Dr. Ambedkar Institute of Technology, Bengaluru-56

(An Autonomous Institution Affiliated to VTU, Belagavi)
(Accredited by NAAC with Grade "A")

Department of Electronics and Communication Engineering

ANALOG ELECTRONIC CIRCUIT PRACTICAL COMPONENT OF IPCC Manual

Prepared By

Dr. SHILPA K.C
Assistant Professor
Dept of ECE

Dr. AMBEDKAR INSTITUTE OF TECHNOLOGY, BENGALURU-560056

(An Autonomous Institution Affiliated TO VTU, BELAGAVI and Accredited by NAAC with Grade "A")

Department of Electronics and Communication Engineering

Vision Statement:

"To excel in education and research in Electronics and Communication Engineering and its related areas through its integrated activities for the society"

Mission Statement:

- To provide the high quality education in Electronics and Communication Engineering discipline and its related areas to meet the growing challenges of the industry and the society through research.
- To be a contributor to technology through value based quality technical education.
- To equip the students with strong foundations of Electronics and communication engineering.

Program Educational Objectives (PEOs):

PEO1: Graduates will have a solid foundation in electronics and communication engineering.
PEO2: Graduates are technically competent and able to analyze, design, develop and implement electronic and communication systems.
PEO3: Graduates will have sufficient breadth in electronics and its related fields so as to enable them to solve general engineering problems.
PEO4: Graduates are capable of communicating effectively and interact professionally with colleagues, clients, employers and the society.
PEO5: Graduates are capable of engaging in life - long learning and to keep themselves abreast of new developments in their fields of practice.

Program Outcomes (POs):

PO1. Exhibit the knowledge of differential equations, vector calculus, complex variables, matrix theory, probability theory, physics, chemistry and basics of electrical and electronics engineering.
PO2. Identify, formulate and solve Electronics & Communication Engineering problems.
PO3. Design Electronic circuits and conduct experiments with Digital and Analog discrete components and integrated circuits.
PO4. Design, verify and validate the digital and analog Electronic systems using simulation software and its implementation on real time applications.
PO5. Visualize, analyze and model complex engineering problems with multidisciplinary tasks using appropriate advanced processors and design tools.
PO6. Understand the impact of engineering solutions in a global and societal context.
PO7. Recognize the impact of engineering solution in legal, cultural and environmental contexts and need for sustainable development.
PO8. Apply the knowledge of professional ethics and responsibilities.
PO9. Demonstrate the management
PO10. Skills and leadership qualities to work as a team member and/or leader in multidisciplinary areas of engineering.
PO11. Communicate effectively through written and oral modes to all levels of society.
PO12. Graduate will develop confidence for self education and ability for lifelong learning
Graduate will be able to design, implement and validate the real time systems applied to physical world with financial constraints.

Sl. No.	Contents	No of Hours	BT Levels
	PART-B : Simulation using PSpice software		
1	Design and verification of voltage follower, inverting amplifier and non-inverting amplifier using OP-AMP.	2	L1- Remembering L2- Understanding L3- Applying L4- Analyzing
2	Design and verification of Integrator and Differentiator using OP-AMP.	2	L1- Remembering L2- Understanding L3- Applying L4- Analyzing
3	Design and Simulation of Function generator to generate square wave and triangular wave generator using OP-AMP.	2	L1- Remembering L2- Understanding L3- Applying L4- Analyzing
4	Analyze Input, Output characteristics of BJT Common emitter configuration and evaluation of parameters.	2	L2- Understanding L3- Applying L4- Analyzing L5- Evaluating
5	Design and Simulation of drain and gate characteristics of a JFET.	2	L1- Remembering L2- Understanding L3- Applying L4- Analyzing
6	Analyze of Static characteristics of SCR	2	L1- Remembering L2- Understanding L3- Applying L4- Analyzing

Course Outcomes:

CO1: Understand the characteristics of various electronic devices and measurement of parameters

CO2: Design and test simple electronic circuits.

CO3: Use of circuit simulation software for the implementation and characterization of electronic circuits and devices.

Course outcomes mapping with programme outcomes.

Cos	Mapping with POs
CO1	PO02,PO03, PO04
CO2	PO02,PO03, PO04
CO3	PO02,PO03, PO04

Note: *The experiments are to be carried using discrete components only.*

*****

Dos and Don'ts in Laboratory

1. Wear your College ID card.

2. Be regular to the Lab.

3. Do not come late to the Lab.

4. Keep your bags in the rack.

5. Take care of your valuable things.

6. Keep your work area clean.

7. Bring Observation book, Record and Manual along with pen, pencil, and eraser Etc., no borrowing from others.

8. Before entering to lab, must prepare for Viva for which they are going to conduct experiment.

9. Do not handle any equipment before reading the instructions/Instruction manuals.

10. Check CRO probe before connecting it.

11. Strictly follow the instructions given by the teacher/Lab Instructor.

12. Before switch on the power supply, must show the connections to one of the faculties or instructors.

13. Avoid loose connection and short circuits.

14. Do not panic if you do not get the output.

15. Do not throw connecting wires on the Floor.

16. Remove the Connections and Return the components to the respective lab instructors.

17. Before leaving the lab, switch off the power supplies of all equipments and keep chairs properly.

Record Writing Format

Unruled Side	*Ruled side*
Circuit Diagram(Pencil)	Date
Design(pen)	Title
Observation table(Pencil) and Entries (pen)	Aim
Expected Graph/ Response(Pencil)	Components required
	Theory
	Procedure
	Result

*****

III Semester ANALOG ELECTRONIC CIRCUITS Course Code: 22ECU303 CIE Marks: 50 Teaching Hours/Week (L:T:P:S): 3:0:2:0 SEE Marks: 50 Total Hours of Pedagogy: 52 Total Marks: 100 Credits: 4 Exam Hours: 3

PRACTICAL COMPONENT OF IPCC 1. Design and verification of voltage follower, inverting amplifier and non-inverting amplifier using OP-AMP.

2. Design and verification of Integrator and Differentiator using OP-AMP.

3. Design and Simulation of Function generator to generate square wave and triangular wave generator using OP-AMP.

4. Analyze Input, Output characteristics of BJT Common emitter

5. Design and Simulation of input and output characteristics of MOSFET

6. Analyze of Static characteristics of SCR

7. Design and Simulation of RC Phase shift Oscillator using FET.

8. Design and Simulation of R-2R Ladder DAC

PART-B: Simulation using PSpice software

Experiment No. B.1

B. 1: *Design and verification of voltage follower, inverting amplifier and non-inverting amplifier using OP-AMP*

B. 1.1: Voltage Follower

Aim: - Design and Verification of OP-AMP as Voltage Follower

Circuit Diagram: OPAMP as Voltage Follower
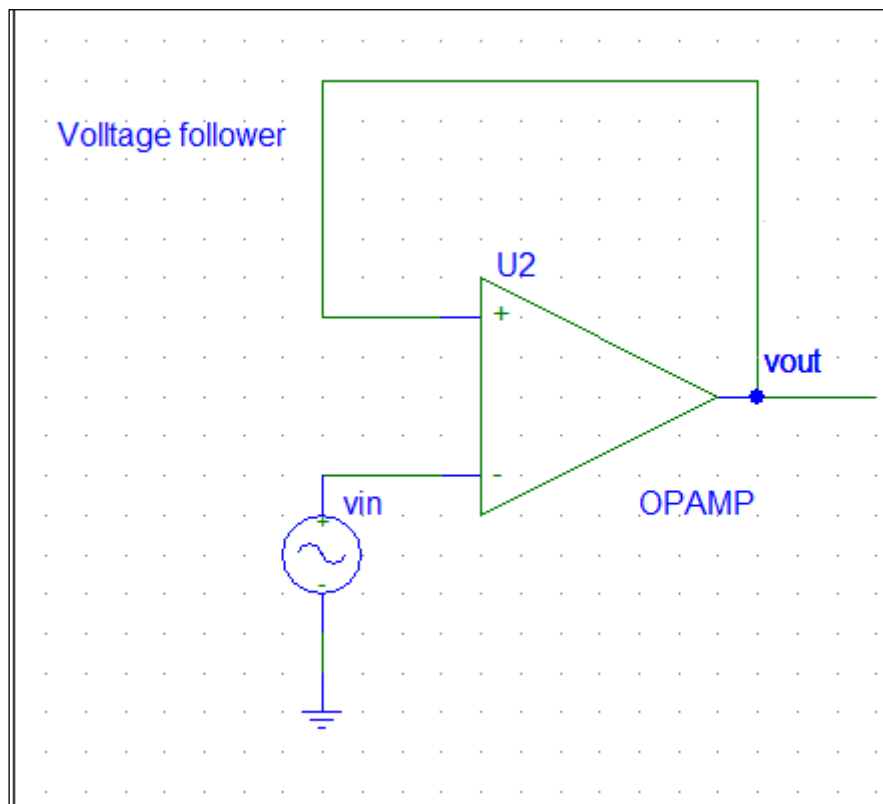
Figure 1.1: Voltage Follower Circuit

Steps to build the Circuit Diagram in PSpice simulation:
1. Click on the *Schematics icon* on the Desktop to open the Simulation window.

2. The Pspice Schematic window is as shown below
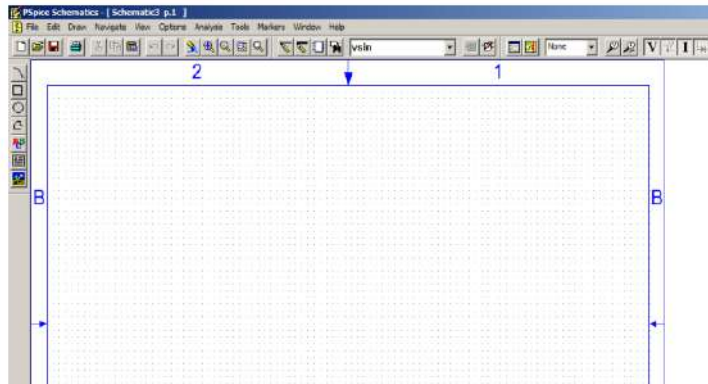
3. Press **Clt+G** (*Parts Browser Basic*) in *Part Name* type **uA741**, and click *Place and Close* option in *Part Browser Basic,* to place Op-amp in the schematic window.

4. Press *Clt+G (Parts Browser Basic)* to select *input sine wave*, in the *Part Name* type *vsin* , and click *Place and Close* option in *Part Browser Basic*, to place *vsin* schematic window.
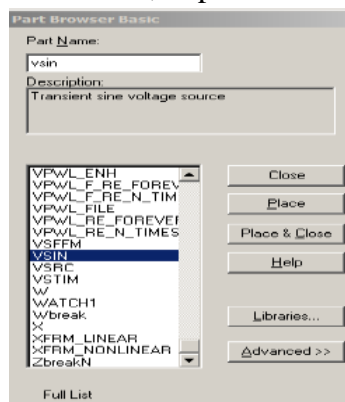
vsin icon

5. Double click on the *vsin icon* to add the following parameter.
 (*Note: Each time adding the parameter always press, Save Attr*)
   a. DC =0

b. AC =0
c. VOFF=0
d. VAMPL =0.5v
e. FREQ = 50Hz
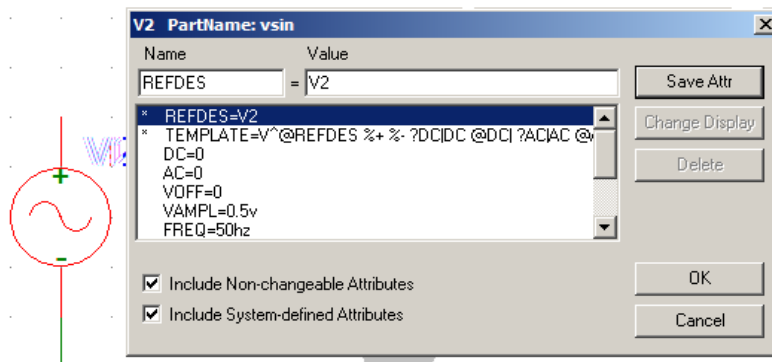
6. Select *Draw wire* option from menu bar to connect wire .
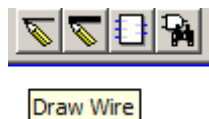
7. Select *Draw wire* option from menu bar to connect wire between the positive of the sine wave is connected to the non-inverting terminal of op-amp *(' – sign' is non- inverting terminal of op-amp)* and negative sign to ground, using *draw wire option*, connect the wires between the positive input terminal and negative ground terminal .

8. To get ground icon, Press *Clt*+**G** (**Parts Browser Basic**) to select *ground icon* in the *Part Name* type *gnd_earth* , and click *Place and Close* option in *Part Browser Basic*, to place ground in schematic window

Ground Icon
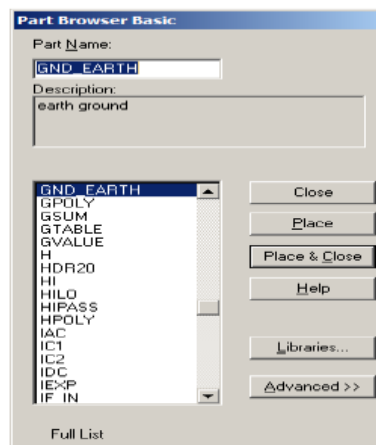
9. Using *draw wire* option, connect the wires between the inverting terminal of op-amp **(' + *sign' is inverting terminal of op-amp)**   to the output of opamp, connect a wire using draw wire at output, to set as output label point.

10. Double click on the output label point to label as *vout.*

11. To perform simulation, the **transient analysis** must be setup.

12. Click on *Analysis* in the *menu bar*, select *Setup option*.
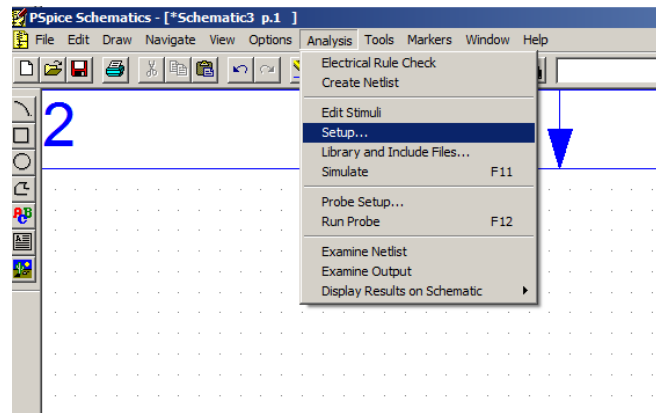
13. In *Analysis Setup*, click on *Transient label*.
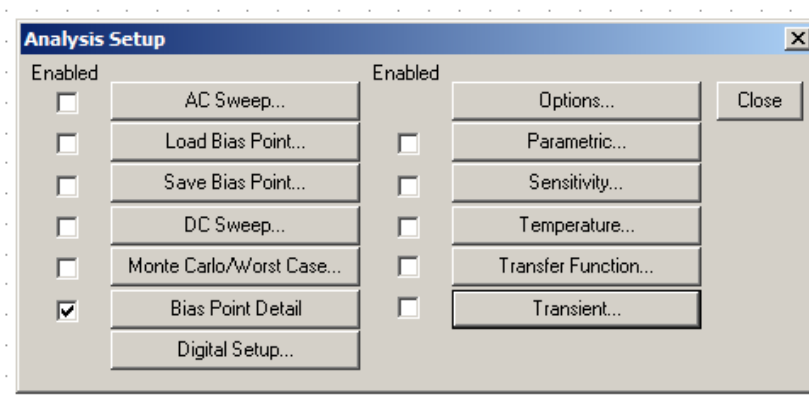
14. In *Transient window*, type

   a.   Print Step = 0

   b.   Final time = 50 ms
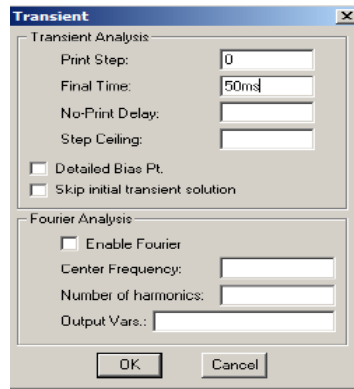
15. Save the schematic file before the simulating.

