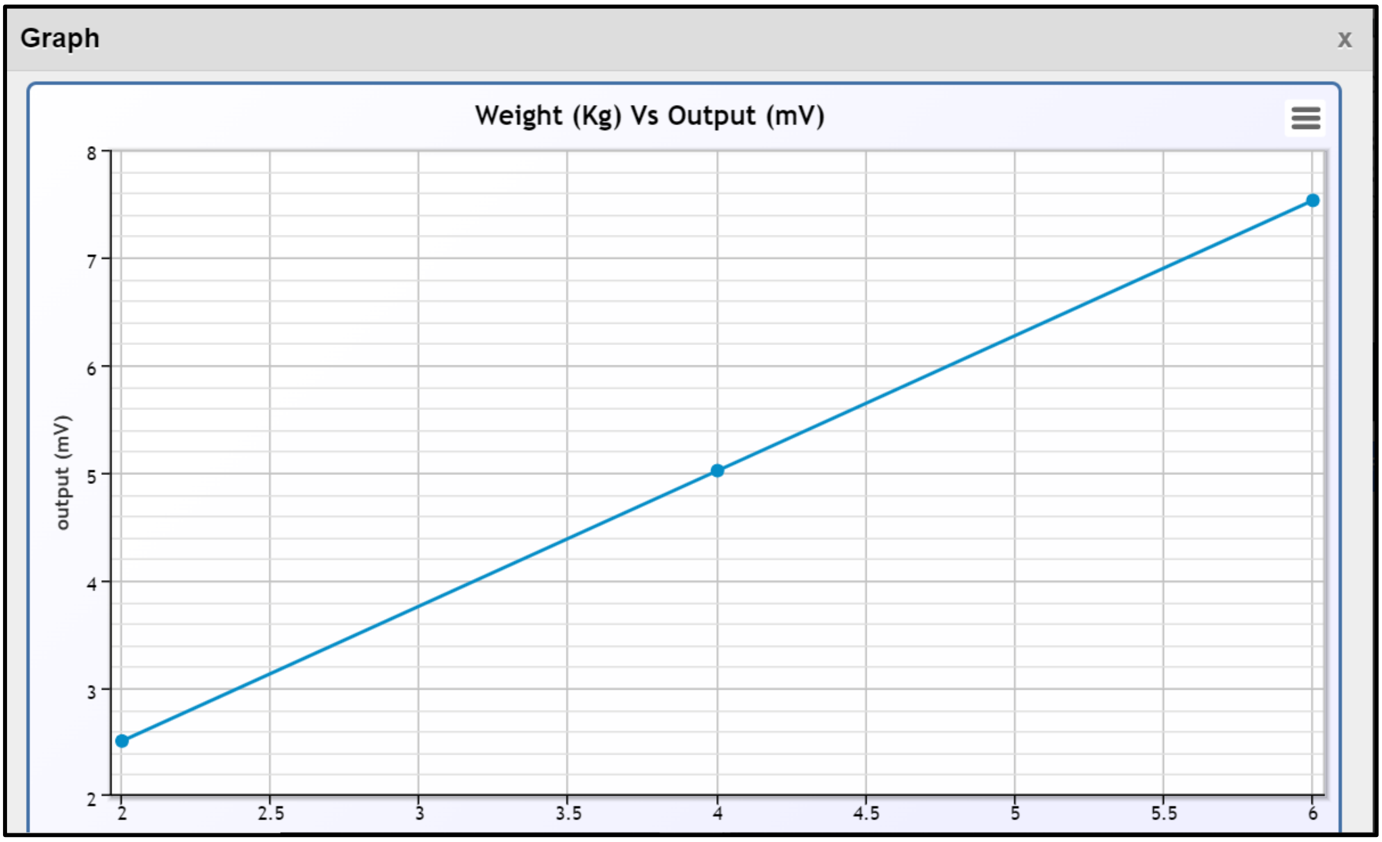


Model Graphs:

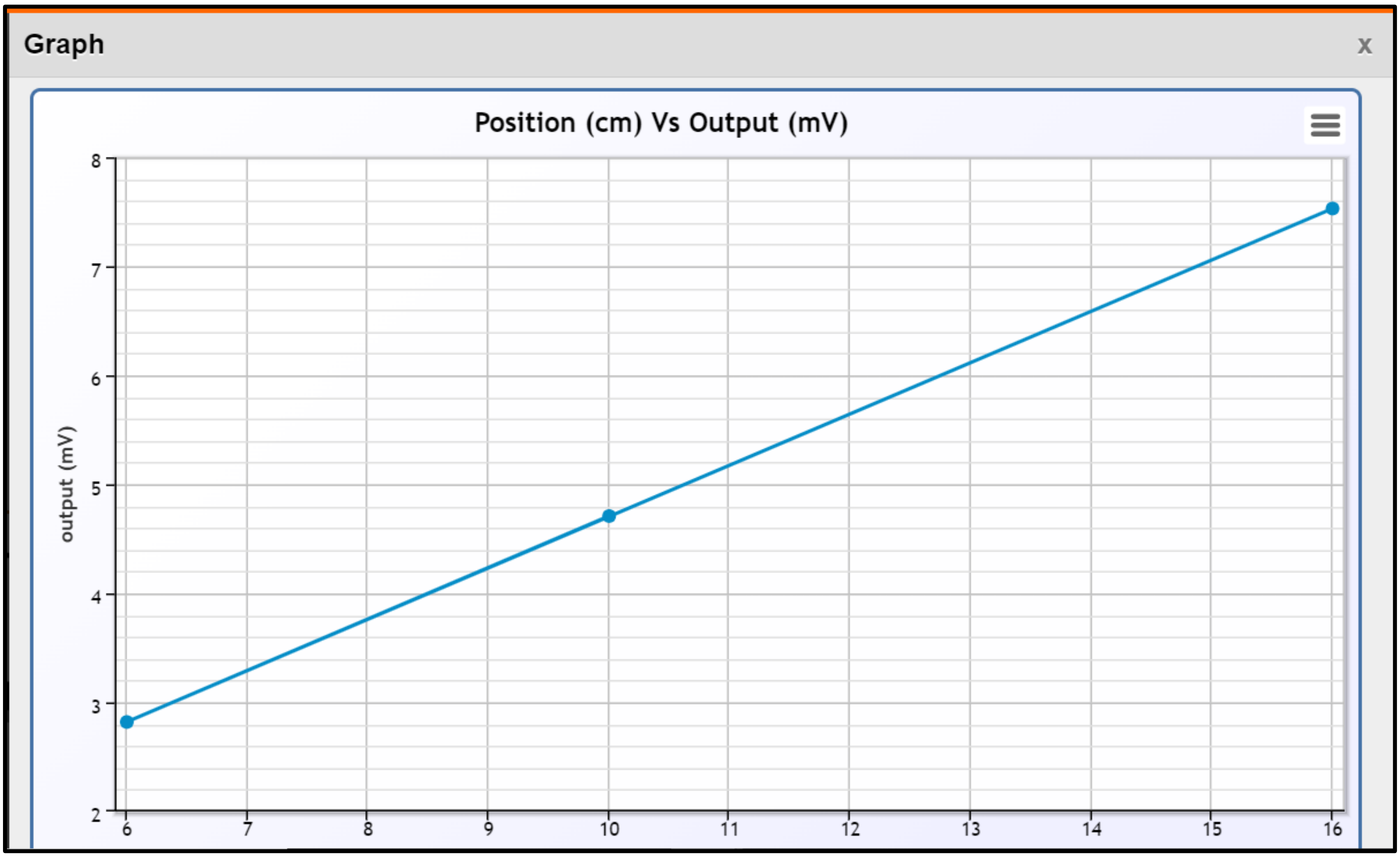


Output Graphs:

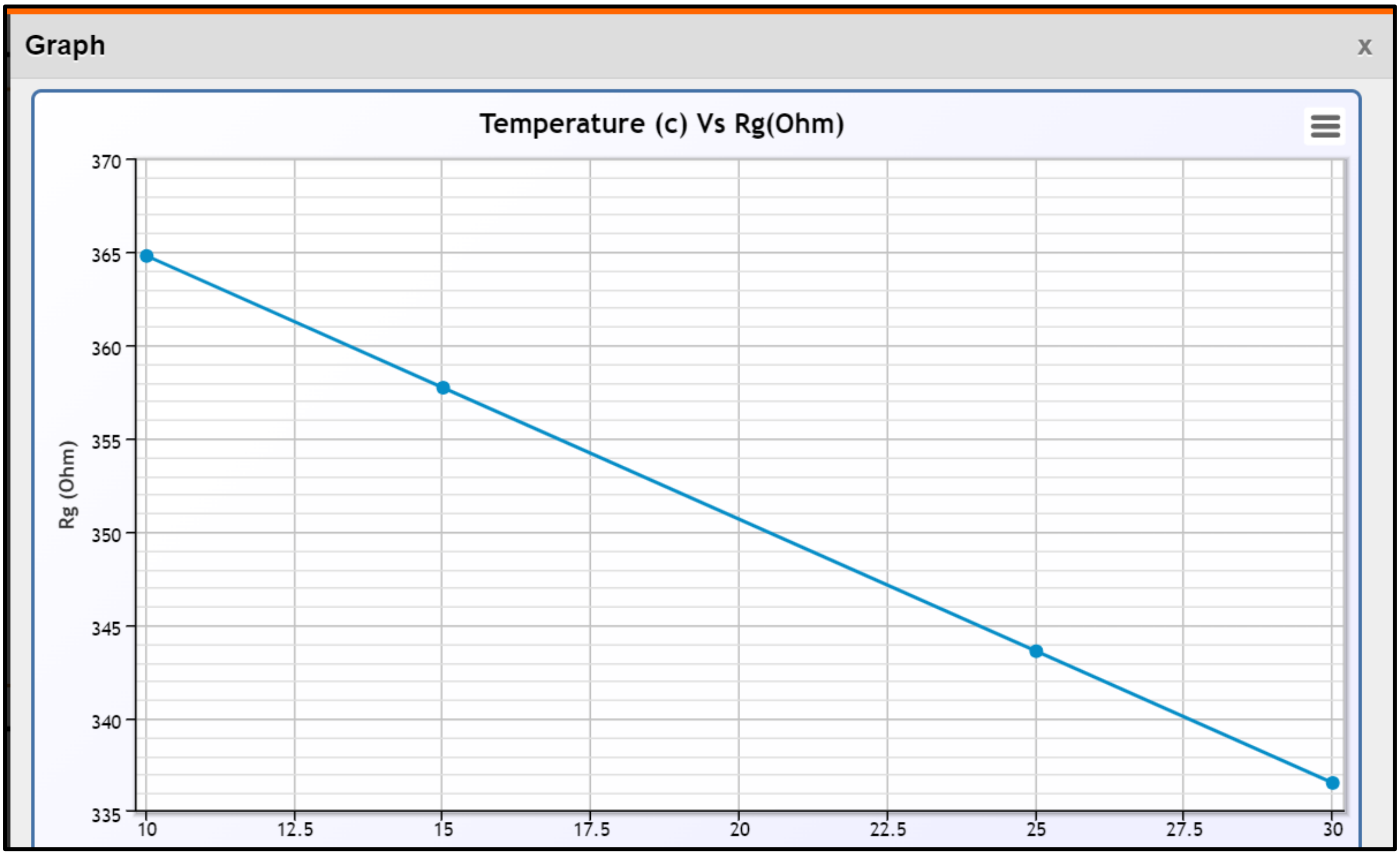
Level-1:



Level-2:



Level-3:



Results :

i. Strain gauges provide precise values with a change in temperature

ii. At constant resistance and gauge factor, the value of output voltage increases with increase in weight attached.

iii. Strain gauges are appropriate for longer periods with certain precautions.

Inferences :

i. Strain gauges are easy to manufacture because of simple components.

ii. Easy to maintain and have a long operating life.

iii. It is totally encapsulated to protect from damages like - handling and installation ins they are used to monitor structures and avoid accidents well in advance in engineering.

Experiment-13

Name: Soma Anirudh

Slot: L19+L20 BECE101P

Date: May 19th, 2022

Register number: 21BCE5537

Experiment number: 13

Linear Variable Differentiable Transformer

Aim: To understand working principle of LVDT

Objective:-

1. Study the relation between core displacement and output of LVDT
2. Understand the effect of change in supply frequency on LVDT performance
3. Understand the effect of change in excitation (supply) voltage on LVDT performance

Software Required : Virtual Lab (https://sl-coep.vlabs.ac.in/)

Theory:

The moveable armature and the outer transformer windings are the two major components of an LVDT. Three windings make up an LVDT. The primary winding is in the centre, while the secondary windings are on the sides. The secondary's are symmetrically positioned around the primary and are identical. In series-opposition, the secondary coils are linked.

The core of an LVDT is the moving element. It's a ferromagnetic armature with a cylindrical shape. It may freely travel along the tube's axis. The core is attached to an item whose movement is being measured on one end, while the other end travels freely inside the hollow bore of the coil.

The main circuit is powered by an alternating current. The amplitude and frequency of this current must be adequate. Primary Excitation is another name for it. The frequency is typically between 1 and 10 kHz. Each secondary is induced with a voltage proportionate to its mutual inductance with the primary by this current. While the induced voltage has the same frequency as the excitation voltage, its amplitude changes depending on the location of the iron core. The voltages generated in the secondary change as the core travels owing to variations in mutual inductance. The coils are linked in series but in the opposite phase, resulting in an output voltage equal to the difference between the two secondary voltages.

When the core is moved in one direction, the voltage in one coil rises relative to the other, causing the output voltage to rise from zero to a maximum value. The main voltage is in phase with this voltage. The output voltage grows from zero to a maximum value when the core goes in the opposite direction, but the phase is reversed from the primary. The output voltage's magnitude is proportional to the distance travelled by the core. The direction of the displacement is indicated by the phase of the voltage.

Residual Voltage: At the null position, the output voltage should be zero. However, a non-zero voltage occurs at the null location due to harmonics in the excitation voltage and stray capacitance coupling between the primary and secondary. The term for this is "residual voltage." It is within acceptable limits if it is less than 1% of full-scale output voltage (which is the common scenario).

<u>*LVDT types based on applications:*</u>

LVDT for General Purposes: can be used in a variety of industrial and research applications. For delicate measuring and quality control applications, use a precision LVDT.

Submersible LVDT: Hermetically sealed for use in corrosive fluids and gases, high temperature and vibrations, and other industrial and research situations.

Types of LVDTs based on their operating range:

<u>Short stroked:</u> full-scale linear ranges from ±0.01 inch (±0.25 mm) to ±0.5 inch (±12.7 mm)

<u>Long stroked</u>: full-scale linear ranges from ±0.5 inch (±12.7 mm) to ±18.5 inch (±470 mm)

Types of LVDT based on excitation used

<u>AC LVDT:</u> AC LVDT is a type of LVDT that is based on excitation. AC LVDTs are stimulated by an AC voltage with a frequency of 50 hertz to 25 kHz, with a nominal value of 2.5 kHz. The carrier frequency is usually set to be at least 10 times higher than the core motion's highest predicted frequency. AC-operated LVDTs are often smaller and more precise than DC-operated LVDTs. They can withstand more dramatic fluctuations in operational temperature than DC LVDT.

