Index

S. No.	Name of the Experiment	Page No.	Date of Experiment	Date of Submission	Remarks
1	Introduction to CRO, DSO and function generator.	1-4			
2.	To generate various signals using function generator and measure amplitude and frequency of these signals using oscilloscope	5-11			
3.	To study and test various active and passive electronic components.	12-14			
4.	To measure voltage and current of series RLC circuit.	15-18			
5	To plot V-I characteristics of p-n junction diode in forward bias.	19-24			
6	To plot V-I characteristics of zener diode in reverse bias	25-31			
7	To measure input and output waveform of half wave rectifier.	32-34			
8	To study input and output waveform of ~~full~~ half wave rectifier. with capacitor	35-37			
9	To study input and output waveform of full wave rectifier.	38-40			

Index

Date

Expt. No. Page No. 1

EXPERIMENT-1

Objective - Introduction to CRO, DSO, and function generator.

Apparatus- CRO, DSO, power supply, AC frequency counter, function generator.

Theory-

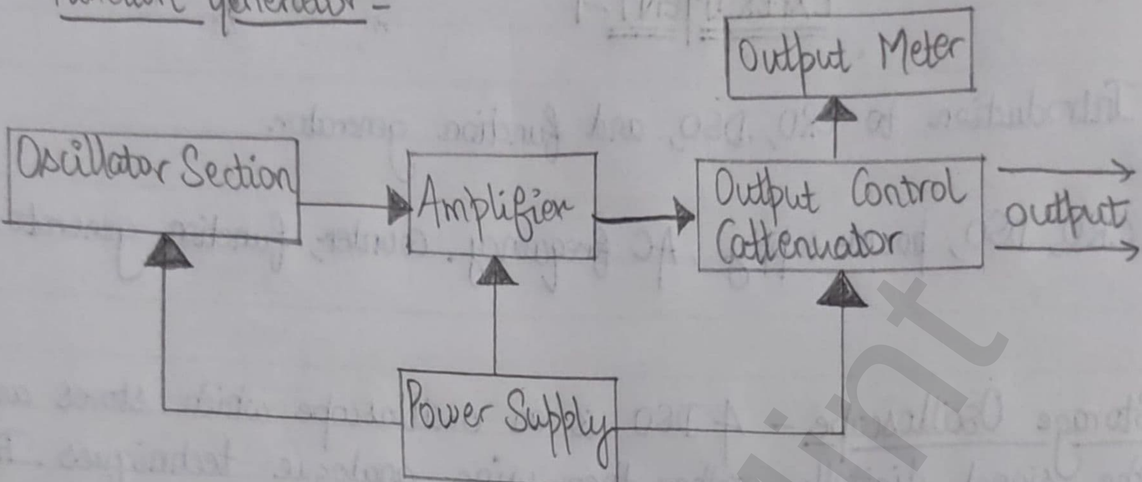
① **Digital Storage Oscilloscope** - A DSO is an oscilloscope which stores and analyses the signal digitally rather than using analogue techniques. It is the most common type of oscilloscope in use because of advanced trigger. The input analogue signal is sampled and then converted into a digital record of amplitude of the signal at each sample time. The sampling frequency should not be less than nyquist rate to avoid aliasing. The principal advantage over analog storage is that the stored traces are as bright as sharply defined and written as quickly as non-stored traces.

② **Cathode Ray Oscilloscope** - are used to observe changes of electrical signal over time, such that voltage and time describe a shape which is continuously graphed against a caliberated scale. The observed waveform can be analysed for such properties as amplitude, frequency, rise time, time interval distribution. Modern digital instruments may calculate and display these properties directly. Calculation of these values require manually measuring waveform against scales built into screen of the instrument.

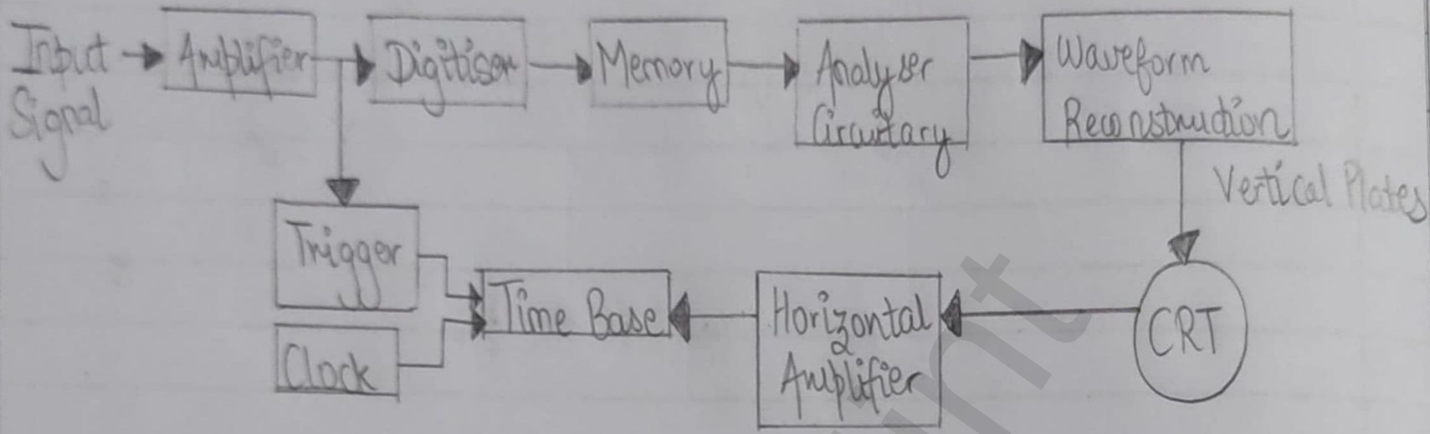
③ **Function Generator** - is an electronic test equipment/software used to generate different types of electrical waveform over a wide range of frequencies. Some of the most common waveforms produced by the function generator are sine, square, triangular. These waveforms can be repetitive

Teacher's Signature

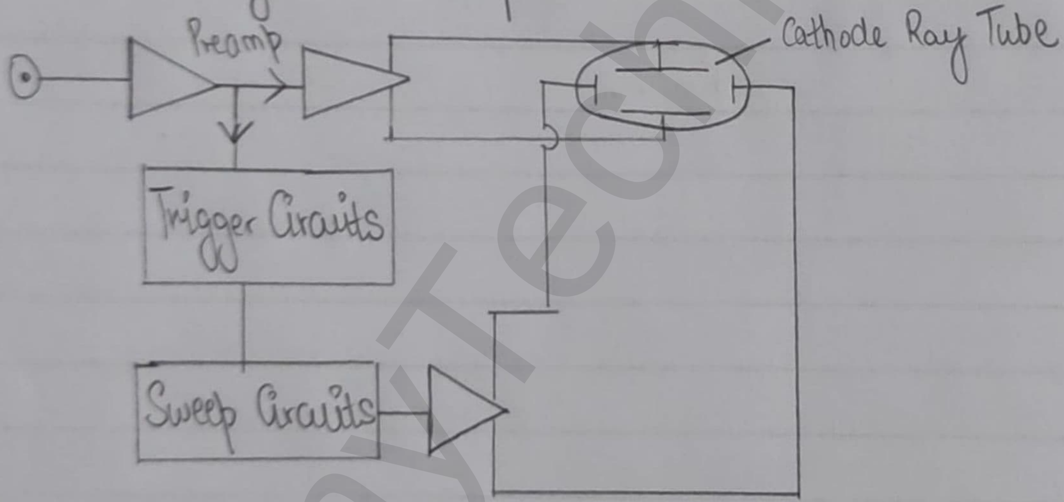
- Function Generator -



Block Diagram - • Digital Storage Oscilloscope



• Cathode Ray Oscilloscope



Date

Expt. No.

or single shot. Although, function generator covers both audio and RF frequencies, but are not suitable for application that need low distortion or stable frequencies.

Result - Electronic Devices like CRO, DSO, frequency generator are studied successfully.

Teacher's Signature

Date ____
Expt. No. ____

EXPERIMENT - 2.

Objective - To generate various signals using function generator and measure amplitude and frequency of these signals using oscilloscope.

Software Used - Multisim

Theory - Oscilloscope is defined as an electronic device that is used to produce visual display corresponding to electric signals. The X-axis represents time period and y-axis represents amplitude. The frequency is calculated by function generator which can generate sinusoidal, triangular and square waveforms with adjustable frequency and amplitude.

Procedure -

1. Connect the circuit.
2. For efficient values of amplitude and frequency on function generator, plot signals and display on oscilloscope.
3. Repeat for triangular and square waveforms.

Precautions -

1. Make proper circuit.
2. Function Generator should be given accurate value.

Result -

All the waveforms are obtained

Sinusoidal waveform -

Triangular Waveform -

Square Waveform -

Teacher's Signature ____

Observation-

- Sinusoidal Waveform

(1) Value on function generator

Frequency - 50 Hz

Amp - 10 Vp

Observation on oscilloscope:

x-axis (Time) scale = 10 ms / Div

y-axis (Amp) Scale = 5 V / Div

Amplitude = 10 (2 scale)

$\text{frequency} = \frac{1}{\text{Time}} = \frac{1}{20 \times 10^{-3}} = \boxed{50\ Hz}$

(2) Value of function generator:

Frequency = 25 Hz

Amp = 5 Vp

Observation on oscilloscope

X axis (Time) Scale = 10 ms/Div

y axis (Time) Scale = 5 v/Div

Amplitude = 5 (1 scale)

$\text{frequency} = \frac{1}{\text{Time}} = \frac{1}{4 \times 10 \times 10^{-3}}$ sec

$\Rightarrow \boxed{25\ Hz}$

- Triangular Waveform

(3) Value on function generator

Amplitude = 5

Frequency = 25 Hz

Observation on Oscilloscope:

X-axis (Time) scale = 10 ms / Div

Y-axis (Amp) scale = 5 V / Div

Amplitude = 5 (1 scale)

$\text{Frequency} = \frac{1}{\text{Time}} = \frac{1}{4 \times 10 \times 10^{-3}}$

$\Rightarrow \boxed{25\ Hz}$

- Square Waveform

(4) Value on function generator

Amplitude = 10

Frequency = 50 Hz

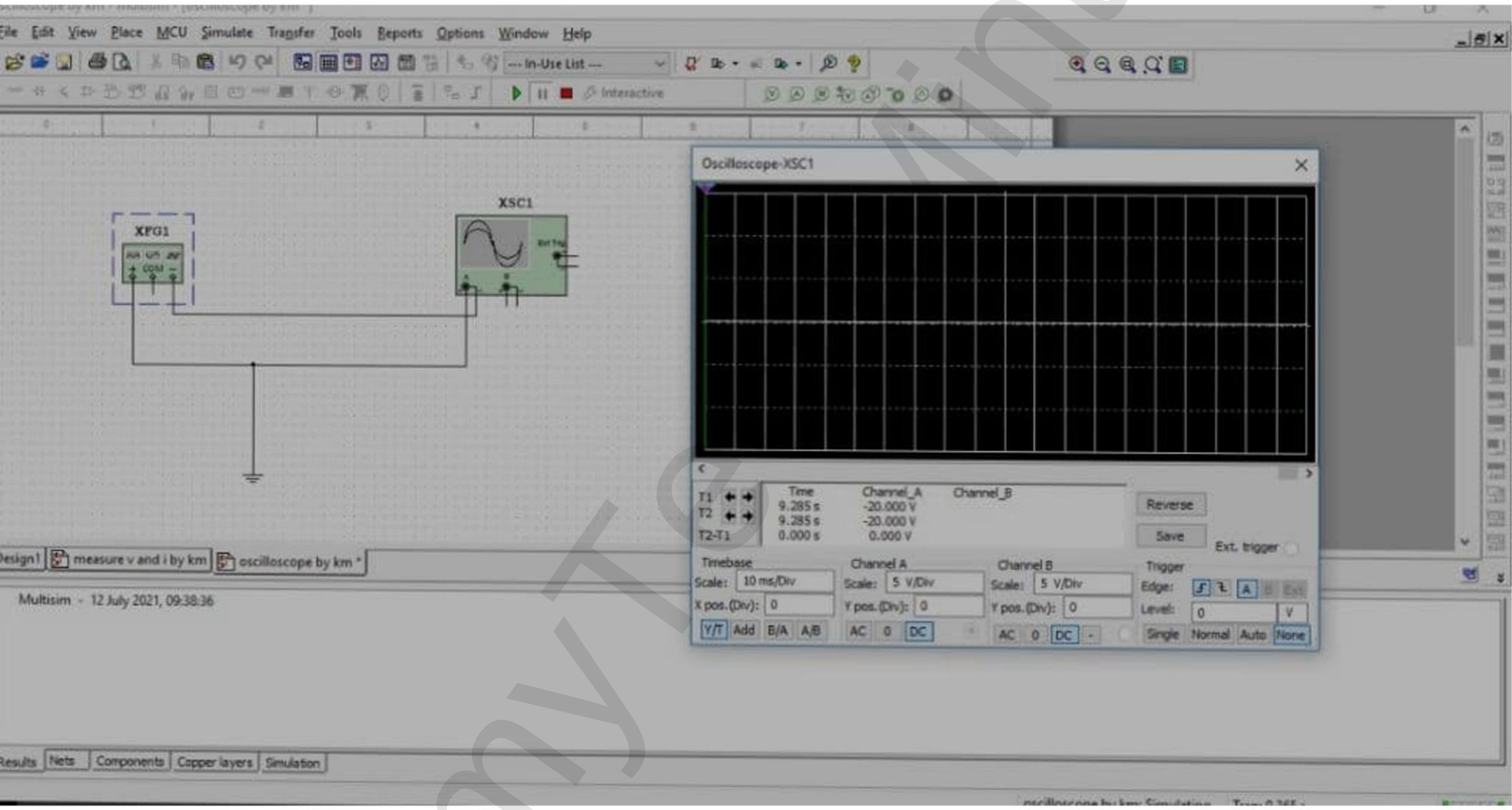
Observation of oscilloscope

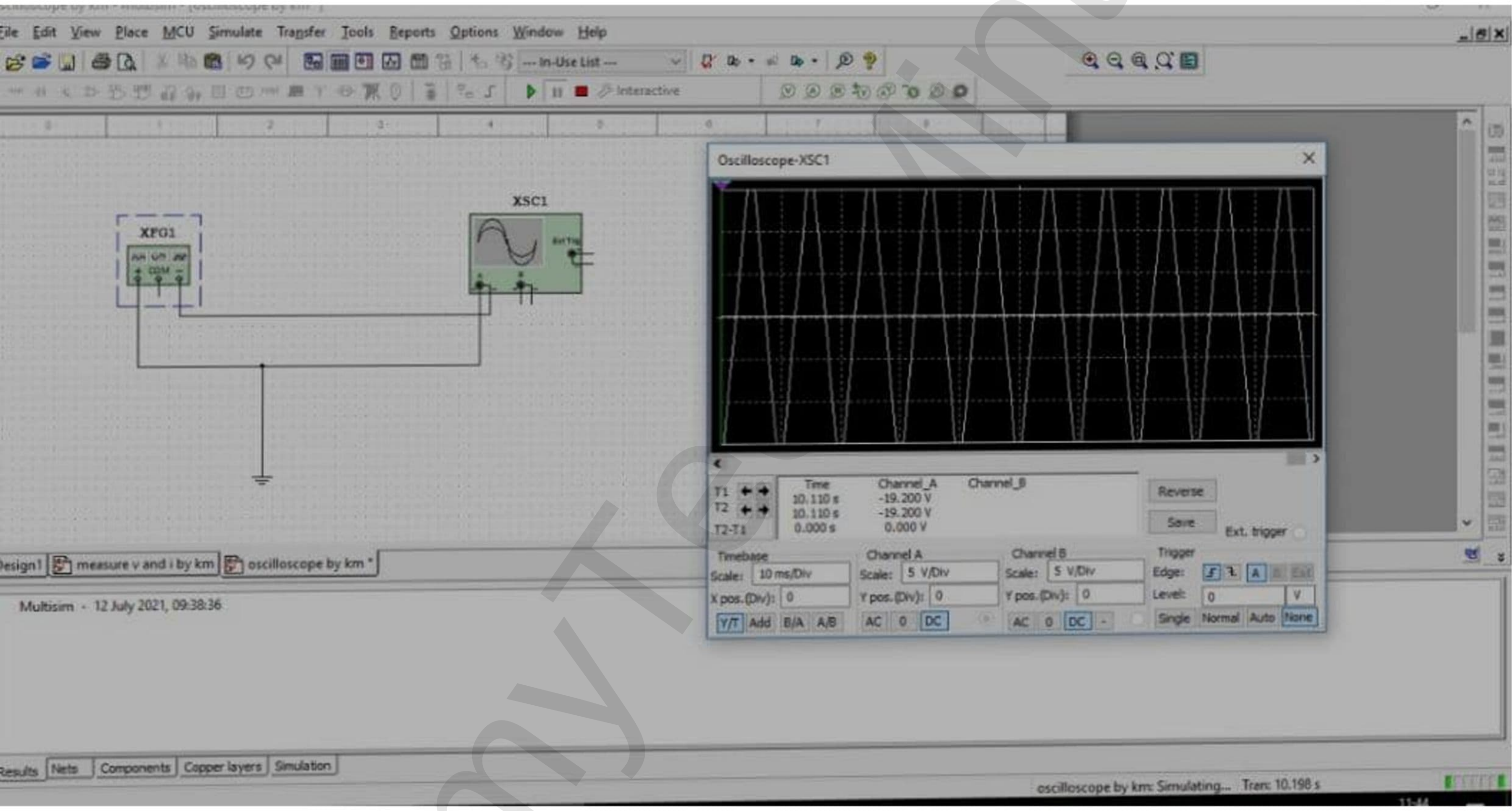
X axis (Time) = 10 ms / Div

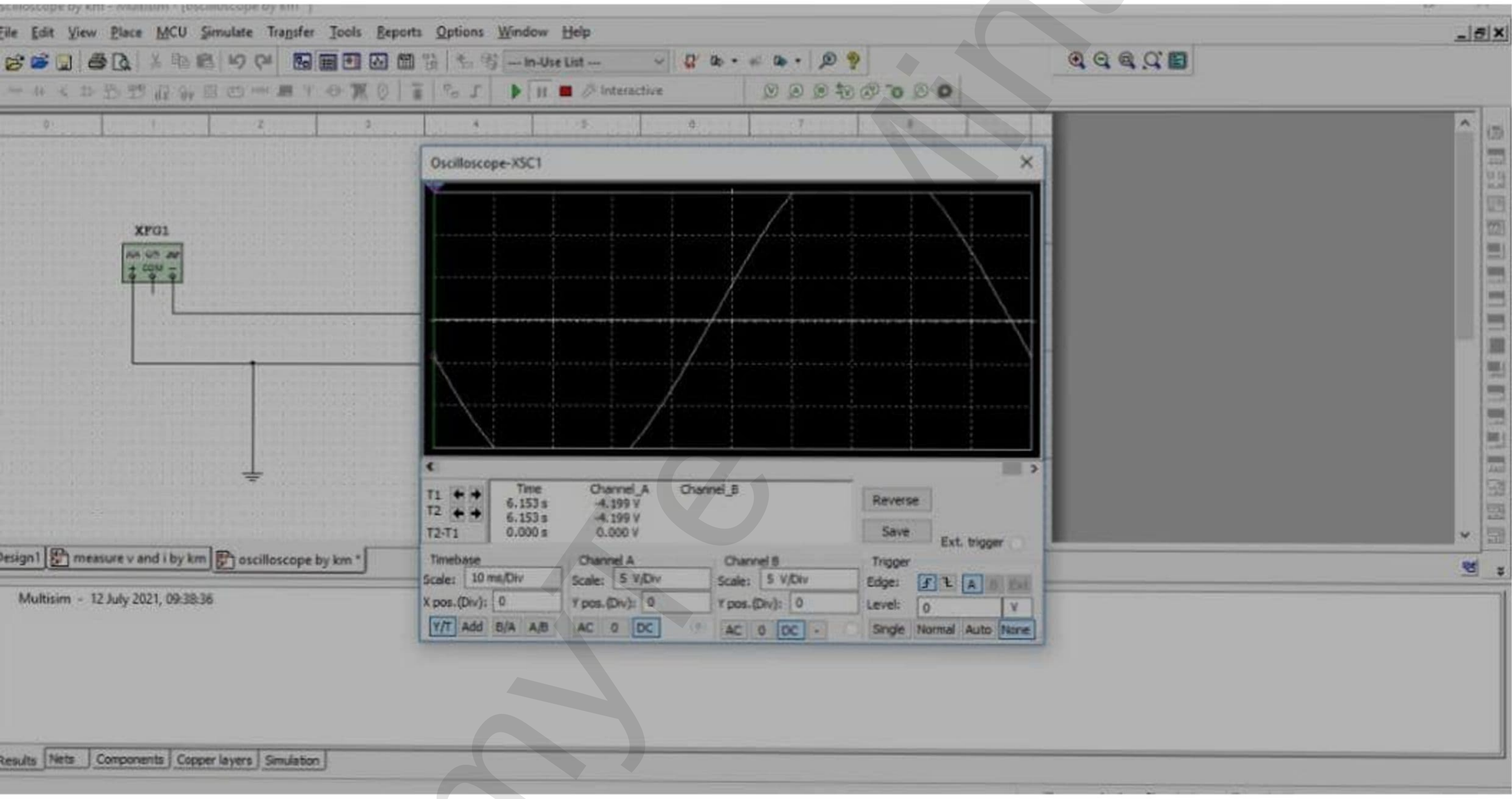
Y axis (Amp) = 5 V / Div

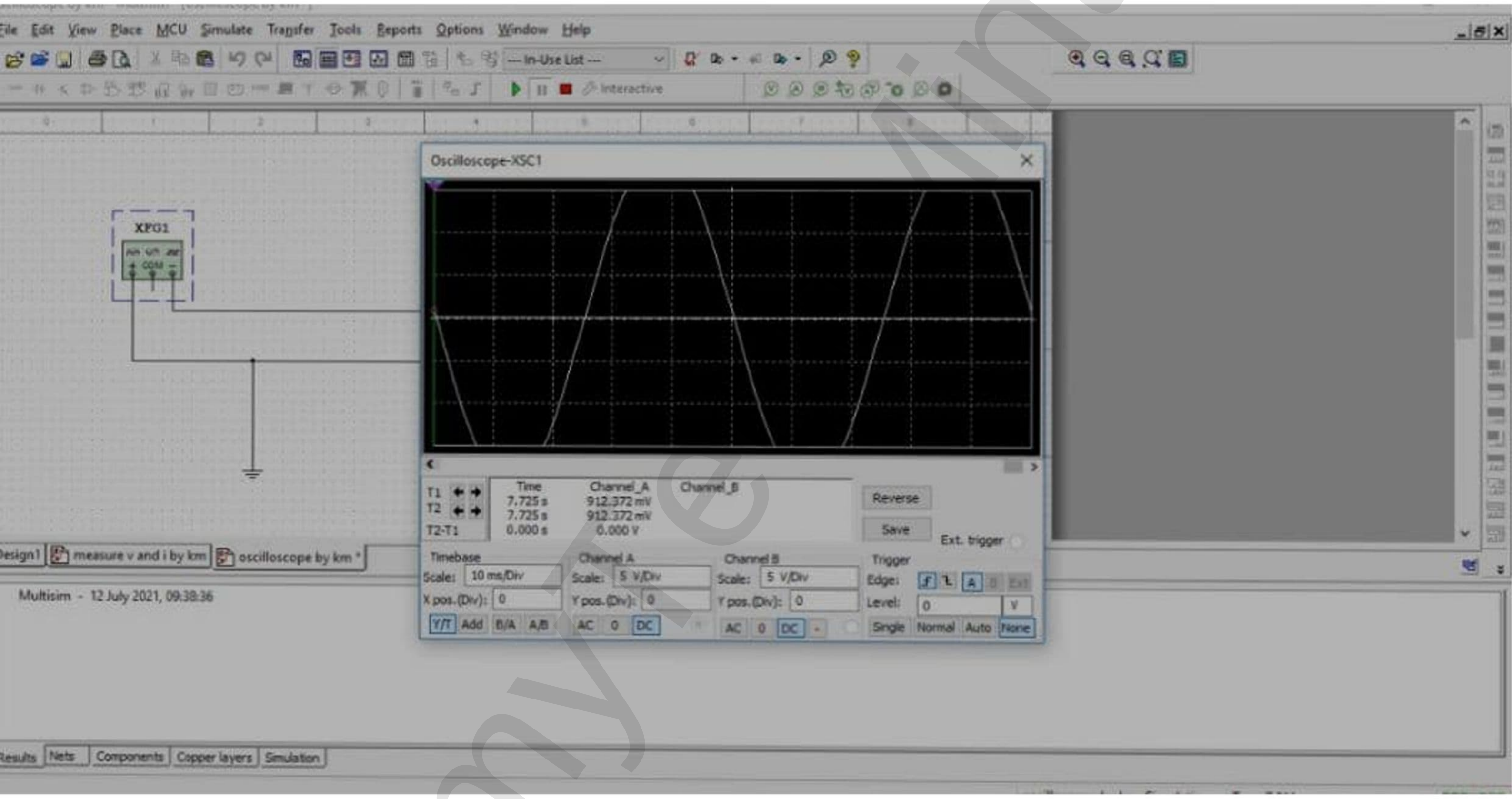
Amplitude = 10 (2 scale)

