BMS COLLEGE OF ENGINEERING ELECTRONICS AND TELECOMMUNICATION

REPORT ON MULTISIM INTERNSHIP

ACKNOWLEDGEMENT

The satisfaction that accompanies the successful completion of this Multisim Internship would be in-complete without mention of the people who made it possible, without constant guidance and encouragement would have a made efforts go in vain. We consider ourselves privileged to express gratitude and respect towards all from the completion of this internship.

We convey thanks to **professors Dr. Kanamani B** mam, **Prof. Gowra P S** mam, **Prof. Archana K** mam, **Prof. Ambika K** mam of Electronics and Telecommunication department for providing encouragement, constant support and guidance which was of a great help for completion of this internship successfully.

TWO WEEKS INTERNSHIP ON MULTISIM

NI Multisim is an electronic schematic capture and simulation program which is part of a suite of circuit design programs, along with NI Ultiboard. Multisim Live is a web-based electronic schematic capture and simulation tool with built- in SPICE based simulation. NI Multisim and Multisim Live are developed and owned by National Instruments.

OBJECTIVE OF INTERNSHIP

The main objective of this internship is that:

- Due to Pandemic situation, students were not able to attend colleges and didn't have a hands on experience with hardware lab implementations. Hence, this internship provided a platform for students to visualize hardware implementation using multi-sim simulation tool.

- In this multi-sim, students will be able to learn and implement the practical circuits and can observe and analyse the output.

- In this internship, Analog as well as Digital circuit implementation will be taught.

- Here, Students will be able to gain knowledge on Analog and Digital circuit theoretically and practically.

- In the final stage of this internship, student will be able to design & execute a mini project.

Schedule for two weeks online internship on multi-sim
Date: 23/08/2021 – 03/09/2021

Date	Faculty handling	Concepts
Day-1 **23/08/2021**	**Archana K**	• Introduction to multi-sim live tool • Diode Characteristics • Half wave rectifier (With & Without filter) • Bridge rectifier (With & Without filter)
Day-2 **24/08/2021**	**Archana K**	• Full wave rectifier • CE configuration – I/P & O/P characteristics • Basic gates – Truth table verification • Half adder – Implementation using basic gates
Day-3 **25/08/2021**	**Archana K**	• Full adder using basic gates • Half adder & Full adder using Nor gates • Half adder & Full adder using NAND gates
Day-4 **26/08/2021**	**Ambika K**	• Op-amp as voltage follower • Inverting & Non-inverting amplifier • Inverting & Non-inverting summing amplifier • Differentiator & Integrator
Day-5 **27/08/2021**	**Ambika K**	• Precision Half wave & Full wave rectifier • Zero crossing detector & Schmitt trigger • First order low pass & High pass filter • Wein bridge oscillator
Day-6 **30/08/2021**	**P S Gowra**	• Clipping circuits (Single / Double ended) • Clamping circuits (Positive/Negative)

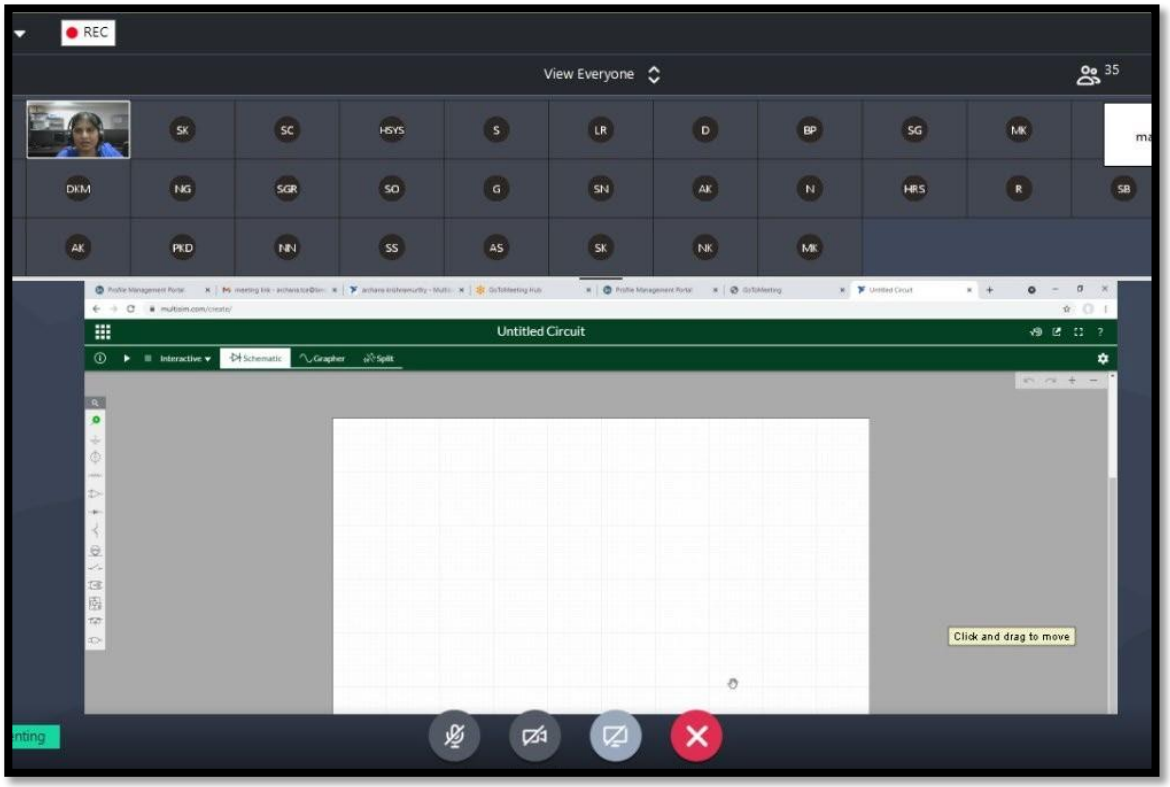
		• Transistor as a Switch • Briefing about mini project and team formation
Day-7 31/08/2021	**P S Gowra**	• Zener as voltage regulator to determine Line / Load regulation • RC coupled Amplifier using multi-sim and virtual lab • BJT as RC phase shift oscillator
Day-8 01/09/2021	**P S Gowra**	• RC coupled amplifier with feedback • Drain & Source characteristics of MOSFET • Finalizing projects and teams
Day-9 02/09/2021	**P S Gowra**	**Project Presentation**
Day-10 03/09/2021	**P S Gowra**	**Project Presentation**

DAY-1: 23/08/2021 – REPORT

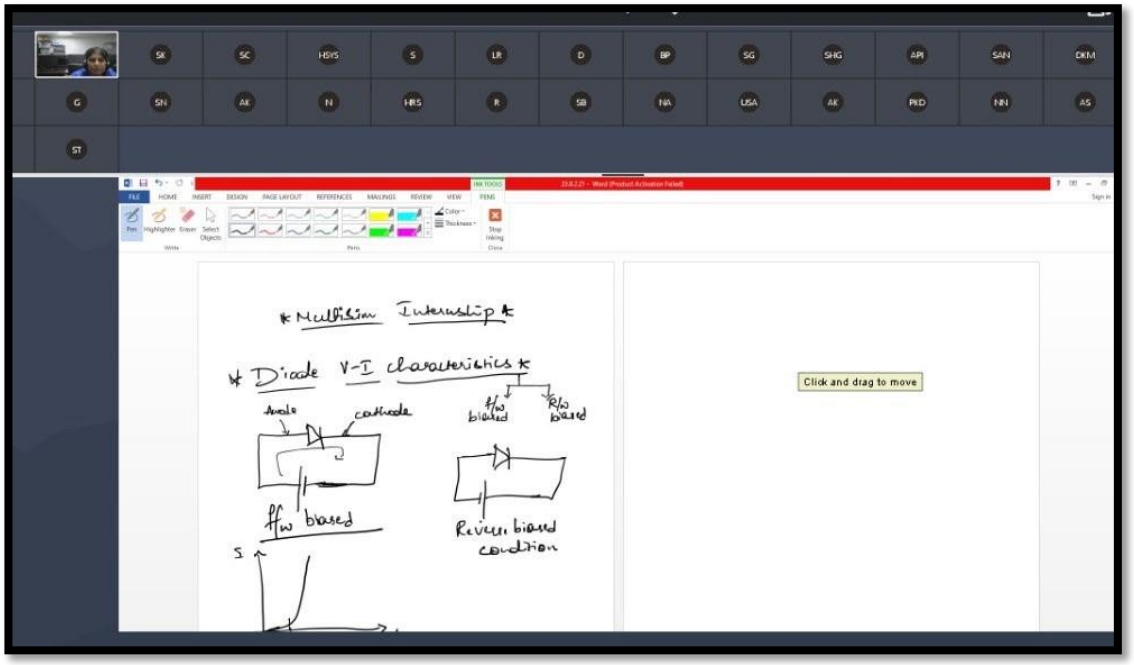
Day -1: The session started at 10:00 am. Faculty handled by Archana K . About 35 students were present for the first day training on multi-sim.

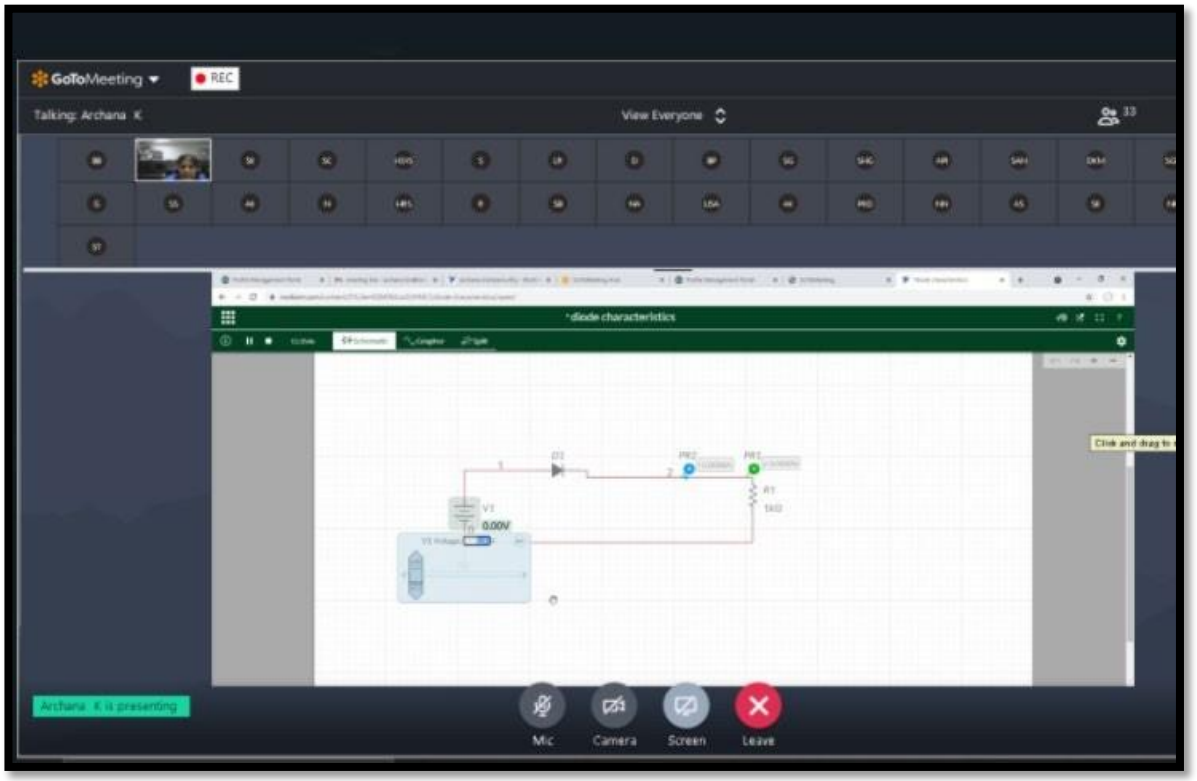
- **Topics Covered:**

  - Introduction about Internship
  - Introduction about Multi-sim platform
  - VI Characteristics of diode
  - Rectifiers: Bridge rectifier, half wave rectifier, Centre tapped rectifier.



- **<u>VI characteristics Diode:</u>** When the forward voltage exceeds the diodes P-N junction internal barrier voltage, which for silicon is about 0.7 volts, avalanche occurs and the forward current increases rapidly for a very small increase in voltage producing a non-linear curve.

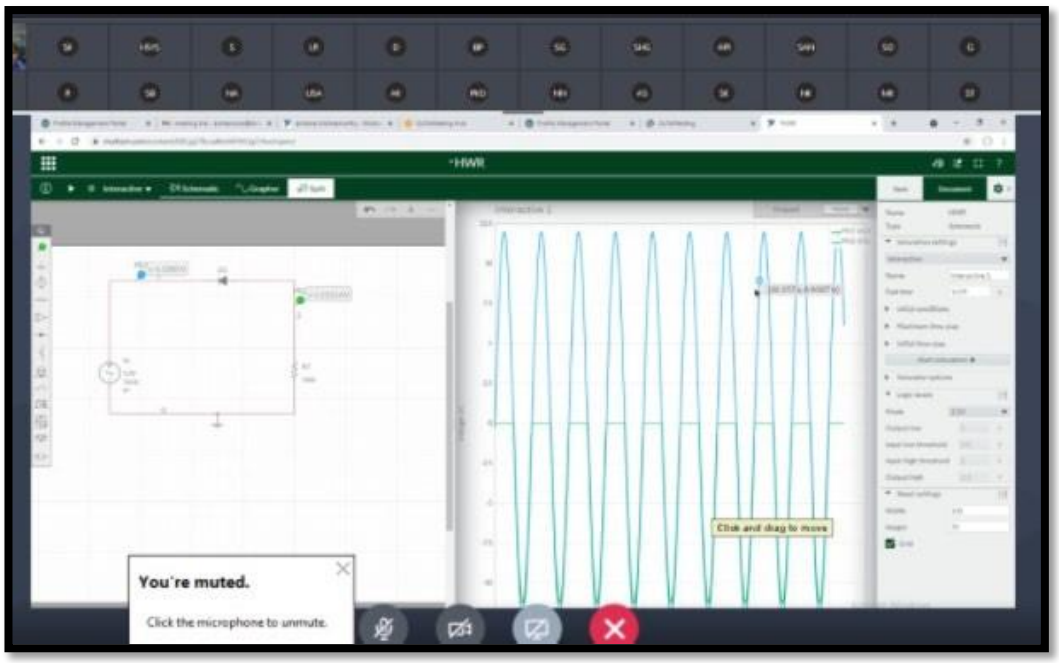




- **Rectifiers:** A rectifier is an electrical device that converts alternating current (AC) into Pulsating DC, which flows in only one direction.

  - **Half wave Rectifier:** In half-wave rectification of a single-phase supply, either the positive or negative half of the AC wave is passed, while the other half is blocked. Because only one half of the input waveform reaches the output, mean voltage is lower.

  - **Full wave Rectifier:** A full-wave rectifier converts the whole of the input waveform to one of constant polarity (positive or negative) at its output. Mathematically, this corresponds to the absolute value function. Full-wave rectification converts both polarities of the input waveform to pulsating DC (direct current), and yields a higher average output voltage. Two diodes and a centre tapped transformer, or four diodes in a bridge configuration and any AC source (including a transformer without centre tap), are needed.

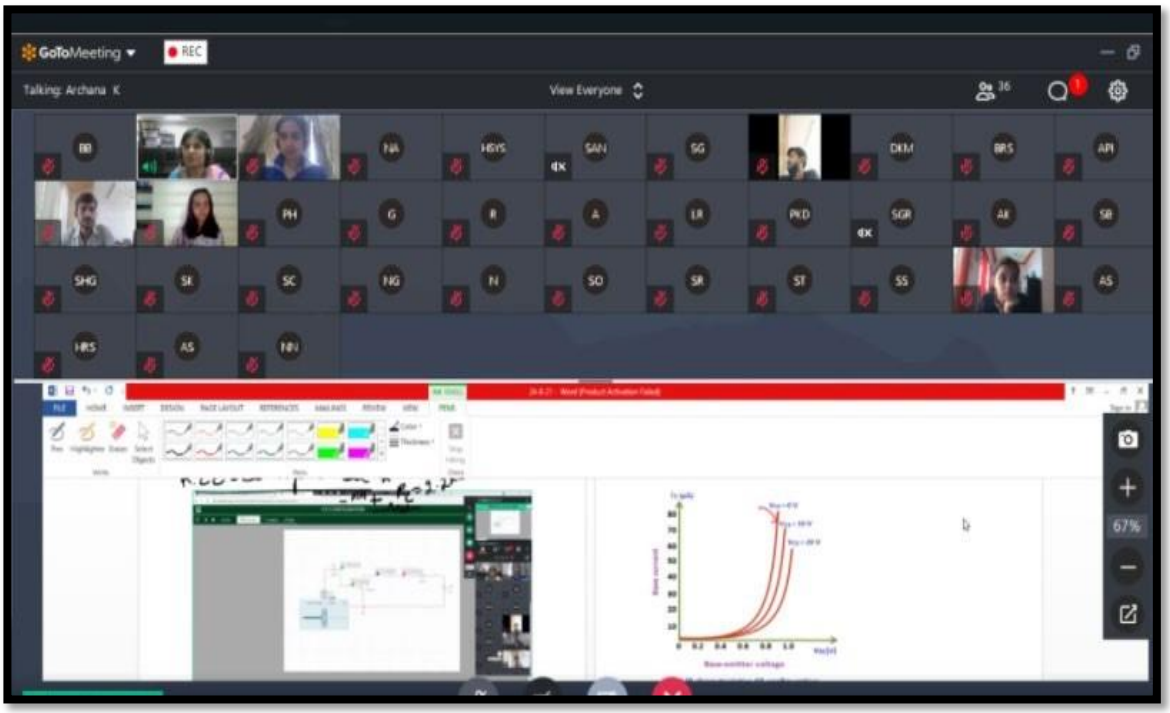


OUTCOME

Students were able to implement the above mentioned content. They could analyze the practical aspects of the circuit using theoretical knowledge on multi-sim platform.

DAY 2: 24-08-2021

The session started at 10:00 am. Faculty handled by Archana K . About 37 students were present for the second day training on multi-sim.

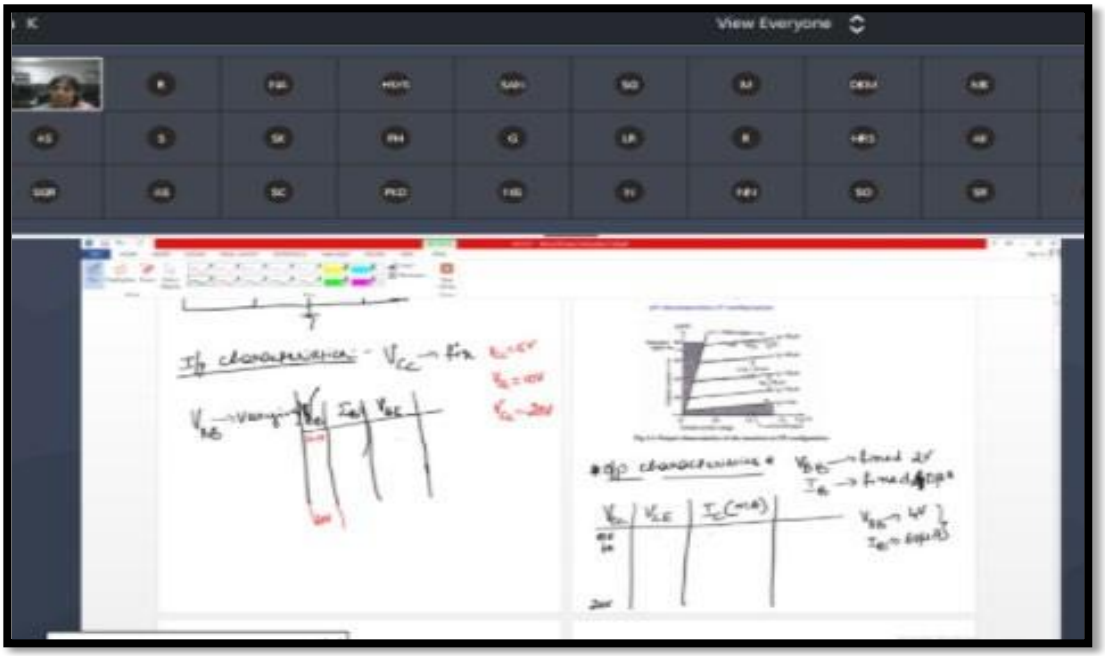


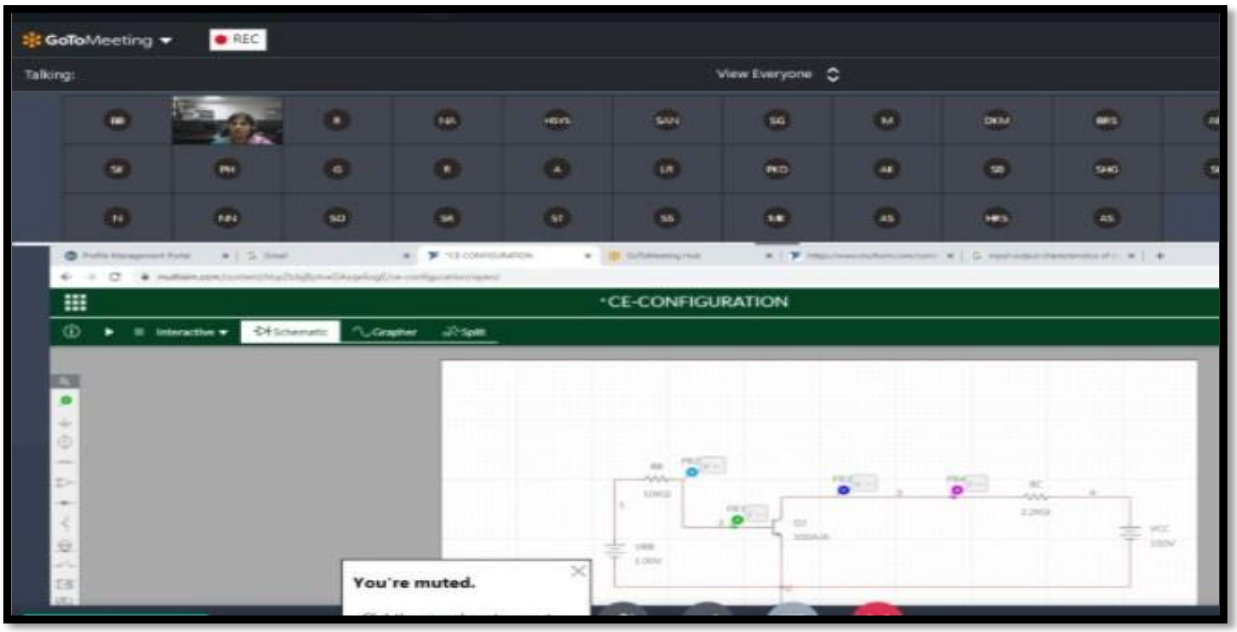
- **Topics Covered:**

  - Common Emitter configuration of BJT transistor.
  - Introduction about digital circuits.
  - AND,OR,NOT NAND and NOR gates.
  - Design and implementation of Half adder.

- **Common Emitter configuration of BJT transistor:**

  The **common emitter** (CE) configuration is the most widely used transistor configuration. The common emitter (CE) amplifiers are used when large current gain is needed. The input signal is applied between the base and emitter terminals while the output signal is taken between the collector and emitter terminals.


View Everyone


GoToMeeting
REC
Talking:
View Everyone
*CE-CONFIGURATION
Interactive
Schematic
Grapher
Split
You're muted.

DIGITAL GATES:

The basic digital electronic circuit that has one or more inputs and single output is known as **Logic gate**. Hence, the Logic gates are the building blocks of any digital system.

- **AND:** An AND gate is a digital circuit that has two or more inputs and produces an output, which is the **logical AND** of all those inputs. It is optional to represent

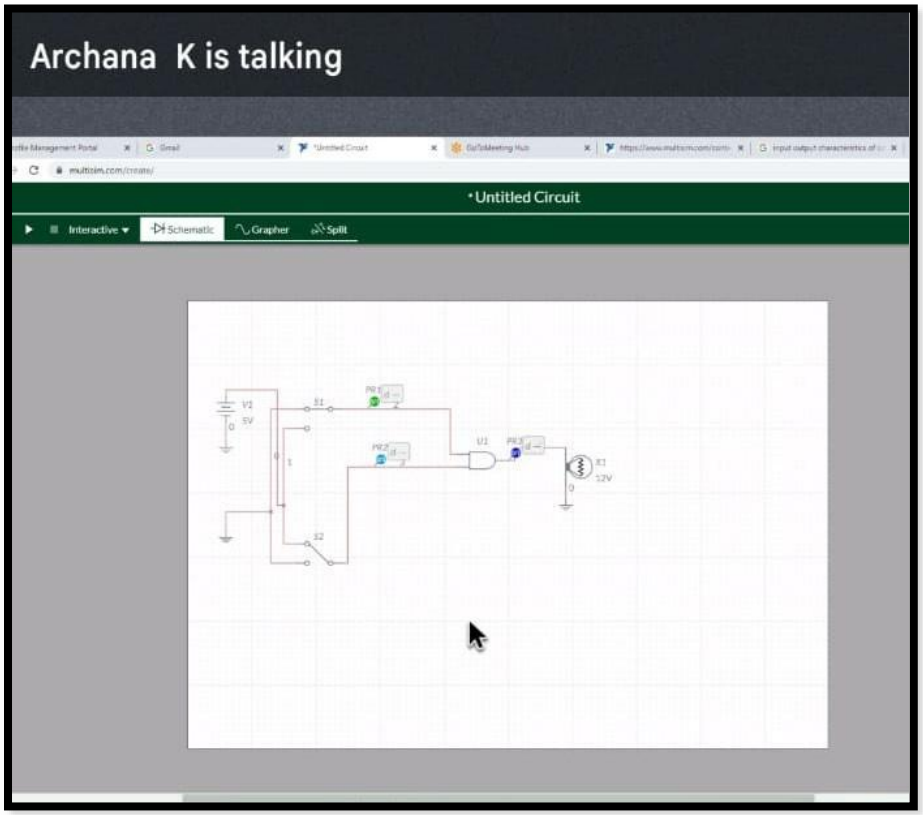
  the **Logical AND** with the symbol.

- **OR:** An OR gate is a digital circuit that has two or more inputs and produces an output, which is the logical OR of all those inputs. This **logical OR** is represented with the symbol '+'.

- **NOT:** A NOT gate is a digital circuit that has single input and single output. The output of NOT gate is the **logical inversion** of input. Hence, the NOT gate is also called as inverter.

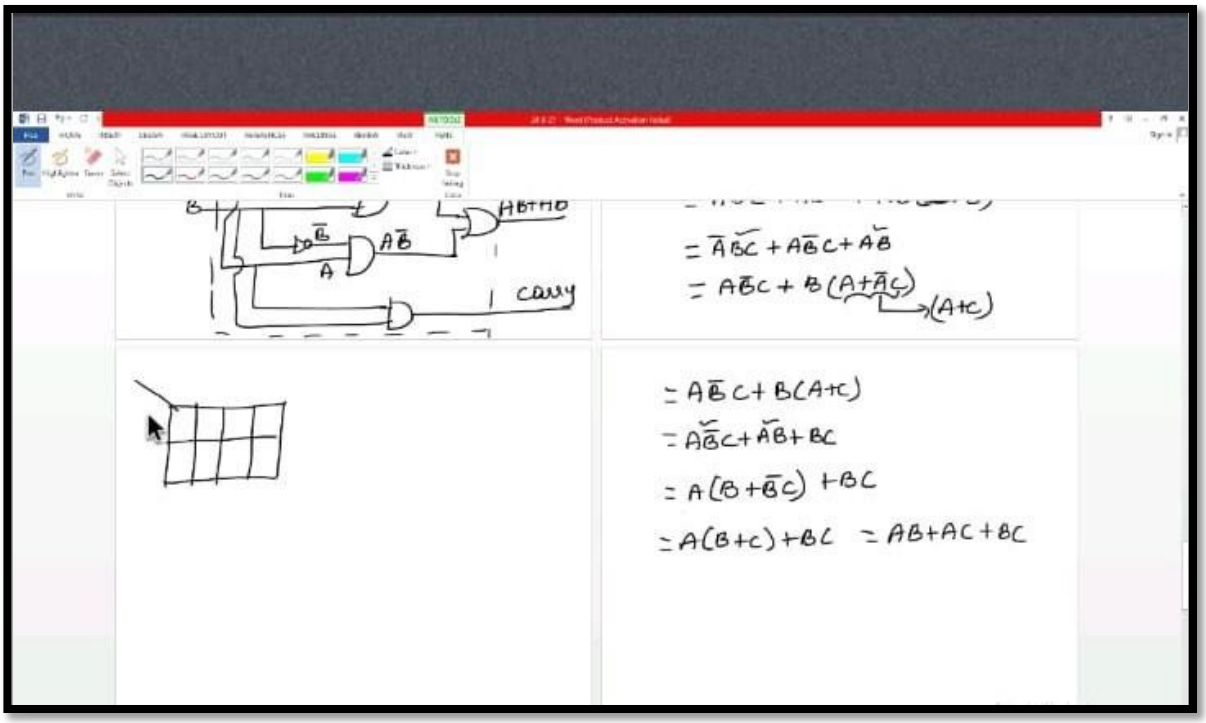
- **NAND:** The NAND (Not – AND) gate has an output that is normally at logic level "1" and only goes "LOW" to logic level "0" when ALL of its inputs are at logic level "1". The Logic NAND Gate is the reverse or "Complementary" form of the AND gate

- **NOR:** A NOR gate is a logic gate which gives a positive output only when both inputs are negative. Like NAND gates, NOR gates are so-called "universal gates" that can be combined to form any other kind of logic gate.