<u>DC LVDT:</u> Onboard oscillator, carrier amplifier, and demodulator circuitry are included in the DC LVDT. The benefits of DC-operated ("DC-to-DC") LVDTs include ease of installation and signal conditioning, as well as the ability to run from dry cell batteries in distant areas and cheaper system costs (especially in multipoint applications). The DC LVDT is temperature restricted, generally functioning between -40°C and +120°C.

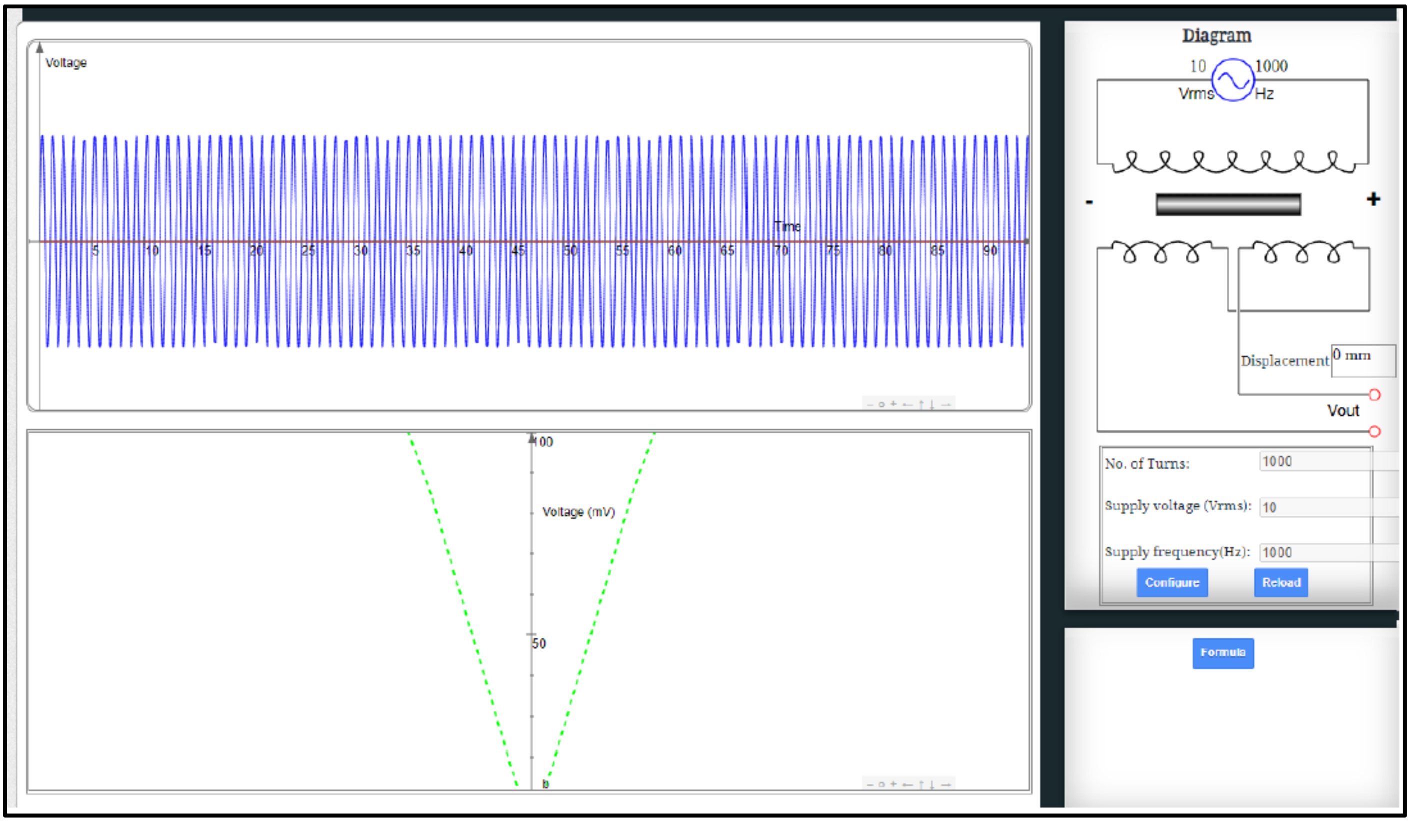
Types of LVDT based on armature:

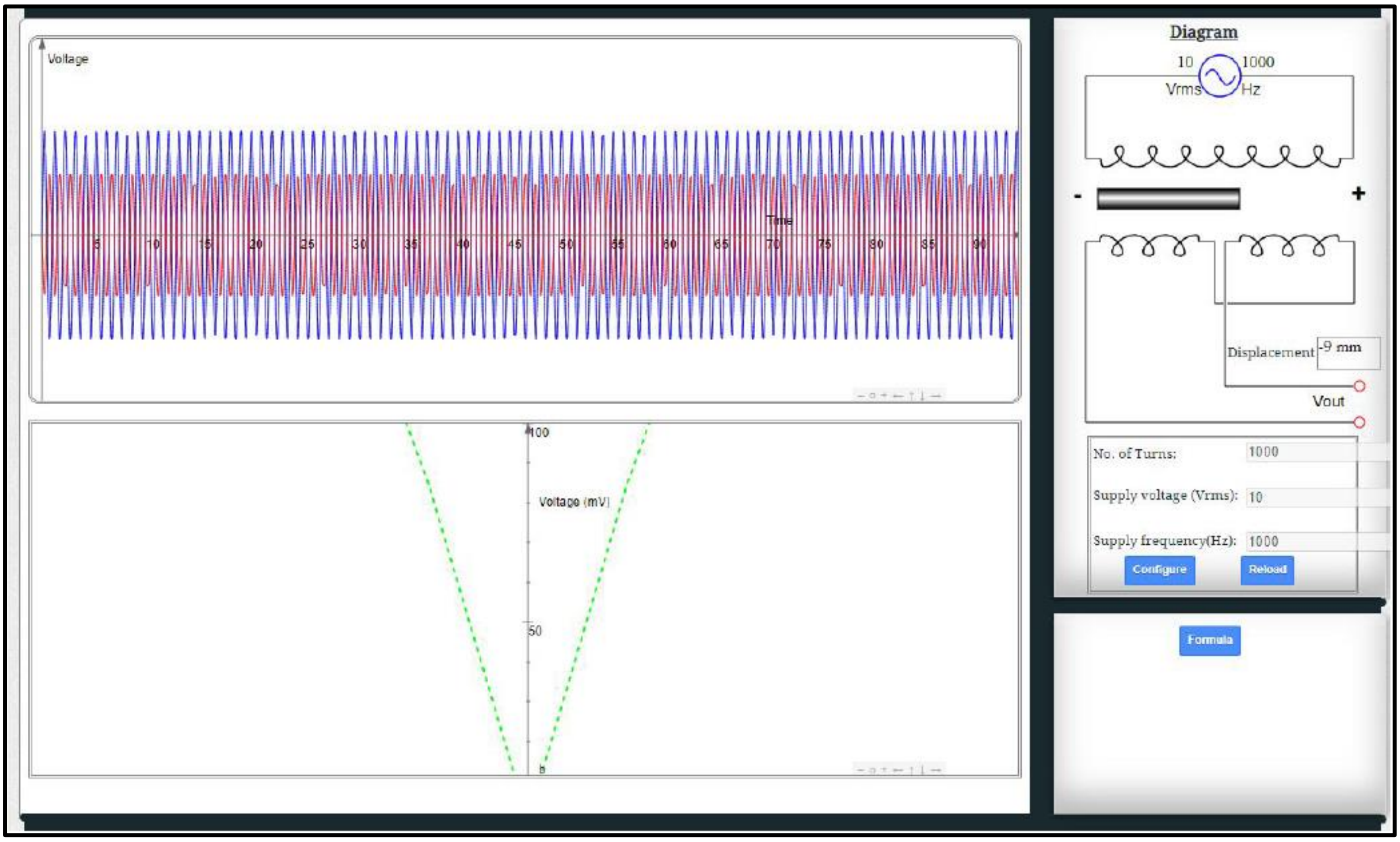
<u>Unguided Armature:</u> The armature fits loosely in the cavity of the coils bore in this design. To ensure adequate movement along the axis, correct installation is required. This enables for smooth, wear-free movement. This kind has an indefinite fatigue life as well as high repeatability and resolution. Short-range, high-speed applications are best served by free armature.