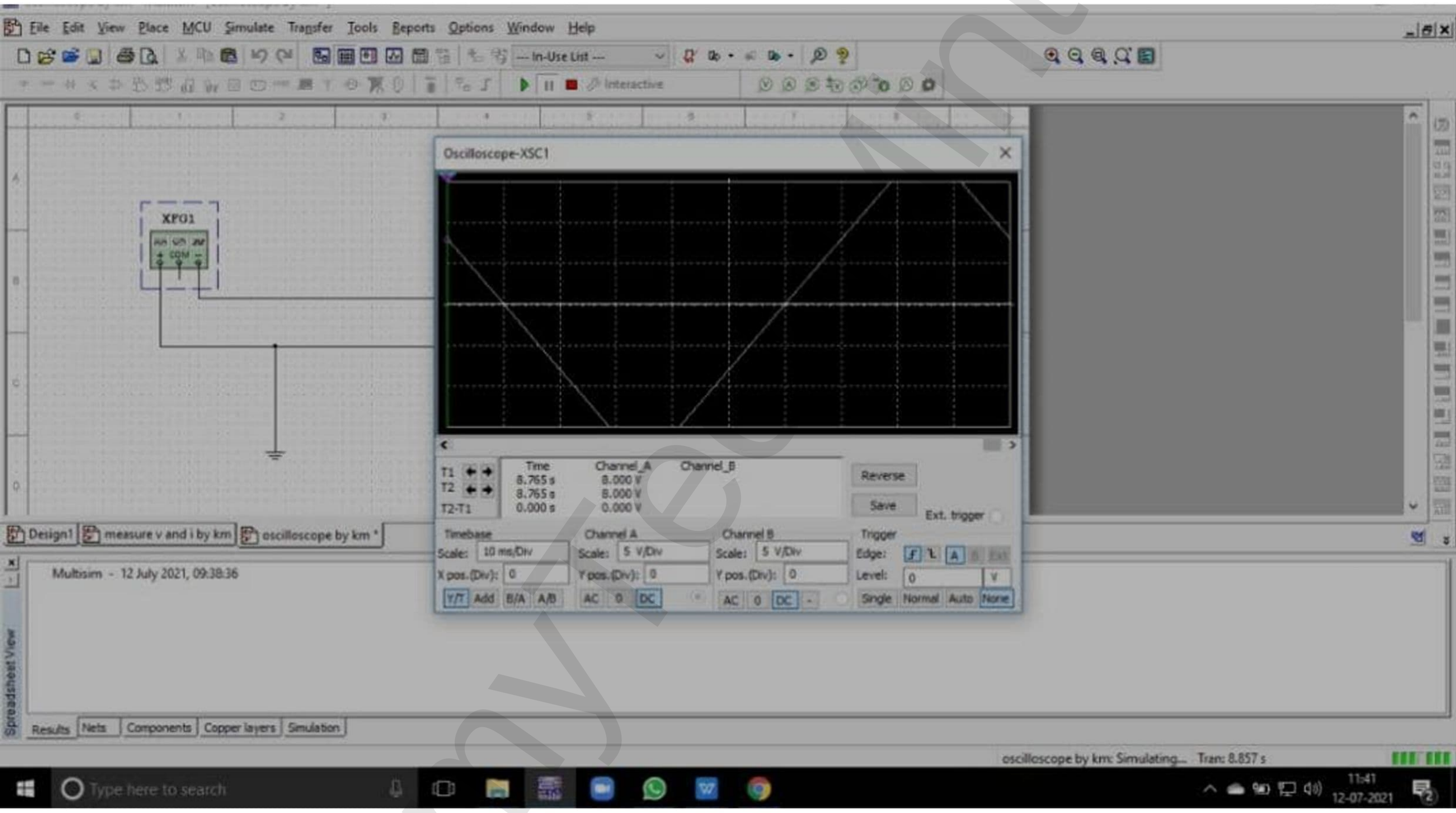
$\text{Freq} = \frac{1}{\text{Time}} \Rightarrow \frac{1}{20 \times 10^{-3}} \Rightarrow \boxed{50\ Hz}$


XFG1
XSC1
Oscilloscope-XSC1
Time
Channel_A
Channel_B
T1
T2
T2-T1
9.285 s
9.285 s
0.000 s
-20.000 V
-20.000 V
0.000 V
Reverse
Save
Ext. trigger
Timebase
Scale: 10 ms/Div
X pos.(Div): 0
Y/T Add B/A A/B
Channel A
Scale: 5 V/Div
Y pos.(Div): 0
AC 0 DC
Channel B
Scale: 5 V/Div
Y pos.(Div): 0
AC 0 DC -
Trigger
Edge:
Level: 0 V
Single Normal Auto None
Multisim - 12 July 2021, 09:38:36
measure v and i by km
oscilloscope by km
Results Nets Components Copper layers Simulation


Oscilloscope-XSC1
XFG1
XSC1
Time
Channel_A
Channel_B
10.110 s
-19.200 V
10.110 s
-19.200 V
0.000 s
0.000 V
Reverse
Save
Ext. trigger
Timebase
Scale: 10 ms/Div
X pos.(Div): 0
Channel A
Scale: 5 V/Div
Y pos.(Div): 0
Channel B
Scale: 5 V/Div
Y pos.(Div): 0
Trigger
Edge:
Level: 0 V
Y/T Add B/A A/B
AC 0 DC
Single Normal Auto None
Multisim - 12 July 2021, 09:38:36
oscilloscope by km: Simulating... Tran: 10.198 s




File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help
Oscilloscope-XSC1
XFG1
Reverse
Save
Ext. trigger
Timebase
Channel A
Channel B
Trigger
Multisim - 12 July 2021, 09:38:36


Oscilloscope-XSC1
Reverse
Save
Ext. trigger
Timebase
Channel A
Channel B
Trigger
Multisim - 12 July 2021, 09:38:36

or single shot. Although, function generator covers both audio and RF frequencies, but are not suitable for application that need low distortion or stable frequencies.

Result - Electronic Devices like CRO, DSO, frequency generator are studied successfully.

EXPERIMENT-3

Aim - To study and test various active and passive electronic components.

Objectives -

1. To get familiar with basic electronic components such as resistor, capacitor, diodes, transistors, LED's.
2. To test and understand the function of various electronic components.

Theory -

1. **Resistors** -

A resistor is a passive electrical component with 2 terminals that are used for either limiting or regulating the flow of electric current in electrical circuits. The main purpose of resistor is to reduce the current flow and to lower the voltage in any particular portion of the circuit. It is made up of copper wires which is coiled around a ceramic rod and the outer part of the resistor is coated with an insulated paint. It is used to step up or lower the voltage signal or vice-versa. The resistors are sensitive to heat, light or other variables.

2. **Capacitors** -

A capacitor is a passive 2-terminal electrical device that possess the ability to store energy in the form of electrical charge. It consists of 2 electrical conductors that are separated by a distance. It stores energy by holding apart pairs of opposite charges. The practical capacitors ~~contain~~ has ability to store charges known as capacitance. Some capacitors have their values printed on them, some are marked with 3-digit codes and few are coloured coded.

Date ____

Expt. No. ____ Page No. 5

③ Diodes-
A diode is a two terminal electronic component that conducts electricity in one direction. It has high resistance in one end and low resistance on the other end. It is used to protect circuits by limiting the voltage and transform AC to DC. Semiconductors [Silicon, Germanium] used to make most of the diodes.

④ Transistors-
A transistor is a type of semiconductor device that can be used to both conduct and insulate electric current or voltage. It is composed of three layers of semiconductor materials. It acts as as switch and an amplifier. Voltage & Current is applied to one pair of terminals of the transistors. Some transistors are packaged individually but many more are found embedded in integrated circuits.

Methods to determine value of the components-

- Diode- measure forward and reverse bias voltages of given diodes and record them.

(i) Forward Bias Voltage- place the Red probe on the anode. Touch the black probe on the cathode and record the reading

(ii) Reverse Bias Voltage- place Red probe on the cathode. Touch the black probe to the anode and record the reading.

- Transistors- Both NPN and PNP transistor looks similar in physical appearance. They can get differentiated by-
Connect the positive lead of your ohm-meter to the base. Touch the other end of your meter to the collector. If we get a reading, transistor is

Teacher's Signature ____

NPN.

If meter reads open-circuit, then connect the negative leads to the base and touch the positive lead to the collector. If we get the reading, then the transistor is PNP.

Teacher's Signature

Expt. No.

EXPERIMENT - 4

Objective - To measure voltage and current ~~in~~ of series RLC circuit.

Software Used - Multisim

Theory - RLC Circuit is an electrical circuit consisting of a Resistor (R), Inductor (L) and Capacitor (C), connected in series/parallel.

Series RLC Circuit - The three components are all in series with voltage source

Procedure -

1. Connect the circuit
2. Set the range on multimeter
3. Measure the voltage by connecting multimeter parallel to R, L and C.
4. Connect multimeter in series to measure current.

Result -

The voltage and current, measured when

	Voltage	Current
R is connected =	43.5V	5.32A
L is connected =	182.12V	2.49A
C is connected =	92.3V	8.3A
Across circuit →	100.055V	2.9A

where, $R = 15\Omega$, $L = 0.2H$, $C = 100\mu F$

Precautions -

1. Make proper circuit.
2. Measure voltage and current carefully.

Teacher's Signature

Observation -

① Value of frequency on function generator = 100 Hz.
Display on frequency counter = 100Hz

② Value of frequency on function generator = 50Hz
Display on frequency counter = 50 Hz.

	Observed	Calculated
Voltage across R	44.17V	43.5 V
Voltage across L	200.87V	182.12 V
Voltage across C	93.45V	92.3 V
Current in Circuit	3.01 V	2.9 V

Error in voltage -

Across R = 44.17 - 43.5 ⇒ 0.67V
Across L = 200.87 - 182.12 ⇒ 18.75V
Across C = 93.45 - 92.3 ⇒ 1.15V

Error in current ⇒ observed - calculated ⇒ 3.01 - 2.9 ⇒ 0.11V

Calculations -

Resistance R = 15Ω

Inductive Resistance, X_L

$X_L = 2\pi f L$ $\quad$ L = 20mH ⇒ 0.2H

$\Rightarrow 2\pi \times 50 \times 0.2 \Rightarrow \boxed{62.8\,\Omega}$

Capacitive Resistance, X_C

$X_C = \frac{1}{2\pi f C} = \frac{1}{2\times\pi\times 50\times 10^{-4}}$, $C = 100\,\mu F \Rightarrow 100\times 10^{-6} F \Rightarrow 10^{-4} F$

$\Rightarrow \boxed{31.83\,\Omega}$

Circuit Impedance (Z) = $\sqrt{R^2+(X_L-X_C)^2} \Rightarrow \sqrt{(15)^2+(62.8-31.83)^2}$

$\Rightarrow$ **34.41 Ω**

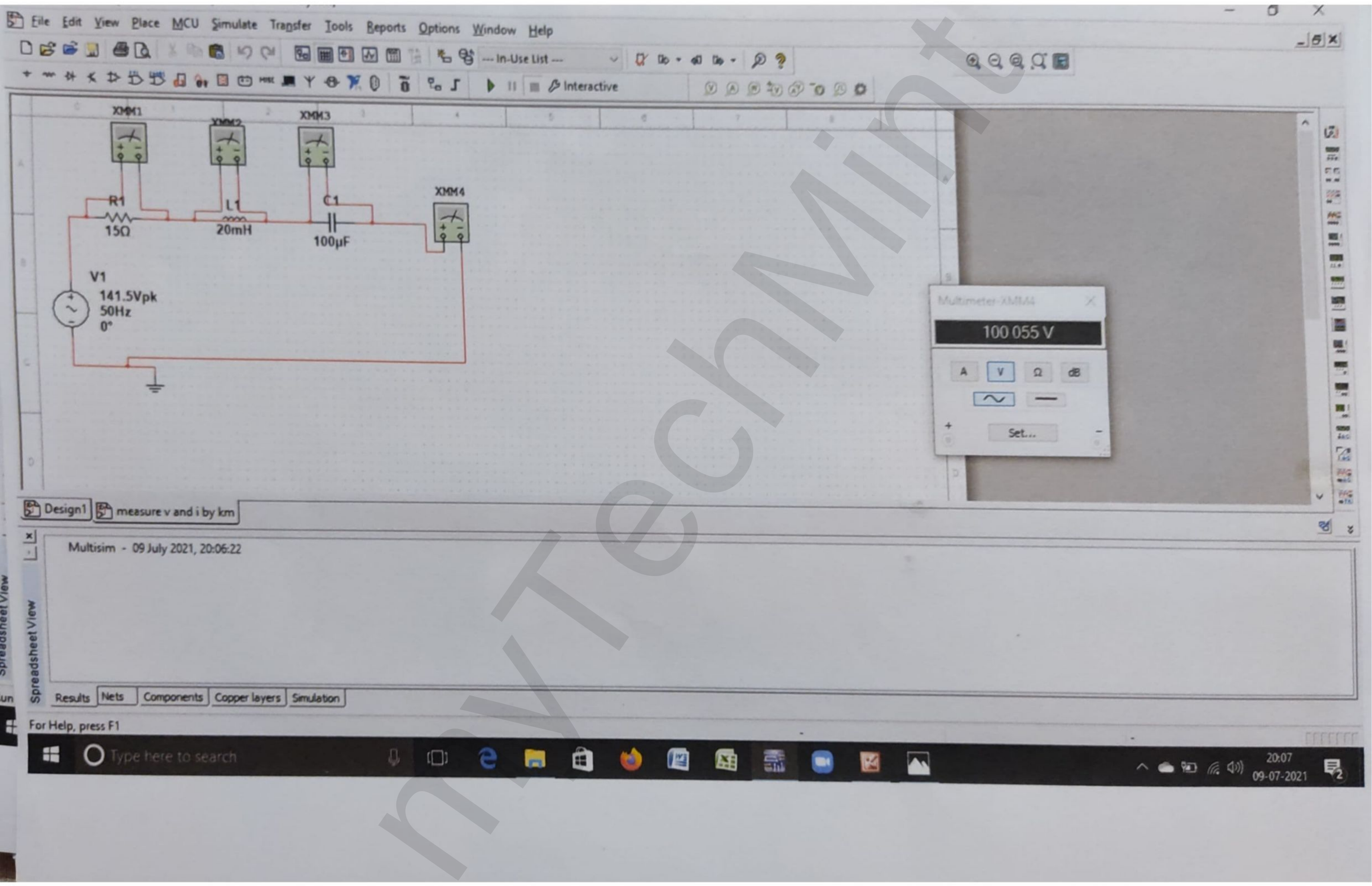
Voltage across RLC, $V_S = 100\ V$

$V_R = IR = 2.9 \times 15 \Rightarrow$ **43.5V**

$V_L = IL = 2.9 \times 62.8 \Rightarrow$ **182.12V**

$V_C = IC = 2.9 \times 31.83 \Rightarrow$ **92.3V**

Current across RLC Circuit $\Rightarrow \frac{V}{Z} \Rightarrow \frac{100}{34.41} \Rightarrow$ **2.9 A**



File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help
XMM1
XMM2
XMM3
XMM4
R1
15Ω
L1
20mH
C1
100µF
V1
141.5Vpk
50Hz
0°
100.055 V
Set...
Design1
measure v and i by km
Multisim - 09 July 2021, 20:06:22
Results
Nets
Components
Copper layers
Simulation
For Help, press F1
20:07
09-07-2021

EXPERIMENT-5

Aim- To plot the V-I characteristics of p-n junction diode in forward bias.

Software Used- Multisim

Theory- Under forward bias condition, when positive terminal of the battery is connected to P-type and negative terminal to the N-type of the PN junction diode.

The applied potential with external battery acts in opposition to the internal potential barrier and disturb the equilibrium. when equilibrium is disturbed, the fermi level is no longer continued across the junction.

Under the condition when the applied potential is more than the internal barrier potential, then depletion region and internal barrier potential disappears.

V-I characteristics of Diode under forward bias - As the forward bias is increased, for $V_f \leq V_0$, the forward current (I_F) is almost zero due to potential barrier prevents the holes from P-region and electrons from N-region to flow across the depletion region in the opposite direction.

For $V_f > V_0$, the potential barrier completely disappears, hence the holes cross the junction from P-type to N-type. This results in relatively large current flow in the external circuit.

At the cut in the voltage, the potential barrier is overcome and the current through the junction starts to increase rapidly.

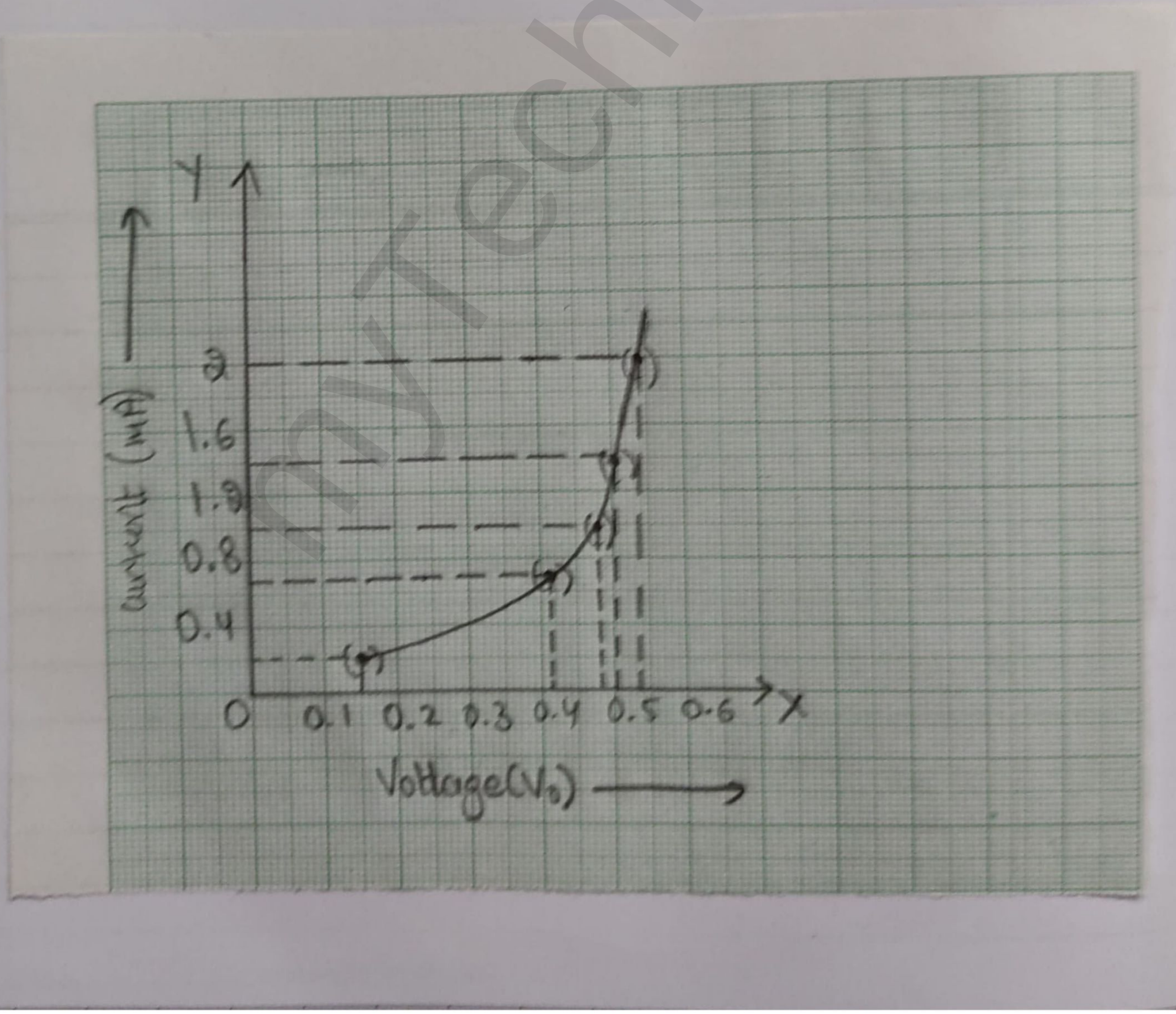
Procedure-

① **Method-1**

- By analysis and Stimulation.

