Experiment No: 1

PSPICE SIMULATION OF SERIES RLC CIRCUITS FOR STEP, PULSE & SINUSOIDAL INPUTS

AIM: To study the responses of series RLC circuits for a given step, pulse & sinusoidal inputs.

SIMULATION TOOLS REQUIRED: PC with PSPICE Software.

SPECIFICATIONS:

Step input: $V_1 = 0$, $T_1 = 0$, $V_2 = 1V$, $T_2 = 1ns$, $V_3 = 1V$, $T_3 = 1ms$

Pulse input: $V_1 = -10V$, $V_2 = 10V$, $T_D = T_R = T_F = 1ns$, PW = 40us, PER = 80us.

Sinusoidal input: $V_{OFF} = 0V$, $V_{AMPL} = 169.7V$, FREQ = 50 Hz.

CIRCUIT DIAGRAMS:

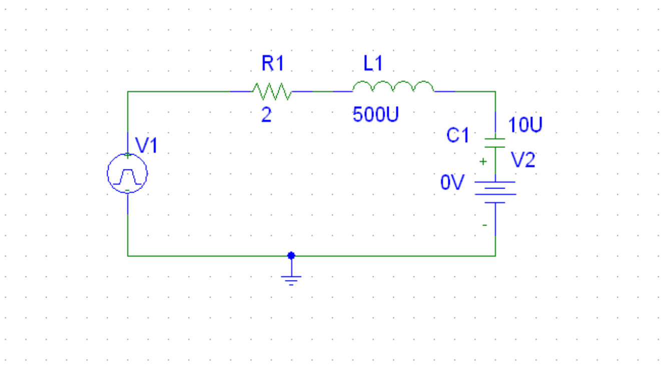


Fig. (1) Pulse input

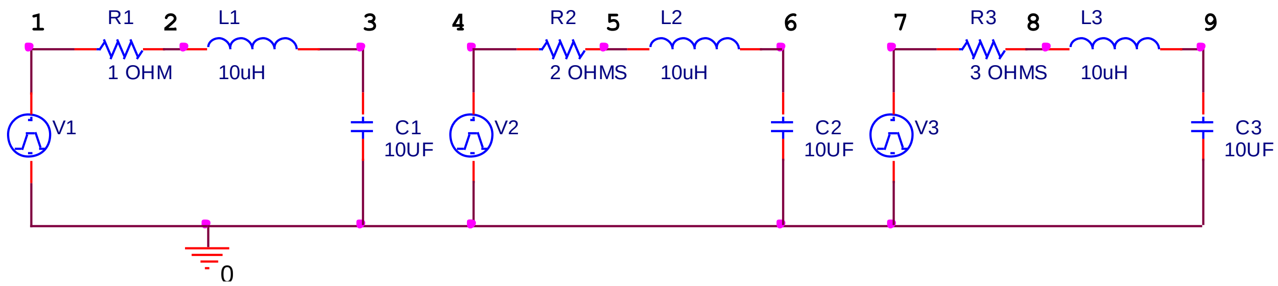


Fig. (2) Series RLC circuit for STEP input

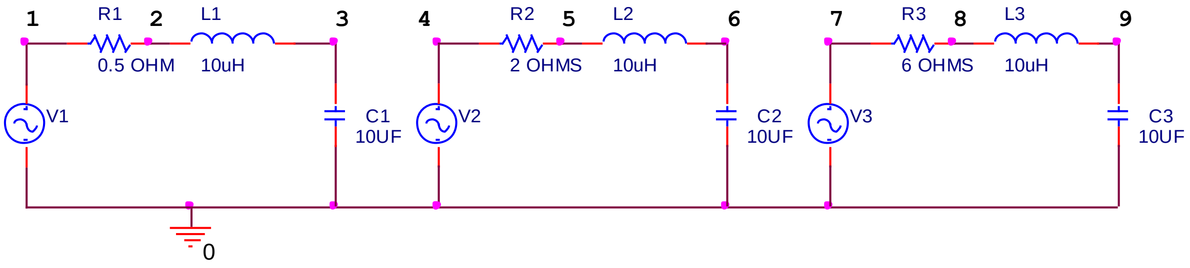


Fig. (3) Series RLC circuit for SINUSOIDAL input

PROCEDURE:

BY ANALYSIS PROGRAM:

1.Write the program in a new text file in PSpice AD.

2. Save the file using the notation filename.cir.

3. Activate the file by opening it.

4. Run the simulation process using blue button.

5. By clicking Add Trace icon, get the required waveform.

12. Enter the name where simulation results are required.

13. Select time domain analysis and set run time to a suitable value.

14. Simulate the file by selecting run from pspice menu

15. If the simulation is successful output wave form are displayed in the probe window

PROGRAMS:

STEP INPUT

```
SIMPLE RLC CIRCUIT FOR STEP INPUT

V1 1 0 PWL(0 0 1NS 1V 1MS 1V)

V2 4 0 PWL(0 0 1NS 1V 1MS 1V)

V3 7 0 PWL(0 0 1NS 1V 1MS 1V)
```

```
R1 1 2 1OHM

L1 2 3 50UH

C1 3 0 10UF

R2 4 5 2OHMS

L2 5 6 50UH

C2 6 0 10UF

R3 7 8 8OHMS

L3 8 9 50UH

C3 9 0 10UF

.TRAN 1US 800US

.PROBE

.END
```

RESULTANTWAVE FORM:

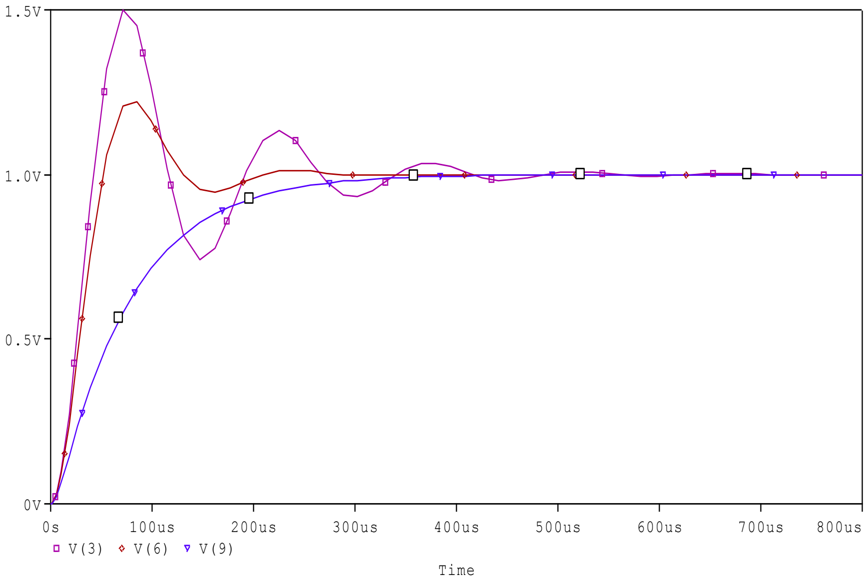


Fig: (1)SERIES RLC CIRCUIT WITH STEP INPUT

PULSE INPUT

```
SIMPLE RLC CIRCUIT FOR STEP INPUT

V1 1 0 PULSE(-10 10 1NS 1NS 1NS 40US 80US)

V2 4 0 PULSE(-10 10 1NS 1NS 1NS 40US 80US)

V3 7 0 PULSE(-10 10 1NS 1NS 1NS 40US 80US)

R1 1 2 1OHM

L1 2 3 10UH

C1 3 0 10UF

R2 4 5 2OHMS

L2 5 6 10UH

C2 6 0 10UF

R3 7 8 3OHMS

L3 8 9 10UH

C3 9 0 10UF

.TRAN 1US 240US

.PROBE

.END
```

RESULTANTWAVE FORM:

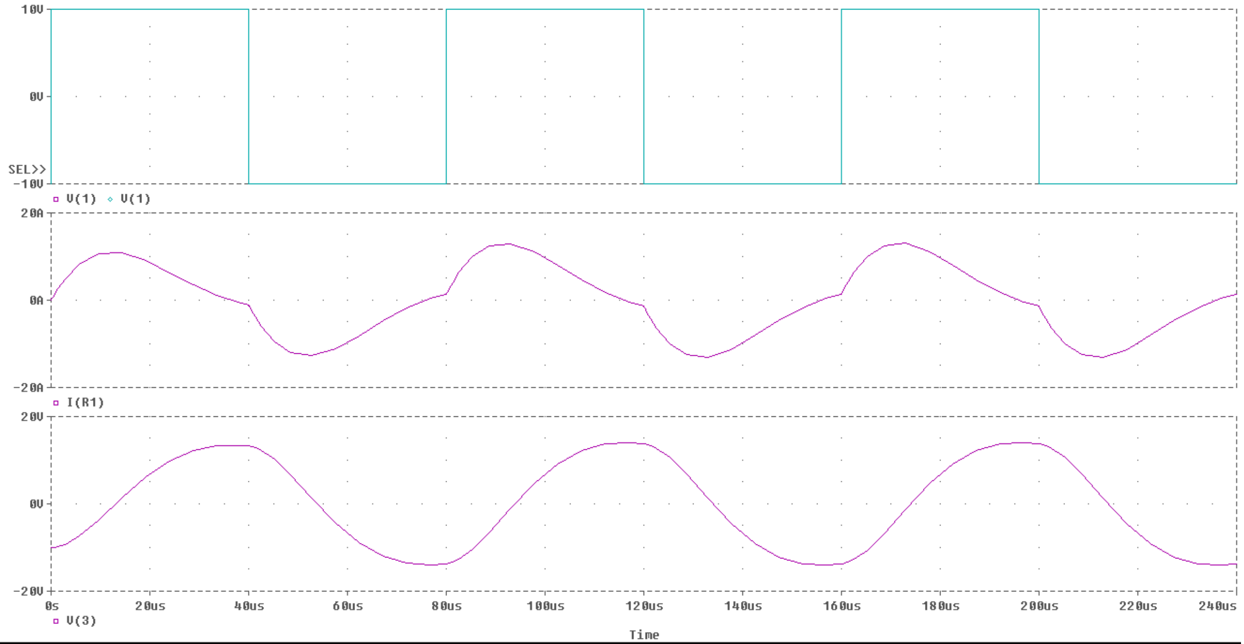


Fig:(2)Simple RLC circuit for step input

SINUSOIDAL INPUT

SIMPLE RLC CIRCUIT FOR SINUSOIDAL INPUT

V1 1 0 SIN(0 169.7V 50)

V2 4 0 SIN(0 169.7V 50)

V3 7 0 SIN(0 169.7V 50)

R1 1 2 0.5OHM

L1 2 3 10UH

C1 3 0 10UF

R2 4 5 2OHMS

L2 5 6 10UH

C2 6 0 10UF

R3 7 8 6OHMS

L3 8 9 10UH

C3 9 0 10UF

.TRAN 1US 60MS

.PROBE

.END

RESULTANT WAVE FORM:

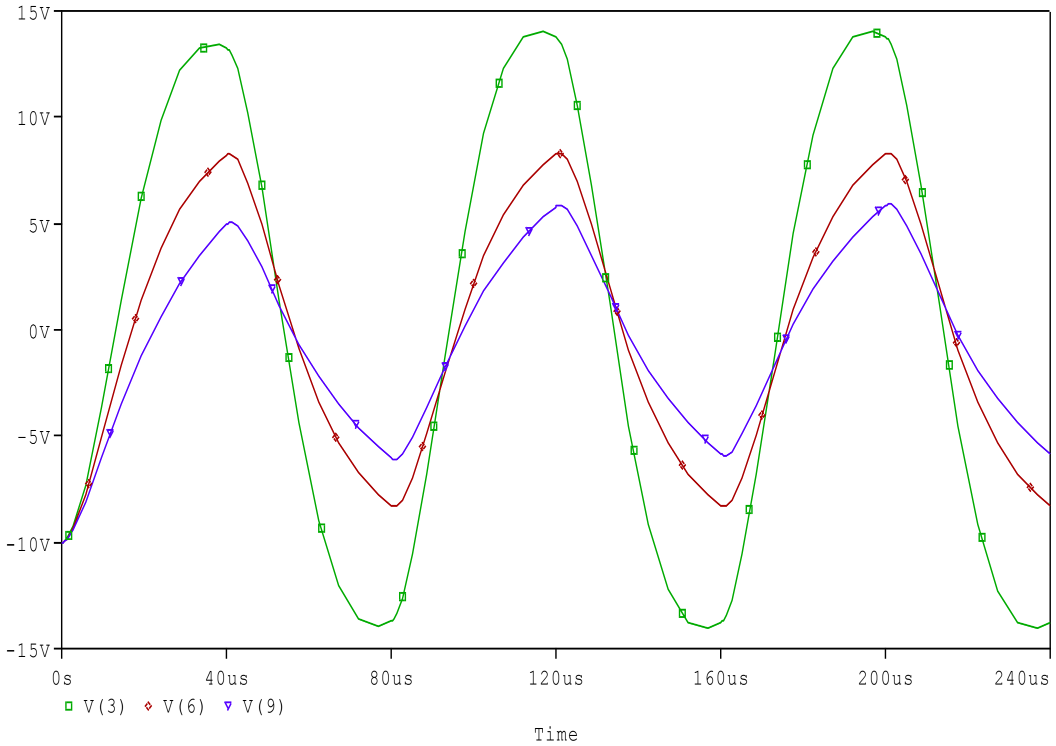


Fig: (3) SERIES RLC CIRCUIT WITH SINUSOIDAL INPUT

RESULT:

Experiment No: 2

PSPICE ANALYSIS OF THREE PHASE CIRCUIT

Date:

AIM: To study the analysis of simple three phase circuit for balanced and unbalanced loads.