16. For *simulation*, Click on *Analysis* in the menu bar, select Simulate option.
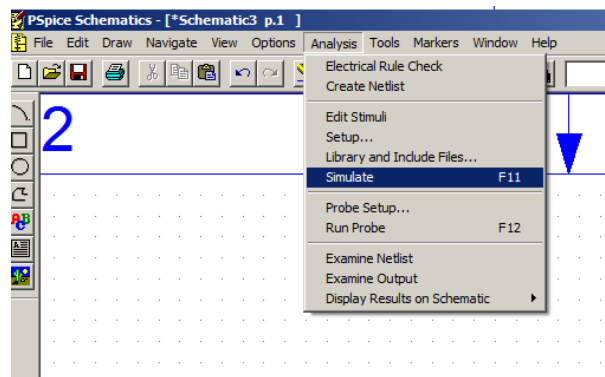
17. Next Click on *Trace* in the menu bar, select *Add Trace*.
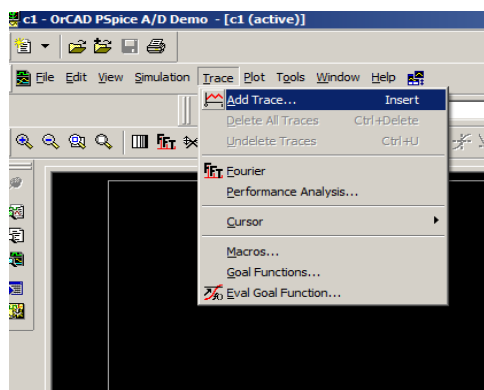
18. Add Trace Window displays the input and output label used in the circuit diagram.

19. Select the input label as V(V2 +) used in the circuit diagram to obtain the input sine wave.

20. Select the output label as V(vout) used in the circuit diagram to obtain the output sine wave.

21. The output wave follows the input wave, the amplitude of the output to the input wave is the same, and hence gain = unity
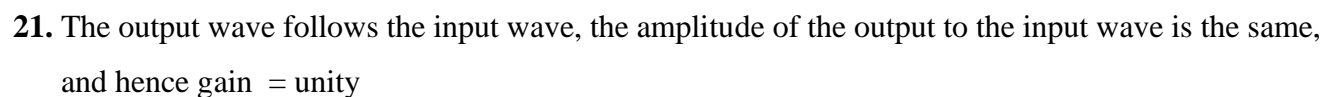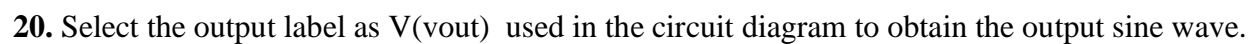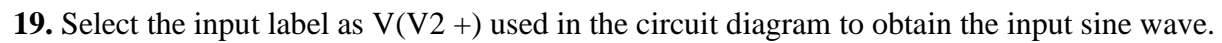
$$A_v = \left(\frac{V_{out}}{V_{in}}\right)$$

$$A_v = 1$$

22. The circuit opamp as voltage follower is simulated.

B. 1.2: Inverting Amplifier

*Aim: - Design and verification of OP-AMP as **Inverting Amplifier***

Circuit Diagram: <u>OPAMP as Inverting Amplifier</u>
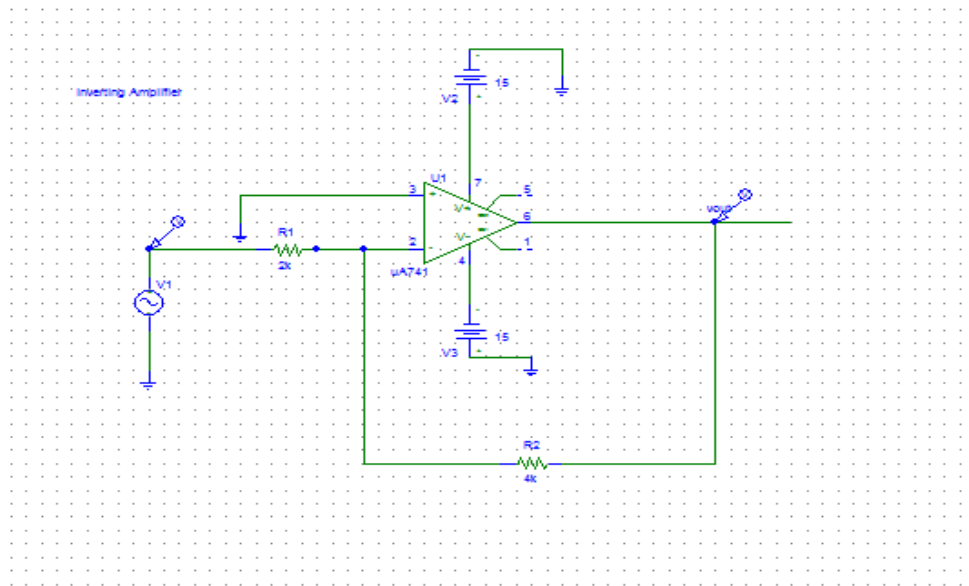
Figure 1.2: Inverting Amplifier Circuit

Design:

$$Let : R_1 = (10K\Omega)$$

$$A_V = 5$$

$$A_V = \left(-\frac{R_f}{R_1}\right)$$

$$R_f = 50K\Omega$$

*Steps to **build the *Circuit Diagram in PSpice simulation:***

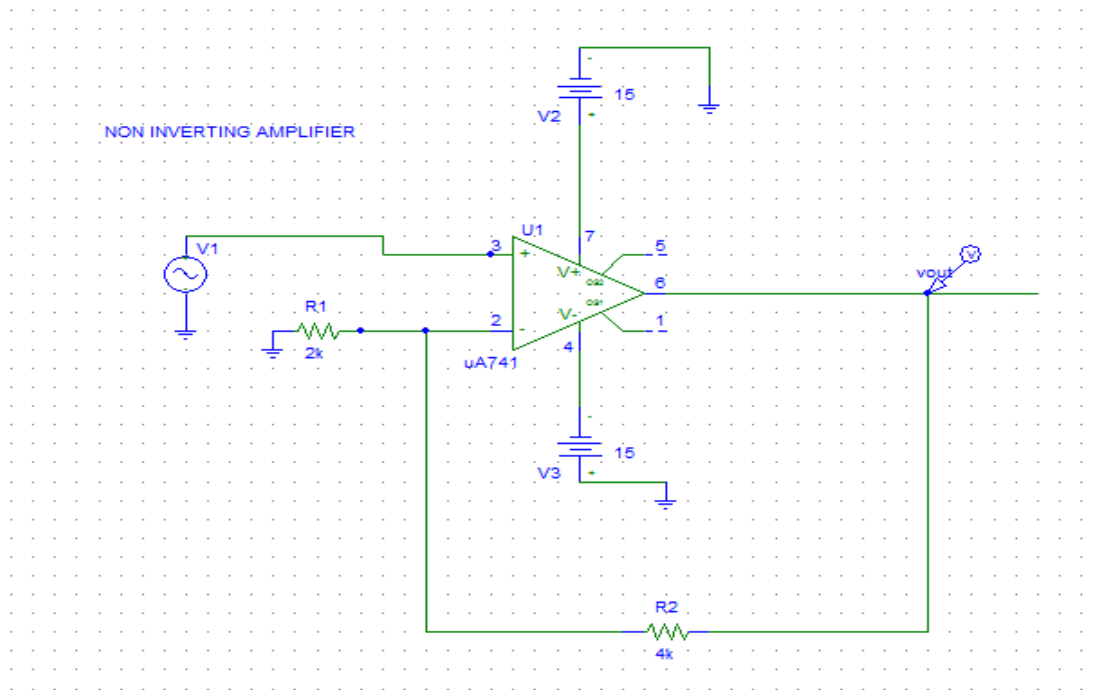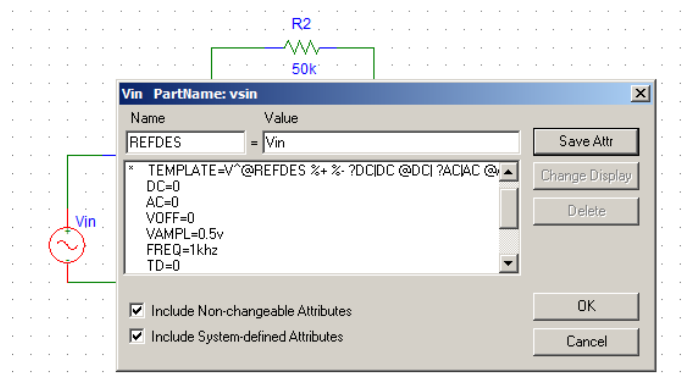1. Press Clt+G *(Parts Browser Basic)* in *Part Name* type *ua741,* and click *Place and Close* option in *Part Browser Basic*, to place Op-amp in the schematic window.
2. Press *Clt+G (Parts Browser Basic)* to select *input sine wave,* in the *Part Name* type *vsin* , and click *Place and Close* option in *Part Browser Basic*, to place vsin schematic window.
3. Double click on the *vsin icon* to add the following parameter

   (Note: Each time adding the parameter always press, Save Attr)
   Name of the input sine wave = Vin
   a. DC =0

   b. AC =0

   c. VOFF=0

   d. VAMPL =0.5v

   e. FREQ = 1KHz

4. The positive of the sine wave is connected to the inverting terminal of op-amp *(' + sign' is inverting terminal of op-amp)* and negative sign to ground, using draw wire option, connect the wires between the positive input terminal and negative ground terminal.

5. Press *Clt+G (Parts Browser Basic)* to select *resistors*, in the *Part Name* type *'r'* , and click *Place and Close* option in *Part Browser Basic*, to place *resistors* in the schematic window.
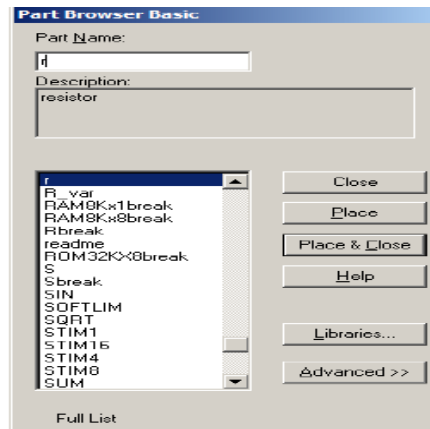
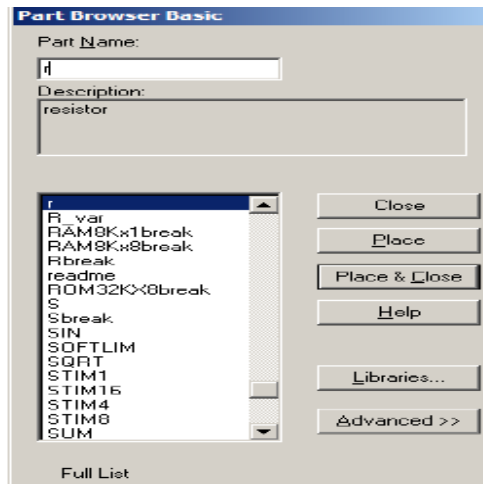6. Double click on the *resistor icon* to add the following parameter .

   a. **R1 = 10K**

   b. **R2= 50K**

7. Select *Draw wire* option from menu bar to connect wire between *inverting terminal ('+ sign' is inverting terminal of op-amp)* and to *R1 resistors* , and *R2 resistors* and output , the label the output name as *invoutput.*

8. In Transient window, type
   a. Print Step = 0
   b. Final time = 5 ms
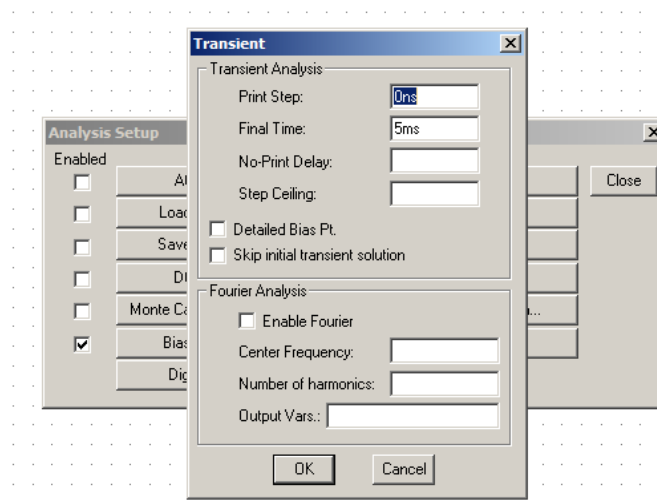
9. Click on Analysis in the menu bar, select Simulate option, next Click on Trace in the menu bar, select Add Trace. Add Trace Window displays the input and output label used in the circuit diagram.
10. Select the input label as V1(Vin) used in the circuit diagram to obtain the input sine wave
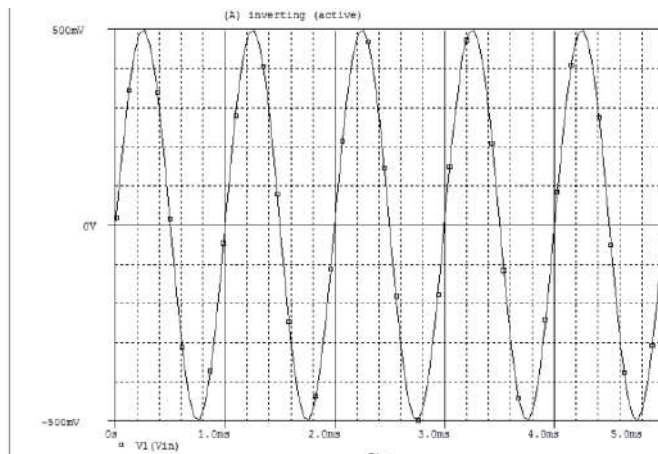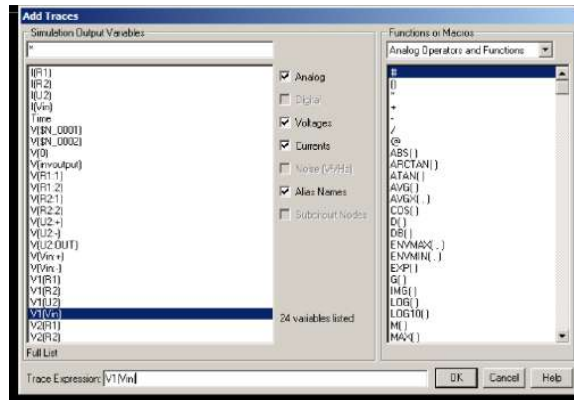
11. Select the output as *V(invoutput)* used in the circuit diagram to obtain the input sine wave.
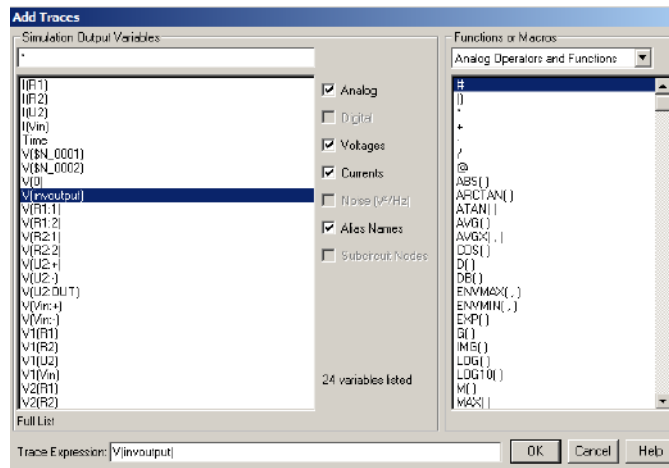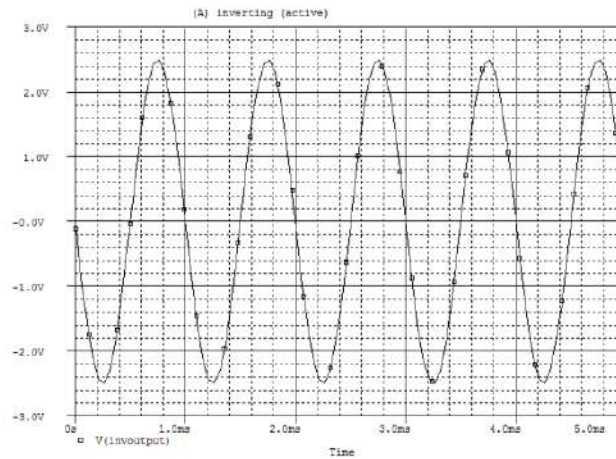
The output vs the input wave is observed, which shows the output wave is phase shift w.r.t to the input sine wave, with **Vout = - 1.25 V**

$$V_{out} = \left(-\frac{R_2}{R_1}\right)Vin$$

$$V_{out} = \left(-\frac{50K}{10K}\right)0.5$$

$$V_{out} = (-1.25\,)V$$

The circuit opamp as inverting amplifier is simulated.
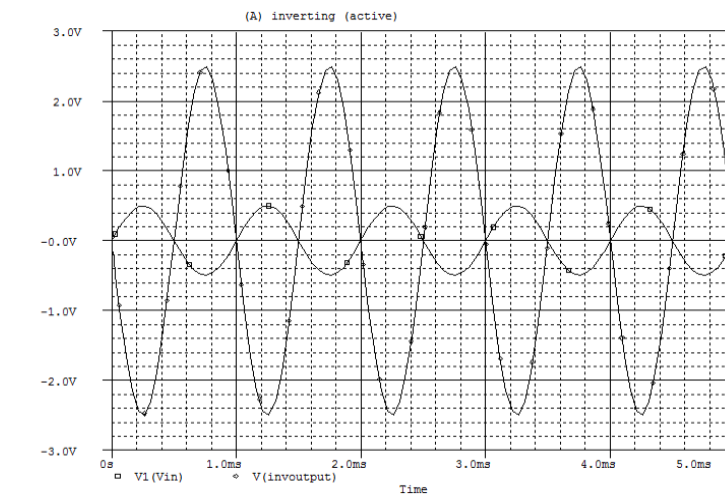
B. 1.3 : Non- Inverting Amplifier

Aim: - *Design and verification of OP-AMP as* **Non-Inverting Amplifier**

Circuit Diagram: *OPAMP as Non-Inverting Amplifier*

Figure 1.3: Non-Inverting Amplifier Circuit

Design :

$Let : R_1 = (10K\Omega)$

$A_V = 6$

$A_V = \left(1 + \dfrac{R_f}{R_1}\right)$

$R_f = 50K\Omega$

Steps to build the Circuit Diagram in PSpice simulation:

1. Press Clt+G *(Parts Browser Basic)* in *Part Name* type *Opamp*, and click *Place and Close* option in *Part Browser Basic*, to place Op-amp in the schematic window.