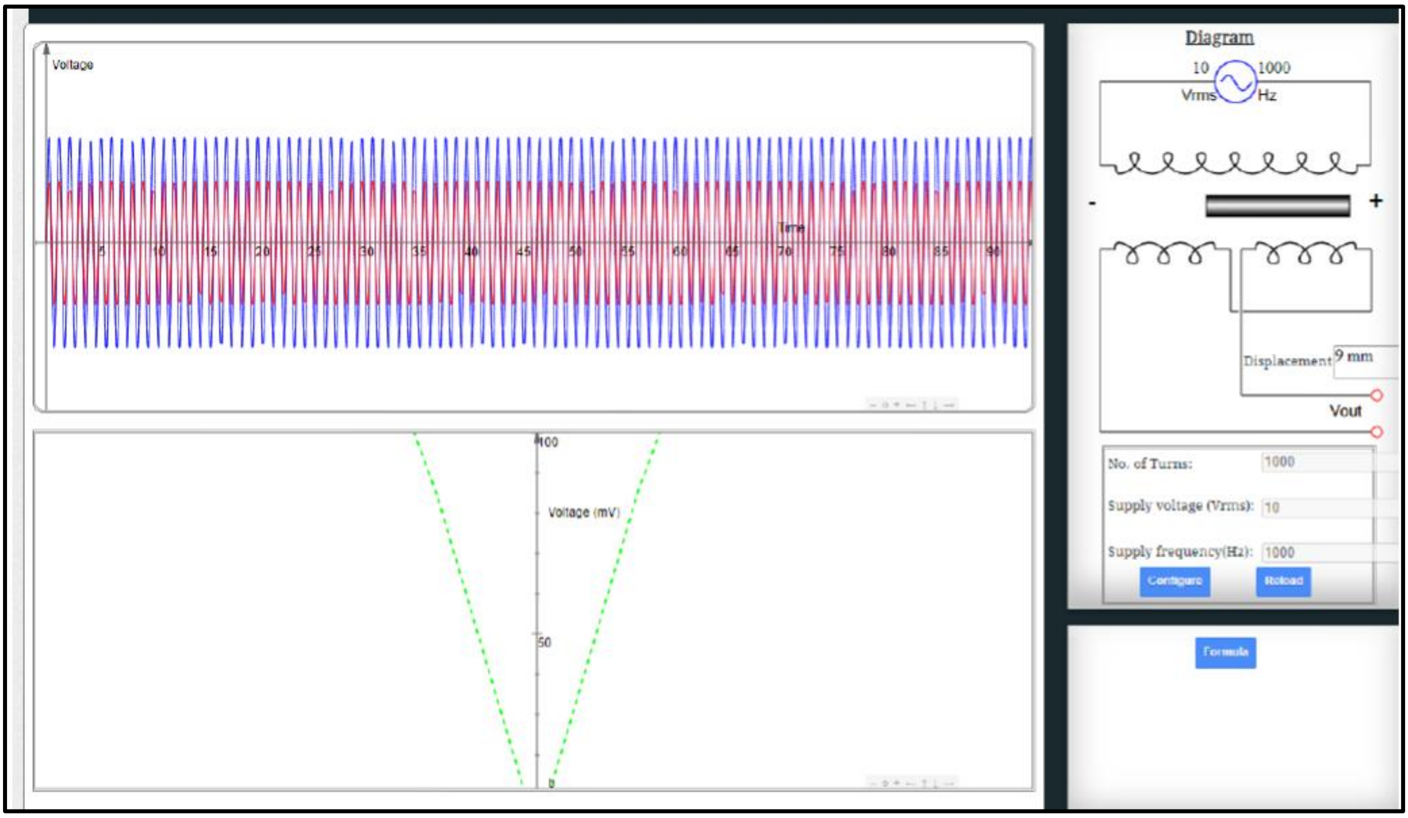
<u>Guided (Captive) Armature:</u> The armature is constrained and guided by a low friction bearing assembly in a guided (captive) armature. These are ideal for working across long distances. The armature is directed to eliminate the chance of misalignment.

<u>Spring Extended Armature:</u> This armature is similar to a guided armature LVDT, but it contains an internal spring that pushes the armature to its utmost extension continually. This ensures that mild and consistent contact is maintained with the measured object.

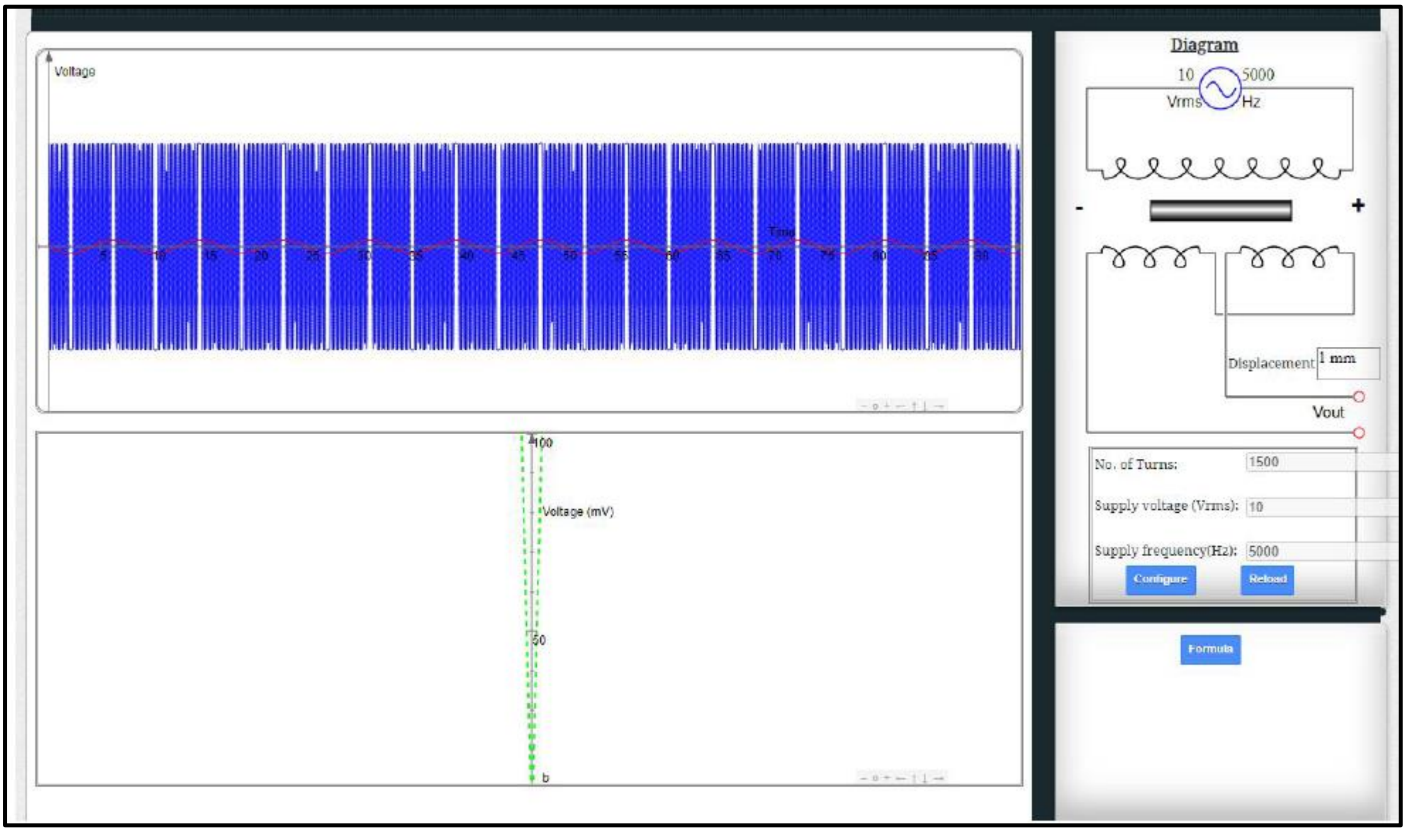
Simulation to study the relation between core displacement and output of LVDT.