SIMULATION TOOLS REQUIRED: PC with PSPICE Software

SPECIFICATIONS: Sinusoidal input: V_{OFF} = 0V, V_{AMPL} = 169.7V, FREQ = 50 Hz.

CIRCUIT DIAGRAMS:

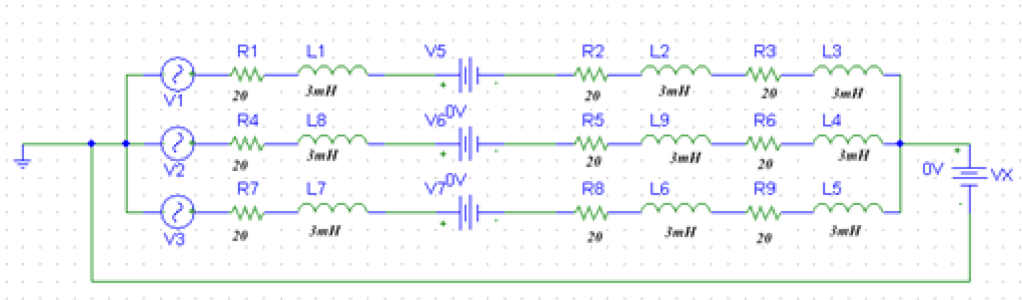


Fig: (1)Three Phase circuit with balanced load

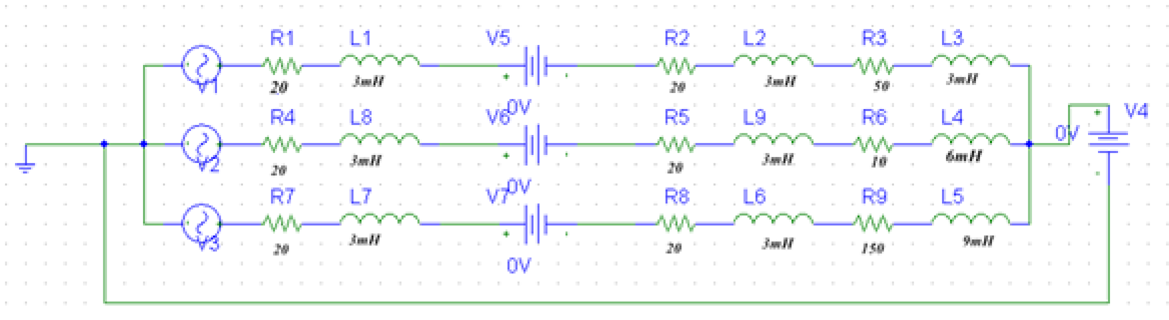


Fig : (2) Three Phase circuit with unbalanced load

PROCEDURE:

FOR PROGRAM:

1. Write the program in a new text file in PSpice AD.
2. Save the file using the notation filename.cir.
3. Activate the file by opening it.
4. Run the simulation process using blue button.
5. By clicking Add Trace icon, get the required waveform.

PROGRAMS:

BALANCED LOAD CONDITION

SIMPLE 3 PHASE CIRCUIT FOR BALANCED LOAD

```
VS1 1 0 sin(0 169.7 50)
VS2 2 0 sin(0 169.7 50 0 0 120)
VS3 3 0 sin(0 169.7 50 0 0 240)
R1 1 4 20
L1 4 7 3mH
R2 7 10 20
L2 10 13 3mH
R3 13 16 20
L3 16 19 3mH
R4 2 5 20
L4 5 8 3mH
R5 8 11 20
L5 11 14 3mH
R6 14 17 20
L6 17 19 3mH
R7 3 6 20
L7 6 9 3mH
R8 9 12 20
L8 12 15 3mH
```

R9 15 18 20

L9 18 19 3mH

VX 19 0 dc 0

.Tran 1us 40ms

.Probe

.End

RESULTANT WAVEFORMS:

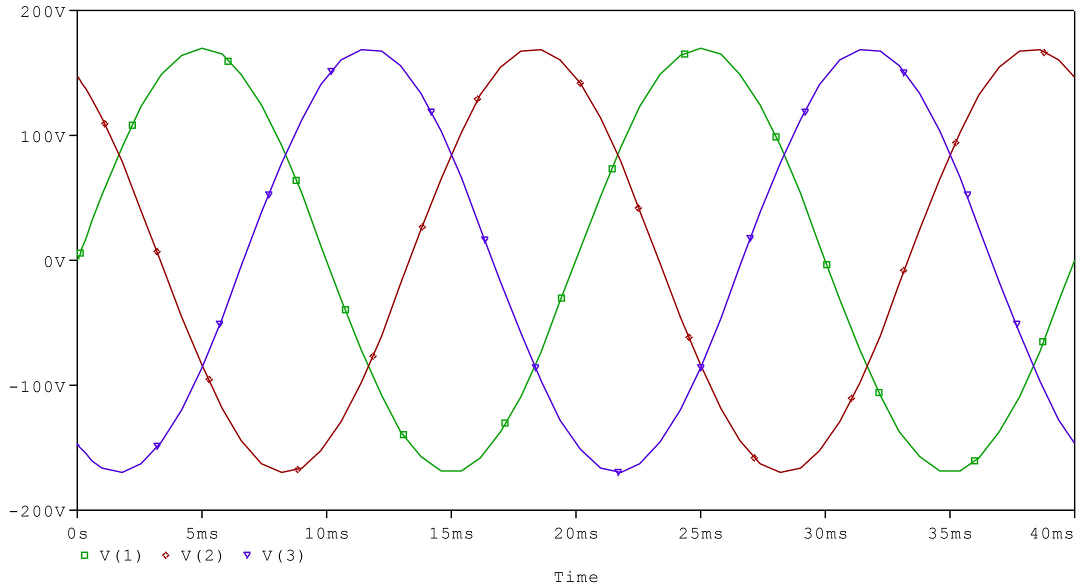


Fig:(1)INPUT WAVEFORM

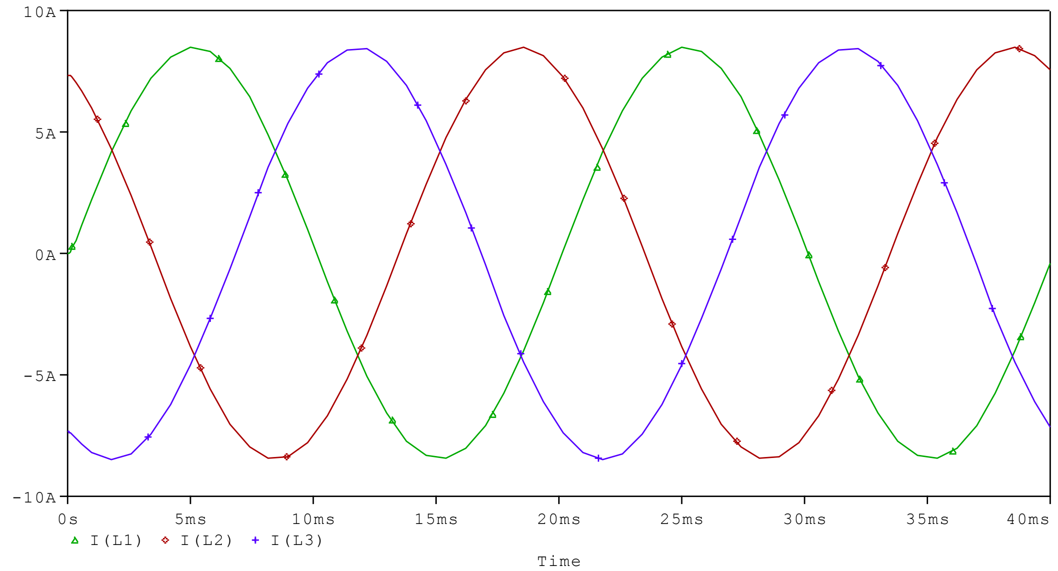


Fig:(2)BALANCED LOAD CONDITION

UNBALANCED LOAD CONDITION

SIMPLE 3 PHASE CIRCUIT FOR UNBALANCED LOAD

```
VS1 1 0 sin(0 169.7 50)
VS2 2 0 sin(0 169.7 50 0 0 120)
VS3 3 0 sin(0 169.7 50 0 0 240)
R1 1 4 20
L1 4 7 3mH
R2 7 10 20
L2 10 13 3mH
R3 13 16 20
L3 16 19 3mH
R4 2 5 20
L4 5 8 3mH
R5 8 11 20
L5 11 14 3mH
R6 14 17 20
L6 17 19 3mH
R7 3 6 50
L7 6 9 3mH
R8 9 12 10
L8 12 15 6mH
R9 15 18 150
L9 18 19 9mH
VX 19 0 dc 0
.Tran 1us 40ms
.Probe
.End
```

RESULTANT WAVEFORMS:

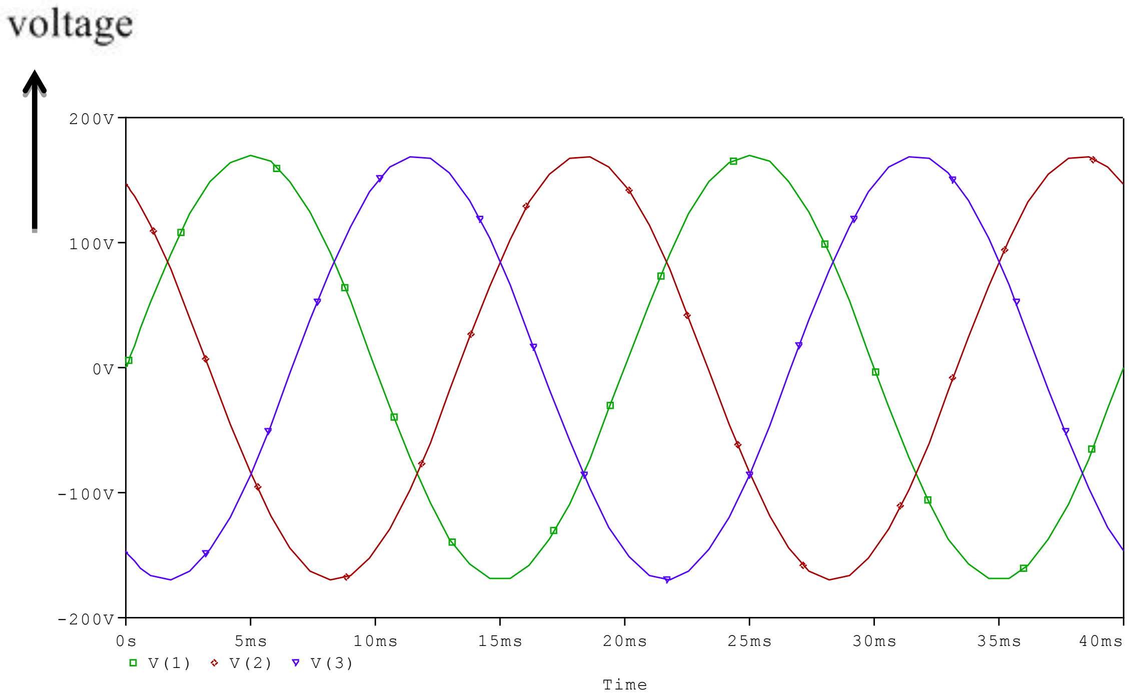


Fig:(1)INPUT WAVEFORM

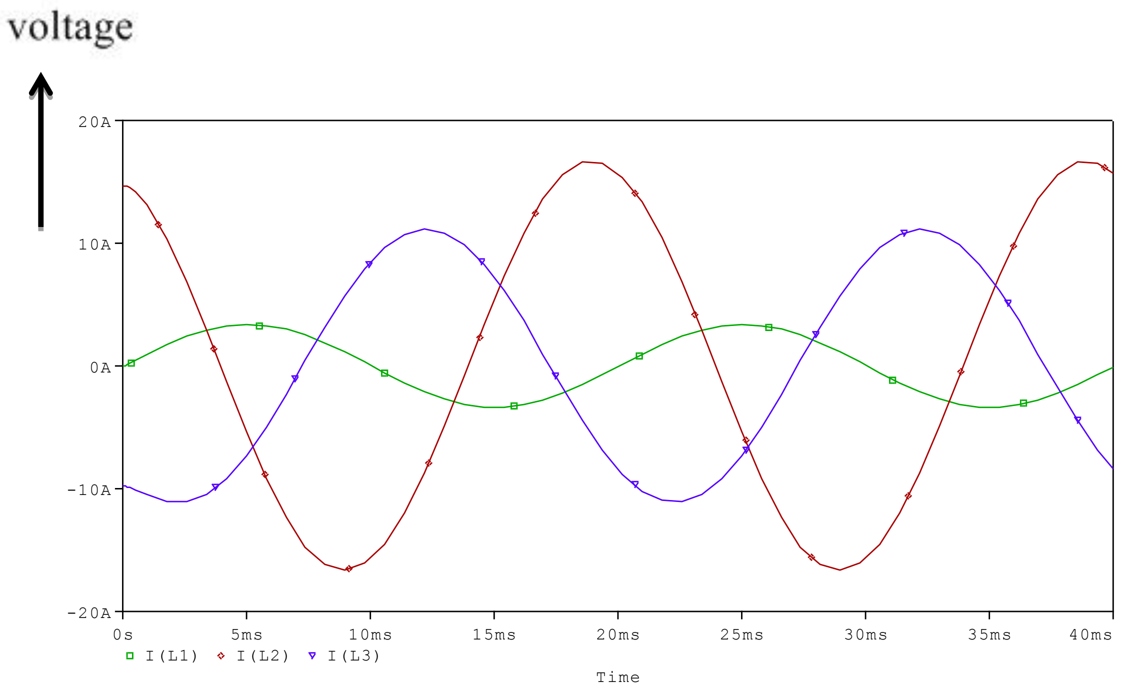


Fig:(3)UNBALANCED LOAD CONDITION

RESULT:

Experiment No: 3.a

PSPICE ANALYSIS OF SINGLE PHASE FULL CONVERTER WITH RLE LOADS

Date:

AIM: To analyze the single phase full converter with RL and RLE Loads.