HALF ADDER:

Half Adder is a combinational logic circuit which is designed by connecting one EX-OR gate and one AND gate. The half adder circuit has two inputs: A and B, which add two input digits and generates a carry and a sum.A half adder is used for adding together the two least significant digits in a binary sum.



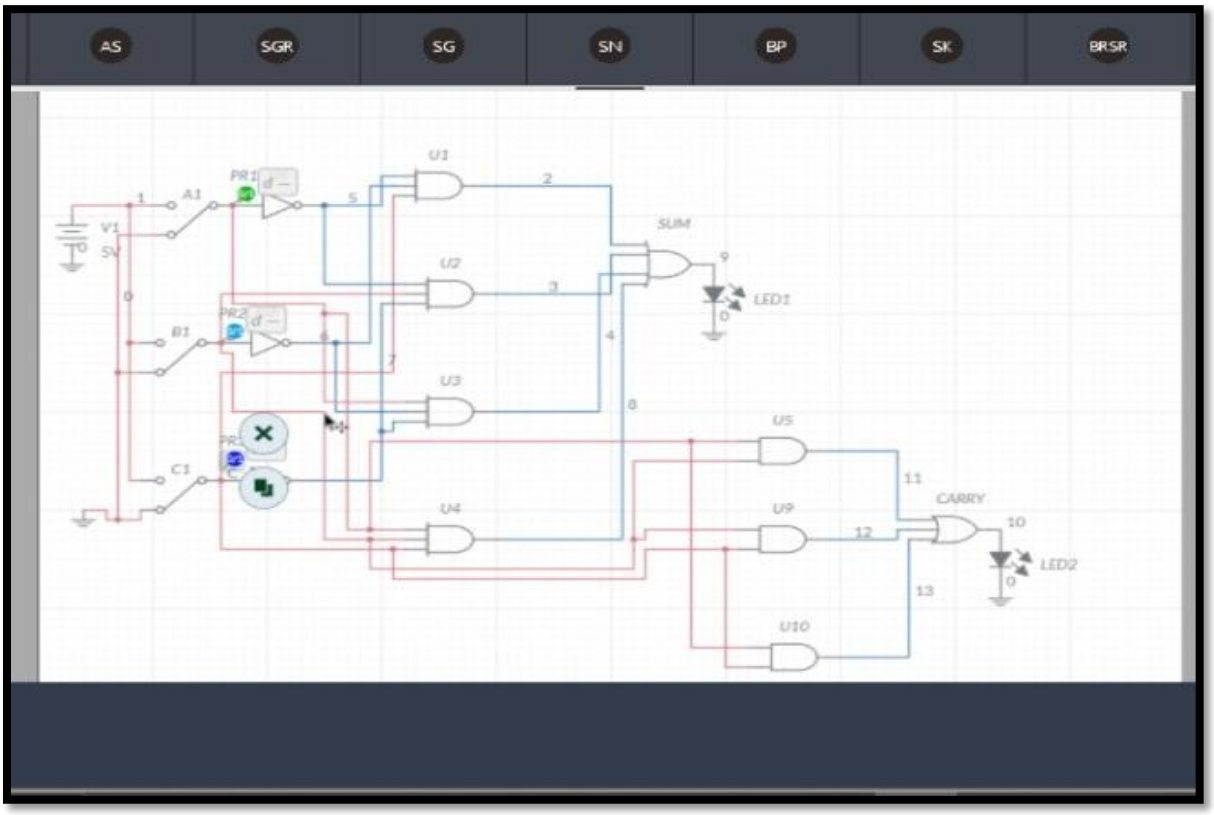
OUTCOME:

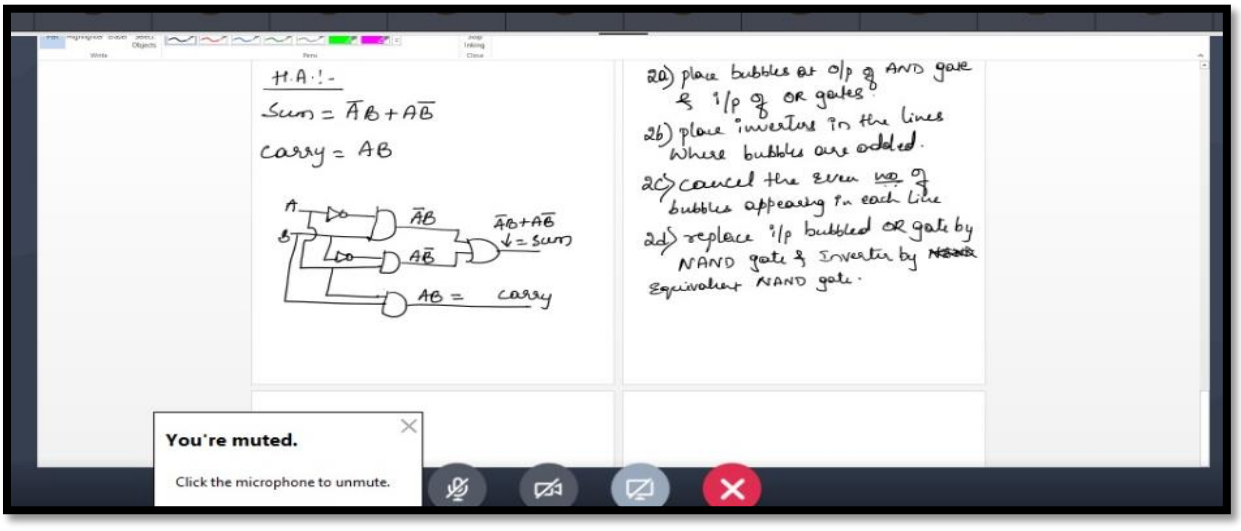
Students were able to implement the above mentioned content. They could analyze the practical aspects of the circuit using theoretical knowledge on multi-sim platform. Also they were able to learn, analyze and design digital circuits.

DAY-3: 25/08/2021 – REPORT

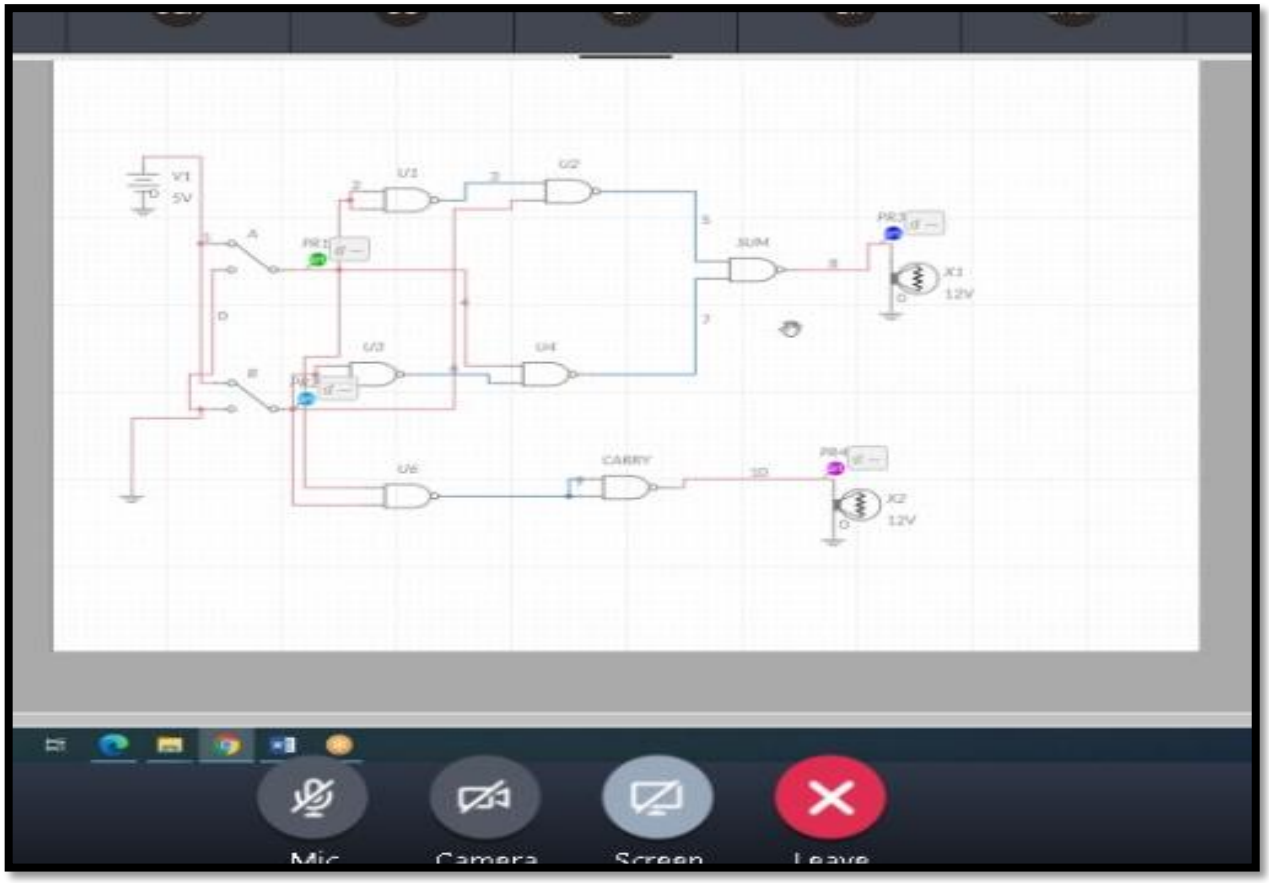
Day -3: The session started at 10:00 am. Faculty handled by Archana K . About 37 students were present for the third day training on multi-sim.

- **Topics Covered:**
  - Full adder using basic gates
  - Half adder and Full adder using NOR gates.
  - Half adder and Full adder using NAND gates

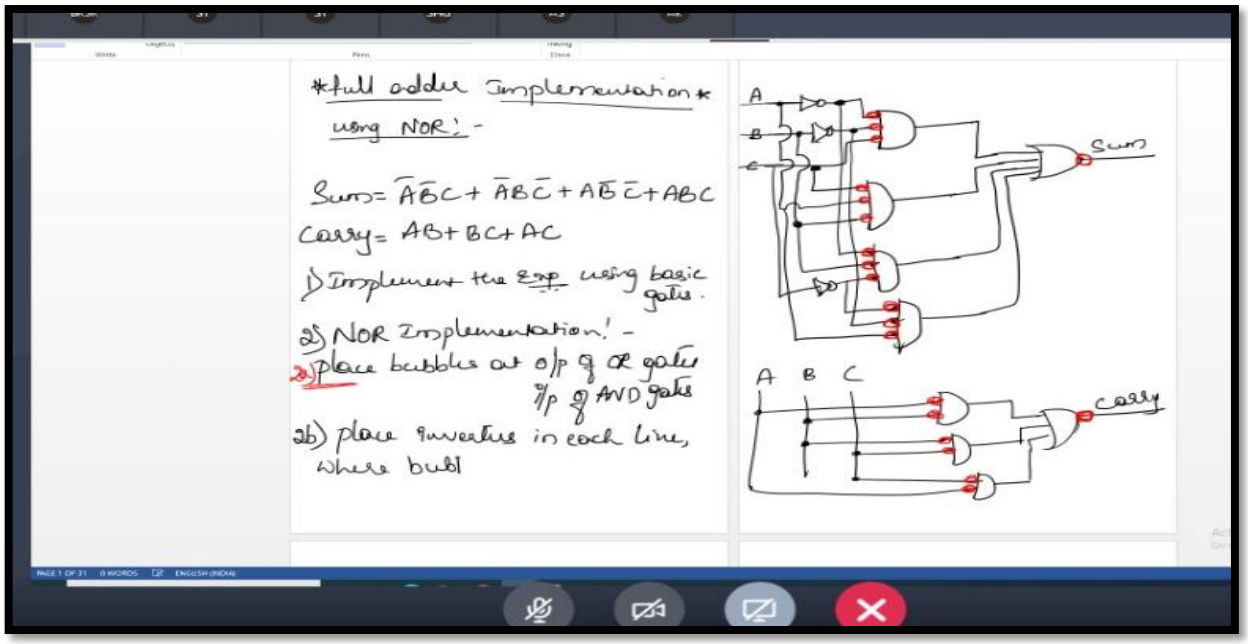




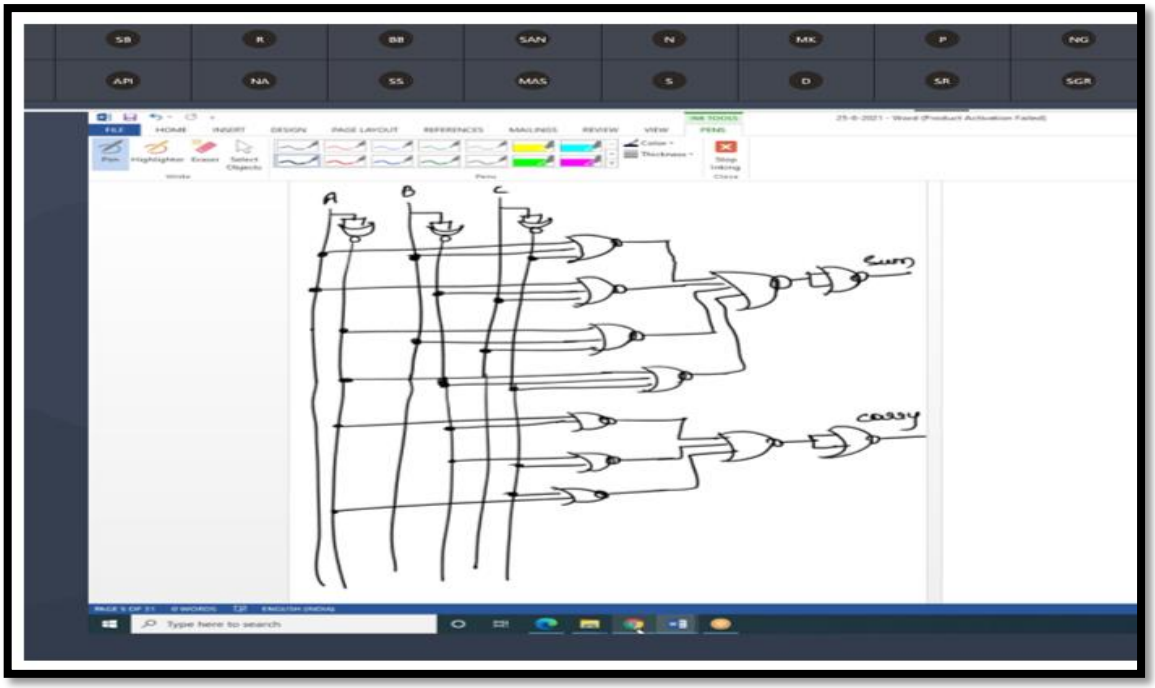
→ **Full Adder:** Full Adder is the adder which adds three inputs and produces two outputs. The first two inputs are A and B and the third input is an input carry as C-IN. The output carry is designated as C-OUT and the normal output is designated as S which is SUM. A full adder logic is designed in such a manner that can take eight inputs together to create a byte-wide adder and cascade the carry bit from one adder to another.

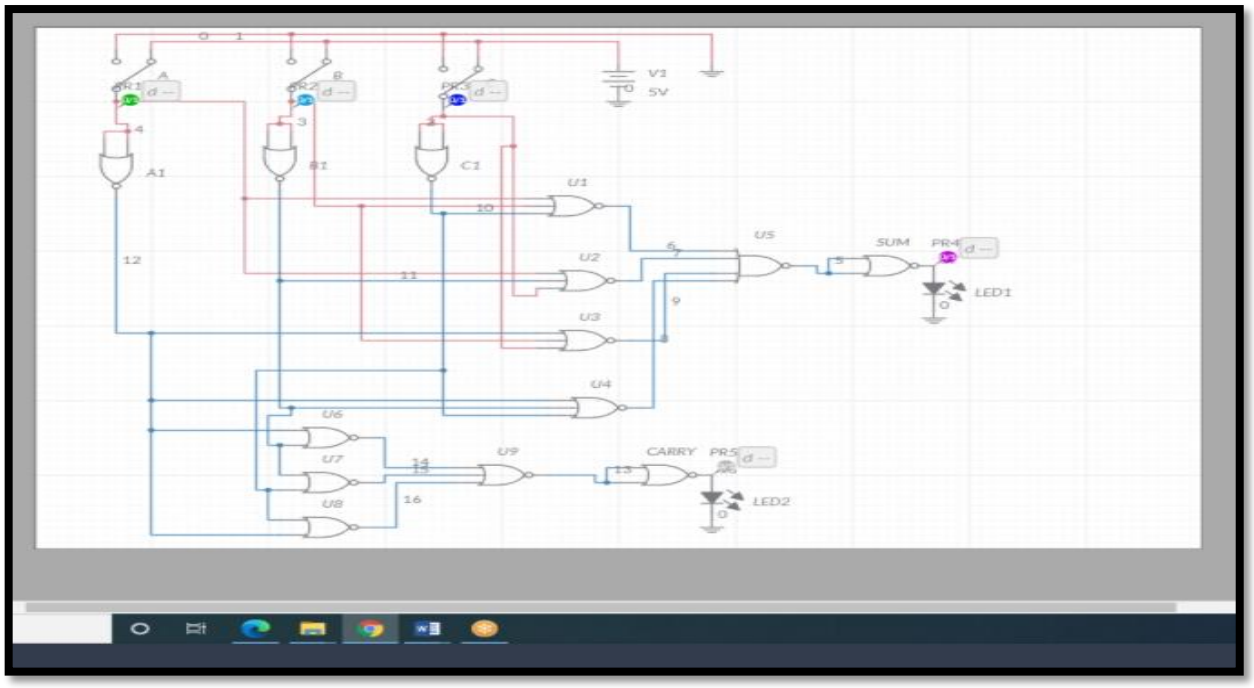


→ **Full Adder using NAND Gates:** a NAND gate is one of the universal gates and can be used to implement any logic design. The circuit of full adder using only NAND gates .Full adder is a simple 1–bit adder. If we want to perform n–bit addition, then we need ‘n’ number of 1–bit full adders should be used in the form of a cascade connection.



→ **<u>Full adder using NOR gates:</u>** a NOR gate is one of the universal gates and can be used to implement any logic design. The circuit of full adder using only NOR gates.

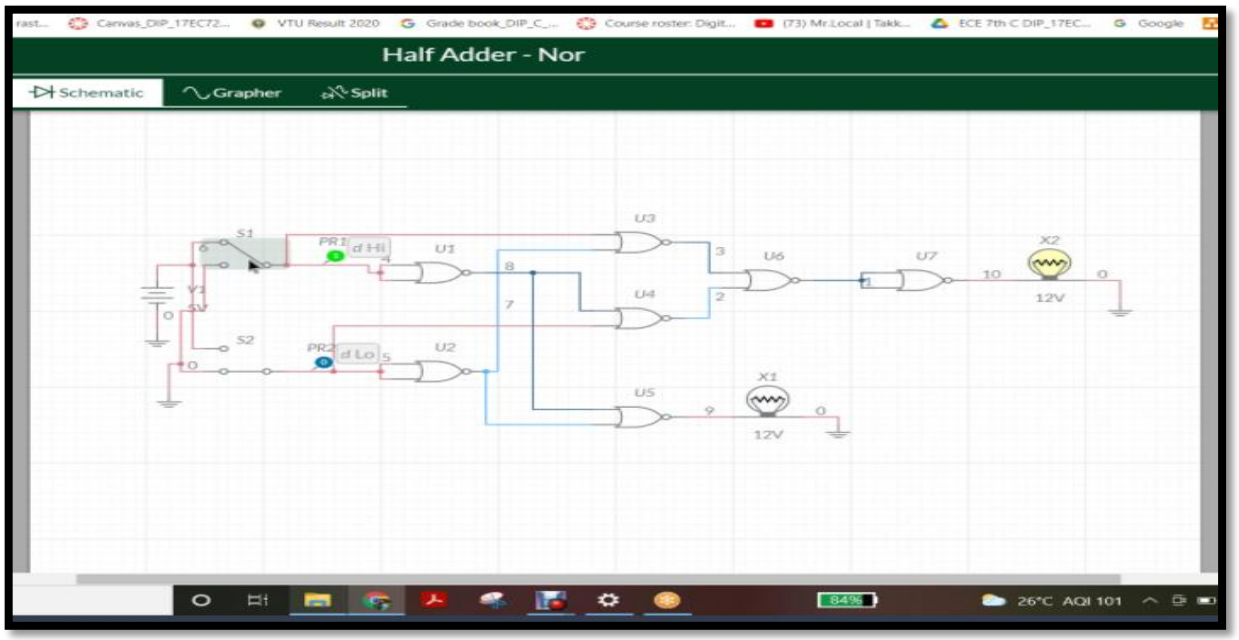




As we know that NAND and NOR are called universal gates as any logic system can be implemented using these two, the half adder circuit can also be implemented using them.

Five NAND gates are required in order to design a half adder.

→ **Half Adder using NOR gates:** Five NOR gates are required in order to design a half adder.



OUTCOME:

Students were able to implement the above mentioned content. They could analyze the practical

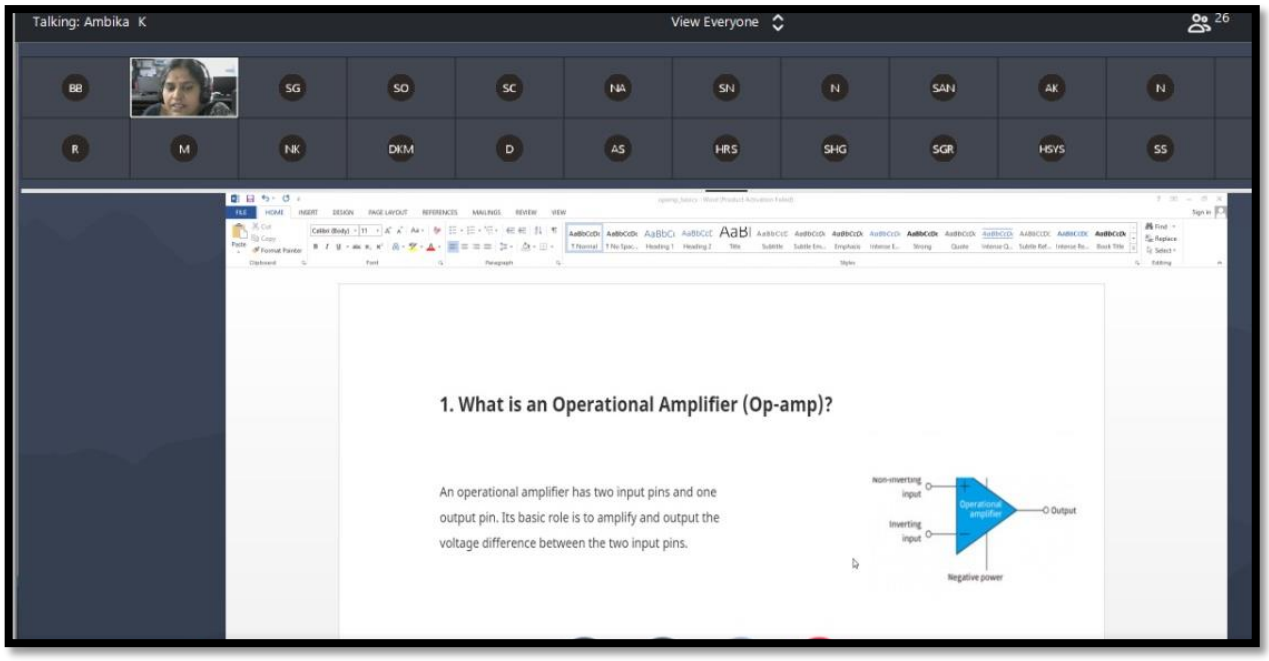
aspects of the circuit using theoretical knowledge on multi-sim platform. They learnt to implement a Full adder & Half adder circuits, its Boolean Equations and also design of Full adder & Half adder using NAND and NOR gates.

DAY-4: 26/08/2021 – REPORT

Day -4: The session started at 10:00 am. Faculty handled by Ambika K. About 37 students were present for the fourth day training on multi-sim.

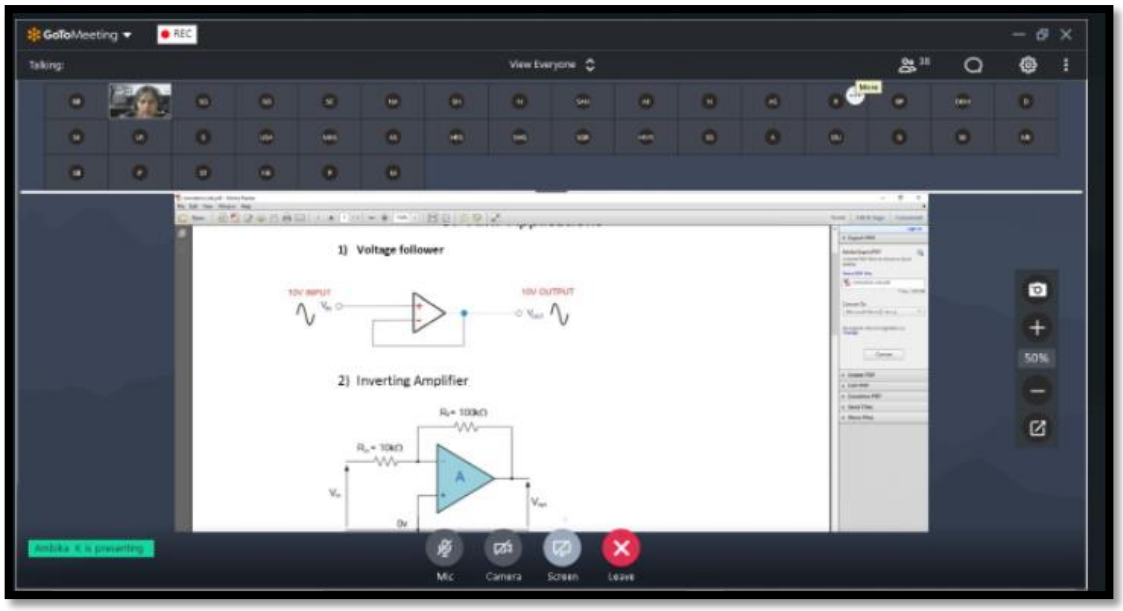
- **Topics Covered:**

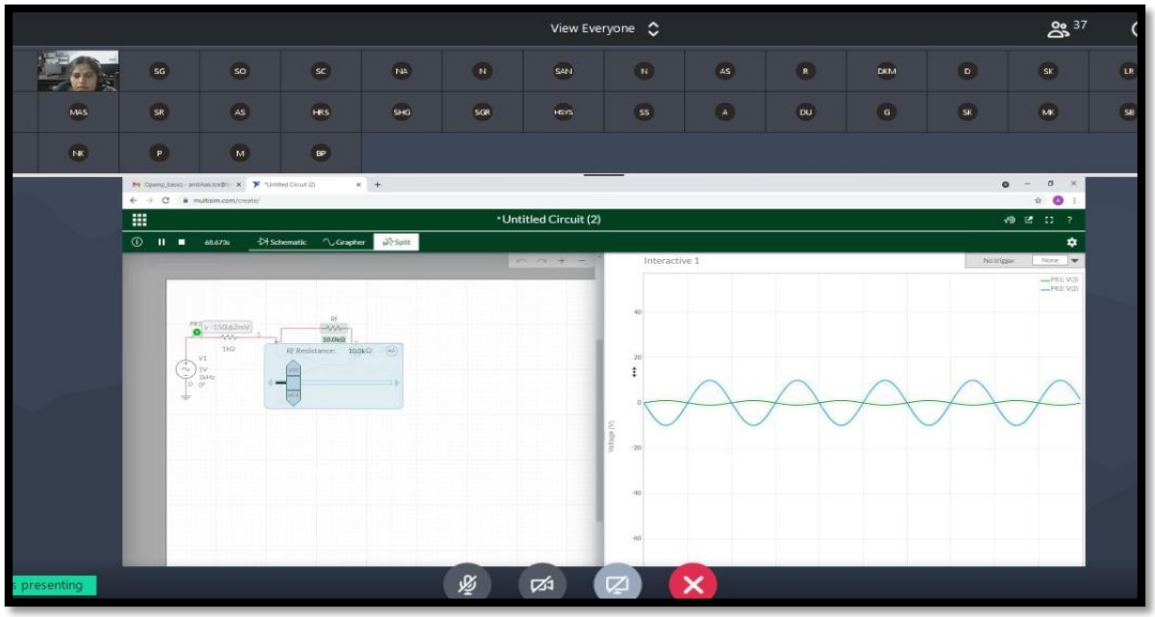
 Introduction to operational amplifiers (op-amp).
Op-amp as voltage follower.
 Inverting and non-inverting amplifiers.
→ Inverting and non-inverting summing amplifers.
→ Differentiator and integrator.