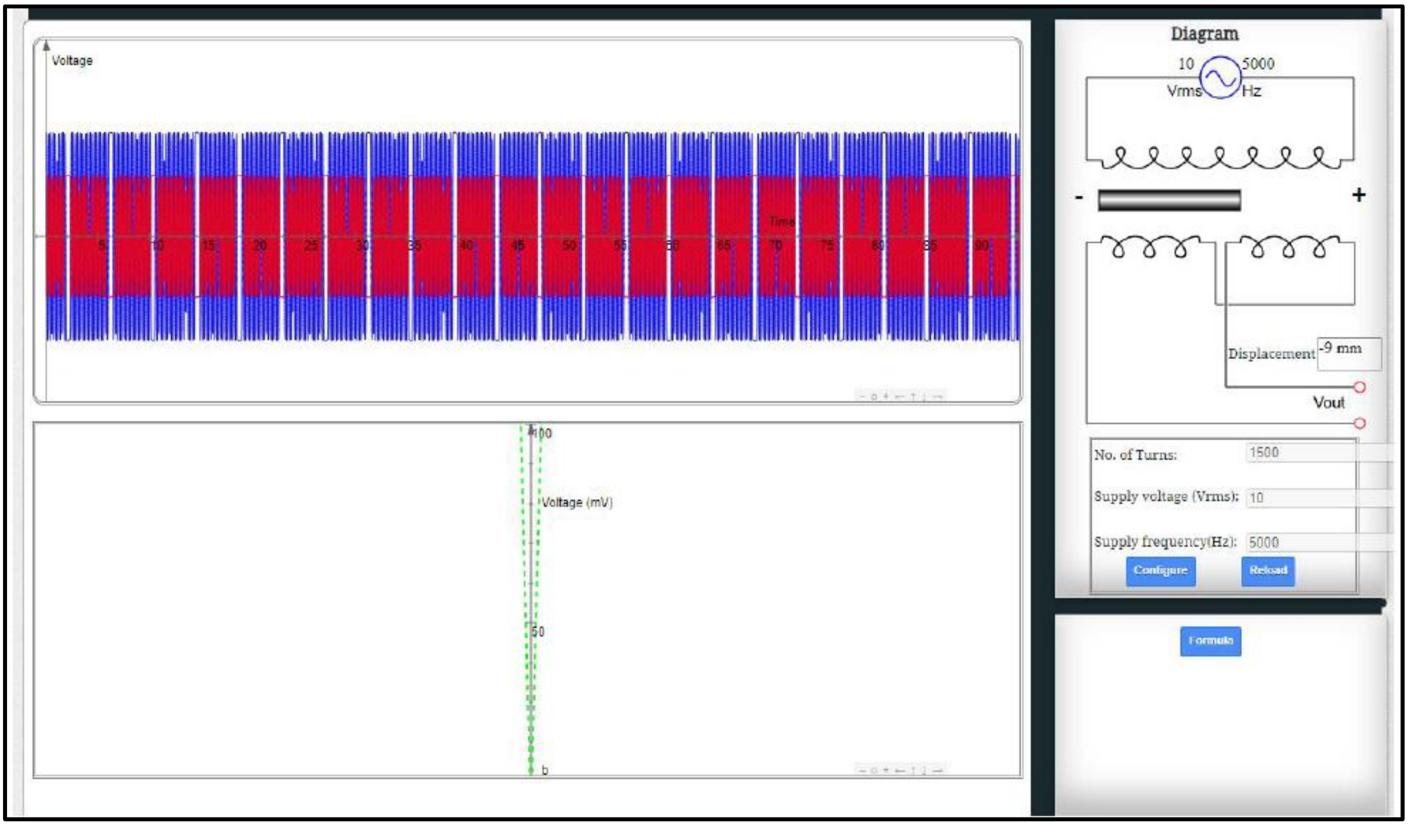

Voltage
Time
5
10
15
20
25
30
35
40
45
50
55
60
65
70
75
80
85
90
100
Voltage (mV)
50
0
Diagram
10
1000
Vrms
Hz
-
+
Displacement
-9 mm
Vout
No. of Turns:
1000
Supply voltage (Vrms):
10
Supply frequency(Hz):
1000
Configure
Reload
Formula


Voltage
Time
5
10
15
20
25
30
35
40
45
50
55
60
65
70
75
80
85
90
100
Voltage (mV)
50
0
Diagram
10
1000
Vrms
Hz
-
+
Displacement
9 mm
Vout
No. of Turns:
1000
Supply voltage (Vrms):
10
Supply frequency(Hz):
1000
Configure
Reload
Formula

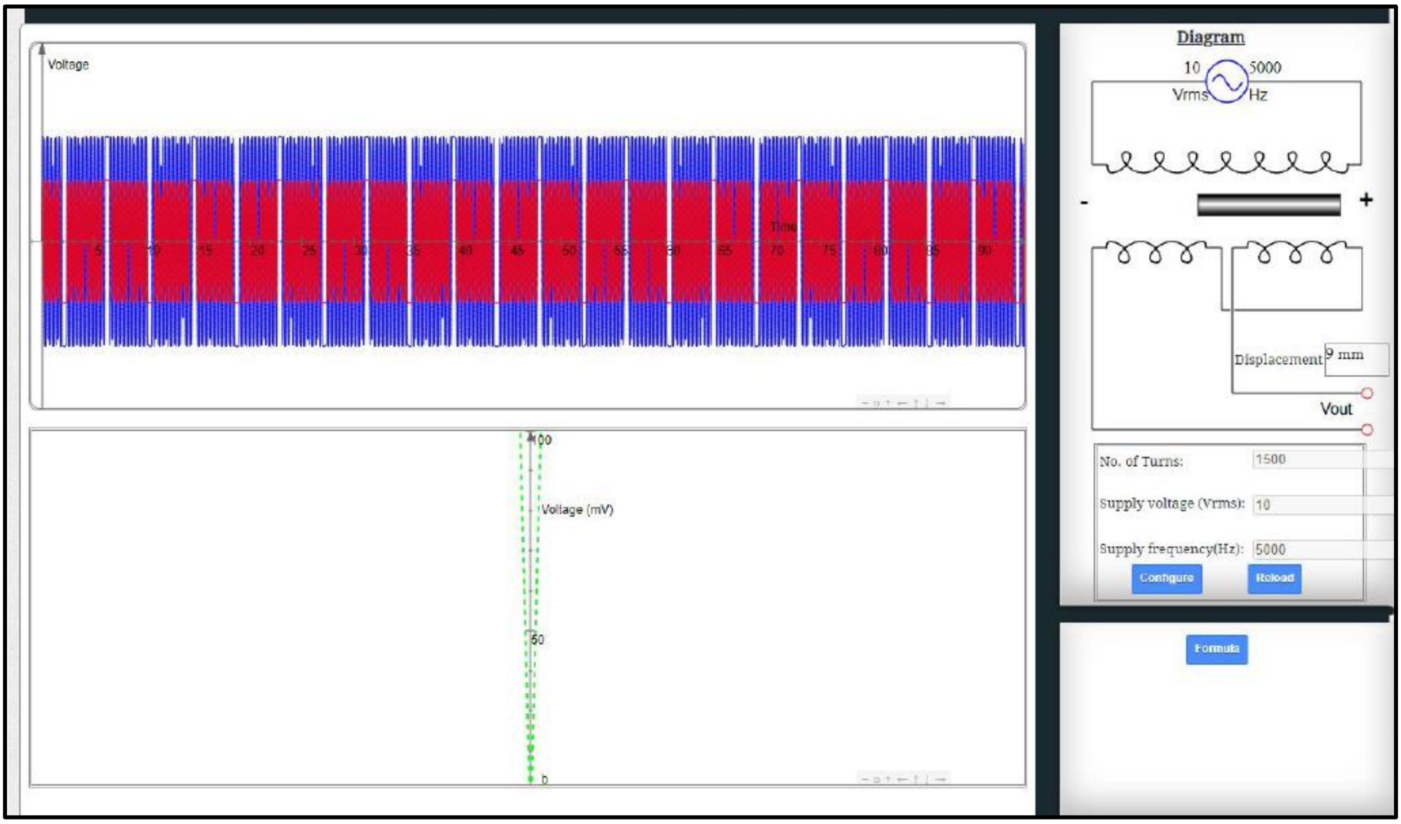
Simulation to understand the effect of change in supply frequency on lvdt performance by changing frequency :