2. Press *Clt+G (Parts Browser Basic)* to select *input sine wave*, in the *Part Name* type *vsin* , and click *Place and Close* option in *Part Browser Basic*, to place vsin schematic window.

3. Double click on the **vsin icon** to add the following parameter

   (Note: Each time adding the parameter always press, Save Attr)
   Name of the input sine wave = Vin
   a. DC =0

   b. AC =0

   c. $V_{OFF} = 0$

   d. VAMPL =0.5v

   e. FREQ = 1KHz

4. The positive of the sine wave is connected to the non-inverting terminal of op-amp *('- sign' is non-inverting terminal of op-amp)* and negative sign to ground, using draw wire option, connect the wires between the positive input terminal and negative ground terminal.

5. *Press Clt+G (Parts Browser Basic) to select resistors, in the Part Name type 'r' , and click Place and Close option in Part Browser Basic,* to place *resistors* in the schematic window.

6. Double click on the **resistor icon** to add the following parameter .

   c. **R1 = 10K**

   d. **R2= 50K**

7. Select **Draw wire** option from menu bar to connect wire *R1 resistors* between *inverting terminal ('+ sign' is inverting terminal of op-amp)* & *R1 another end to ground,* and between *R2 resistors* and output , the label the output name as *noninvoutput.*

8. In Transient window, type

9. Print Step = 0

10. Final time = 5 ms
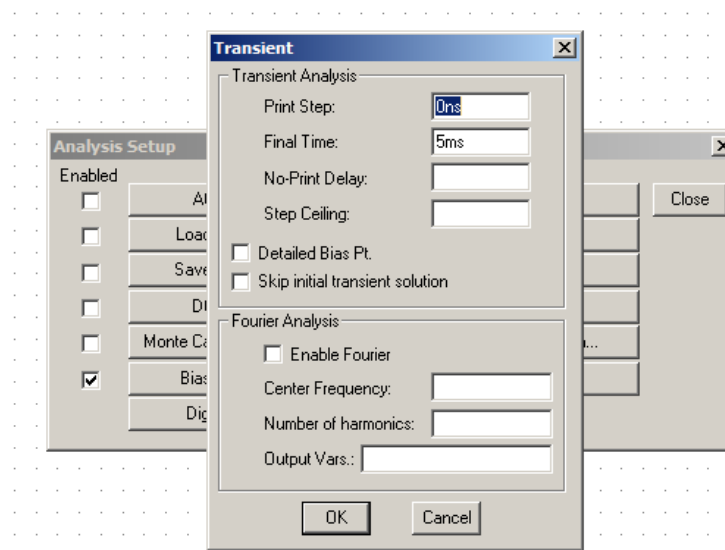
11. Click on Analysis in the menu bar, select Simulate option, next Click on Trace in the menu bar, select Add Trace. Add Trace Window displays the input and output label used in the circuit diagram.

12. Select the input label as *V1(Vin)* used in the circuit diagram to obtain the input sine wave
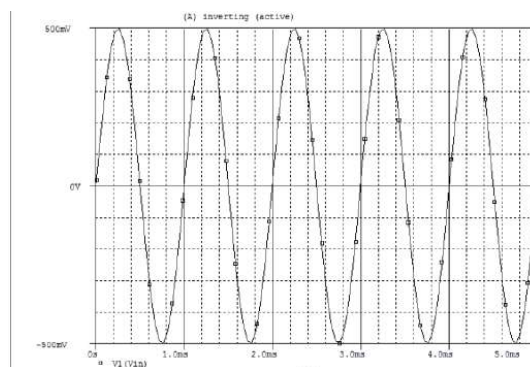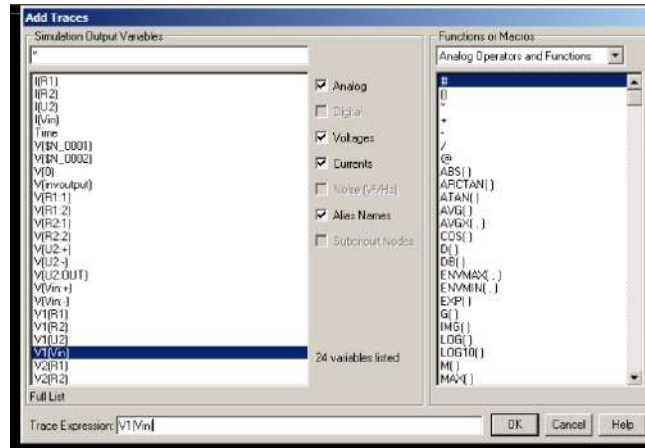
13. Select the output as *V(noninvoutput)* used in the circuit diagram to obtain the input sine wave.
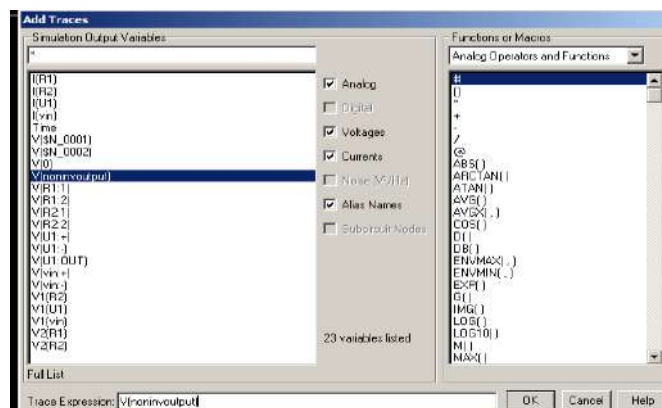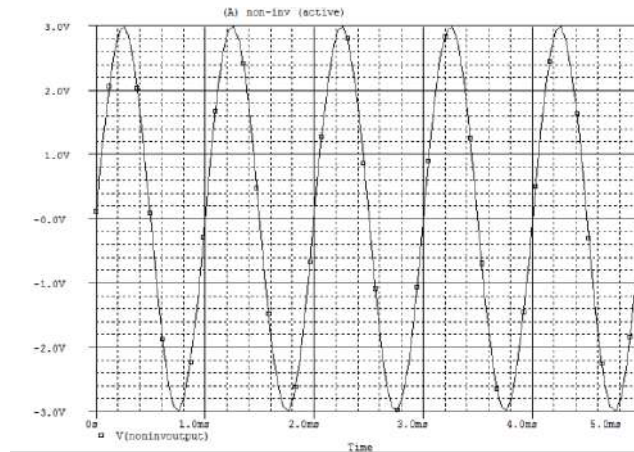
The output vs the input wave is observed, which shows the output wave is in-phase shift wrt to the input sine wave, with **Vout = 3 V**

$$V_{out} = \left(1 + \frac{R_2}{R_1}\right) Vin$$

$$V_{out} = \left(1 + \frac{50K}{10K}\right) 0.5$$

$$V_{out} = (3)V$$
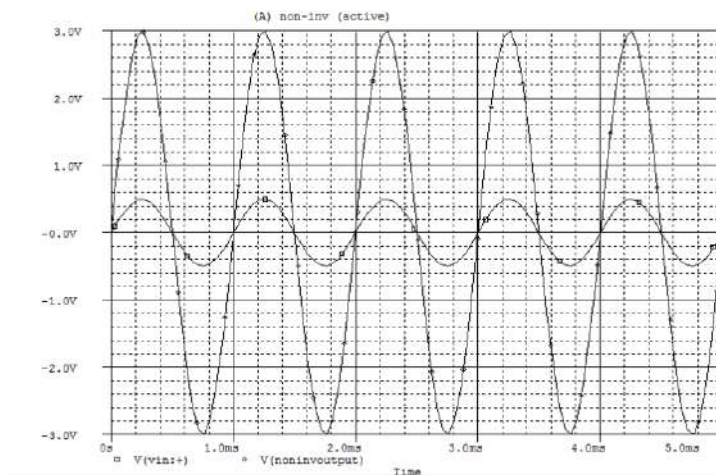
14. The circuit opamp as non- inverting amplifier is simulated.

*****

Experiment No. B.2

Design and verification of Integrator and Differentiator using OP-AMP.

B.2.1 Design and verification of Integrator using OP-AMP.

Aim: - *Design and Verification of Integrator using OP-AMP*
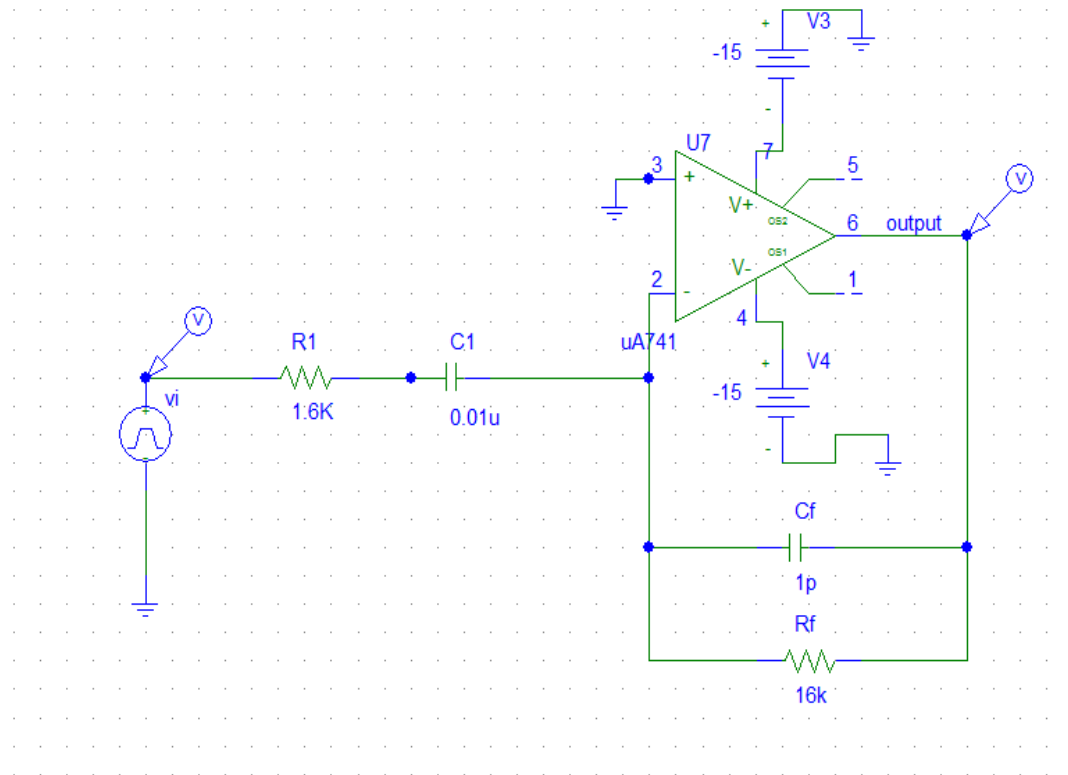
**Circuit Diagram: OPAMP as Integrator**
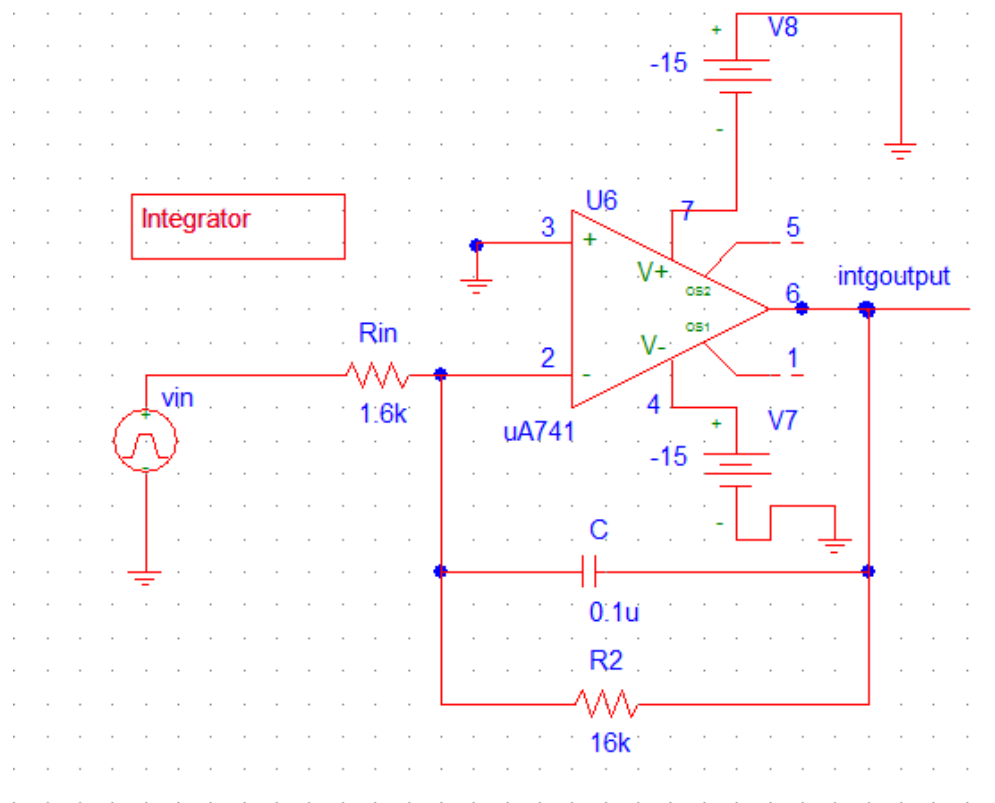

Figure 2.1: Integrator Circuit

Settings:

1. Press **Clt+G** (**Parts Browser Basic**) to select **input square wave**, in the **Part Name** type **vpulse** , and click **Place and Close** option in **Part Browser Basic**, to place vpulse schematic window.
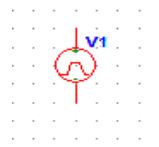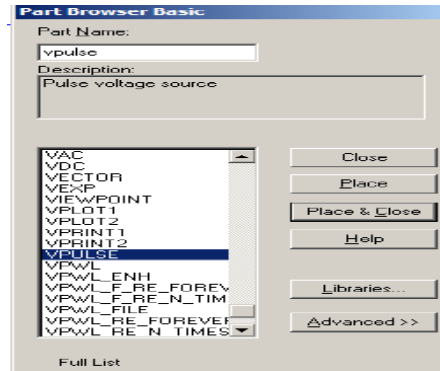
vpulse icon

2. Double click on the **vpulse icon** to add the following parameter.
   (Note: Each time adding the parameter always press, Save Attr).

$V1 = 5V$

$V2 = -5V$

$TD = 0$

$TR = 1ns$

$TF = 1ns$

$PW = 1ms$

$PER = 2ms$
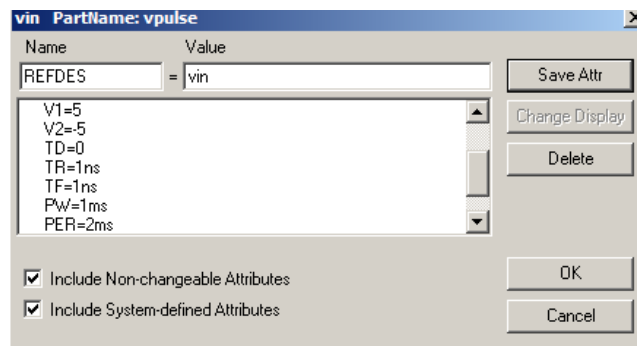
3. Double click of the opamp , to adjust + **Vcc and –Vcc** parameter.