→ **Op-amp as Voltage follower:**

Voltage Follower is simply **a circuit in which output follows the input**, means output voltage remains same as input voltage. It is also commonly known as Unity gain Opamp Amplifier or Opamp Buffer.



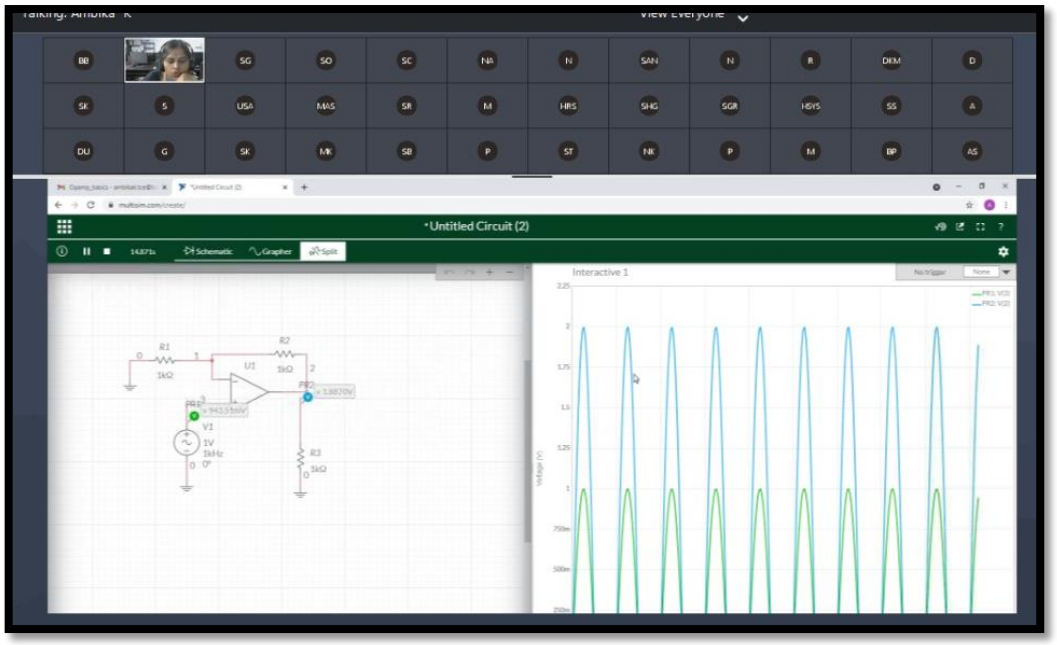


→ **Op-amp as Inverting Amplifier:**

An inverting amplifier (also known as an inverting operational amplifier or an inverting op-amp) is **a type of operational amplifier circuit which produces an output which is out of phase with respect to its input by 180º**

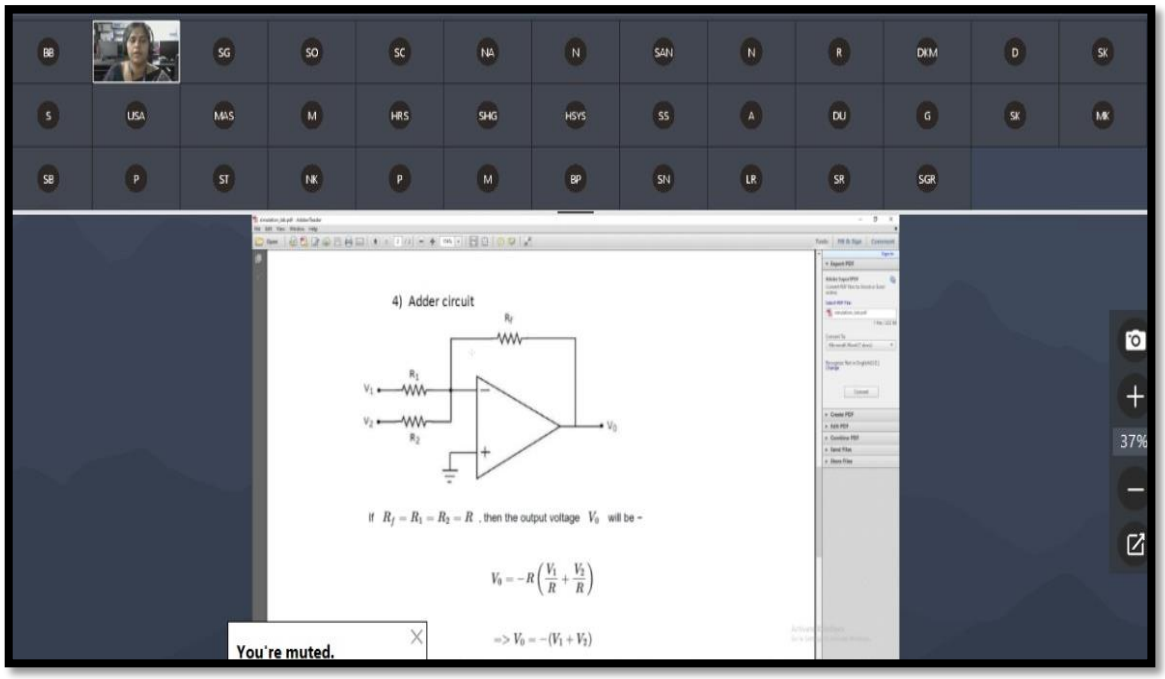
→ **Op-amp as non - Inverting Amplifier:**

A non-inverting amplifier is **an op-amp circuit configuration that produces an amplified output signal** and this output signal of the non-inverting op-amp is in-phase with the applied input signal. In other words, a non-inverting amplifier behaves like a voltage follower circuit.



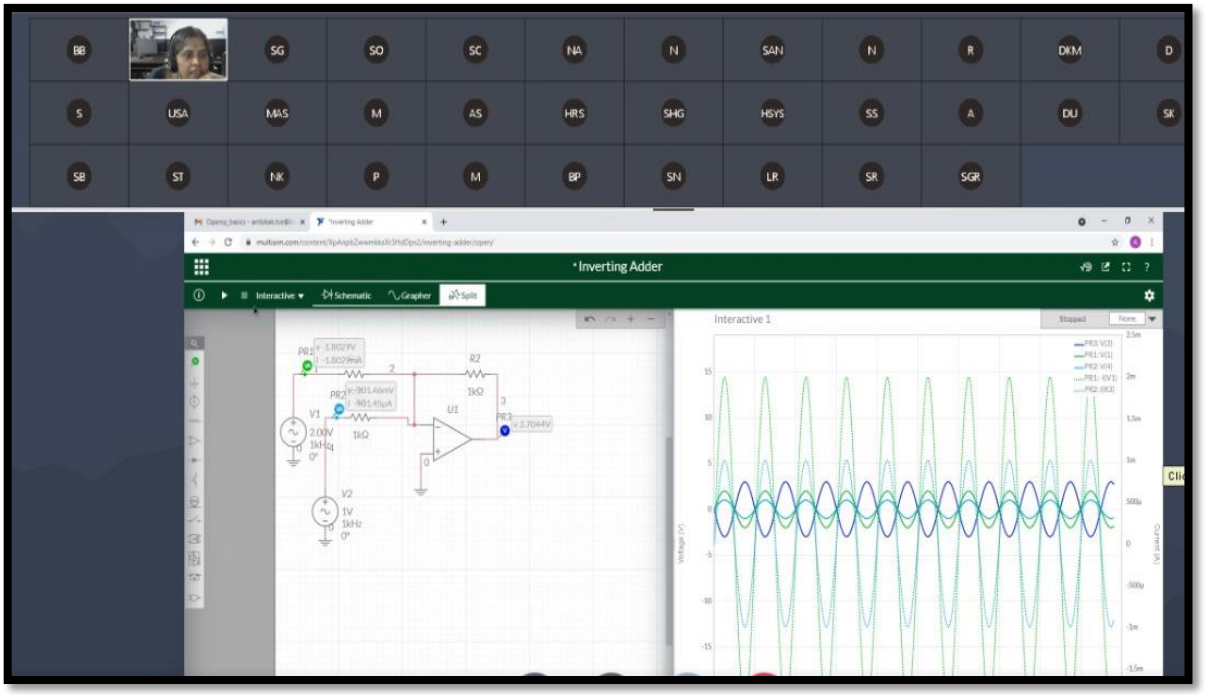
→ **Op-amp as Inverting summing Amplifier:**

The Summing Amplifier. The Summing Amplifier is another type of operational amplifier circuit configuration that is used to **combine the voltages present on two or more inputs** into a single output voltage.



→ **Op-amp as non - Inverting summing Amplifier:**

A Non-Inverting Summing Amplifier can also be constructed using the Non-Inverting Amplifier configuration of the Op Amp. Here, the input voltages are applied to the non-inverting input terminal of the Op Amp and a part of the output is fed back to the inverting input terminal, through voltage-divider-bias feedback.

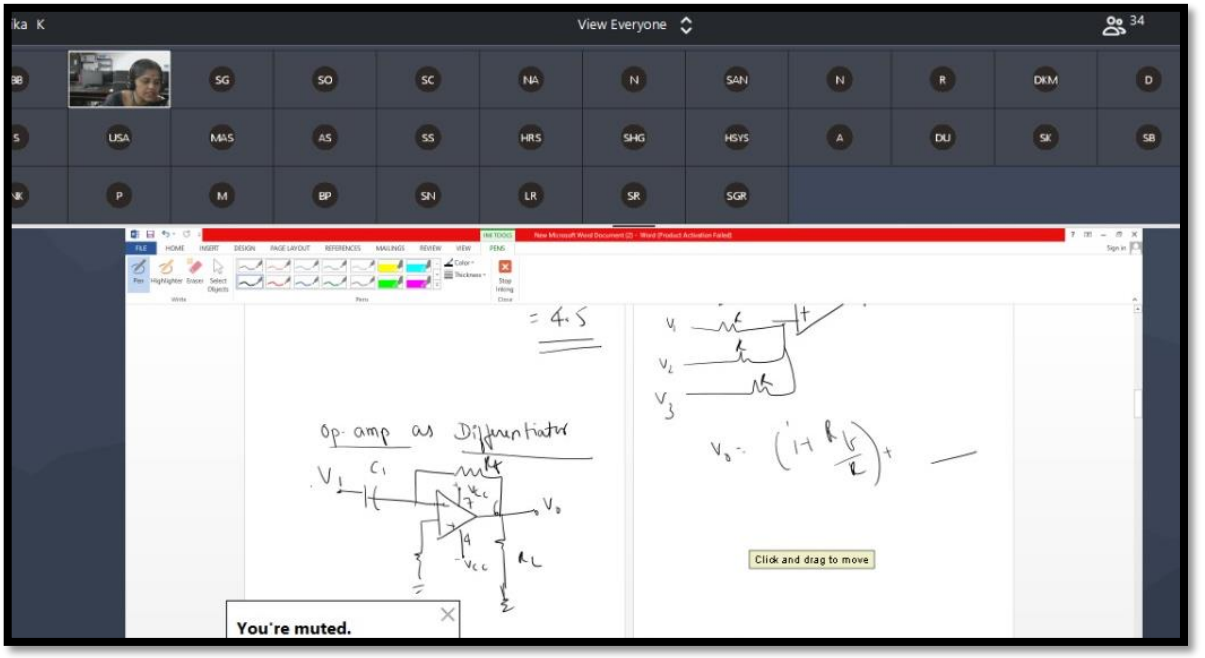


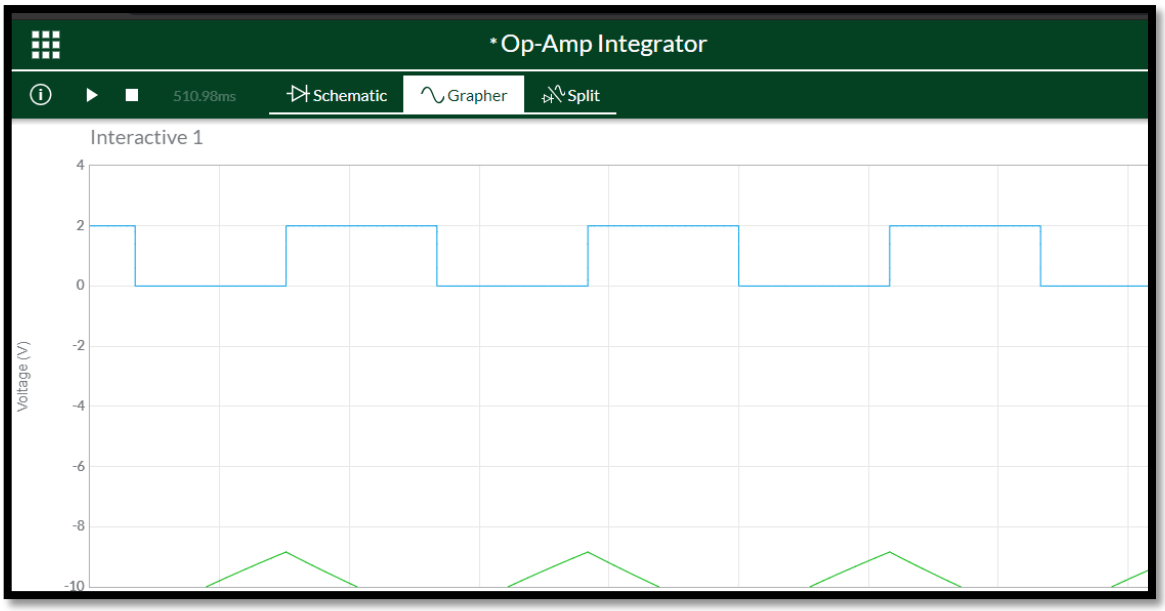
→ **Op-amp as differentiator and integrator:**

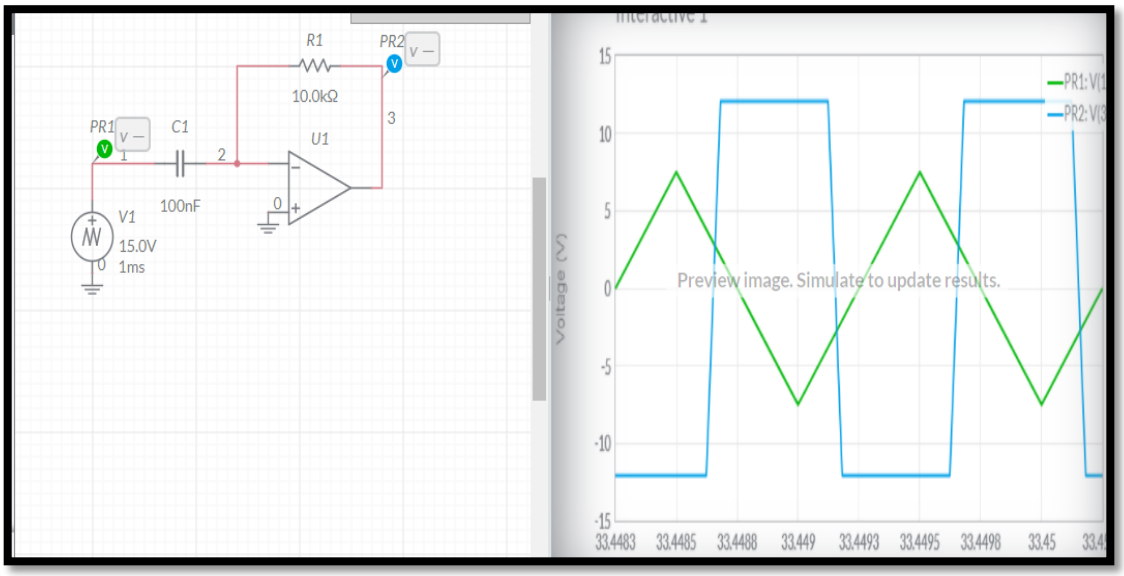
An op-amp differentiator or a differentiator amplifier is **a circuit configuration which is inverse of the integrator circuit**. It produces an output signal where the instantaneous amplitude is proportional to the rate of change of the applied input voltage.

→ **Op-amp as differentiator and integrator:**

An operational amplifier (op-amp) integrator is **an operational amplifier circuit that performs the mathematical operation of integration with respect to time**—this means the output voltage is proportional to the input voltage integrated over time.







OUTCOME:

Students were able to learn operational amplifiers and their applications. They implemented and analyzed the mathematical functions such as summing, differentiation and integration using opamps.

DAY-5: 27/08/2021 – REPORT

Day -5: The session started at 10:00 am. Faculty handled by Ambika K . About 37 students were present for the fifth day training on multi-sim.

- **Topics Covered:**

  - Precision Half wave and Full wave rectifier.
  - First order High & Low pass filter.

 Zero crossing detector & Schmitt trigger.

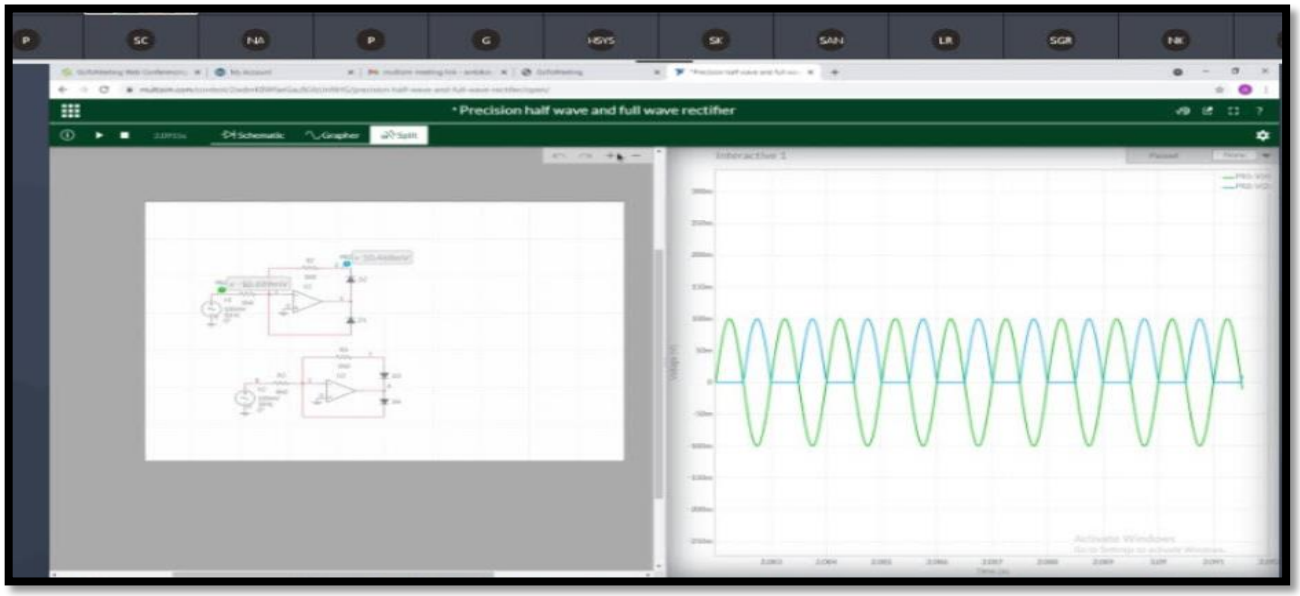
→ Wein bridge oscillator

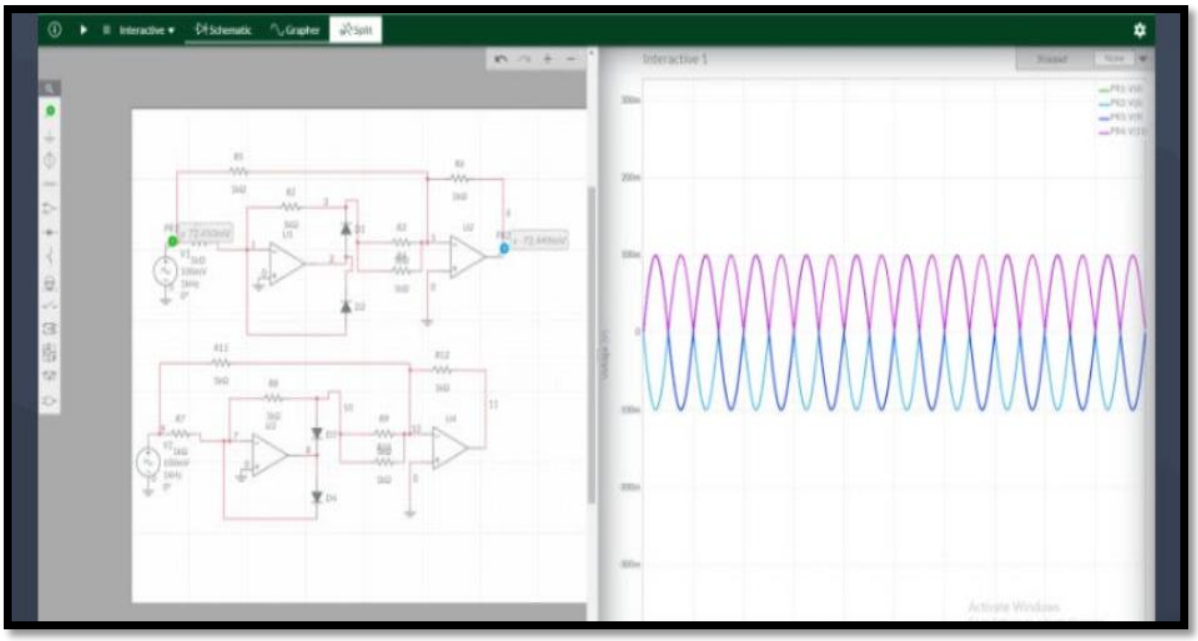
→ Multisim installation

- **Precision Half wave and Full wave rectifier:**

→ Precision half-wave rectifiers are commonly used with other op amp circuits such as a peak-detector or bandwidth limited non-inverting amplifier to produce a DC output voltage. This configuration has been designed to work for sinusoidal input signals between 0.2mVpp and 4Vpp at frequencies up to 50 kHz.

→ The Precision Full Wave Rectifier circuits accept an Ac signal at the input, inverts either the negative or the positive half, and delivers both the inverted and non-inverted halves at the output.





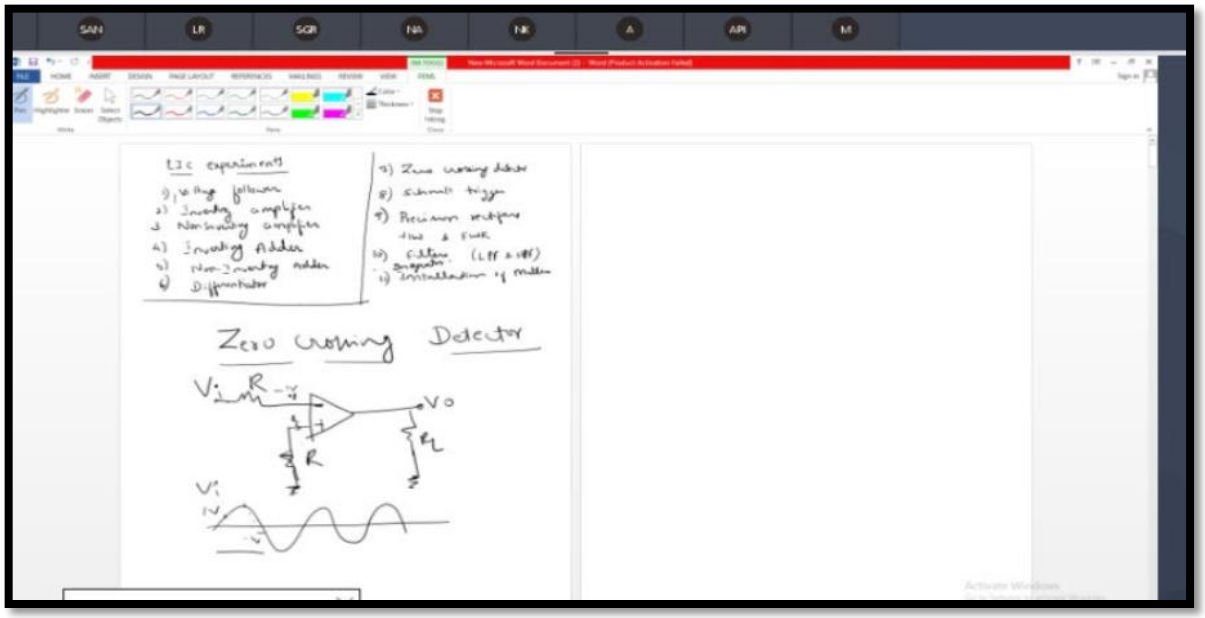
- **First order High pass and Low pass filter:**

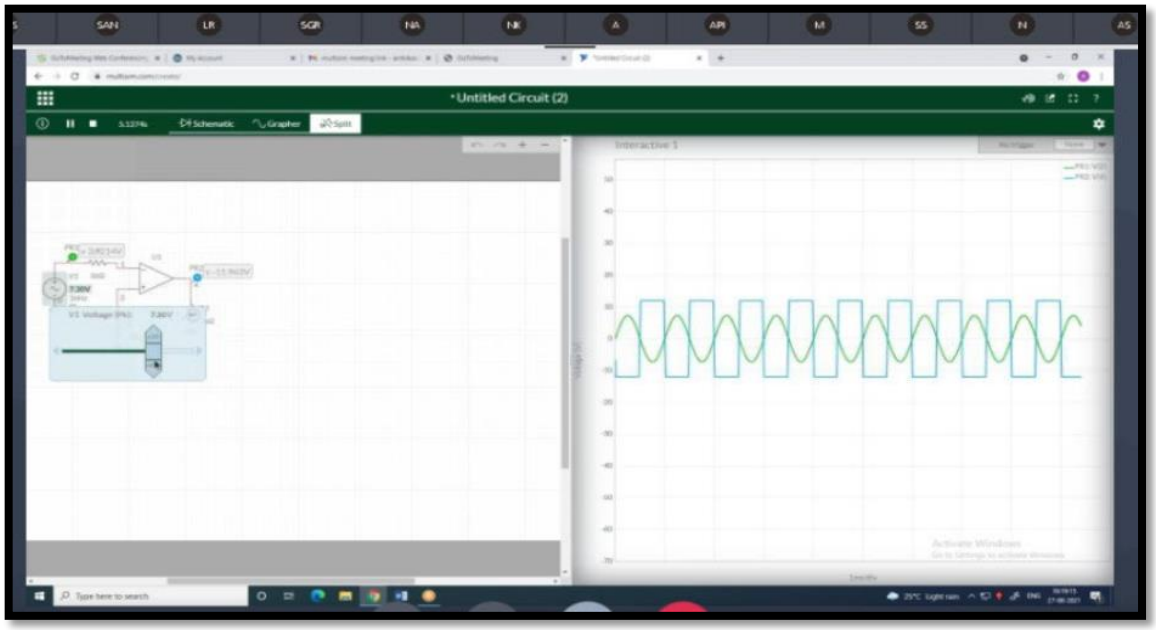
→ First-order high pass filter, consists simply of a passive filter followed by a non-inverting amplifier. The frequency response of the circuit is the same as that of the

passive filter, except that the amplitude of the signal is increased by the gain of the amplifier.

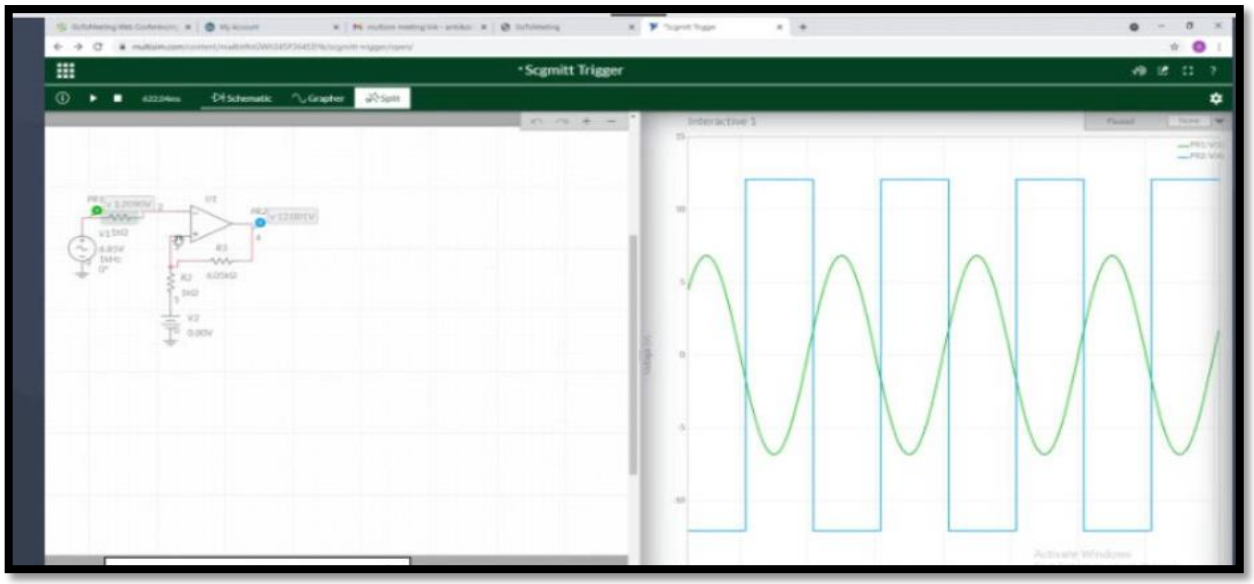
→ First-order low pass active filter, consists simply of a passive RC filter stage providing a low frequency path to the input of a non-inverting operational amplifier. ... If a voltage gain greater than one is required we can use the following filter circuit.