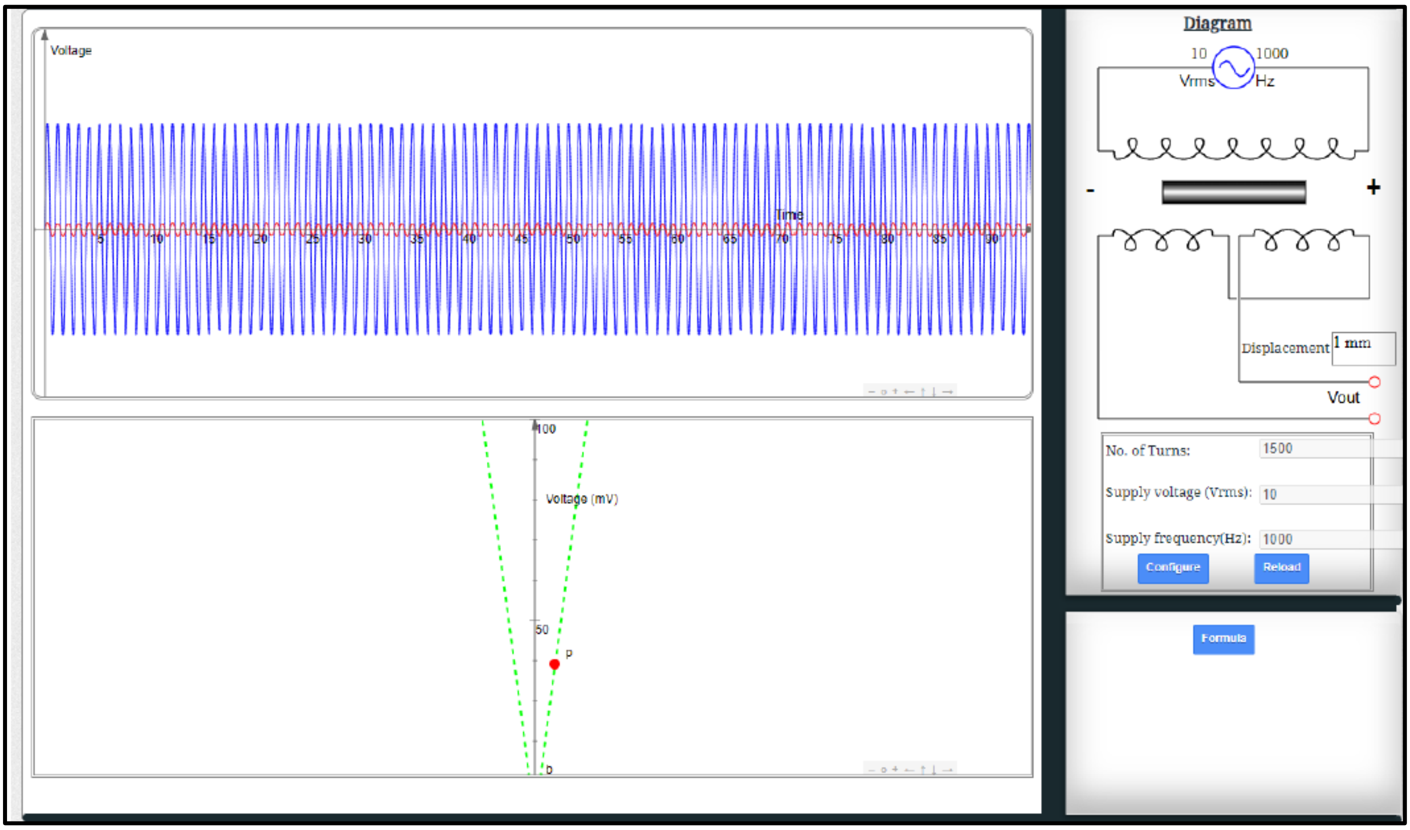
Negative displacement



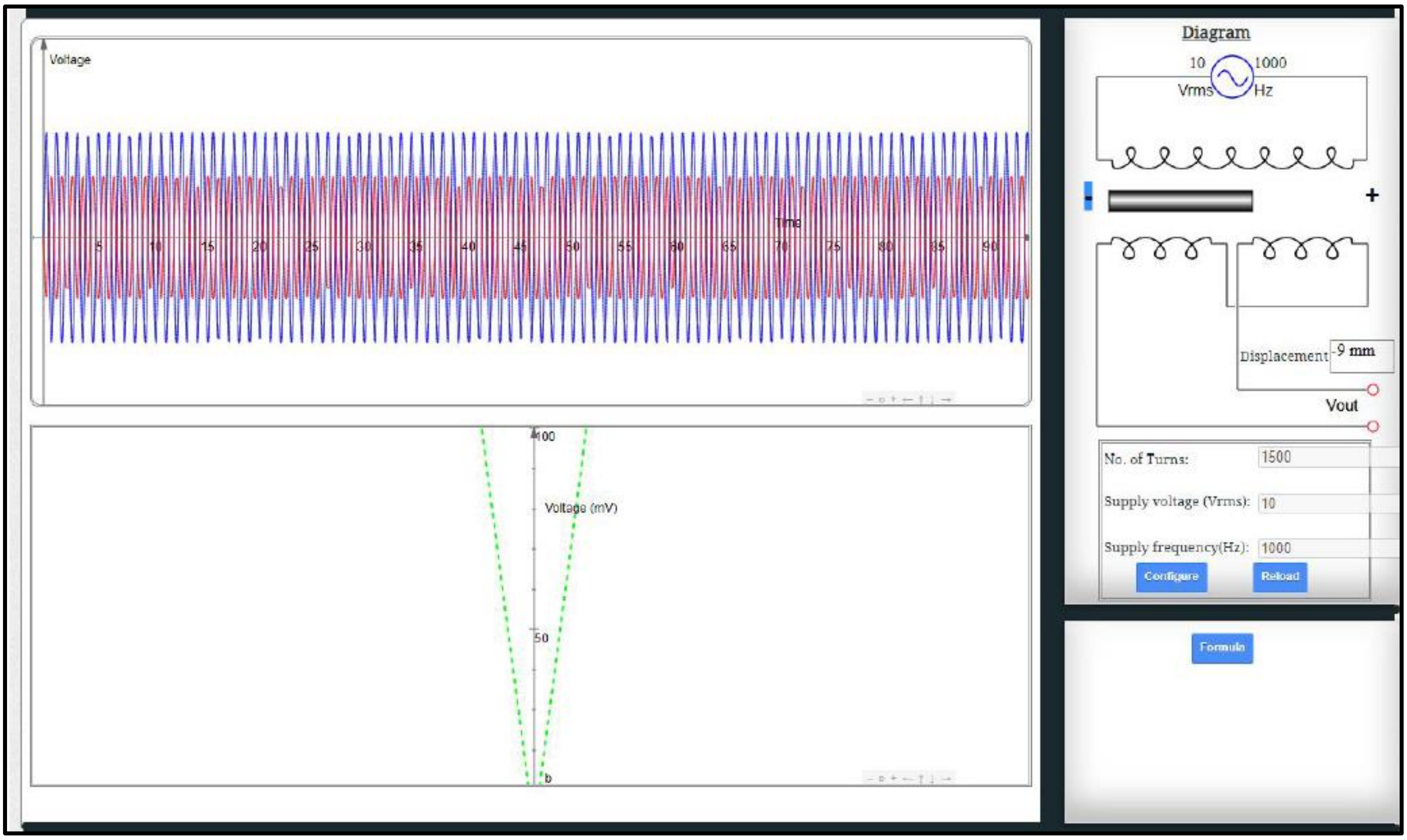
Positive displacement



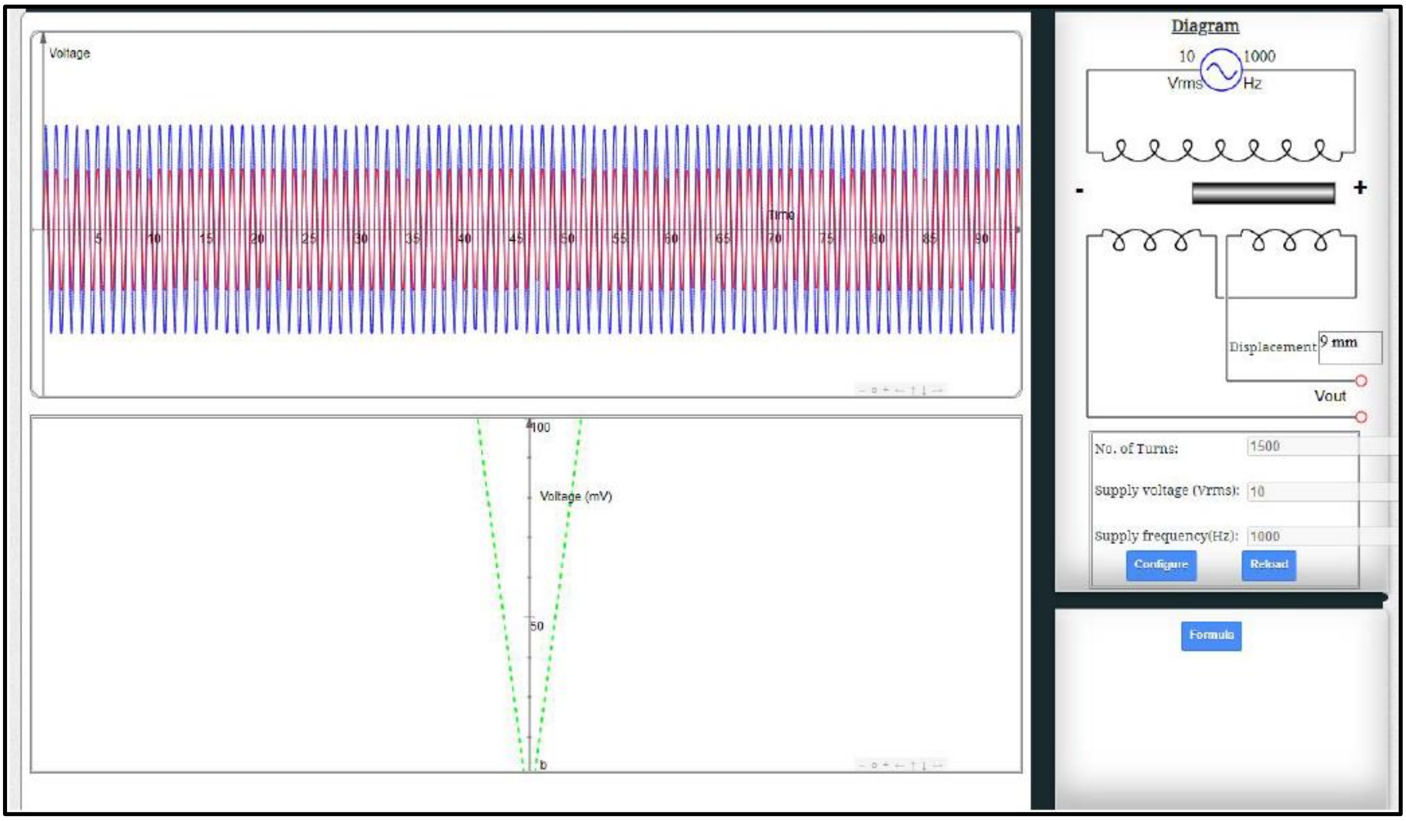
Simulation to understand the effect of change in excitation (supply) voltage on lvdt performance by changing number of turns:



Negative displacement



Positive displacement:



Experiment-14

Name: Soma Anirudh

Slot: L19+L20 BECE101P

Date: May 26th, 2022

Register number: 21BCE5537

Experiment number: 14

Challenging Expt (NTC thermistors)

Aim: To understand working principle of thermistors

Software Required : LtSpice

Theory:

NTC stands for "Negative Temperature Coefficient."