Observation Table

SNo.	Voltage (V_i)	V_o (V)	I_o (mA)
1	0.2V	0.15V	0.7mA
2	0.7V	0.41V	0.7mA
3	1V	0.48V	1 mA
4	1.4V	0.50V	1.4mA
5	2V	0.53V	2mA



* Make proper connection with DC power ~~circuit~~ source, a diode and a resistor and then ground the circuit.
* Click on stimulate on the task bar of Multisim.
* Click on analysis and Stimulation, then on DC ~~swap~~ sweep adjust the voltage and add $I(R_1)$ to the DC operating circuit.
* Then save and Run.

Result-1- The V-I graph will be formed and will be displayed on the grapher view.

② Method 2-

- Traditional Multimeter Method.

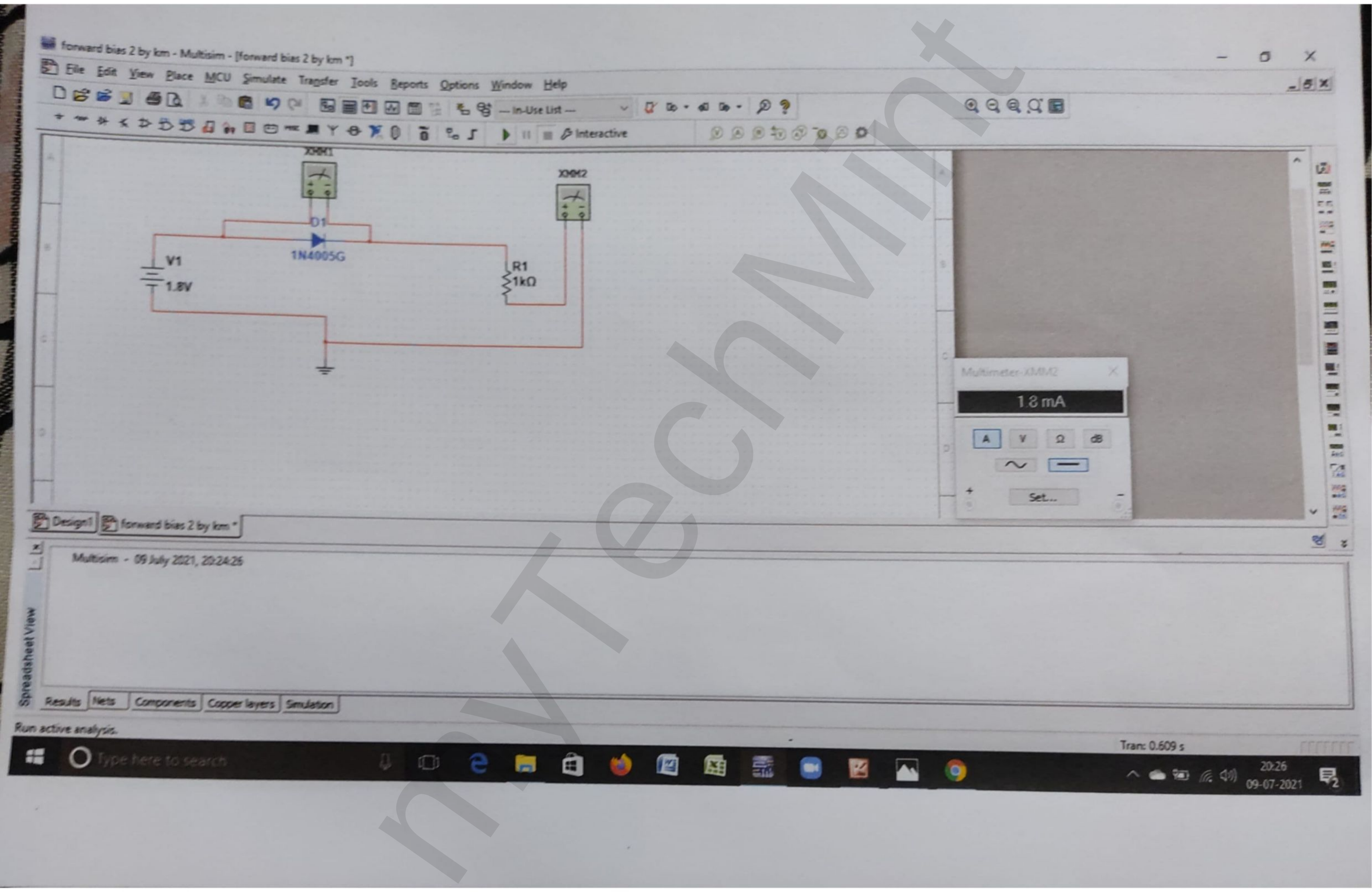
* Make a proper circuit with DC power source, diode, resistor and ground the circuit.
* Connect the multimeter parallel to the diode used.
* Use multimeter as an ammeter and connect with the resistor and the power source.
* Now run the simulation.
* Now take the readings by changing the voltage across the circuit.
* Repeat the step 5 till 4 readings.

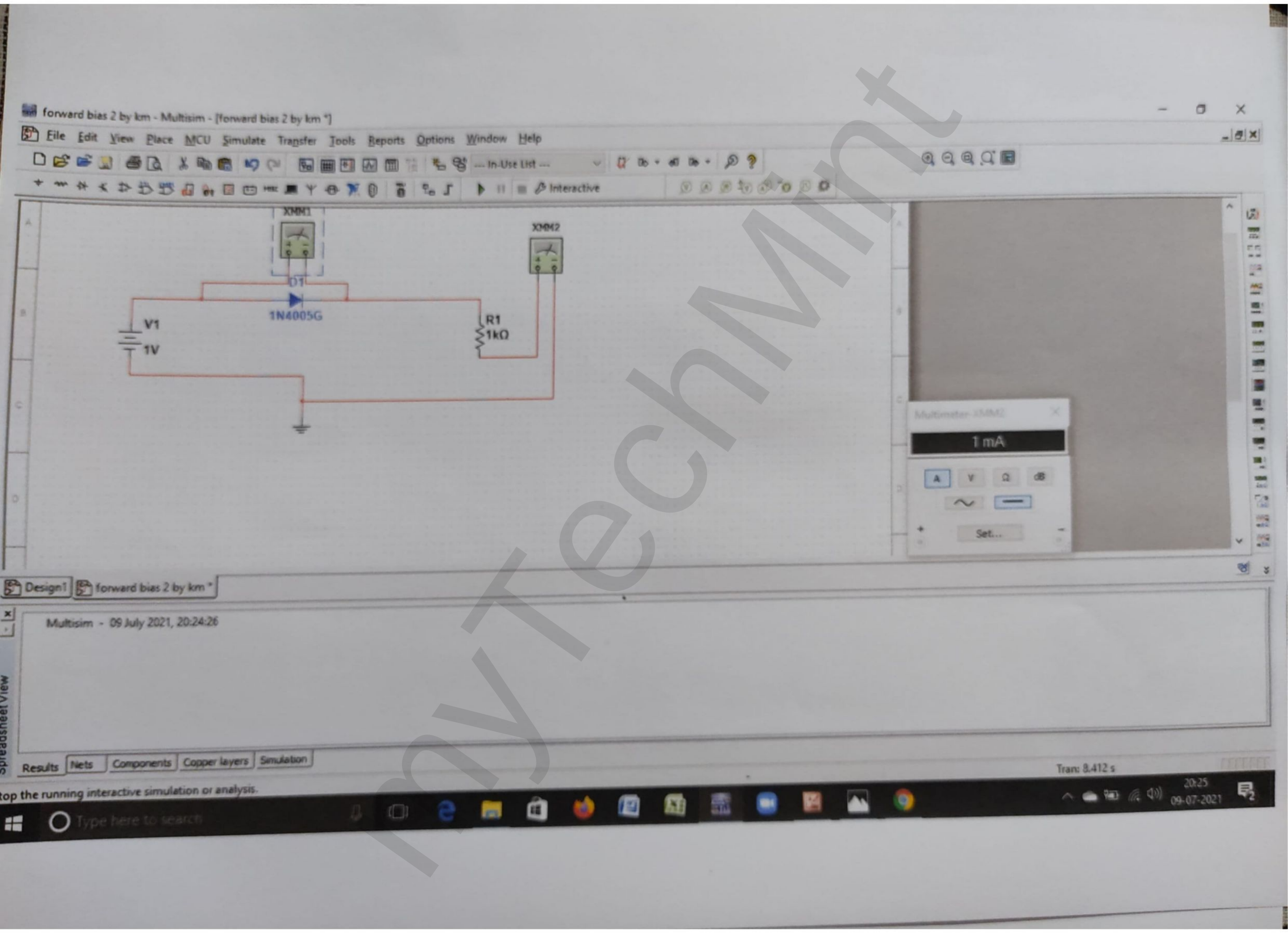
Result 2- The observation table will be formed which form the V-I characteristic graph.

Precautions-

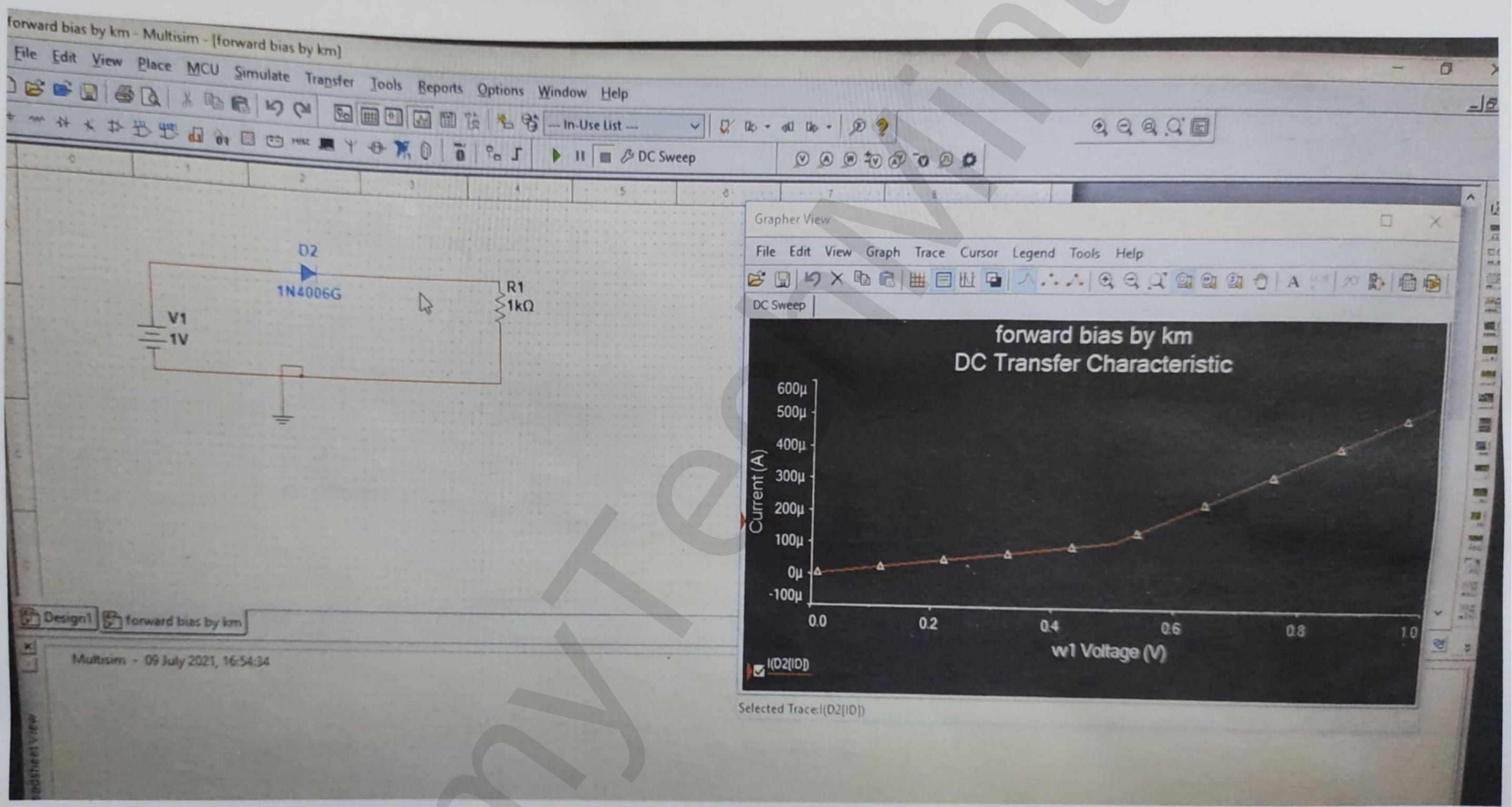
① Make the circuit very carefully
② Do proper simulation.
③ Note down the readings carefully.

Teacher's Signature


forward bias 2 by km - Multisim - [forward bias 2 by km *]
File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help
XMM1
XMM2
D1
1N4005G
V1
1.8V
R1
1kΩ
Multimeter-XMM2
1.8 mA
Set...
Design1
forward bias 2 by km *
Multisim - 09 July 2021, 20:24:26
Results Nets Components Copper layers Simulation
Run active analysis.
Tran: 0.609 s
20:26
09-07-2021



forward bias 2 by km - Multisim - [forward bias 2 by km *]
File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help
XMM1
XMM2
D1
1N4005G
V1
1V
R1
1kΩ
1 mA
Set...
Design1
forward bias 2 by km *
Multisim - 09 July 2021, 20:24:26
Results Nets Components Copper layers Simulation
Tran: 8.412 s


forward bias by km - Multisim - [forward bias by km]
File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help
D2
1N4006G
R1
1kΩ
V1
1V
Grapher View
forward bias by km
DC Transfer Characteristic
Current (A)
w1 Voltage (V)
I(D2[ID])
Selected Trace:I(D2[ID])
Multisim - 09 July 2021, 16:54:34

EXPERIMENT-6.

Aim- To plot the V-I characteristics of zener diode in reverse bias.

Software Used- Multisim

Theory- A zener diode is constructed for operation in the reverse breakdown region.
The relation between I-V is almost linear in the case.

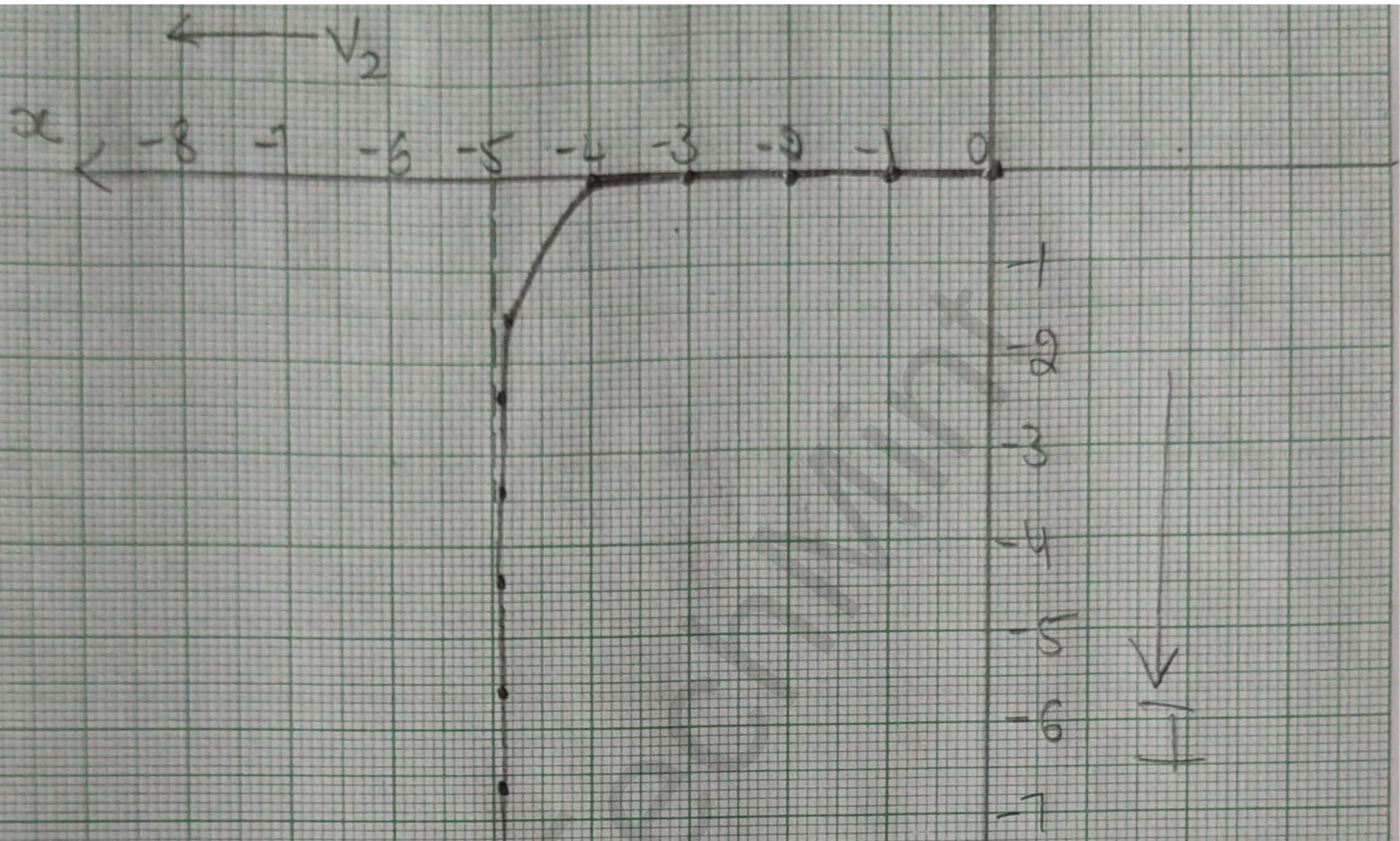
$$V_z = V_{z0} + I_z r_z$$

where r_z is the dynamic resistance of the zener at the operating point. V_{z0} is the voltage at which the straight line approximation of the V-I characteristics interacts the horizontal axis. After reaching a certain voltage, called breakdown voltage, the current increases widely even for a small change in voltage. Hence, there is no appreciable change in voltage. So when we plot the graph, we should get a curve near to y-axis and almost parallel to it for long.
After zener potential V_z, there will be a sudden change and graph will become downwards.

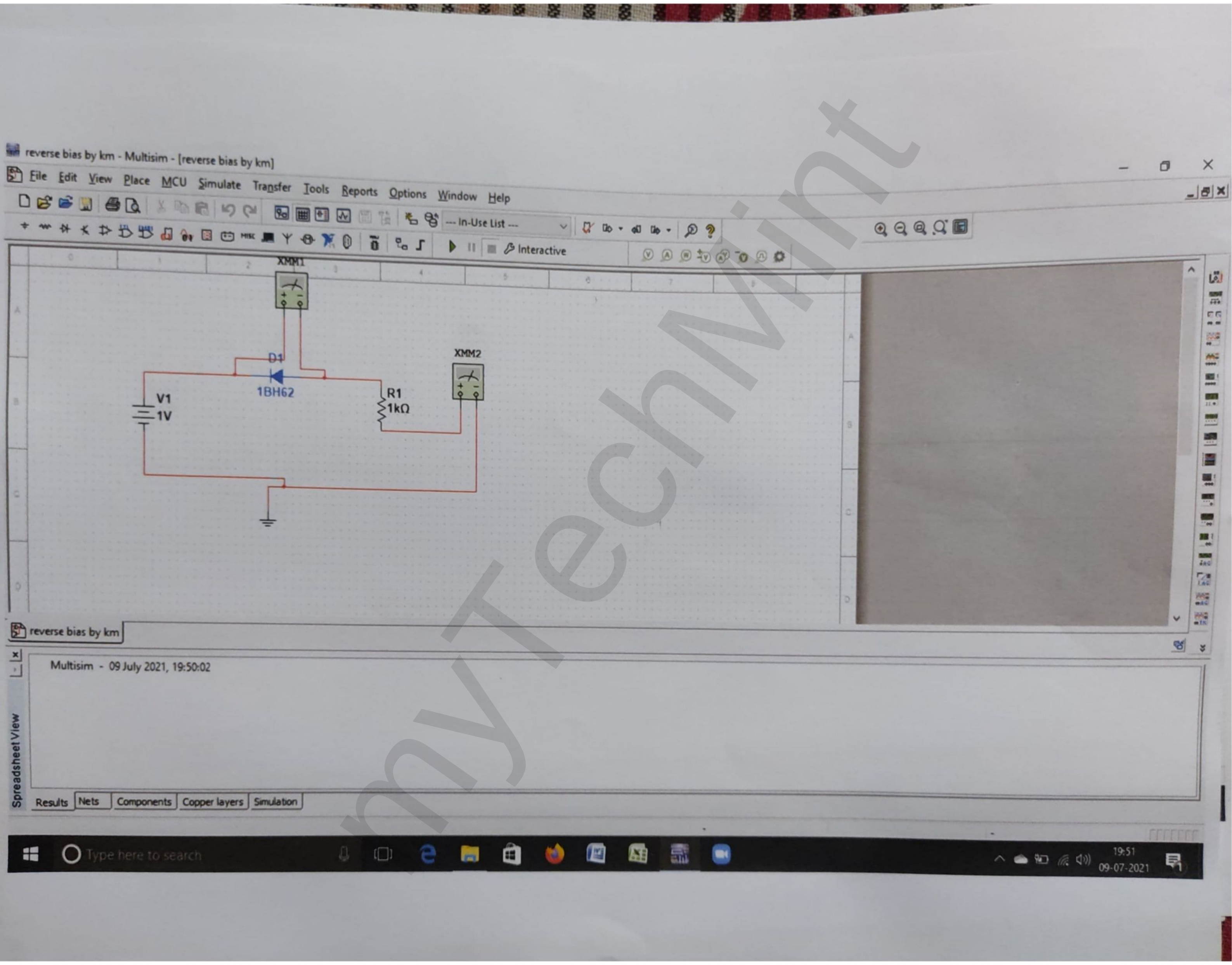
Procedure-
1. Make a proper circuit with DC power, zener diode of any choice, resistor of 1KΩ and then ground the circuit.
2. Attach a multimeter parallel to the zener diode.
3. Add a multimeter as ammeter and complete the circuit.
4. Click on run and note down the readings by changing the voltage supplied.
5. Note atleast 14 readings.