- **<u>Zero crossing detector:</u>** A zero-crossing detector or ZCD is one type of voltage comparator, used to detect a sine waveform transition from positive and negative that coincides when the I/P crosses the zero voltage condition. The applications of the Zero Crossing Detector are phase meter and time marker generator.





- **<u>Schmitt Trigger:</u>** Schmitt trigger is a comparator circuit with hysteresis implemented by applying positive feedback to the noninverting input of a comparator or differential amplifier. It is an active circuit which converts an analog input signal to a digital output signal. The circuit is named a "trigger" because the output retains its value until the input changes sufficiently to trigger a change.



- **Wein Bridge Oscillator:** A Wien bridge oscillator is a type of electronic oscillator that generates sine waves. It can generate a large range of frequencies. The oscillator is based on a bridge circuit originally developed by Max Wien in 1891 for the measurement of impedances. The bridge comprises four resistors and two capacitors. The oscillator can also be viewed as a positive gain amplifier combined with a bandpass filter that provides positive feedback.

OUTCOME:

Students were able to implement the above mentioned content. They could analyze the practical aspects of the circuit using theoretical knowledge on multi-sim platform. They were able to learn operational amplifiers and their applications. They learnt some application such as wein bridge oscillator, precision half wave & full wave, first order high pass & low pass filter.

DAY-6: 30/08/2021 – REPORT

Day -6: The session started at 10:00 am. Faculty handled by P.S Gowra . About 30 students were present for the sixth day training on multi-sim.

- **Topics Covered:**

 Clipping circuits (single/double ended).
 Transistor as a Switch
→ Briefing about mini project.

- **CLIPPING CIRCUITS:**

A clipper is a device that removes either the positive half (top half) or negative half (bottom half), or both positive and negative halves of the input AC signal. In other words, a clipper is a device that limits the positive amplitude or negative amplitude or both positive and negative amplitudes

of the input AC signal. In some cases, a clipper removes a small portion of the positive half cycle or negative half cycle or both positive and negative half cycles.

The clipping of a waveform is the most common technique that applies to the input signals to adapt them so that they may lie within the operating range of the electronic circuits. The clipping of waveforms can be done by eliminating the portions of the waveform which crosses the input range of the circuit.

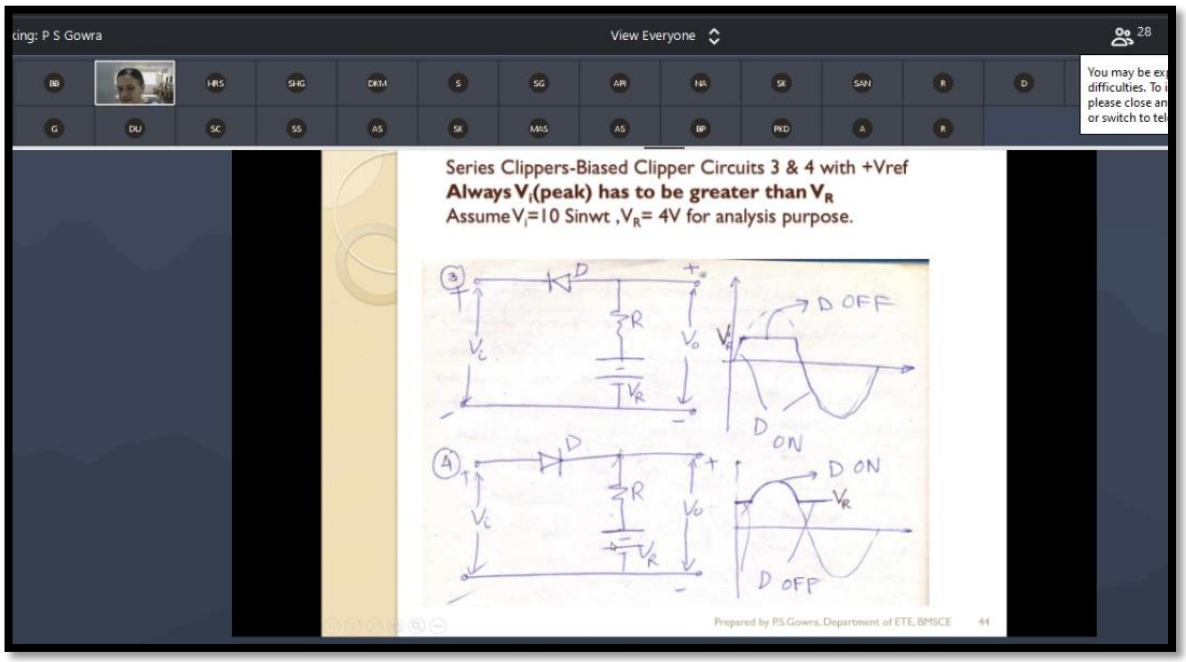
Clippers can be broadly classified into two basic types of circuits. They are:

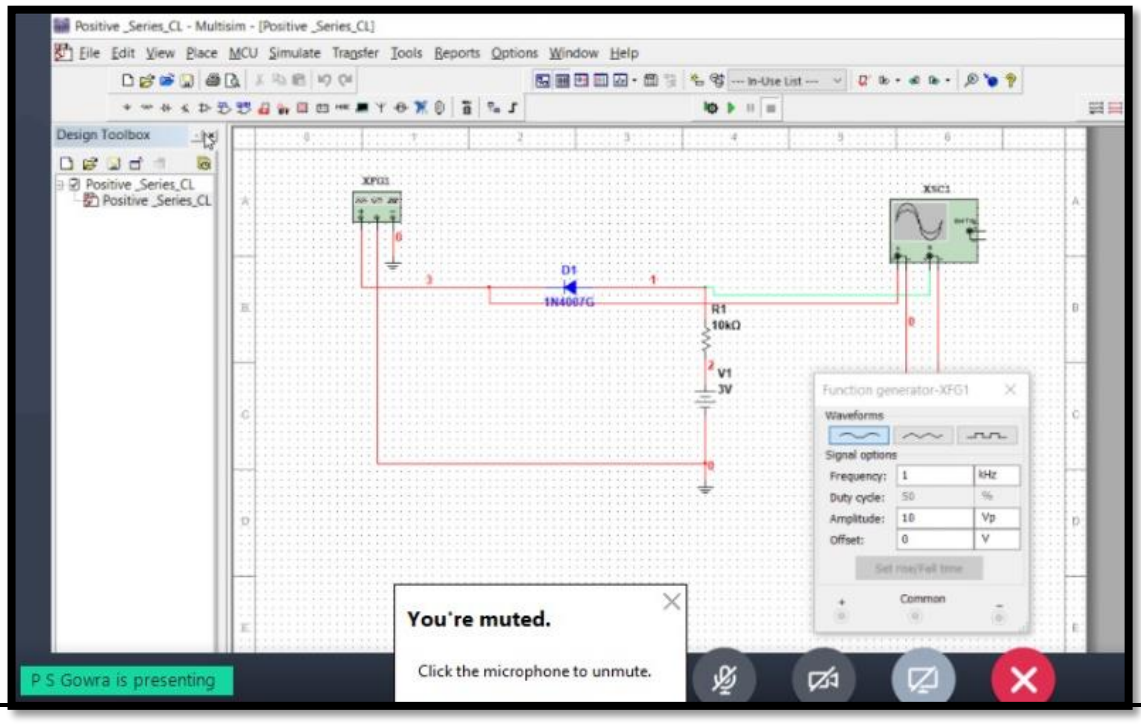
→ Series Clippers

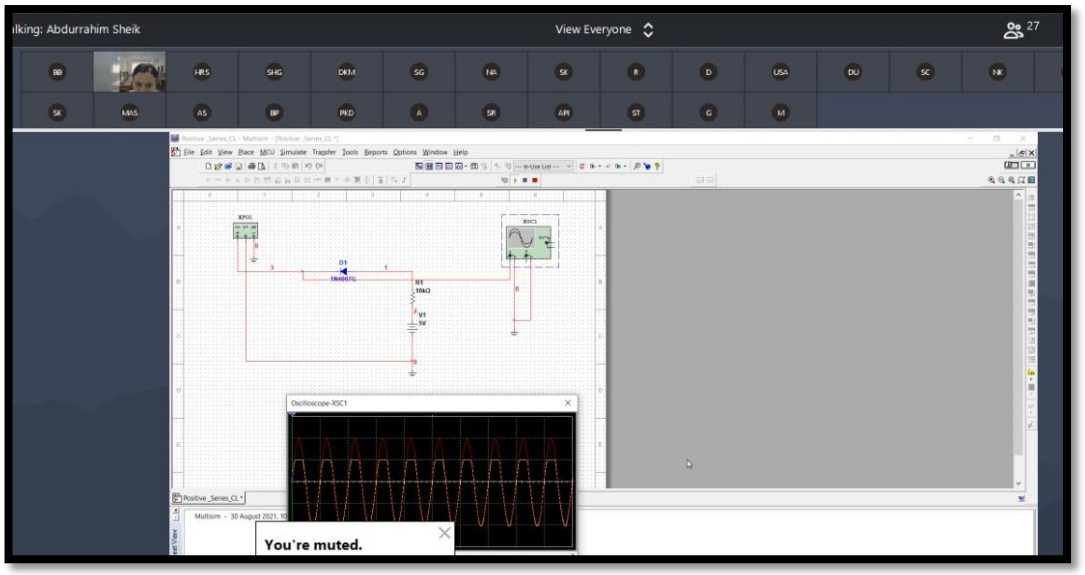
→ Shunt or Parallel Clippers

Series clipper circuit contains a power diode in series with the load connected at the end of the circuit.

The shunt clipper contains a diode in parallel with the resistive load.

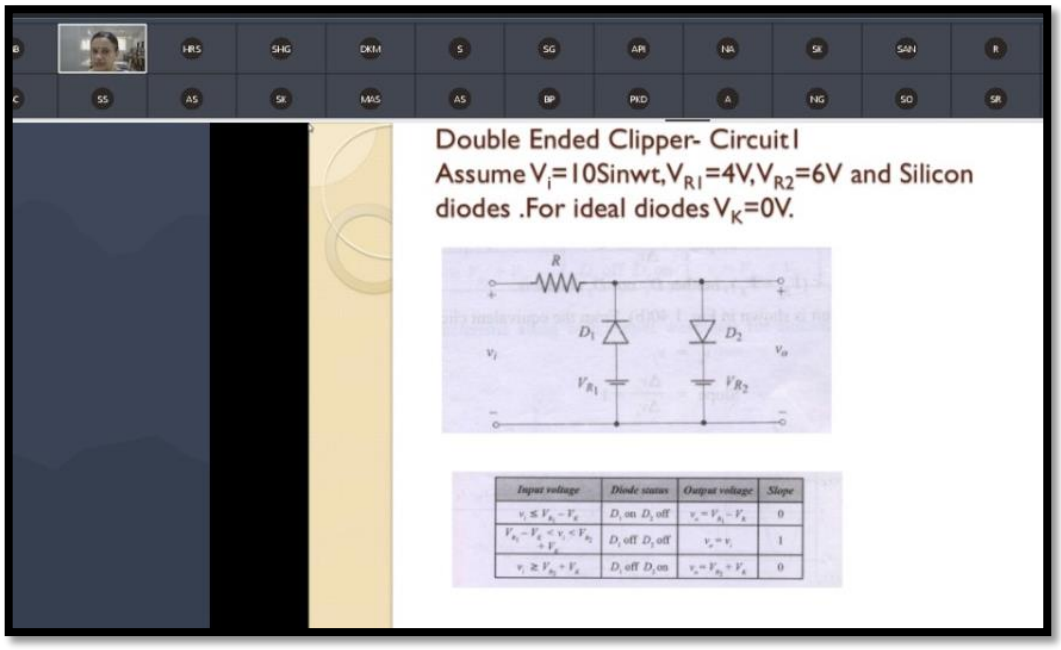


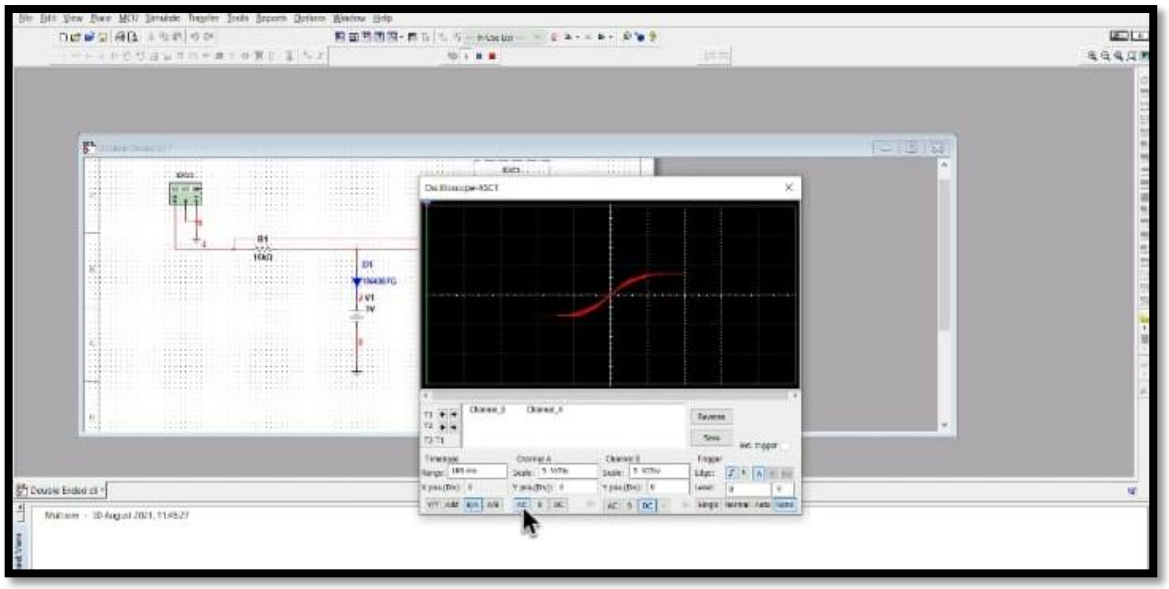




- **DOUBLE ENDED CLIPPERS:**

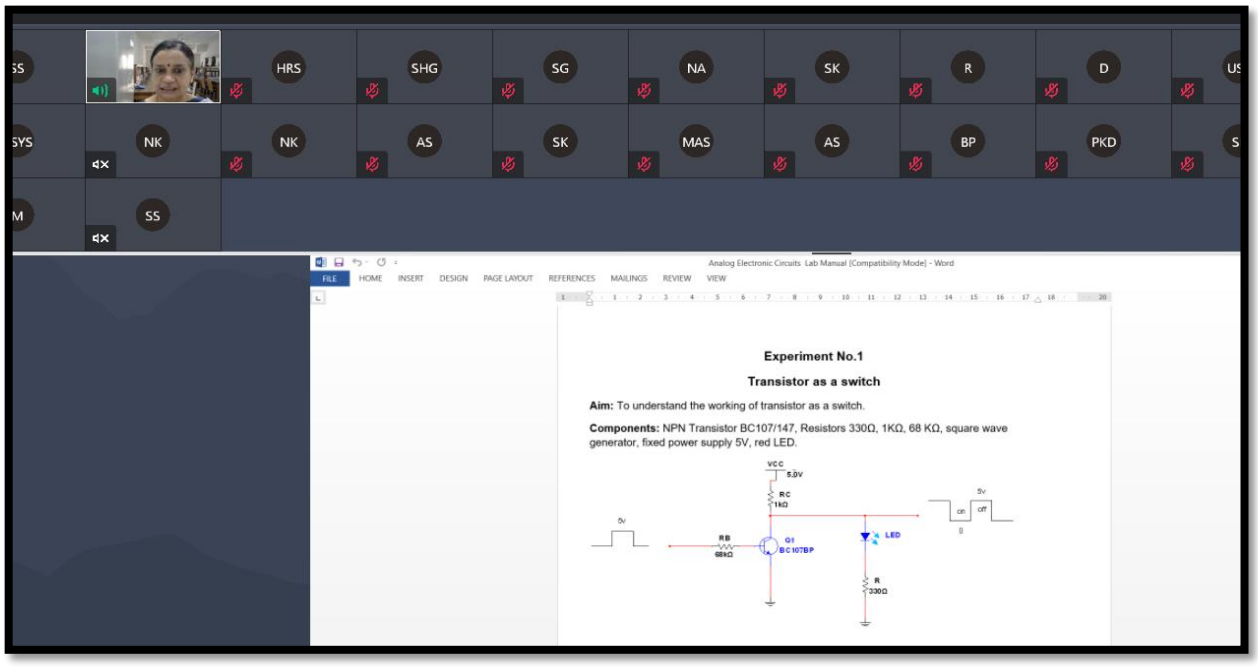
Sometimes it is desired to remove a small portion of both positive and negative half cycles. In such cases, the dual clippers are used.
The dual clippers are made by combining the biased shunt positive clipper and biased shunt negative clipper.

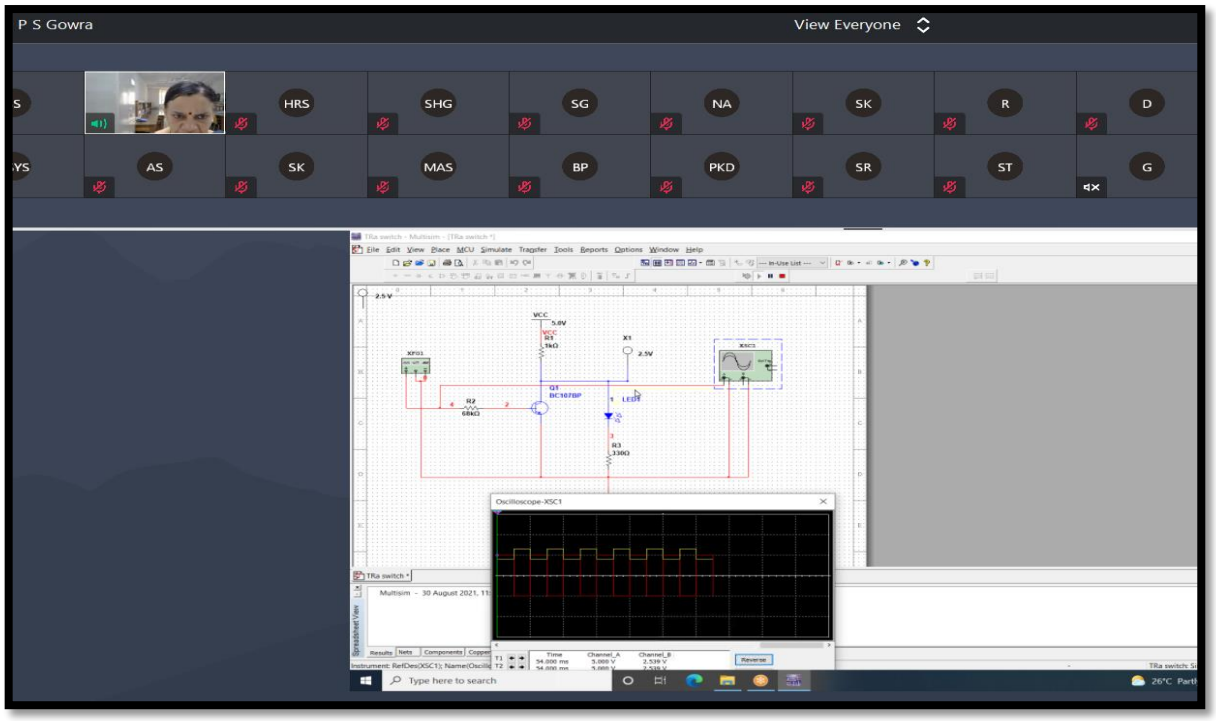




- **TRANSISTOR AS A SWITCH:**

Using a transistor as a switch is the simplest application of the device. A transistor can be extensively used for switching operation either for opening or closing of a circuit. Meanwhile, the basic concept behind the operation of a transistor as a switch relies on its mode of operation. Generally, the low voltage DC is turned on or off by transistors in this mode.
Both PNP and NPN transistors can be utilized as switches. A basic terminal transistor can be handled differently from a signal amplifier by biasing both NPN and PNP bipolar transistors by an “ON / OFF” static switch. One of the main uses of the transistor to transform a DC signal “On” or “OFF” is solid-state switches.





OUTCOME:

Students were able to implement the above mentioned content. They could analyze the practical aspects of the circuit using theoretical knowledge on multi-sim platform. They were able to learn clipping circuits and transistors its working and applications and were also able to analyze transistor working as switch with respective waveforms.

DAY-7: 31/08/2021 – REPORT

Day -7: The session started at 10:00 am. Faculty handled by P.S Gowra . About 30 students were present for the sixth day training on multi-sim.

- **Topics Covered:**

  Zener as Voltage regulator to determine Line/Load regulation.

  → Clampers

  RC coupled amplifier using multi-sim & virtual lab

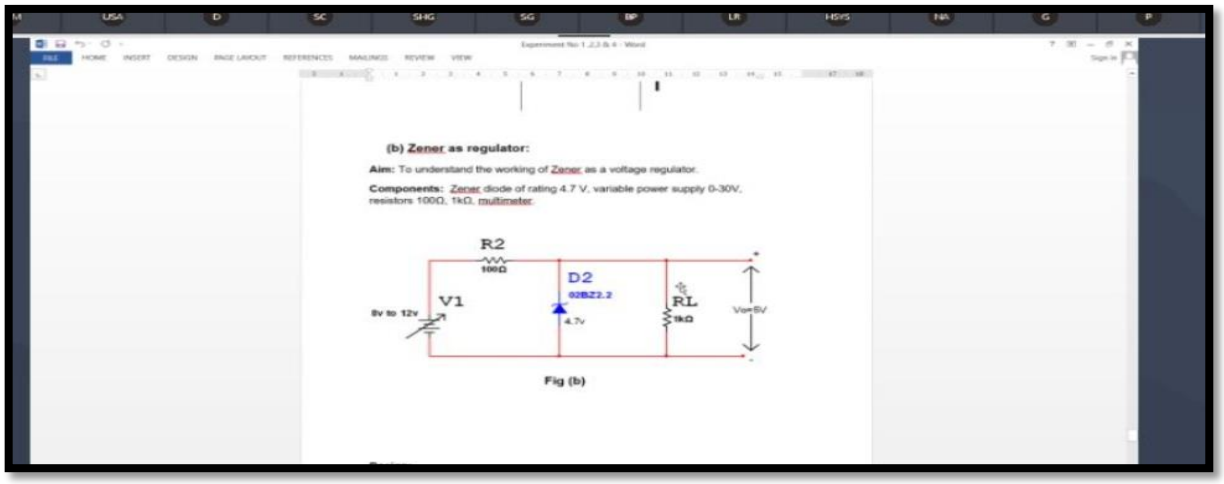
  → BJT as RC phase shift oscillator.

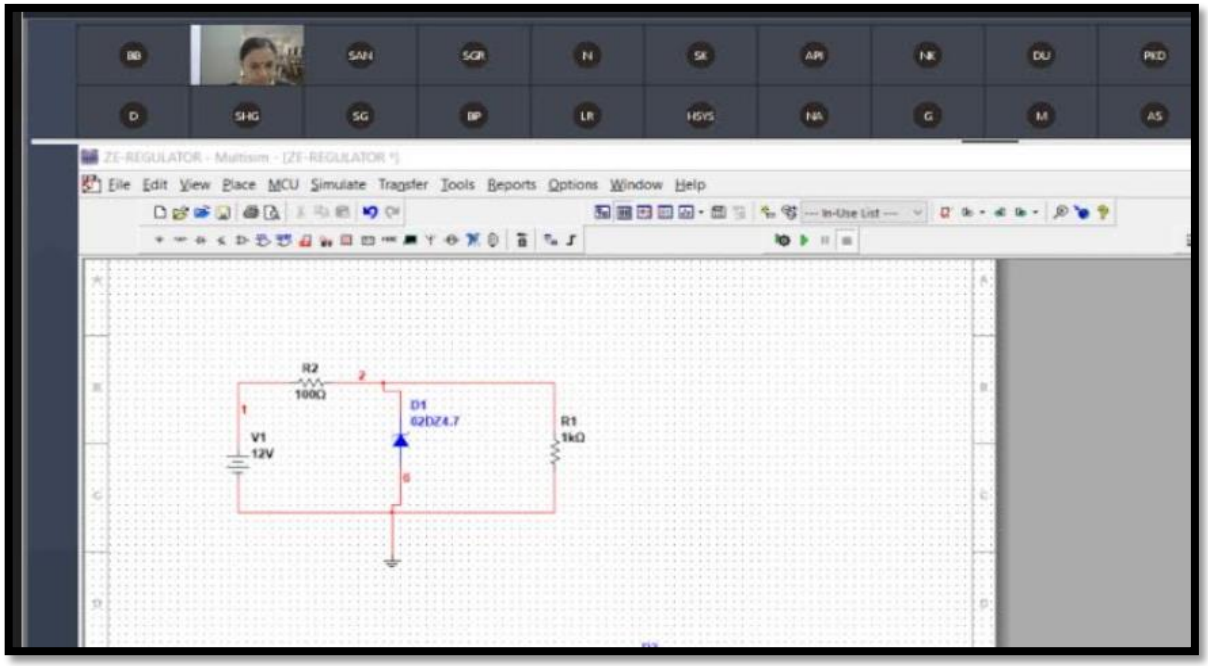
- **Zener as Voltage regulator:**

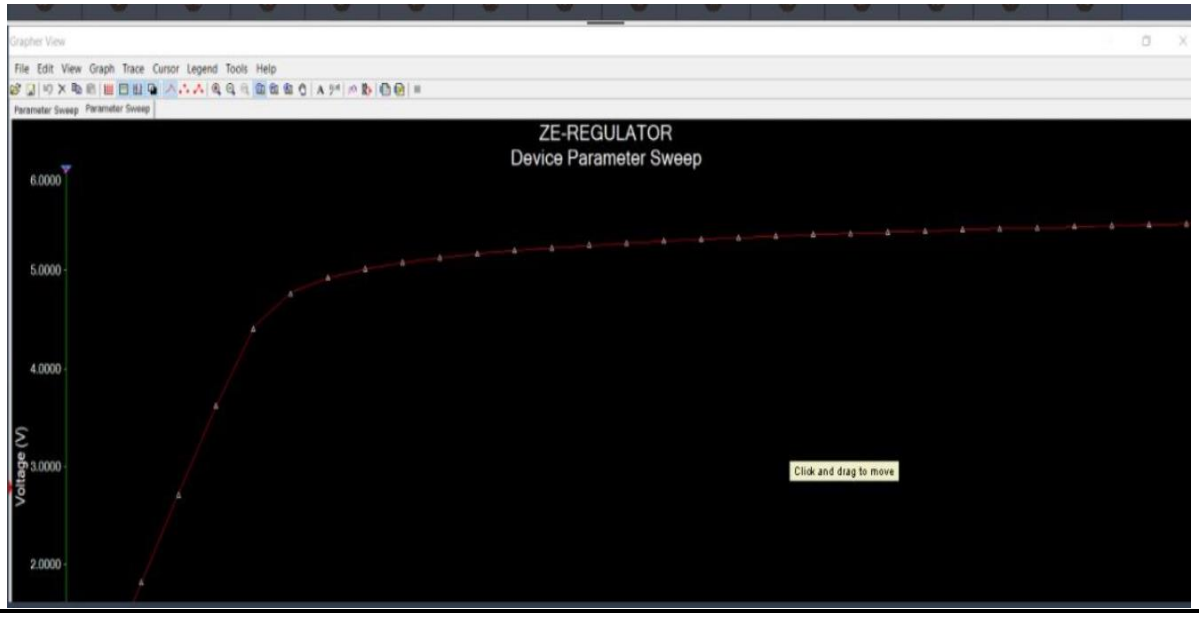
→ The Zener voltage regulator consists of a current limiting resistor R_S connected in series with the input voltage V_S with the Zener diode connected in parallel with the load R_L in this reverse biased condition. The stabilized output voltage is always selected to be the same as the breakdown voltage V_Z of the diode.