NTC thermistors are resistors having a negative temperature coefficient, which implies that as the temperature rises, the resistance lowers.

They're mostly employed as current-limiting devices and resistive temperature sensors.

The temperature sensitivity coefficient is approximately five times that of silicon temperatu re sensors (silistors) and 10 times that of resistance temperature detectors (RTDs).

NTC sensors are commonly utilised in temperatures ranging from 55 to 200 degrees Celsius.

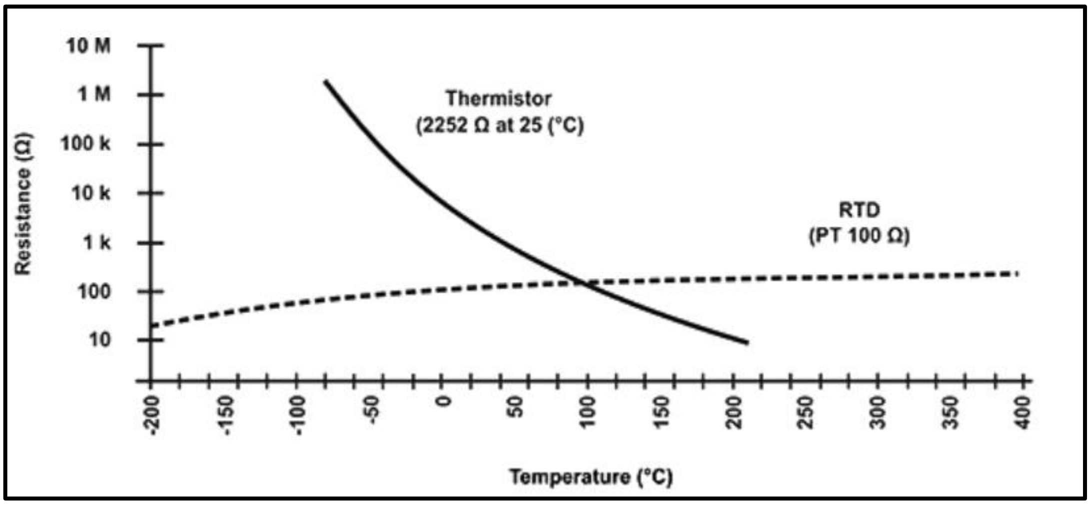
When employing analogue circuits to reliably monitor temperature, the non-linearity of the relationship between resistance and temperature demonstrated by NTC resistors created a significant obstacle.