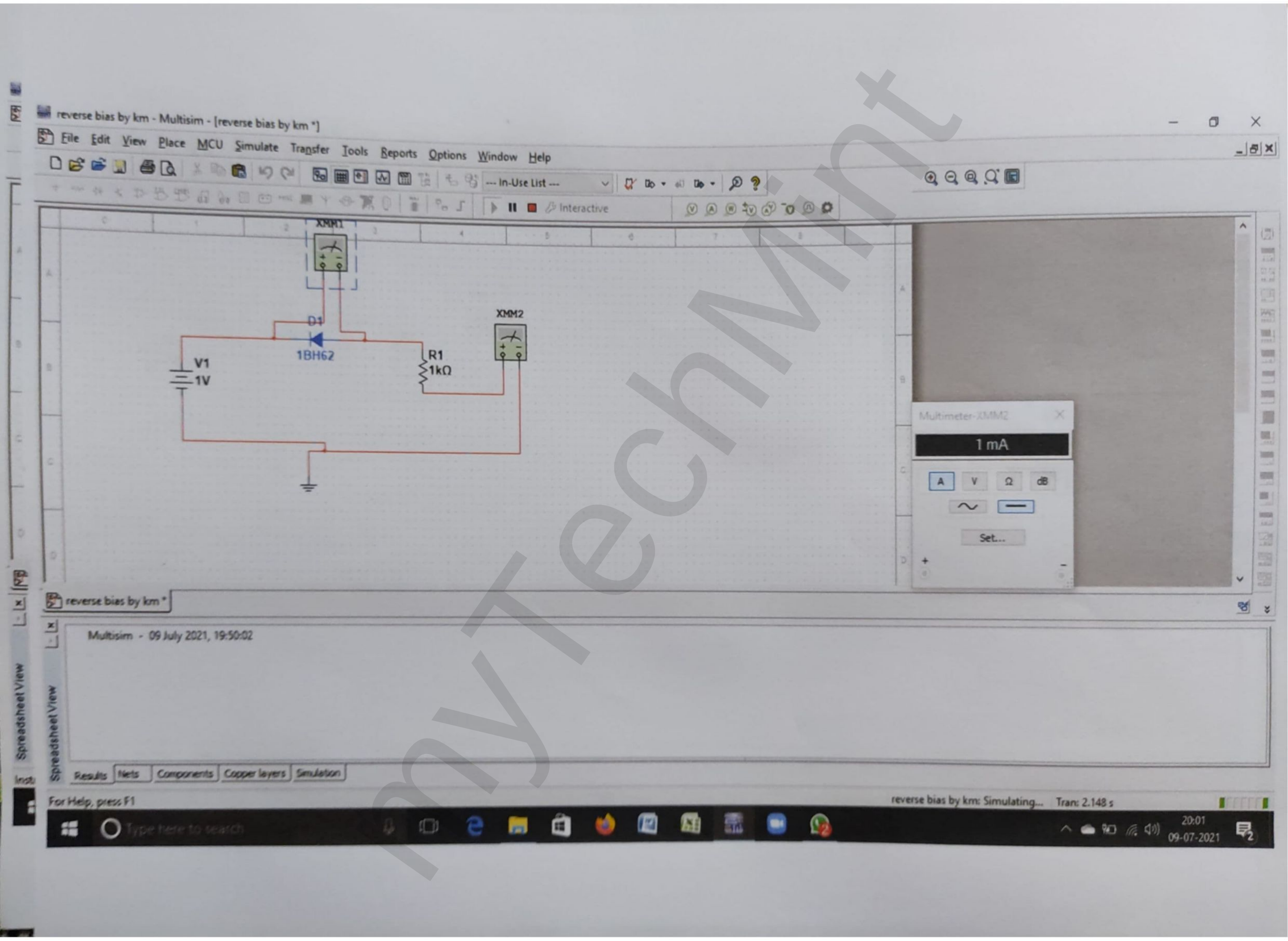
Observation

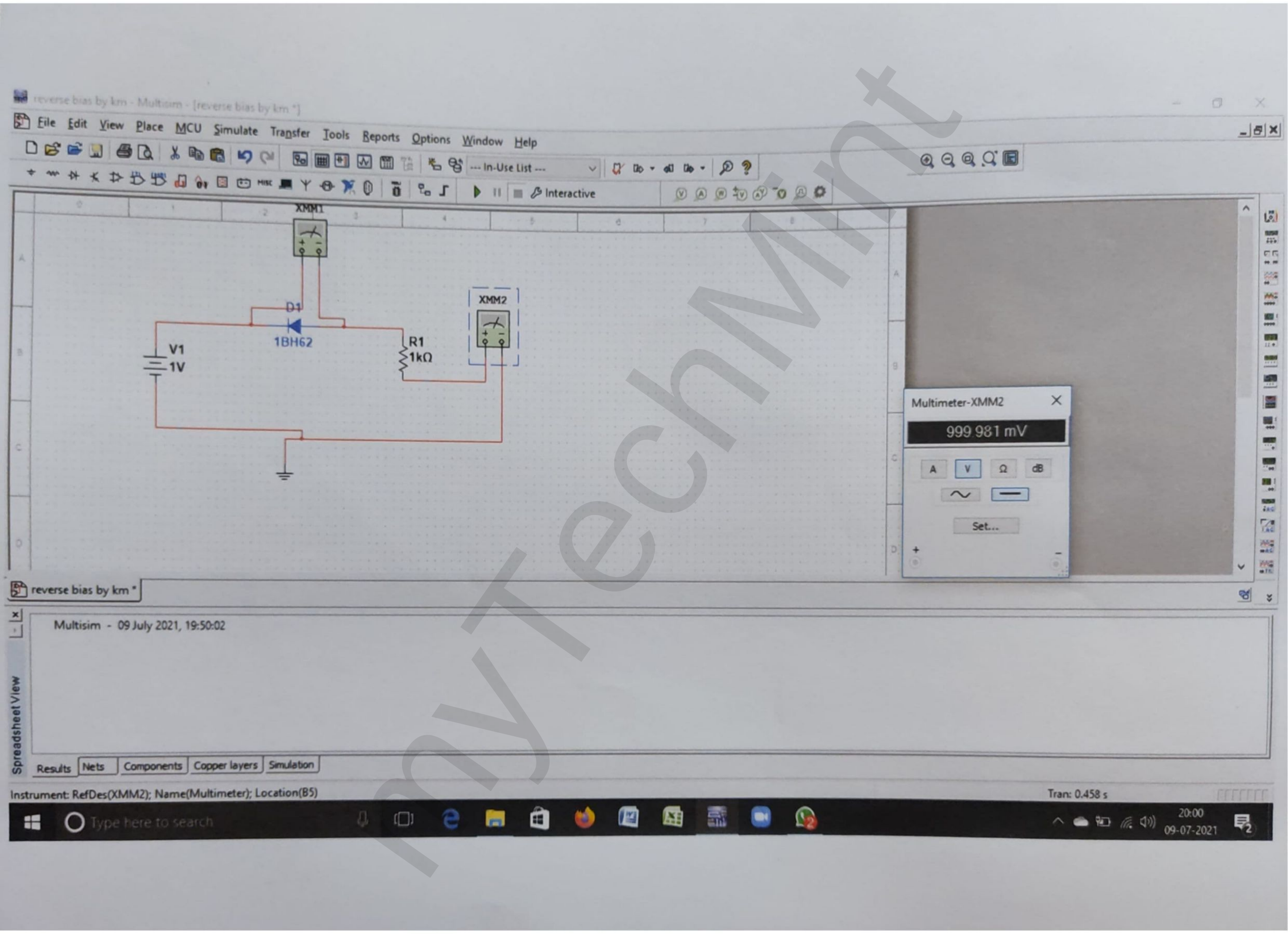
S.No.	Voltage Input (V_L)	V_Z (V)	Current I_Z (mA)
1	1V	-0.9 V	0
2	2V	-2.2V	0
3	3V	-3.2V	0
4	4V	-4.2V	0
5	5V	-4.6V	-0.3 mA
6	6V	-4.61V	-1.3 mA
7	7V	-4.67V	-2.3 mA
8	8V	-4.68V	-3.3 mA
9	9V	-4.72V	-4.02 mA
10	10V	-4.7V	-5.3 mA
11	11V	-4.73V	-6.5 mA
12	12V	-4.72V	-7.1 mA
13	13V	-4.74V	-8.4 mA
14	14V	-4.77V	-9.2 mA



V2
x
-8
-7
-6
-5
-4
-3
-2
-1
0
-1
-2
-3
-4
-5
-6
-7
-8
-9
-10
-11
I
y


reverse bias by km - Multisim - [reverse bias by km]
File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help
--- In-Use List ---
Interactive
XMM1
D1
1BH62
V1
1V
R1
1kΩ
XMM2
reverse bias by km
Multisim - 09 July 2021, 19:50:02
Spreadsheet View
Results
Nets
Components
Copper layers
Simulation
Type here to search
19:51
09-07-2021


reverse bias by km - Multisim - [reverse bias by km *]
File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help
XMM1
D1
1BH62
V1
1V
R1
1kΩ
XMM2
Multimeter-XMM2
1 mA
Set...
Multisim - 09 July 2021, 19:50:02
Results Nets Components Copper layers Simulation
For Help, press F1
reverse bias by km: Simulating...
Tran: 2.148 s
20:01
09-07-2021


reverse bias by km - Multisim - [reverse bias by km *]
File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help
--- In-Use List ---
Interactive
XMM1
D1
1BH62
V1
1V
R1
1kΩ
XMM2
Multimeter-XMM2
999.981 mV
A V Ω dB
Set...
reverse bias by km *
Multisim - 09 July 2021, 19:50:02
Spreadsheet View
Results Nets Components Copper layers Simulation
Instrument: RefDes(XMM2); Name(Multimeter); Location(B5)
Tran: 0.458 s
Type here to search
20:00
09-07-2021
myTechMint

Result - The breakdown potential / zener potential is

Precautions -

1. Carefully connect the circuit
2. Do not forget to ground the circuit
3. Note down the readings carefully.

EXPERIMENT-7

Aim - To plot input and output waveforms of half wave rectifier.

Software Used - Multisim

Theory - The conversion of AC into DC is called Rectification. Electronic devices can convert AC to DC power with high efficiency.
During the positive half cycle, the diode is forward biased and it conducts and current flow through load resistor. During negative half cycle, the diode is reverse biased and is equivalent to open circuit, hence current through load resistance is 0.
Thus, diode conducts only for one half cycle and results in half wave rectification.

$$V_{in} = V_p \sin \omega t, \quad 0 \le t \le T$$
$$V_{out} = V_{pi} \sin \omega t, \quad 0 \le t \le T$$

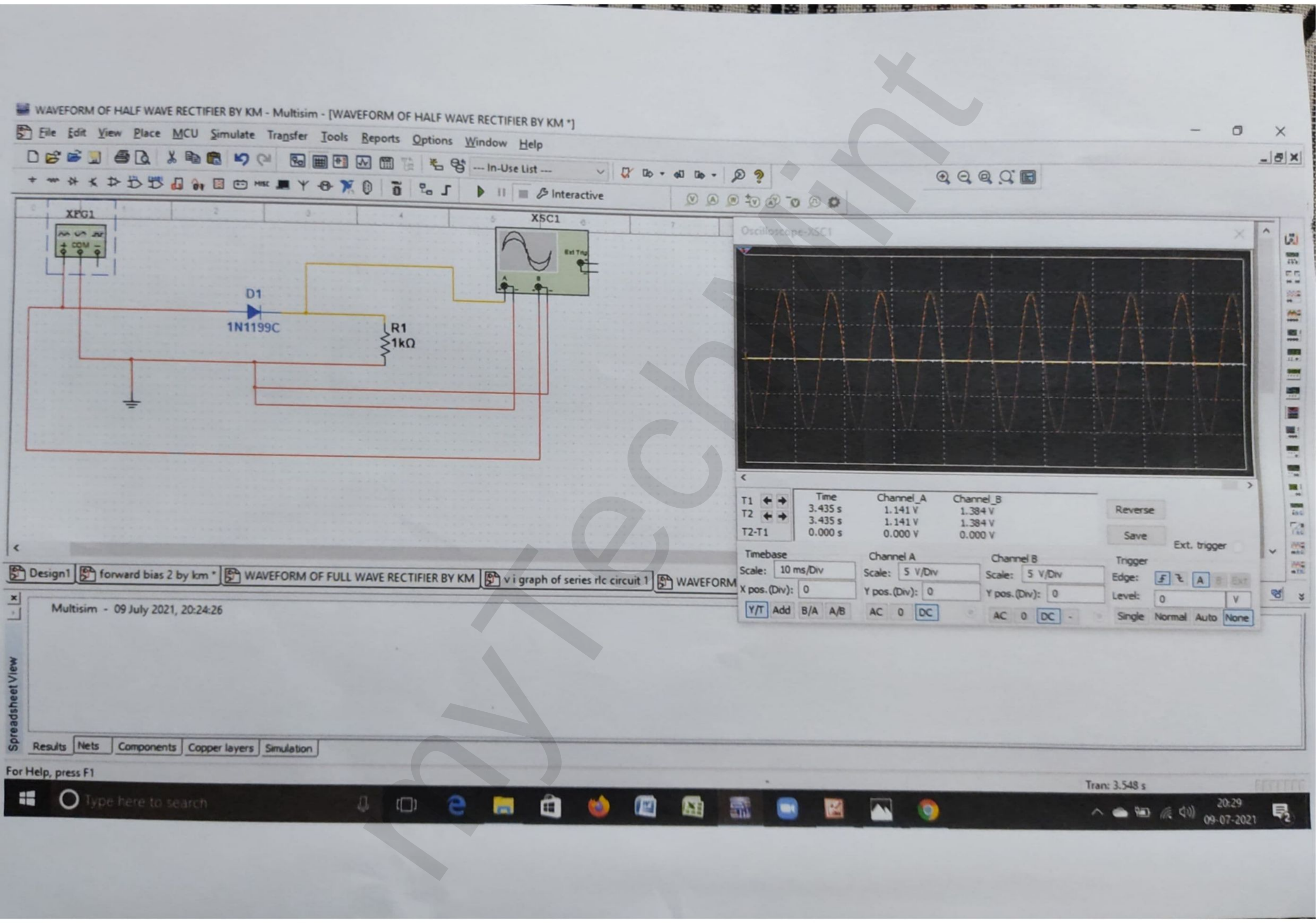
Average dc value of half wave rectified in sine wave voltage -

$$V_{avg} = \frac{1}{T}\left[\int_0^{T/2} V_{pi} \sin \omega t \, d(\omega t) + \int_{T/2}^{T} 0 \cdot d(\omega t)\right]$$

$$\Rightarrow \frac{V_{PL}}{\pi}$$

Procedure -

1. Make proper circuit with diode, resistor, function generator, oscilloscope and ground the circuit.
2. Make proper connection between diode, resistor, function generator. Resistor should be in series with diode.


WAVEFORM OF HALF WAVE RECTIFIER BY KM - Multisim - [WAVEFORM OF HALF WAVE RECTIFIER BY KM *]
File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help
XFG1
XSC1
D1
1N1199C
R1
1kΩ
Oscilloscope-XSC1
Time
Channel_A
Channel_B
T1 3.435 s 1.141 V 1.384 V
T2 3.435 s 1.141 V 1.384 V
T2-T1 0.000 s 0.000 V 0.000 V
Reverse
Save
Ext. trigger
Timebase
Scale: 10 ms/Div
X pos.(Div): 0
Channel A
Scale: 5 V/Div
Y pos.(Div): 0
Channel B
Scale: 5 V/Div
Y pos.(Div): 0
Trigger
Edge:
Level: 0 V
Y/T Add B/A A/B
AC 0 DC
Single Normal Auto None
Design1
forward bias 2 by km
WAVEFORM OF FULL WAVE RECTIFIER BY KM
v i graph of series rlc circuit 1
Multisim - 09 July 2021, 20:24:26
Results Nets Components Copper layers Simulation
For Help, press F1
Tran: 3.548 s
20:29
09-07-2021

(3) Connect oscilloscope with function generator and do proper connection and ground the circuit.
(4) Run the circuit on the software
(5) Set respective value to frequency and click on oscilloscope to see the waveform output
(6) To see more clearly, click on the output connecting wire and go to properties and change the colour.
(7) Now input and output waveform will be seen on the oscilloscope.

Result - We get input, output waveform of half wave rectifier in different colour.

Precautions -

(1) Make connections carefully in the circuit.
(2) Do not forget to ground the circuit
(3) Note observation carefully.

EXPERIMENT-8

Aim - To plot input and output waveforms of half wave rectifier with capacitor.

Software Used - Multisim

Theory - All rectifiers contains output of considerable amount of ripple in addition to the DC component. In order to avoid AC components, a filter is connected at the output of the rectifier.
Capacitor input filter, choke input filter, RC, CRC, LC and CLC filters are used.
Capacitor input filters are simplest and cheap. A high value capacitor C is connected in shunt with load resistor R_L. Capacitor charges to peak voltage V_m, when half cycle appears to be at the output.
After peak value is passed, the capacitor discharges through load resistor slowly since the diode is reverse biased by the capacitor voltage. Before, the voltage of capacitor drops substantially, next output arrives and capacitor recharge to peak.

Rms value of filtered output was a triangular wave.

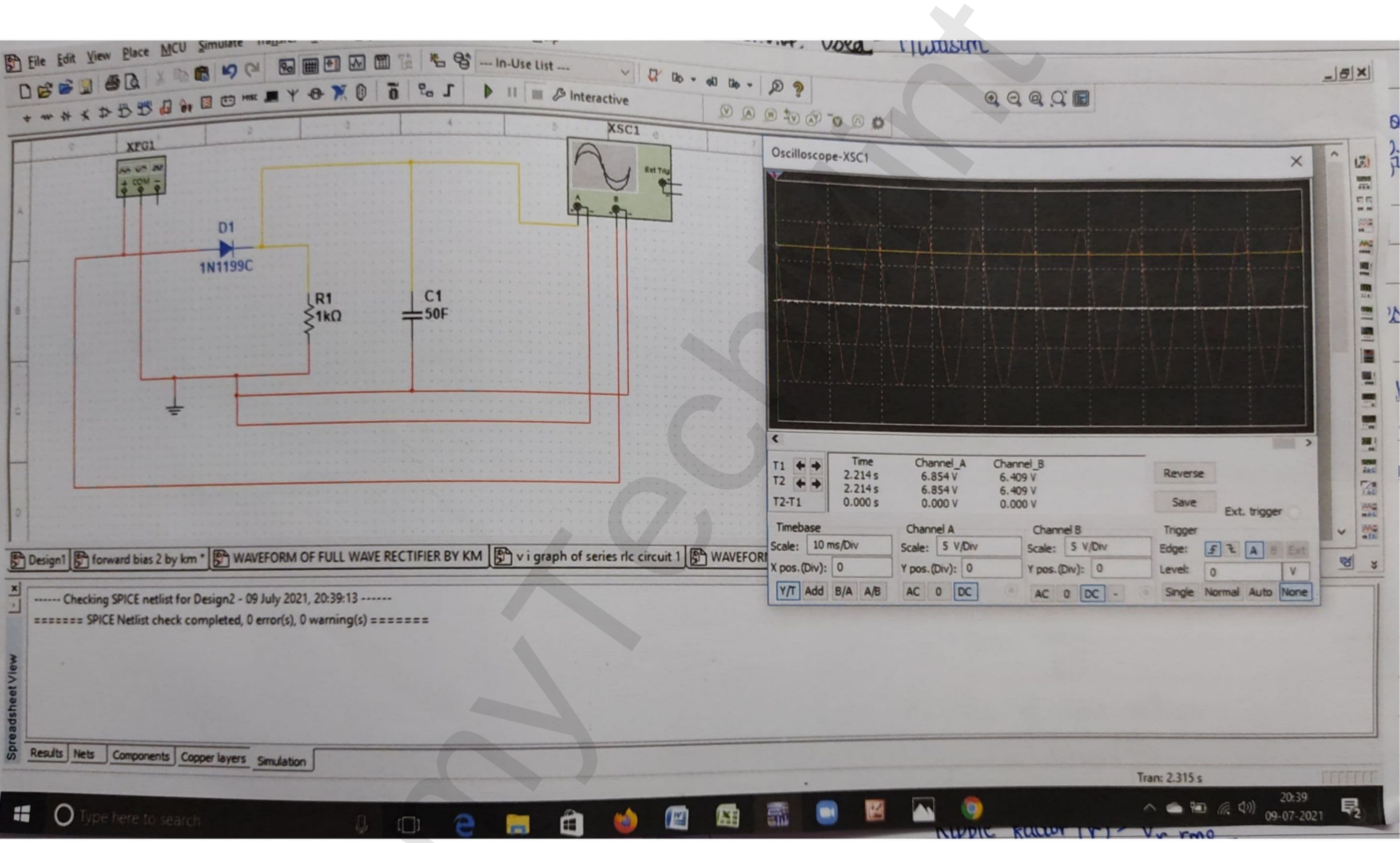
$$V_{RMS} = \frac{V_{rpp}}{2\sqrt{3}}$$

V_{rpp} → peak to peak value of ripple voltage.

$$V_{dc} = V_m - (V_{rpp}/2)$$

$$\text{Ripple factor } (r) = \frac{V_{r,rms}}{V_{dc}}$$

Teacher's Signature


XFG1
D1
1N1199C
R1
1kΩ
C1
50F
XSC1
Oscilloscope-XSC1
Time
Channel_A
Channel_B
T1
2.214 s
6.854 V
6.409 V
T2
2.214 s
6.854 V
6.409 V
T2-T1
0.000 s
0.000 V
0.000 V
Reverse
Save
Ext. trigger
Timebase
Scale: 10 ms/Div
X pos.(Div): 0
Channel A
Scale: 5 V/Div
Y pos.(Div): 0
Channel B
Scale: 5 V/Div
Y pos.(Div): 0
Trigger
Edge:
Level: 0 V
Single Normal Auto None
------ Checking SPICE netlist for Design2 - 09 July 2021, 20:39:13 ------
======= SPICE Netlist check completed, 0 error(s), 0 warning(s) =======
Tran: 2.315 s

Date

Expt. No.   Page No. 18

Procedure -

1. Make proper circuit connection with diode, resistor, function generator, oscilloscope and capacitor as filter.
2. Resistor should be connected in series with diode and capacitor should be connected in parallel to output circuit.
3. Connect oscilloscope with the function generator and another with capacitor and then ground the circuit.
4. After making proper connection and connecting oscilloscope, capacitor, diode and resistor.
5. Run the stimulation to see input - output waveform.
6. Click on function generator and provide necessary frequency to the circuit.
7. Click on the oscilloscope to see the waveform produced.
8. Changing the value of capacitor, you can see the output waveform filtered.
9. If you want to see output waveform clearly, you can go to properties and change colour.
10. Now, desired output waveform will be on the screen.

Result - The filtered input and output waveform is obtained on the oscilloscope.

Precautions -

1. Make circuit carefully, make each and every connection properly.
2. Do not forget to ground the circuit.
3. Note the waveform carefully.

Teacher's Signature

EXPERIMENT-9

Aim - To study input and output waveforms of full wave Rectifier.

Software Used - Multisim

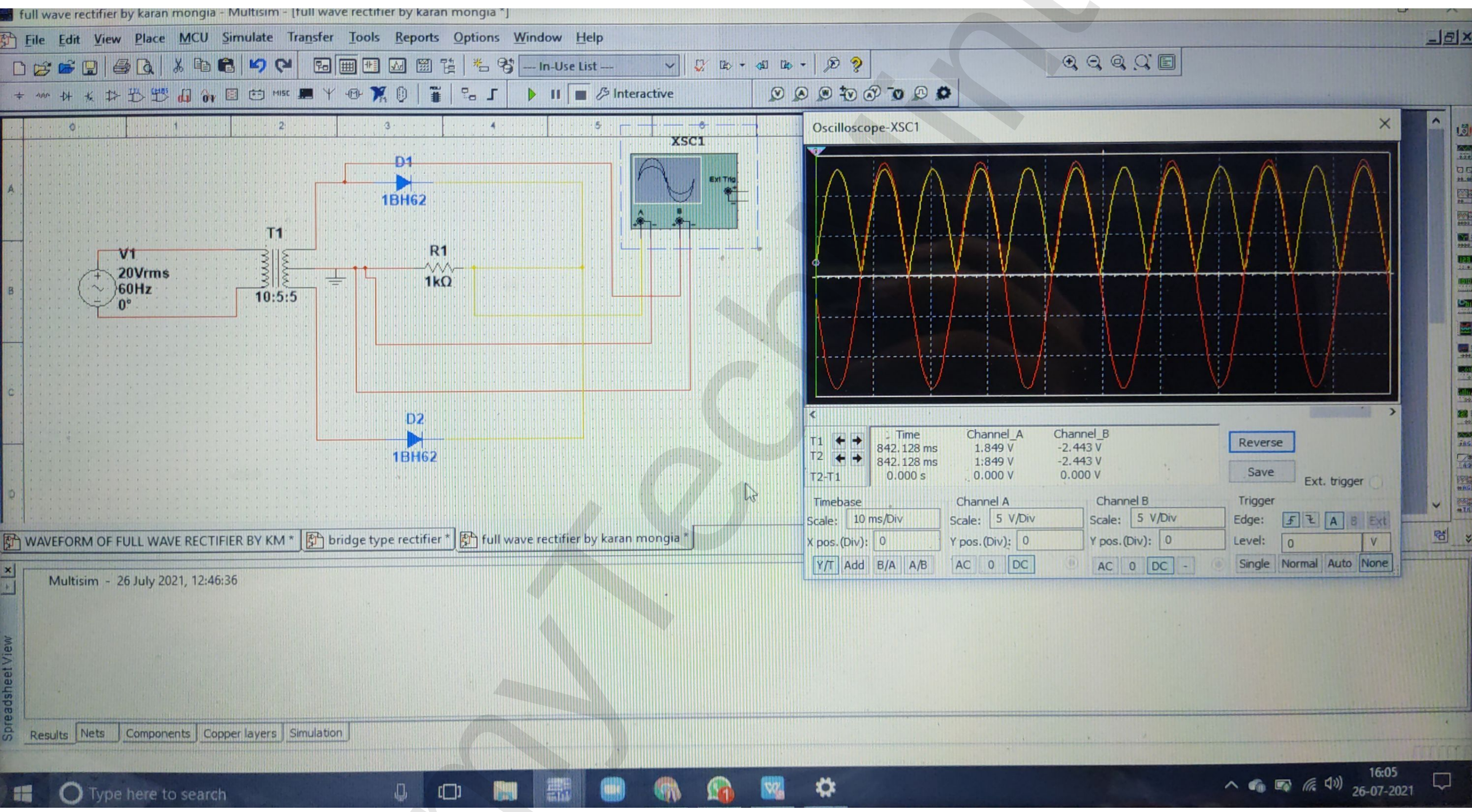
Theory - A device is capable of converting a sinusoidal input waveform into a unidirectional waveform with non-zero average component is called rectifier. A full wave rectifier with resistive load is shown in the circuit diagram. It consists of 2 half wave rectifiers connected to the common load. One rectifies during positive half cycle of input and other rectifying the negative half cycle. The transformer supplies 2 diodes (D_1 and D_2) with sinusoidal input voltages that are equal in magnitudes but opposite in phase.
During input positive half cycle, diode D_1 is ON and D_2 is OFF. During negative half cycle, D_1 is OFF and D_2 is ON. The output of the full wave rectifier contains both ac and dc components.
The undesirable ac components/ripple can be minimized using filters.

$$\text{Ripple factor } (r) = \sqrt{\left[\frac{V_{PL}/2}{2V_{PL}/\pi}\right]^2 - 1} \Rightarrow 0.482.$$

Procedure -

1. Make proper circuit with diodes, function generator, resistor, oscilloscope and ground the circuit.
2. Make proper connections such that diodes are parallel to the resistor
3. Connect oscilloscope with function generator and do proper connection and ground the circuit.
4. Run the circuit on the software.

Teacher's Signature


full wave rectifier by karan mongia - Multisim - [full wave rectifier by karan mongia *]
File
Edit
View
Place
MCU
Simulate
Transfer
Tools
Reports
Options
Window
Help
--- In-Use List ---
Interactive
XSC1
D1
1BH62
T1
V1
20Vrms
60Hz
0°
10:5:5
R1
1kΩ
D2
1BH62
Oscilloscope-XSC1
Time
Channel_A
Channel_B
T1
842.128 ms
1.849 V
-2.443 V
T2
842.128 ms
1.849 V
-2.443 V
T2-T1
0.000 s
0.000 V
0.000 V
Reverse
Save
Ext. trigger
Timebase
Scale:
10 ms/Div
X pos.(Div):
0
Y/T
Add
B/A
A/B
Channel A
Scale:
5 V/Div
Y pos.(Div):
0
AC
0
DC
Channel B
Scale:
5 V/Div
Y pos.(Div):
0
AC
0
DC
-
Trigger
Edge:
Level:
0
V
Single
Normal
Auto
None
WAVEFORM OF FULL WAVE RECTIFIER BY KM *
bridge type rectifier *
full wave rectifier by karan mongia *
Multisim - 26 July 2021, 12:46:36
Spreadsheet View
Results
Nets
Components
Copper layers
Simulation
Type here to search
16:05
26-07-2021

Date

Expt. No. Page No.

(5) Set respective value to frequency and click on the oscilloscope to see waveform output.

(6) To see properly, click on the output connecting wire and go to properties and change the colour.

(7) Now input and output waveform will be seen on the oscilloscope.

Result - We get input, output waveform of full wave rectifier in different colours.

Precautions -

(1) Make connections carefully in circuit.
(2) Do not forget to ground the circuit.
(3) Note observation carefully.

EXPERIMENT- 10

Aim- To plot input and output waveforms of bridge type rectifier

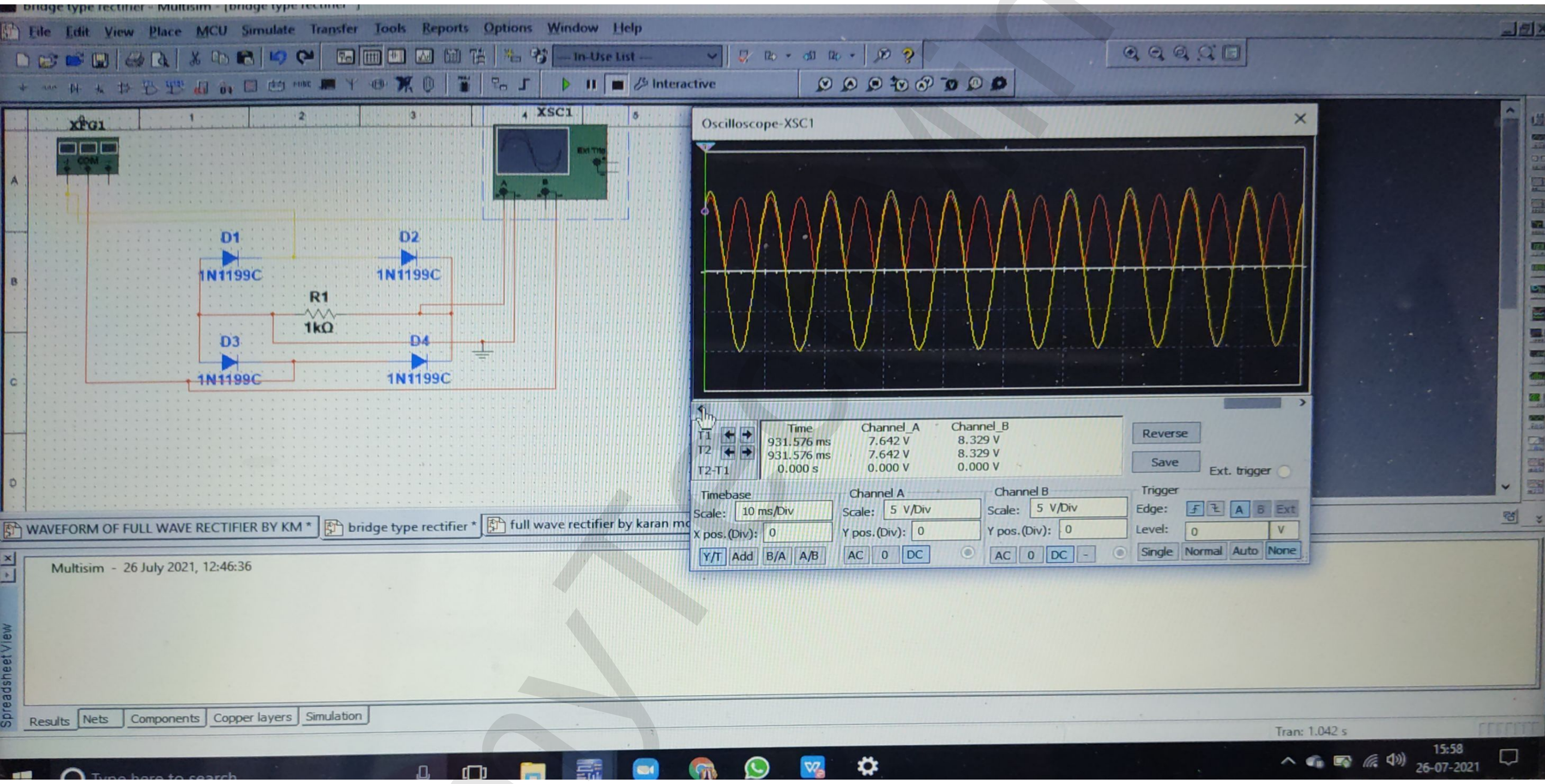
Software Used- Multisim

Theory- A device is capable of converting sinusoidal input waveform into unidirectional waveform with non-zero average components is called rectifiers.
A practical full wave rectifier with resistive load. It consists of two half wave rectifiers connected to a common load. One rectifier during positive half cycle of the input and other rectifying the negative half cycle. The two diodes (D_1, D_2) with sinusoidal input voltages that are equal in magnitude but opposite in phase. During input positive half cycle, diode D_1 is on and diode D_2 is off. During input negative half cycle, diode D_1 is off and D_2 is on. The output of full wave rectifier contains both ac and dc components.

Procedure-

1. Make proper circuit using diode, resistor, function generator, oscilloscope
2. Add 4 diodes, 2 in series respectively and 2 in parallel to them, make a proper connection of diodes and connect them.
3. Take a resistor and connect between the diodes connection.
4. Connect the function generator, connect the positive terminal between D_1 and D_2 and negative terminal in between D_3 and D_4.
5. Connect the oscilloscope, connect (A) positive terminal to resistor branch and negative terminal to the other side of resistor.
6. Now connect B, positive terminal of oscilloscope to function generator's (+ve) terminal, and negative terminal to the negative of function generator across D_3 and D_4.
7. Now ground the circuit, connect the ground to output terminal.

Teacher's Signature ________


Oscilloscope-XSC1
XFG1
XSC1
D1
D2
D3
D4
1N1199C
R1
1kΩ
Time
Channel_A
Channel_B
T1
T2
T2-T1
931.576 ms
7.642 V
8.329 V
0.000 s
0.000 V
Reverse
Save
Ext. trigger
Timebase
Scale: 10 ms/Div
X pos.(Div): 0
Channel A
Scale: 5 V/Div
Y pos.(Div): 0
Channel B
Trigger
Edge:
Level: 0 V
Y/T Add B/A A/B
AC 0 DC
Single Normal Auto None
WAVEFORM OF FULL WAVE RECTIFIER BY KM *
bridge type rectifier *
Multisim - 26 July 2021, 12:46:36
Results Nets Components Copper layers Simulation
Tran: 1.042 s
15:58
26-07-2021

⑧ Now run the stimulation and observe input-output waveform.

Result - The input-output waveform will appear on the oscilloscope. Now change the frequency to 100 Hz and output waveforms will appear on the oscilloscope.

Precautions -

① Make proper connection among diodes and function generator.
② Make proper connections of oscilloscope.
③ Do not forget to ground the circuit
④ Make proper frequency & range and note the observation carefully.

EXPERIMENT-11

Aim - To plot input and output waveforms of bridge type rectifier with capacitor (as filter).

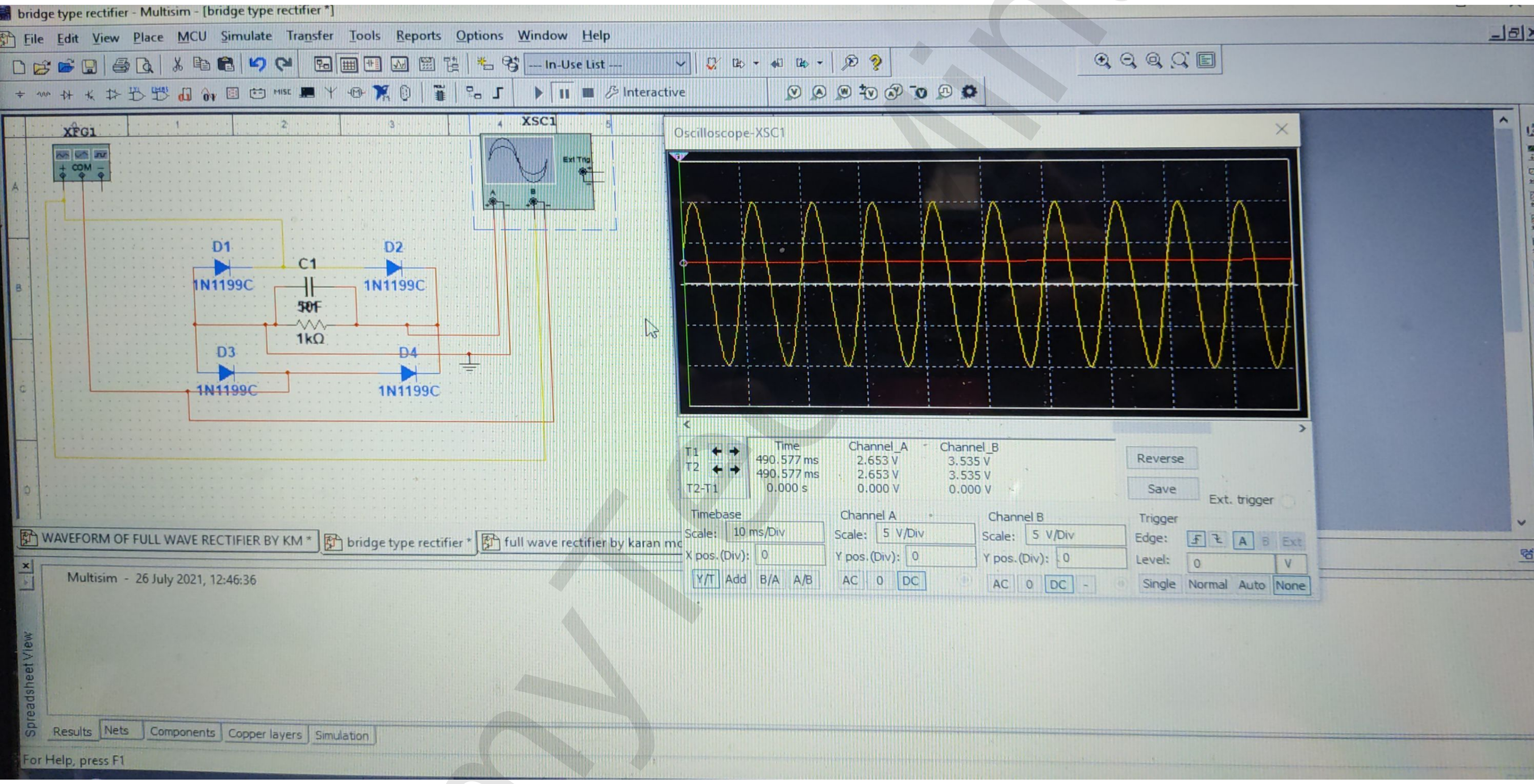
Software Used - Multisim

Theory - The full wave rectifier rectifies both positive and negative half cycles of the input AC signal. The DC signal obtained at the output still contains some ripples. To reduce these ripples at the output, we use a filter.
The filter is an electronic device that converts pulsating direct current into pure direct current. The filter (capacitor) is connected parallel to load resistor. The charging of the capacitor takes place as we provide voltage and maximum value of input supply voltage. The capacitor filter reduces ripples form on the process of charging and discharging. Then a proper output is obtained with filtered waveform.

Procedure -

1. Make proper circuit using 4 diodes, resistor, capacitor, function generator and oscilloscope.
2. Make proper connection of diodes, connect D_1 and D_2 in series and D_3 and D_4 in series and D_1 and D_3, D_2 and D_4 are all parallel.
3. Connect the resistor across diodes.
4. Now connect capacitor parallel to resistor connected.
5. Now connect function generator positive terminal across D_1 and D_2 and negative across D_3 and D_4.
6. Now connect the oscilloscope (A) positive terminal to resistor branch and negative to other side of resistor branch.
7. Now connect the oscilloscope (B) positive terminal to function generator positive branch and negative to negative terminal of function generator.

Teacher's Signature ______



(8) Now connect the circuit, connect ground to oscilloscope (A) terminal branch.
(9) Now run stimulation and note input-output waveform on the oscilloscope.
(10) Change the value of capacitor to 50F, and note the result's waveform.

Result - Now, we observe fully filtered waveform outputs on the oscilloscope. The input-output waveform ripple free waveforms on the oscilloscope.

Precautions -

(1) Make proper connections.
(2) Connect the function generator and oscilloscope properly
(3) Do not forget to ground the circuit.
(4) Note the observations carefully.

EXPERIMENT-12.

Aim - To study input and output waveforms of center tapped full wave rectifier with capacitor (as filter).

Software Used - Multisim

Theory - The full wave rectifier rectifies both positive and negative half cycles of input AC signal. The DC signal obtained at the output contains some ripples. To reduce ripples at the output, we use a filter.
The filter is an electronic device that converts pulsating direct current into pure direct current. The filter (capacitor) is connected parallel to the load resistor. The capacitor filter is to short the ripples to the ground and blocks the pure DC. The charging of the capacitor takes place as we provide voltage and maximum value of the input supply voltage. The capacitor filter reduces ripples on the process of charging and discharging. Then proper output is obtained with filtered waveform.