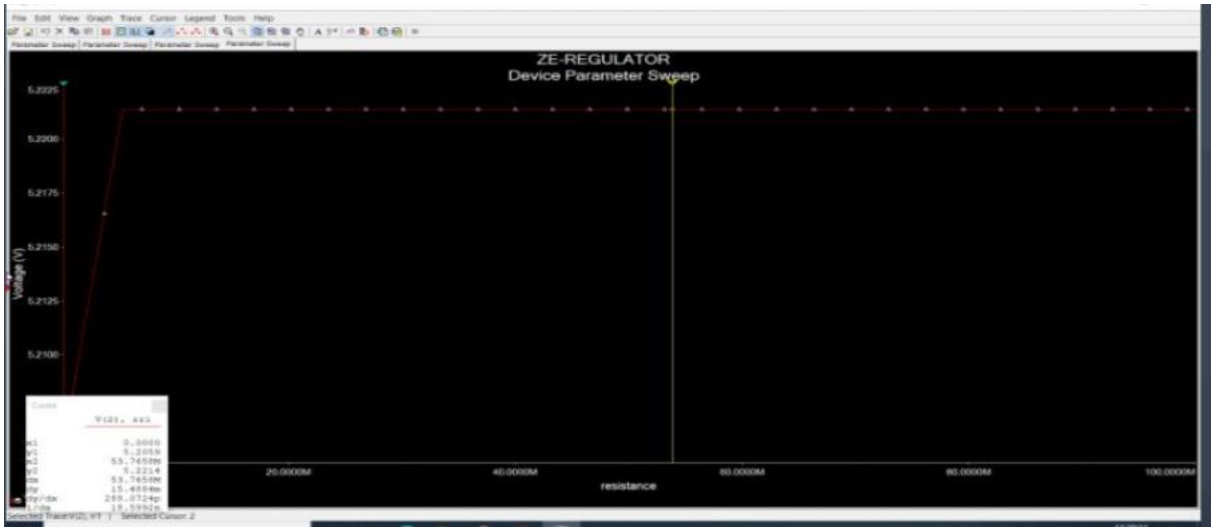
→ Line regulation: Series resistance and load resistance are fixed, only input voltage is changing. Output voltage remains the same as long as the input voltage is maintained above a minimum value.

→ Load Regulation: Input voltage is fixed and the load resistance is varying. Output volt remains same, as long as the load resistance is maintained above a minimum value.



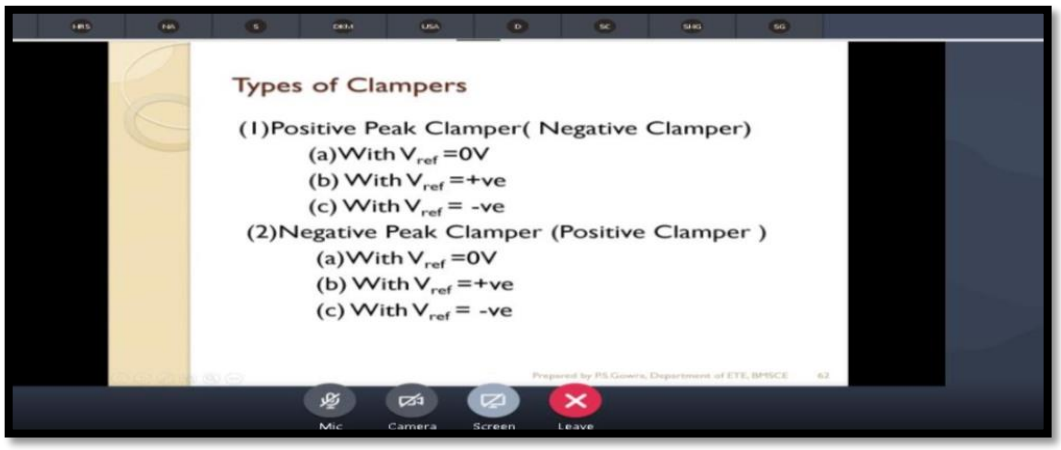


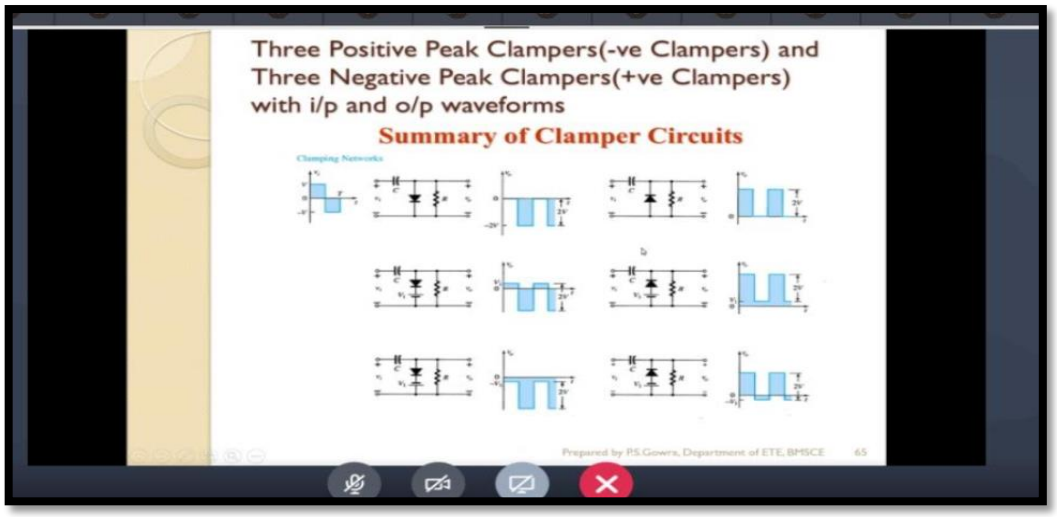




- **Clampers:**

Clampers can also be referred as DC restorers. Clamping circuits are designed to shift the input waveform either above or below a DC reference level without altering the shape of the waveform. This shifting of the waveform results in a change in the DC average voltage of the input waveform. The levels of peaks in the signal can be shifted using the clamper circuit, hence clampers can also be referred as level shifters.

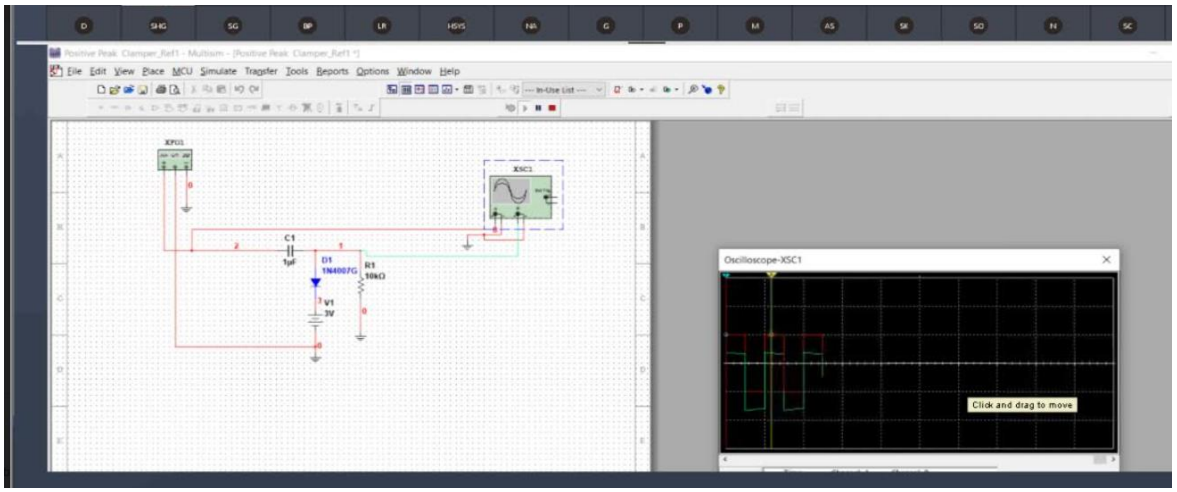


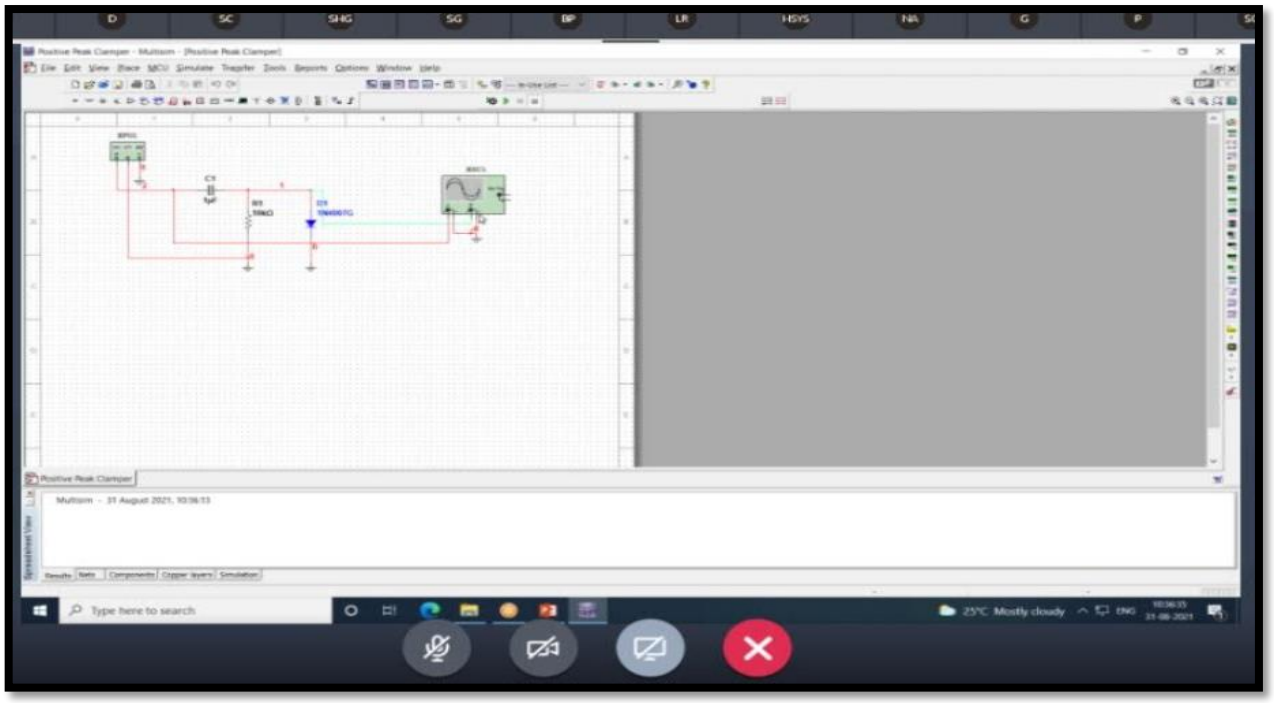


Clampers can be broadly classified into two types. They are:

→ Positive Clampers
→ Negative Clampers

Positive Clamper: This type of clamping circuit shifts the input waveform in a positive direction, as a result the waveform lies above a DC reference voltage.
Negative Clamper: This type of clamping circuit shifts the input waveform in a negative direction, as a result the waveform lies below a DC reference voltage.



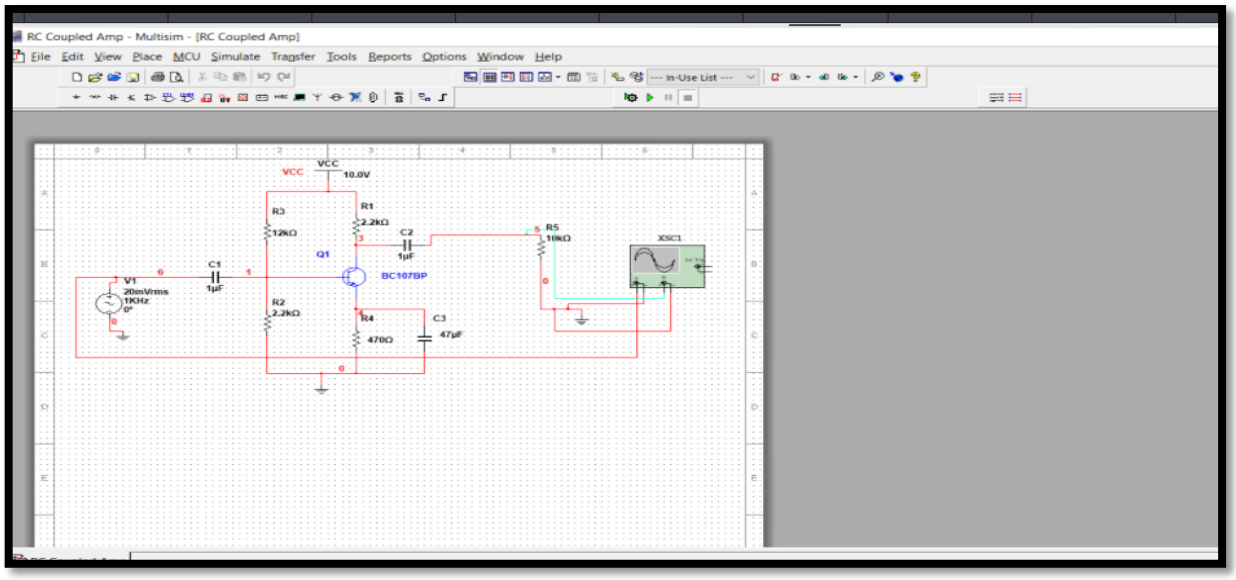


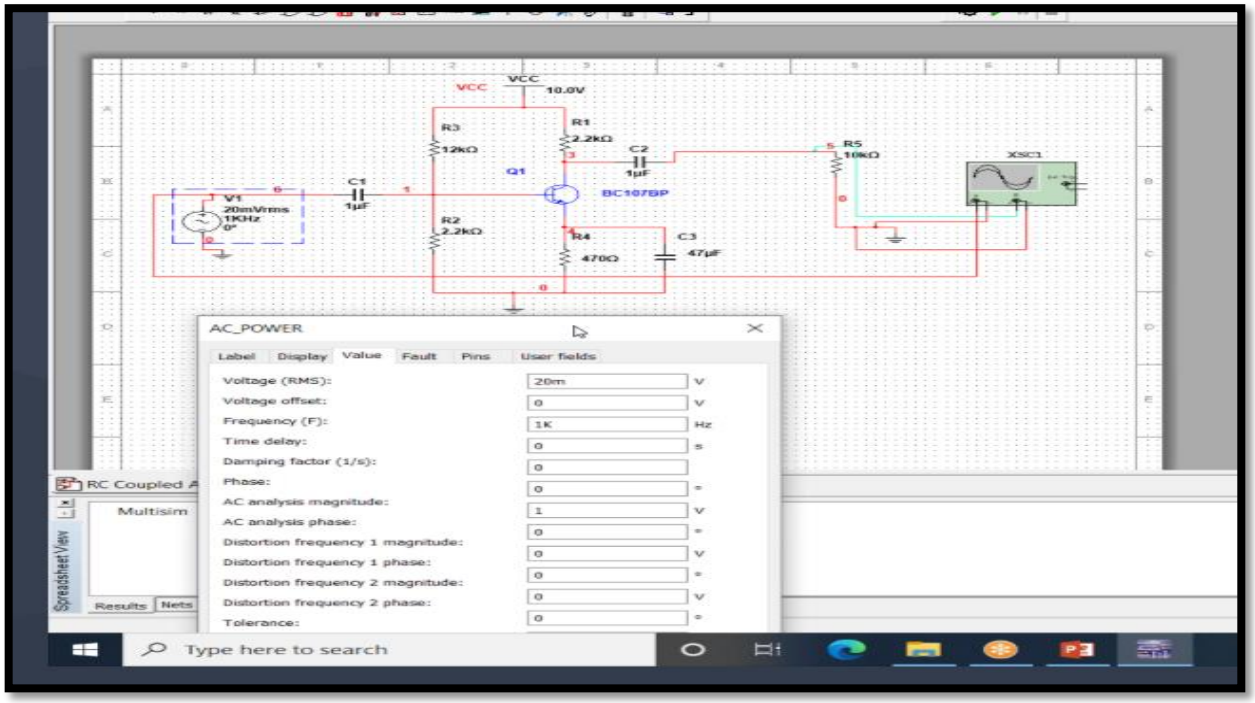
- **RC coupled amplifier:**

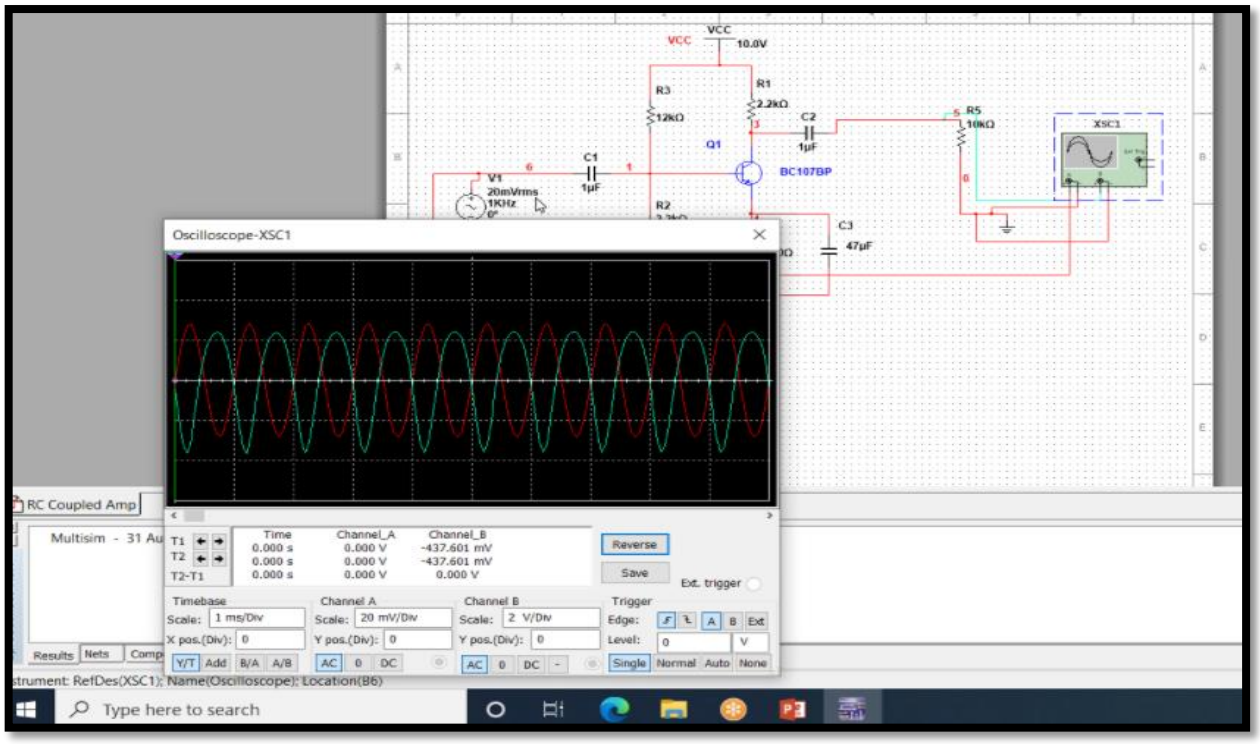
A Resistance Capacitance (RC) Coupled Amplifier is basically a multi-stage amplifier circuit extensively used in electronic circuits. Here the individual stages of the amplifier are connected together using a resistor–capacitor combination due to which it bears its name as RC Coupled. The design of individual stages of the RC coupled amplifiers is similar to that in the case of common emitter amplifiers in which the resistors R_1 and R_2 form the biasing network while the emitter resistor RE form the stabilization network.

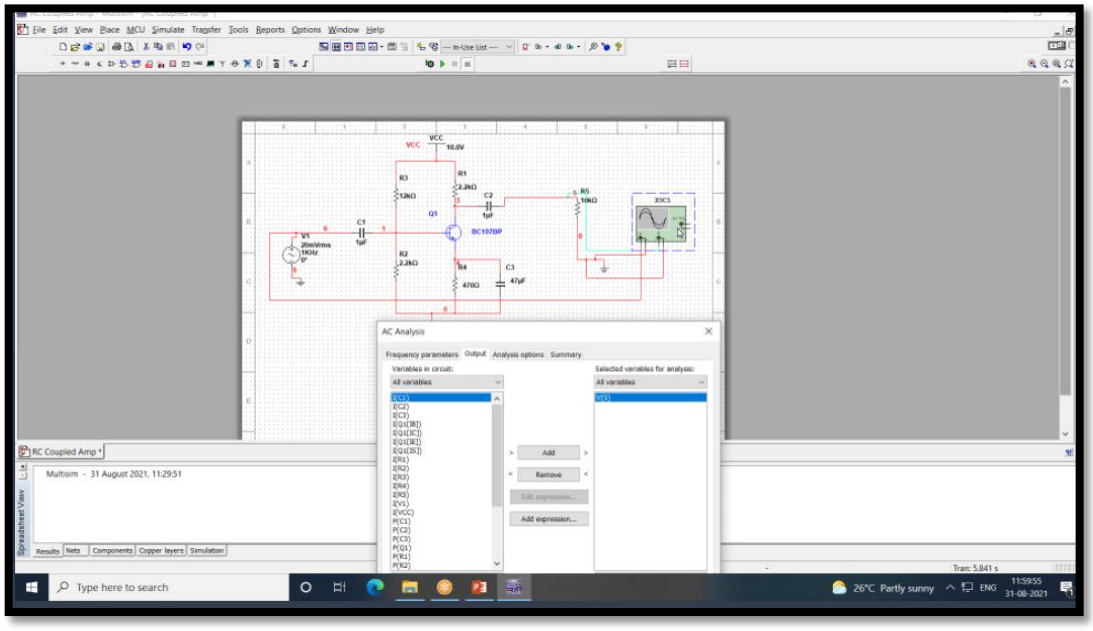
The C_E is also called bypass capacitor which passes only AC while restricting DC, which causes only DC voltage to drop across R_E while the entire AC voltage will be coupled to the next stage.

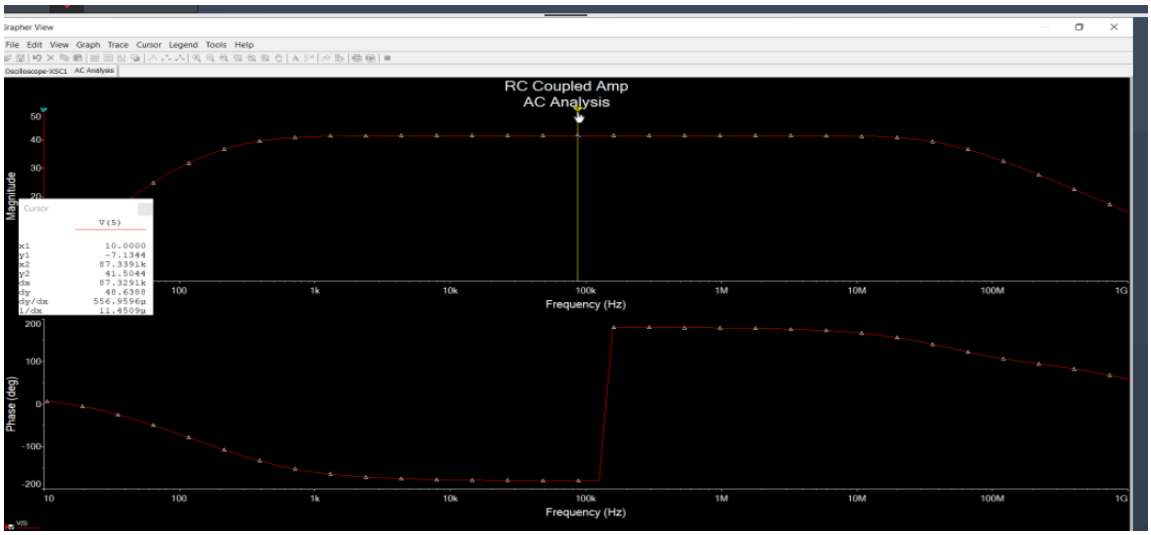
Further, the coupling capacitor C_C also increases the stability of the network as it blocks the DC while offers a low resistance path to the AC signals, thereby preventing the DC bias conditions of one stage affecting the other. In addition, in this circuit, the voltage drop across the collector-emitter terminal is chosen to be 50% of the supply voltage V_{CC} in order to ensure appropriate biasing point.

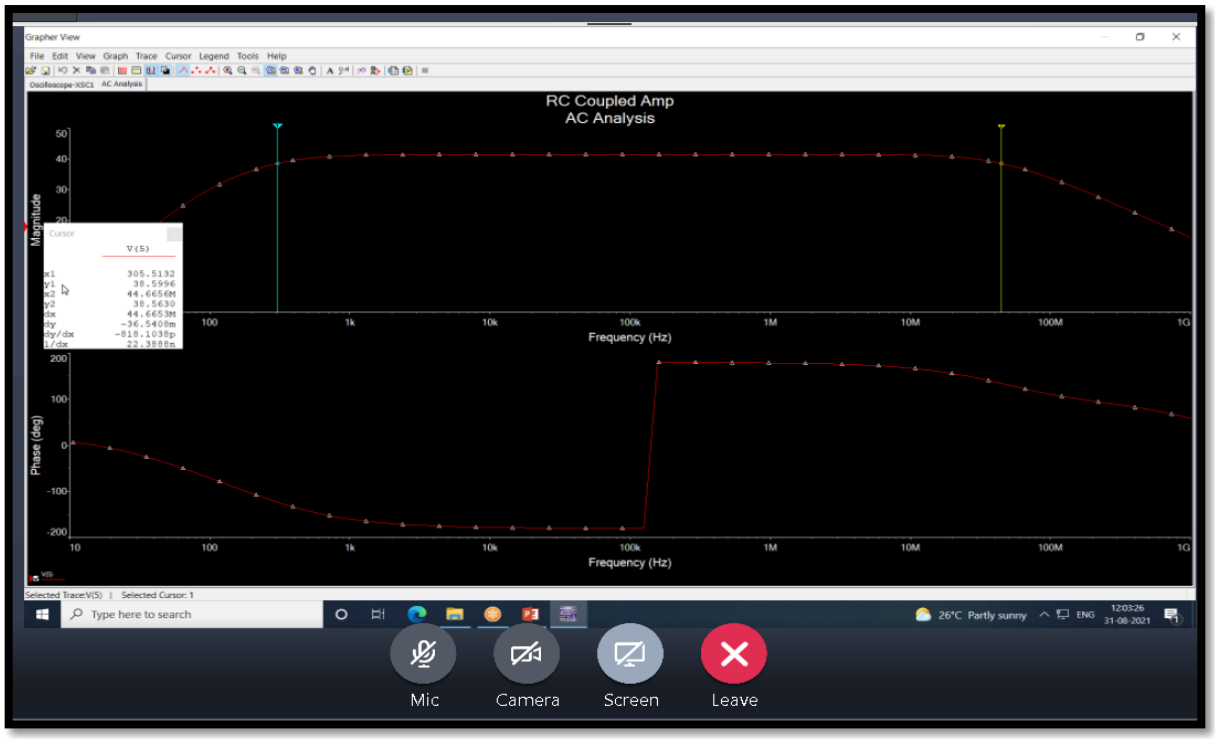

RC Coupled Amp - Multisim - [RC Coupled Amp]
File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help
VCC
10.0V
R1
2.2kΩ
R3
12kΩ
C2
1µF
R5
10kΩ
XSC1
Q1
BC107BP
C1
1µF
V1
20mVrms
1KHz
0°
R2
2.2kΩ
R4
470Ω
C3
47µF


VCC
10.0V
R1
2.2kΩ
R3
12kΩ
C2
1µF
R5
10kΩ
XSC1
Q1
BC107BP
C1
1µF
V1
20mVrms
1KHz
0°
R2
2.2kΩ
R4
470Ω
C3
47µF
AC_POWER
Label Display Value Fault Pins User fields
Voltage (RMS): 20m V
Voltage offset: 0 V
Frequency (F): 1K Hz
Time delay: 0 s
Damping factor (1/s): 0
Phase: 0 °
AC analysis magnitude: 1 V
AC analysis phase: 0 °
Distortion frequency 1 magnitude: 0 V
Distortion frequency 1 phase: 0 °
Distortion frequency 2 magnitude: 0 V
Distortion frequency 2 phase: 0 °
Tolerance: 0 °
RC Coupled A
Multisim
Spreadsheet View
Results Nets
Type here to search


VCC
10.0V
R3
12kΩ
R1
2.2kΩ
C2
1µF
R5
10kΩ
XSC1
Q1
BC107BP
C1
1µF
V1
20mVrms
1KHz
R2
C3
47µF
Oscilloscope-XSC1
RC Coupled Amp
Multisim - 31 Au
Time
Channel_A
Channel_B
T1
T2
T2-T1
0.000 s
0.000 V
-437.601 mV
0.000 s
0.000 V
-437.601 mV
0.000 s
0.000 V
0.000 V
Reverse
Save
Ext. trigger
Timebase
Scale: 1 ms/Div
X pos.(Div): 0
Channel A
Scale: 20 mV/Div
Y pos.(Div): 0
Channel B
Scale: 2 V/Div
Y pos.(Div): 0
Trigger
Edge:
Level: 0 V
Single
Normal
Auto
None
Results
Nets
Type here to search