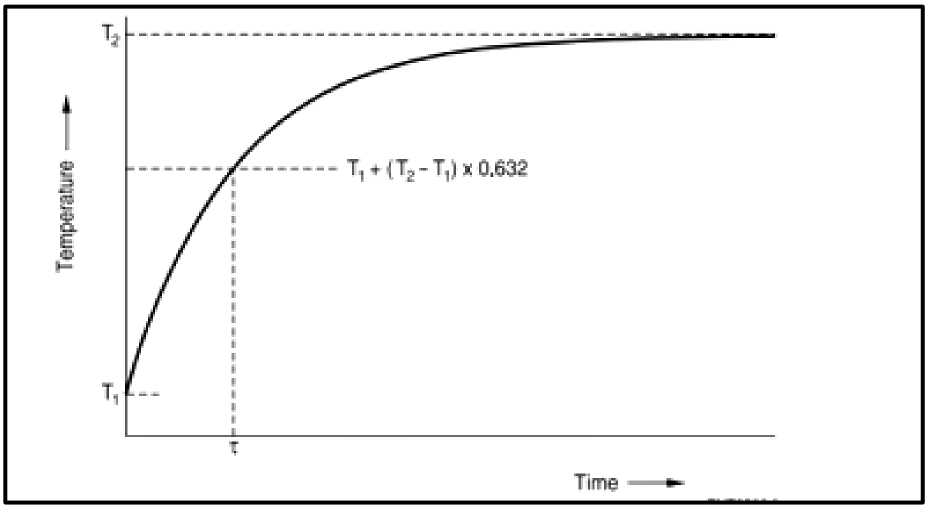
However, the rapid development of digital circuitry alleviated this difficulty by allowing precise values to be computed by interpolating lookup tables or solving equations.

NTC thermistors are often composed of ceramics or polymers, unlike RTDs (Resistance Tem perature Detectors), which are built of metals.

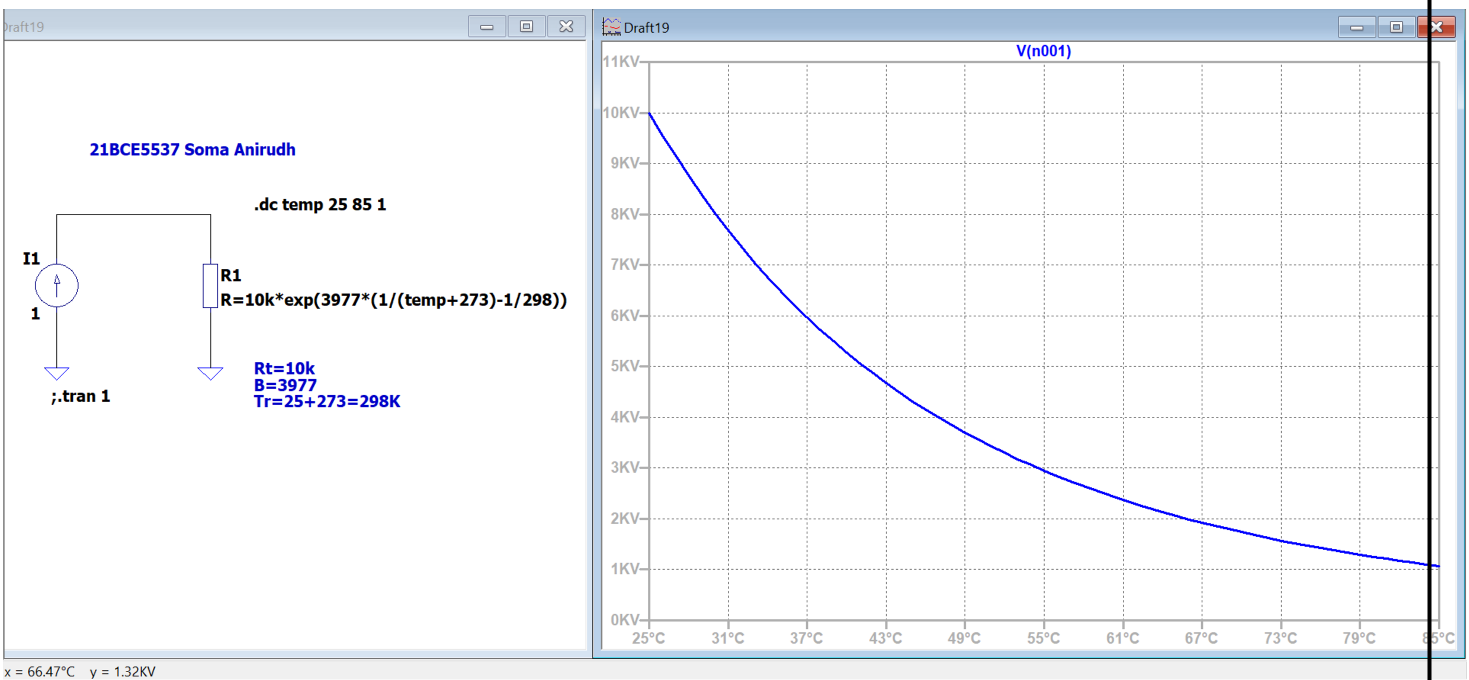
NTC thermistors are made from a variety of materials, which result in a wide range of temp erature responses and other performance characteristics.

Model Graph:

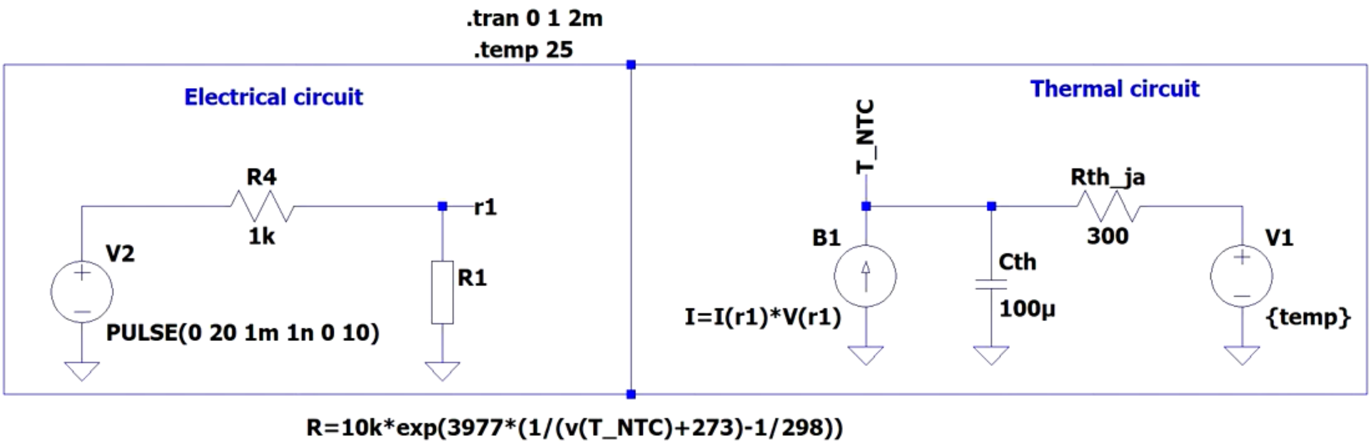




LtSpice Simulations:



Circuit:



Output: