NCR CAMPUS, MODINAGAR

DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

Analog and Digital Electronics Laboratory (18CSS201J)

Title of Experiment	**: Experiment No. 1 : Half-Wave and Full Wave Rectifiers**
Name of the candidate	**: ANANYA GUYPTA**
Register Number	**: RA1911003030265**
Branch-Section	**: CSE-I**

Experiment No. 1 : Half-Wave and Full Wave Rectifiers

Aim: Design and implement half and full wave rectifiers using PSPICE.
Apparatus Require: PSPICE Software
Theory: we will simulate a simple half-wave rectifier circuit.
There are two key differences: The source is sinusoidal and there is a diode in the circuit.

The sinusoidal source VSIN is in the SOURCE library.

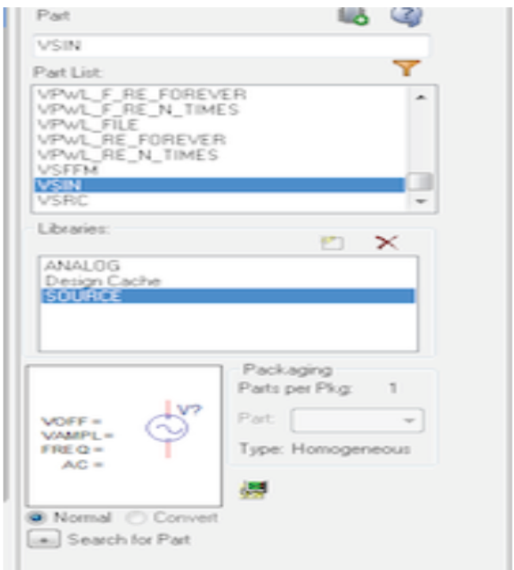
The parameters are:

VOFF - DC offset voltage. (Just like the DC offest on the function generators in lab.) Generally, we set this to zero.

VAMPL - amplitude (peak or magnitude) of the sine wave.

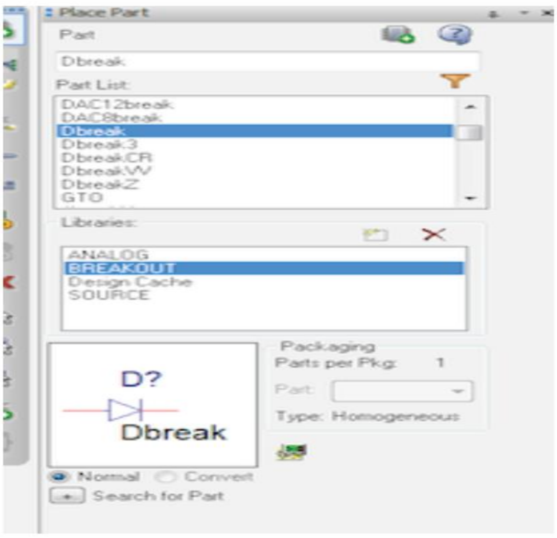
FREQ - frequency in hertz.

AC - set it to zero.



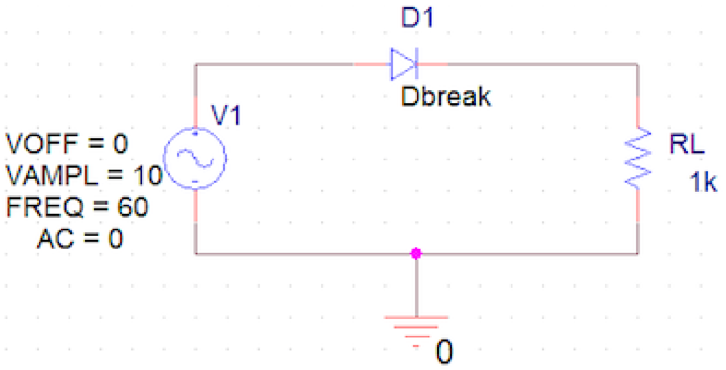
There are many diode models that are available. The one most useful to us is the "Dbreak" model, which is located in the "BREAKOUT" library. Therefore, we need to add that library to the project.
The BREAKOUT models are useful because we can easily change the parameters of the devices (i.e. we can "break out" the parameters and adjust them). Most of the other models for electronic components are not easily adjustable.

Add a the BREAKOUT library to your collection and then choose the Dbreak model for the diode. (This is the simplest diode model.)



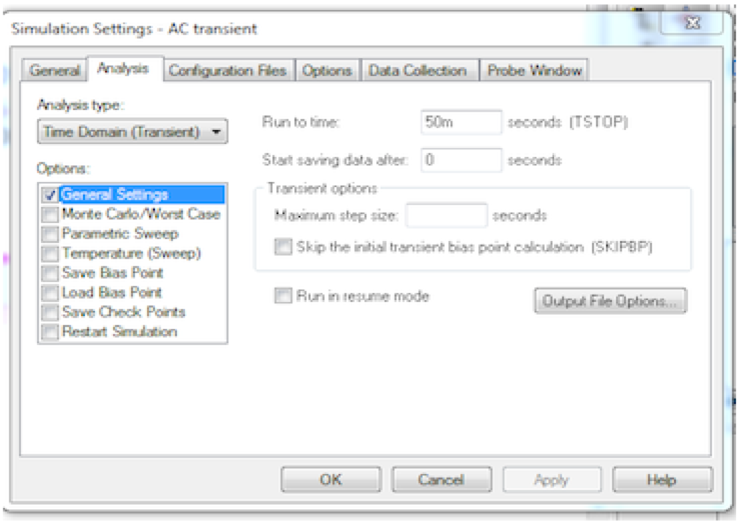
Build the circuit:

Gather the components for a simple half-wave rectifier (VSIN, Dbreak, and resistor), and wire them together, as shown below. Don't forget the ground. For the sinusoid, we use an amplitude of 10 V and frequency of 60 Hz, as if the source were being provided by a transformer.

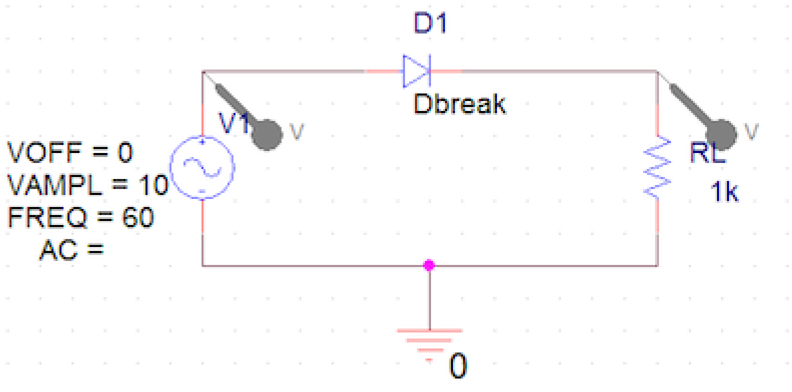


Set up the simulation:

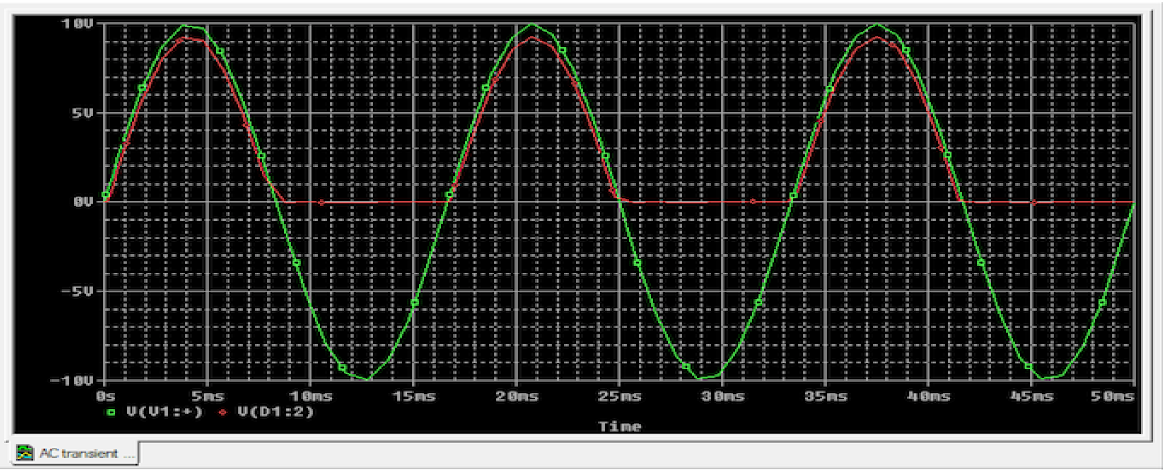
Again, will use a transient analysis, so that we can see how the voltages change with time. Set the "Run to time" to 50 ms, which will be a bit more than 3 periods of the sine wave.



Add voltage probes:



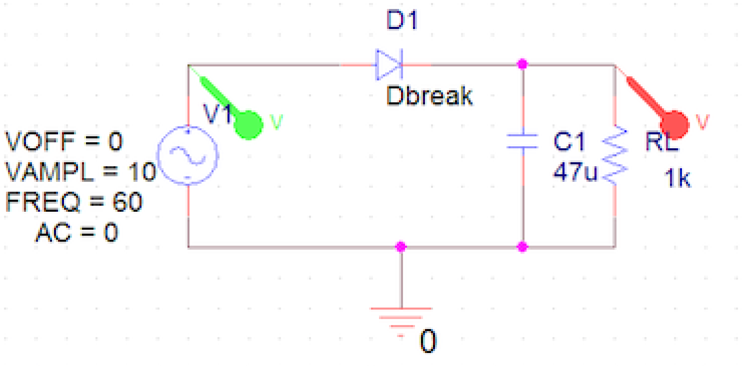
Run the simulation and observe the results:

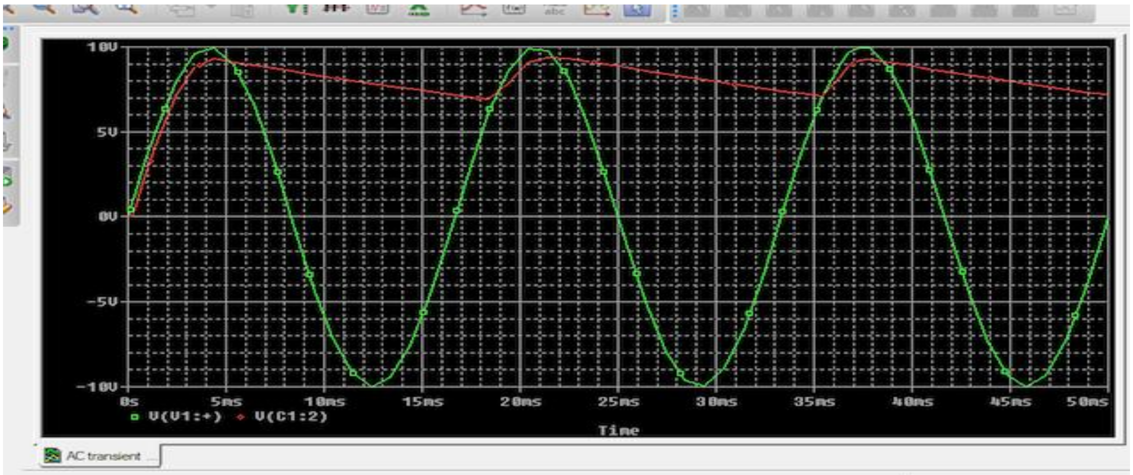


The result is the waveform of a classic half-wave rectifier. Note that the traces are not really smooth. When PSPICE chose the interval, the step size was a bit too large to make a really smooth graph. This might be an instance when it would be useful to go back and enter a value for the maximum step size in the simulation profile.

Peak rectifier

Add a 47-µF capacitor in parallel with the load resistor to make a peak rectifier.

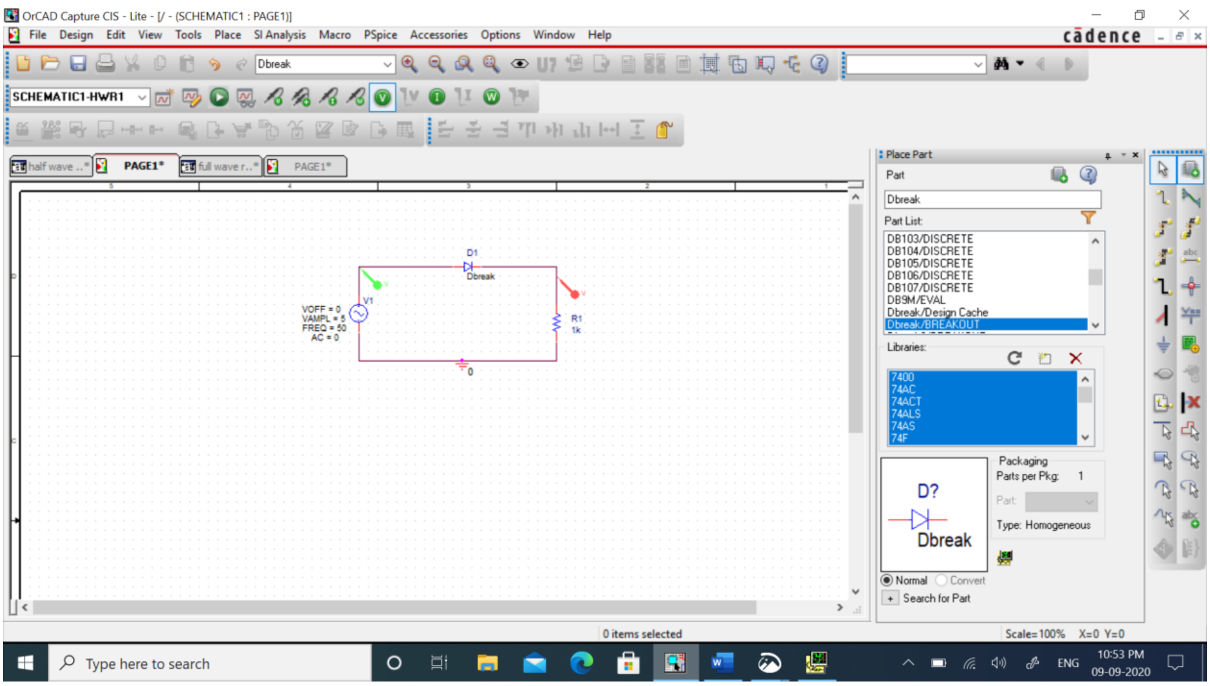




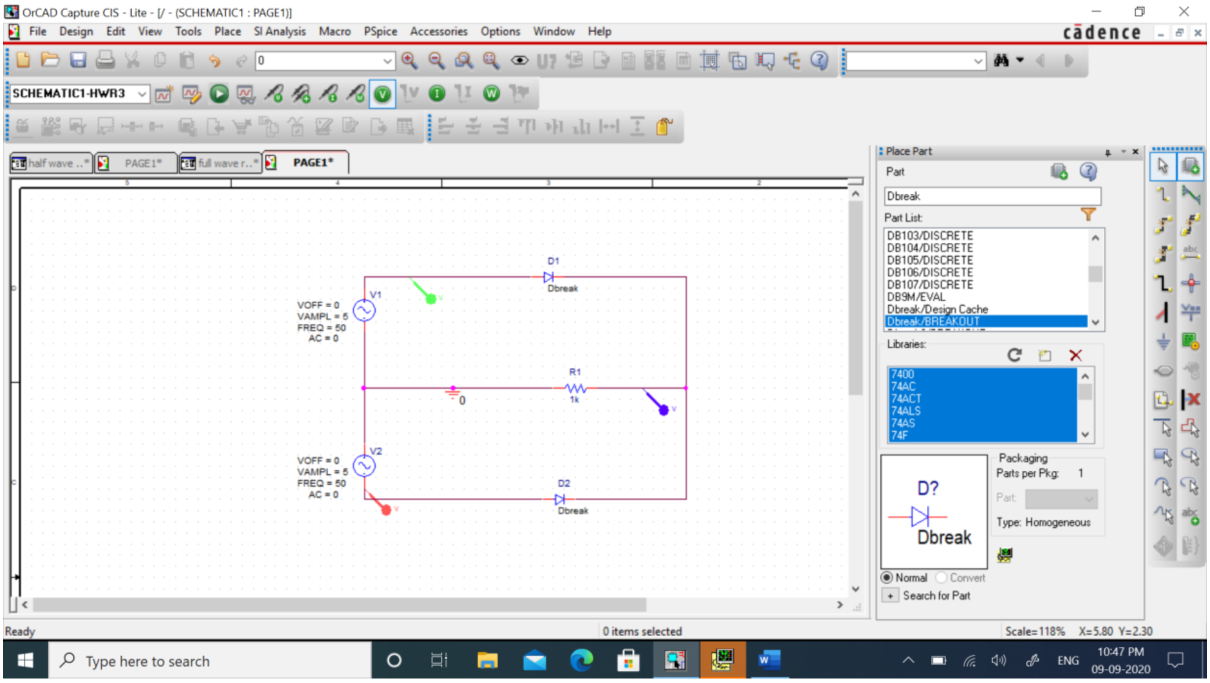
If we needed, we could use the plot cursors to measure various points on the output voltage curve to determine the ripple voltage.

CIRCUIT DIAGRAM:

HALF WAVE RECTIFIER:



FULL WAVE RECTIFIER:



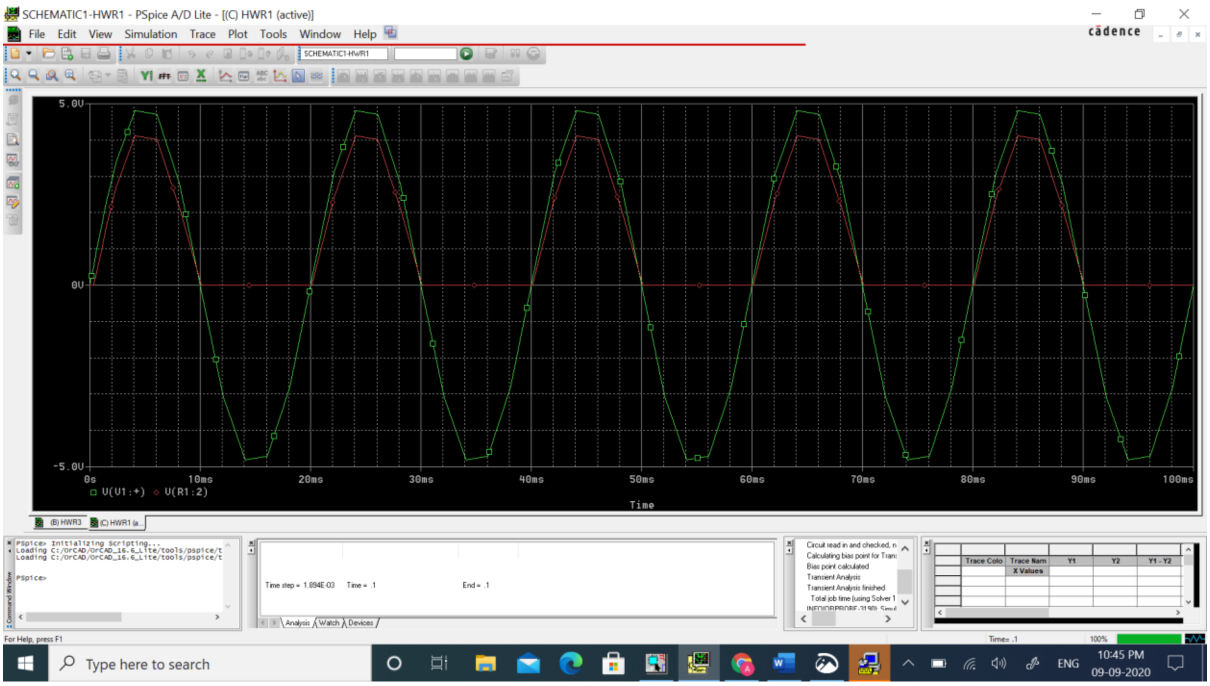
Program:

```
*HALF WAVE rectifier
VIN 2 0 sin(0 220V 50HZ)
RL 5 0 500
RS 2 1 10
L1 1 0 2000uH
L2 3 0 20uH
K1 L1 L2 0.99999
D1 3 5 D1N4009
.model D1N4009 D(Is=544.7E-21 N=1 Rs=.1 Ikf=0 Xti=3 Eg=1.11 Cjo=4p M=.3333
+ Vj=.75 Fc=.5 Isr=30.77n Nr=2 Bv=25 Ibv=100u Tt=2.885n)
.tran 0.2m 200m
.plot tran v(3), v(5)
.probe
.end
*FULL WAVE rectifier
Vin 2 0 sin(0 230V 50HZ)
RL 5 4 1000
RS 2 1 10
```

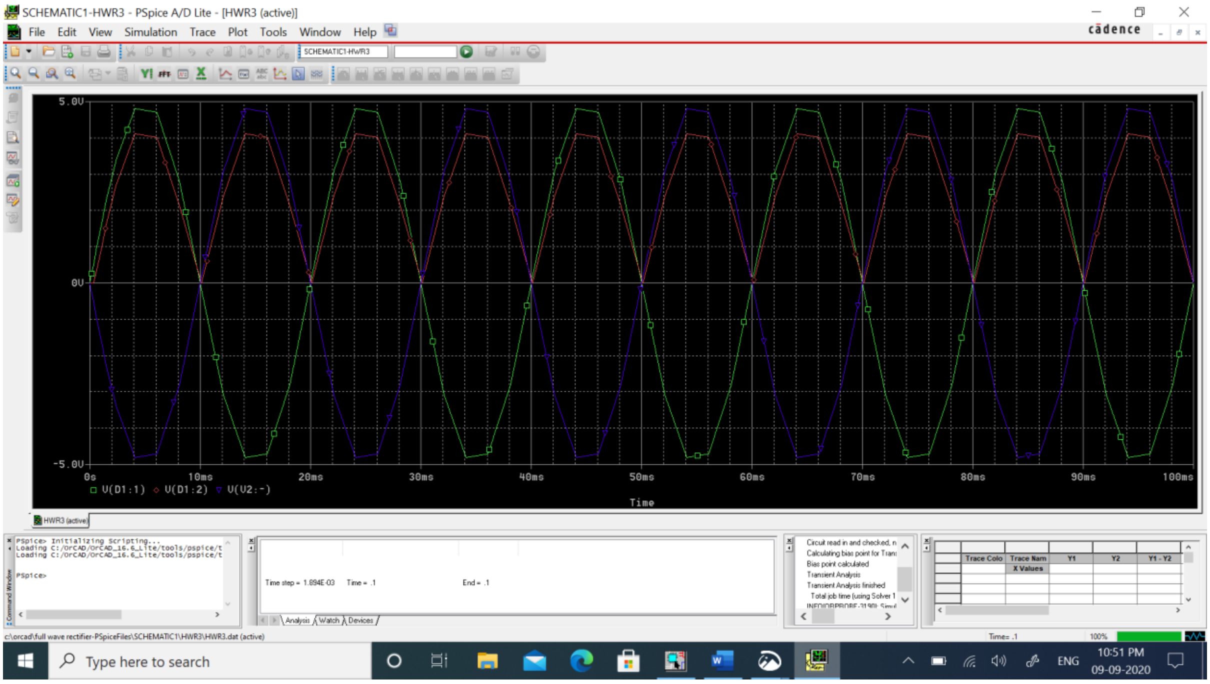
L1 1 0 2000
L2 3 4 10
L3 4 0 10
K1 L1 L2 L3 0.99
D1 0 5 D1N4009
D2 3 5 D1N4009
.model D1N4009
D(Is=544.7E-21 N=1 Rs=.1 Ikf=0 Xti=3 Eg=1.11 Cjo=4p M=.3333
+ Vj=.75 Fc=.5 Isr=30.77n Nr=2 Bv=25 Ibv=100u Tt=2.885n)
.tran 0.2ms 200ms
.probe
.end

Result:

HALF WAVE RECTIFIER:



FULL WAVE RECTIFIER:



Result: We are able to visualise the expected output of Half-wave and Full-wave Rectifier circuit as given in theory.

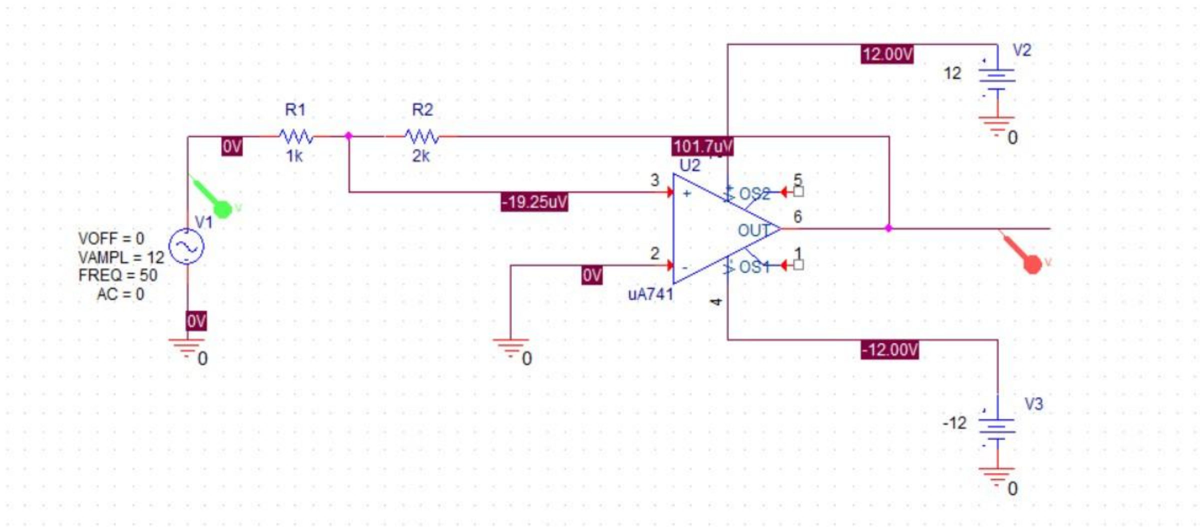
Title of Experiment	**: Experiment No. 2 : Schimitt Triffer Using Op-Amp using PSPICE**
Name of the candidate	**: ANANYA GUPTA**
Register Number	**: RA1911003030265**
Branch-Section	**: CSE-I**

Experiment No. 2 : Schimitt Triffer Using Op-Amp using PSPICE

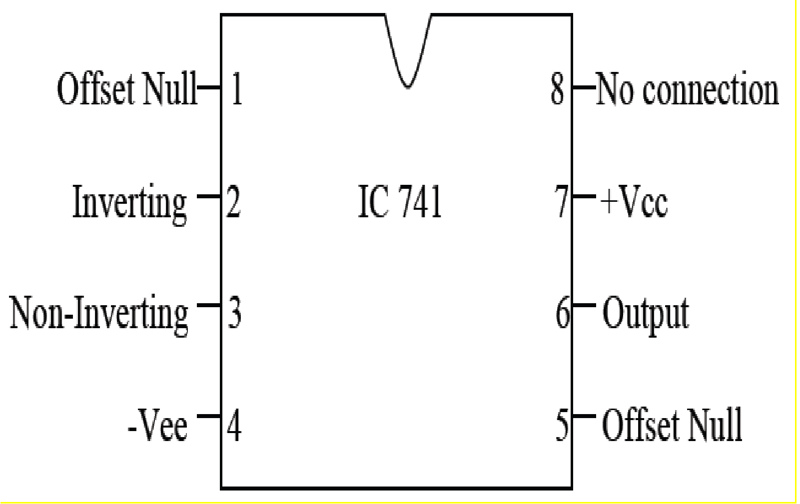
Aim: Design and implement schimitt trigger using op-amp using PSPICE.
Apparatus Require: PSPICE Software

Theory:
Schmitt trigger is useful in squaring of slowly varying i/p waveforms.Vin is applied to inverting terminal of op-amp .Feedback voltage is applied to the non-inverting terminal. LTP is the point at which output changes from high level to low level .This is highly useful in triangular waveform generation, wave shape pulse generator, A/D convertor etc.

Circuit Diagram:



Pin Diagram:



Procedure:

The connections are made as shown in the circuit diagram. The signal which has to be made square is applied to the inverting terminal . Here the i/p is a sine waveform. The supply voltage is switched ON and the o/p waveform is recorded through CRO. The UTP and LTP are also found and the theoretical and
practical values are verified.

LTP = R1/ (R1 + R2) X(-Vsat)

UTP = R2 /(R1 + R2) X(+Vsat)

Design :

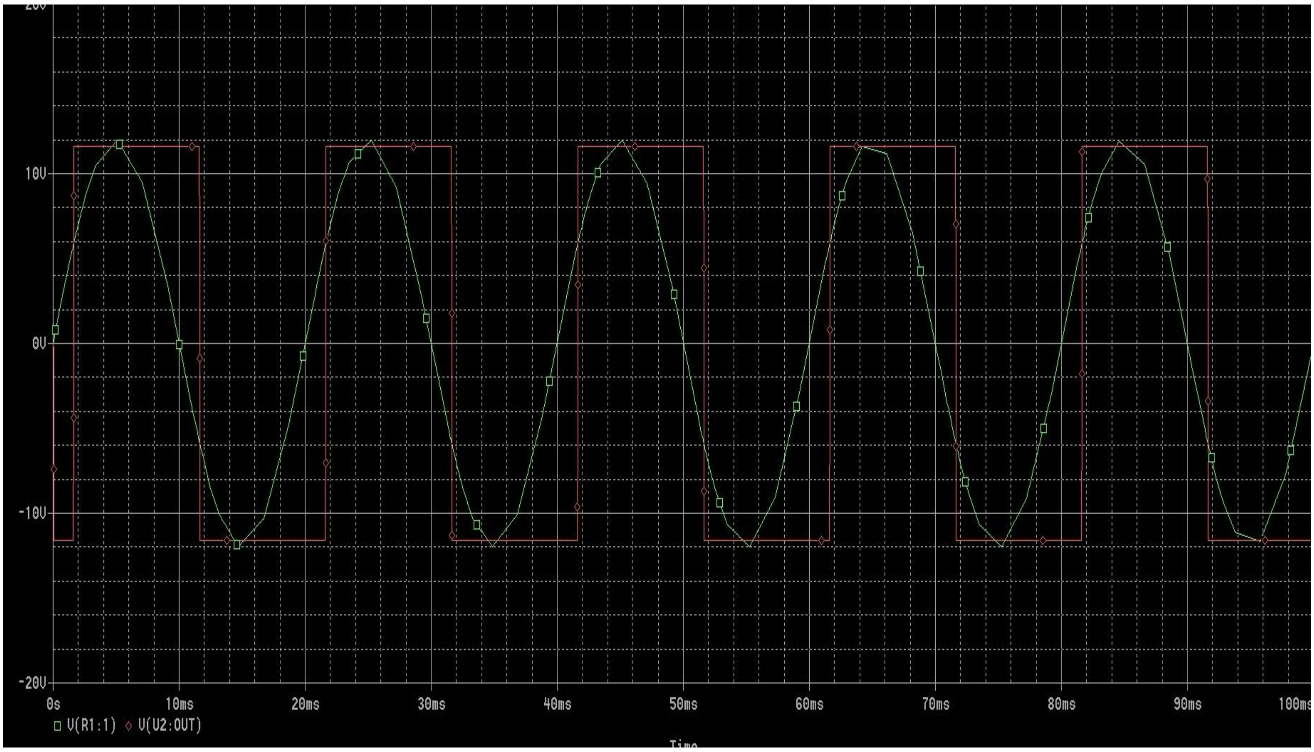
+Vsat= +Vcc=15v

-Vsat= -Vee= -15v

Program:

```
.LIB.EVAL.LIB
VCC 5 0 DC 15
VEE 0 6 DC 15
VIN 1 0 SIN(0 4 100)
R1 3 0 10K
R2 3 4 100K
R3 1 2 10K
RL 4 0 10K
X1 3 2 5 6 4 UA741
.TRAN 0 30MS
.OP
.PROBE
.END
```

Result:


20V
10V
0V
-10V
-20V
0s
10ms
20ms
30ms
40ms
50ms
60ms
70ms
80ms
90ms
100ms
V(R1:1)
V(U2:OUT)