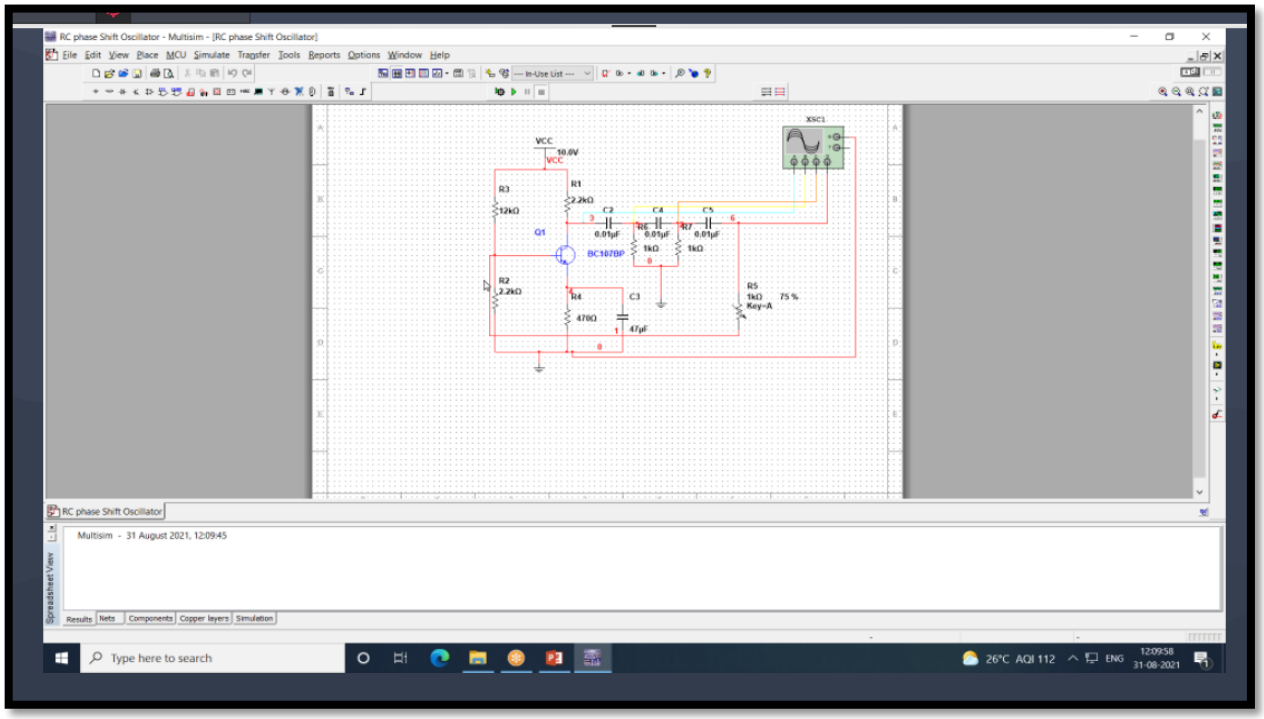

AC Analysis
Frequency parameters
Output
Analysis options
Summary
Add
Remove
RC Coupled Amp
Multisim - 31 August 2021, 11:29:51
Type here to search
26°C Partly sunny
ENG
11:59:55
31-08-2021

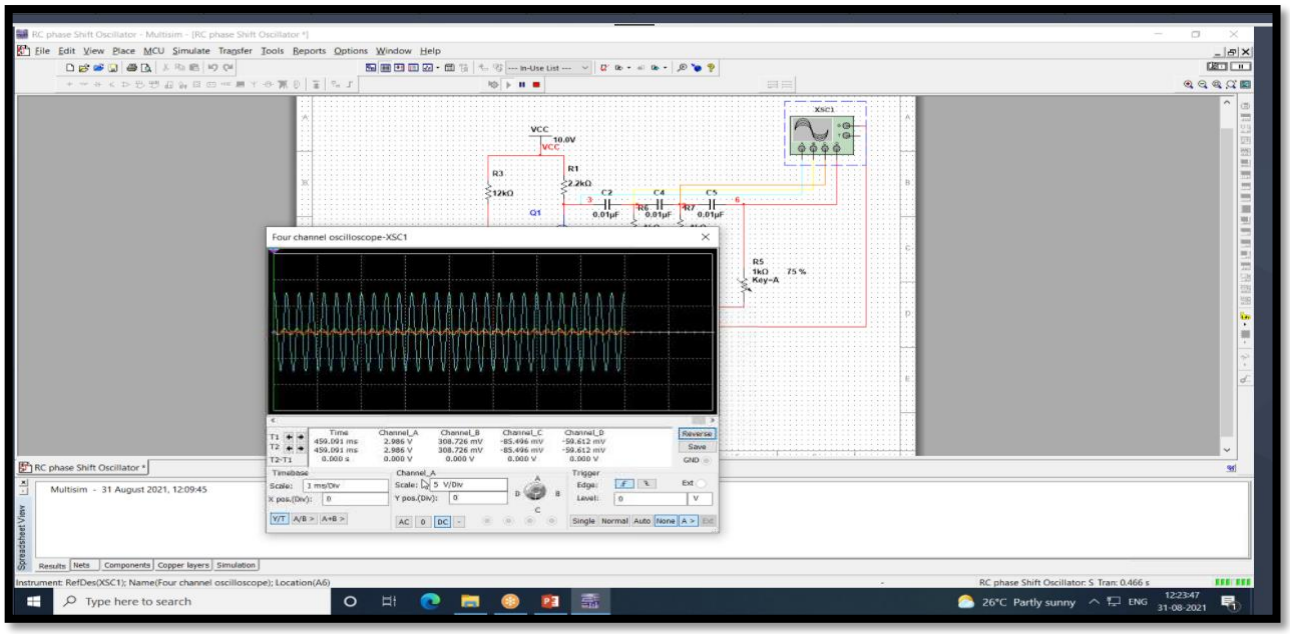

RC Coupled Amp
AC Analysis
Magnitude
Phase (deg)
Frequency (Hz)
Frequency (Hz)
10
100
1k
10k
100k
1M
10M
100M
1G

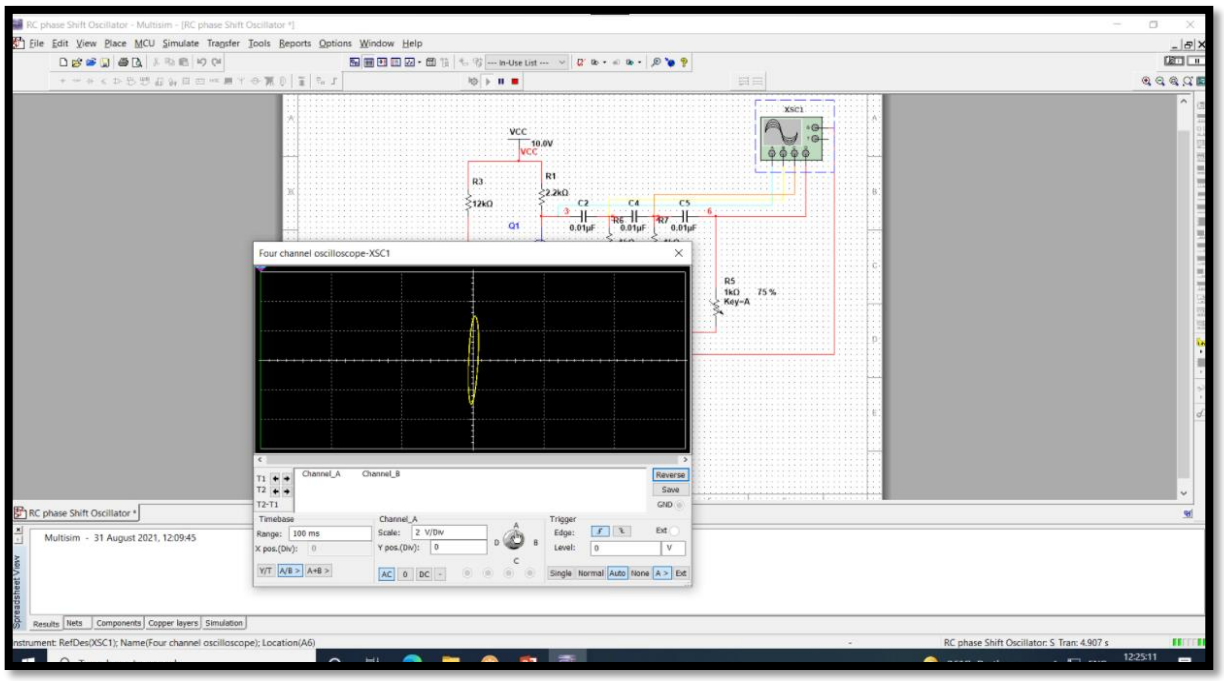


- **<u>BJT as RC Phase shift oscillator:</u>**

RC phase-shift oscillator circuit can be built with a resistor as well as a capacitor. This circuit offers the required phase shift with the feedback signal. They have outstanding frequency strength and can give a clean sine wave for an extensive range of loads. Preferably an easy RC network can be expected to include an o/p which directs the input with 90 degree. The following RC phase shift oscillator circuit using BJT can be built by cascading 3-RC phase shift networks; each provides a 60^0 phase shift. In the circuit, the RC which is known as the collector resistor stops the transistor's collector current.







OUTCOME:

Students were able to implement the above mentioned content. They could analyze the practical aspects of the circuit using theoretical knowledge on multi-sim platform. They were able to learn Zener as voltage regulator and also determine it has line and load regulations. They also learnt RC coupled amplifier in multi-sim as well as in virtual lab. They learnt BJT as RC phase shift oscillator. Also clamping circuits were also taught to implement.

DAY-8: 01/09/2021 – REPORT

Day -8: The session started at 10:00 am. Faculty handled by P.S Gowra . About 33 students were present for the sixth day training on multi-sim.

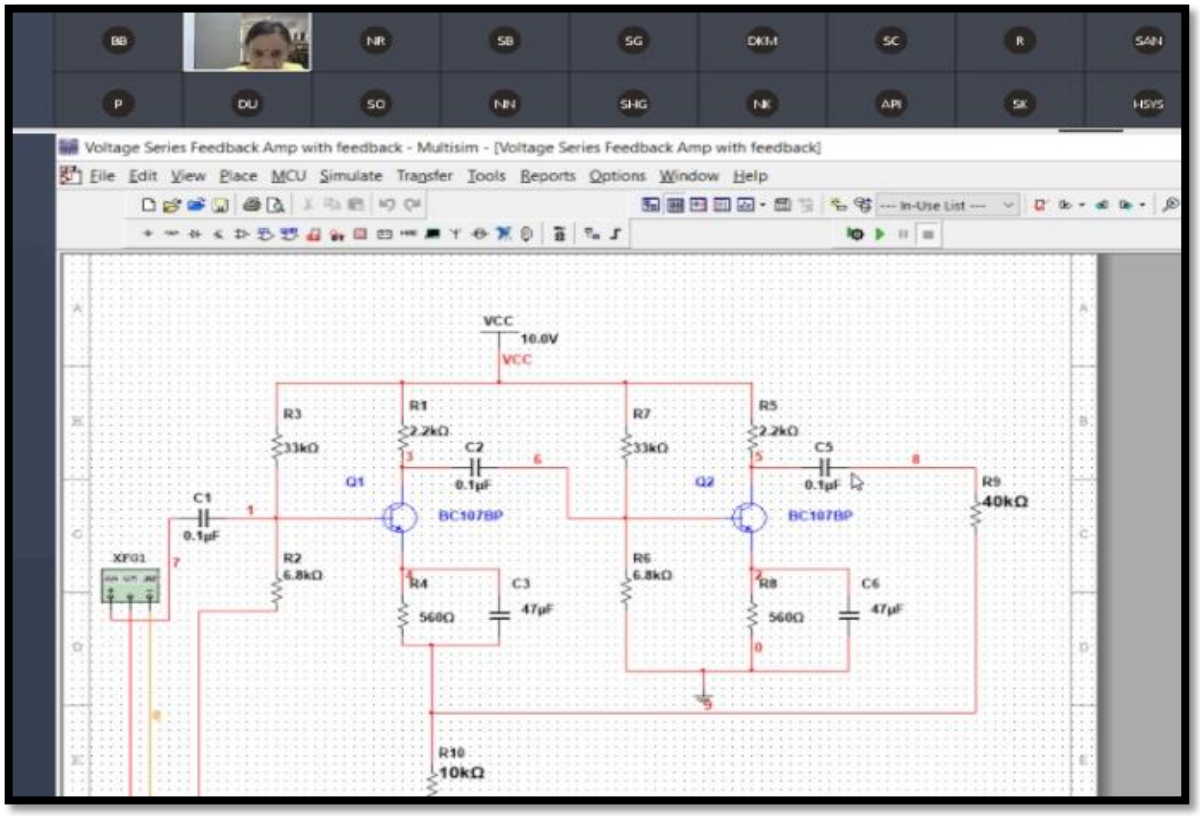
- **Topics Covered:**

 Voltage series with and without feedback.
Drain and source characteristics of MOSFET.
→ RC coupled circuits in Virtual lab.

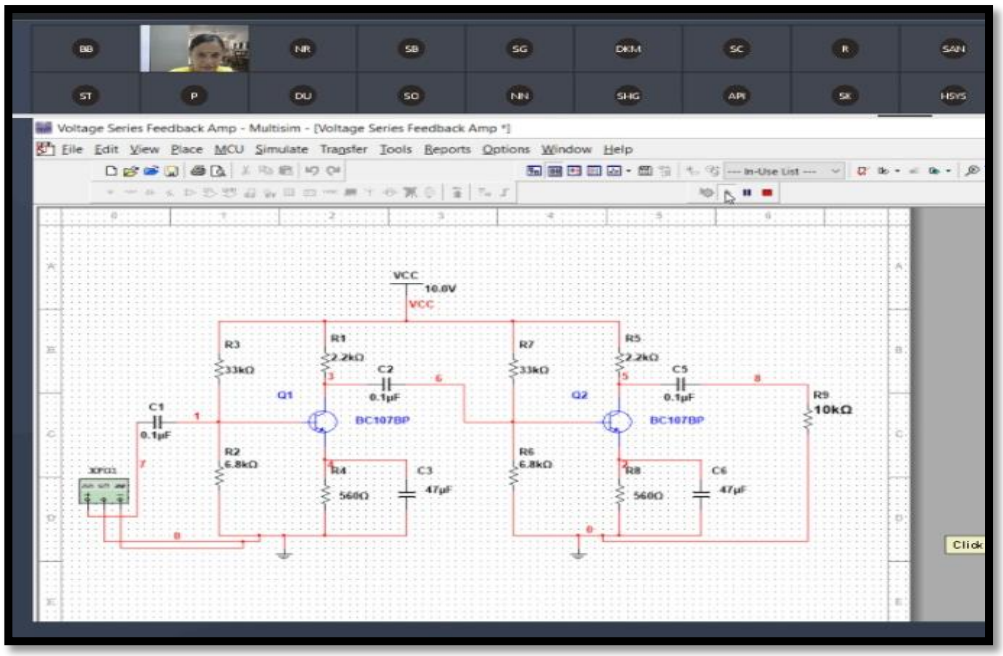
- **Voltage series feedback amplifier with feedback**:

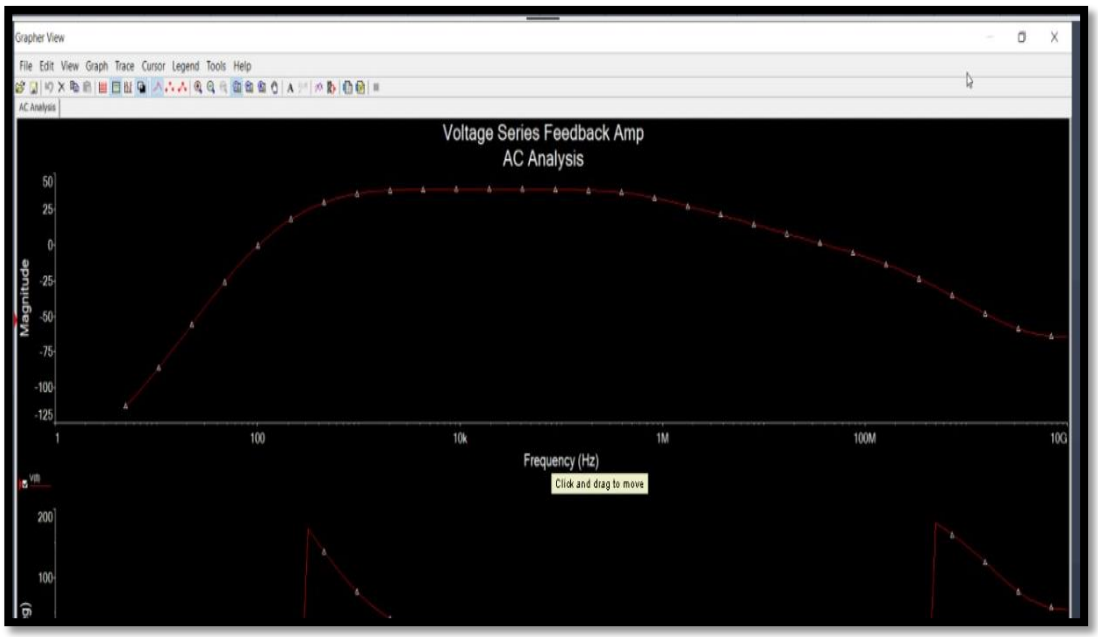
When any increase in the output signal results into the input in such a way as to cause the decrease in the output signal, the amplifier is said to have negative feedback. The advantages of providing negative feedback are that the transfer gain of the amplifier with feedback can be stabilized against variations in the hybrid parameters of the transistor or the parameters of the other active devices used in the circuit. The most advantage of the negative feedback is that by proper use of this, there is significant improvement in the frequency response and in the linearity of the operation of the amplifier. This disadvantage of the negative

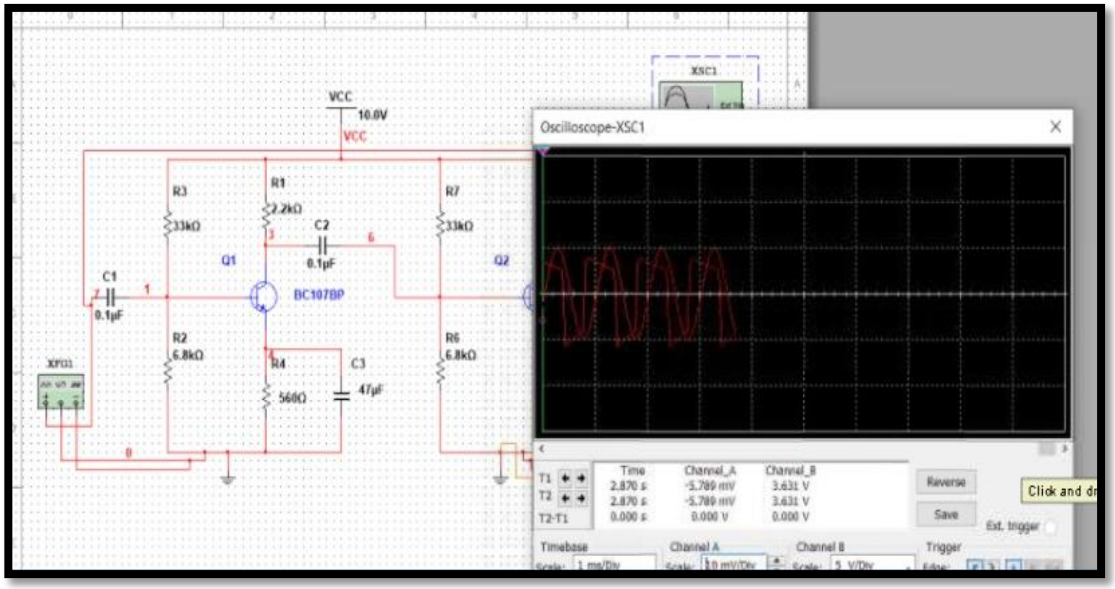
feedback is that the voltage gain is decreased. In Voltage-Series feedback, the input impedance of the amplifier is increased and the output impedance is decreased. Noise and distortions are reduced considerably.



- **Voltage series feedback amplifier without feedback**:


Voltage Series Feedback Amp - Multisim - [Voltage Series Feedback Amp *]
File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help
VCC
10.0V
R3
33kΩ
R1
2.2kΩ
C2
0.1µF
Q1
BC107BP
C1
0.1µF
R2
6.8kΩ
R4
560Ω
C3
47µF
R7
33kΩ
R5
2.2kΩ
C5
0.1µF
Q2
BC107BP
R9
10kΩ
R6
6.8kΩ
R8
560Ω
C6
47µF
XFG1
Click


Grapher View
File Edit View Graph Trace Cursor Legend Tools Help
AC Analysis
Voltage Series Feedback Amp
AC Analysis
Magnitude
50
25
0
-25
-50
-75
-100
-125
1
100
10k
1M
100M
10G
Frequency (Hz)
Click and drag to move
200
100

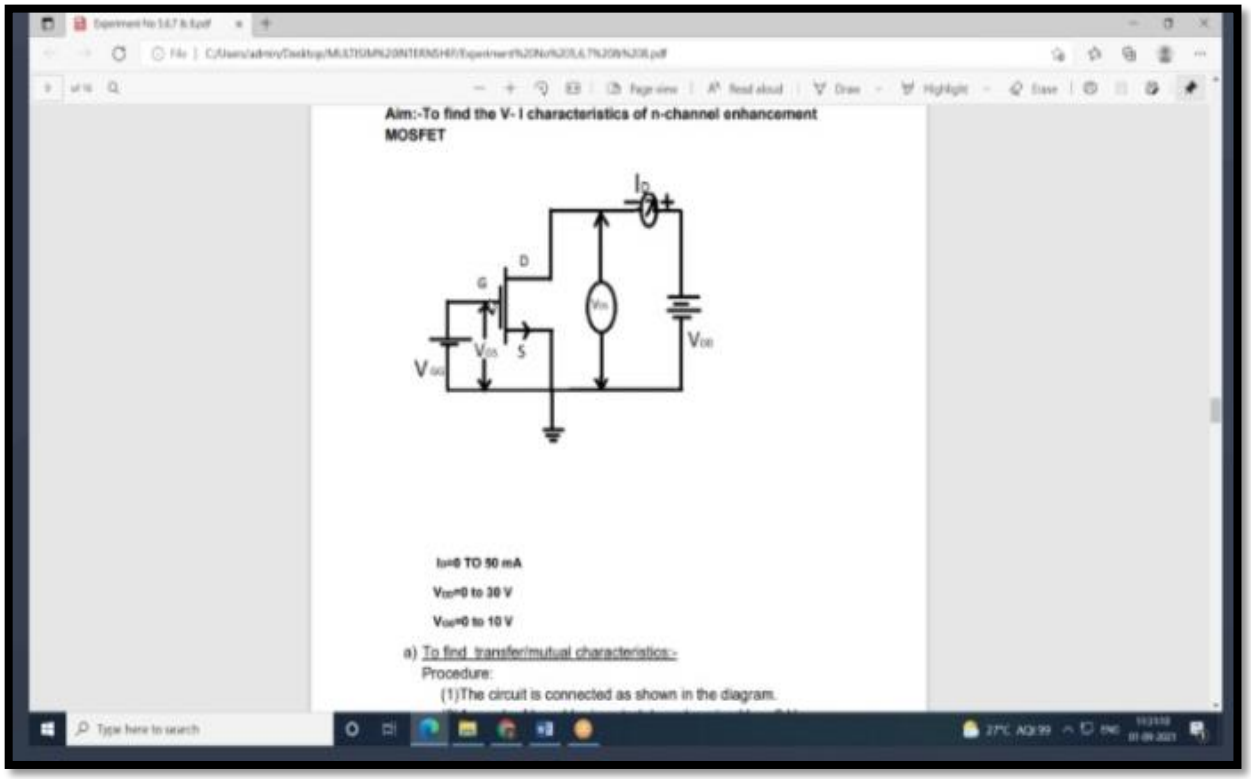

XSC1
Oscilloscope-XSC1
VCC
10.0V
R3
33kΩ
R1
2.2kΩ
C2
0.1µF
Q1
BC107BP
C1
0.1µF
R2
6.8kΩ
R4
560Ω
C3
47µF
R7
33kΩ
Q2
R6
6.8kΩ
XFG1
Time
Channel_A
Channel_B
T1
T2
T2-T1
2.870 s
-5.789 mV
3.631 V
0.000 s
0.000 V
Reverse
Save
Timebase
Channel A
Channel B
Trigger
Click and dr

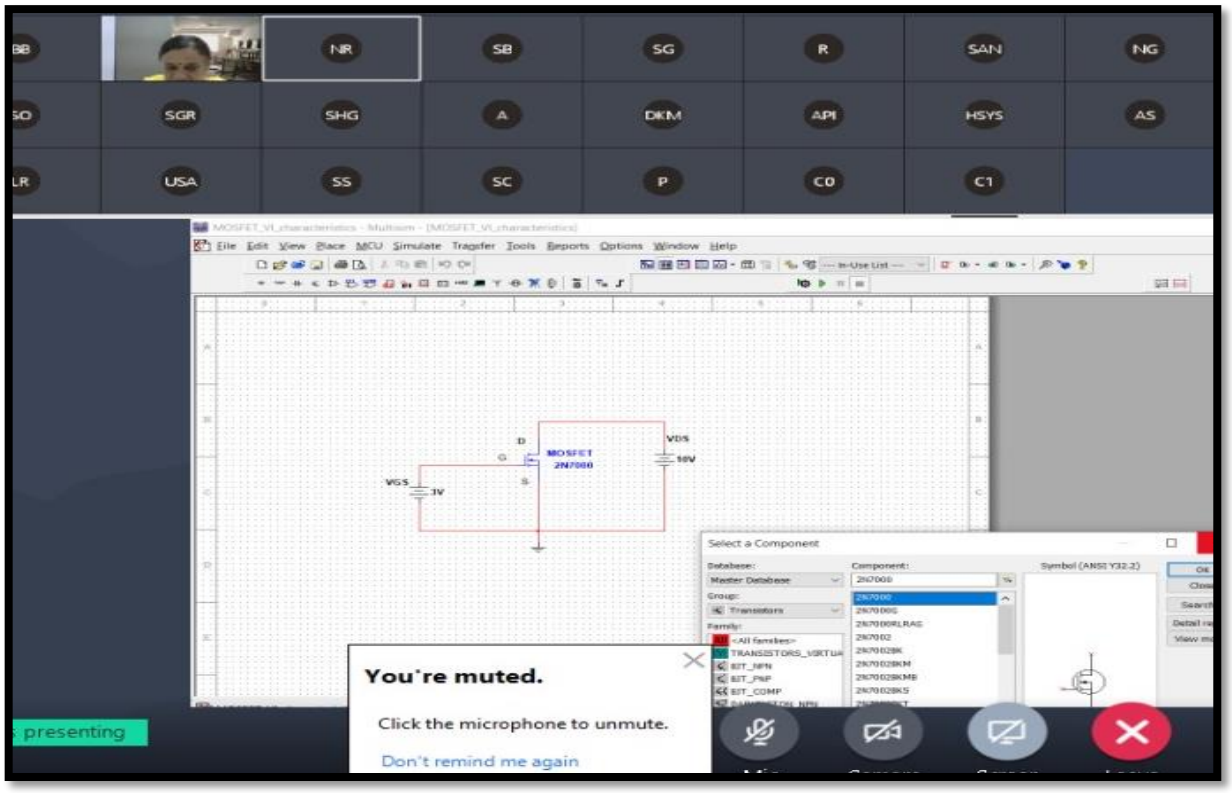
- **<u>Drain and source characteristics of MOSFET</u>** :

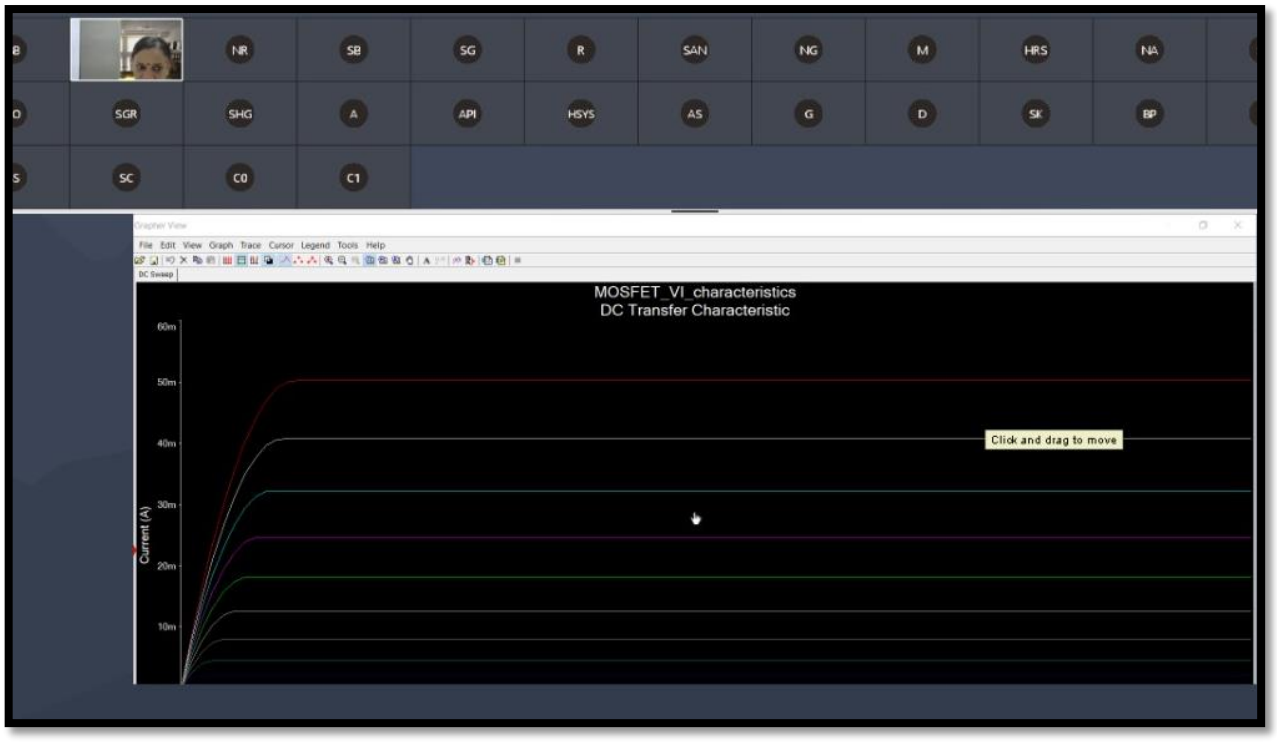
MOSFET stands for Metal Oxide Silicon Field Effect Transistor or Metal Oxide Semiconductor Field Effect Transistor.