SIMULATION TOOLS REQUIRED: PC with PSPICE Software.

SPECIFICATIONS:

Sinusoidal input: V_{OFF} = 0V, V_{AMPL} = 169.7V, FREQ = 50 Hz.T_1 and T_2: V_1 = 0V, V_2 = 100V,

T_D = 3333.34us, T_R = T_F = 1ns, PW = 100us, PER = 20ms. T_3 and T_4: V_1 = 0V, V_2 = 100V,

T_D = 13333.34us, T_R = T_F = 1ns, PW = 100us, PER = 20ms. Firing circuit: R_G = 50 Ω, V_X, V_Y = 0V, R_T = 1 Ω, C_T = 10uf, R_{ON} = 0.0125, R_{OFF} = 10E+5, V_{ON} = 0.5V, V_{OFF} = 0V, I_S = 2.2E-15, BV = 1800V,

TT = 0 sec.

CIRCUIT DIAGRAMS:

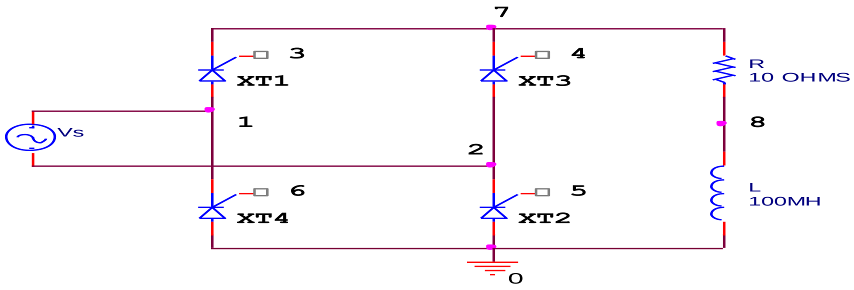


Fig . (1) Single Phase full converter with RLE load

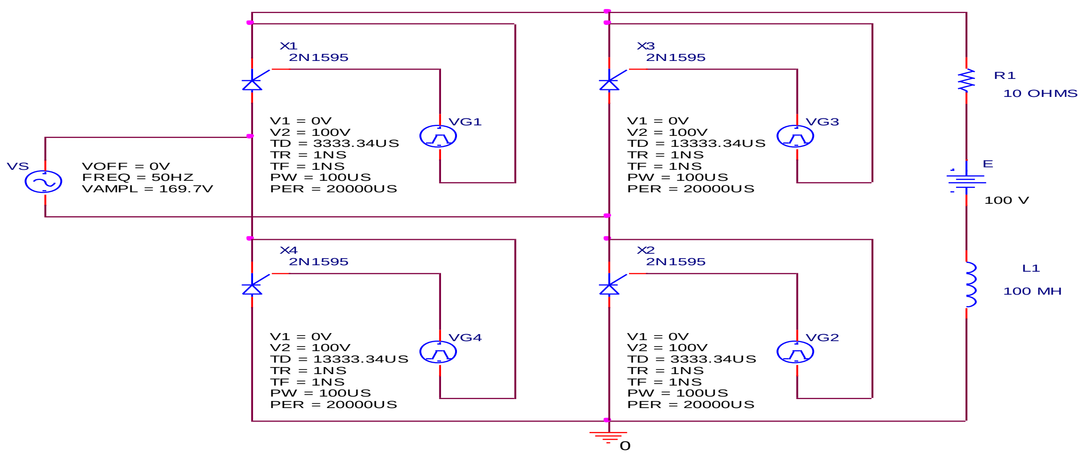


Fig: (2) Single Phase full converter with RL load

PROCEDURE:

FOR ANALYSIS USING PROGRAM:

1 Write the program in a new text file in PSpice AD.
2. Save the file using the notation filename.cir.
3. Activate the file by opening it.
4. Run the simulation process using blue button.
5. By clicking Add Trace icon, get the required waveform.

PROGRAMS:SIGLE-PHASE FULL CONVERTER CIRCUIT WITH RL LOAD

```
VS1 1 2 SIN(0 169.7V 50HZ)

R1 7 8 10OHM

L1 8 0 100MH

VG1 3 7 PULSE(0 100V 3333.34US 1NS 1NS 100US 20000US)

VG3 4 7 PULSE(0 100V 13333.34US 1NS 1NS 100US 20000US)

VG2 5 2 PULSE(0 100V 3333.34US 1NS 1NS 100US 20000US)

VG4 6 1 PULSE(0 100V 13333.34US 1NS 1NS 100US 20000US)

XT1 1 7 3 7 SCR

XT2 0 2 5 2 SCR

XT3 2 7 4 7 SCR

XT4 0 1 6 1 SCR

.SUBCKT SCR 1 2 3 2

S1 1 5 6 2 SMOD

RG 3 4 50OHMS

VX 4 2 DC 0V

VY 5 7 DC 0V

DT 7 2 DMOD

RT 6 2 1OHM

CT 6 2 10UF

F1 2 6 POLY(2) VX VY 0 50 11

.MODEL SMOD VSWITCH(RON=0.0125 ROFF=10E+5 VON=0.5V VOFF=0V)
```

```
.MODEL DMOD D(IS=2.2E-15 BV=1800 TT=0)

.ENDS SCR

.TRAN 1US 60MS

.PROBE

.END
```

RESULTANT WAVE FORM:

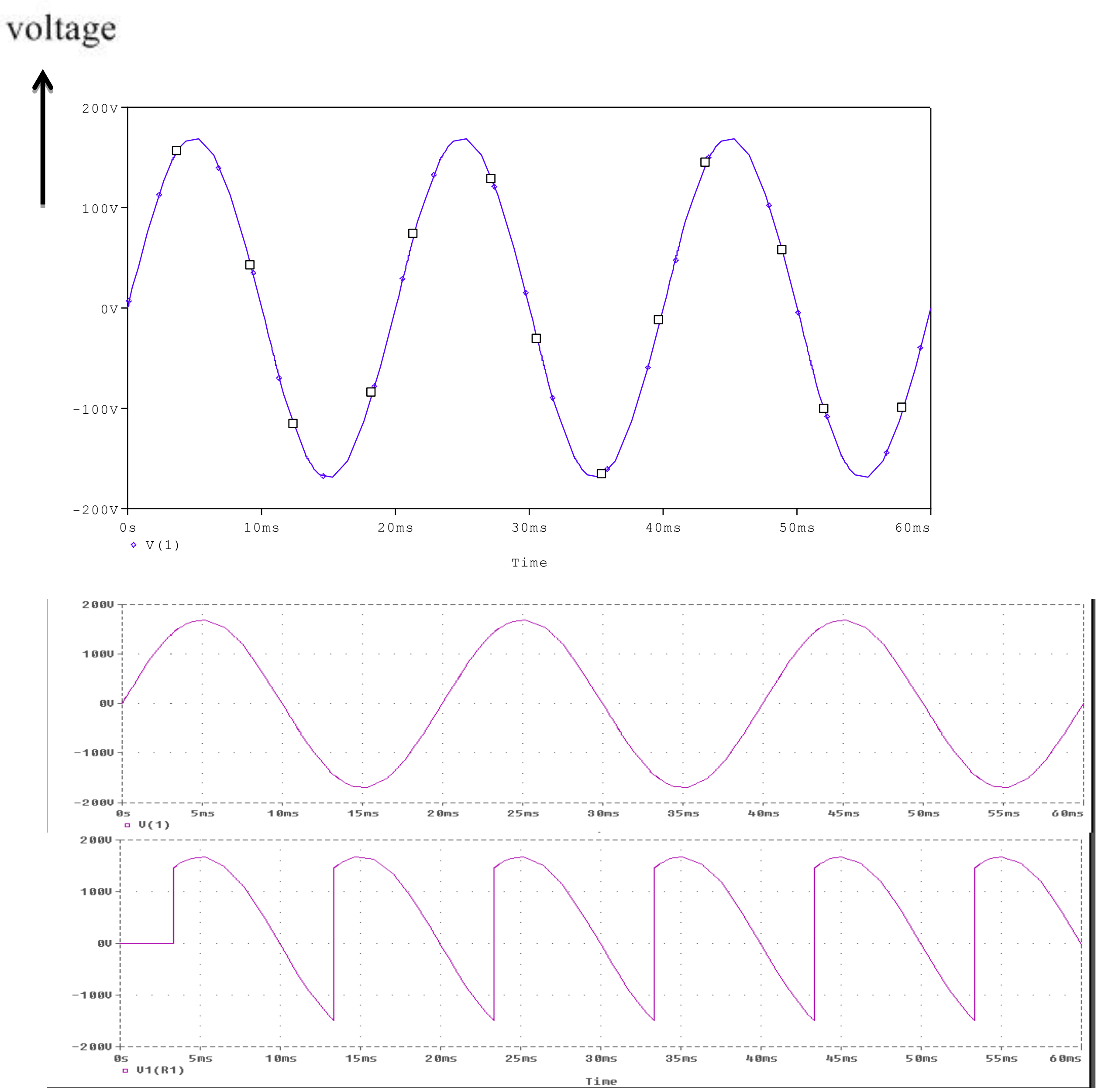


FIG: SIGLE-PHASE FULL CONVERTER CIRCUIT WITH RL LOAD

WITH RLE LOAD

```
SINGLE-PHASE FULL CONVERTER CIRCUIT WITH RLE LOAD
VS1 1 2 SIN(0 169.7V 50HZ)
R1 7 8 10OHM
L1 8 9 100MH
VDC 9 0 DC 100V
VG1 3 7 PULSE(0 100V 3333.34US 1NS 1NS 100US 20000US)
VG3 4 7 PULSE(0 100V 13333.34US 1NS 1NS 100US 20000US)
VG2 5 2 PULSE(0 100V 3333.34US 1NS 1NS 100US 20000US)
VG4 6 1 PULSE(0 100V 13333.34US 1NS 1NS 100US 20000US)
XT1 1 7 3 7 SCR
XT2 0 2 5 2 SCR
XT3 2 7 4 7 SCR
XT4 0 1 6 1 SCR
.SUBCKT SCR 1 2 3 2
S1 1 5 6 2 SMOD
RG 3 4 50OHMS
VX 4 2 DC 0V
VY 5 7 DC 0V
DT 7 2 DMOD
RT 6 2 1OHM
CT 6 2 10UF
F1 2 6 POLY(2) VX VY 0 50 11
.MODEL SMOD VSWITCH(RON=0.0125 ROFF=10E+5 VON=0.5V
VOFF=0V)
.MODEL DMOD D(IS=2.2E-15 BV=1800 TT=0)
.ENDS SCR
.TRAN 1US 60MS
.PROBE
.END
```