   (Note: Each time adding the parameter always press, Save Attr)

   a. $VPOS = +15V$

   b. $VNEG = -15V$

4. In Transient window, type
   a. Print Step = 10us
   b. Final time = 60 ms
   c. No-Print Delay = 50ms
   d. Select initial transient solution
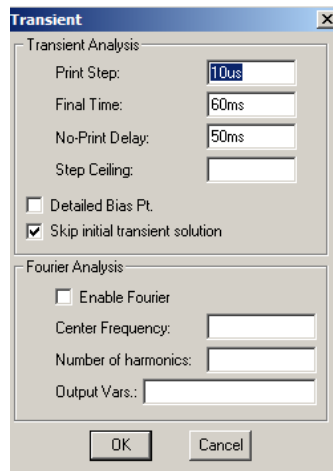
Procedure

1. Connections are made as per the circuit diagram.
2. Apply the square input signal.
3. The output voltage is simulated and the input and output voltage waveforms are obtained as shown.
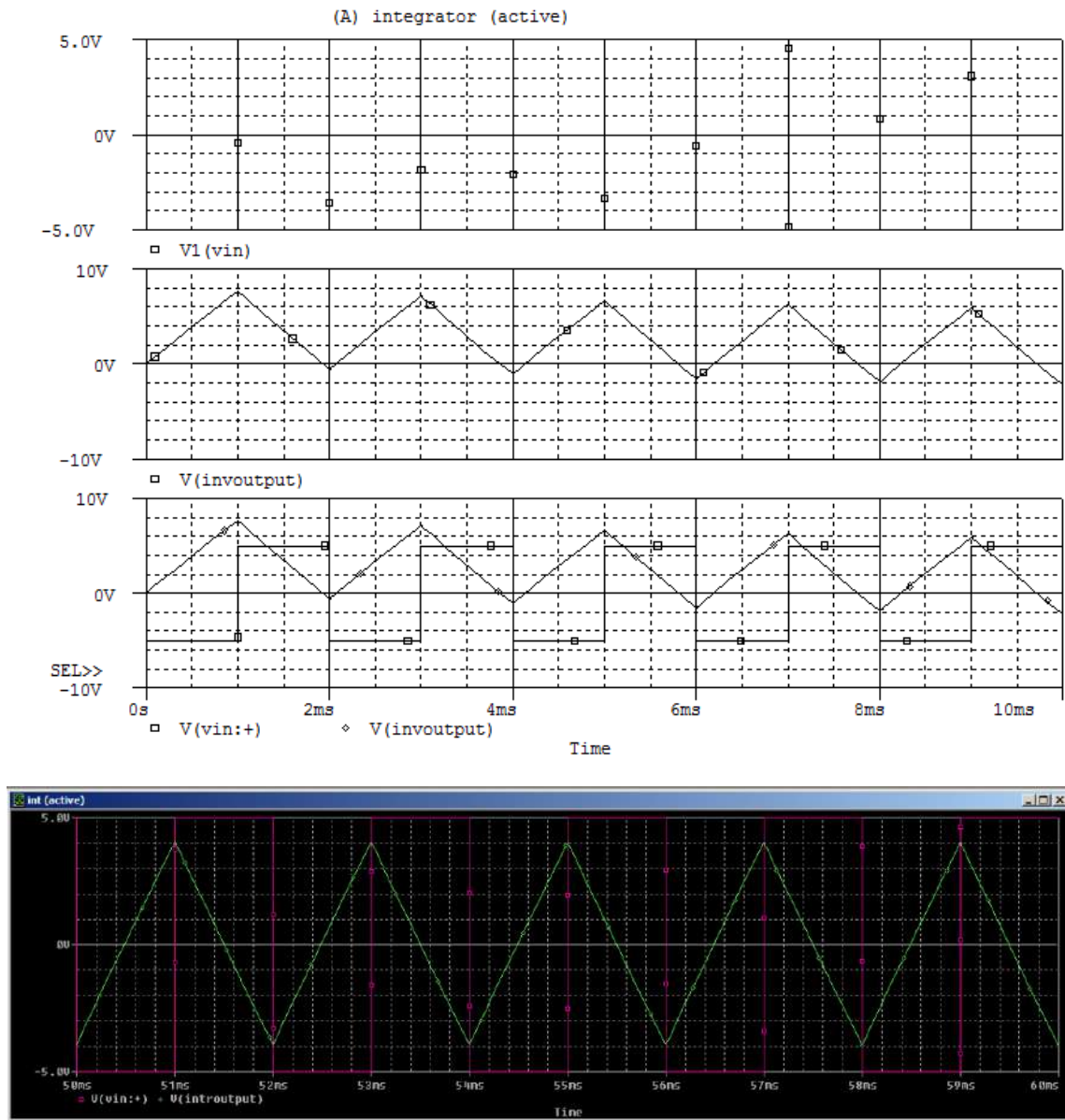
Figure 2.2: Output waveforms

*****

B. 2.2 : *Design and verification of Differentiator using OP-AMP*

Aim: - Design and Verification of Differentiator using OP-AMP
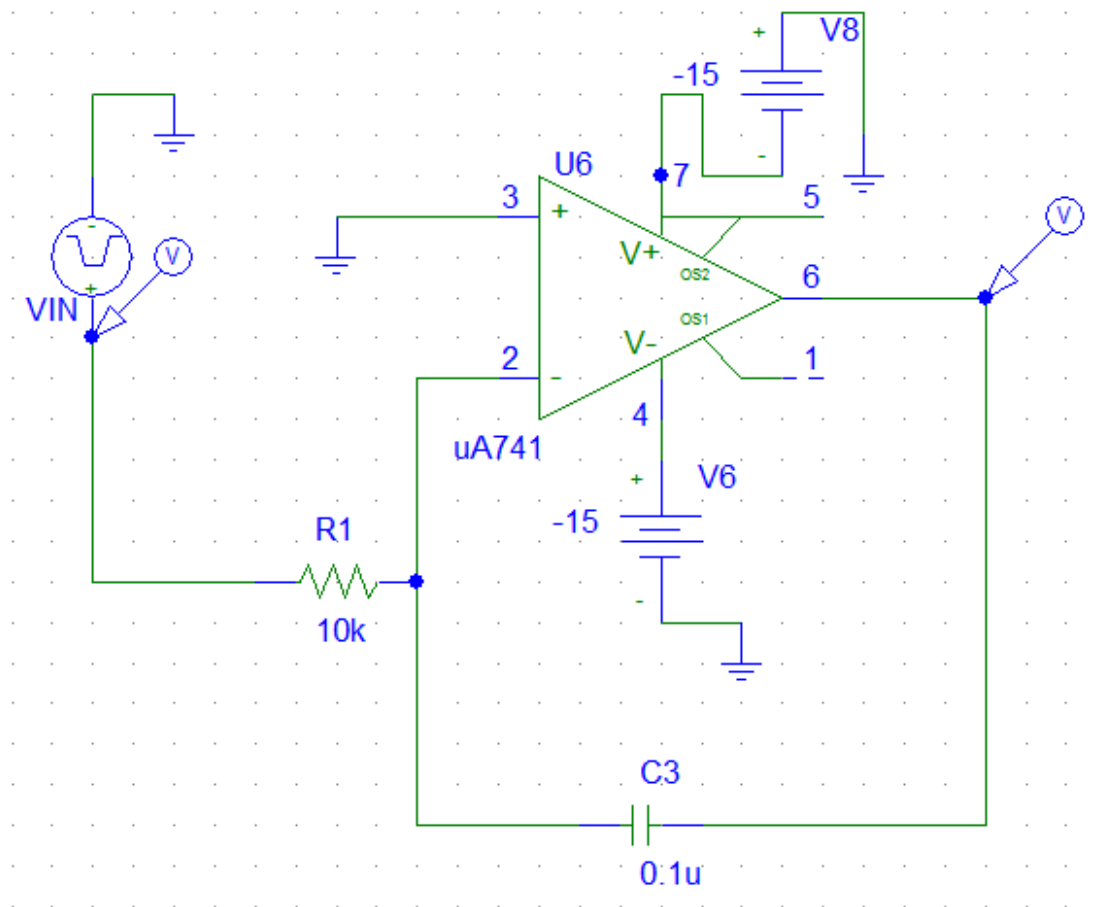
Circuit Diagram : <u>OPAMP as Differentiator</u>

Figure 2.1: Differentiator Circuit

Settings:

1. Press **Clt+G** (**Parts Browser Basic**) to select **input square wave**, in the **Part Name** type **vpulse,** and click **Place and Close** option in **Part Browser Basic**, to place vpulse schematic window.
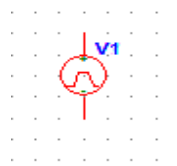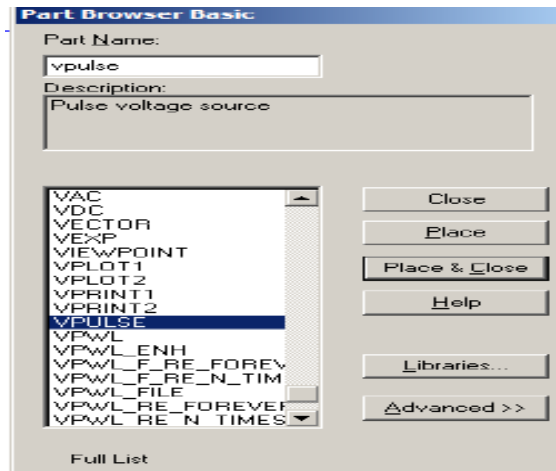
vpulse icon

2. Double click on the **vpulse icon** to add the following parameter.

   (Note: Each time adding the parameter always press, Save Attr).
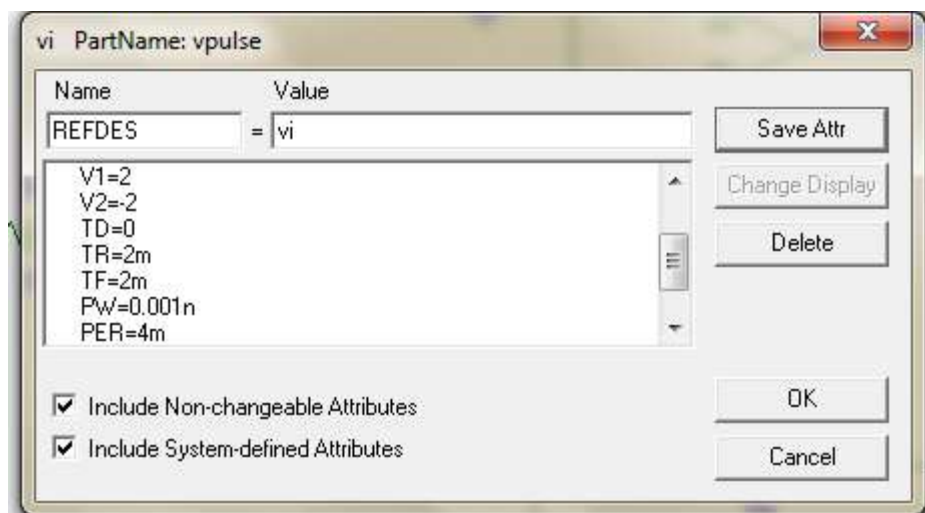
3. In Transient window, type
   a. Print Step = 10us
   b. Final time = 60 ms
   c. No-Print Delay = 50ms
   d. Select initial transient solution
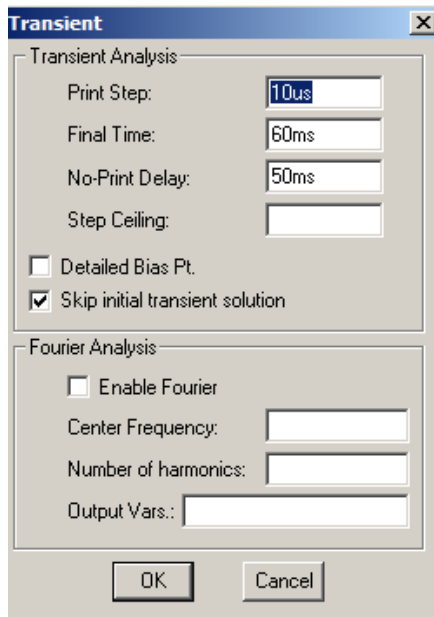
Procedure

1. Connections are made as per the circuit diagram.
2. Apply the square input signal.
3. The output voltage is simulated and the input and output voltage waveforms are obtained as shown.
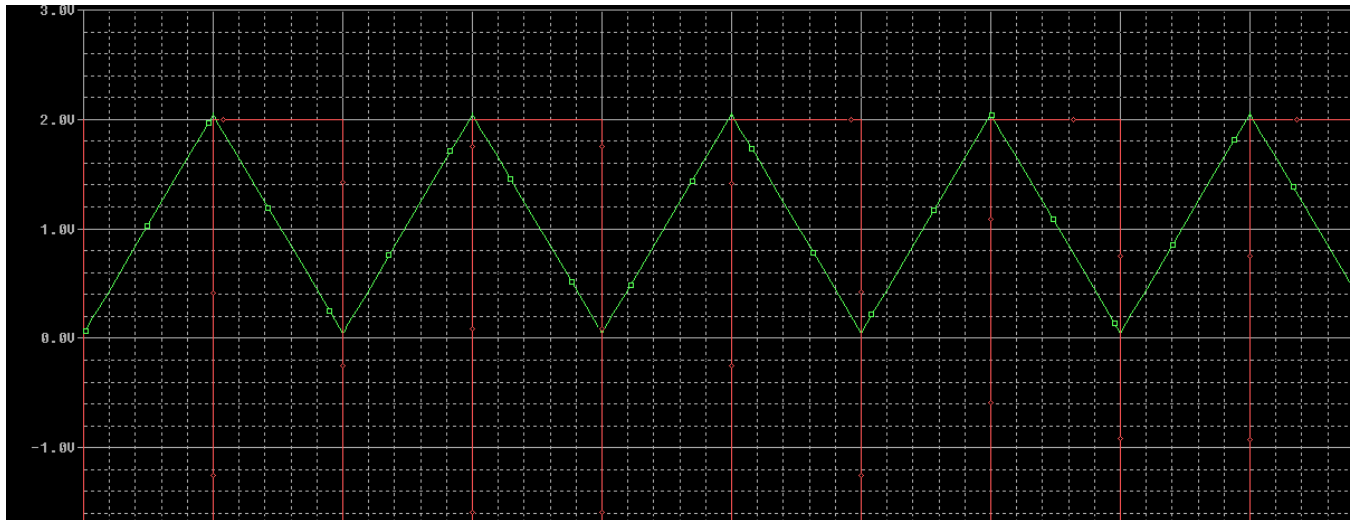

*****
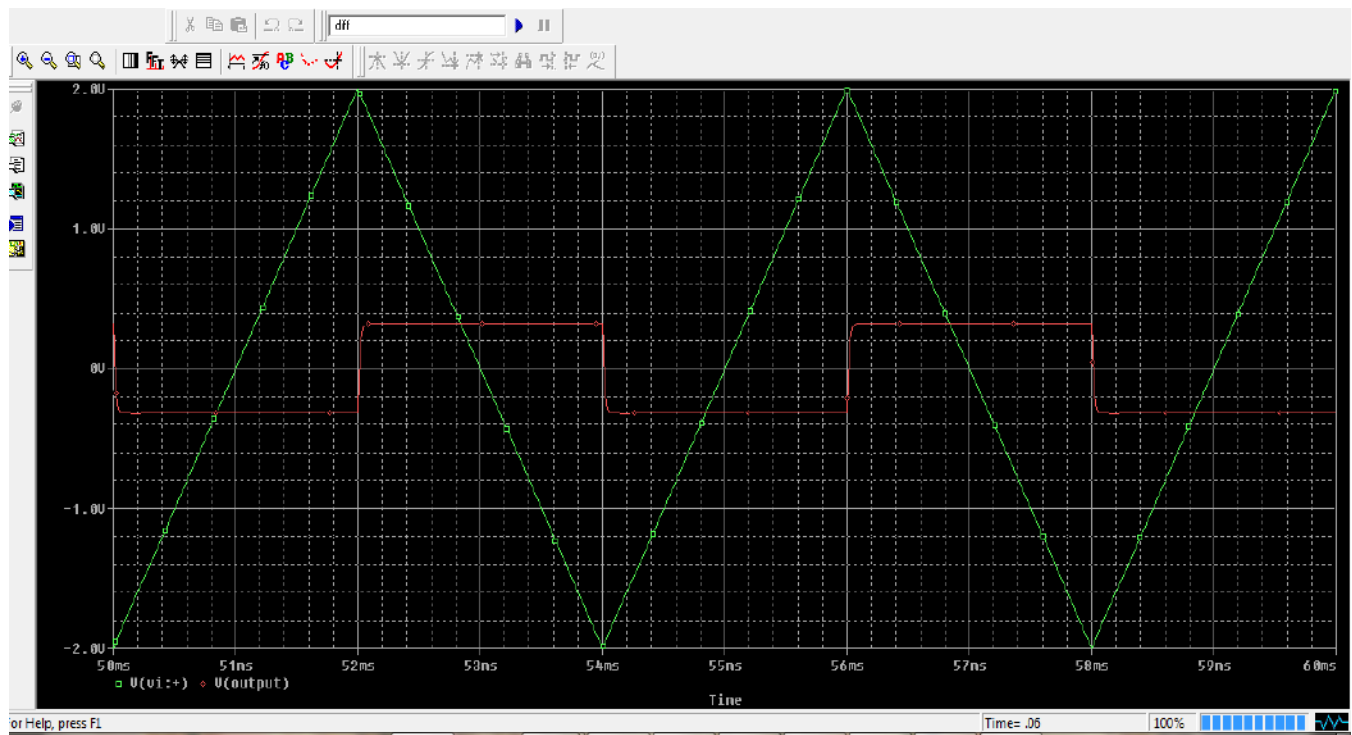
Experiment No. B.3

Design and Simulation of Function generator to generate square wave and triangular wave generator using OP-AMP.

