Name of the Experiment.: Diode Characteristics Curve

Circuit diagram:

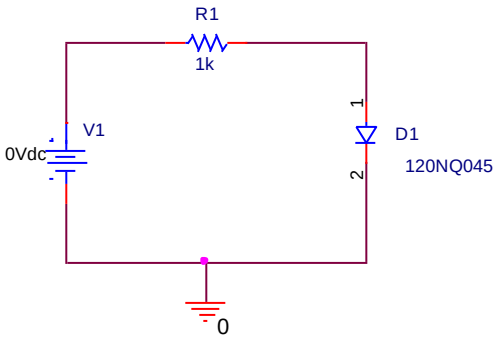
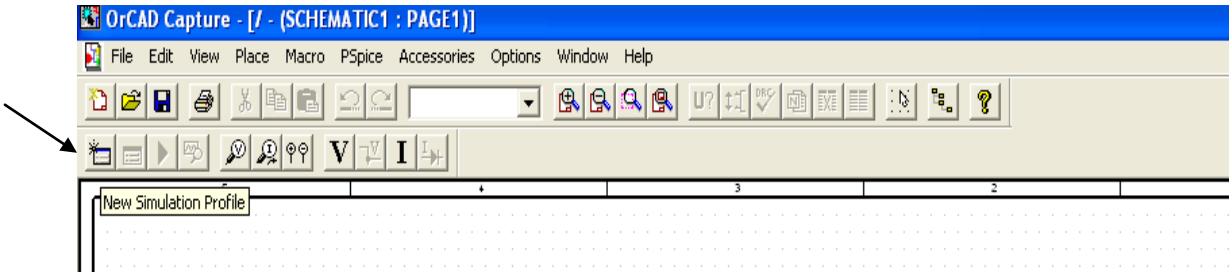


Fig. Circuit diagram for diode characteristic curve

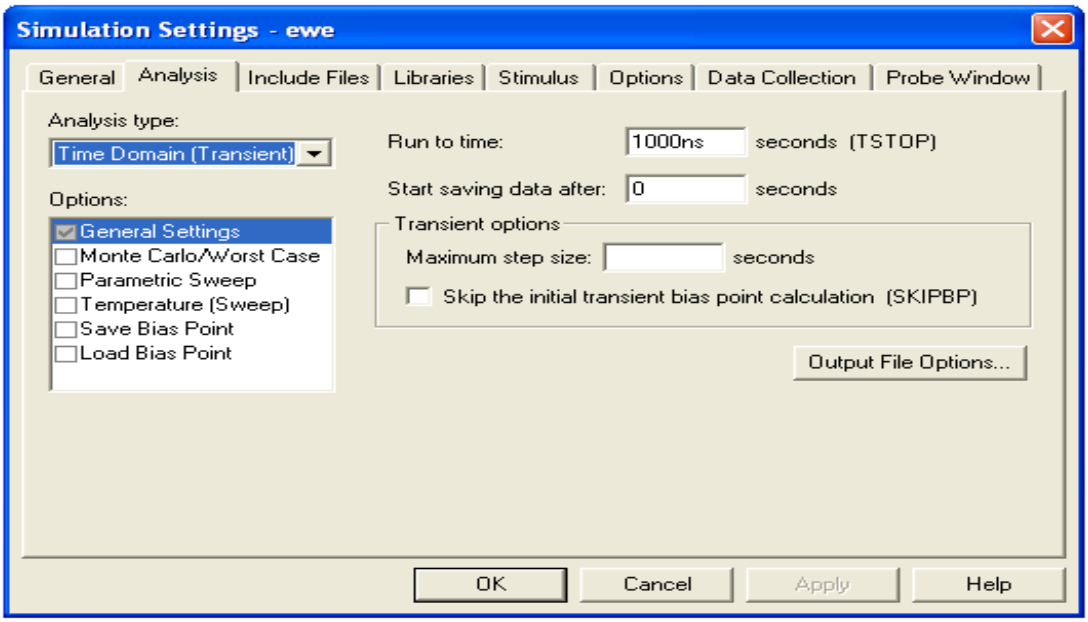
Procedure:

1. First go to OrCAD release 9.1> Capture CIS.Ink
2. Then go to File>New>Project
3. Select Analog or Mixed Signal Circuit Wizard
4. Give appropriate name
5. Set Location like F:\Shamimul\LAB
6. After clicking ok we get Analog Mixed-Mode Project Wizard.
7. From here we have to select required libraries and add to right hand side.
8. Now from place> parts or from right hand side clicking the button or shift+P we can get place parts.
9. Now selecting libraries and different parts we have to place the required parts or devices in the circuit wizard.
10. using place wire or Shift+W we can get the option to connect the circuit.
11. Now we have to select appropriate GND by place GND or Shift+G
12. Select all the libraries and choose 0/SOURCE GND.
13. Now go to new simulation profile and give a name and click create.

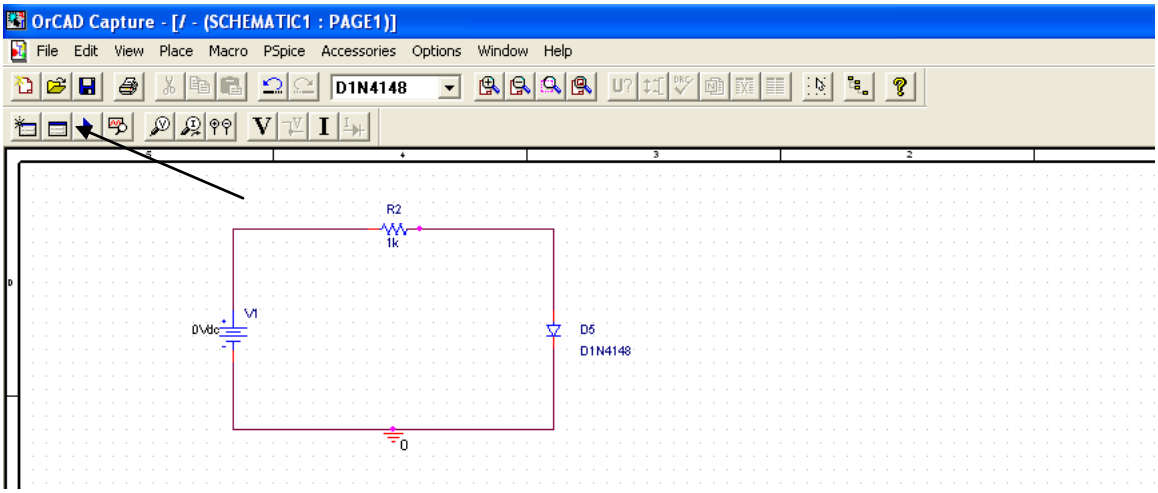


14. Now we will get Simulation Setting – ewe.

15. From Simulation setting> analysis > dc sweep> primary sweep. Assign Voltage source name set start value end value and increment. Here set initial value 0V, End value 10V and Increment 0.1V.



16. We have already drawn the following circuit.

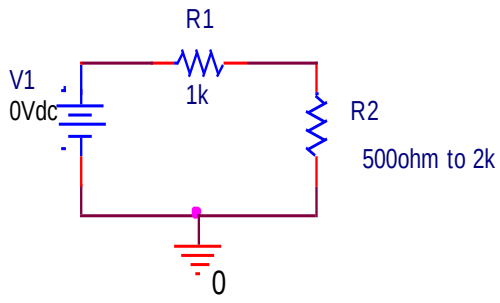


17. Now click Run Pspice showing by the arrow above

18. Now to set axis go to plot>axis setting>axis variable. Set anode voltage as axis. **19.** From add trace set diode current as y-axis.

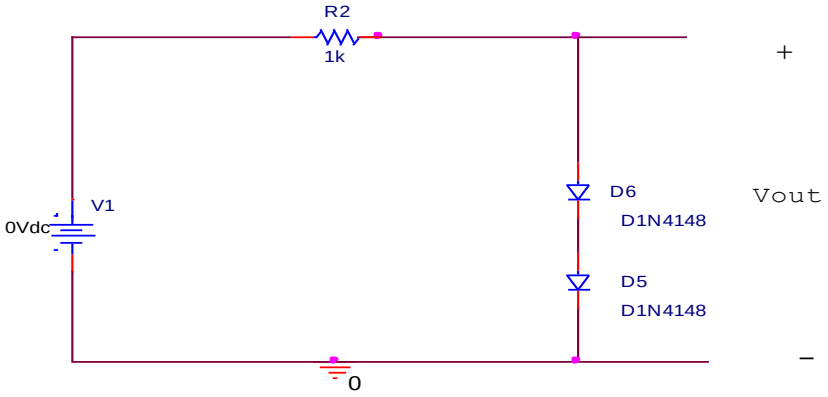
Exercise:

1.

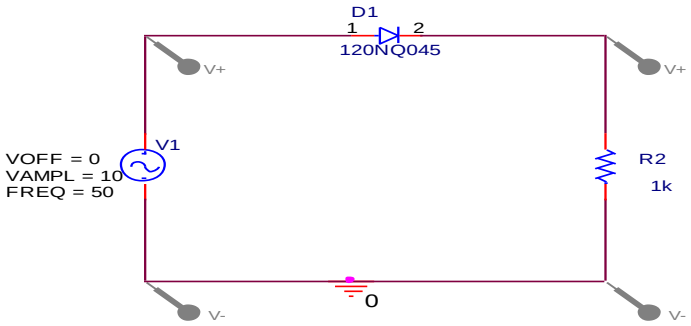


Plot the output current vs voltage for Varying input voltage from zero to ten volt for the resistor R_1.

2. Plot the output voltage across diode D6 and D5 vs current for the following circuit.



Name of the Experiment.: Half wave Rectifier Circuit



Procedure:

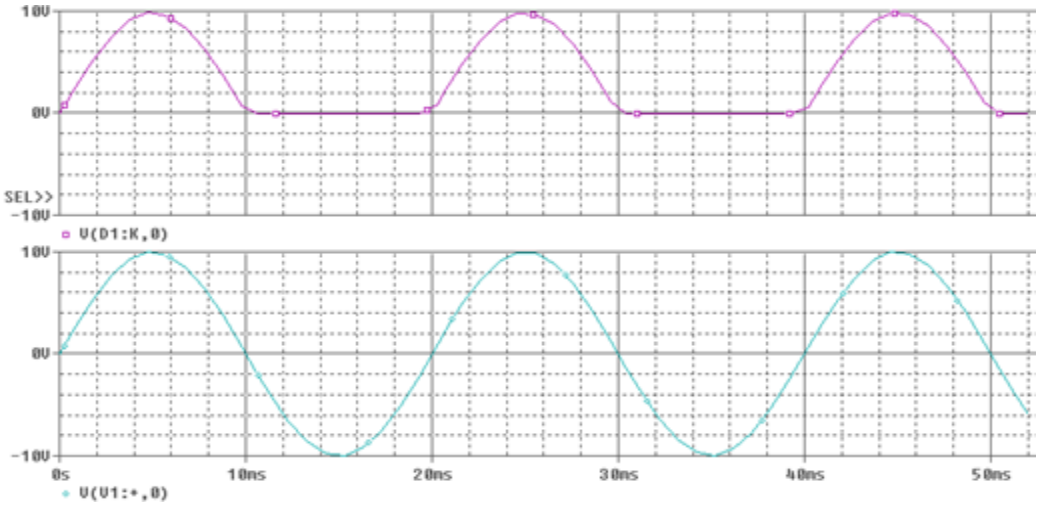
1. Construct the above circuit.
2. Here we use Vsin as source. Set V_{OFF}=0, V_{AMP}=10, FREQ=50.
3. After adding New Simulation Profile, we get simulation setting window. And from there we set Analysis>Analysis type> Time domain(Transiant)
4. Set ,

Run to time: 52ms (The time we like to observe);

Start saving data after 0sec.

Maximum step size: .001

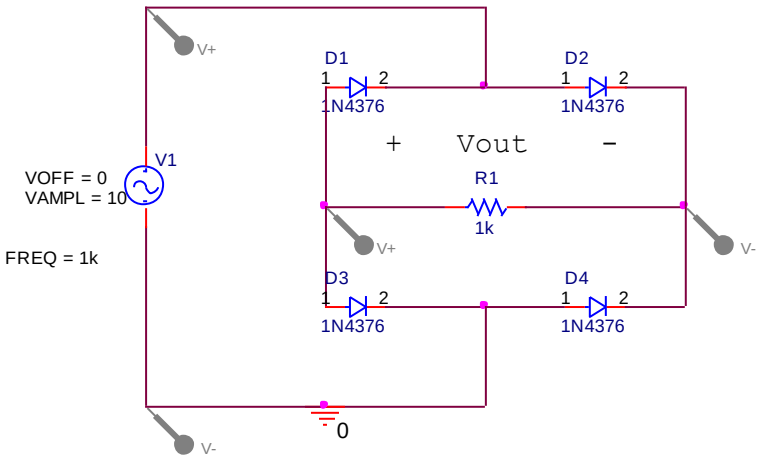
5. Now run the simulation circuit. And we observe the following output



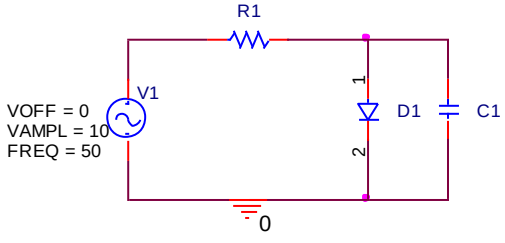
From plot > add new window we get two separate windows for two outputs.

Exercise:

1. Show input output wave shape curve for the following full wave bridge rectifier circuit.

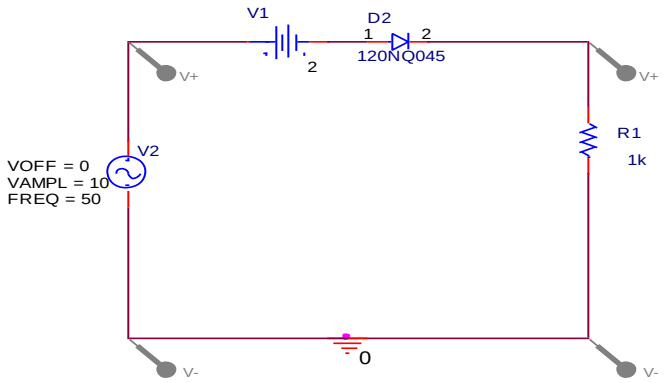


2.



Observe the input and output wave shapes of the above rectifier circuit for different values of capacitance.

Name of the Experiment.: Clipper Circuit



Procedure:

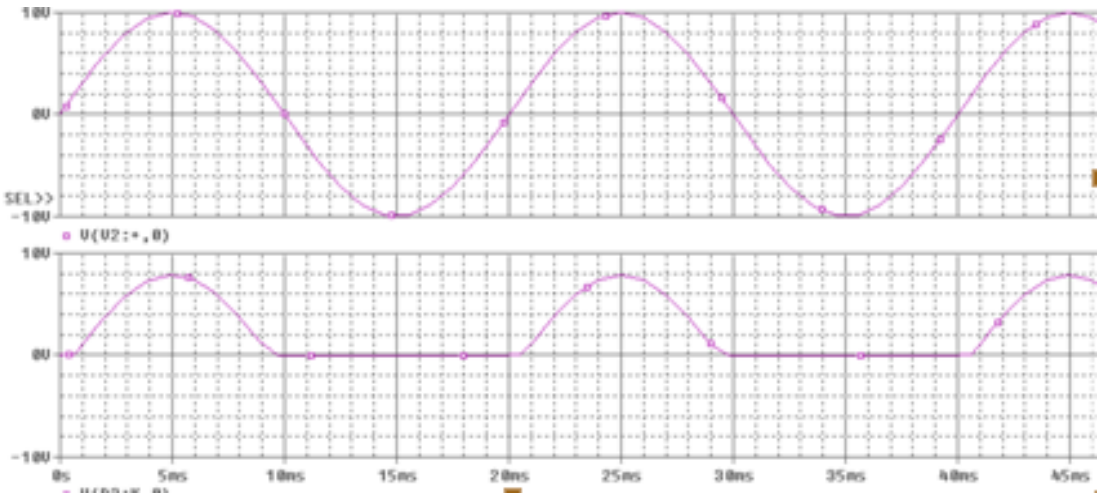
1. Construct the above circuit.
2. Here we use Vsin as source. Set V_{OFF}=0, V_{AMP}=10, FREQ=50.
3. After adding New Simulation Profile, we get simulation setting window. And from there we set Analysis>Analysis type> Time domain (Transiant)
4. Set ,

Run to time: 52ms (The time we like to observe);

Start saving data after 0sec.

Maximum step size: .001

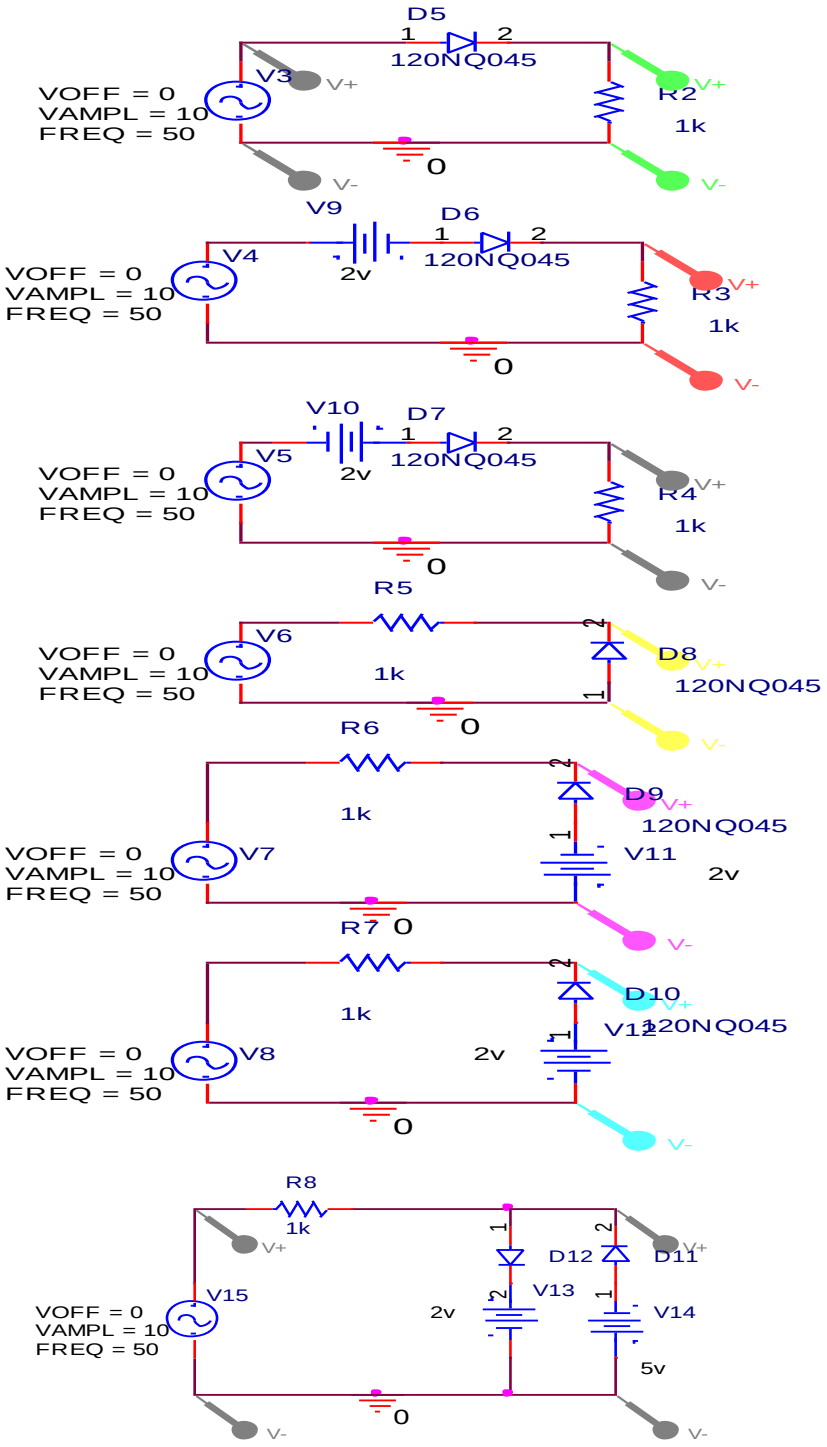
5. Now run the simulation circuit. And we observe the following output



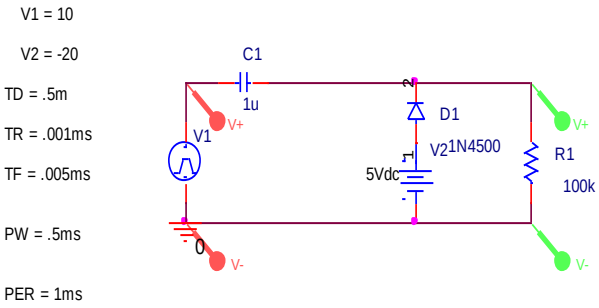
From plot > add new window we get two separate windows for two outputs.

Exercise:

1. Show the input output waveform of the following clipper circuits



Name of the Experiment.: Clamper Circuit



Where,

V1=Maximum voltage at +ve direction

V2=Maximum voltage at –ve direction

TD= Time duration from 0 to end of 1st pulse

TR= Time for rise.

TF= Time for fall.

PW=Pulse width/ time of half cycle

PER= Total time period.

Procedure:

1. Construct the above circuit.

2. Here we use Vpulse as source. Set C_1=1uf, R_1=100k, V_1=10V, V_2=-20V,TD=.5ms, TR=.001ms, TF=.005ms, PW= .5ms, PER=1ms.

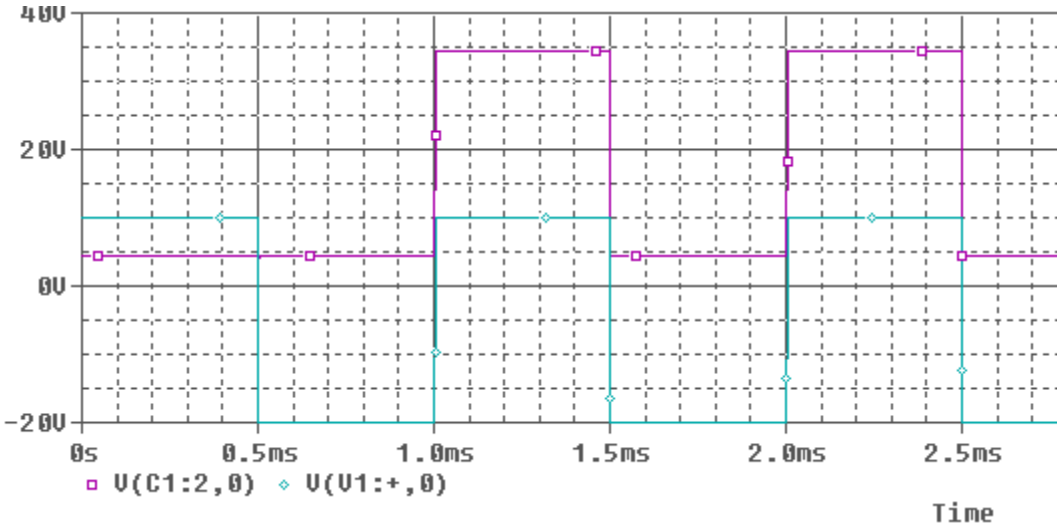
3. After adding New Simulation Profile, we get simulation setting window. And from there we set Analysis>Analysis type> Time domain(Transiant)

6. Set ,Run to time: 52ms (The time we like to observe);

Start saving data after 0sec.

Maximum step size: .001

7. Now run the simulation circuit. And we observe the following output

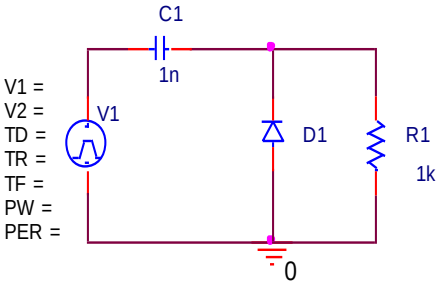


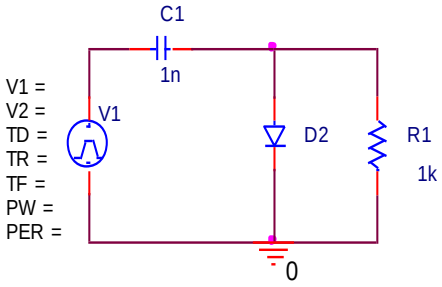
From plot > add new window we can get two separate windows for two outputs

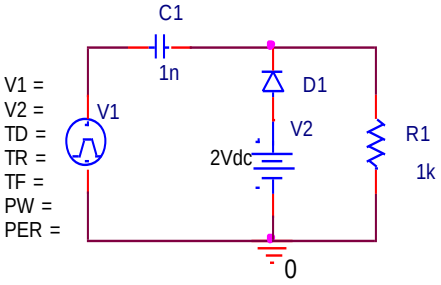
Exercise:

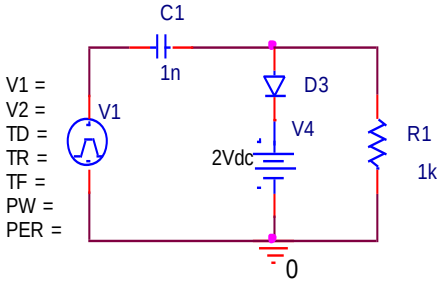
Find the output voltage wave forms for the following clampers.

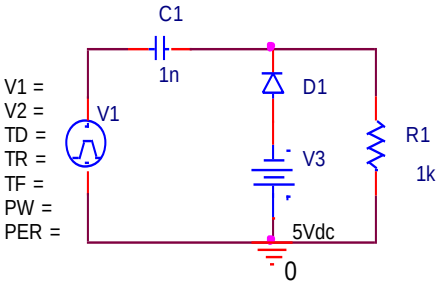
Use Vpulse as source. Set C_1=1uf, R_1=100k, V_1=10V, V_2=-10V, TD=.5ms, TR=.001ms, TF=.005ms, PW= .5ms, PER=1ms.

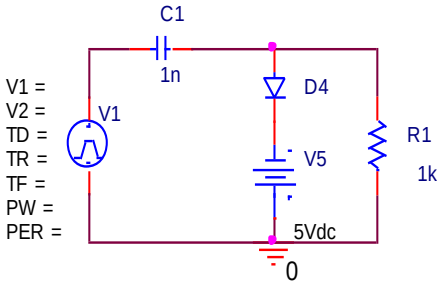




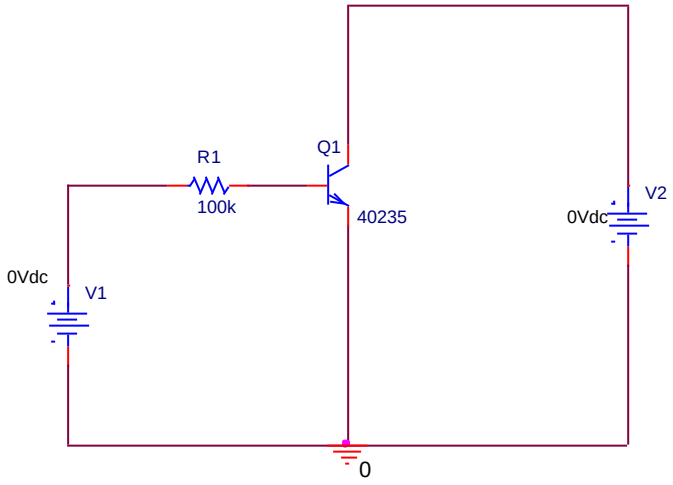








Name of the Experiment.: Input Output characteristics of BJT (CE)



Procedure:

1. Construct the above circuit.
2. After adding New Simulation Profile, we get simulation setting window. And from there we set Analysis>Analysis type> Dc Sweep
3. Set , (for getting output characteristics curve)

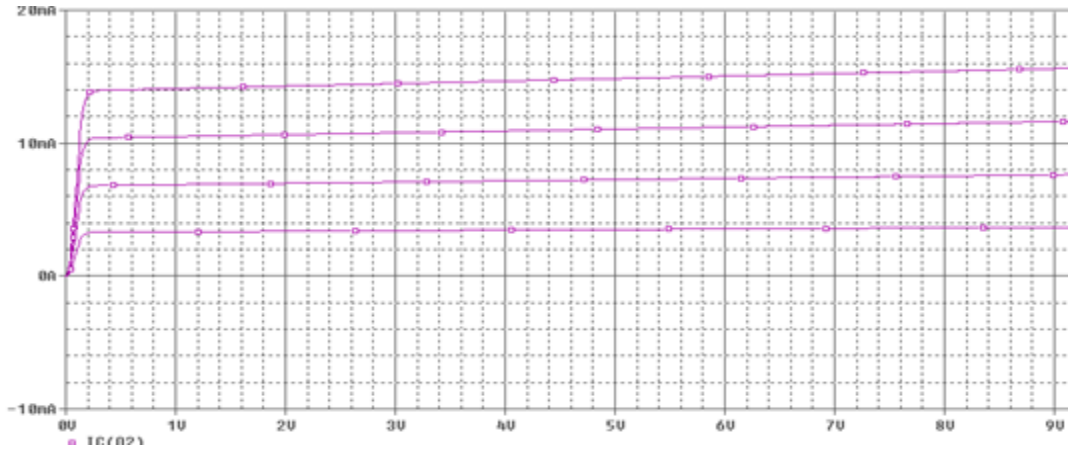
Primary Sweep: V2 (as we like to vary V_{CE}) from 0 to 10V with increment .01.

Secondary sweep: V1 (as we like to vary I_B) from 2.7 to 10V with increment 2.

For both case sweep type is linear.

4. Now run the simulation circuit. From add trace select collector current as y-axis.

And we get the following output



5. Again for getting input characteristic curve we have to go to simulation setting.

Now, set,

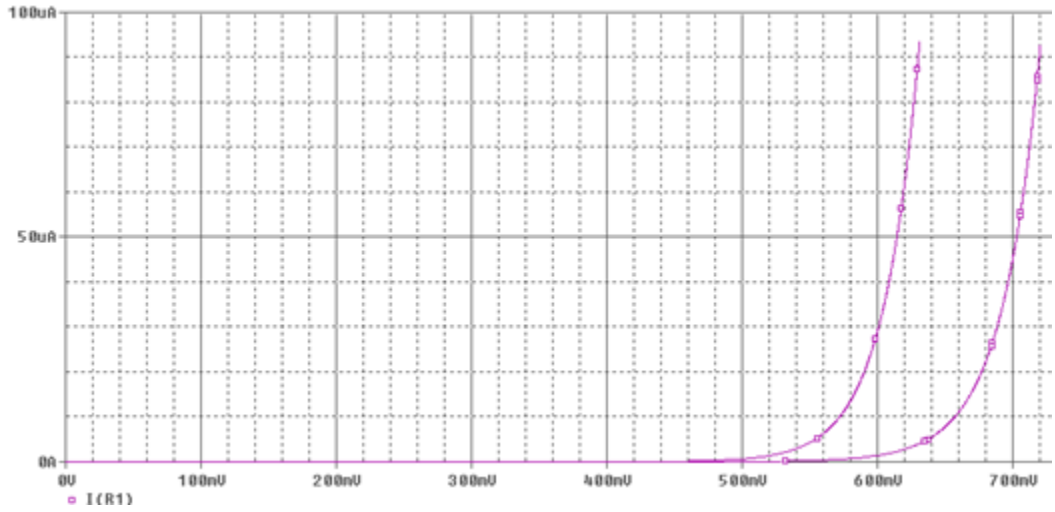
Primary sweep: V1 (as we like to set V_{BE}) from 0 to 10 with increment .01

Secondary sweep: V2 (as we like to set V_{CE}) from 0 to 25 with increment 5

6. Now click run.

7. From plot>axis setting> X axis> axis variable> set V [Q2: b] as we like to set V_{BE} as x axis.

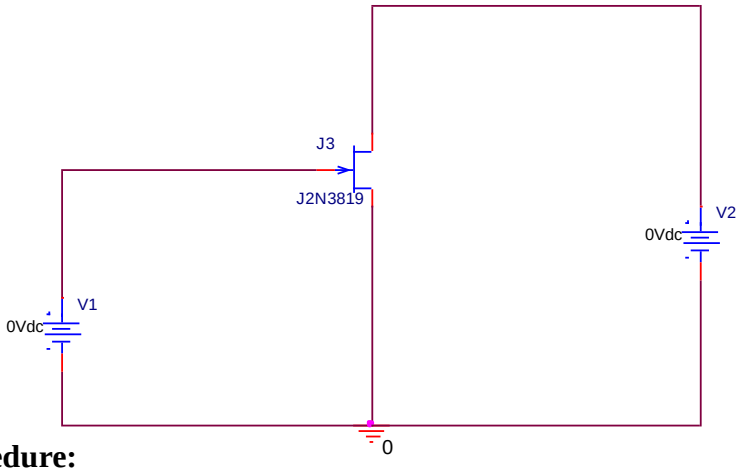
8. Now, from add trace set I_B base current as y-axis and click ok, and so we get the following output



Exercise:

1. Find the input output characteristics for p-channel BJT.
2. Find the input output characteristics for common base configuration.

Output Characteristics of FET:



Procedure:

1. Construct the above circuit.

2. After adding New Simulation Profile, we get simulation setting window. And from there we set Analysis>Analysis type> Dc Sweep

3. Set , (for getting output characteristics curve)

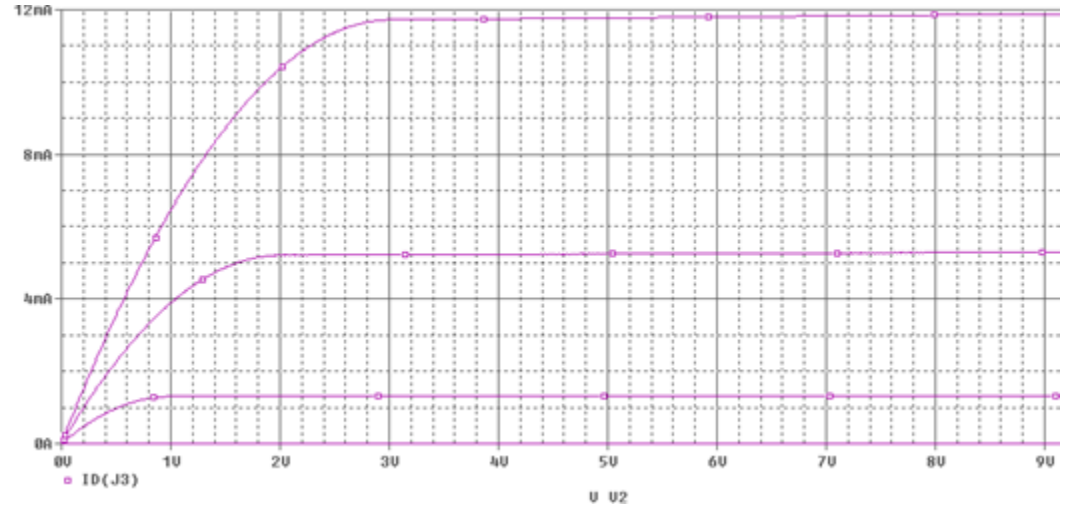
Primary Sweep : V2 (as we like to vary V_{DS}) from 0 to 10V with increment .01.

Secondary sweep: V1 (as we like to vary V_{GS}) from 0 to -5V with increment -1.

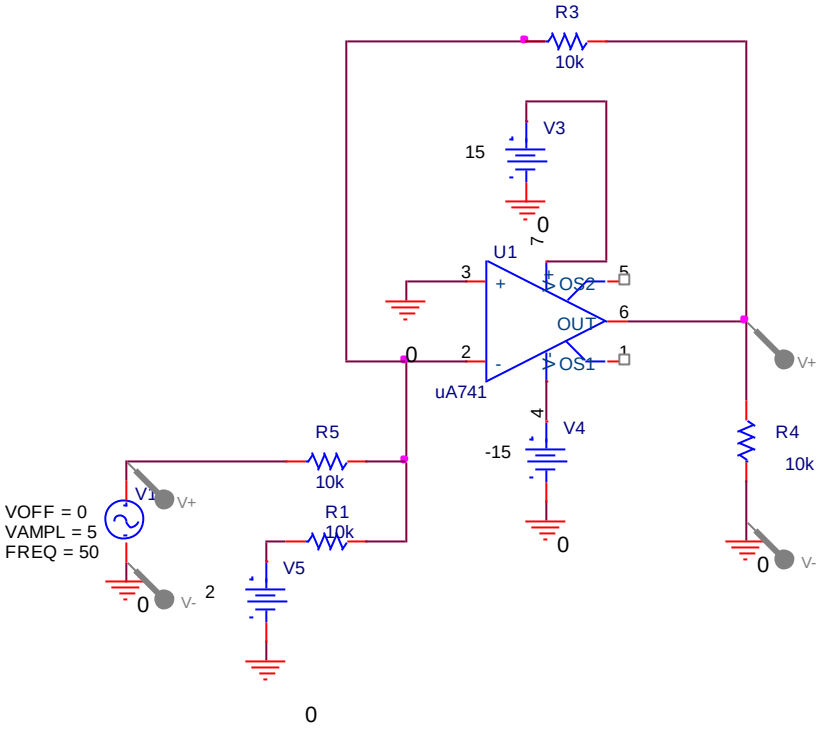
For both case sweep type is linear.

4. Now run the simulation circuit. From add trace select drain current as y-axis.

And we get the following output



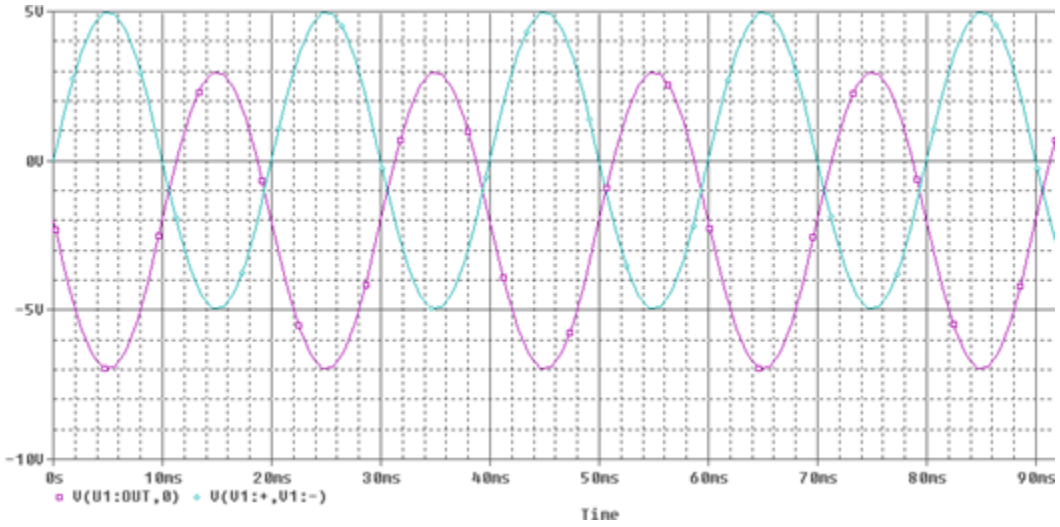
Name of the Experiment.: OPAMP DC offsetting AC signal



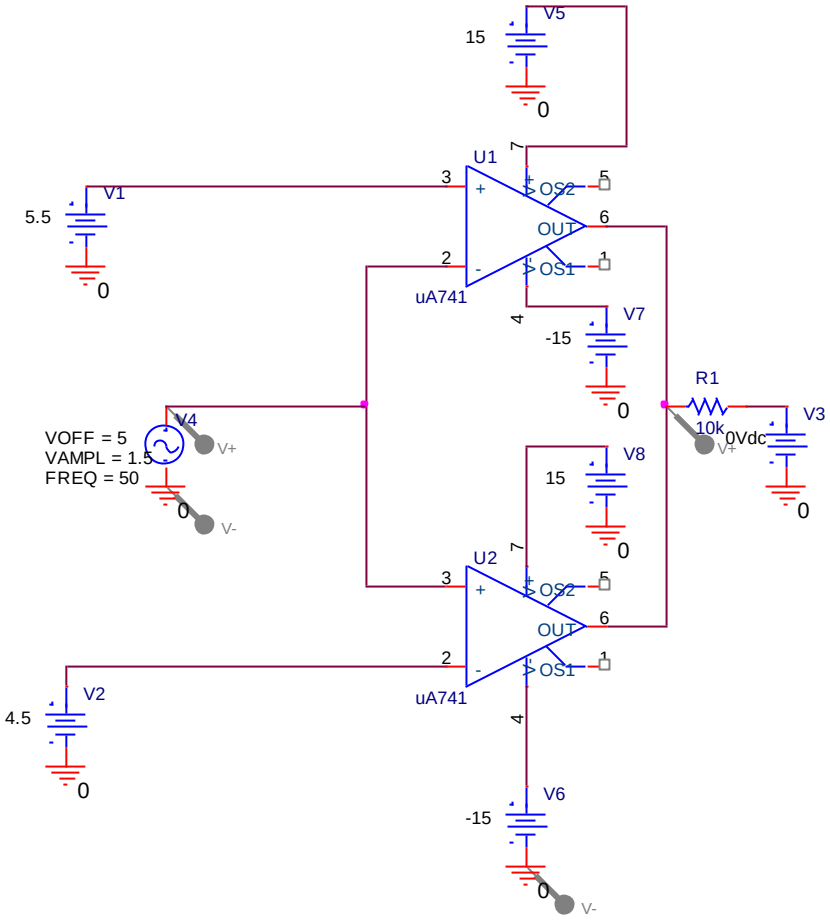
Procedure:

1. Construct the above circuit using uA741.

2. Get the following output.



Name of the Experiment.: Window Detector Circuit



Procedure:

1. Construct the following circuit.
2. Set V_{AMP}=1.5, Freq =50, V_{OFF}= 5

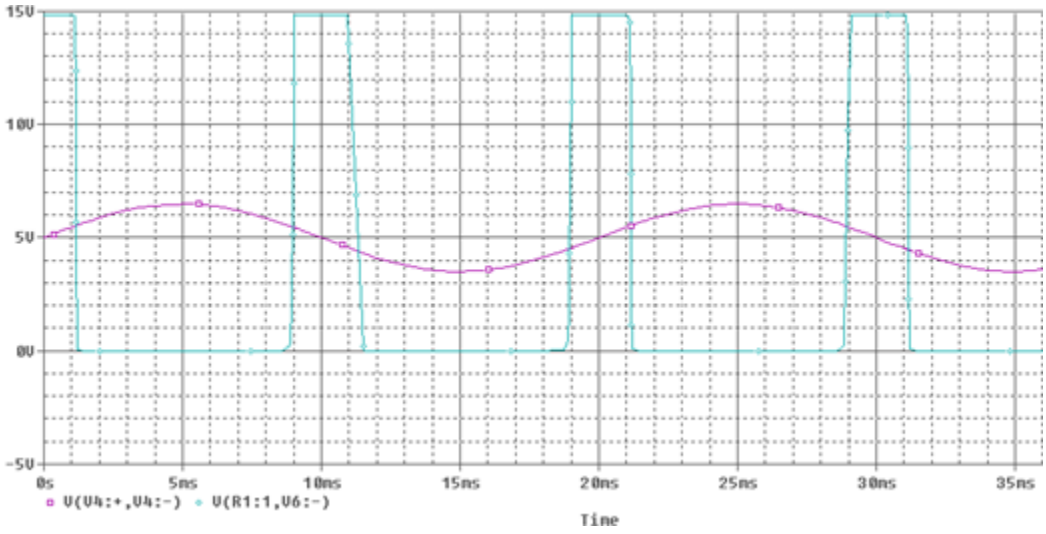
3. After adding New Simulation Profile, we get simulation setting window. And from there we set Analysis>Analysis type> Time domain(Transiant)
4. Set ,

Run to time: 40ms (The time we like to observe);

Start saving data after 0sec.

Maximum step size: .001

5. Now run the simulation circuit. And we observe the following output



Exercises for OPAMPS:

a) Input vs output characteristics curves of Zero crossing detectors (P-20).

b) Input vs output characteristics curves of positive voltage level detectors.

c) Phase shifters (P-137).

d) Free running multivibrators (P-152).

e) One shot multivibrators (P-156).

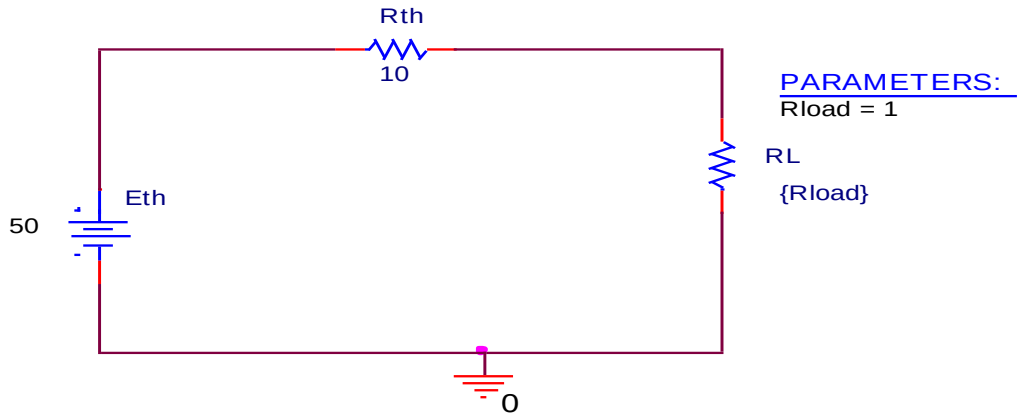
f) Unipolar triangular wave generator (P-163).

g) Inverting linear half wave rectifiers using OPAMPS (P-190).

h) Full wave rectifiers using OPAMPS (P-195).

Name of the Experiment.: Global Parameter

Maximum Power Transfer Theorem



Maximum Power Transfer Theorem:

A load will receive maximum power from a bilateral network when its load resistance value is exactly equal to the Thevinin resistance of the network as seen by the load.

RL= Rth

I= Eth/(Rth+Rload)

$PL=I^2.Rload$

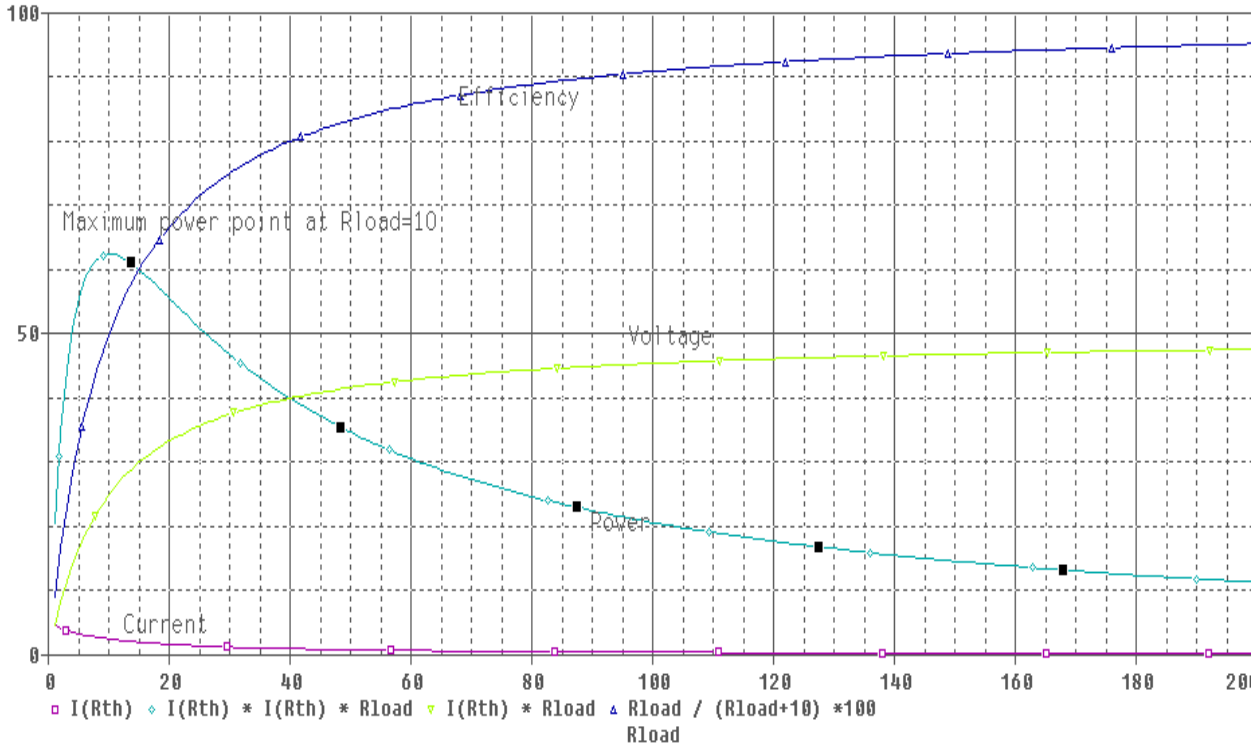
VL=I. Rload

Efficiency=PL/Ps= (Rload/Rload+Rth)×100%

Procedure:

1. Construct the above circuit. From Place part select param from the Special library. Then place it near the RL.
2. Double Click on the value of RL. Write {Rload}. Click ok.
3. Double Click on PARAM to get the 'Property editor' tool bar. Then click new to get 'add new property' tool bar. Give the name Rload and then click ok. Now go to Disply button to get 'Display Properties' and select Name and value.
4. Now × the 'property editor' toolbar.
5. Now put the value of Rload=1, below the PARAM.
6. Now go to New simulation profile and give a name and create to get a simulation setting. Go to Analysis>DC Sweep Select Global parameter. Name: Rload. Initial value: 1 , End value: 200. Increment: 1.
7. Now run the project.

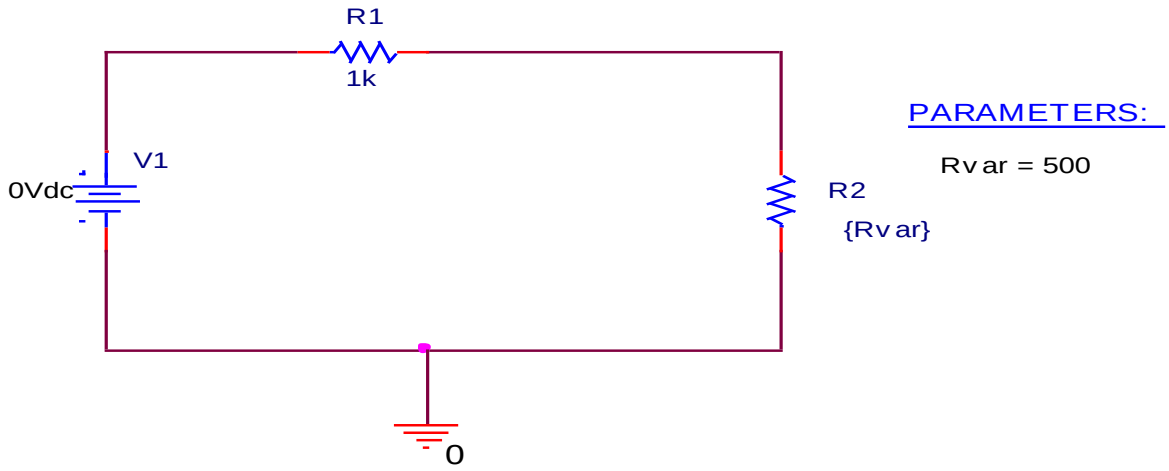
8. From Add trace select I(Rth) to get the current curve, select I(Rth)* Rload to get the voltage curve. Select I(Rth)*I(Rth)*Rload to get Power curve. To get the efficiency curve write Rload/(Rload+10)*100. To get input power Eth* I(Rth).



Exercise:

If R_L=10 fixed. And Rth varied from 0 to 200 ohm. Then now plot the curves for current, voltage, power, efficiency, Input power.

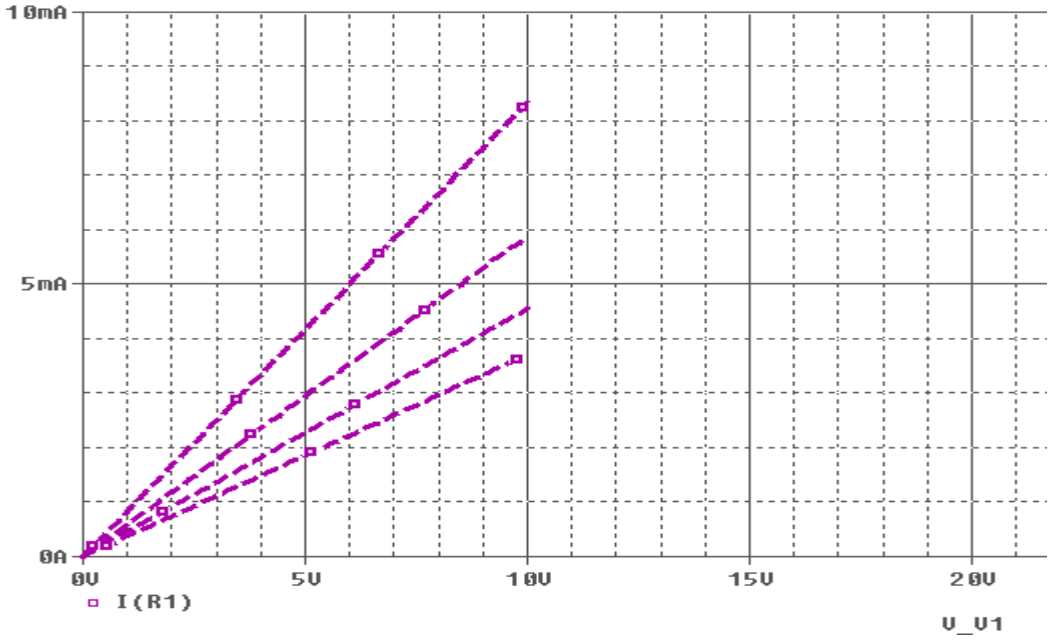
Ohms Law:



Procuedure:

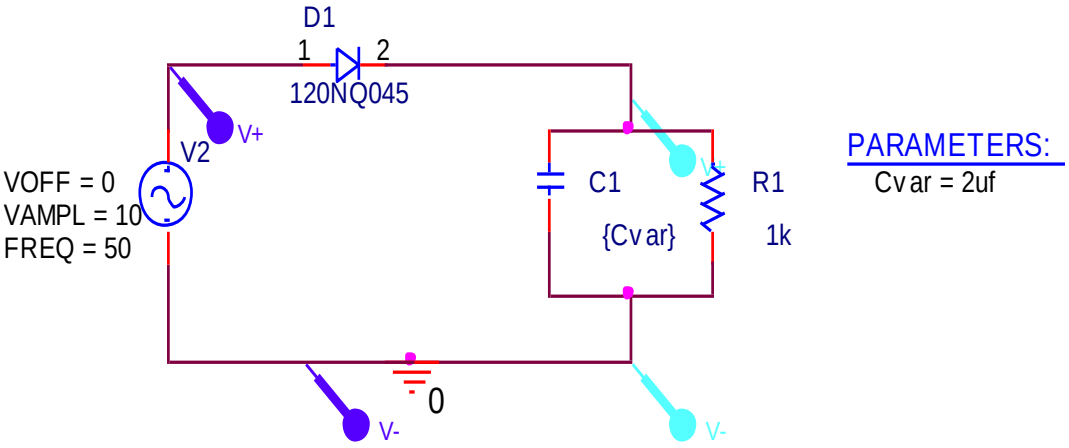
1. From the library special select param and place it beside R2.

2. Set the value of R2={Rvar}. From the property editor of Parameters add new column named Rvar and give a value 500.
3. Go to DC Sweep, **Primary sweep:** V1: 0 to 10 with .1 increment. **Secondary sweep>parametric sweep>Global parameter:** Rvar: 200 to 2000 with increment 500.
4. Run the project and select I(R2) to get the following curve.



Rectifier Circuit:

Rectifier circuit using variable capacitance:

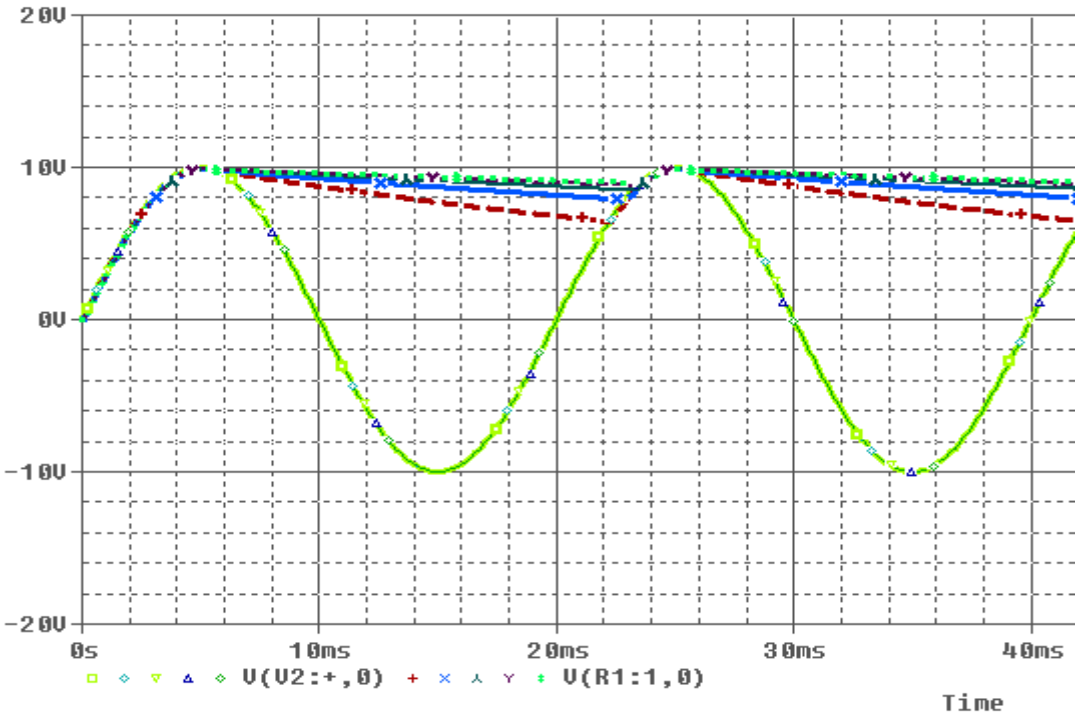


Procedure:

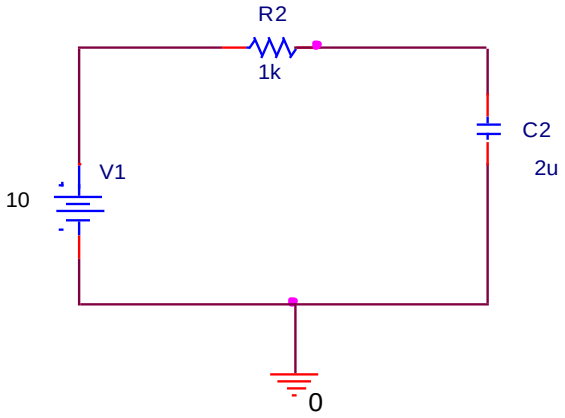
1. Here the value of the capacitor is varied as global parameter.

2. In **time domain(transient)>General setting:** 100m, .0001

Time domain(transient)>parametric sweep: Cvar: 40u, 200u,40u

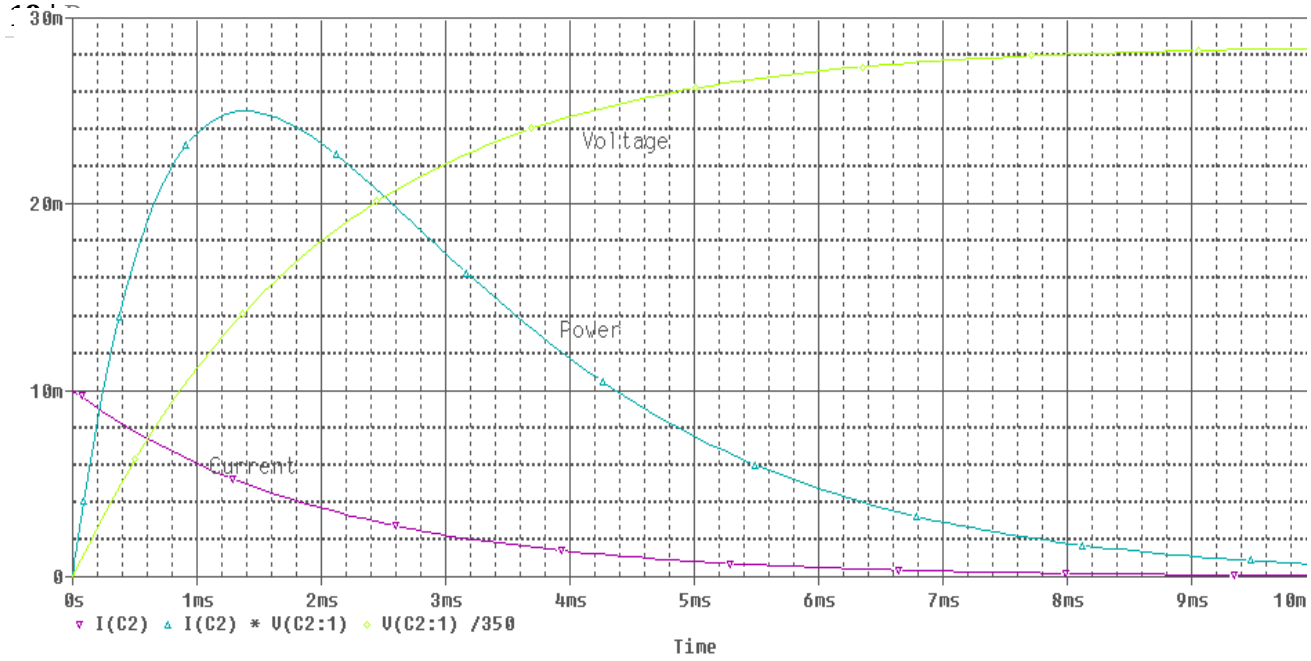


Name of the Experiment.: Power Absorbed by a capacitor

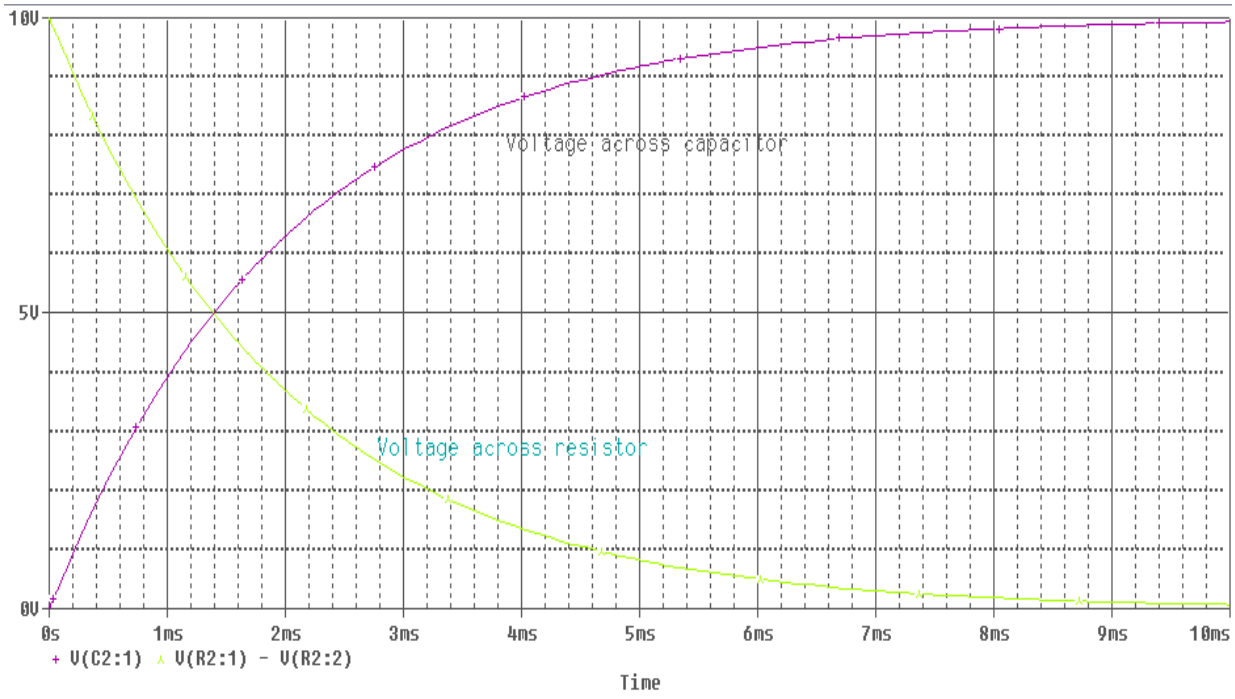


Procedure:

1. Construct the upper circuit.
2. Set run to time 10ms at simulation setting. Step size .1ms. select skip initial transient.
3. Output voltage across the source and capacitor can be observed by oscilloscope probe or from add trace to observe power select current × voltage. To observe the voltage waveform normalize it by 250.
4. Observe voltage across capacitor, current and power.



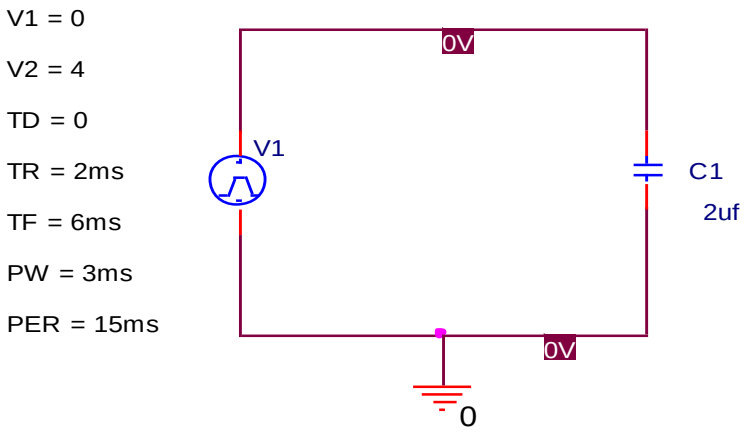
5. Voltage across capacitor and resistor can be observed.



Exercise:

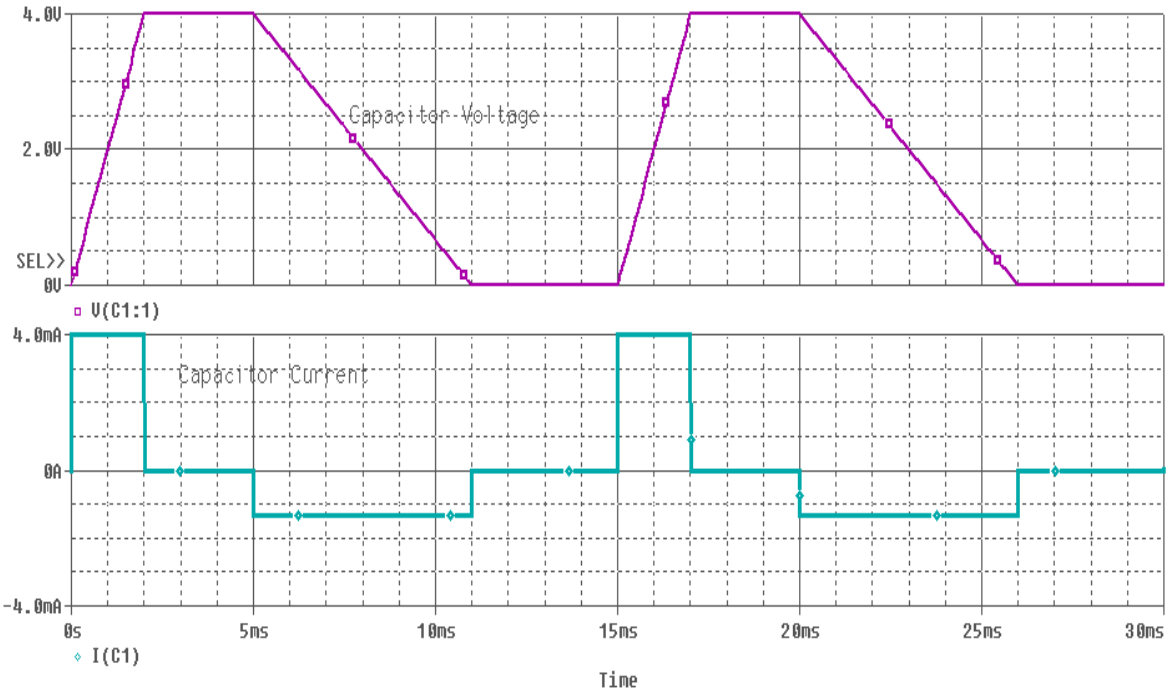
Find the voltage, current and power wave shape using an inductor.

Name of the Experiment.: Capacitor and inductor current wave shape

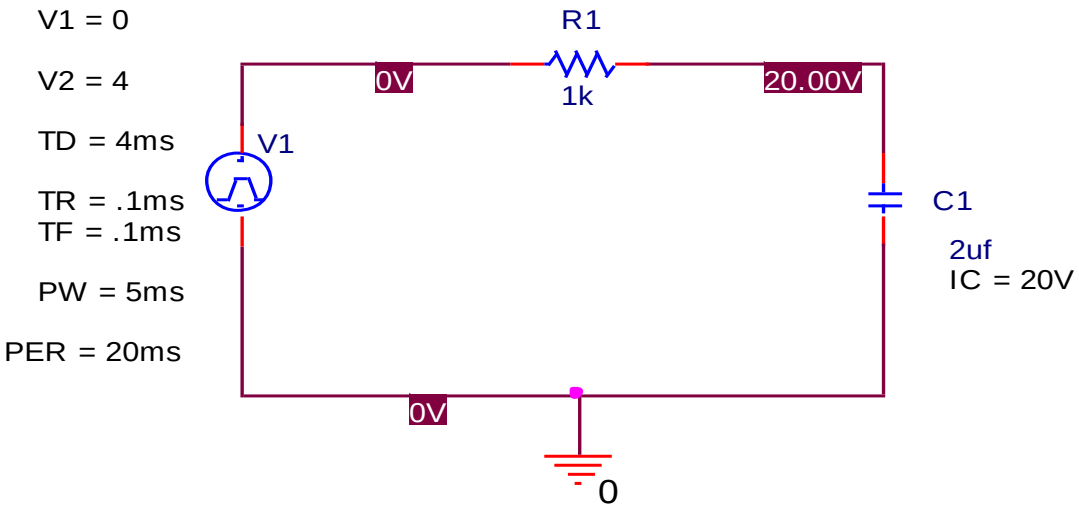


$i_C = C.\ dv_C/dt$

So, the current wave is the differentiation of the voltage wave.

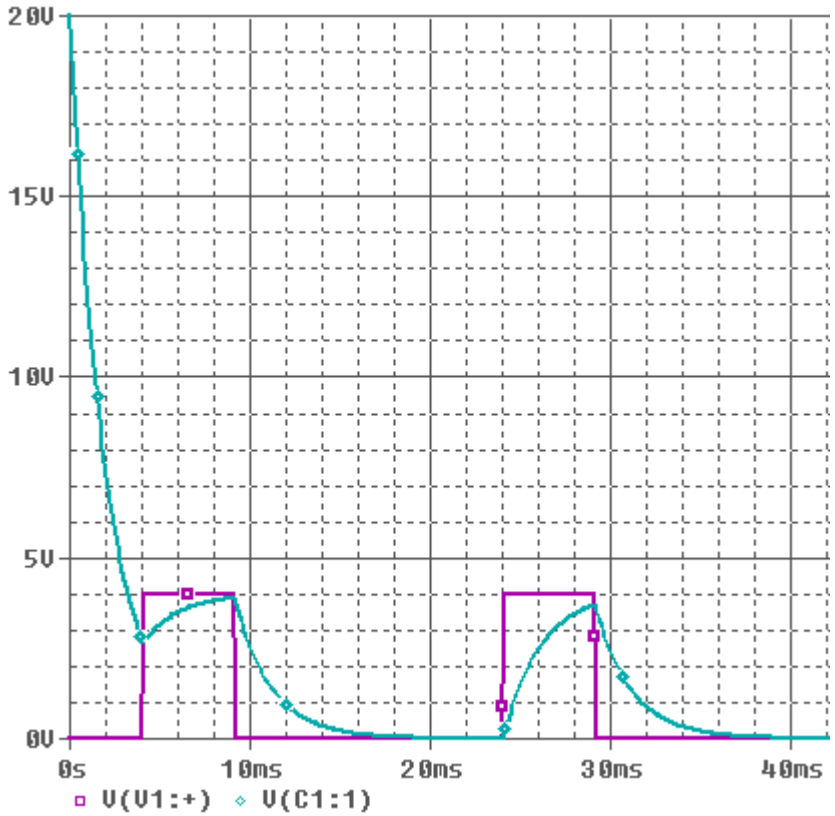


Capacitor Voltage with Initial condition

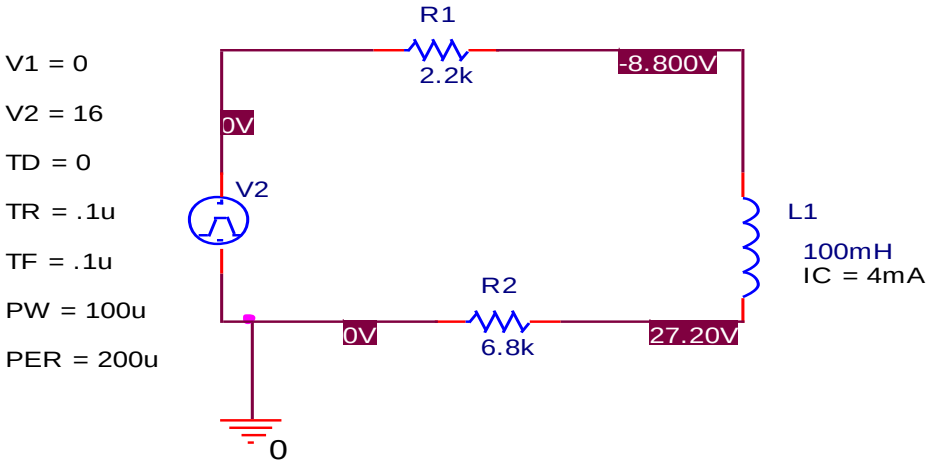


Procedure:

1. Construct the above circuit, set IC(Initial condition). Observe the input voltage wave form and voltage across the capacitor.



Inductor Current with Initial Condition

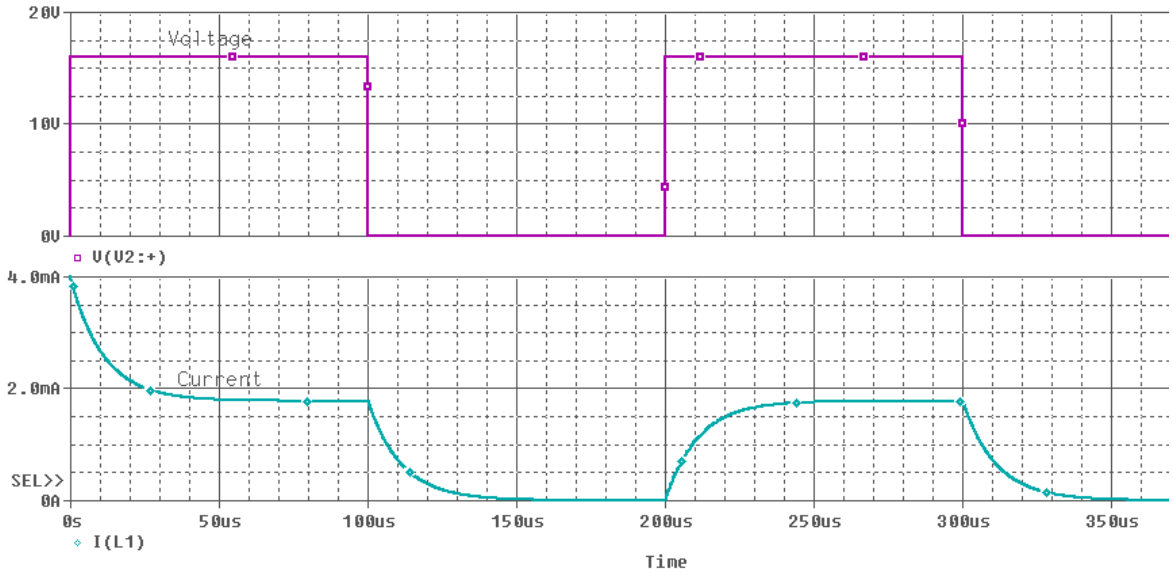


Procedure:

1. Draw the above circuit. And put the value as mentioned in the circuit diagram. To give the initial condition to the capacitor double click on the inductor to get the property editor tool bar. Then add new property write IC. Go to display and select name and value. Then minimize or cut the property editor. Now assign the initial condition as 4mA.

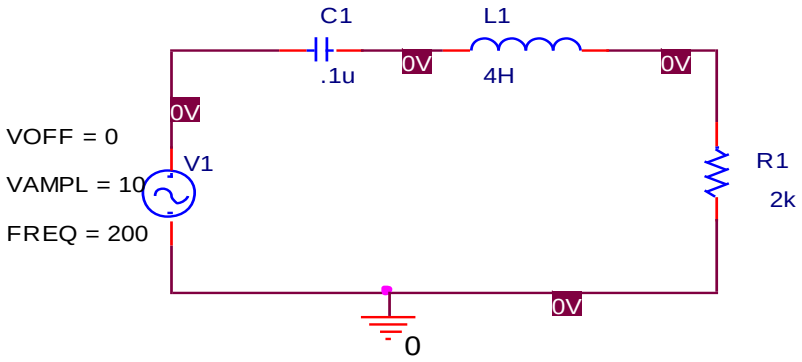
2. Set run to time as 400us and step size .1us.

3. Now run the project and from add trace select input voltage and current wave shape of the circuit.



Name of the Experiment.: Inductors and Capacitors in AC Circuit

Voltage current phase shift :



Frequency=200Hz

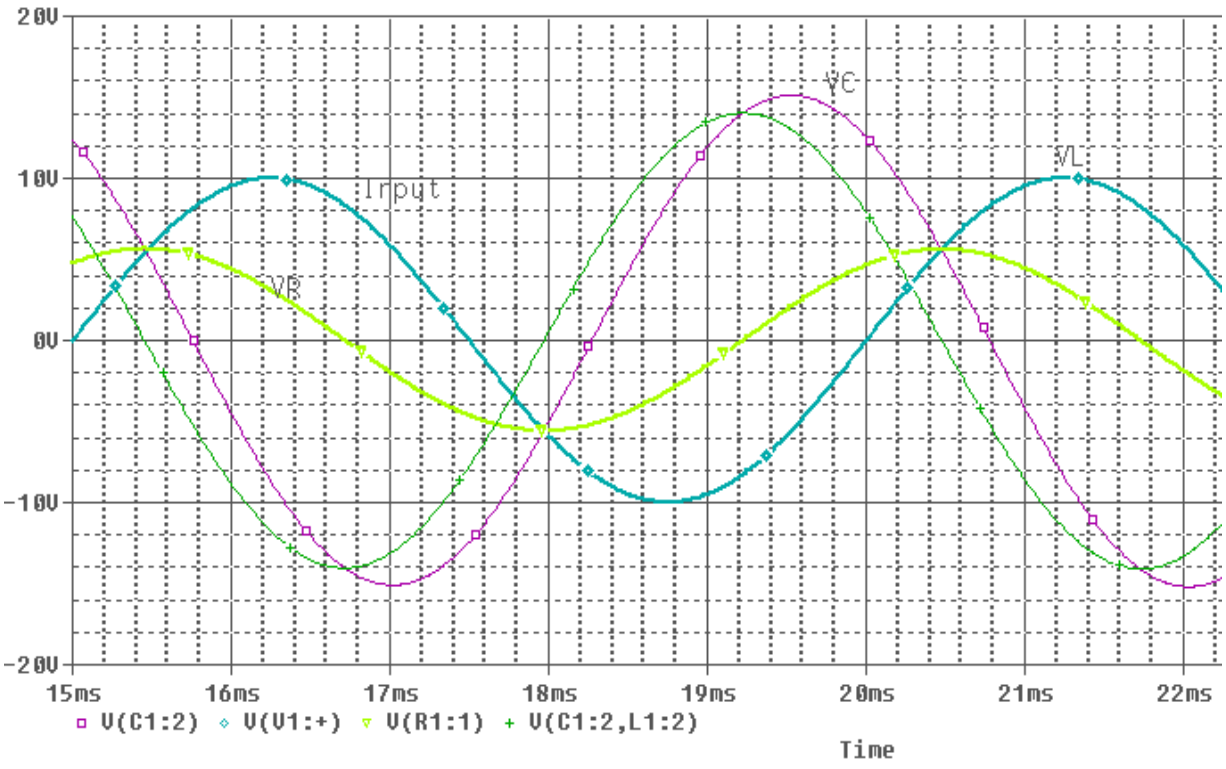
Time period= 1/f = 1/200 =.005=5ms

R1=2 K ohm

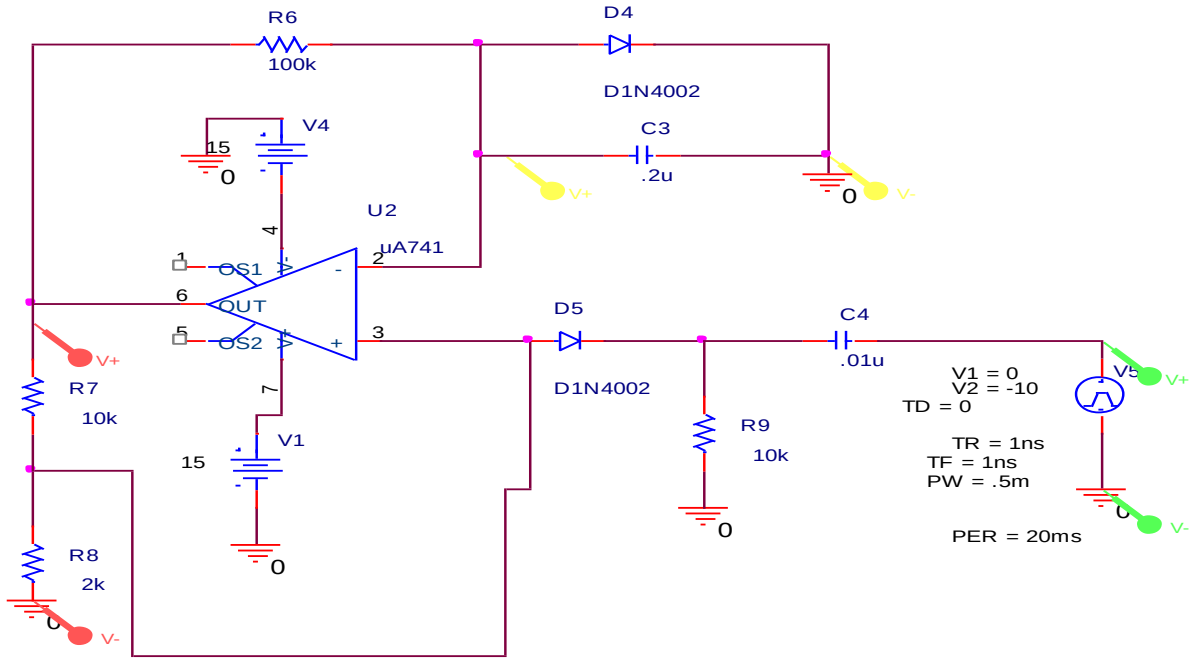
ω= 2pi*frequency= 2*3.14*200= 1256

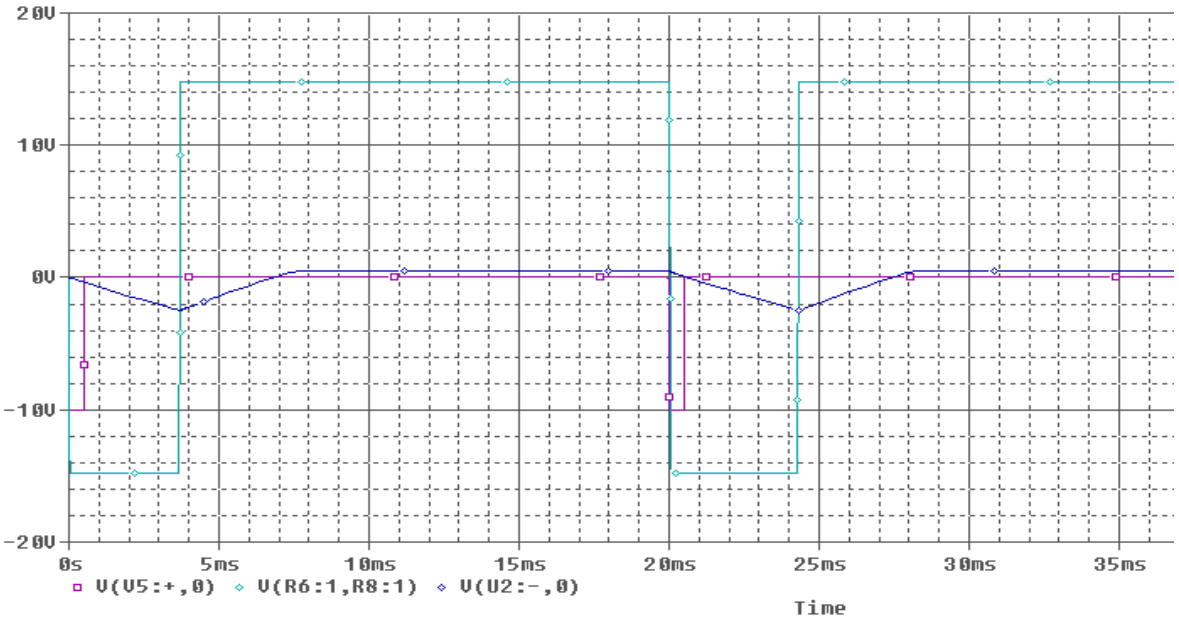
X_L= 1256*4= 5024 ohm

X_C=1/(1256*.1u)=10^6/125.6= 7961ohm



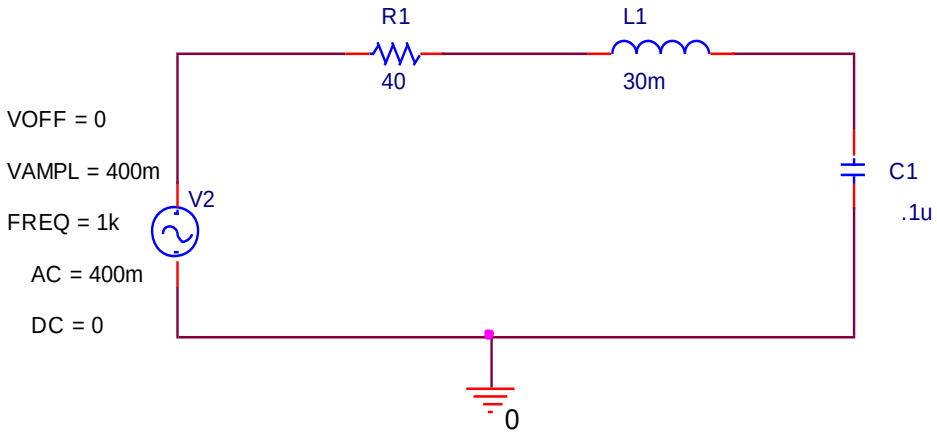
Name of the Experiment.: Free Running and One Shot Multivibrator





Name of the Experiment.: Frequency Response Curves

Series Resonance Circuits

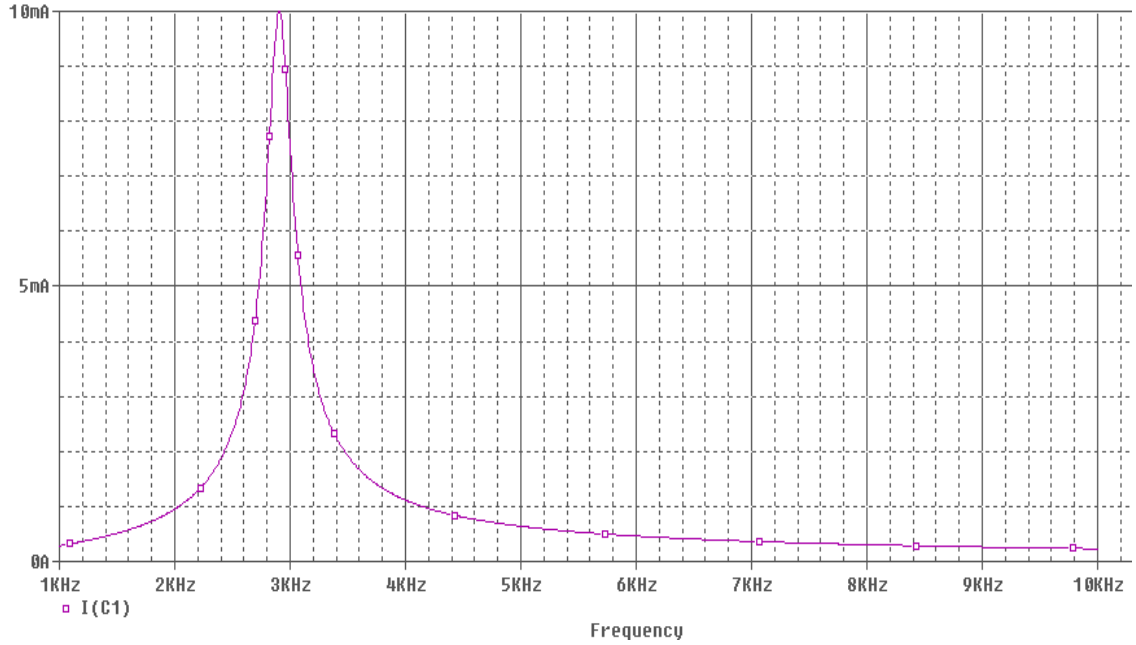


Procedure:

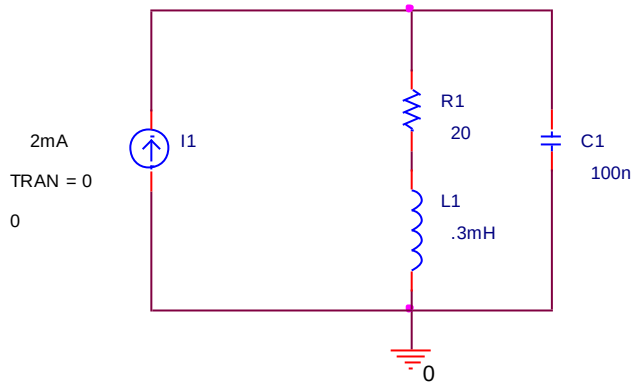
1. Draw the above \circuit and use VSIN as source. By double clicking on the source apply all the value from the property editor window.

2. At simulation setting go to AC sweep/ Noise. Set start frequency 1K. End frequency 10k and no of points 100. Select linear.

3. Now run the project and from add trace select I(C) and observe the following wave shape.

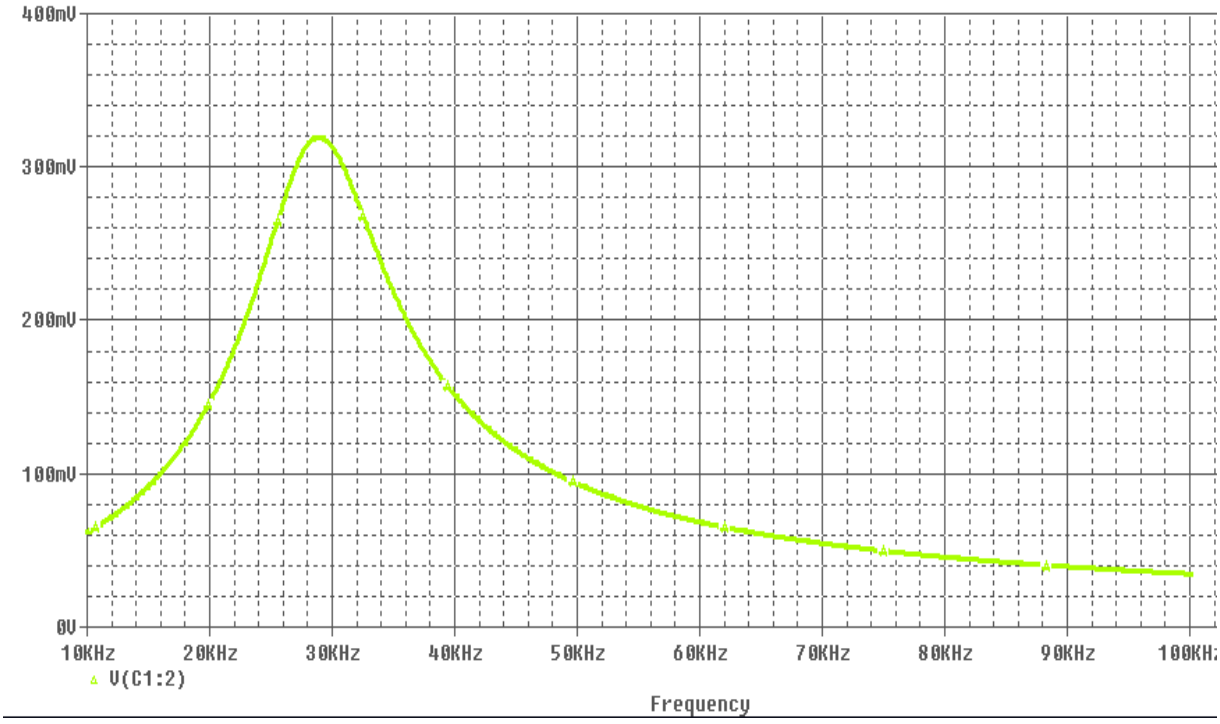


Parallel Resonance Circuit

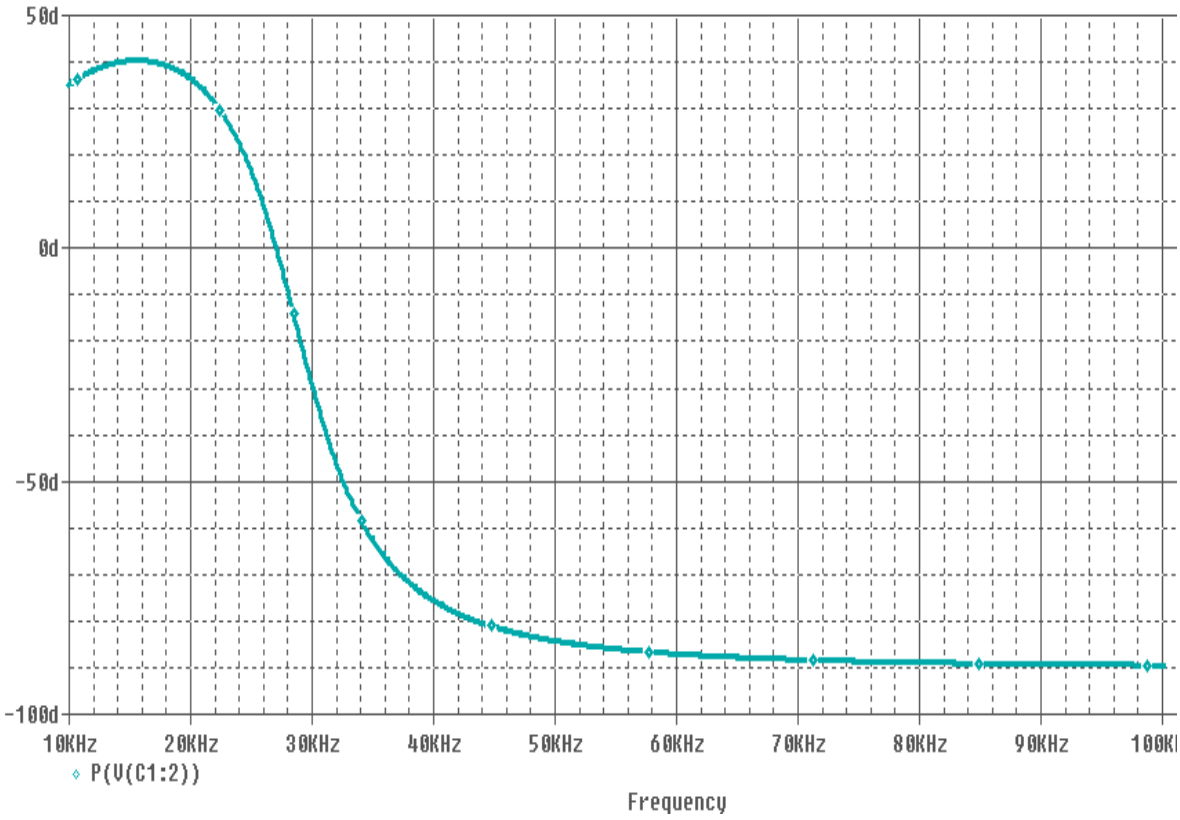


Procedure:

1. Construct the above circuit
2. Use ISRC as source.
3. At simulation setting set start frequency 10K, end frequency 100k, and no of points 1000.
4. Now run the project. Select output voltage across the capacitor and observe the wave shape.



5. Delete the previous trace and now write P(V(C1:2)) to get the following figure of phase angle at different frequencies.



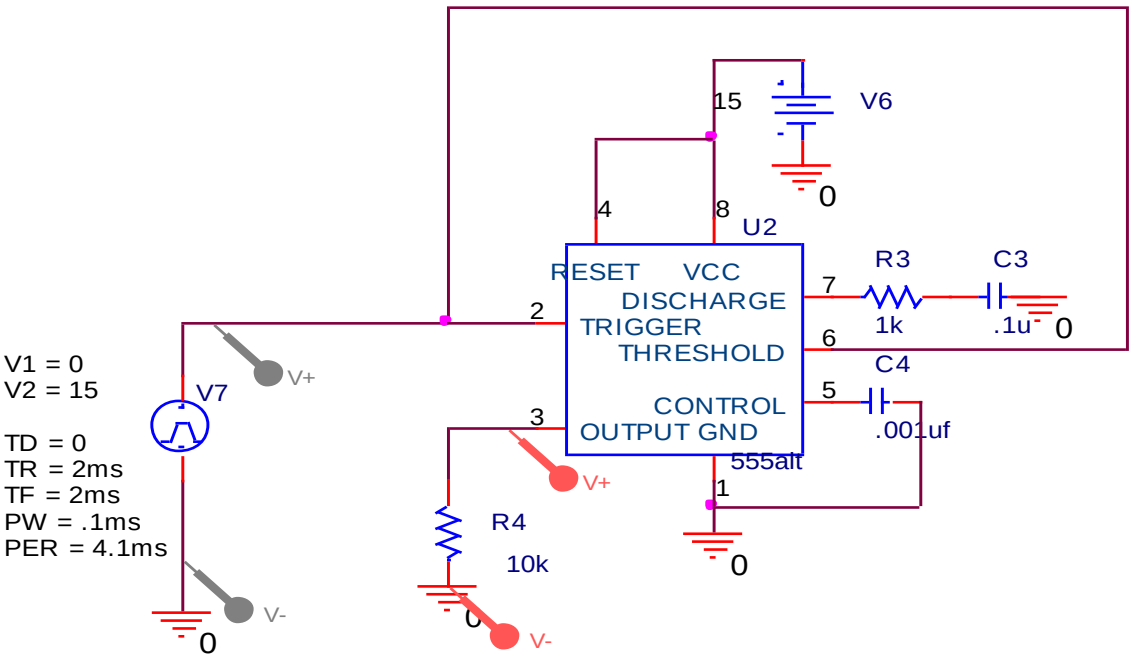
Name of the Experiment.: Use of 555 timer IC

Understanding the principle of operation:

Operating States: $V_{UT}=2V_{CC}/3$, $V_{LT}=V_{CC}/3$,

Operating States	Trigger Pin 2	Threshold Pin 6	State of terminals	
			Output 3	Discharge 7
A	Below V_{LT}	Below V_{UT}	High	Open
B	Below V_{LT}	Above V_{UT}	High	Open
C	Above V_{LT}	Below V_{UT}	Remember last state	
D	Above V_{LT}	Above V_{UT}	Low	Ground

Experimental Circuit:

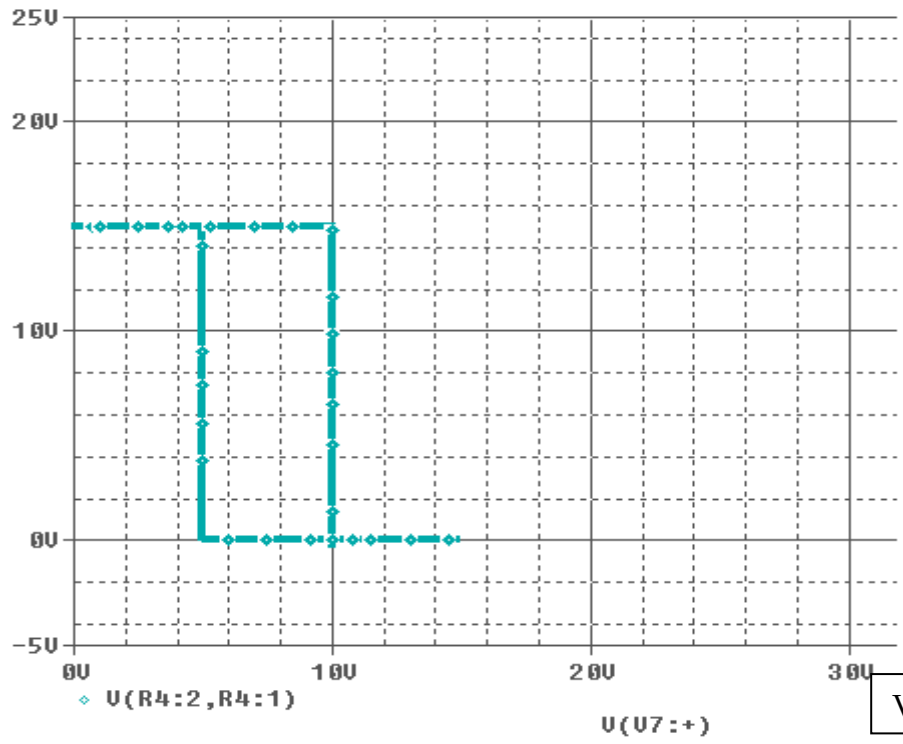


Description:

1. add the library ANL_MISC to get the timer IC. Here we use 555alt to get the required 555 timer.

2. Construct the above circuit and observe the input and output wave shpe with respect to time and input vs output curve. The output wave shapes are shown below.

Vout



Vinput

Fig. Input vs output curve

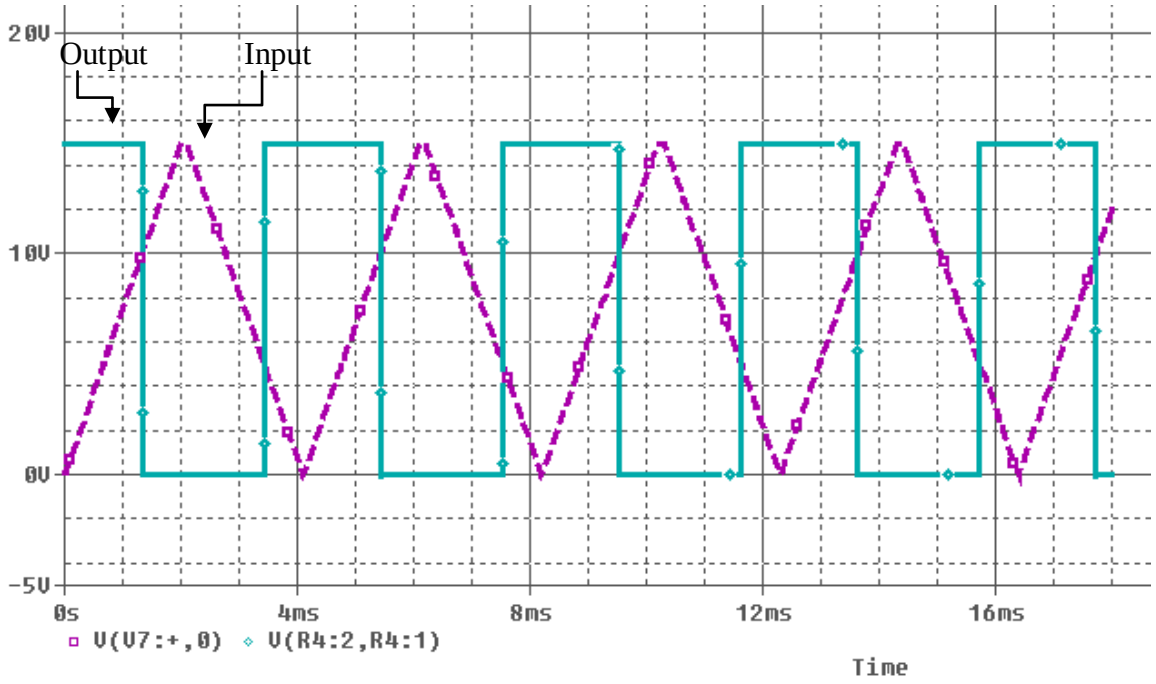
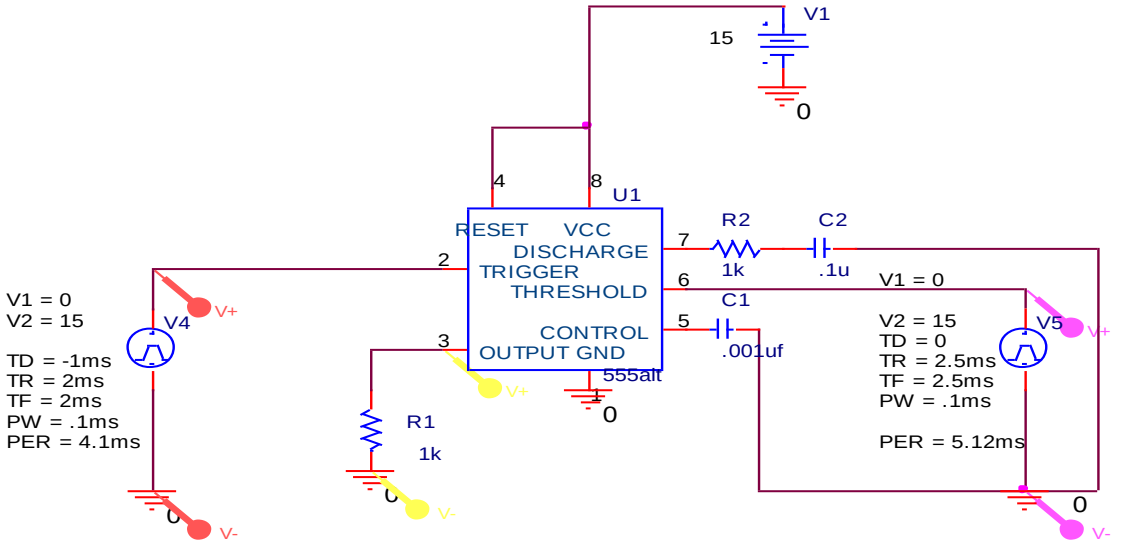


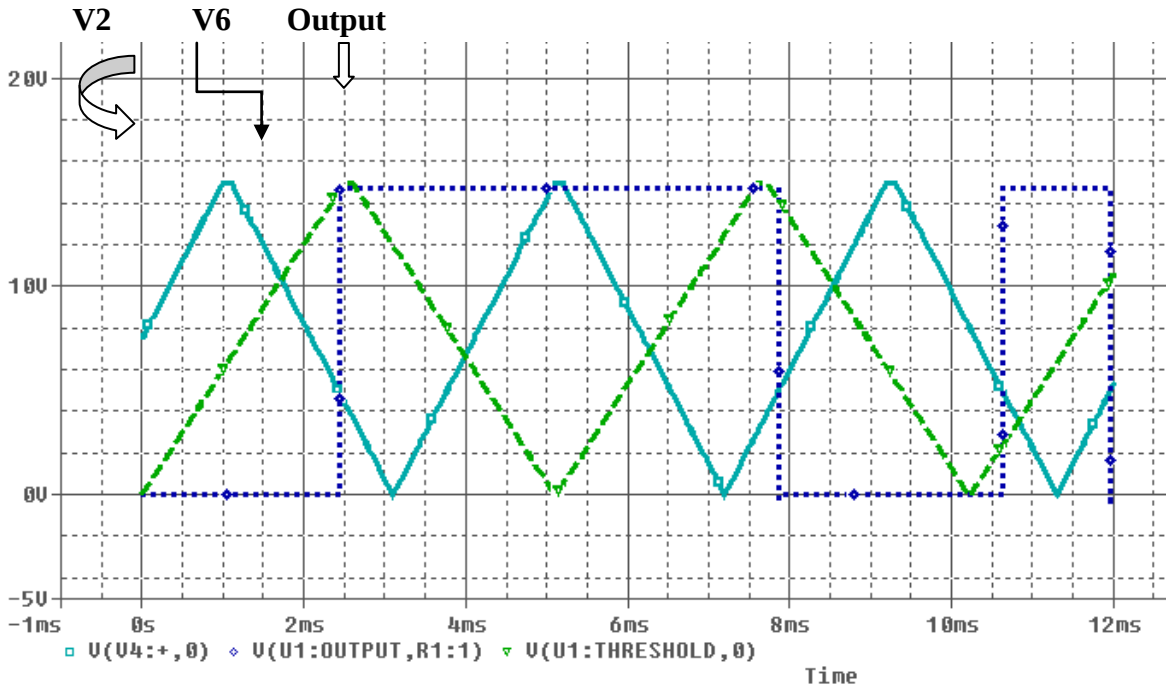
Fig. Input and output curve of the timer circuit

Circuit diagram:



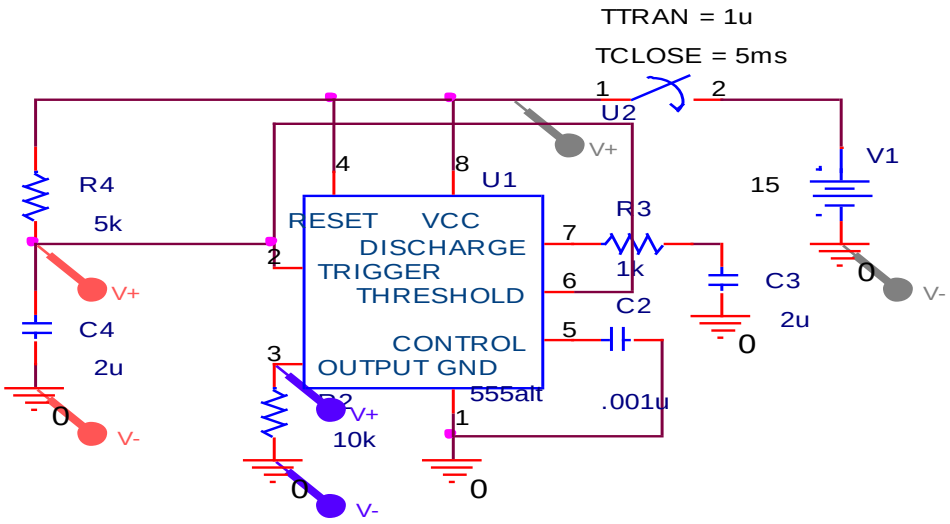
Procedure:

1. Construct the above circuit diagram and observe the following wave shapes.

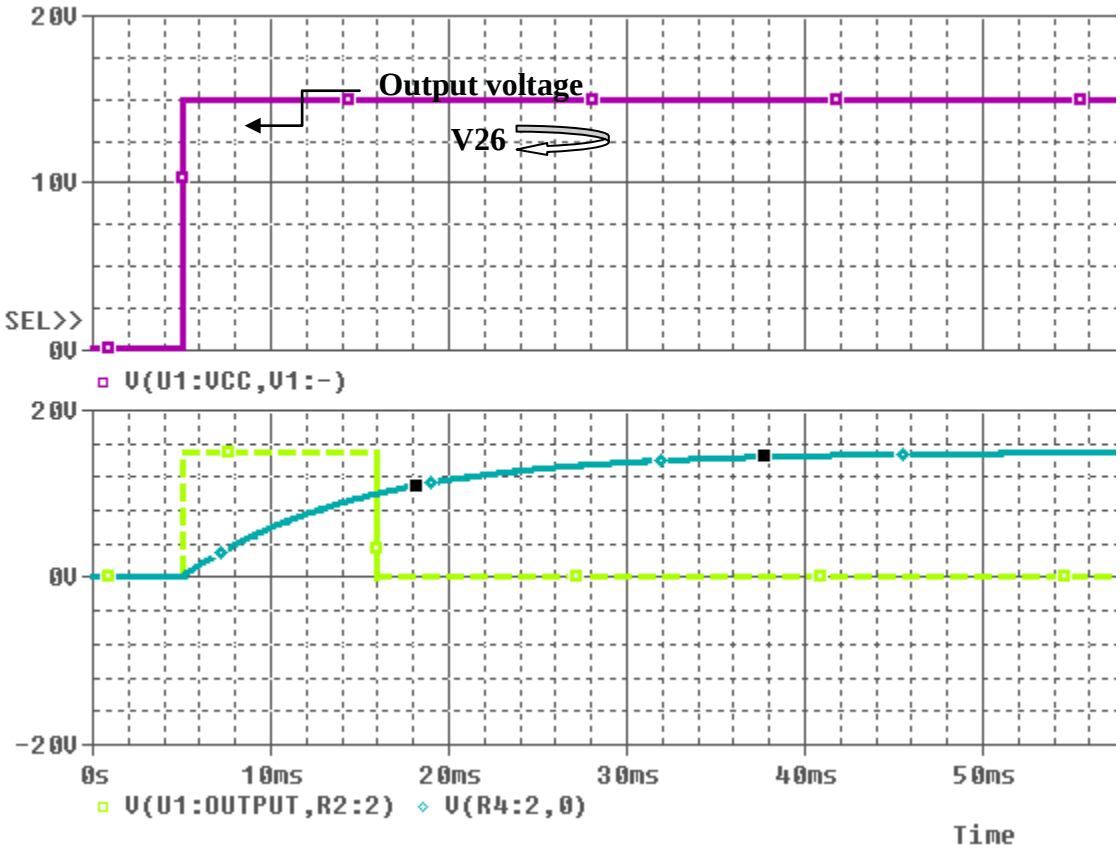


Power on time delay circuit:

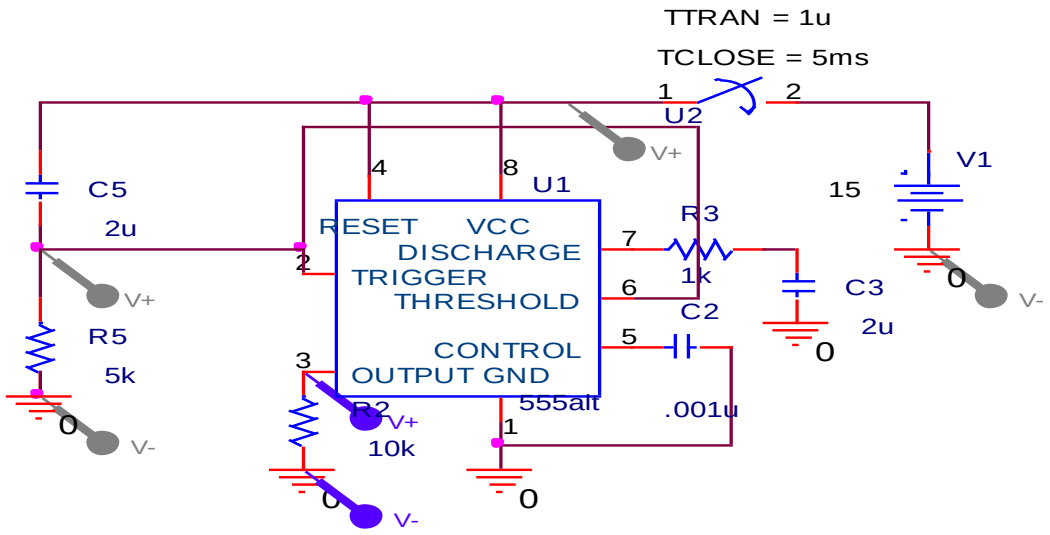
Circuit I:

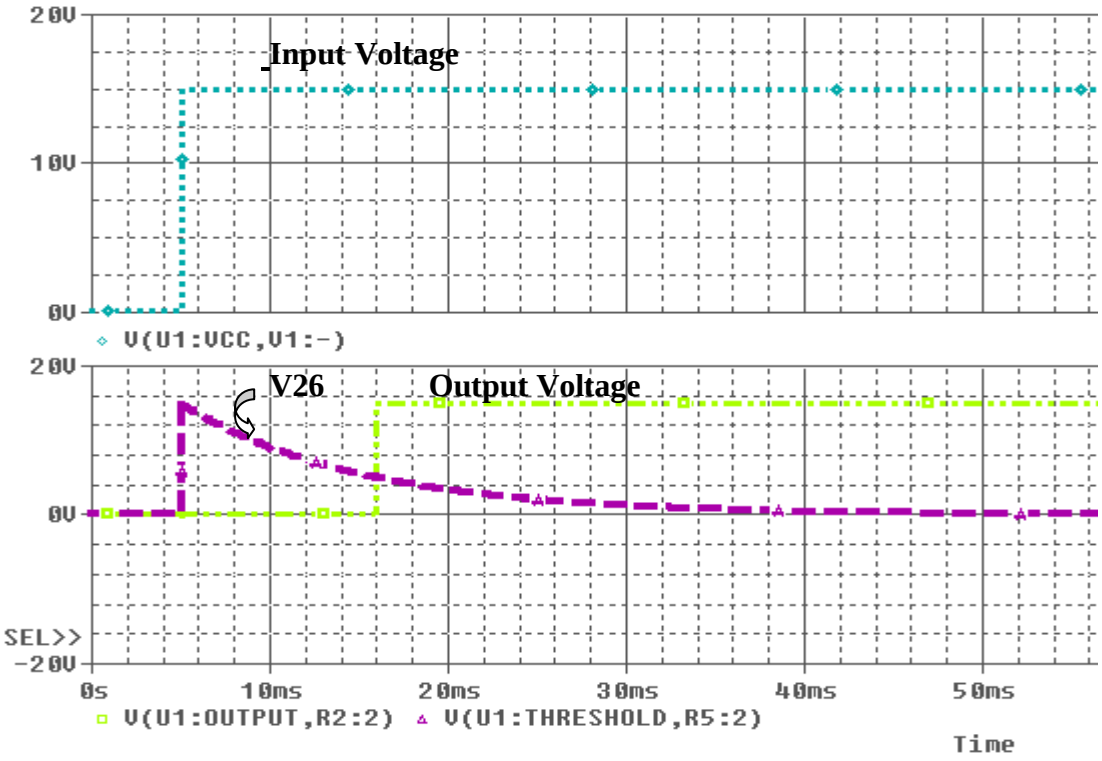


Input Voltage

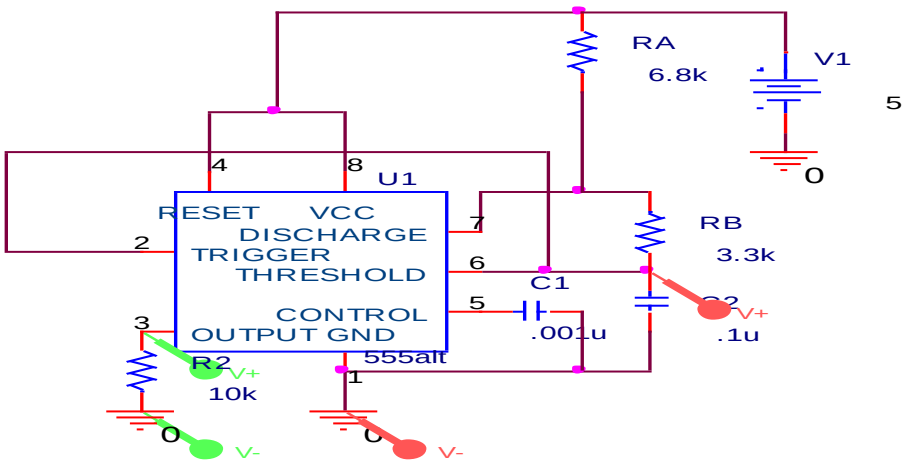


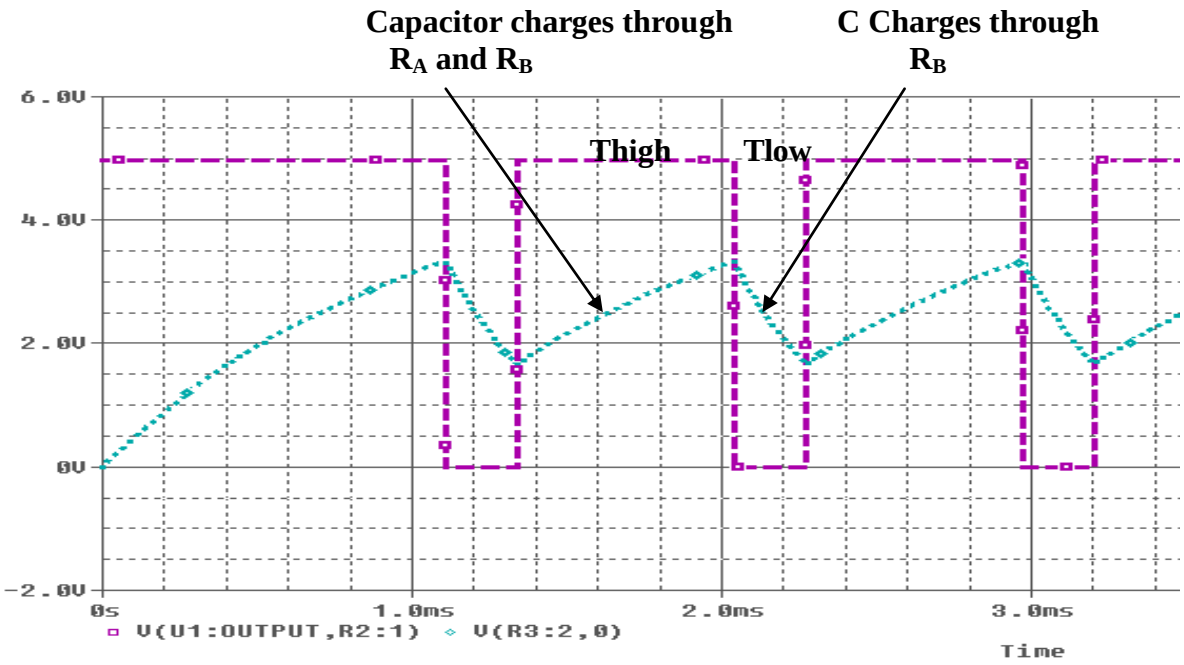
Circuit II:



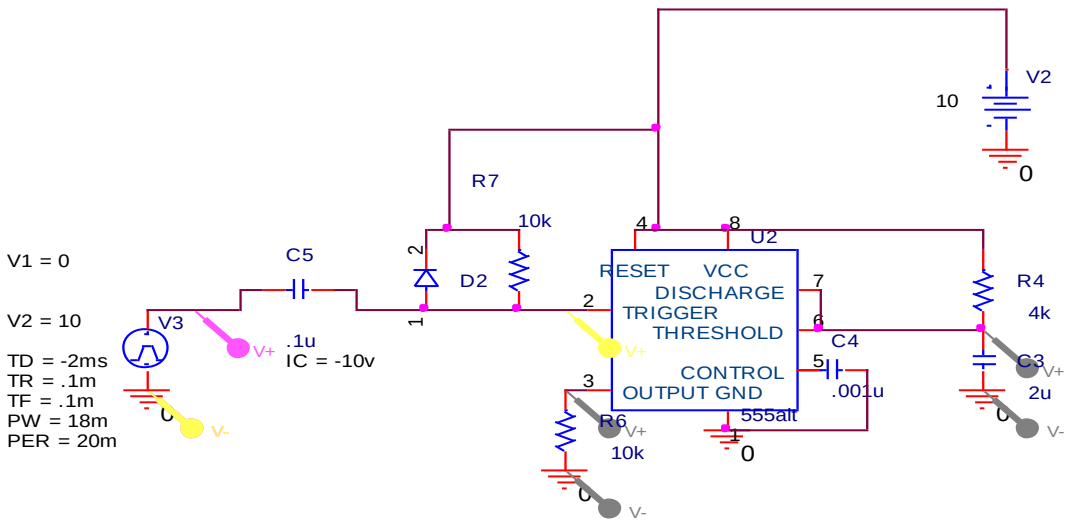


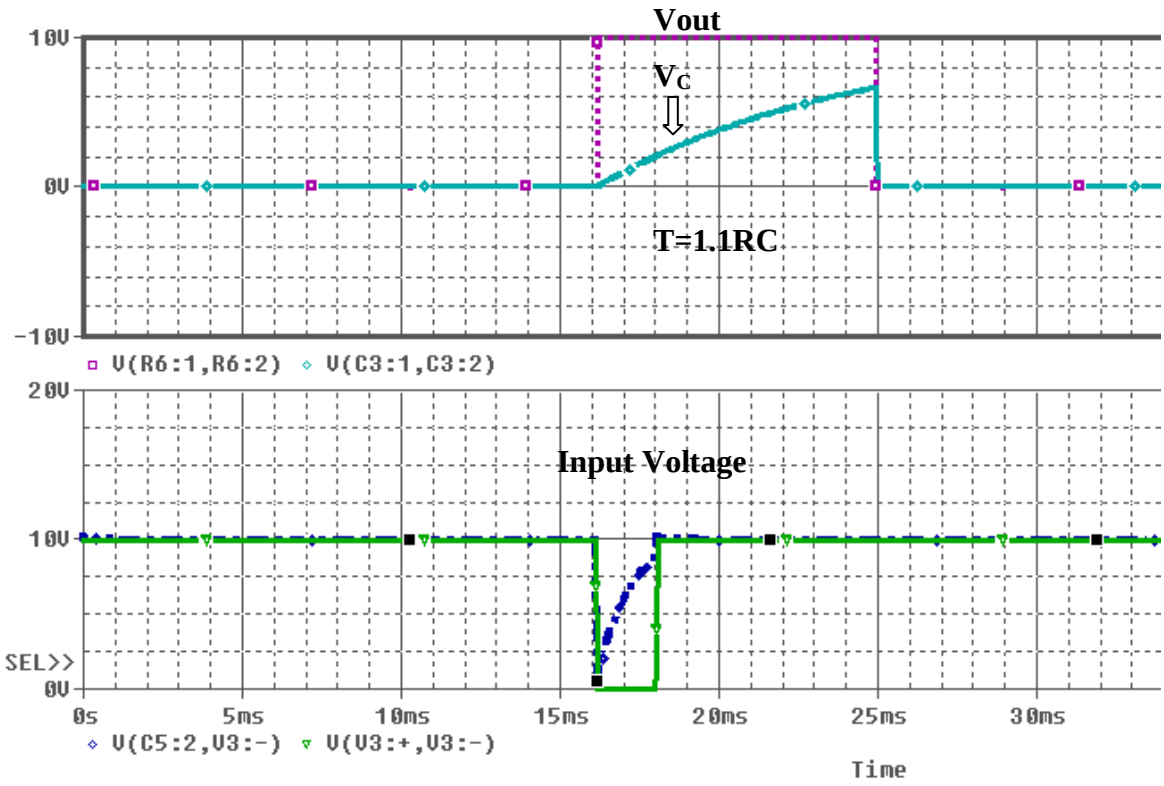
Name of the Experiment.: Free Running and One Shot operation Using 555 Timer





One Shot Operation:




Vout
10V
0V
-10V
V_C
T=1.1RC
V(R6:1,R6:2)
V(C3:1,C3:2)
20V
10V
SEL>>
0V
Input Voltage
0s
5ms
10ms
15ms
20ms
25ms
30ms
V(C5:2,U3:-)
V(U3:+,U3:-)
Time