Procedure

1. Make proper connection using AC power source, centre tapped transformer, 2 diodes, resistor and oscilloscope.
2. Connect AC power supply to the two terminals of centre tapped transformer.
3. Now attach top and bottom terminal of transformer of 2 diodes ~~centered~~ connected parallel to each other.
4. Now attach diodes together, and attach resistor of 1 KΩ to one end of diode connection and other with centre branch of transformer.
5. Now attach capacitor and connect parallel to the resistor.
6. Now attach oscilloscope A(+ve) terminal with resistor one end and (-ve) terminal to the other end of resistor.
7. Attach the oscilloscope's B(+ve) terminal to Diode D_1 and (-ve) terminal to the

Teacher's Signature ____

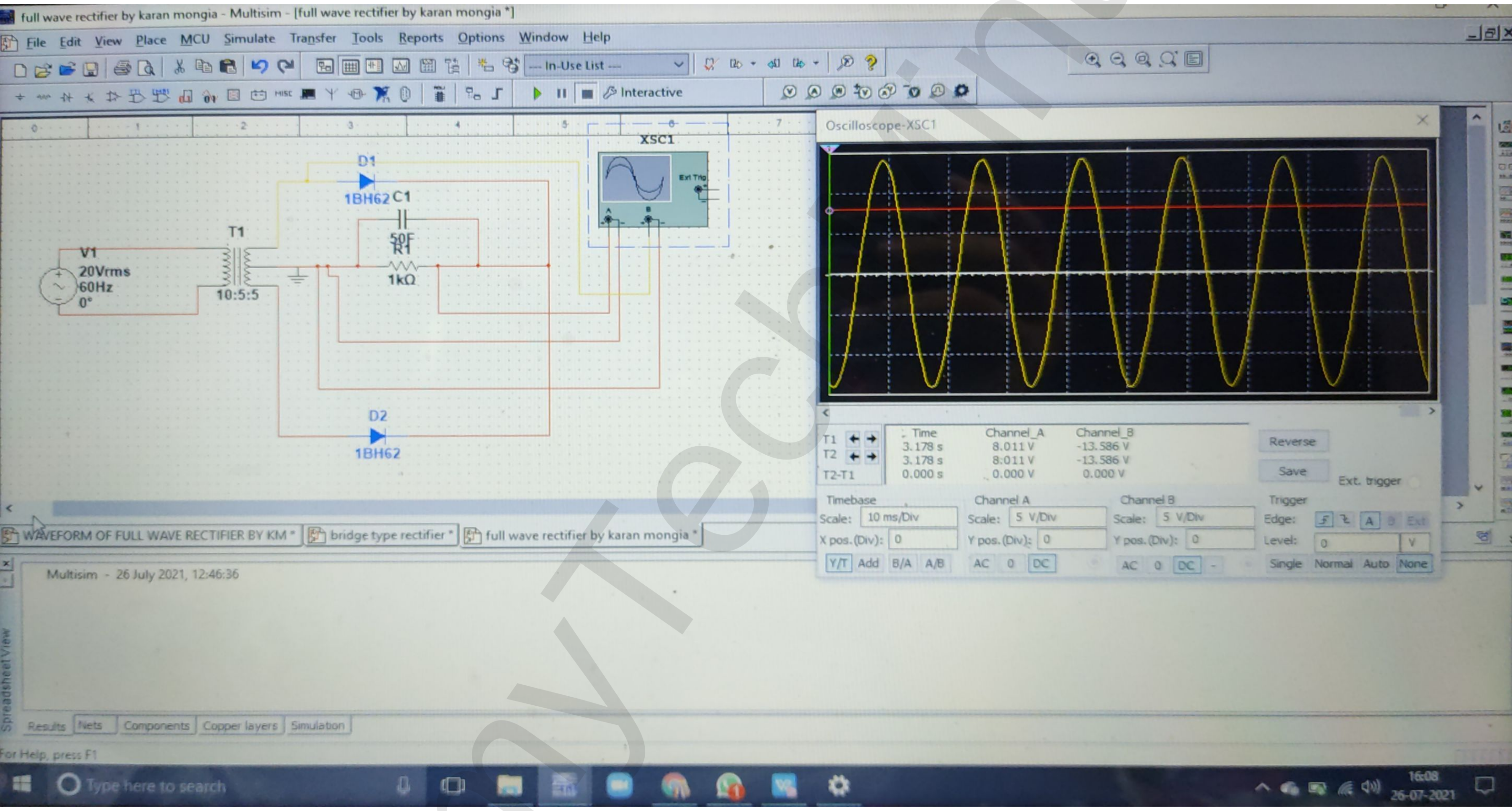
centre branch of the transformer.

8. Now ground the circuit, attach ground to the centre branch of the transformer.
9. Adjust the V_{rms} value of ac source between 20-40 and also adjust value of capacitor between 30-50. Farad
10. Now run the stimulation and observe the waveforms.

Result - The filtered waveforms are obtained on oscilloscope. We get the desired input-output ripple free waveforms.

Precautions -

1. Make connections properly as in the procedure.
2. Do not forget to ground the circuit
3. Adjust V_{rms} value carefully
4. Adjust capacitor value between 20-50 Farad
5. Note the observations carefully.


full wave rectifier by karan mongia - Multisim - [full wave rectifier by karan mongia *]
File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help
Interactive
D1
1BH62
C1
50F
R1
1kΩ
T1
V1
20Vrms
60Hz
0°
10:5:5
XSC1
D2
1BH62
Oscilloscope-XSC1
Time
Channel_A
Channel_B
T1
T2
T2-T1
3.178 s
3.178 s
0.000 s
8.011 V
8.011 V
0.000 V
-13.586 V
-13.586 V
0.000 V
Reverse
Save
Ext. trigger
Timebase
Scale: 10 ms/Div
X pos.(Div): 0
Channel A
Scale: 5 V/Div
Y pos.(Div): 0
Channel B
Scale: 5 V/Div
Y pos.(Div): 0
Trigger
Edge:
Level: 0
Y/T Add B/A A/B
AC 0 DC
Single Normal Auto None
WAVEFORM OF FULL WAVE RECTIFIER BY KM *
bridge type rectifier *
full wave rectifier by karan mongia *
Multisim - 26 July 2021, 12:46:36
Results Nets Components Copper layers Simulation
For Help, press F1
Type here to search
16:08
26-07-2021

EXPERIMENT- 13

Aim - To plot input and output characteristic of Common Base Transistor.

Software Used - Multisim

Theory - In Common Base configurations, the base terminals of the transistor will be common between input and output terminals. This configuration offers low input impedance, high output impedance, high resistance and high voltage gain.

The input characteristics of common base configuration circuit which describes variation of emitter current, ie base emitter voltage, V_{BE} keeping collector-base voltage, V_{CB} constant.

$$R_{in} = \frac{\Delta V_{BE}}{\Delta I_E} \Big| V_{CE} = constant$$

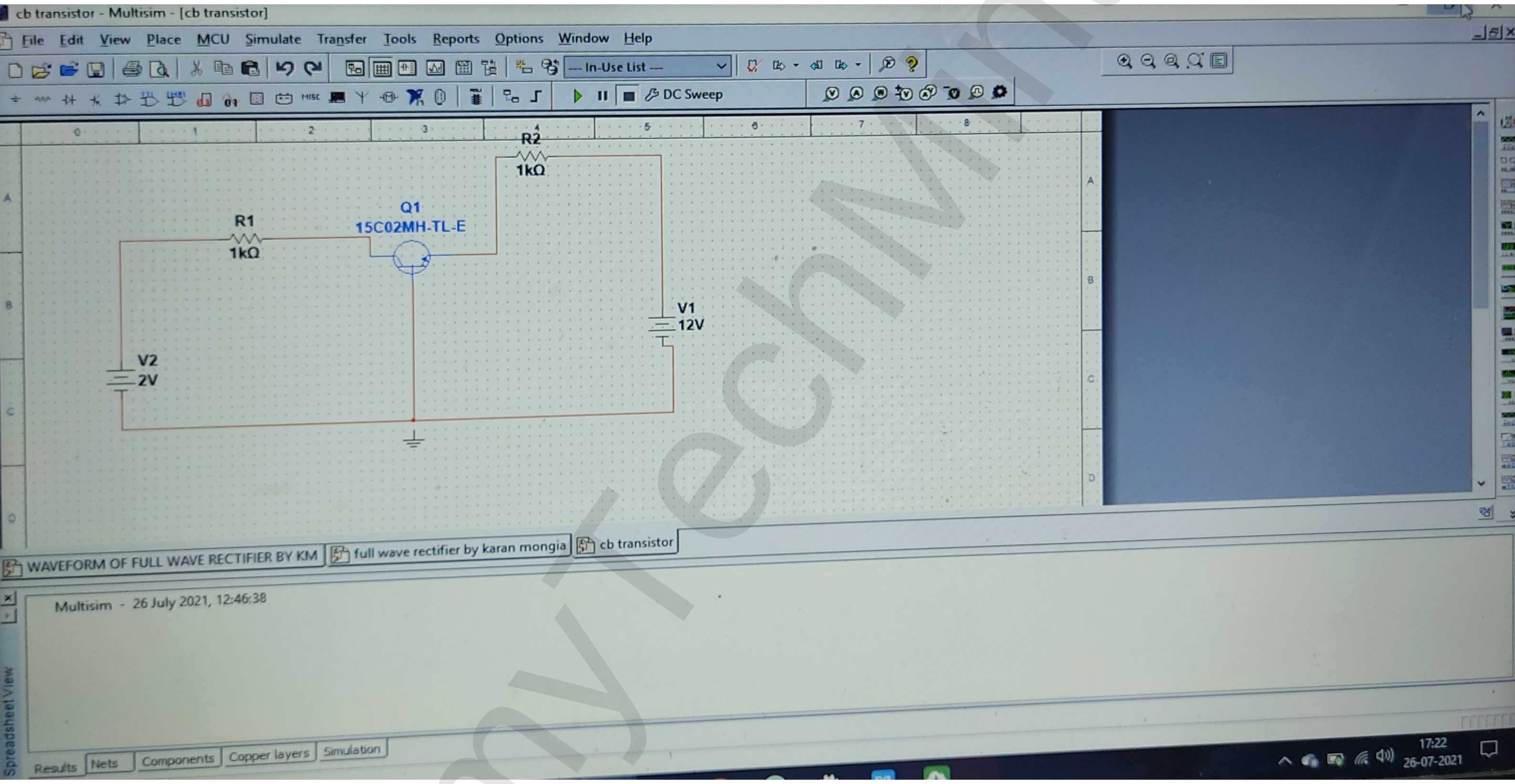
The output characteristics of CB configuration show the variation of collector current, I_C with V_{CB} when emitter current, I_E is held constant, V_{CB} as constant. It can be expressed as-

$$\alpha = \frac{\Delta I_C}{\Delta I_E} \Big| V_{CB} = constant.$$

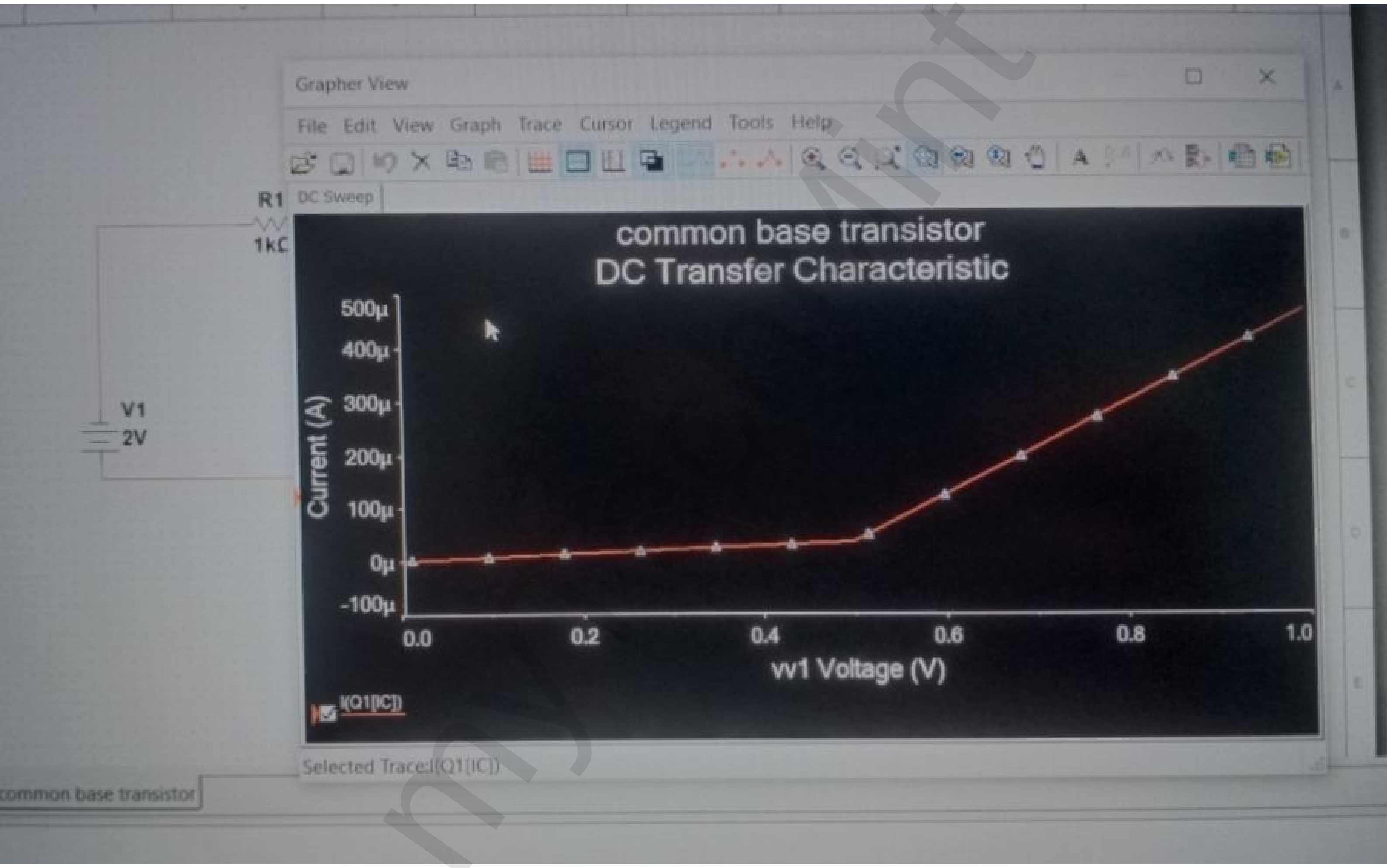
Procedure -

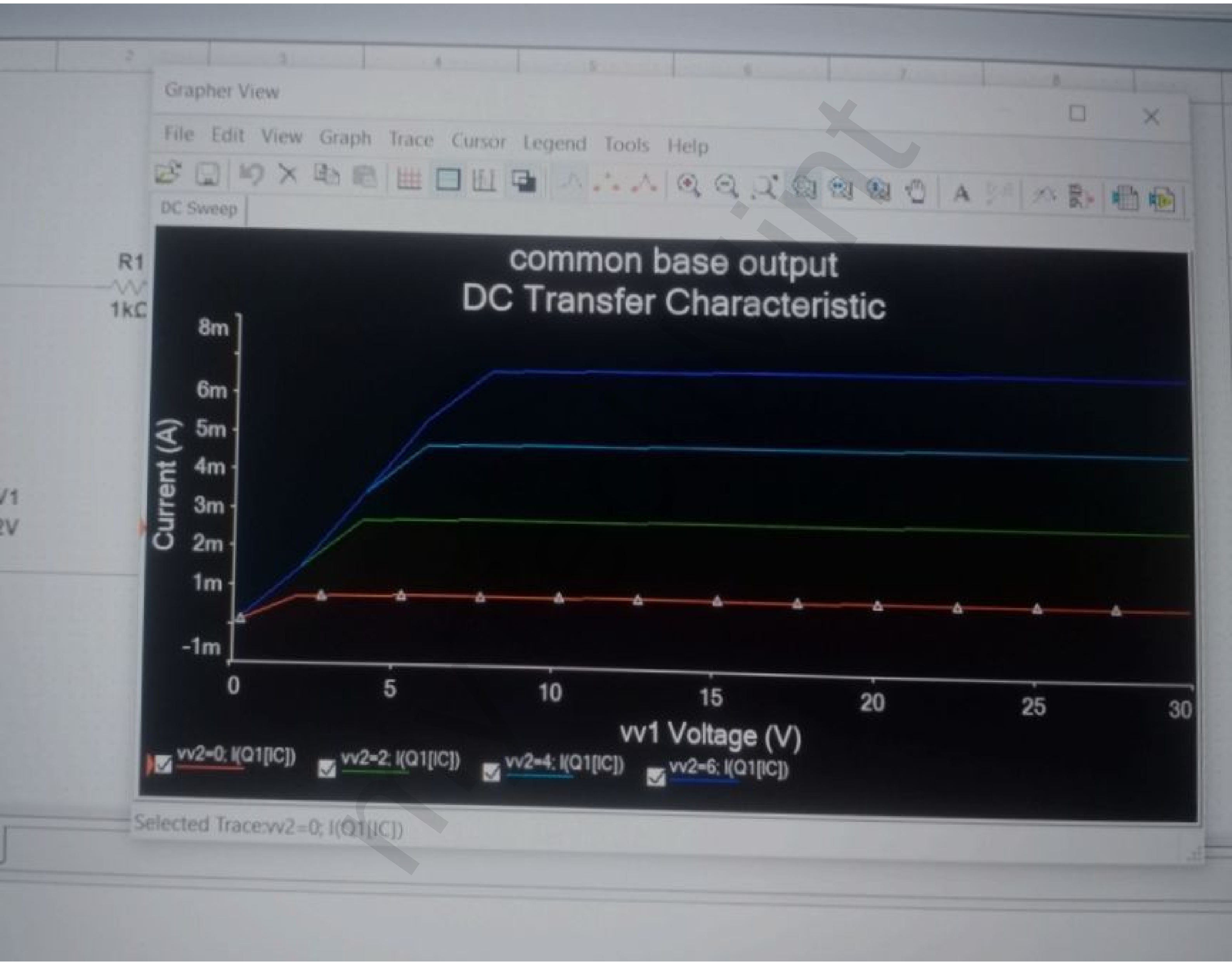
1. Make proper circuit using transistor of common base, two resistor and DC power supply in input and output and ground. $V_1 = 2V$, $V_2 = 12V$
2. Now connect common base transistor and connect it with resistor connected in input and other end to the other resistor.
3. Connect DC power source in the input to the resistor and other DC power source of output to output resistor. Connect input power source in negatively biased.

Teacher's Signature



cb transistor - Multisim - [cb transistor]
File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help
--- In-Use List ---
DC Sweep
R2
1kΩ
Q1
15C02MH-TL-E
R1
1kΩ
V1
12V
V2
2V
WAVEFORM OF FULL WAVE RECTIFIER BY KM
full wave rectifier by karan mongia
cb transistor
Multisim - 26 July 2021, 12:46:38
Results
Nets
Components
Copper layers
Simulation
Spreadsheet View
17:22
26-07-2021


Grapher View
File Edit View Graph Trace Cursor Legend Tools Help
DC Sweep
R1
1kΩ
V1
2V
common base transistor
DC Transfer Characteristic
Current (A)
500µ
400µ
300µ
200µ
100µ
0µ
-100µ
0.0
0.2
0.4
0.6
0.8
1.0
vv1 Voltage (V)
I(Q1[IC])
Selected Trace:I(Q1[IC])
common base transistor


Grapher View
File Edit View Graph Trace Cursor Legend Tools Help
DC Sweep
common base output
DC Transfer Characteristic
Current (A)
8m
6m
5m
4m
3m
2m
1m
-1m
0
5
10
15
20
25
30
vv1 Voltage (V)
vv2=0; I(Q1[IC])
vv2=2; I(Q1[IC])
vv2=4; I(Q1[IC])
vv2=6; I(Q1[IC])
Selected Trace:vv2=0; I(Q1[IC])
R1
1kΩ

④ Now connect the base to the power source connection and ground the circuit.
⑤ Now click on simulation, go on analyse and simulation select DC sweep for input characteristic change V_1, start value 0, stop value 1 and increment of 0.5. Now click on output and select I (Q1[IC]) and now run, the simulation.
⑥ For the output characteristic, go to DC sweep again and now in V_1, start value = 0V, stop value = 30V, Increament = 2V
Now select use both source.
For V_2, start value = 0V, stop value = 7V, Increment = 2V
In output, select I(Q1(IC)) and now run the simulation.

Result- The desired input-output characteristics would be obtained on DC Transfer characteristic for common base output and input respectively.

Precautions-
① Make the circuit properly with all the connection done properly.
② Do not forget to ground the circuit
③ Change the respective values of V_1 and V_2 in the simulation.
④ Note the output carefully.

Teacher's Signature

EXPERIMENT-14

Aim - To plot input and output characteristic of common emitter transistor.

Software Used - MULTISIM

Theory - In Common Emitter (CE) configuration, the emitter is the common terminal. Hence the output between the base and emitter while output is between the collector and the emitter.

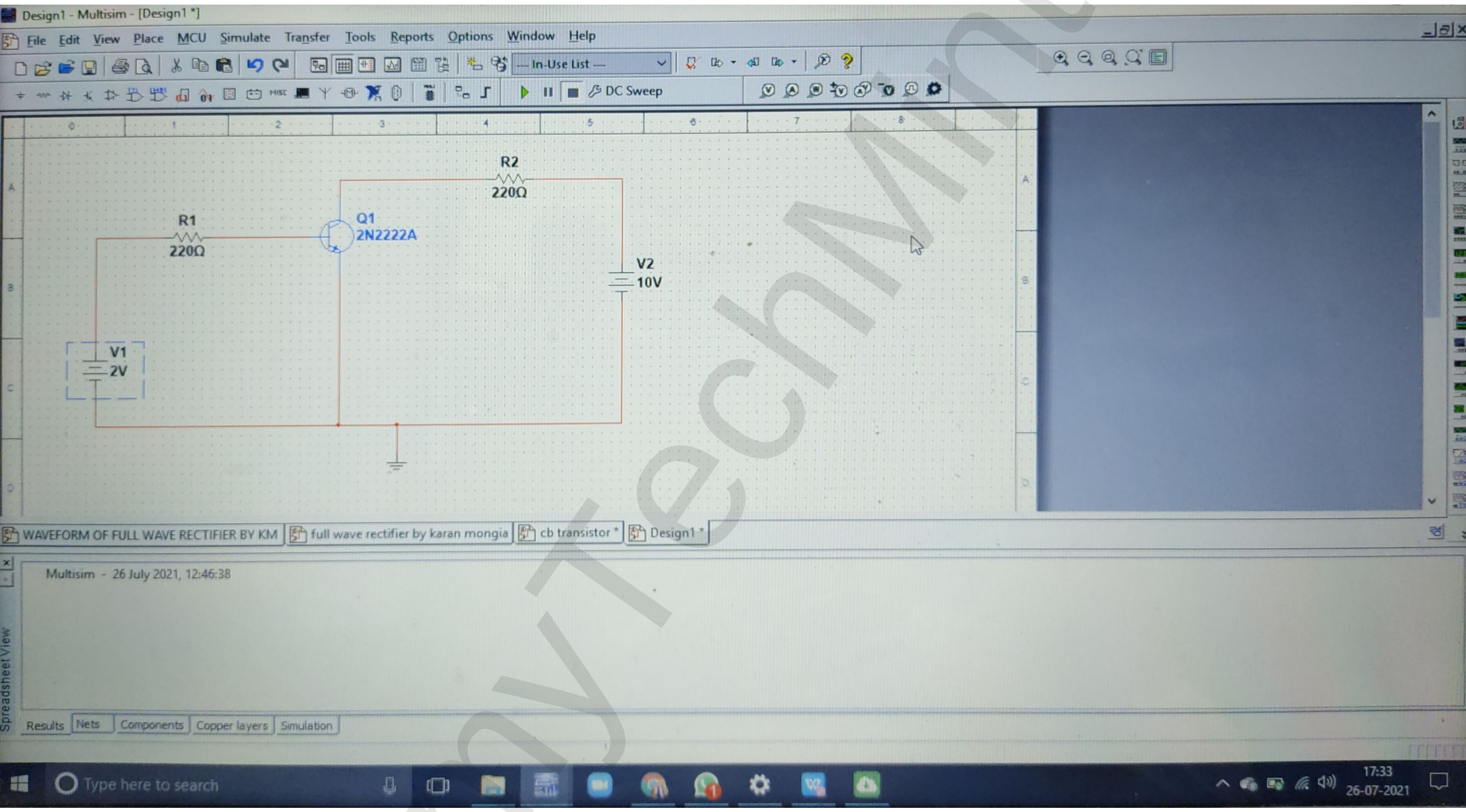
- **Input Characteristics** - The variation of the Base Current (I_B) with base emitter voltage (V_{CE}).
- **Output Characteristics** - The variation of collector current (I_C) with common-emitter voltage (V_{CE}).

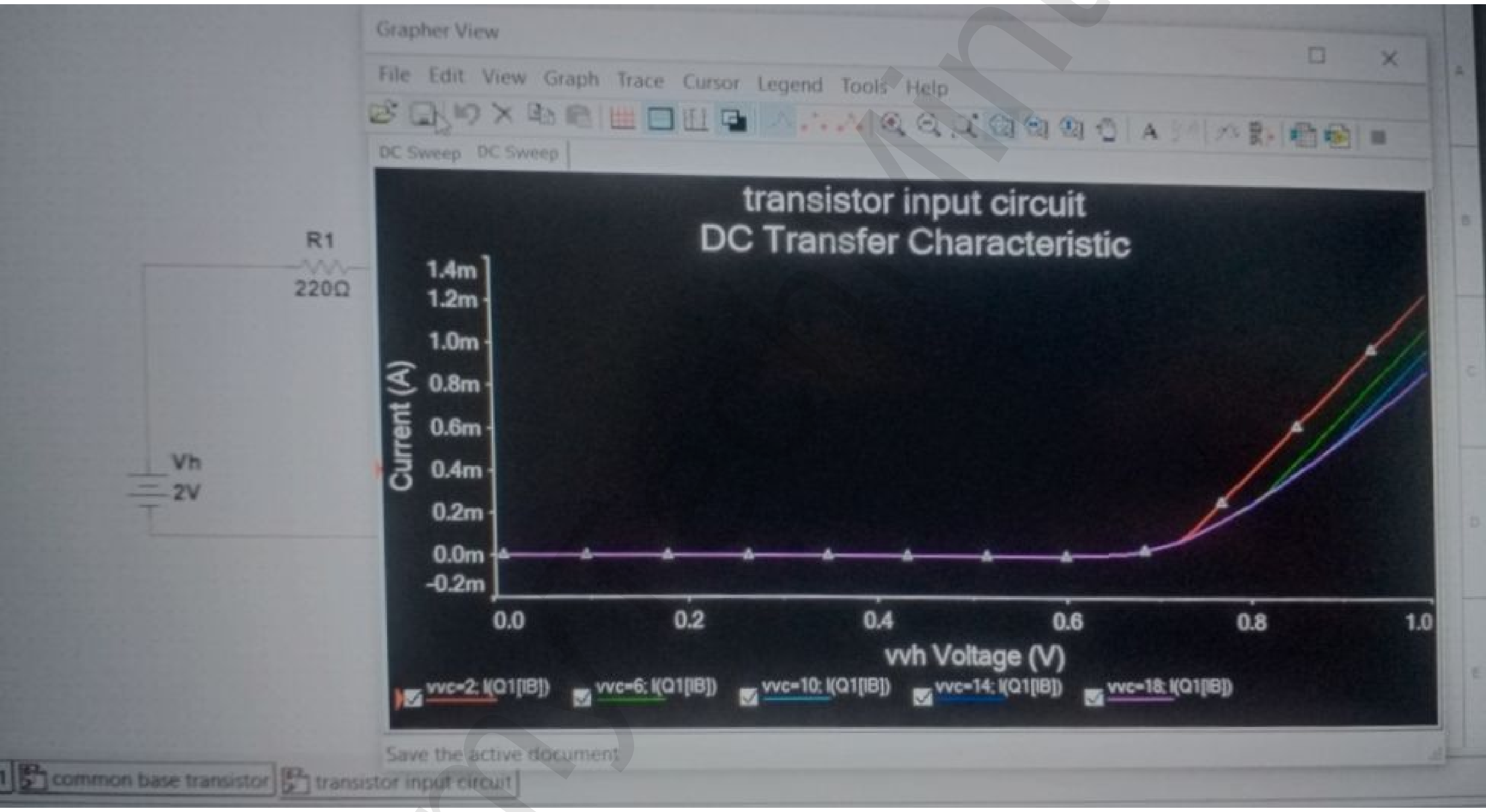
It is observed that the output characteristics are controlled by the input characteristics. Hence, the collector current changes with the base ~~current changes~~ current.

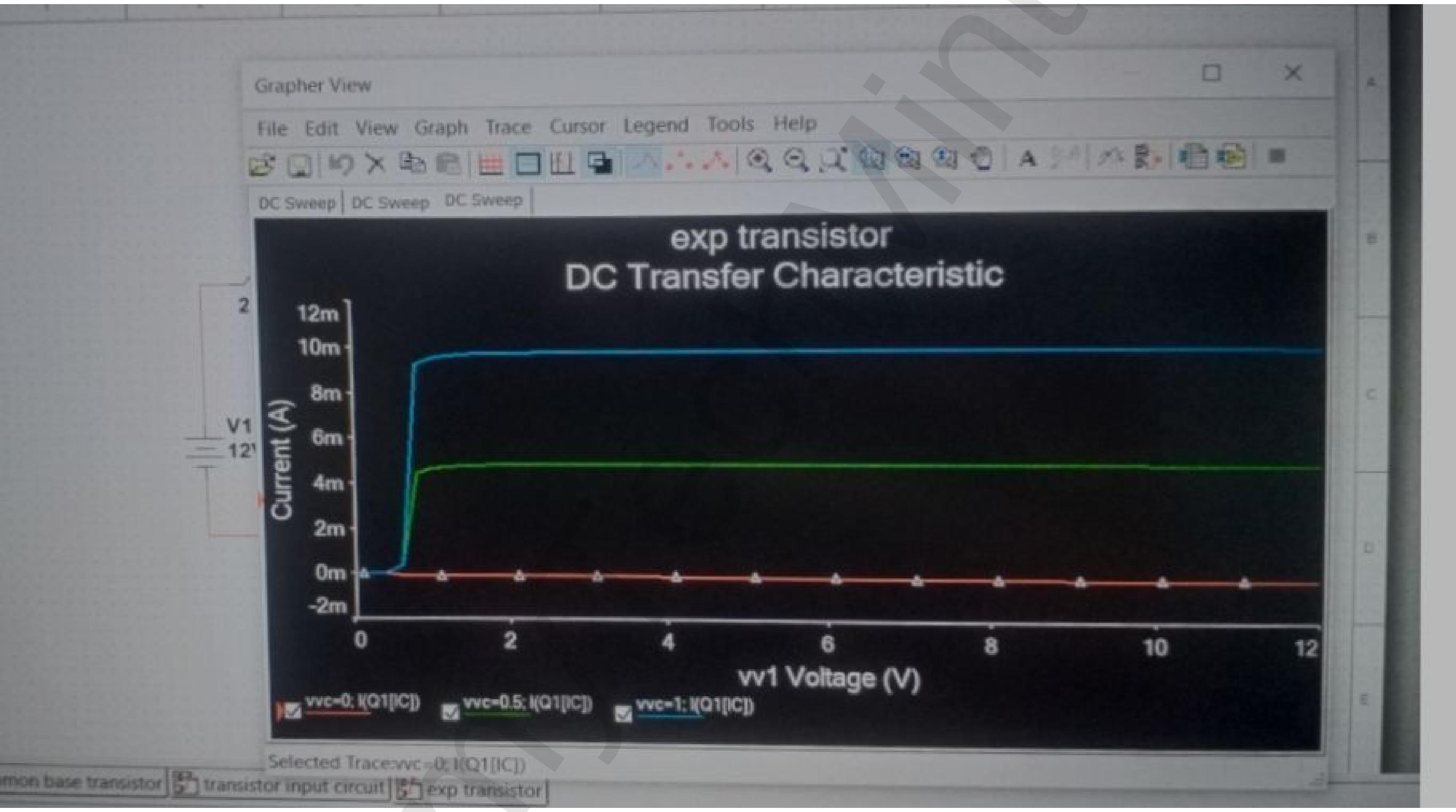
Procedure -

1. Make proper circuit with transistor, dc power source, resistance and ground. Set $V_1 = 12V$ and $V_2 = 10V$.
2. Now connect DC power source in input and in output.
3. Connect the resistances in input and output side of the transistor.
4. Connect one end of resistor to common emitter type transistor other to power source.
5. Connect other resistor with the collector and other to power source.
6. Now connect emitter to power source connection, then ground the circuit.
7. For input characteristic, click on simulation and go to analyse and simulation, select DC sweep, now adjust V_1, start value = 0V, stop value = 12V, Increment = 0.2V
8. For output characteristic, select (IQ1 [IC]) in output and then run simulation.
9. Now for common emitter output characteristics, select DC sweep, follow step 7, Then select use of source 2, start value = 0V, stop value = 1V, Increment = 0.5V

Teacher's Signature


Design1 - Multisim - [Design1 *]
File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help
--- In-Use List ---
DC Sweep
R2
220Ω
R1
220Ω
Q1
2N2222A
V2
10V
V1
2V
WAVEFORM OF FULL WAVE RECTIFIER BY KM
full wave rectifier by karan mongia
cb transistor *
Design1 *
Multisim - 26 July 2021, 12:46:38
Spreadsheet View
Results
Nets
Components
Copper layers
Simulation
Type here to search
17:33
26-07-2021


Grapher View
File Edit View Graph Trace Cursor Legend Tools Help
DC Sweep DC Sweep
transistor input circuit
DC Transfer Characteristic
Current (A)
vvh Voltage (V)
vvc=2: I(Q1[IB])
vvc=6: I(Q1[IB])
vvc=10: I(Q1[IB])
vvc=14: I(Q1[IB])
vvc=18: I(Q1[IB])
R1
220Ω
Vh
2V
Save the active document
common base transistor
transistor input circuit


Grapher View
File Edit View Graph Trace Cursor Legend Tools Help
DC Sweep
DC Sweep
DC Sweep
exp transistor
DC Transfer Characteristic
Current (A)
12m
10m
8m
6m
4m
2m
0m
-2m
0
2
4
6
8
10
12
vv1 Voltage (V)
vvc=0; I(Q1[IC])
vvc=0.5; I(Q1[IC])
vvc=1; I(Q1[IC])
Selected Trace:vvc=0; I(Q1[IC])
transistor input circuit
exp transistor

in output, select [IQ, (IC)] and run simulation.

Result - The desired input, output characteristic is obtained on the Grapher view and both the graph are been obtained clearly.

Precautions-

1. Make connection properly with all sources.
2. Do not forget to ground the circuit.
3. Simulate circuit properly with all correct values in the procedure.
4. Note input-output characteristics properly and adjust the reading properly.
5. Simulate with all correct values.

Teacher's Signature

EXPERIMENT- 15

Aim- To study and verify the truth table of Basic logic Gates.

Software Used - MULTISIM

Theory - A digital logic gate is an electronic device that makes logical decisions based on different combination of digital signals present on its input logic gates. They are referred as Transistor - Transistor logic or TTL gates.
Basic logic Gates are AND, OR, NOT, EXOR.

AND GATE - A multi-input circuit which the output 1 only if all inputs are 1. The representation is in figure 1. It gives high output (1) given by A.B.

OR GATE - A multi-input circuit that gives a high output 1. A plus(+) is used to show OR operations.

NOT GATE - The output is 0 when input is 1 and the output is 1 when input is 0. The symbolic 'A' or A with bar over the top ($\bar{A}$).

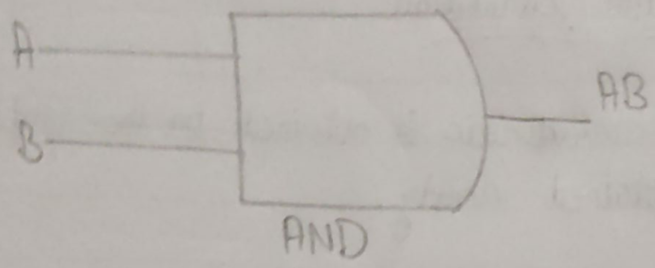
EXOR GATE - The output of exclusive OR gate, is 0 when it has 2 inputs as same and its output is 1 when it's 2 inputs are different called as Anti-coincidence gate, represented at fig. ④

Procedure -

① Go to TTL, in the component bar and select desired TTL for AND, OR, NOT, EXOR gates respectively 7408N, 7432N, 7404N, 7486N.

② Now click on place source and go to all components and search the Integrated Digital Constant and add it for all gates respectively for desired gate