B. 3.1: Design and Simulation of Function generate triangular wave using OP-AMP

Aim: Design and Simulation of Function generate triangular wave using OP-AMP

Circuit Diagram: *TRIANGULAR WAVE GENERATOR*

Figure 3.1: Triangular wave generator Circuit

Settings:

1. Press **Clt+G** (**Parts Browser Basic**) to select **input square wave**, in the **Part Name** type **vpulse** , and click **Place and Close** option in **Part Browser Basic**, to place vpulse schematic window.
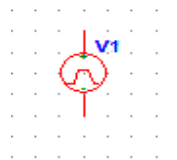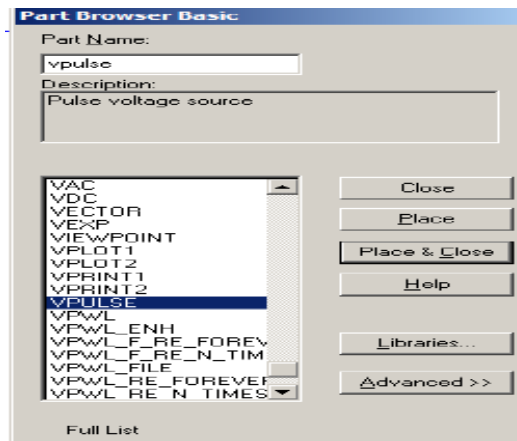
vpulse icon

2. Double click on the **vpulse icon** to add the following parameter.
   (Note: Each time adding the parameter always press, Save Attr).

$$DC = 1$$
$$AC = 1$$
$$V1 = 2V$$
$$V2 = -2V$$
$$TD = 0$$
$$TR = 1ns$$
$$TF = 1ns$$
$$PW = 1ms$$
$$PER = 2ms$$

3. In Transient window, type
   a. Print Step = 10us
   b. Final time = 60 ms
   c. No-Print Delay = 50ms
   d. Select initial transient solution.

Procedure

- Connections are made as per the circuit diagram.
- Apply the square input signal.
- The output voltage is simulated and the input and output voltage waveforms are obtained as shown.

Figure 3.2: Output Triangular Waveform

*****

B. 3.2 : Design and Simulation of Function generate square wave using OP-AMP

Aim: Design and Simulation of Function generate square wave using OP-AMP
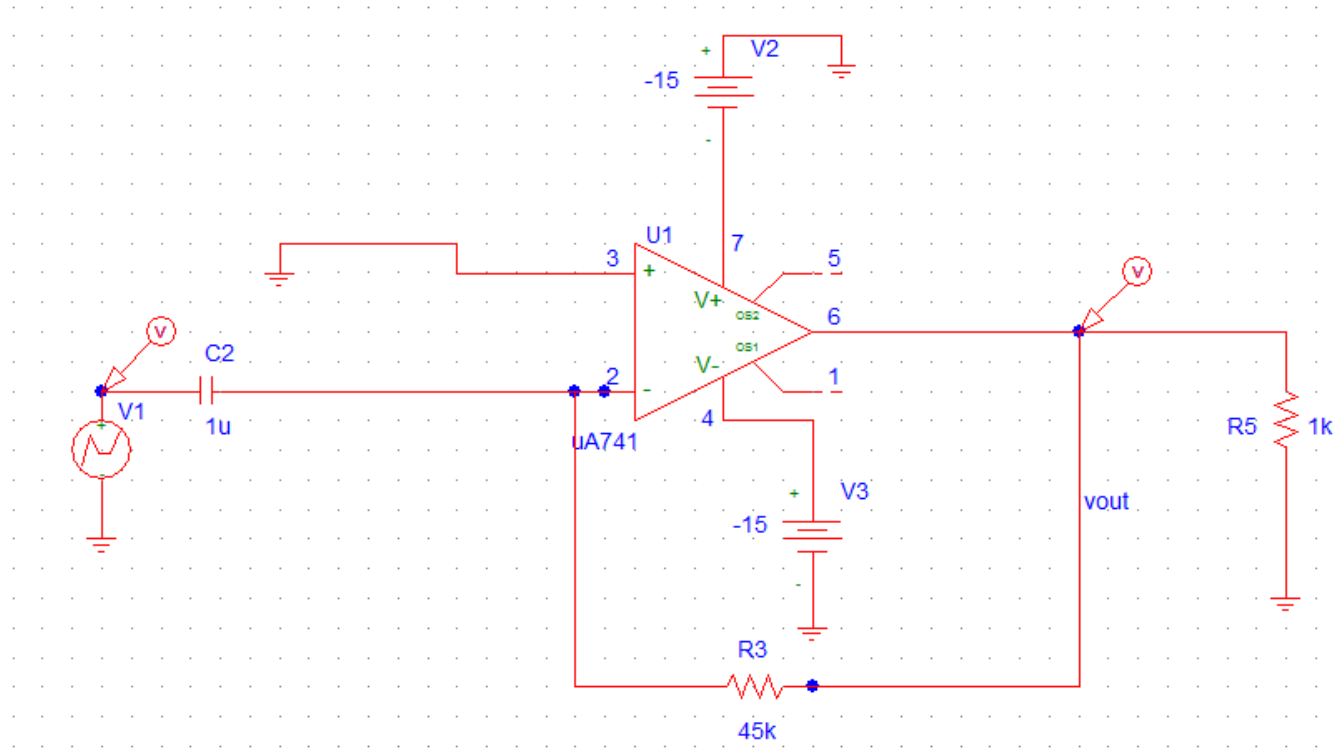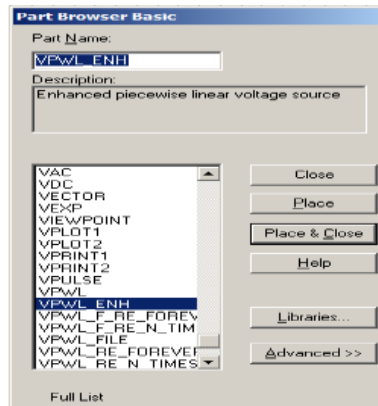
Circuit Diagram: SQUARE WAVE GENERATOR

Figure 3.3: Square Wave Generator Circuit

Settings:

1. Press **Clt+G** (**Parts Browser Basic**) to select **input triangular wave**, in the **Part Name** type **VPWL_ENH** , and click **Place and Close** option in **Part Browser Basic**, to place vpulse schematic window.

VPWL_ENH icon

2. Double click on the *vpwl_enh icon* to add the following parameter.
(Note: Each time adding the parameter always press, Save Attr).

$DC = 1$

$AC = 1$

$TSF = 1$

$VSF = 1$

First_nPairs $= 0.1$ns 0.1mv

Second_nPairs $= 1$ms 1v

Third_nPairs $= 2$ms 0v

Repeat $= 10$
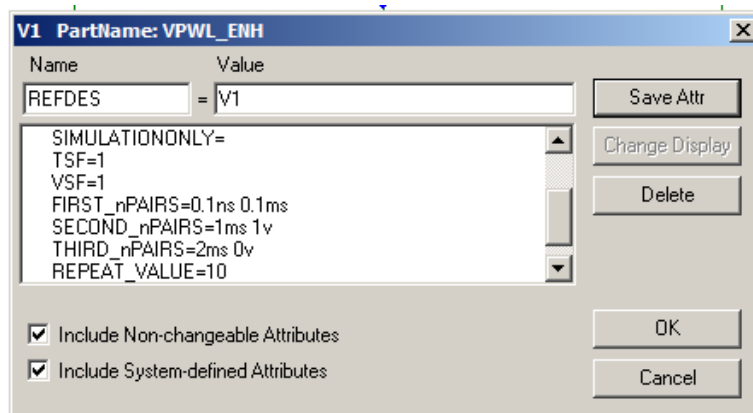
3. In Transient window, type

    a. Print Step $= 0$ns

    b. Final time $= 10$ ms

4. Select initial transient solution.

Procedure
- Connections are made as per the circuit diagram.
- Apply the triangular input signal.
- The output voltage is simulated and the input and output voltage waveforms are obtained as shown.
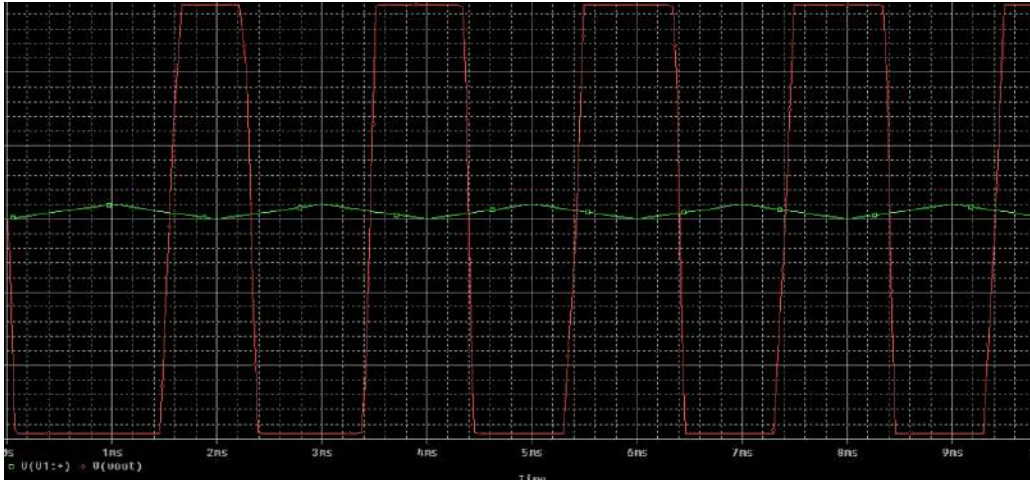
Figure 3.4: Square Wave Generator Waveform

****

Experiment No. B.4

Analyze Input, Output characteristics of BJT Common emitter configuration and evaluation of parameters.

Aim: - Analyze Input, Output characteristics of BJT Common emitter configuration and evaluation of parameters.

Circuit Diagram:
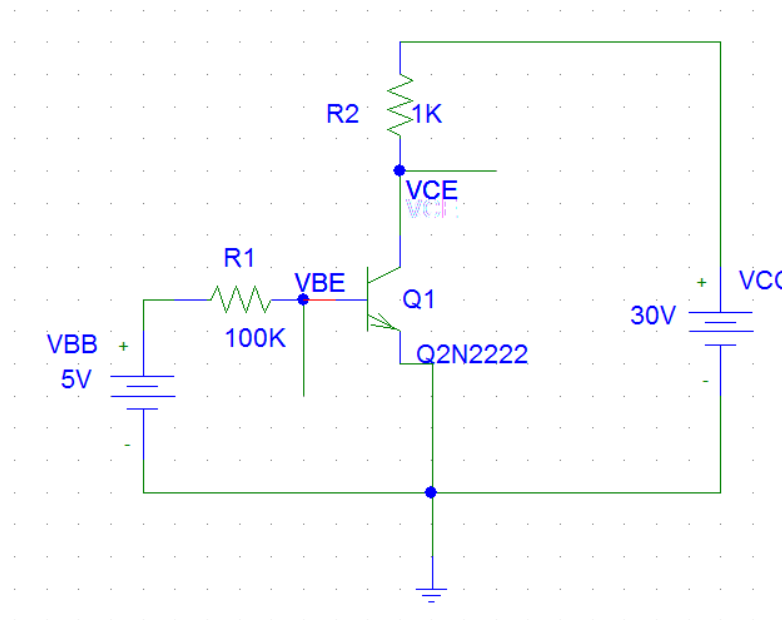
Figure 4.1: BJT Common Emitter Circuit

Steps to draw the Circuit Diagram in PSpice simulation:

1. Press Clt+G (**Parts Browser Basic**) in **Part Name** type **Q2N2222**, and click **Place and Close** option in **Part Browser Basic**, to place npn= transistor in the schematic window.

2. To add input DC voltage and bias voltage VCC, follow the below steps:

3. Press **Clt+G** (**Parts Browser Basic**) to select **dc voltage** , in the **Part Name** type **vdc** , and click **Place and Close** option in **Part Browser Basic**, to place vdc at the input side and bias voltage in the schematic window.

4. Double click on the **vdc icon** at input side to add the following parameter.

(Note: Each time adding the parameter always press, Save Attr)

Name of the input dc voltage = VBB
   a.   DC =5V

5.   Double click on the **vdc icon** for the bias voltage

   Name of the input dc voltage = VBB
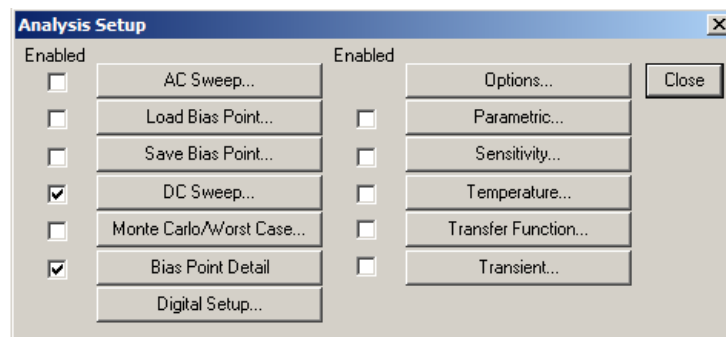   a.   DC =30V

6.   Double click on the **resistor icon** to add the following parameter .

   b.   R1 = 100K

   c.   R2= 1K

7.   Add label as VBE at input side, and VCE at output side, as shown in circuit diagram to obtain the characteristic curve

8.   Click on *Analysis* in the menu bar, select *Setup option*, Select *DC Sweep option.*

9.   In *DC Sweep* , Select the Voltage source, and type

Name = VBB
Start Value : 0
End Value = 5
Increment = 0.5.
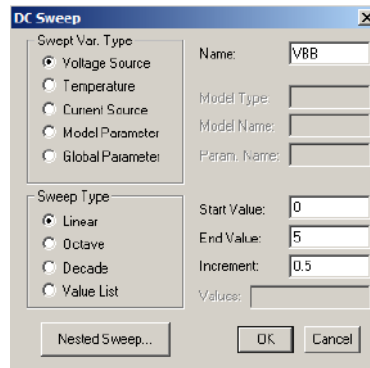
10. In *DC Sweep* , Select the *Nested Sweep*