The drain characteristics of a MOSFET are drawn between the drain current $\mathbf{I_D}$ and the drain source voltage $\mathbf{V_{DS}}$. The characteristic curve is as shown below for different values of inputs.

Actually when $\mathbf{V_{DS}}$ is increased, the drain current $\mathbf{I_D}$ should increase, but due to the applied $\mathbf{V_{GS}}$, the drain current is controlled at certain level. Hence the gate current controls the output drain current.

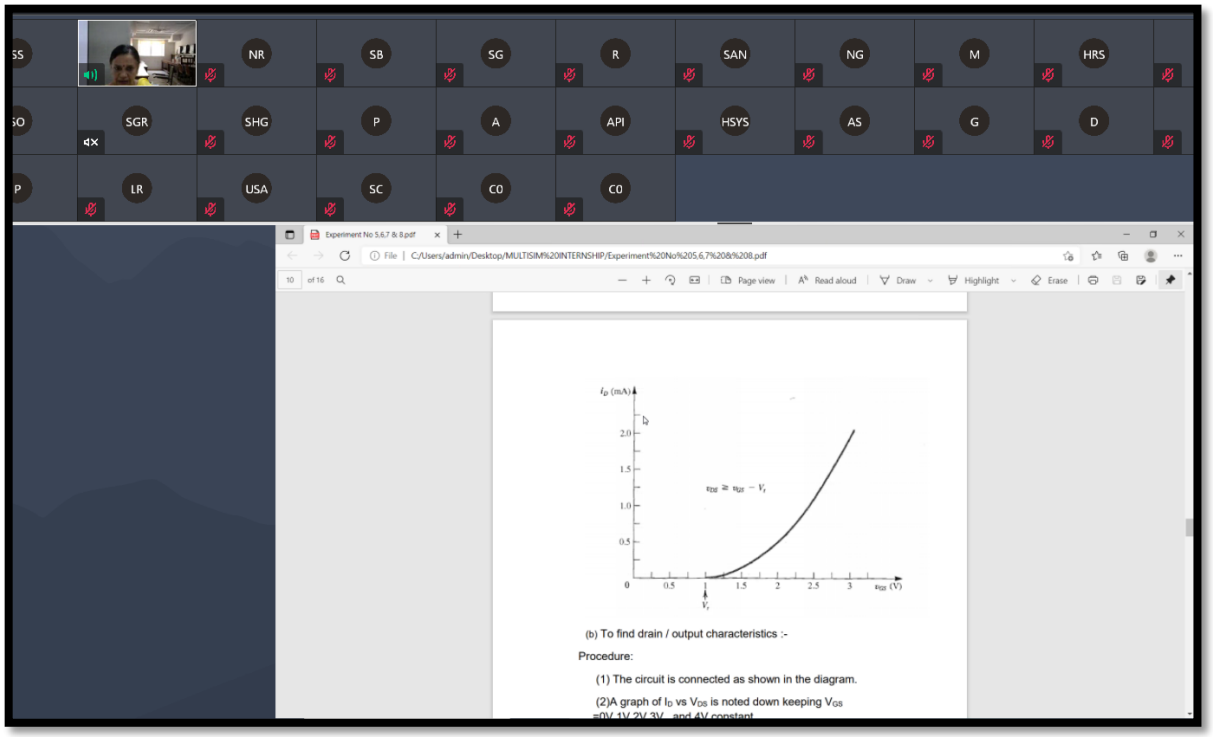


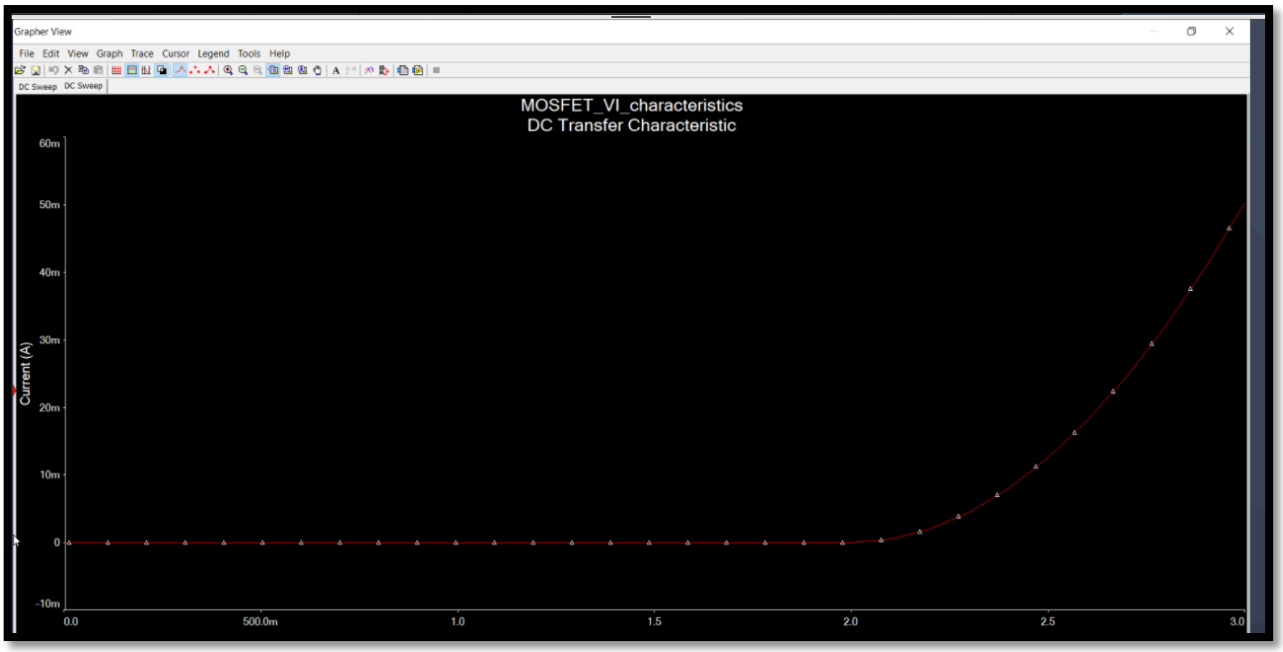




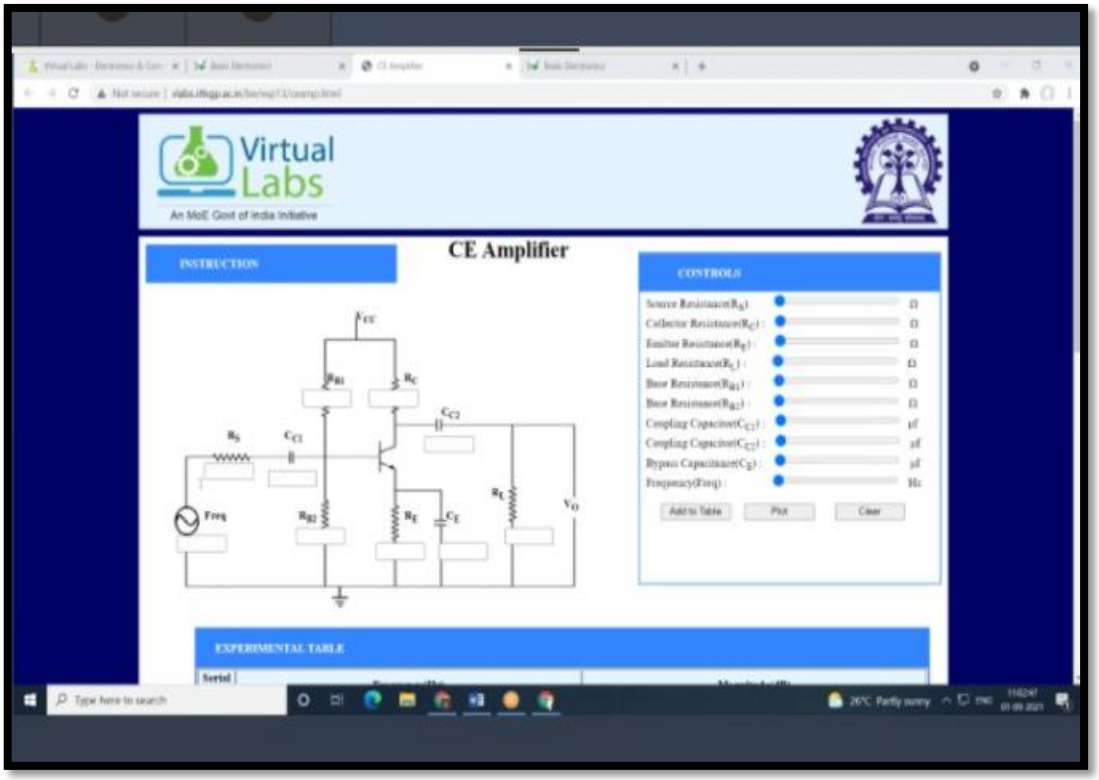
Transfer Characteristics

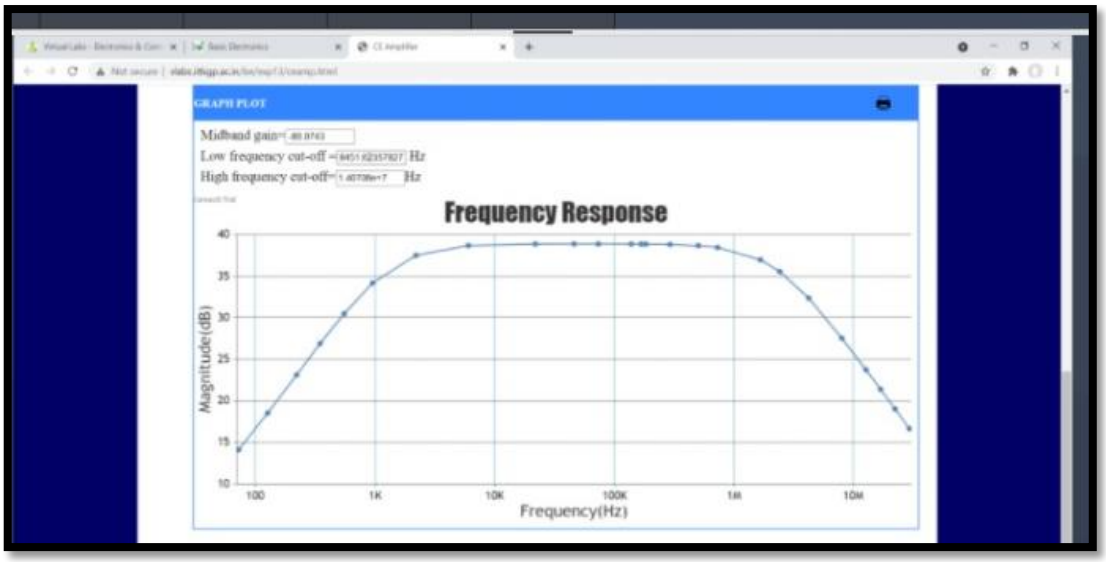
Transfer characteristics define the change in the value of $\mathbf{V_{DS}}$ with the change in $\mathbf{I_D}$ and $\mathbf{V_{GS}}$ in both depletion and enhancement modes. The below transfer characteristic curve is drawn for drain current versus gate to source voltage.

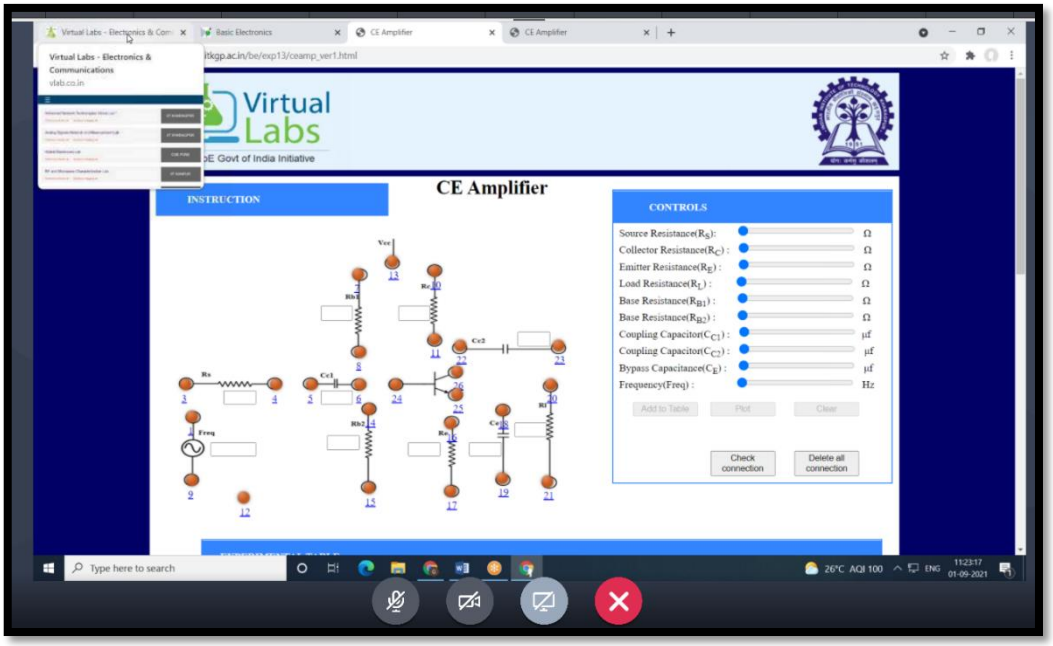




- **<u>RC coupled circuits in Virtual lab:</u>**





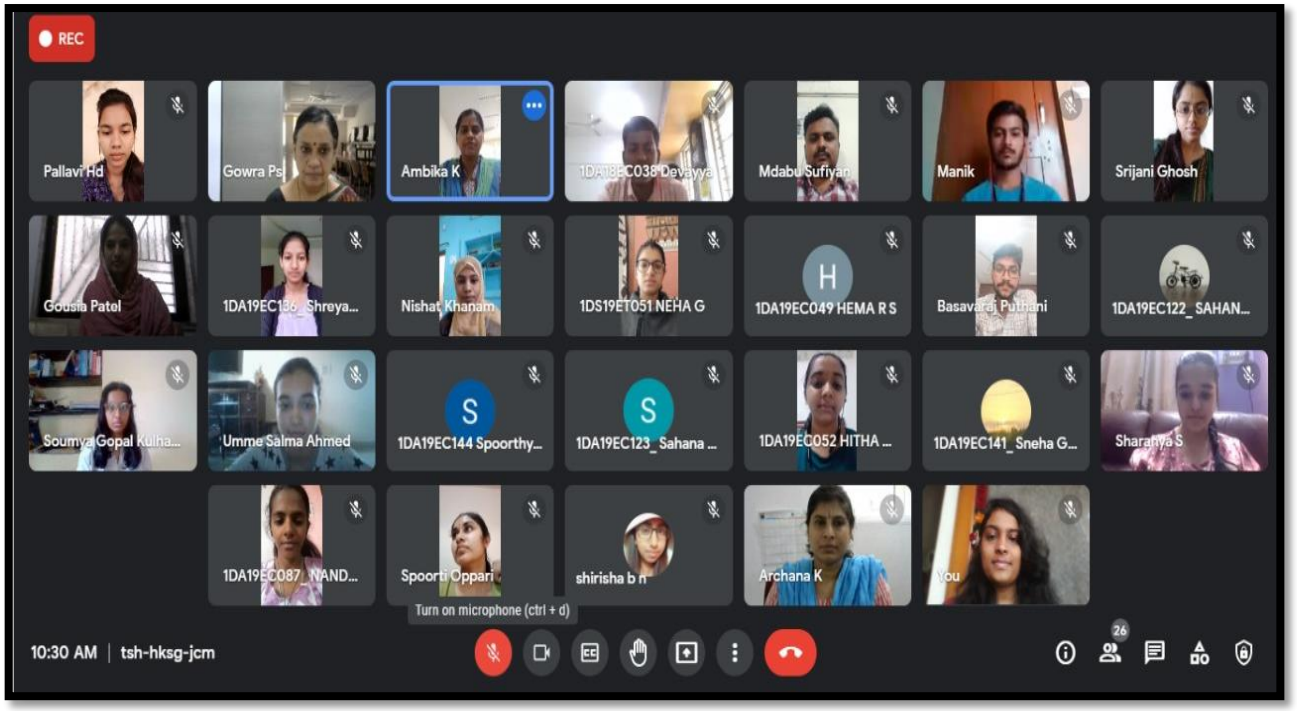


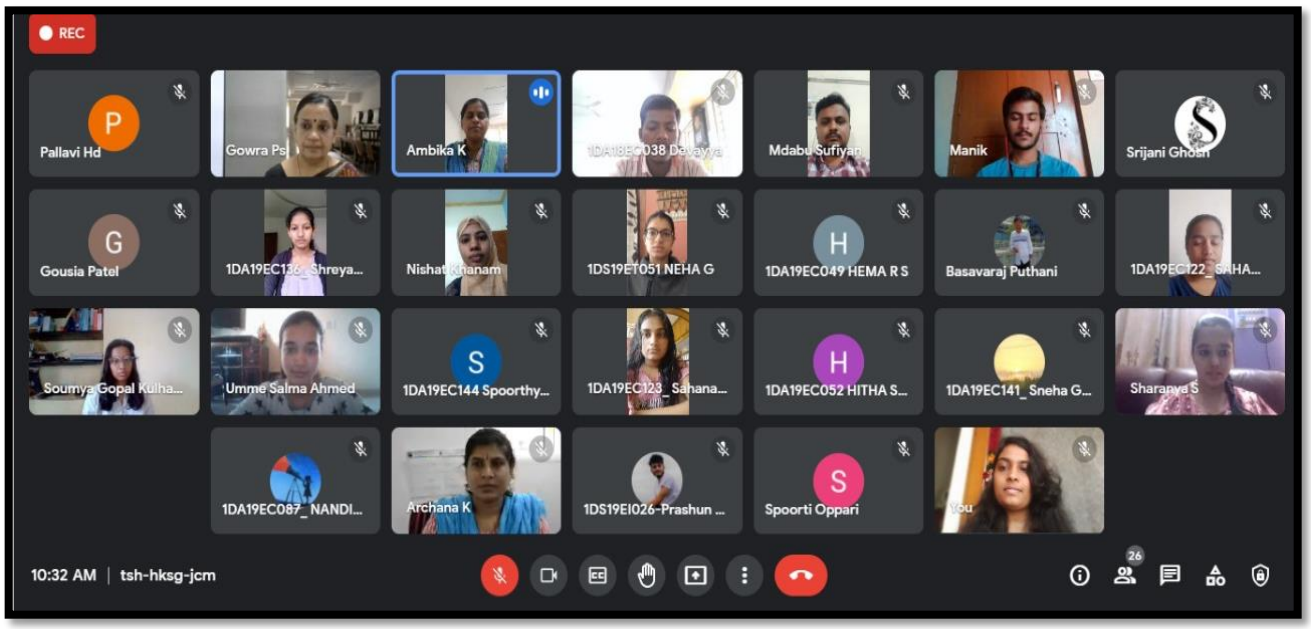
OUTCOME:

Students were able to implement the above mentioned content. And they have learnt to rig up the circuit in both multisim and virtual labs.
Students also gained knowledge on DC and AC analysis, frequency responses as well as transfer characteristics of given circuit or IC.

DAY-8,9,10: 02/09/2021-4/09/2021

As mentioned earlier in the schedule three days from 2nd September to 4th September students presented their project under the supervision of P.S.Gowra mam. Students at the end of the internship were able to design and implement mini-project using multisim and presented, executed and explained them in the meet.





PROJECT TOPICS AND SCHEDULE

BMS COLLEGE OF ENGINEERING
ELECTRONICS AND TELECOMMUNICATION

Project Groups & Project Title

Schedule for Project presentation

DATE	Project Groups	
02/09/2021	**Srijani Ghosh**	Function Generator
	Dinesh U	DC Regulated power supply
	Hema R S **Hitha Shree Y S**	Sine wave to Square wave converter
	Shashidhar S **Suparna B K**	Band pass filter
	Sahana K S **Sneha G R** **Spoorthi H G**	Hartley oscillator
	Nanditha A **Sahana A N** **Shreya C**	Automatic Door Bell
	Anusha T A **Siri Ranganath**	Password security Instrumentation amplifier
	Neha G **Prashun Kumar Dash**	Mobile phone charger
	Rashmi T	Low pass filter
	Adrian P Issac	Simple D to A converter using op-amp

DATE	Project Groups	
03/09/2021	Nishat Khanam Maheshwari	Traffic light controller with timers and counters
	Spoorti Oppari Basavaraj Puthani	Automatic Washroom light project
	Avyukt Sharma Shirisha	3 Phase sine wave generator
	Divya Abdurrahim	Op-Amp Transistor PWM
	Umme Salma Ahmed	PWM LED Dimmer using NE555 timer.
	Soumya Kulhali Nimisha MR	FM Listening bugger circuit
	Sandeep T Likith P Reddy	Mobile Detector circuit

04/09/2021	Devayya Manik	Mobile Jammer circuit
	Gousia Begum Pallavi Md Abu Sufiyan	Simple battery level indicat using op-amp

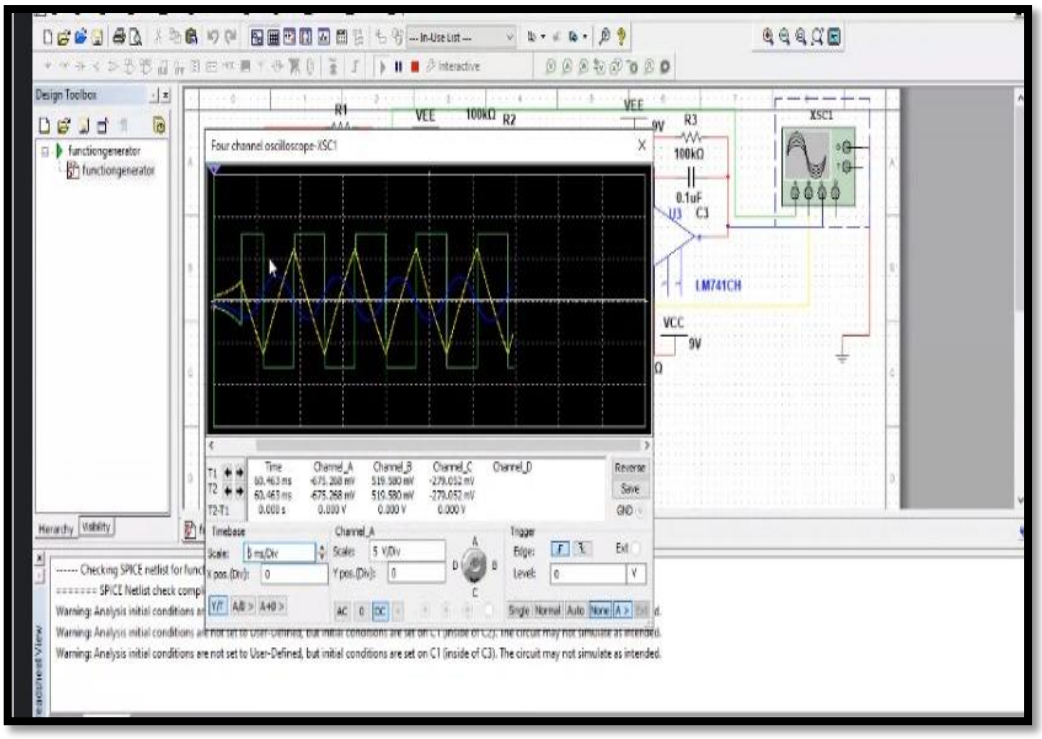
OUTCOME

The students learnt to use the multisim software and bridge the gap between theory and real world. By working on project students gained knowledge on various real world applications on analog as well as digital electronics. And also they were able to analyze graphs, frequencies and other characteristics of the electronic components.

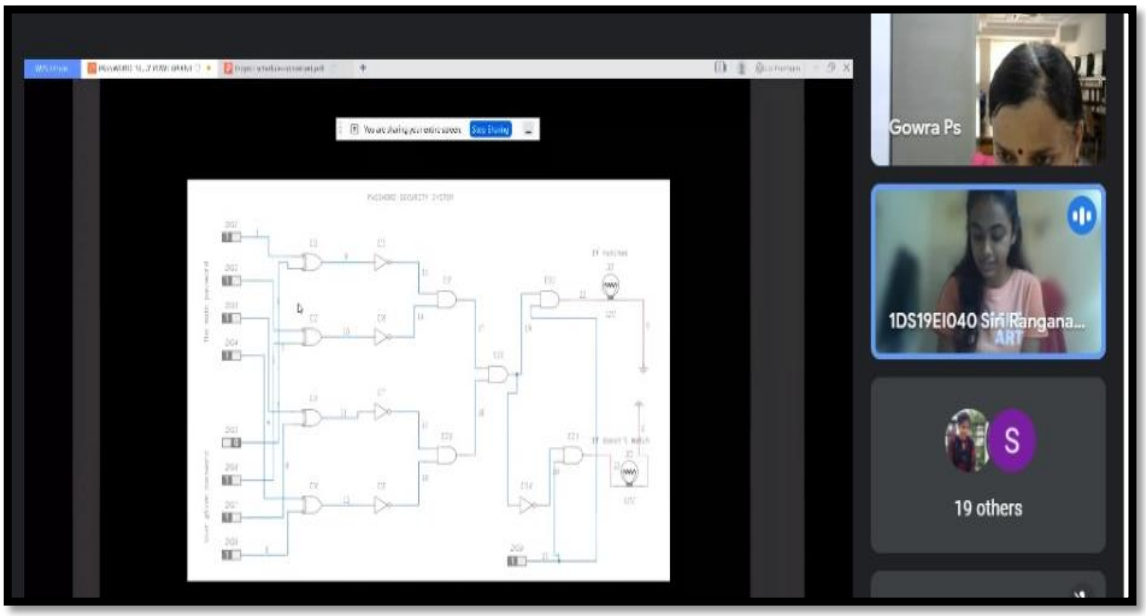
The session was very interactive as well as informative.

PHOTO GALLERY OF THE PROJECT PRESENTATION

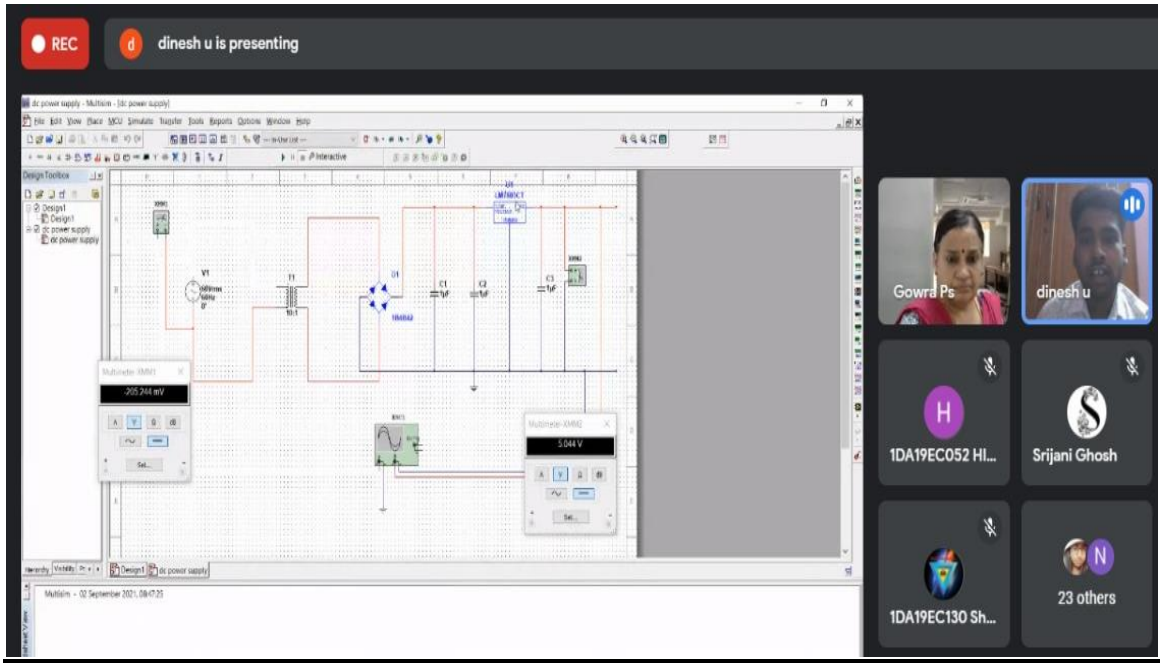
- **Function generator:**



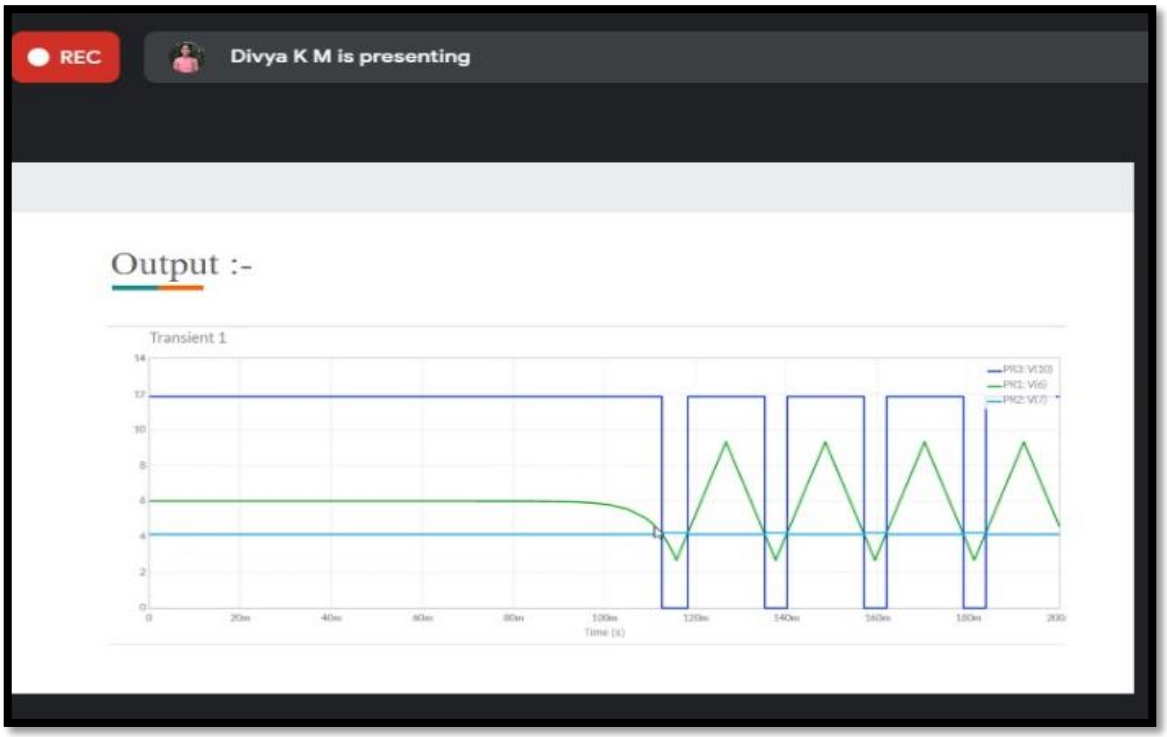
- **Password security instrumentation amplifier:**



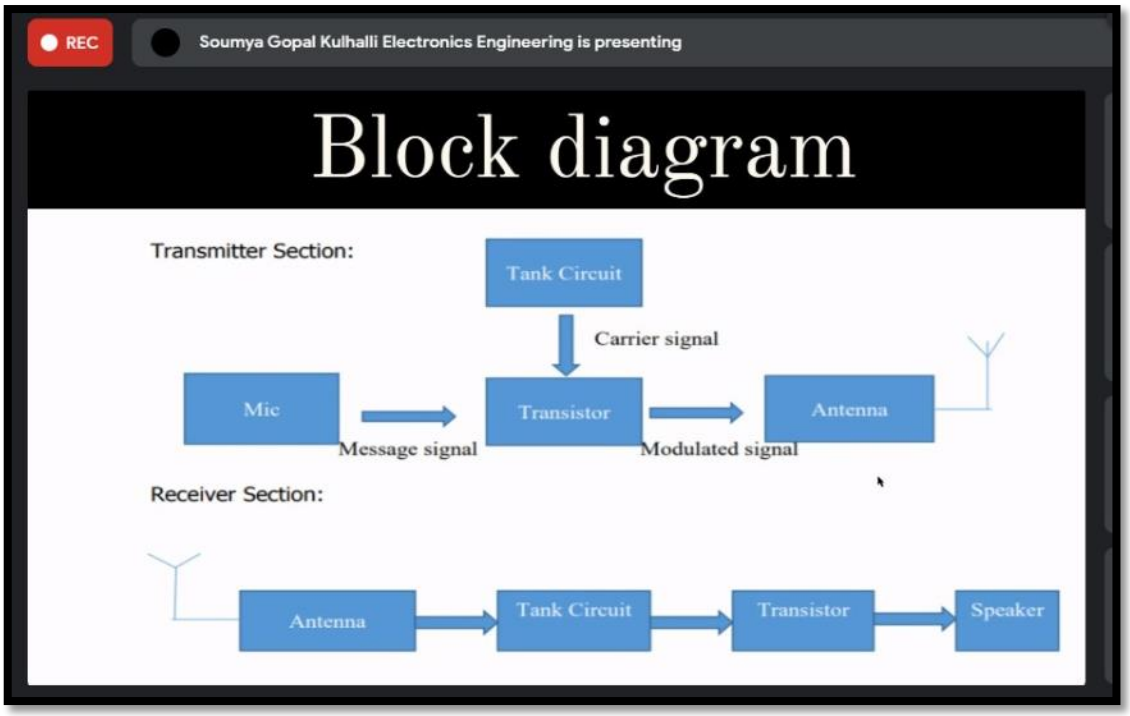
- **DC regulated power supply:**



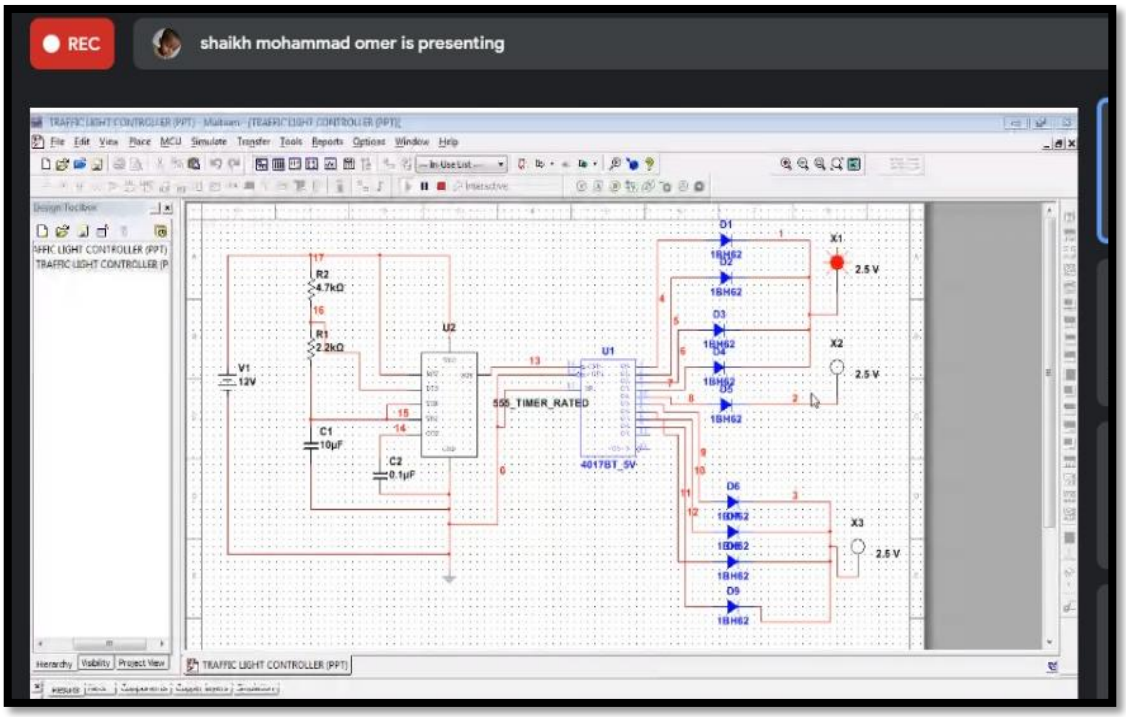
- **Pulse width modulation:**



- **FM listening bugger circuit:**



- **Traffic light controller with timers and counters:**



ATTENDANCE LIST

SL.NO	NAME	23-Aug	24-Aug	25-Aug	26-Aug	27-Aug	30-Aug	31-Aug	01-Sep	02-Sep	03-Sep	04-Sep	Eligib	Percentage
1	Abdurrahim sheik	AB	P	AB	P	P	P	P	P	P	P	P	YES	82%
2	Abhijith k	P	P	P	AB	AB	AB	AB	AB	AB	AB	AB	NO	28%
3	Adrian P Isaac	P	P	P	P	P	P	P	P	P	P	P	YES	100%
4	Avyukt Sharma	P	P	P	P	P	P	AB	AB	AB	P	P		73%
5	B R SRIRANGARAJU	AB	P	P	P	P	AB	AB	AB	P	AB	AB	NO	45%
6	BASAVARAJ PUTHANI	P	P	P	P	AB	P	P	P	P	P	P	YES	91%
7	Devayya	P	AB	P	P	AB	P	P	P	P	P	P	YES	82%
8	Dinesh U	P	AB	AB	AB	P	P	P	P	P	P	P		73%
9	Divya K M	P	P	P	P	P	P	P	P	P	P	P	YES	100%
10	Gousia	P	P	P	P	P	P	P	P	P	P	P	YES	100%
11	Hema R S	P	P	P	P	P	P	P	P	P	P	P	YES	100%
12	Hitha Shree Y S	P	P	P	P	P	AB	P	P	P	P	P	YES	91%
13	Likhith P Reddy	P	P	P	P	P	AB	P	P	P	P	P	YES	91%
14	Maheshwari M Birapure	AB	AB	AB	P	AB	AB	AB	AB	P	P	P	NO	36%
15	Manik	AB	P	P	P	P	AB	P	P	P	P	P	YES	82%
16	Manjiri Kulkarni	P	P	P	P	AB	AB	AB	AB	AB	AB	AB	NO	36%
17	N Suraj Naik	AB	AB	P	P	AB	AB	AB	AB	AB	AB	AB	NO	18%
18	Nandan.N	P	P	AB	AB	AB	P	P	P	AB	AB	AB	NO	46%
19	Nanditha A	P	P	P	P	P	P	P	P	P	P	P	YES	100%
20	NEHA G	P	P	P	P	AB	P	P	P	P	P	P	YES	91%
21	Nimisha MR	P	P	P	P	P	AB	P	P	P	P	P	YES	91%
22	Nishat khanam	P	AB	P	P	P	P	P	P	P	P	P	YES	91%
23	Prashun kumar Dash	P	P	P	P	P	P	P	P	P	P	P	YES	100%
24	Rashmi T	P	P	P	P	AB	P	P	P	P	P	P	YES	91%
25	Sahana KS	P	P	P	P	P	P	P	P	P	P	P	YES	100%
26	Sahana.A.N.	P	P	P	P	P	P	P	P	P	P	P	YES	100%
27	Sandeep T	P	P	P	P	P	AB	P	P	P	P	P	YES	91%
28	Shashidhar S	AB	P	P	P	P	P	P	P	P	P	P	YES	91%
29	Shirisha B N	P	P	P	P	AB	AB	AB	AB	P	P	P		64%
31	Shreya. C	P	P	P	P	P	P	AB	P	P	P	P	YES	91%
33	Sneha G R	AB	P	P	P	P	AB	P	P	P	P	P	YES	82%
34	Soumya Kulhali	P	P	P	P	P	P	P	P	P	P	P	YES	100%
35	Spoorthy.H.G	P	P	P	P	P	P	P	P	P	P	P	YES	100%
36	Spoorti oppari	P	P	P	P	P	P	P	P	P	P	P	YES	100%
37	Srijani ghosh	P	P	P	P	P	P	P	P	P	P	P	YES	100%
38	Suparna bk	P	P	P	P	P	AB	AB	AB	P	P	P		73%
39	Umme Salma Ahmed	P	AB	P	P	AB	P	P	P	P	P	P	YES	82%
40	ANUSHA TA	AB	P	P	P	P	P	P	P	P	P	P	YES	91%
41	Siri Ranganath	AB	P	P	P	P	P	P	AB	P	P	P	YES	82%
42	MD ABU SUFIYAN	AB	AB	P	P	P	P	P	AB	P	P	P		73%
42	PALLAVI HD	-	P	P	P	P	AB	P	AB	P	P	P	YES	80%

PERMISSION LETTER

8/16/2021 Two week online Internship program - to_office@bmsce.ac.in - BMS College of Engineering Mail

Hod Tce 2:43 PM (4 m
to Principal, Viceprincipal, Faculties TCE, me

Dear Principal and Viceprincipal Sir

Pleased to inform that Three of our faculty:

P S Gowra
Ambika
Archana

would like to offer a Two Week online Internship program for students from 2nd and 3rd year students from any college, including our college. The event brochure is attached.

We plan to charge a nominal amount of Rs 300/- towards the Internship

We would like to share 30 % of the amount collected with college, and use the 70% towards department IRG (to be used for funding procurement of academic accessories for online teaching or any other expenditure)

The faculty would not like to claim any honorarium for this activity.

Thanking you
with regards
B Kanmani

B Kan
16/8/21

Approved
Alida
16/8/21

STUDENT DETAILS

Sl.No	Name	College Name
		List Of Students attending Multisim Internship
1	Abdurrahim sheik	St.Joseph College of Engineering
2	Abhijith k	BMS College of Engineering
3	Adrian P Isaac	University Visvesvaraya College of Engineering
4	Avyukt Sharma	BMS College of Engineering
5	B R SRIRANGARAJU	Atria Institute of Technology
6	BASAVARAJ PUTHANI	University Visvesvaraya College of Engineering
7	Devayya	Dr.Ambedkar Institute of Technology
8	Dinesh u	Govt. Sri Krishnarajendra Silver Jubilee Technology Institute
9	Divya K M	BMS College of Engineering
10	Gousia	PDA college of Engineering kalaburgi
11	Hema R S	Dr.Ambedkar Institute of Technology
12	Hitha Shree Y S	Dr.Ambedkar Institute of Technology
13	Likhith P Reddy	BMS College of Engineering
14	Maheshwari M Birapure	PDA college of Engineering kalaburgi
15	Manik	University Visvesvaraya College of Engineering
16	Manjiri Kulkarni	Dayananda Sagar College Of Engineering
17	N Suraj Naik	Dayananda Sagar College Of Engineering
18	Nandan.N	Dayananda Sagar Academy of Technology and Management
19	Nanditha A	Dr.Ambedkar Institute of Technology
20	NEHA G	Dayananda Sagar College Of Engineering
21	Nimisha MR	BMS College of Engineering
22	Nishat khanam	Poojya Doddappa Appa College of Engineering Kalaburagi
23	Prashun kumar Dash	Dayananda Sagar College Of Engineering
24	Rashmi T	CMR Institute of Technology
25	Sahana KS	Dr Ambedkar Institute of Technology
26	Sahana.A.N.	Dr.Ambedkar Institute of Technology
27	Sandeep T	BMS College of Engineering
28	Shashidhar S	Dr.Ambedkar Institute of Technology
29	Shirisha B N	BMS College of Engineering
30	Shreya. C	Dr.Ambedkar Institute of Technology
31	Sneha G R	Dr.Ambedkar Institute of Technology
32	Soumya Gopal Kulhali	BMS College of Engineering
33	Spoorthy.H.G	Dr.Ambedkar Institute of Technology
34	SPOORTI OPPARI	SDM College of Engineering and Technology
35	Srijani ghosh	BMS College of Engineering
36	Suparna bk	Dr.Ambedkar Institute of Technology
37	Umme Salma Ahmed	Dayananda Sagar College Of Engineering
38	ANUSHA.TA	Dayananda Sagar College Of Engineering
39	Siri Ranganath	Dayananda Sagar college of Engineering
40	MD ABU SUFIYAN	PDA Engineering College,kalaburagi
41	PALLAVI HD	PDA College of Engineering kalaburgi

ACCOUNT DETAILS

College Name	Amount Received
BMS College of Engineering	Rs 200/-
Total Students : 9	**Rs 1800/-**

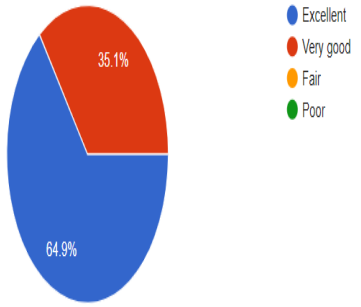
College Name	Amount Received
Other Colleges	Rs 300/-
Total Students : 32	**Rs 9600/-**

Total Amount Received: Rs 11,400/-
Revenue for College: Rs 3420/-
Revenue for Department: Rs 7980

FEEDBACK RESPONSES

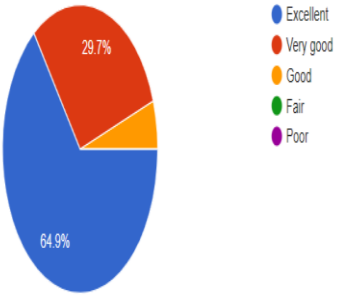
Speaker 1: Archana K -1) knowledge about the topic

37 responses



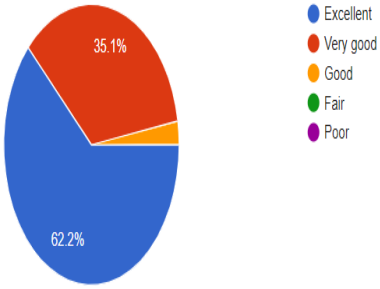
Speaker 1: Archana K- 2)Confidence level of teaching

37 responses



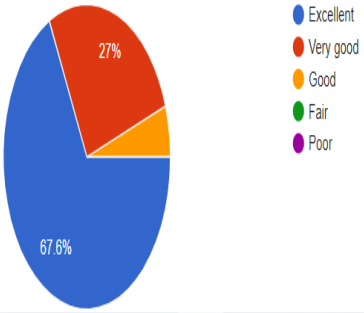
Speaker 1: Archana K- 3)Ability to make you understand the concepts?

37 responses



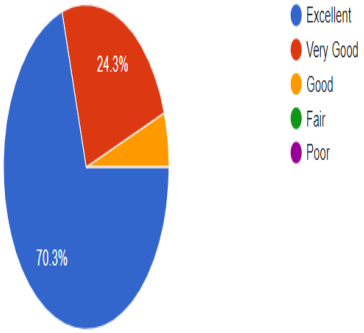
Speaker 2: Ambika K- 1)knowledge about the topic

37 responses



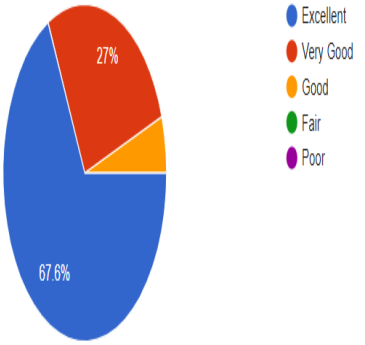
Speaker 2: Ambika K- 2)Confidence level of teaching

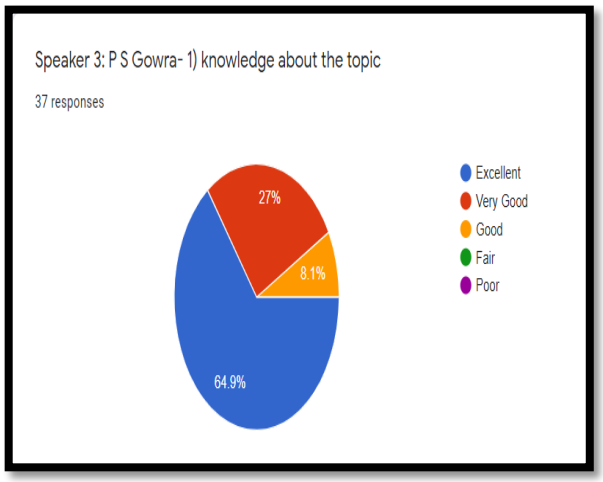
37 responses

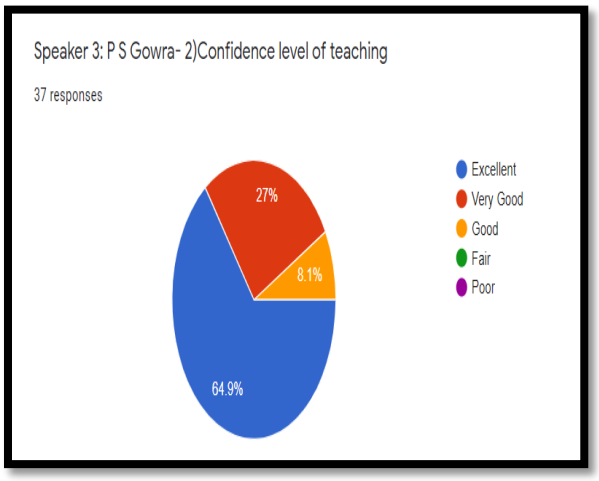


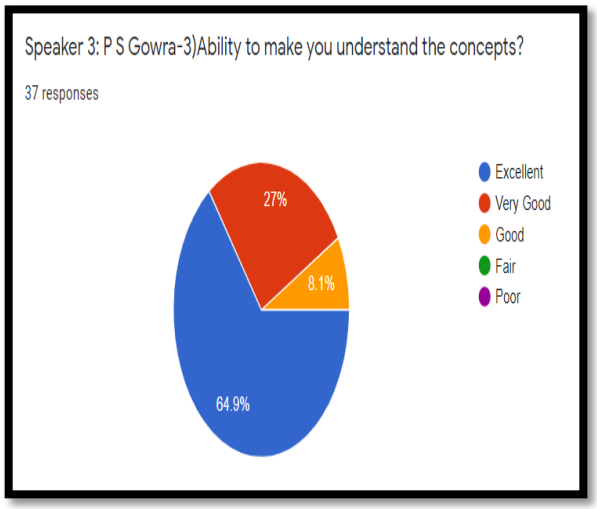
Speaker 2: Ambika K- 3)Ability to make you understand the concepts?

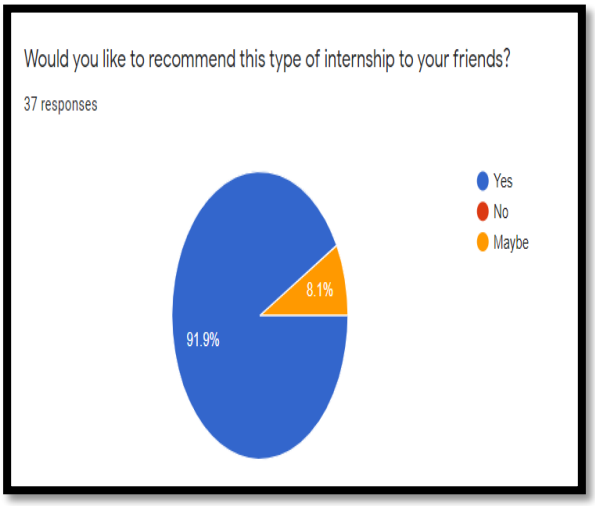
37 responses

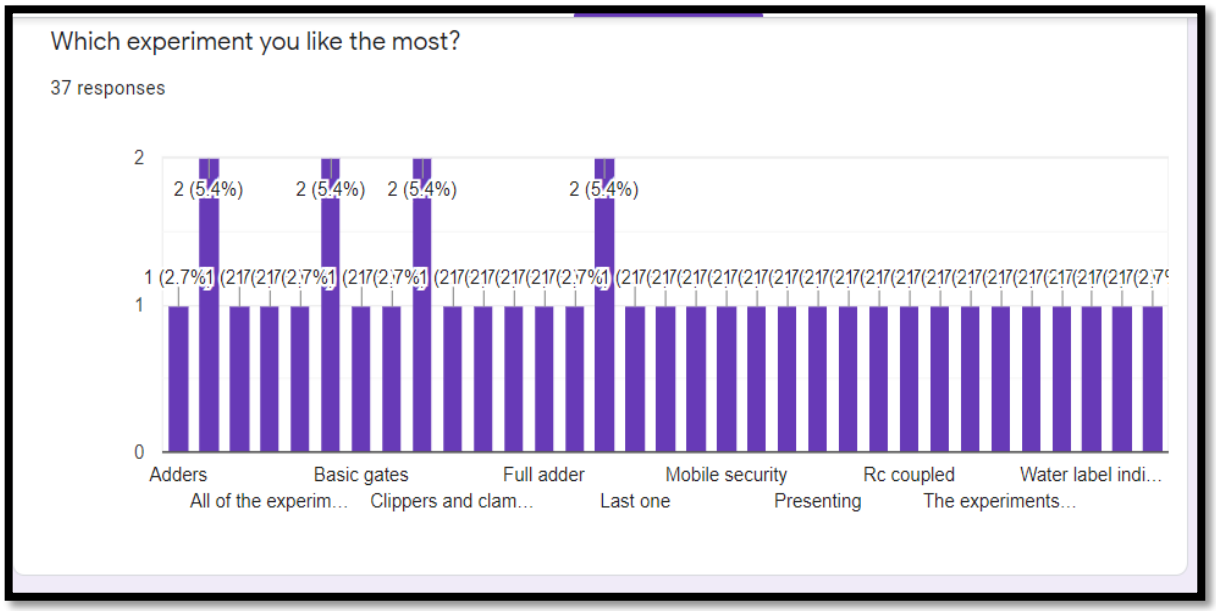



Speaker 3: P S Gowra- 1) knowledge about the topic
37 responses
27%
8.1%
64.9%
Excellent
Very Good
Good
Fair
Poor


Speaker 3: P S Gowra- 2)Confidence level of teaching
37 responses
27%
8.1%
64.9%
Excellent
Very Good
Good
Fair
Poor

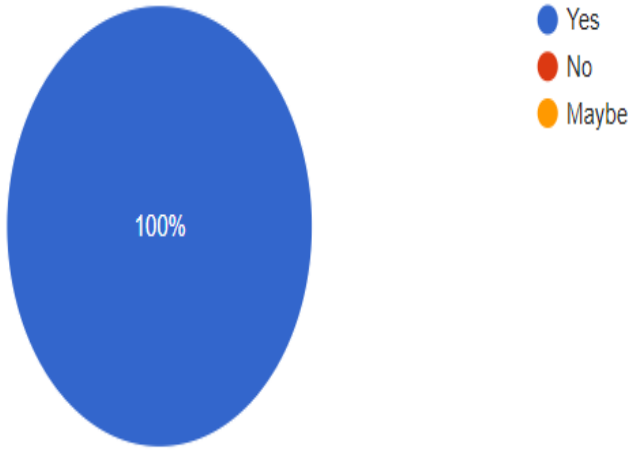

Speaker 3: P S Gowra-3)Ability to make you understand the concepts?
37 responses
27%
8.1%
64.9%
Excellent
Very Good
Good
Fair
Poor


Would you like to recommend this type of internship to your friends?
37 responses
8.1%
91.9%
Yes
No
Maybe


Which experiment you like the most?
37 responses
2
1
0
2 (5.4%)
2 (5.4%)
2 (5.4%)
2 (5.4%)
1 (2.7%)
Adders
Basic gates
Full adder
Mobile security
Rc coupled
Water label indi...
All of the experim...
Clippers and clam...
Last one
Presenting
The experiments...

Was the project presentation helpful in your learning

37 responses



Overall feedback about program

37 responses