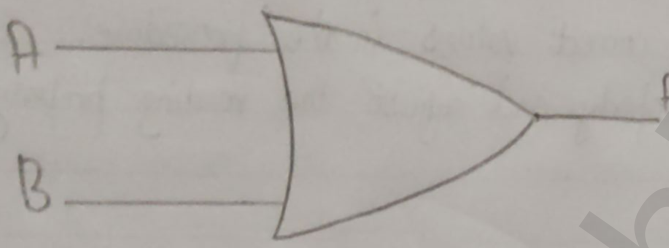
AND Gate



Truth Table

A	B	A.B
0	0	0
0	1	0
1	0	0
1	1	1

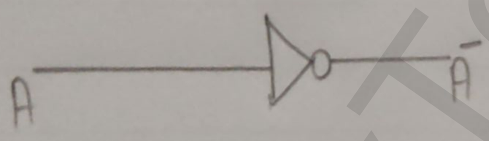
OR Gate



Truth Table

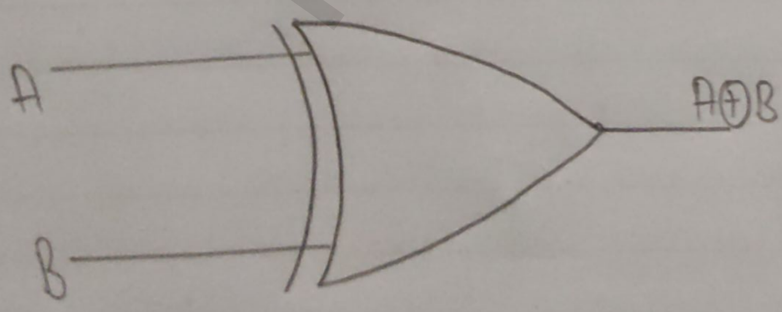
A	B	A+B
0	0	0
0	1	1
1	0	1
1	1	1

NOT Gate

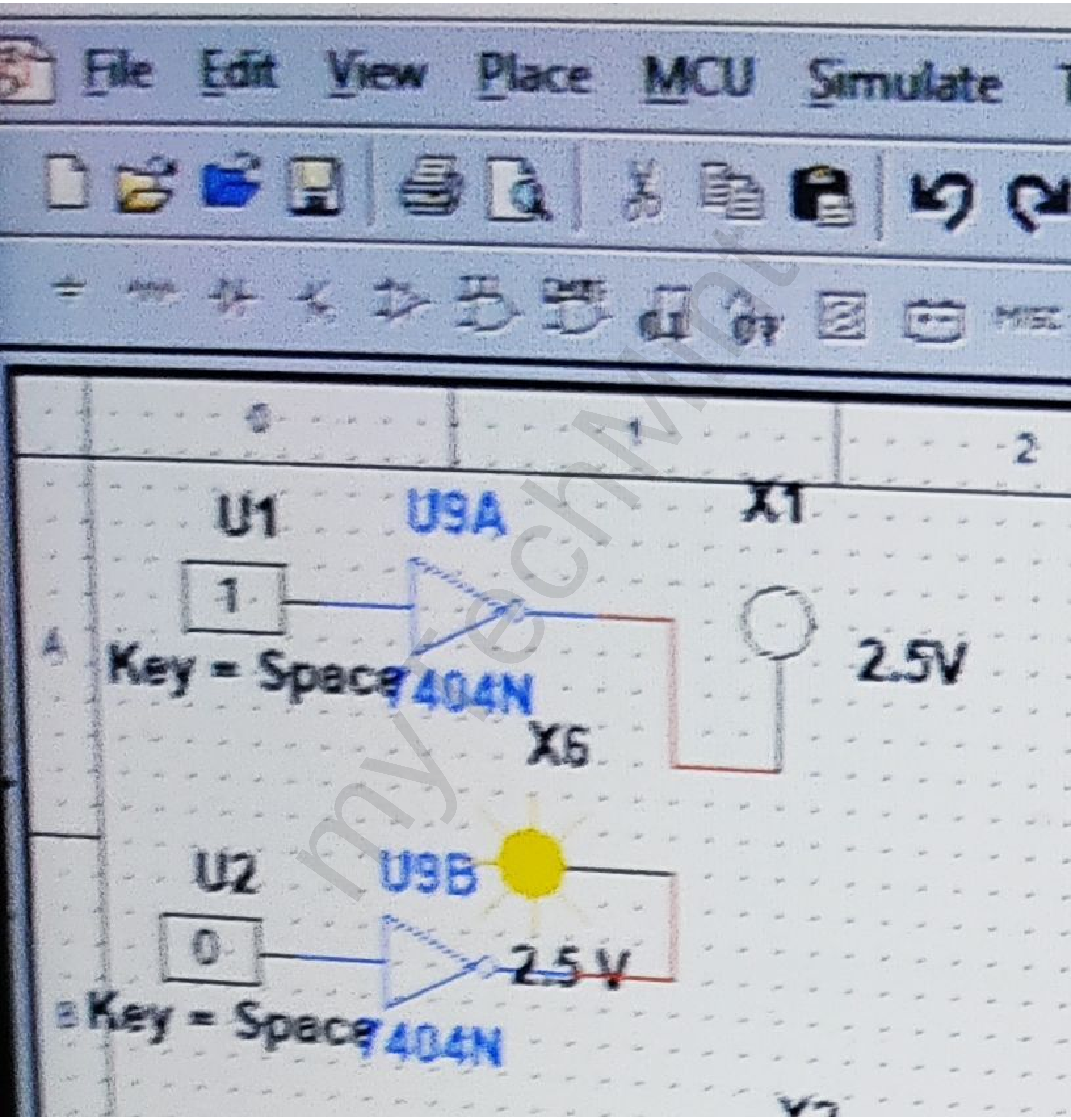


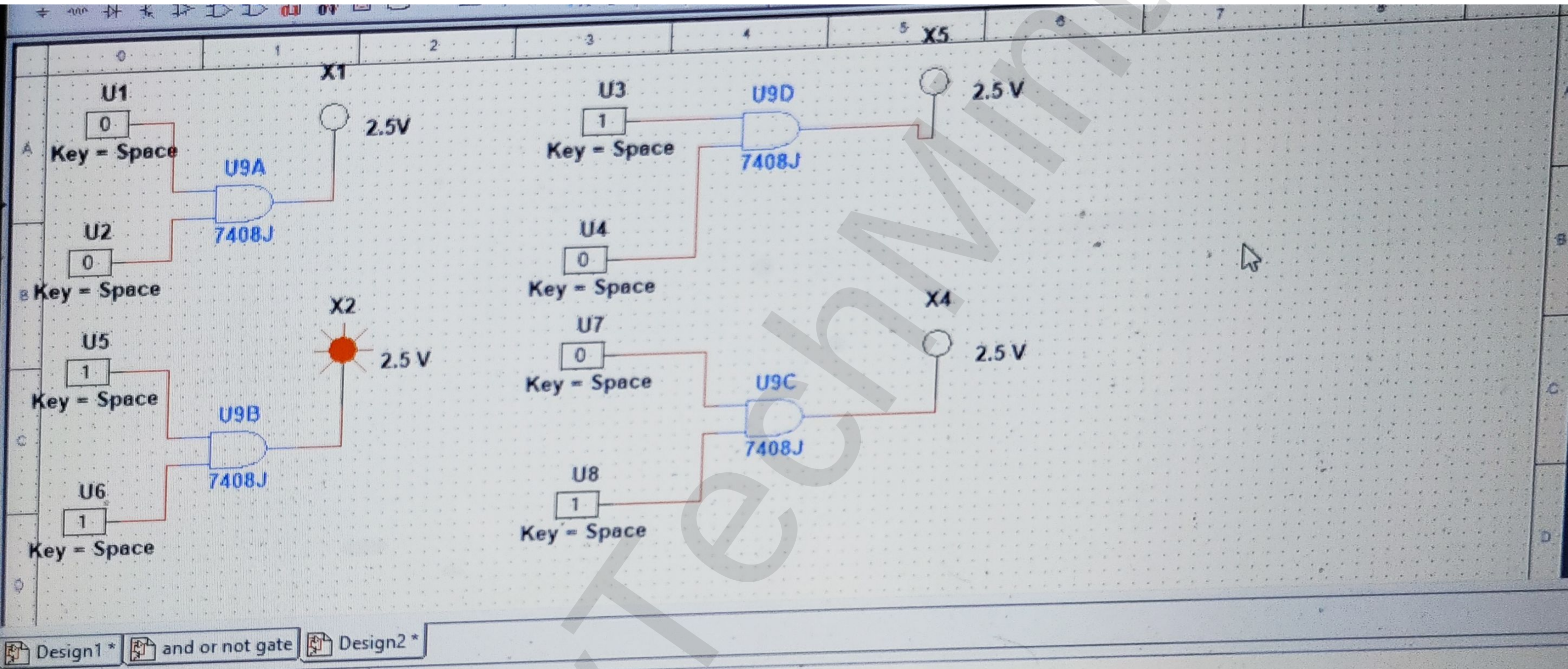
A	$\bar{A}$
0	1
1	0

EXOR Gate

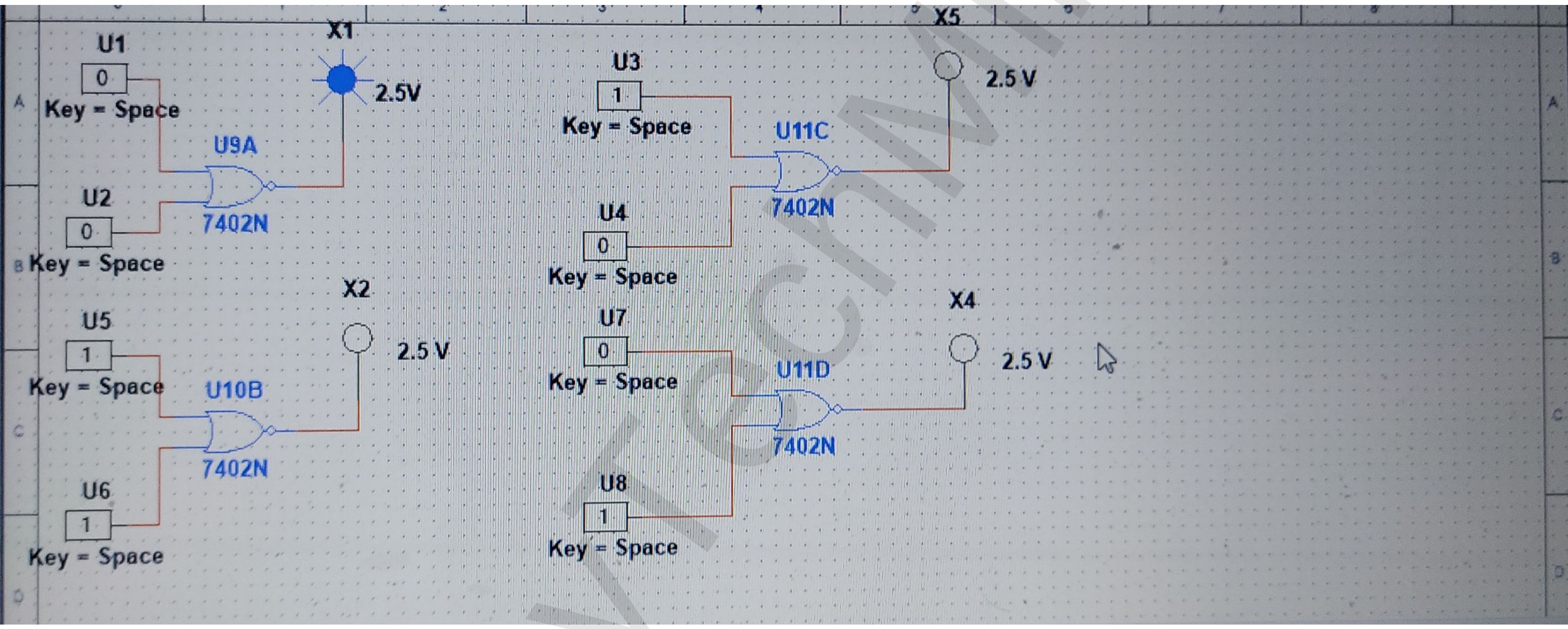


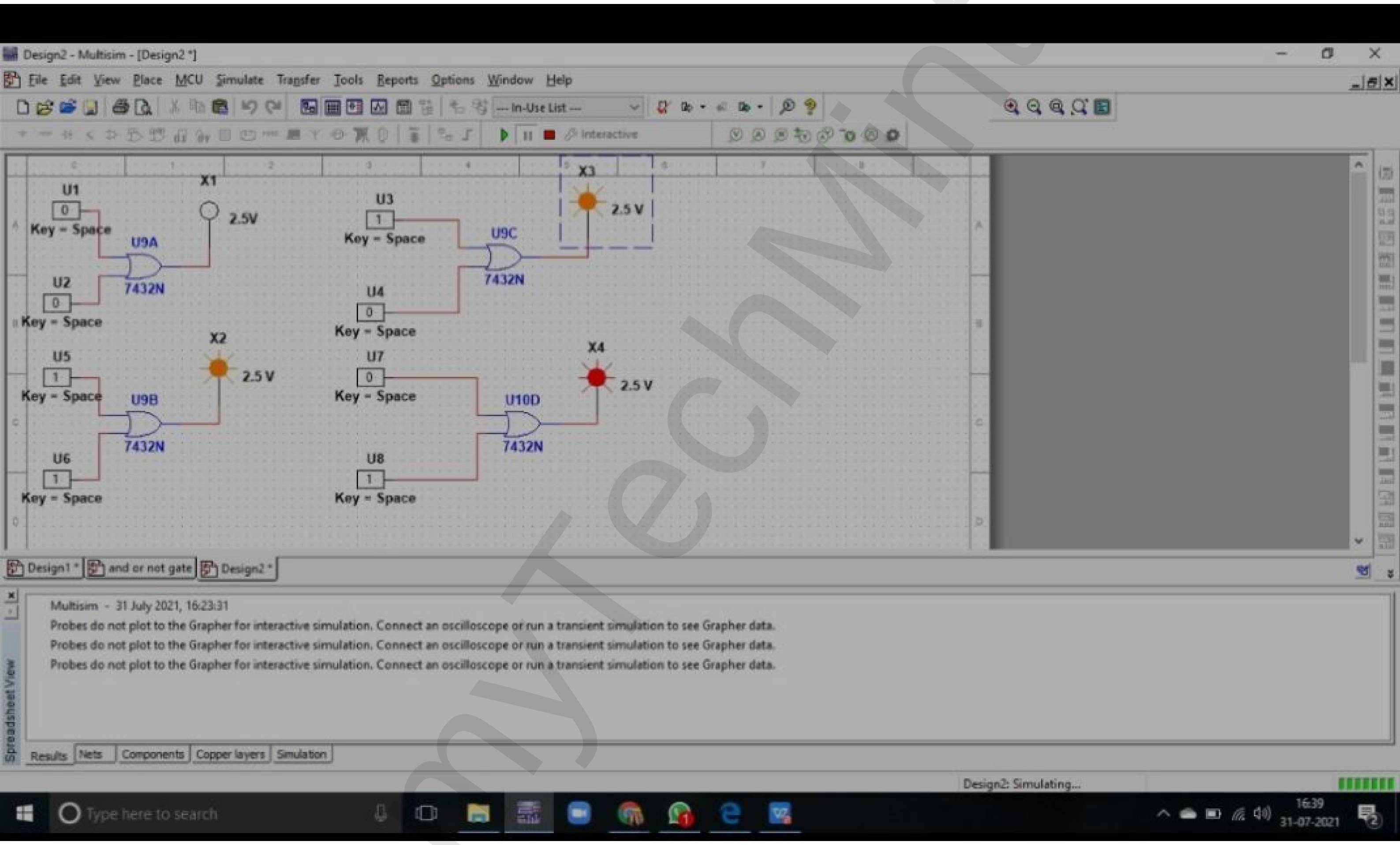
A	B	$A \oplus B$
0	0	0
0	1	1
1	0	1
1	1	0


File
Edit
View
Place
MCU
Simulate
U1
U9A
X1
1
Key = Space
7404N
2.5V
X6
U2
U9B
0
2.5 V
Key = Space
7404N



Multisim - 31 July 2021, 16:23:31
Probes do not plot to the Grapher for interactive simulation. Connect an oscilloscope or run a transient simulation to see Grapher data.
Probes do not plot to the Grapher for interactive simulation. Connect an oscilloscope or run a transient simulation to see Grapher data.
Probes do not plot to the Grapher for interactive simulation. Connect an oscilloscope or run a transient simulation to see Grapher data.
Probes do not plot to the Grapher for interactive simulation. Connect an oscilloscope or run a transient simulation to see Grapher data.
Probes do not plot to the Grapher for interactive simulation. Connect an oscilloscope or run a transient simulation to see Grapher data.
Probes do not plot to the Grapher for interactive simulation. Connect an oscilloscope or run a transient simulation to see Grapher data.


U1
0
Key = Space
X1
2.5V
U9A
7402N
U2
0
Key = Space
X2
2.5 V
U5
1
Key = Space
U10B
7402N
U6
1
Key = Space
U3
1
Key = Space
U11C
7402N
U4
0
Key = Space
U7
0
Key = Space
U11D
7402N
U8
1
Key = Space
X5
2.5 V
X4
2.5 V


Design2 - Multisim - [Design2 *]
File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help
U1
Key = Space
X1
2.5V
U9A
7432N
U2
Key = Space
U3
Key = Space
U9C
7432N
U4
Key = Space
X3
2.5 V
U5
Key = Space
U9B
7432N
U6
Key = Space
X2
2.5 V
U7
Key = Space
U10D
7432N
U8
Key = Space
X4
2.5 V
Design1 * and or not gate Design2 *
Multisim - 31 July 2021, 16:23:31
Probes do not plot to the Grapher for interactive simulation. Connect an oscilloscope or run a transient simulation to see Grapher data.
Probes do not plot to the Grapher for interactive simulation. Connect an oscilloscope or run a transient simulation to see Grapher data.
Probes do not plot to the Grapher for interactive simulation. Connect an oscilloscope or run a transient simulation to see Grapher data.
Results Nets Components Copper layers Simulation
Design2: Simulating...
Type here to search
16:39
31-07-2021

③ Now again go to place source all components and search PROBE and desired digital LED light source which ever colour you wish.
④ Now connect the Interactive Digital Constant to Gates and connect the PROBE to the Gates
⑤ The desired output, click on the digital constant to change ~~output~~ values from 0 to 1 or vice versa.
⑥ Now click on RUN to get output

Result – The desired output will be shown on the screen and now by changing values, you can obtain desired truth table of desired circuit by glowing PROBES

Precautions -
① Make proper connection of gates.
② Give inputs carefully to get proper output
③ Use digital probe for output.
④ Obtain Truth table carefully.

EXPERIMENT-16

Aim - To study and verify truth table of Universal Gates.

Software Used - Multisim

Theory - Universal Gate is a gate which can implement any function without use of any other gate types. NAND and NOR gates are universal gates. They are economical and easier to fabricate and are base gates used in all IC Digital Logical families.

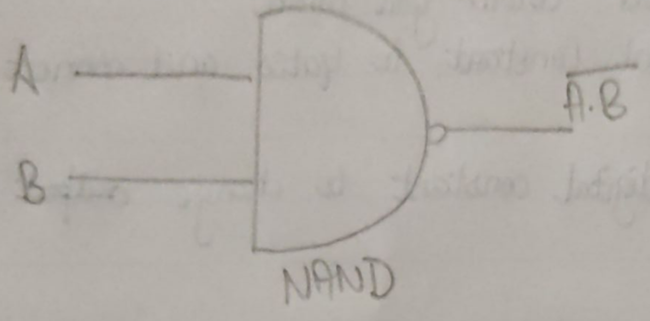
NAND gate - It is AND followed by invert. This is a NOT-AND Gate equal to AND gate followed by NOT Gate. The output of all NAND gates is high if any of the input is low, represented as $\overline{A.B}$.

NOR gate - It is OR followed by Inverter. This is NOT-OR gate equal to OR-gate followed by NOT gate. The outputs of NOR gates are low if any of the input is high. The symbol is OR gate with small circle on output, represented by $\overline{A+B}$.

Procedure -

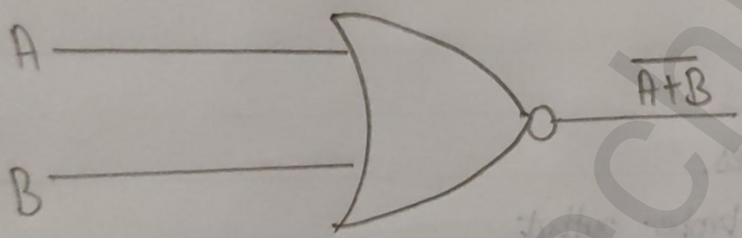
1. Go to TTL in the source bar and select desired TTL for NAND and NOR, 7400N and 7402N respectively.
2. Now click on place source, select all components and search Interactive Digital Constant connect it to the desired requirement of Gate.
3. Now again click on place source, select all components and search PROBE, add PROBE to output of the gate. Connect only digital Probe.
4. Now connect Interactive Digital Constant to input of the gate.
5. For changing value of input from 0 to 1, or vice-versa, click on digital constant.

NAND Gate

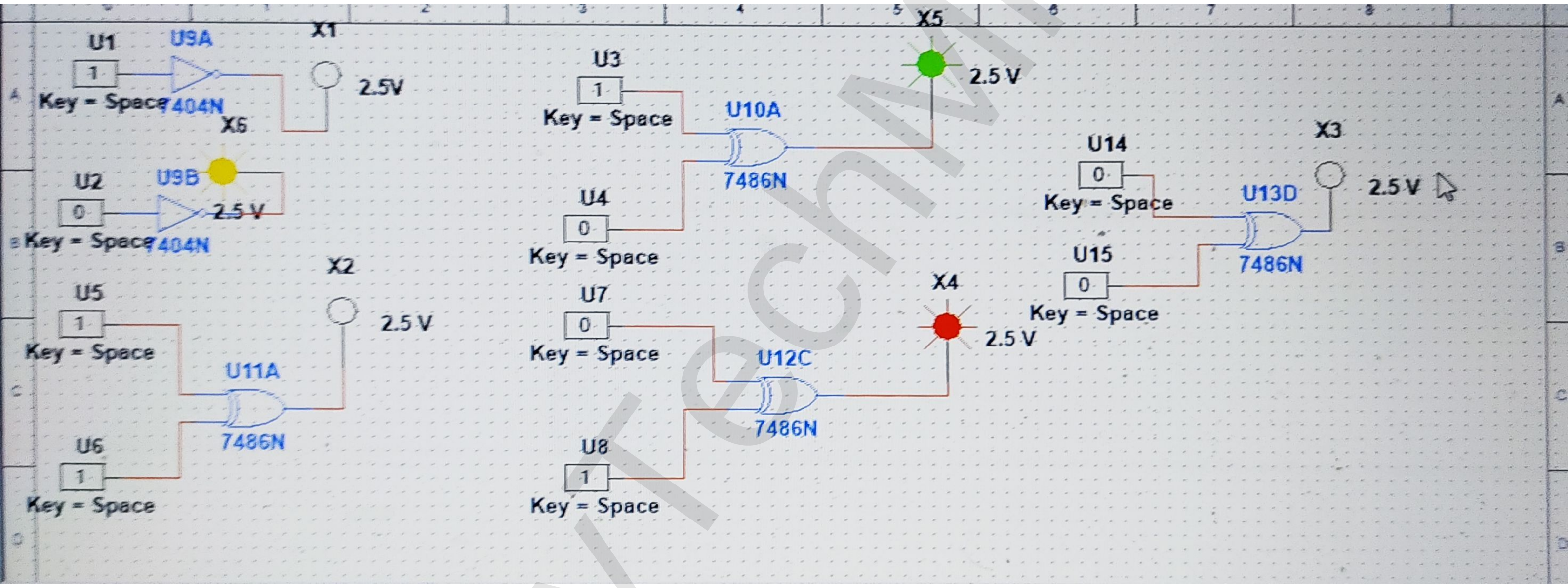


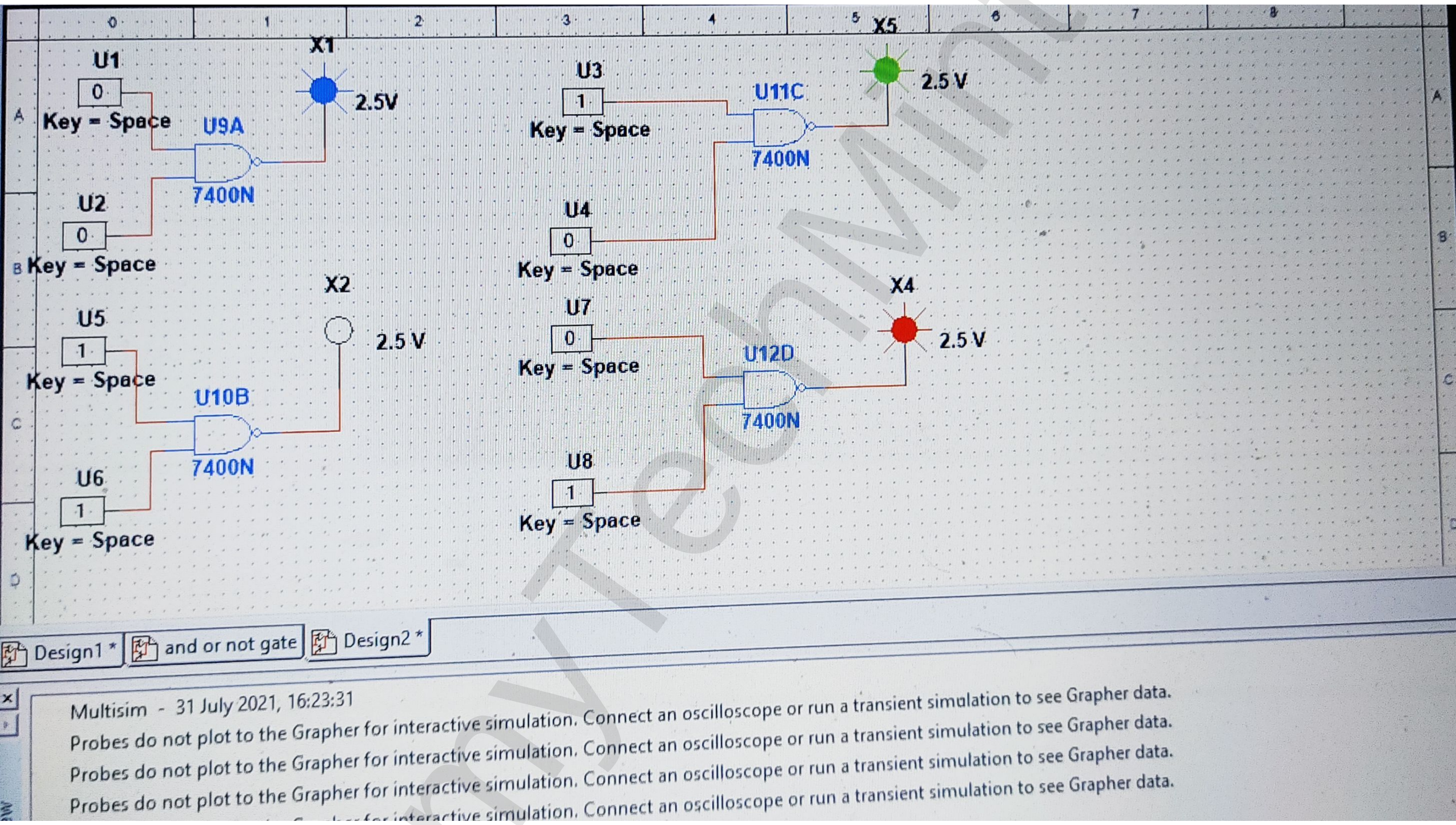
A	B	$\overline{A.B}$
0	0	1
0	1	1
1	0	1
1	1	0

NOR Gate



A	B	$\overline{A+B}$
0	0	1
0	1	0
1	0	0
1	1	0


U1
U9A
X1
1
Key = Space
7404N
2.5V
X6
U2
U9B
0
2.5 V
Key = Space
7404N
X2
U5
2.5 V
1
Key = Space
U11A
U6
7486N
1
Key = Space
U3
X5
1
2.5 V
Key = Space
U10A
7486N
U4
0
Key = Space
U7
X4
0
Key = Space
U12C
2.5 V
7486N
U8
1
Key = Space
U14
X3
0
2.5 V
Key = Space
U13D
U15
7486N
0
Key = Space


U1
0
Key = Space
X1
2.5V
U9A
7400N
U2
0
Key = Space
X2
2.5 V
U5
1
Key = Space
U10B
7400N
U6
1
Key = Space
U3
1
Key = Space
U11C
7400N
X5
2.5 V
U4
0
Key = Space
U7
0
Key = Space
X4
2.5 V
U12D
7400N
U8
1
Key = Space
Design1 *
and or not gate
Design2 *
Multisim - 31 July 2021, 16:23:31
Probes do not plot to the Grapher for interactive simulation. Connect an oscilloscope or run a transient simulation to see Grapher data.
Probes do not plot to the Grapher for interactive simulation. Connect an oscilloscope or run a transient simulation to see Grapher data.
Probes do not plot to the Grapher for interactive simulation. Connect an oscilloscope or run a transient simulation to see Grapher data.
Probes do not plot to the Grapher for interactive simulation. Connect an oscilloscope or run a transient simulation to see Grapher data.

⑥ Now click on RUN, then desired output will be on the screen.

Result - The Digital PROBE will glow on correct input according to Gate characteristics, and output will be obtained by glowing PROBES.

Precautions-

① Make connections properly
② Use correct TTL for NAND and NOR .
③ Use digital probe for output.
④ Note truth table carefully.