11. In *DC Nested Sweep* , Select the *Voltage source*, and enter

Name = VCC
Start Value: 0
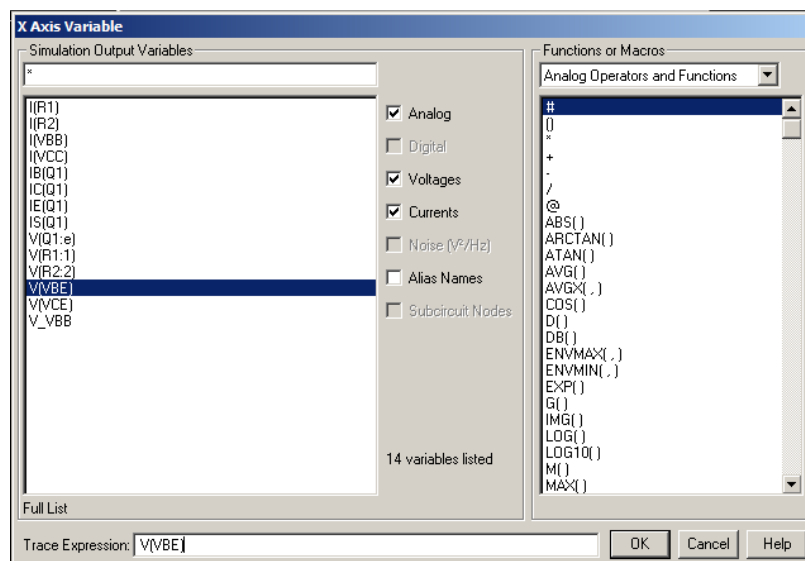*End Value = 15*
Increment = 5
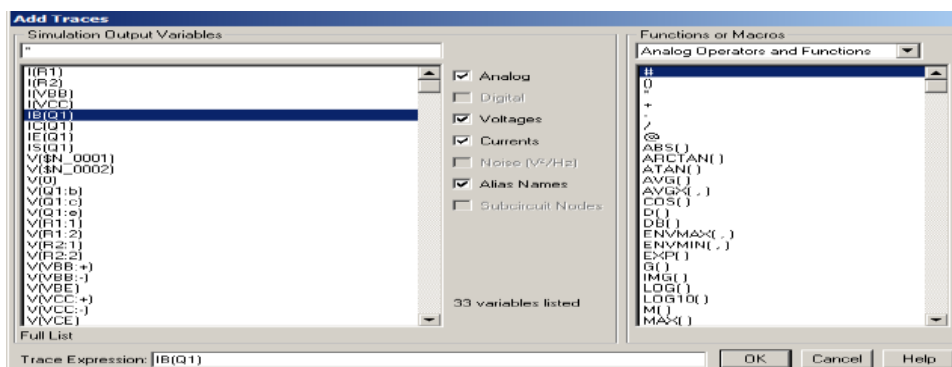**Enable : Enable Nested Sweep.**
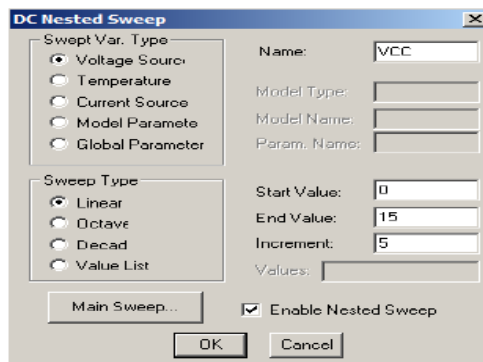
12. Click on Analysis in the menu bar, select Simulate option, next Click on Plot in the menu bar, select Axis Setting. In Axis Setting click on Axis variable.
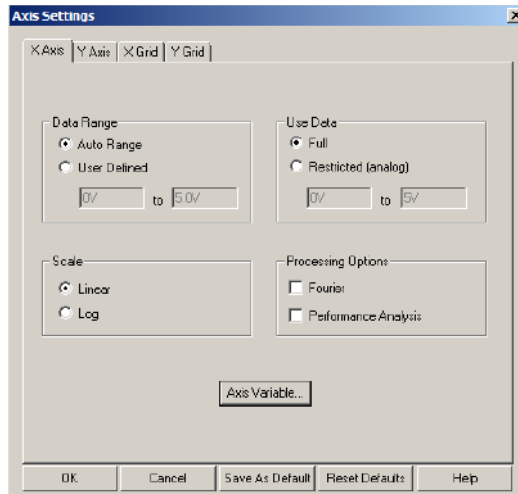
13. In Axis Variable, disable Alias Names, and Select V(VBE) and Click OK. The X-axis VBE would be displayed.

14. Next Click on Trace in the menu bar, select Add Trace.

15. Select IB(Q1).

16. The input characteristics curve (IB vs VBE) is plotted as shown in figure
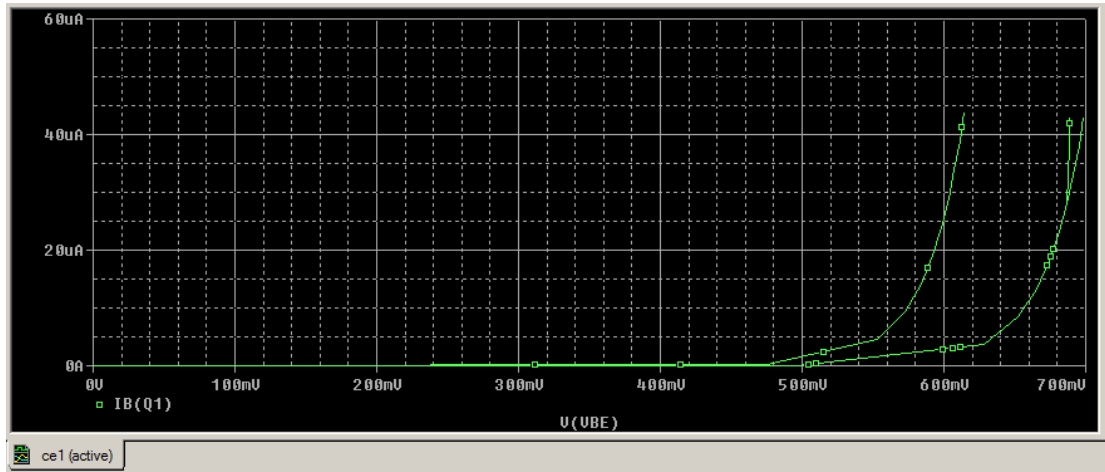
Figure 4.2: The input characteristics waveform

17. The input characteristics curve (IB vs VBB) is plotted as shown in figure
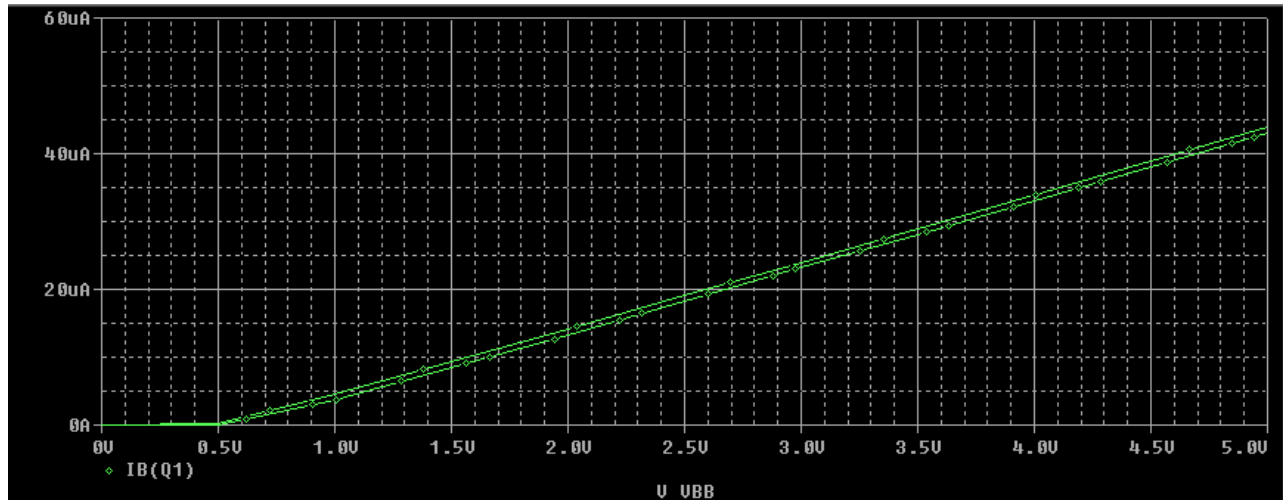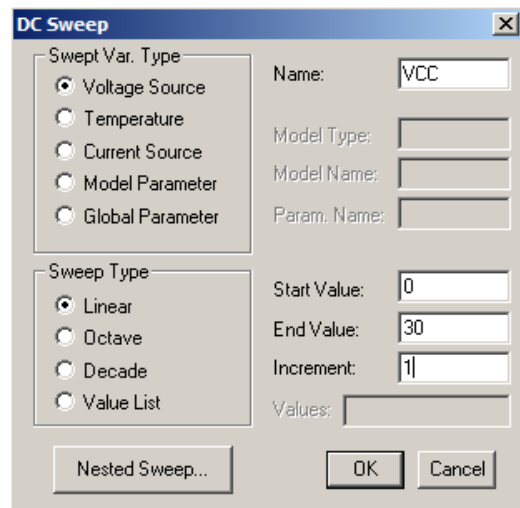

Figure 4.3: The input characteristics waveform

To plot output characteristics curve

18. In *DC Sweep* , Select the *Voltage source*, and type

    a. **Name = VCC**
    b. **Start Value : 0**
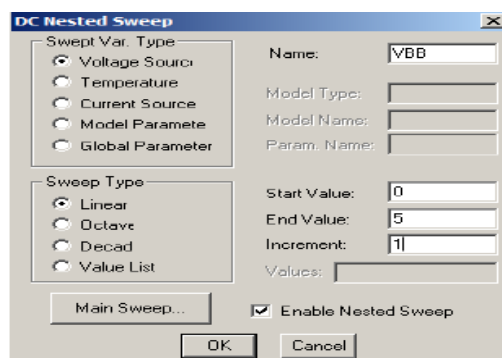   c. **End Value = 30**
    d. **Increment = 1**

19. In *DC Sweep* , Select the *Nested Sweep*

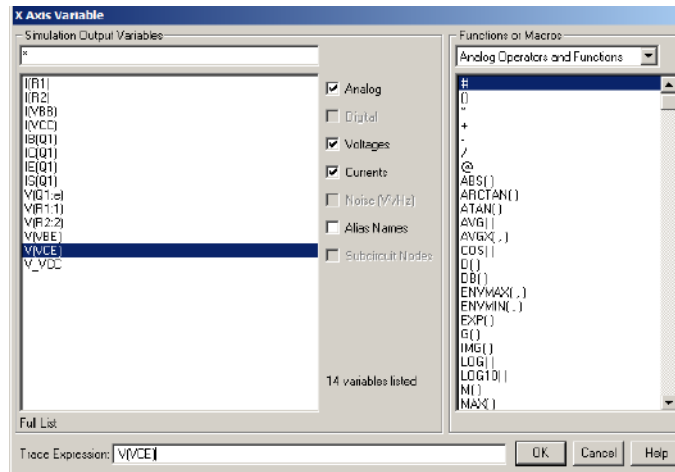20. In *DC Nested Sweep* , Select the *Voltage source*, and type

    a. **Name = VBB**
    b. **Start Value: 0**
    c. **End Value = 5**
    d. **Increment**   =1

Enable : Enable Nested Sweep.

To plot output characteristics, In simulation select V(VCE) in Axis Variable

21. Next Click on Trace in the menu bar, select Add Trace. Select IC(Q1).

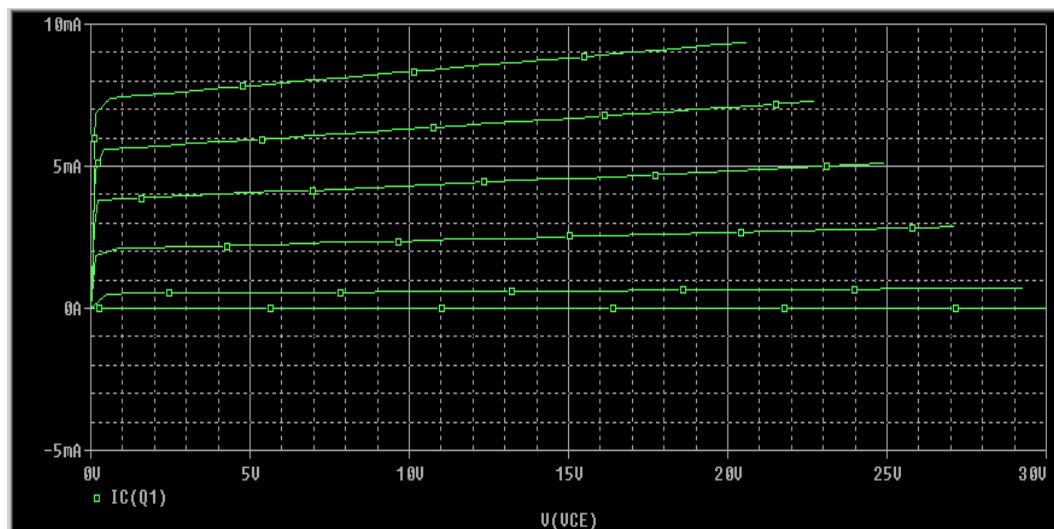22. The output characteristics curve (IC vs VCE) is plotted as shown in figure

Figure 4.4: Output characteristic waveform

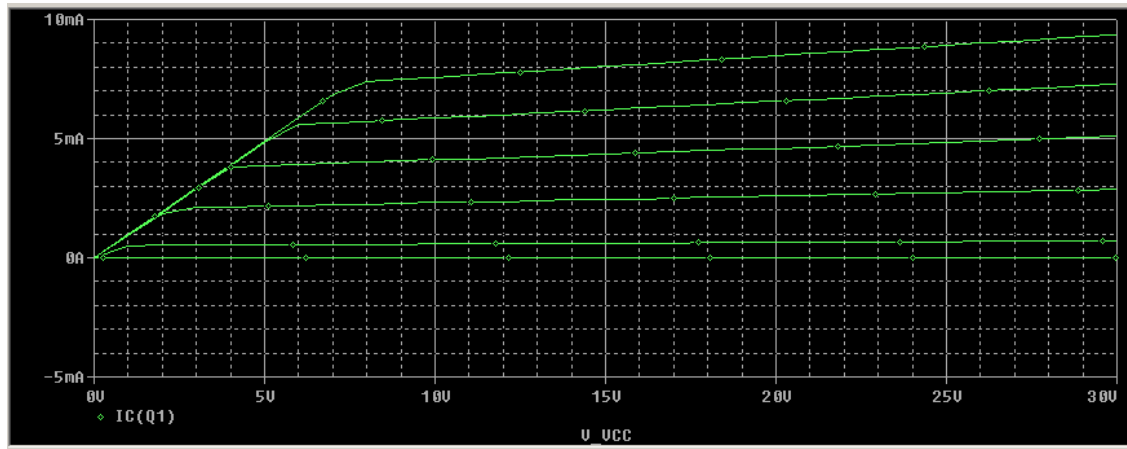23. The input characteristics curve (Ic vs VCC) is plotted as shown in figure

Figure 4.5: Output characteristic waveform

Procedure
- Connections are made as per the circuit diagram.
- The input and output characteristics curve are obtained as shown.

*****

Experiment No. B.5

Analyze drain and gate characteristics of JFET

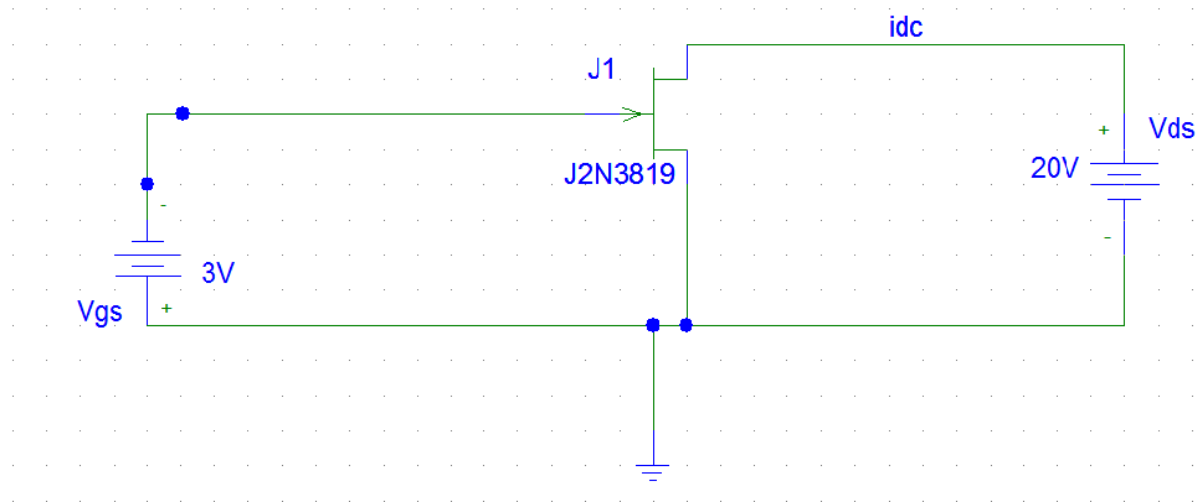*Aim: -* *Analyze drain and gate characteristics of JFET*

Circuit Diagram:

Figure 5.1: JFET circuit

Parameter Settings

1. **Input values:**

Source : Vdc	Source: Vdc
Name : Vgs	*Name: Vds*
DC = 0V	DC = 20V

2. ***DC sweep parameters for drain characteristics***

   To add values for Vgs, select Nested Sweep, and click on enable nested sweep.
   Vgs values to be entered in Nested Sweep only

Vds	*Vgs*
Name Vdc	Name Vgs
Start values: 0V	Start values: 0V
End Value: 20V	End Value: 20V
Increment: 1V	Increment: 1V

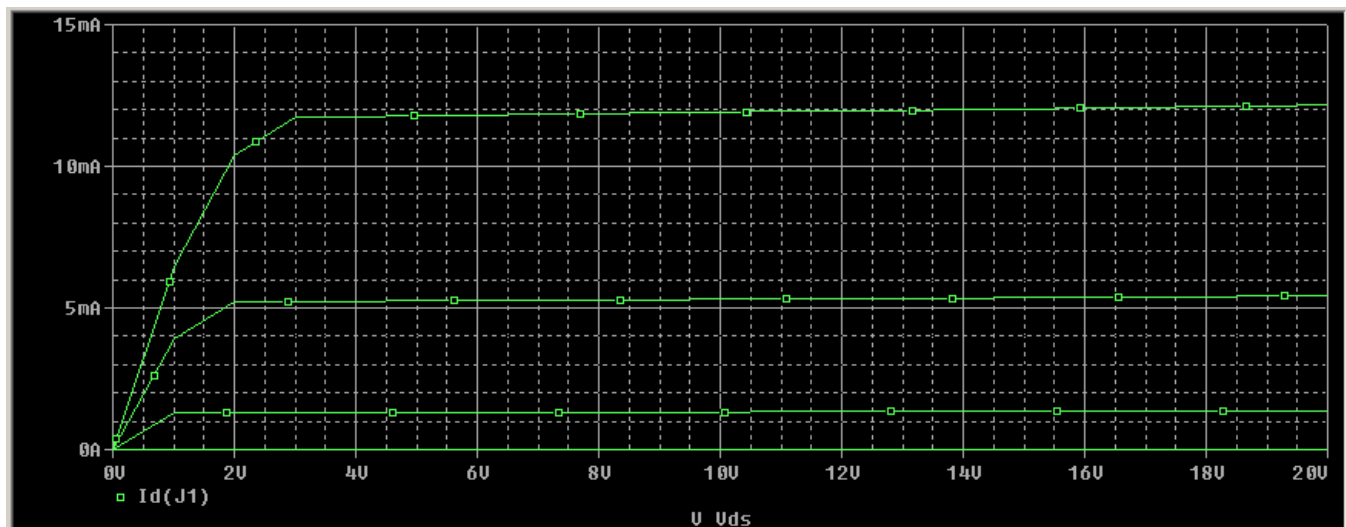3. To analyze the drain characteristic, plot the graph of Vds vs Idc.

Figure 5.2: Drain characteristics waveform

1. *DC sweep parameters for gate characteristics*

   add values for Vgs only, disable nested sweep.

Vgs
Name Vgs
Start values: 0V
End Value: 5V
Increment: 1V

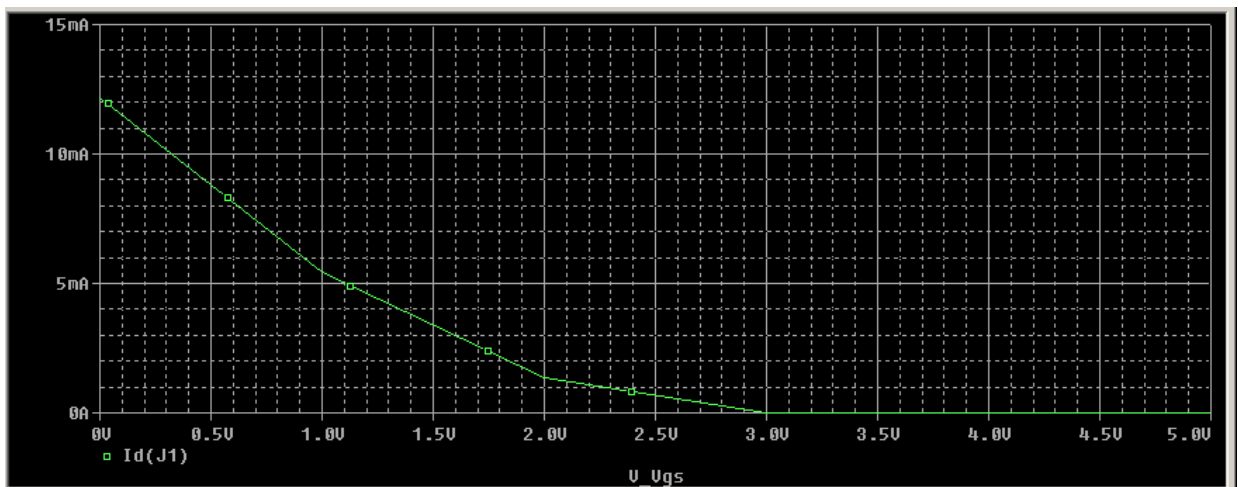2. To analyze the gate characteristic plot the graph of Vgs vs Idc.

Figure 5.3: Gate characteristics waveform

Procedure

- The circuit is built as per the circuit diagram.
- The drain and gatecharacteristics curve are obtained as shown.

****

Experiment No. B.6

Analyze of Static characteristics of SCR

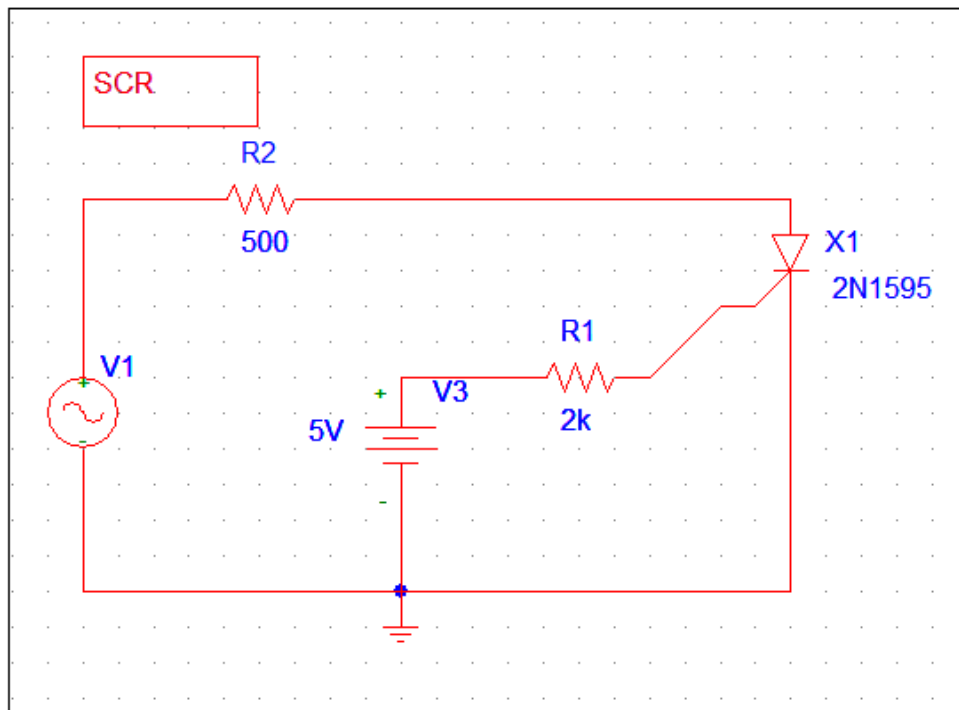*Aim: -* *Analyze of Static characteristics of SCR*

Circuit Diagram: Silicon Controlled Rectifier (SCR)

Figure 6.1: SCR cirucit

Parameter Settings
   1. *Input values:*

Source : Vdc	Source: VSIN
Name : V3	*Name: V1*
DC = 5V	DC = 0
	AC=0
	VOFF = 0
	VAMPL = 12V
	FREQ = 50

2. *Input values:*
   - *R1= 2K*
   - *R2= 500*

3. *SCR name : 2N1595.*

3. *Transient analysis*

| Print step: 0ns |
| Final Time: 100ms |

4. To analyze the scr characteristic:

a.   Add Traces: Select I(R2)
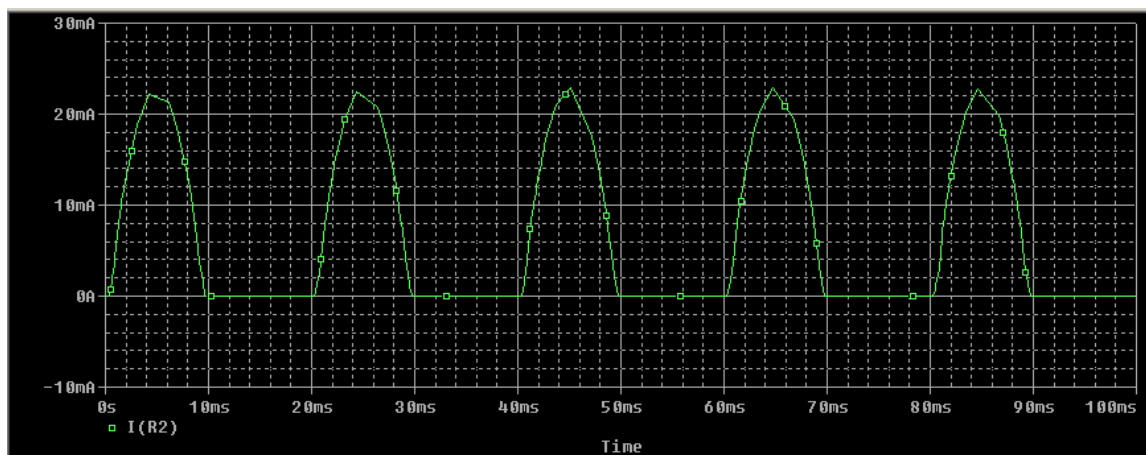
Figure 6.2: SCR characteristics waveform

b.   Select Plot: Axis Setting: Axis Variable: Select V(X1 : A)
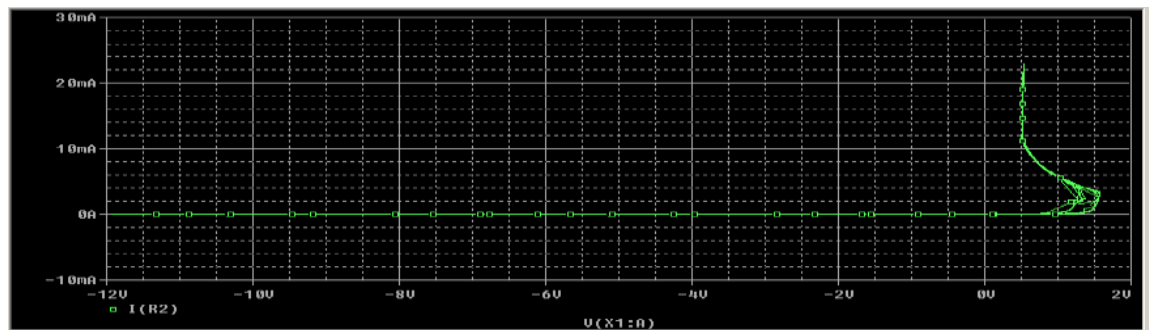
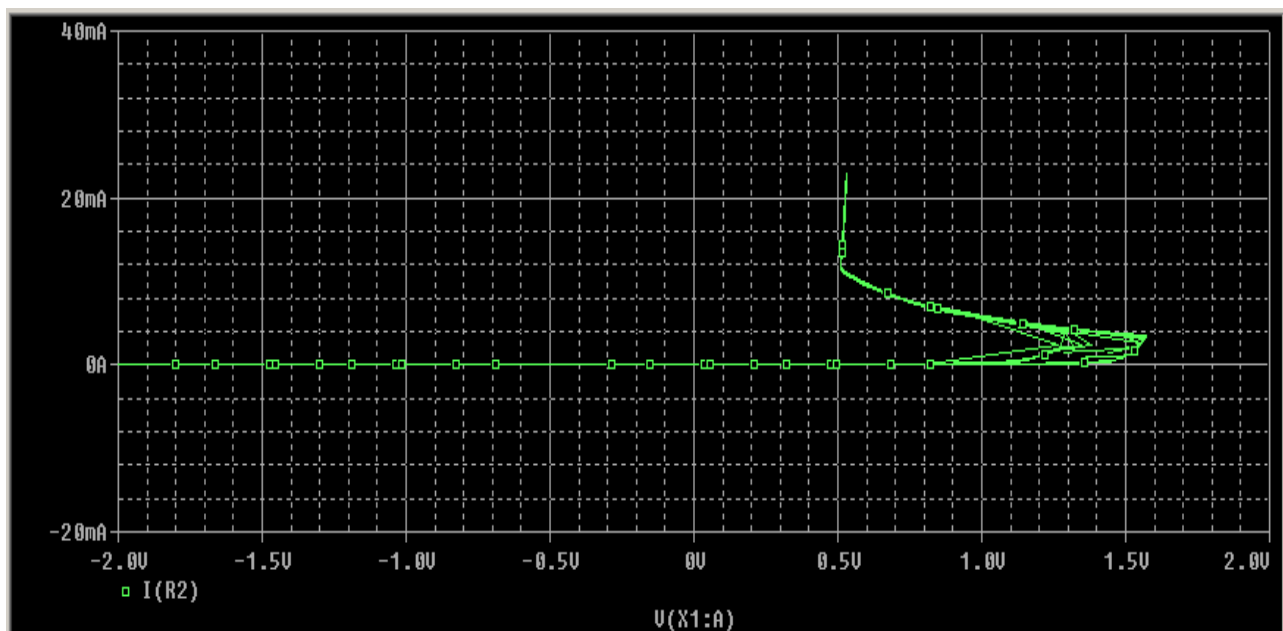   *c.* Select Plot: Axis Setting: Axis Variable: Select User Defined: Range -2 to 2.0V
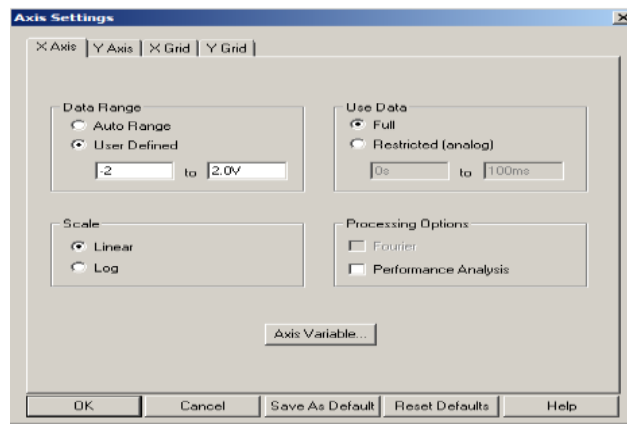

Figure 6.3: Forward characteristics of SCR waveform

Procedure

- The circuit is built as per the circuit diagram.
- The SCR characteristics are obtained as shown.

RC PHASE SHIFT OSCILLATOR

Aim: - *Design and verification of OP-AMP as* **RC PHASE SHIFT OSCILLATOR**
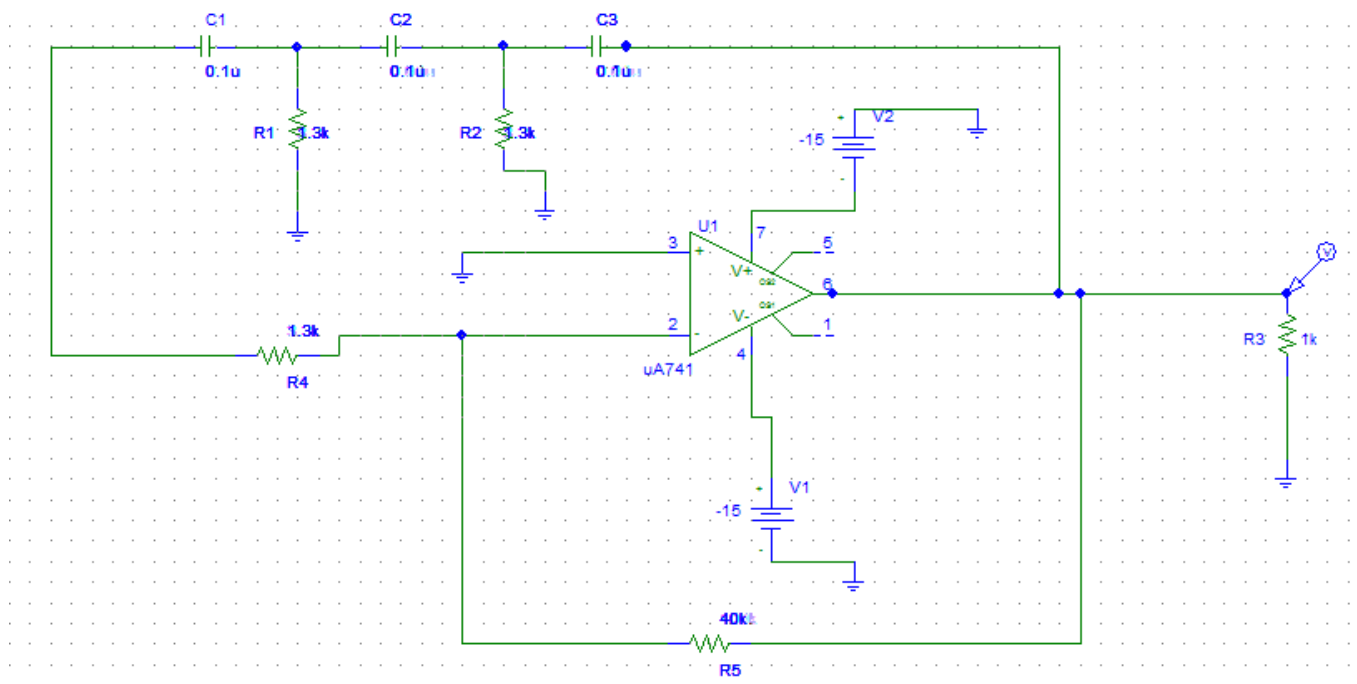
Circuit Diagram:

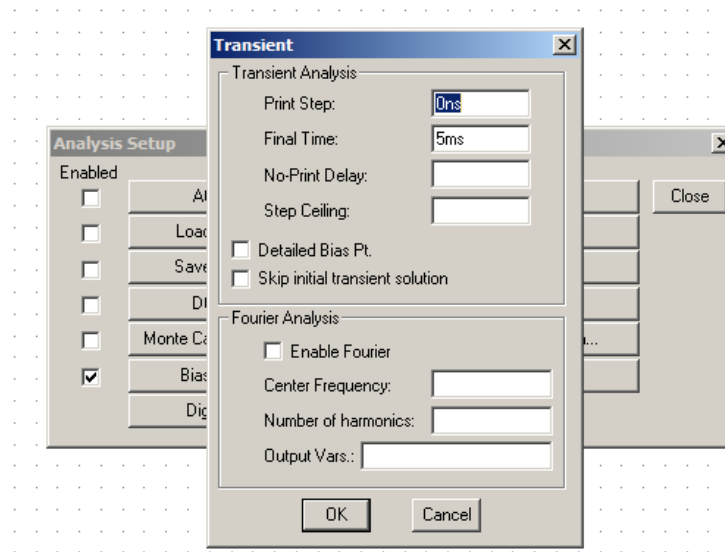*Figure: RC PHASE SHIFT OSCILLATOR CIRCUIT*

Frequency of oscillation

$$f_{\text{oscillation}} = \frac{1}{2\pi RC\sqrt{6}}$$
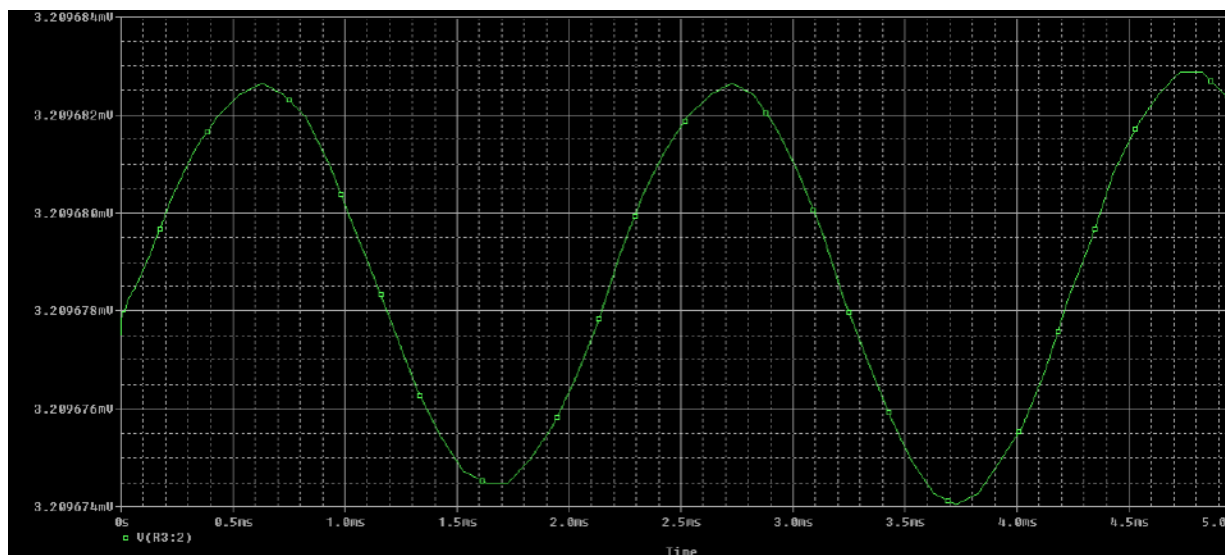
- **R = 1.3*K***
- **C = 0.1u**

    T = 1/f = 2ms

In Transient window, type

1. Print Step = 0

2. Final time = 5 ms

The circuit opamp OSCILLATOR is simulated.

****

R-2R DIGITAL TO ANALOG CONVERTER

*Aim: - Design and verification of OP-AMP as **R-2R DIGITAL TO ANALOG CONVERTER***
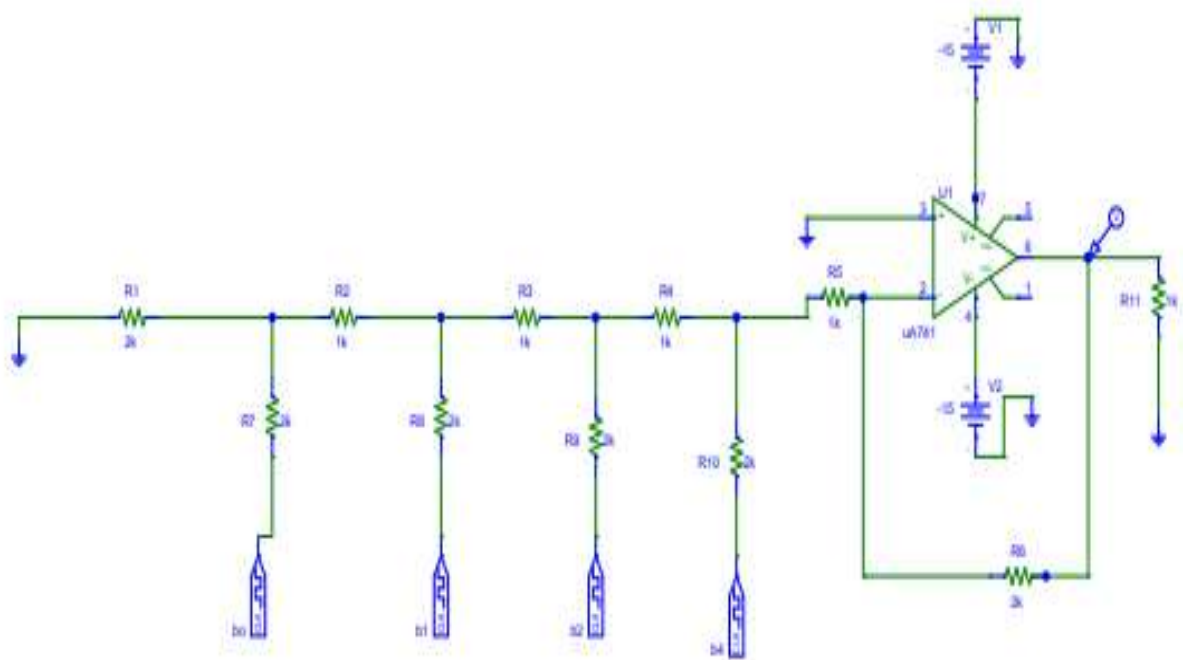
Circuit Diagram:

Figure: **R-2R DIGITAL TO ANALOG CONVERTER**

$$Vo = Vref \sum_{i=0}^{i = N-1} \frac{B_i}{2^{(N-i)}}$$

$$V_{ref}\left(\frac{B0}{2^n} + \frac{B1}{2^{n-1}} + \frac{B2}{2^{n-2}} + \cdots + \frac{Bn-2}{2^2} + \frac{Bn-1}{2^1}\right)$$

DIGCLOCK:

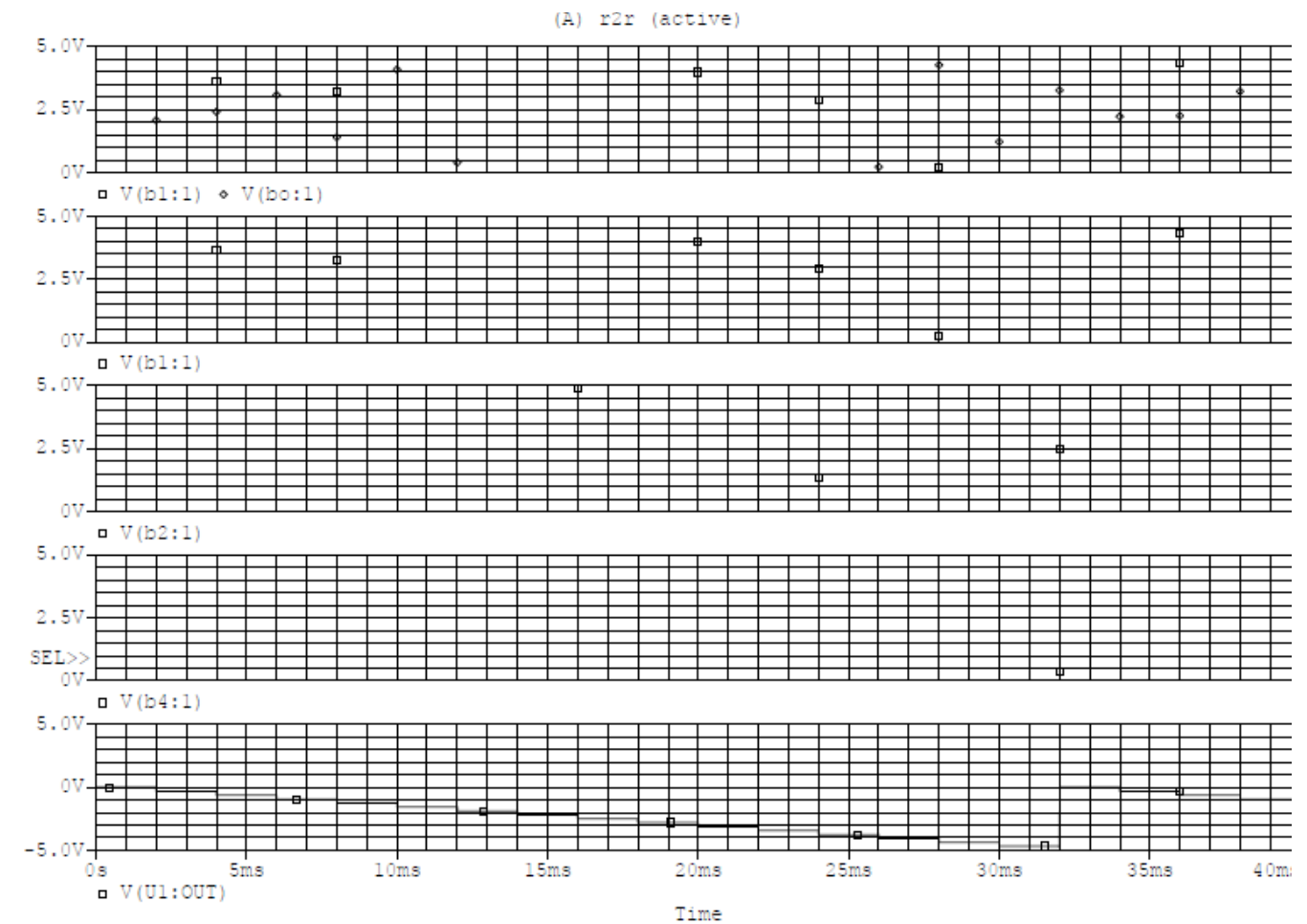| ON TIME	2ms	4ms	8ms	16ms
OFF TIME	2ms	4ms	8ms	16ms

In Transient window, type

1. Print Step = 0
2. Final time = 40 ms

The circuit R2R is simulated.

(A) r2r (active)

(A) r2r (active)

V(U1:OUT)
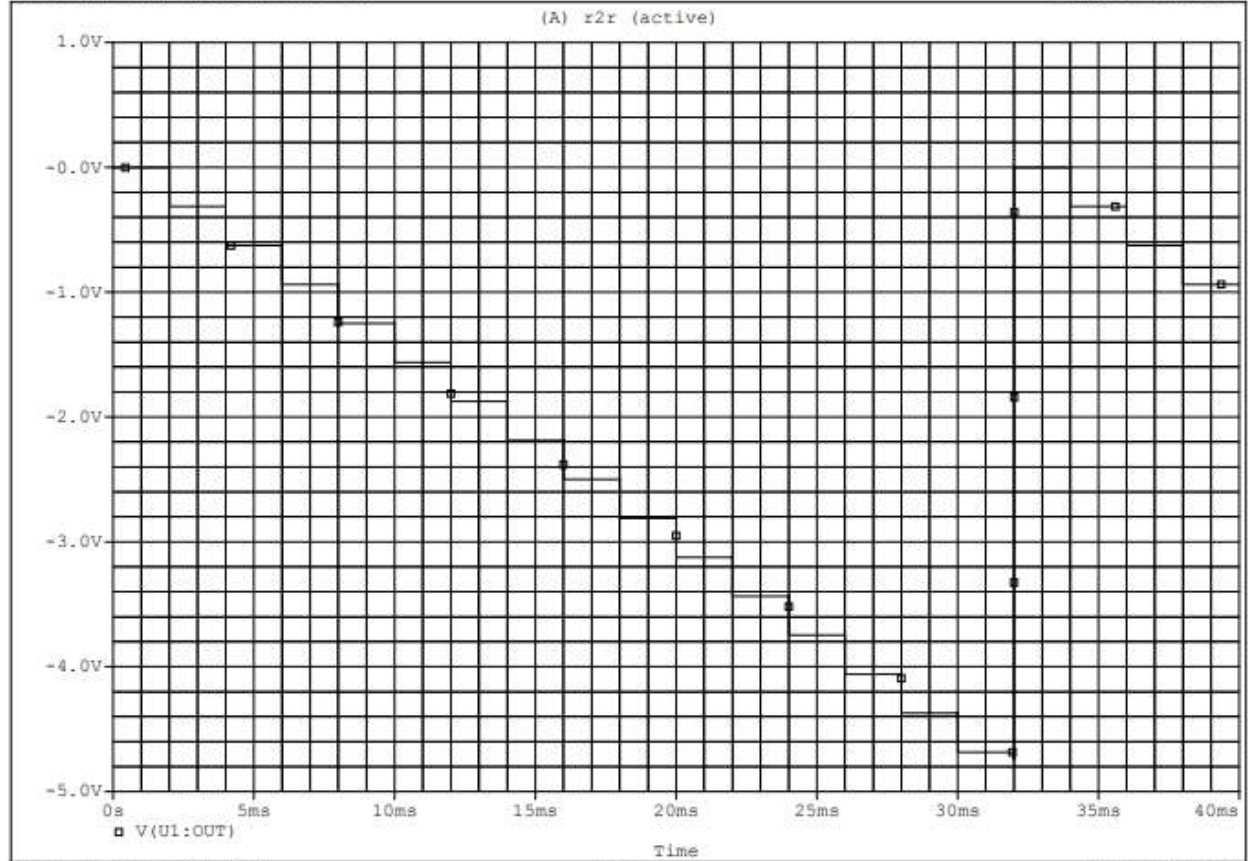
Time

Date: December 20, 2023

Page 1

Time: 13:05:36