RESULTANT WAVE FORMS FOR FULL CONVERTER:

voltage

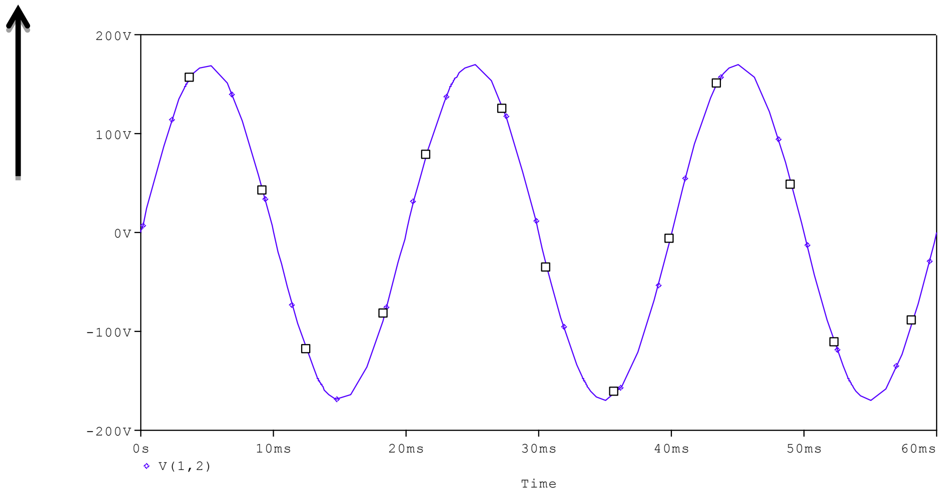


Fig: INPUT WAVEFORM

voltage

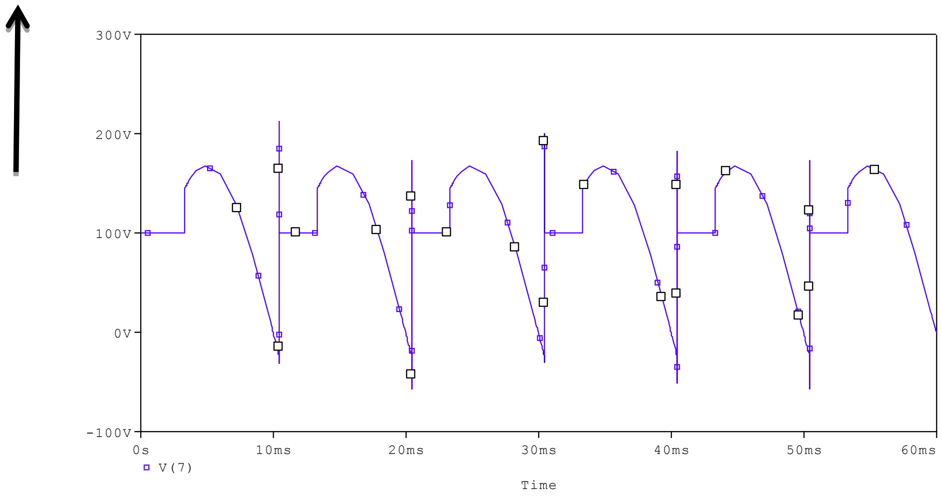


Fig:OUTPUT WAVEFORM WITH RLE LOAD

RESULT:

Experiment No: 3.b

PSPICE ANALYSIS OF SINGLE PHASE AC VOLTAGE CONTROLLER WITH RLLOAD

AIM: To analyze the single phase full converter with RL and RLE Loads.

SIMULATION TOOLS REQUIRED: PC with PSPICE Software

SPECIFICATIONS:

Sinusoidal input: V_{OFF} = 0V, V_{AMPL} = 169.7V, FREQ = 50 Hz.

T_1: V_1 = 0V, V_2 = 100V, T_D = 3333.34us, T_R = T_F = 1ns, PW = 100us, PER = 20ms.

T_2: V_1 = 0V, V_2 = 100V, T_D = 13333.34us, T_R = T_F = 1ns, PW = 100us, PER = 20ms.

Firing circuit: R_G = 50 Ω, V_X, V_Y = 0V, R_T = 1 Ω, C_T = 10uf, R_{ON} = 0.0125, R_{OFF} = 10E+5, V_{ON} = 0.5V, V_{OFF} = 0V, I_S = 2.2E-15, BV = 1800V, TT = 0 sec.

CIRCUIT DIAGRAM:

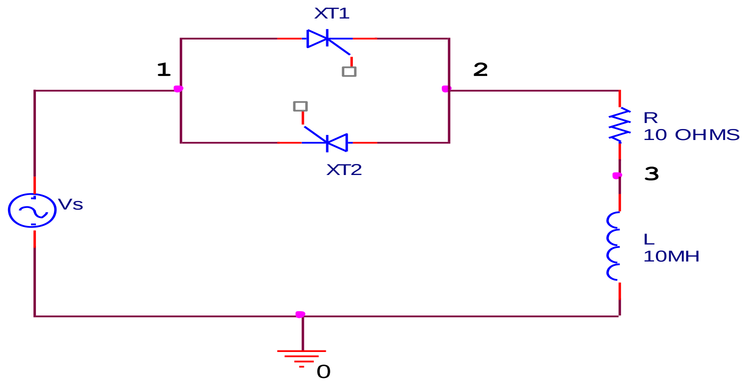


Fig: (1) Single Phase AC VOLTAGE CONTROLLER with RL load

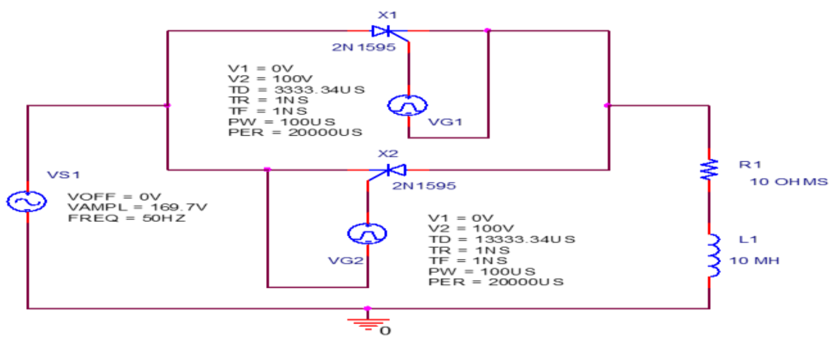


Fig: (2)CIRCUIT DIAGRAM FOR ANALYSIS USING CIRCUIT

PROCEDURE:

FOR ANALYSIS USING PROGRAM

1.Write the program in a new text file in PSpice AD.
2. Save the file using the notation filename.cir.
3.Activate the file by opening it.
4.Run the simulation process using blue button.
5.By clicking Add Trace icon, get the required waveform.

PROGRAM:

```
SINGLE-PHASE AC VOLTAGE CONTROLLER CIRCUIT WITH RL LOAD
VS1 1 0 SIN(0 169.7V 50HZ)
R1 2 3 10OHM
L1 3 0 10MH
VG1 4 2 PULSE(0 100V 3333.34US 1NS 1NS 100US 20000US)
VG2 5 1 PULSE(0 100V 13333.34US 1NS 1NS 100US 20000US)
XT1 1 2 4 2 SCR
XT2 2 1 5 1 SCR
.SUBCKT SCR 1 2 3 2
S1 1 5 6 2 SMOD
RG 3 4 50OHMS
VX 4 2 DC 0V
VY 5 7 DC 0V
DT 7 2 DMOD
RT 6 2 1OHM
CT 6 2 10UF
F1 2 6 POLY(2) VX VY 0 50 11
.MODEL SMOD VSWITCH(RON=0.0125 ROFF=10E+5 VON=0.5V
VOFF=0V)
.MODEL DMOD D(IS=2.2E-15 BV=1800 TT=0)
.ENDS SCR
.TRAN 1US 60MS
.PROBE
.END
```

RESULTANT WAVEFORMS:

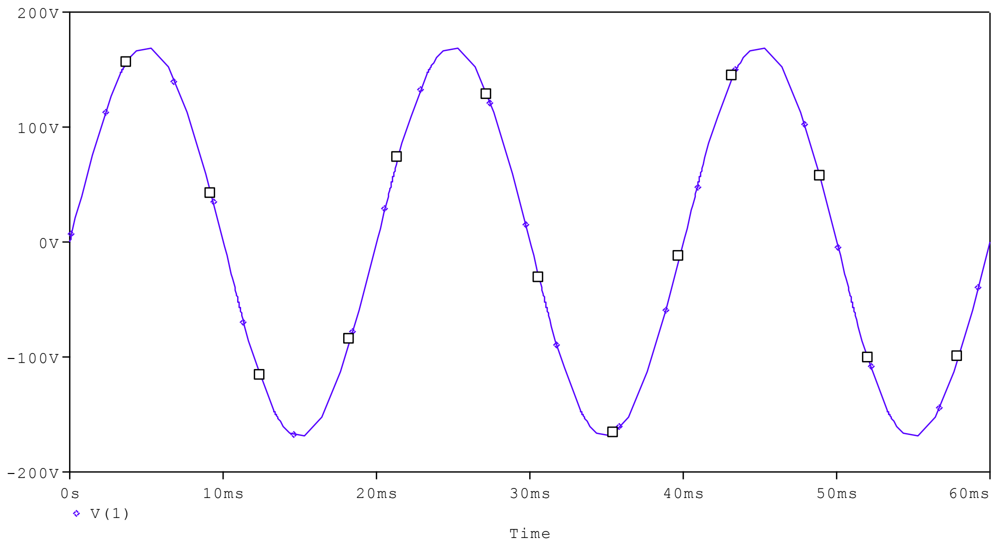


FIG:INPUT WAVEFORM

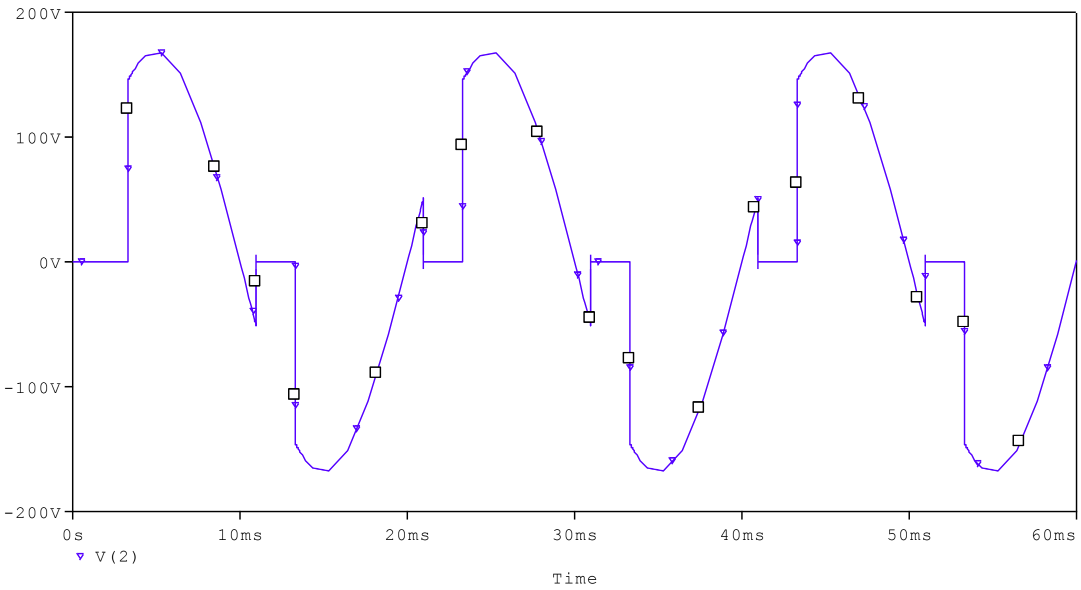


FIG:OUTPUT WAVEFORM

RESULT: .

Experiment No: 4

STABILITY ANALYSIS OF LINEAR TIME INVARIANT SYSTEMS

(Bode, Root Locus, Nyquist plots using MATLAB)

AIM: To analyze the stability of given linear time invariant systems using MATLAB.

SIMULATION TOOLS REQUIRED: PC with MATLAB Software.

PROCEDURE:

1. Open the MATLAB command window clicking on the MATLAB icon.
2. Click on file menu and open new M file.
3. Enter the MATLAB code.
4. Click on the debug menu and run the code.
5. Then copy the obtained plot.

PROGRAMS:

BODE PLOT:

```
num=input('enter the numerator:');

den=input('enter the denominator:');

sys=tf(num,den);

disp(sys);

bode(sys);

[gm,pm,weg,wep]=margin(sys);

if((gm<0)|(pm<0));

   disp('system is unstable');

else

   disp('system is stable');
```

```
end;

clear;
```

RESULTANT WAVE FORMS:

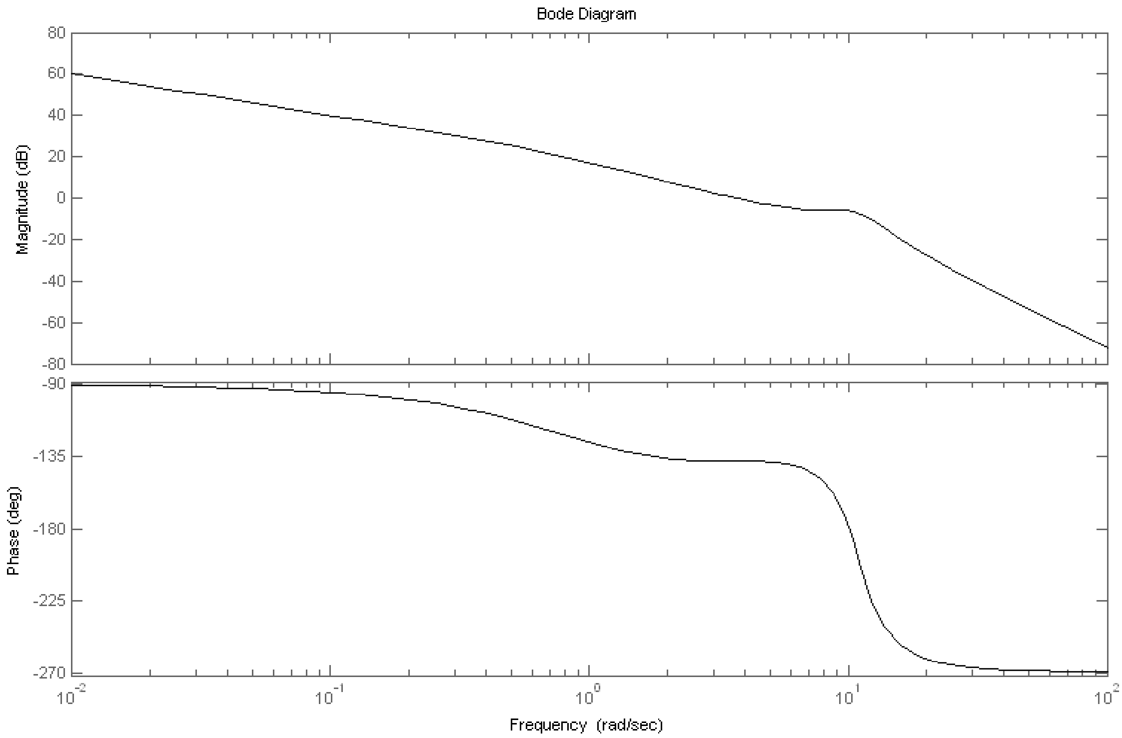


Fig: BODE PLOT

- G(S)=75(1+0.25)/s(s^2+16s+100)
- A=[15 75]
- B=[1 16 100 0]
- System is stable
- Gm=inf
- Pm=91.6644
- Weg=inf
- Wep=0.7573

NYQUIST PLOT:

```
clear;
num=input('enter the numerator:');
den=input('enter the denominator:');
sys=tf(num,den);
disp(sys);
nyquist(sys);
[gm,pm,weg,wep]=margin(sys);
if((gm<0)|(pm<0));
   disp('system is unstable');
else
   disp('system is stable');
end;
clear;
```

RESULTANT WAVE FORMS:

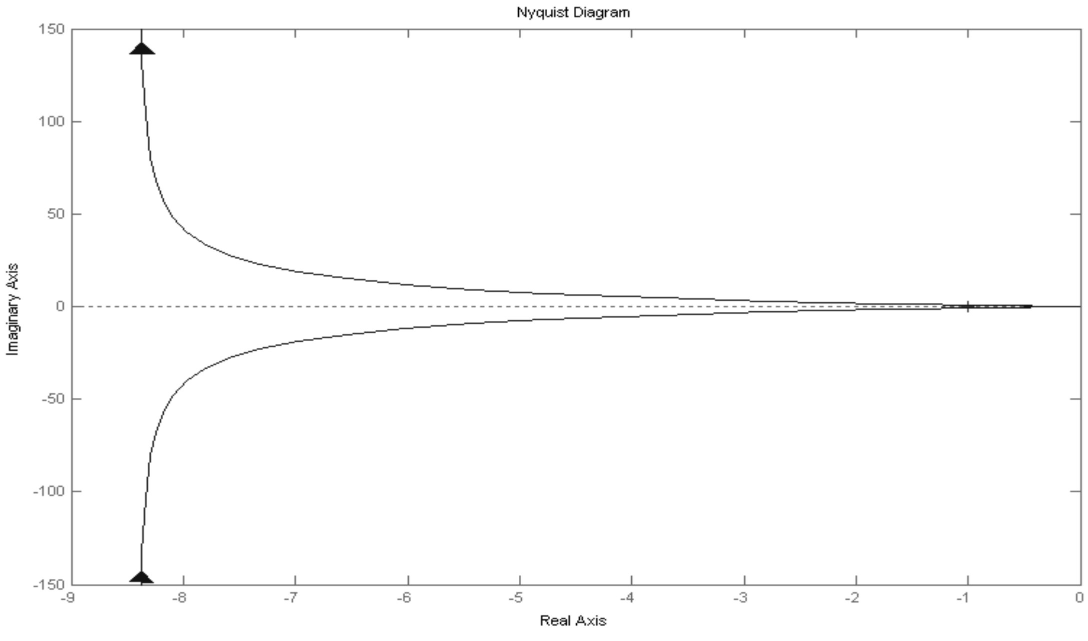


Fig: NYQUIST PLOT

- G(S)H(S)=(s+2)/(s+1)(s-1)
- A=[1 2]
- B=[1 0 -1]
- System is unstable
- Gm=0.50
- Pm=29.7131
- Gef=0
- Pef=1.1414

ROOT LOCUS PLOT:

```
num=input('enter the numerator:');
den=input('enter the denominator:');
sys=tf(num,den);
rlocus(sys);
[r,k]=rlocus(sys);
[m,n]=size(r);
for i=1:n
for j=1:m
if real(r(j,i)>0)
        strl=strcat('the given system is unstable at k',num2str(k(j)));
        disp(strl);
break;
end;
end;
end;
```

RESULTANT WAVE FORMS:

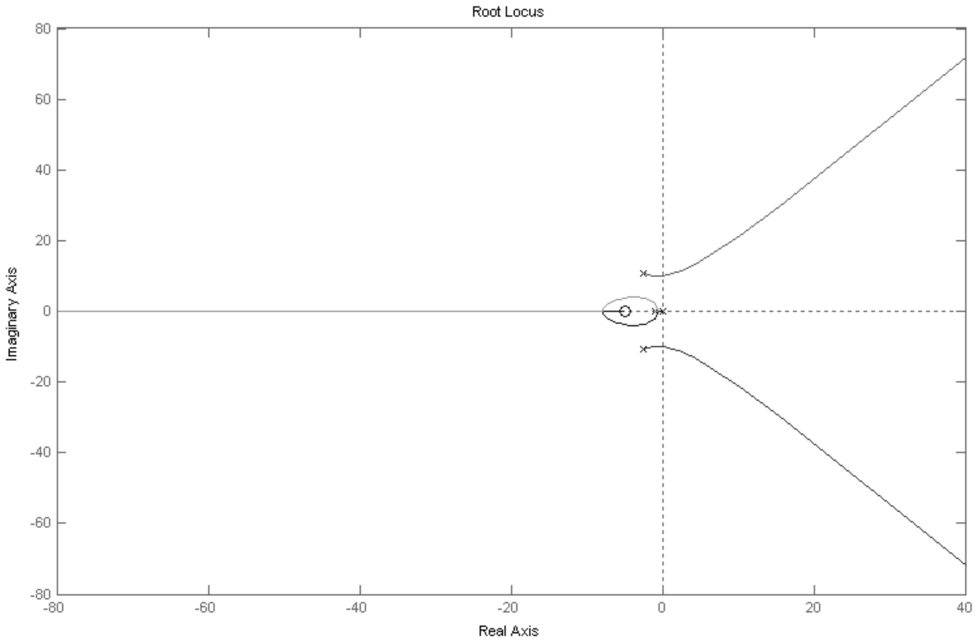


Fig:**ROOT LOCUS PLOT**

- G(s)=k(s+9)/s(s^2+4s+11)
- A=[1 9]
- B=[1 4 11 0]
- System is unstable
- K=13.47
- K=40.54
- K=106.21
- K=278.26
- K=279
- K=310984.34
- K=inf

RESULT:

Experiment No: 5

POWER FLOW SOLUTION OF POWER SYSTEM

Date:

AIM:

To understand, in particular, the mathematical formulation of power flow model in complex form and a simple method of solving power flow problems of small sized system using gauss- siedal iterative algorithm.

Software Required: Matlab Software

Procedure:

1. Enter the command window of mtlab.
2. Create a new M file by selecting File --- new ---M-file
3. Type & Save program in the editor window
4. Execute program by selecting Tools-Run

Steps for writing the program:

1. The Line Impedence are converted are converted to Admittances
2. Y is then initiated to Zero
3. In the first loop the line data is searched and off diagonal elements are entered
4. In the Nested Loop, the line data is searched to find the elements connected to a bus & diagonal elements are formed.

Program:

```
clear
zdata=[1 2 .02 .04
   1 3 .01 .03
   2 3 .0125 .025];
nl=zdata(:,1);
nr=zdata(:,2);
R=zdata(:,3);
X=zdata(:,4);
nbr=length(zdata(:,1));
nbus=max(max(nl),max(nr));
Z=R+(1*j*X);
y=ones(nbr,1)./Z;
Y=zeros(nbus,nbus);
for k=1:nbr
if nl(k)>0 & nr(k).0
```

```
        Y(nl(k),nr(k))=Y(nl(k),nr(k))-y(k);
        Y(nl(k),nr(k))=Y(nr(k),nl(k))
end;
end;
for n=1:nbus
for k=1:nbr
if nl(k)==n|nr(k)==n
            Y(n,n)=Y(n,n)+y(k);
else,end;
end;
end;
Y
a1=input('enter p2 in MW:');
b1=input('enter q2 in MVA:');
a2=input('enter p3 in MW:');
b2=input('enter q3 in MVA:');
pu=input('enter base in MVA:');
p2=(a1/pu);
q2=(b1/pu);

p3=(a2/pu);
q3=(b2/pu);
dx1=1+(0*j);
dx2=1+(0*j);
v1=1.05;
v2=1+(0*j);
v3=1+(0*j);
iter=0;
disp('iter v2 v3');
while abs(dx1)>=0.00001 & abs(dx2)>=0.00001 & iter<15;
    iter=iter+1;
    g1=(((p2-(j*q2))/conj(v2))+(-Y(1,2)*v1)+(-Y(2,3)*v3))/Y(2,2);
    g2=(((p3-(j*q3))/conj(v3))+(-Y(1,3)*v1)+(-Y(2,3)*g1))/Y(3,3);
    dx1=g1-v2;
    dx2=g2-v3;
    v2=v2+dx1;
```

```
    v3=v3+dx2;
    fprintf('%g',iter), disp([v2,v3]);
end;
```

Result:

Experiment No: 6(a)

MODELING OF TRANSFORMER

Date:

AIM: Modelling of Transformer

SIMULATION TOOLS REQUIRED: PC with PSPICE Software.

MEASUREMENTS

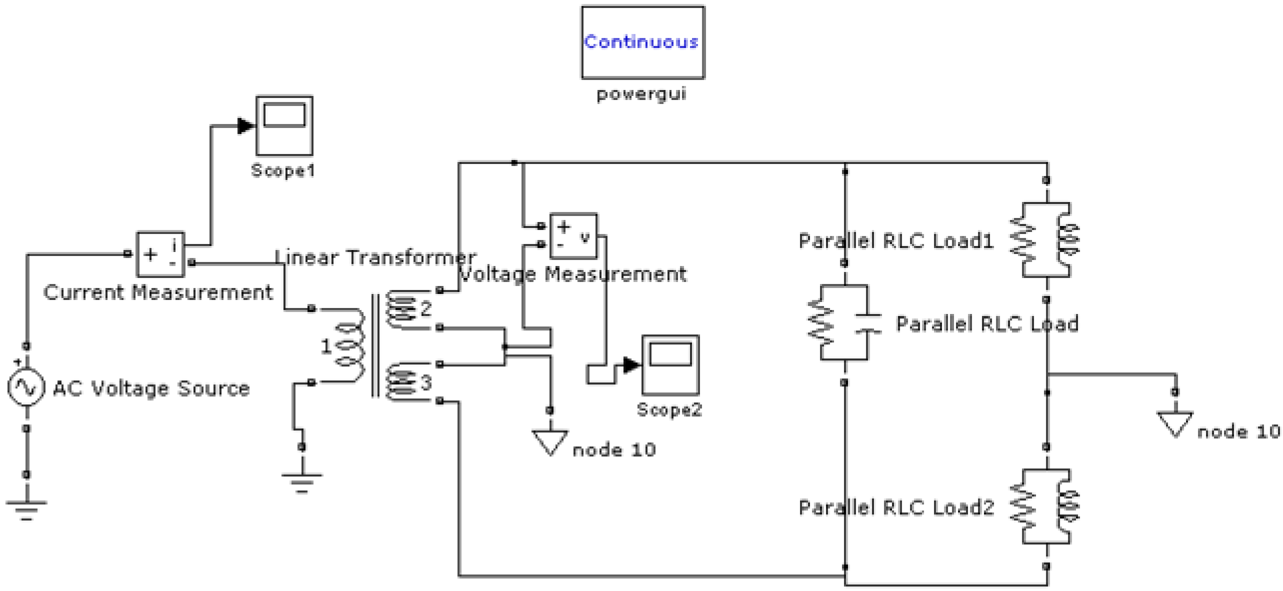
Select Winding voltages to measure the voltage across the winding terminals of the Linear Transformer block.

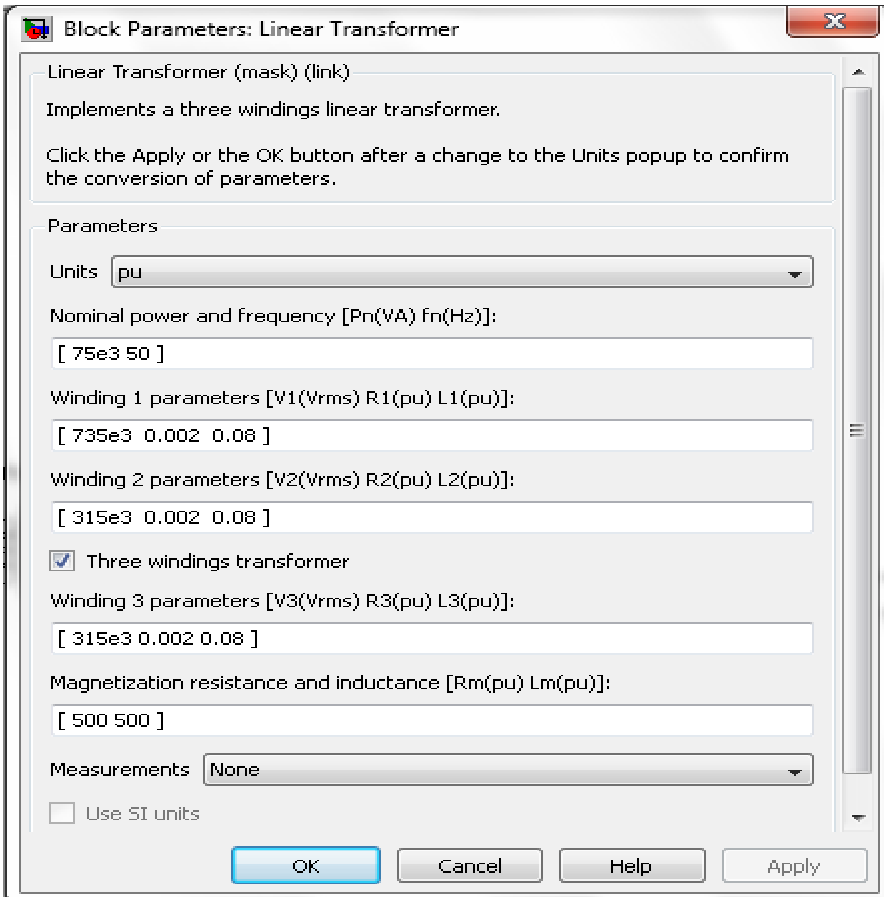
Select Winding currents to measure the current flowing through the windings of the Linear Transformer block.

Select Magnetization current to measure the magnetization current of the Linear Transformer block. Select All voltages and currents to measure the winding voltages and currents plus the magnetizationcurrent.

Place a Multi meter block in your model to display the selected measurements during the simulation.

CIRCUIT DIAGRAM:





RESULTANT WAVE FORMS:

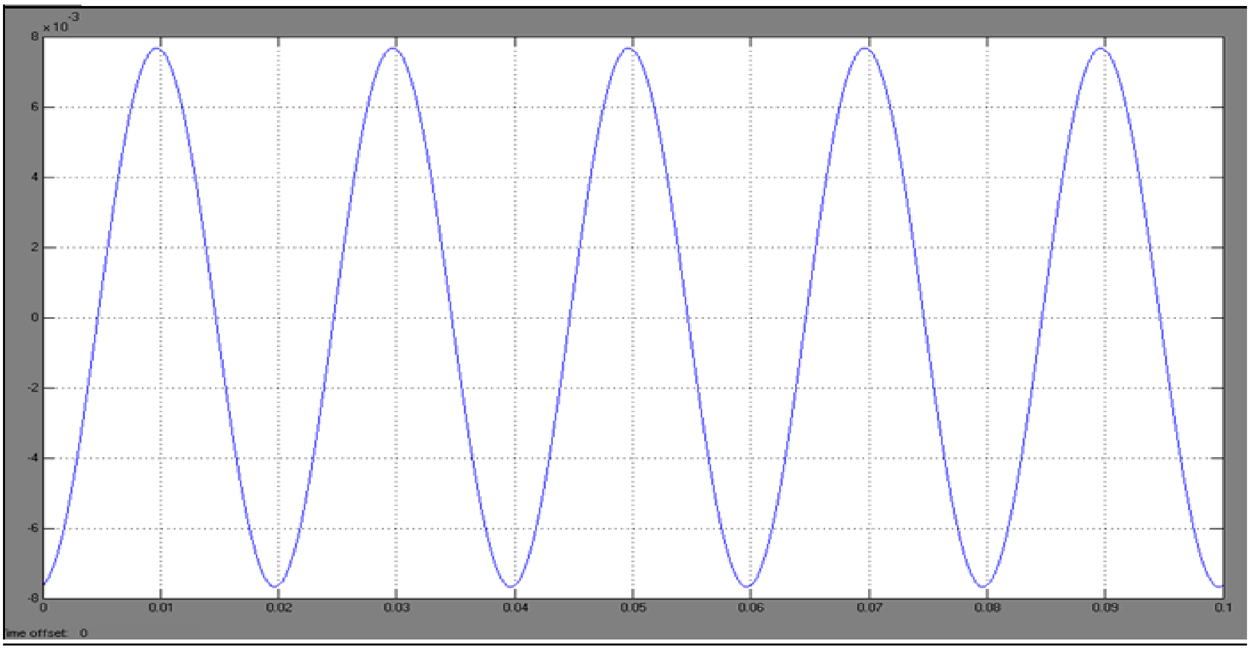


Fig: output voltage waveform

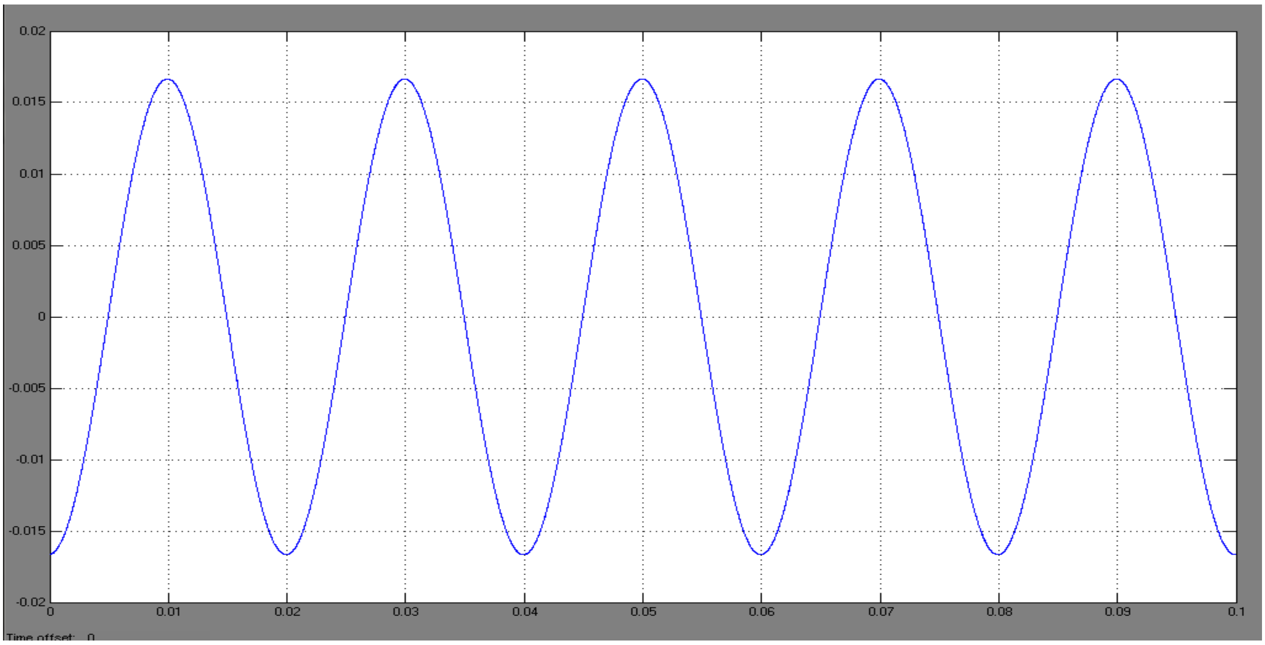


Fig: input current wave form

RESULT:

Experiment No: 6(b)

SIMULATION OF LOSSY TRANSMISSION LINE

Date: 26-8-2013

AIM: To understand modelling and performance of medium transmission lines.

SIMULATION TOOLS REQUIRED: PC with PSPICE Software.

PROBLEM:

A three phase overhead line 200km long R = 0.16 ohm/km and Conductor diameter of 2cm with spacing 4,5,6m transposed. Find A,B,C,D constants ,sending end voltage, current ,power factor and power when the line is delivering full load of 50MW at 132kV ,0.8 pf lagging , transmission efficiency , receiving end voltage and regulation.

PROGRAM:

```
ab=input('value of ab');

bc=input('value of bc');
ca=input('value of ca');
pr=input('receving end power in mw');
vr=input('receving end voltage in kv');
pfr=input('receving end powerfactor');
l=input('length of the line in km');
r=input('resistance/ph/km');
f=input('frequency');
D=input('diameter in m');
rad=D/2;
newrad=(0.7788*rad);
deq=(ab*bc*ca)^(1/3);
L=2*10^(-7)*(log(deq/newrad))
C=(2*pi*8.854*10^-12)/log(deq/rad)
XL=2*pi*f*L*l*1000;
rnew=r*l
Z=rnew+1i*(XL);
Y=1i*(2*pi*f*C*l*1000);
A=1+((Y*Z)/2);
D=A;
B=Z;
C=Y*(1+(Y*Z)/4);
vrph=(vr*10^3)/1.732;
irold=(pr*10^6)/(1.732*vr*10^3*0.8);
k=sin(acos(pfr));
ir=irold*(pfr-(1i*k));
vs=((A*vrph)+(B*ir));
is=((C*vrph)+(D*ir));
angle(vs);
```

```
angle(is);
f=angle(vs);
u=angle(is);
PFS=cos(f-u)
eff=((pr*10^6)/(3*abs(vs)*abs(is)*PFS))*100
reg=((abs(vs)/abs(A))-(abs(vrph)/abs(vrph)))*100
abs(vs)
abs(is)
```

OUTPUT FILE:

```
value of ab: 4
value of bc: 5
value of ca: 6
receving end power in MW : 50
receving end voltage in KV : 132
receving end powerfactor : 0.8
length of the line in Km : 200

resistance/ph/Km : 0.16

frequency:  50

diameter in m : 0.02

L =  1.2902e-006

C = -2.5419e-006 +5.5725e-004i

rnew =    32

A =   0.9772 + 0.0090i

B =  32.0000 +81.0657i

C = -2.5419e-006 +5.5725e-004i

ans =  9.5677e+004

ans =  244.2088

PFS =    0.8067

eff =   88.4248

reg =  9.7909e+00
```

RESULT:

.

Experiment No: 7(a)

PSPICE simulation of Op-Amp based Differentiator Circuit

Date:

AIM: PSPICE simulation of Op-Amp based Differentiator Circuit

SIMULATION TOOLS REQUIRED: PC with PSPICE Software.

CIRCUIT DIAGRAMS:

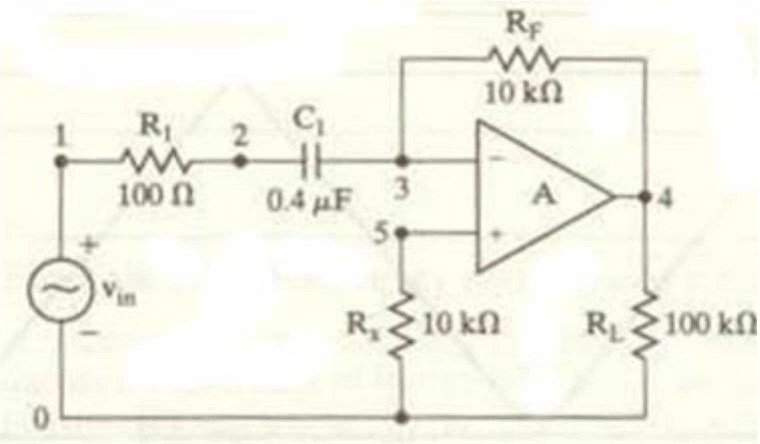


FIG: Op-Amp based Differentiator Circuit

Model Graphs:

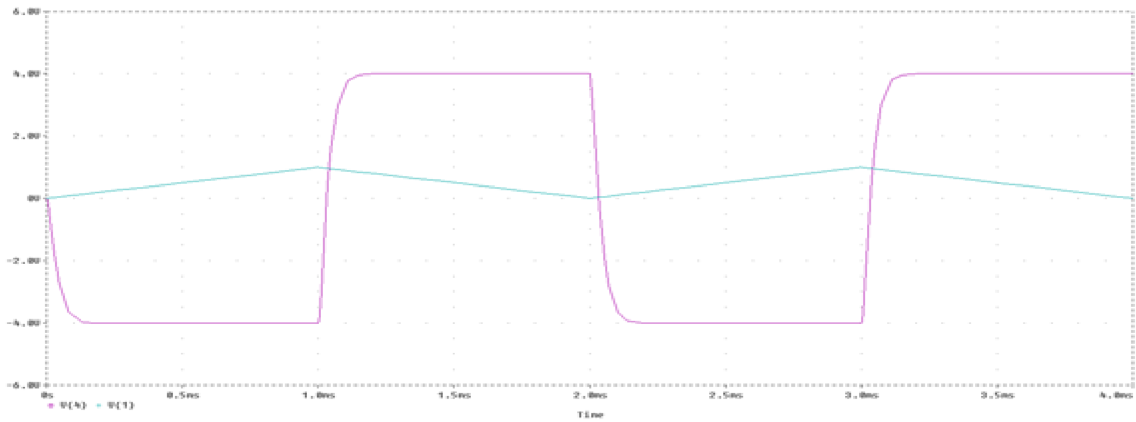


Fig: Op-Amp based Differentiator

Program:

```
vin 1 0 pwl (0 0 1m 1 2m 0 3m 1 4m 0)
r1 1 2 100
rf 3 4 10k
rx 5 0 10k
rl 4 0 100k
c1 2 3 0.4u
*calling subcircuit OPAMP
 xa1 3 5 4 0 OPAMP
*OP AMP sub circuit definition
.subckt OPAMP 1 2 7 4
ri 1 2 2.0e6
gb 4 3 1 2 0.1m
r1 3 4 10k
c1 3 4 1.5619u
ea 4 5 3 4 2e+5
ro 5 7 75
.ends OPAMP
.tran 10u 4m
.plot tran v (4) v(1)
.probe
.end
```

RESULTANT WAVE FORMS:

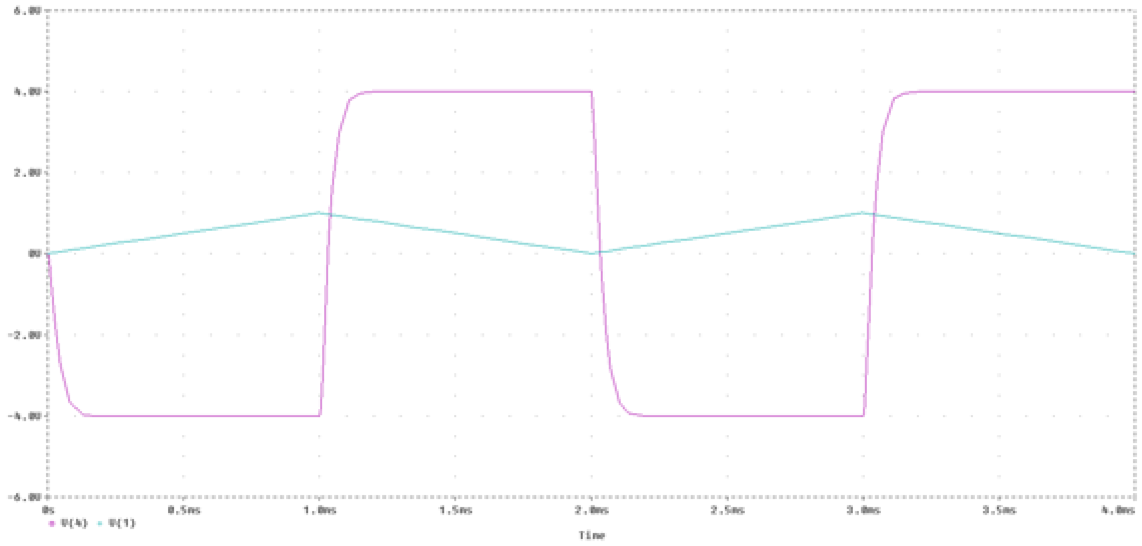


FIG:WAVE FOREM OF Op-Amp based Differentiator Circuit

RESULT:

PSPICE simulation of Op-Amp based Integrator Circuit

Date:

AIM: PSPICE simulation of Op-Amp based Integrator Circuit.

Circuit Diagram:

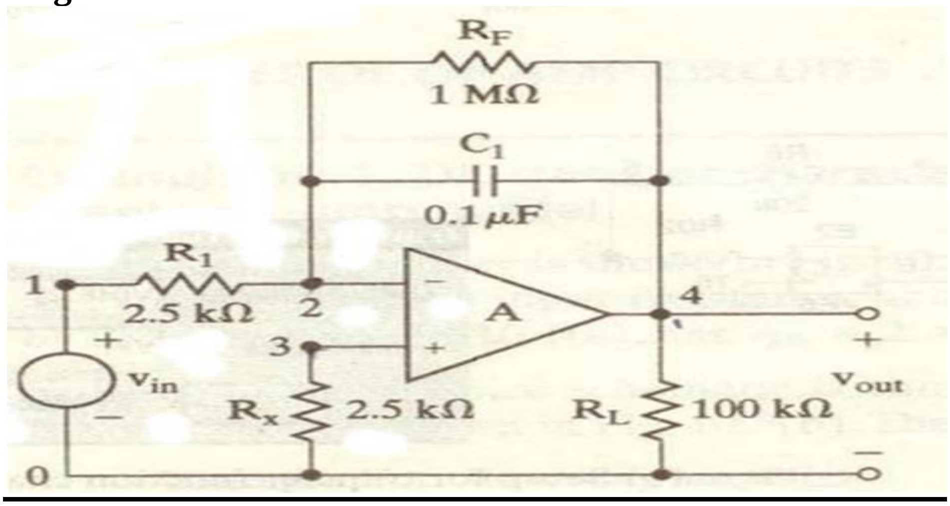


FIG: Op-Amp based Integrator Circuit

Model Graph:

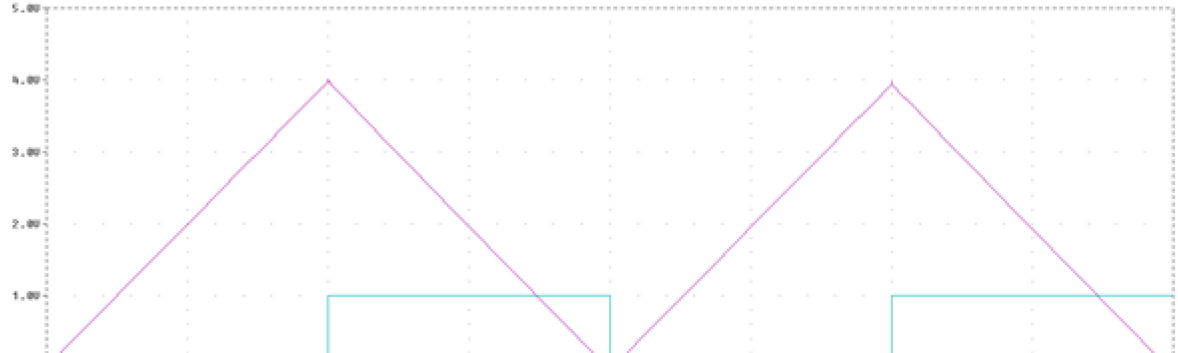


Fig: Op-Amp based Integrator

Program:

Integrator Circuit

```
vin 1 0 pwl(0 0 1n -1 1m -1 1.0001m 1 2m 1 + 2.0001m -1 3m  -1 3.0001m 14m 1  )
r1 1 2 2.5k

rf 2 4 1meg

rx 3 0 2.5k

rl 4 0 100k

c1 2 4 0.1u

*calling subcircuit OPAMP

xa1 2 3 4 0 OPAMP

*OP AMP sub circuit definition

.subckt OPAMP 1 2 7 4

ri 1 2 2.0e6

gb 4 3 1 2 0.1m

r1 3 4 10k

c1 3 4 1.5619u

ea 4 5 3 4 2e+5

R0 5 7 75

.ends OPAMP

.tran 50u 4m

.plot tran v (4) v(1)

.probe

.end
```

RESULTANT WAVE FORMS:

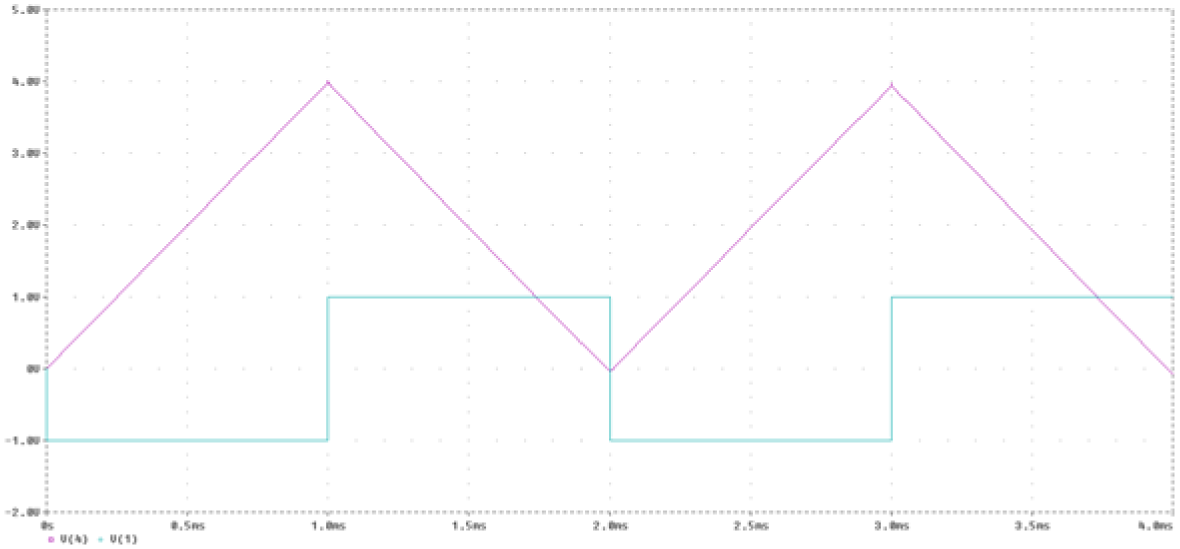


FIG: Op-Amp based Integrator Circuit

Result:

Expt No: 8(a)

PSpice Simulation of DC Circuits-Thevenin's Equivalent

Date:

AIM:

To write a program for the simulation of dc circuit for determining the thevenin's equivalent using PSPICE.

Circuit Diagram:

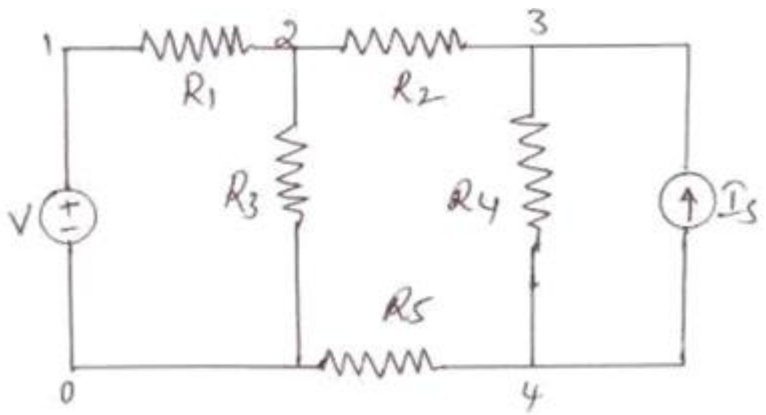


FIG : Thevenin's Equivalent CIRCUIT

Program:

```
PSpice Simulation of DC Circuits-Thevenin's Equivalent

vin 1 0 dc 10

 is 4 3 dc 2

 ix 4 3 dc 0

 r1  1 2 5

 r2  2 3 10

 r3  2 0 20

 r5  4 0 10

 .dc vin 10 10 10

 .print dc v(3,4)
```

.end

RESULT:

PSpice Simulation of DC circuits-Transfer function

Date:

AIM:PSpice Simulation of DC circuits-Transfer Function

Circuit Diagram:

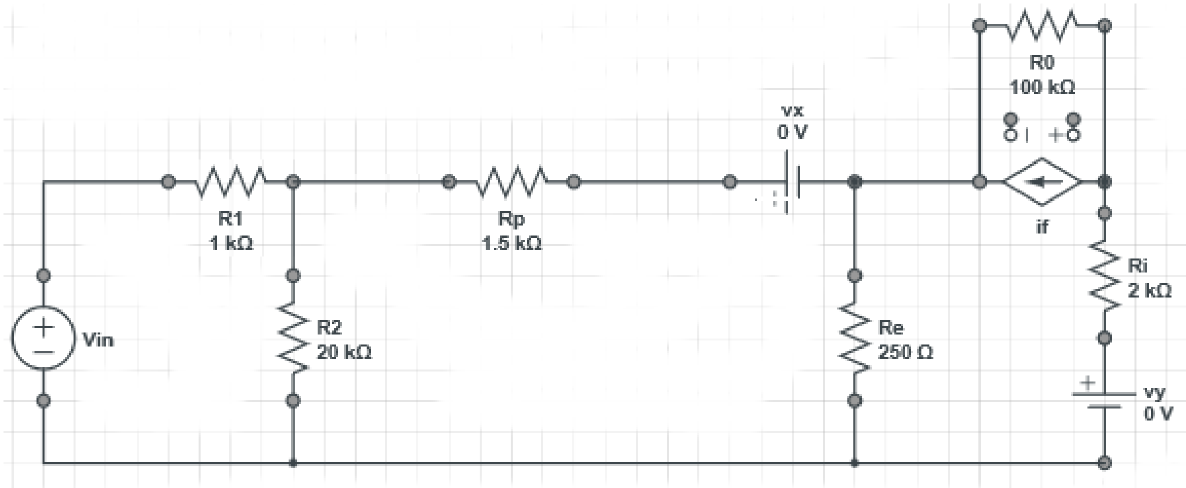


FIG: DC circuits-Transfer function CIRCUIT

Program:

vin 1 0 dc 1

r1 1 2 1k

r2 2 0 20k

rp 2 6 1.5k

re 3 0 250

f1 4 3 vx 40

ro 4 3 100k

rl 4 5 2k

vx 6 3 dc 0

```
vy 5 0 dc 0
.tf v (4) vin
.end
```

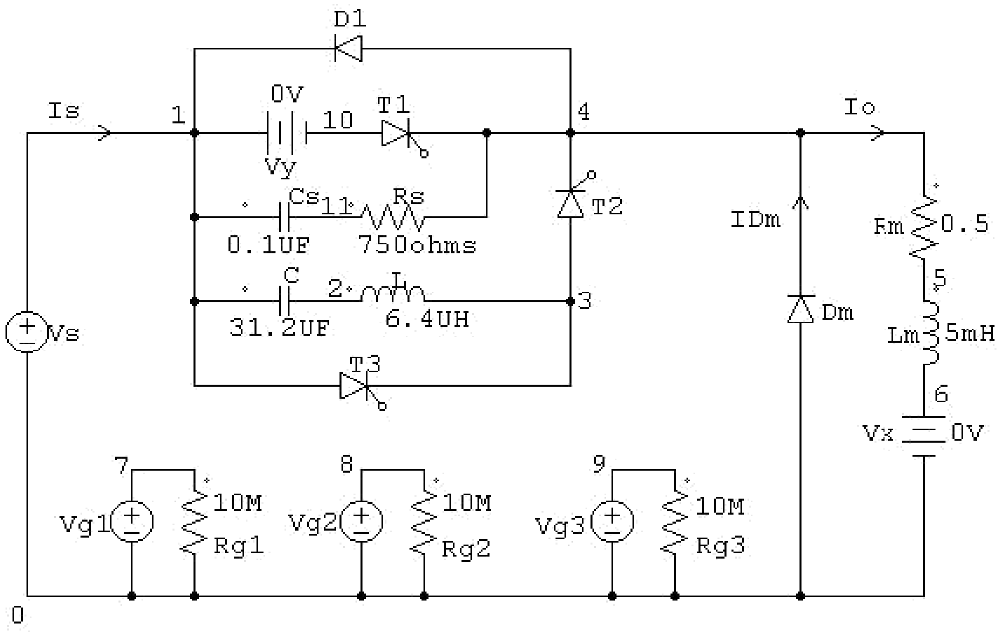
RESULT:

Expt. No: 9(a)

RESONANT PULSE COMMUTATION CIRCUIT

AIM: Pspice analysis of resonant pulse commutation circuit.

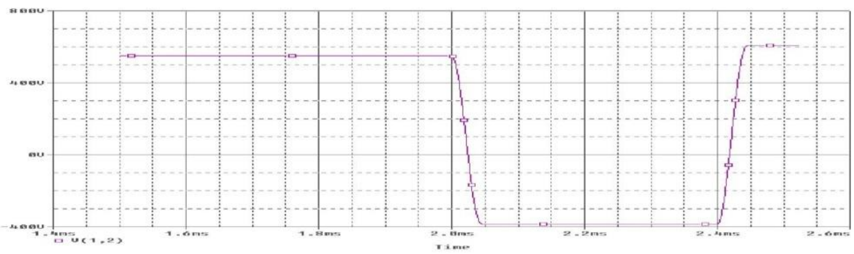
CIRCUIT DIAGRAM:

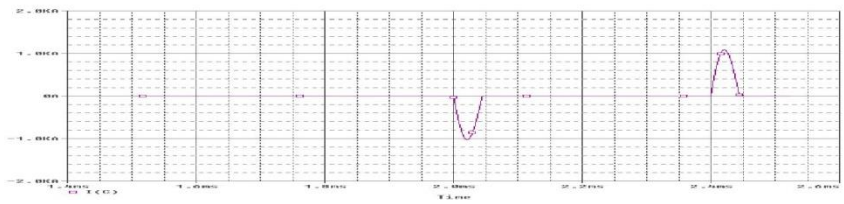


Resonant Pulse Commutation Circuit

FIG: **RESONANT PULSE COMMUTATION CIRCUIT**

Model Graphs:





PROGRAM:

```
RESONANT PULSE COMMU TATION CIRCUIT
VS 1 0 DC 200V
VG1 7 0 PULSE (0 100V 0 1US 1US 0.4MS 1MS)
VG2 8 0 PULSE (0 100V 0.4MS 1US 1US 0.6MS 1MS)
VG3 9 0 PULSE (0 100V 0 1US 1US 0.2MS 1MS)
RG1 7 0 10MEG
RG2 8 0 10MEG
RG3 9 0 10MEG
CS 10 11 0.1UF
RS 11 4 750
C 1 2 31.2UF IC=200V
L 2 3 6.4UH
D1 4 1 DMOD
DM 4 0 DMOD
.MODEL DMOD D (IS=1E-25 BV=1000V)
RM 4 5 0.5
LM 5 6 5MH
VX 6 0 DC 0V
VY 1 10 DC 0V
XT1 10 4 7 0 DCSCR
XT2 3 4 8 0 DCSCR
XT3 1 3 9 0 DCSCR
.SUBCKT DCSCR 1 2 3 4
DT 5 2 DMOD
ST 1 5 3 4 SMOD
.MODEL DMOD D (IS=1E-25 BV=10000V)
.MODEL SMOD VSWITCH (RON=0.1 ROFF=10E+6 VON=10V VOFF=5V)
.ENDS DCSCR
.TRAN 0.5US 3MS 1.5MS 0.5US
.PROBE
.options (abstol=1.000u reltol=0.01 vntol=0.1 ITL5=20000)
.END
```

RESULTANT WAVE FORM:

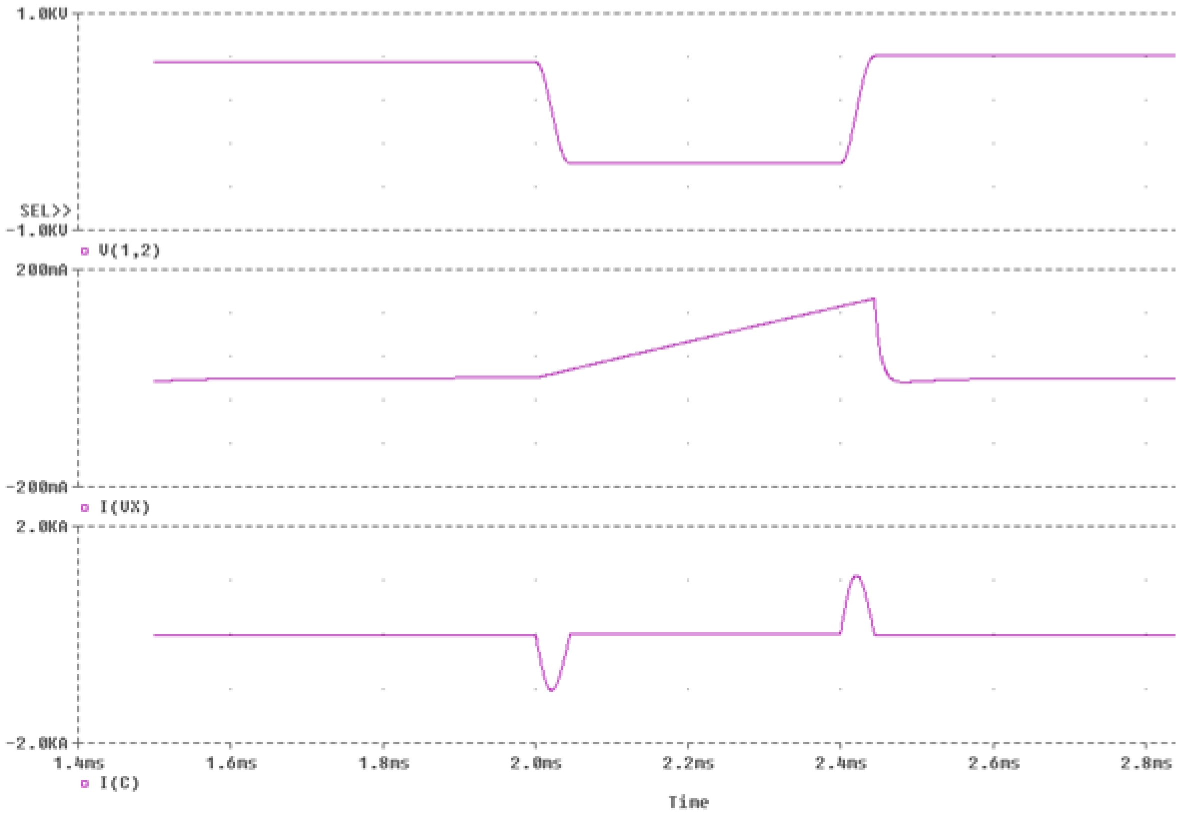


FIG: **WAVE FORMOF RESONANT PULSE COMMUTATION CIRCUIT**

Expt No: 9(b**)**

BUCK CHOPPER

AIM: To analyse Buck chopper circuit using Pspice.

CIRCUIT DIAGRAM:

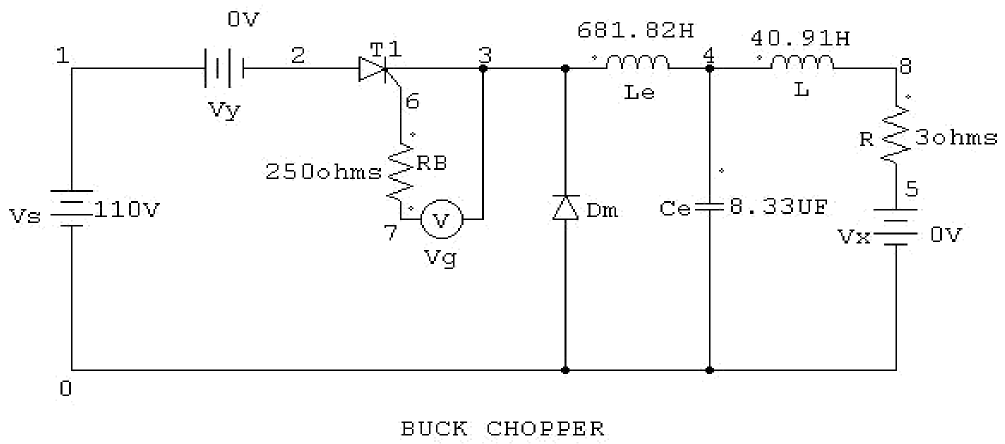
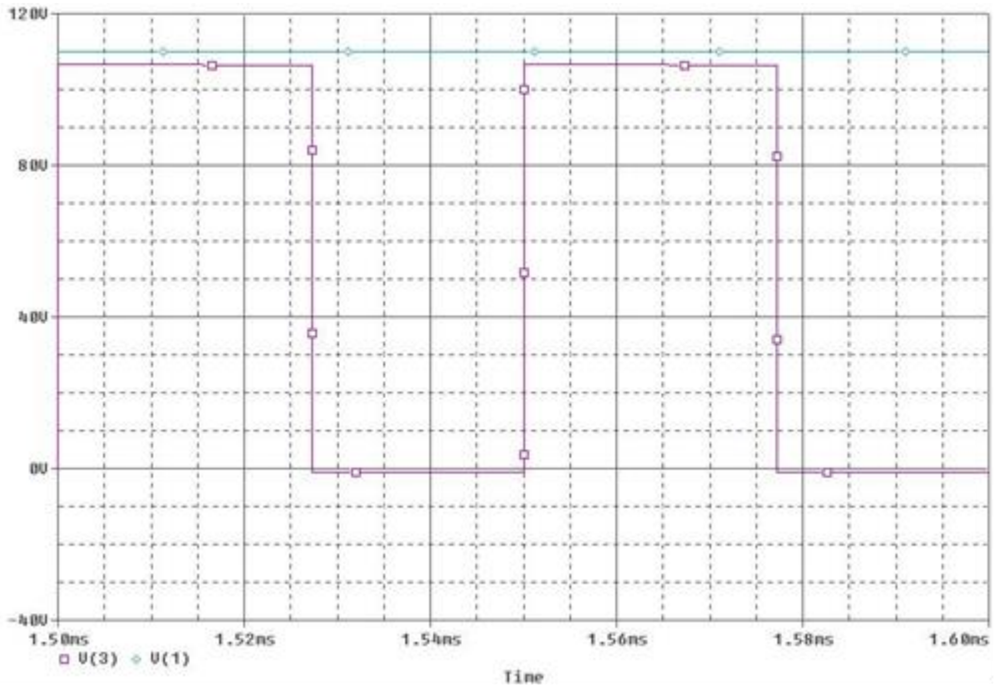


FIG: BUCK CHOPPER CIRCUIT

MODEL GRAPH:



PROGRAM:

```
BUCK CHOPPER
VS 1 0 DC 110V
VY 1 2 DC 0V
VG 7 3 PULSE(0 20 0 0.1NS 0. 1NS 27.28US 50US)
RB 7 6 250
LE 3 4 681.82UH
CE 4 0 8.33UF IC=60V L 4 8 40.91UH
R 8 5 3
VX 5 0 DC 0V
DM 0 3 DMOD
.MODEL DMOD D(IS=2.2E-15 BV=1000V TT=0)
XT1 2 3 6 3 DCSCR
.SUBCKT DCSCR 1 2 3 4
DT 5 2 DMOD
ST 1 5 3 4 SMOD
.MODEL DMOD D(IS=1E-25 B V=1000V)
.MODEL SMOD VSWITCH(R ON=0.1 ROFF=10E+8 VON=10V VOFF=5V)
.ENDS DCSCR
.TRAN 1US 1.6MS 1.5MS 1US
.PROBE
.END
```

RESULTANT WAVE FORM:

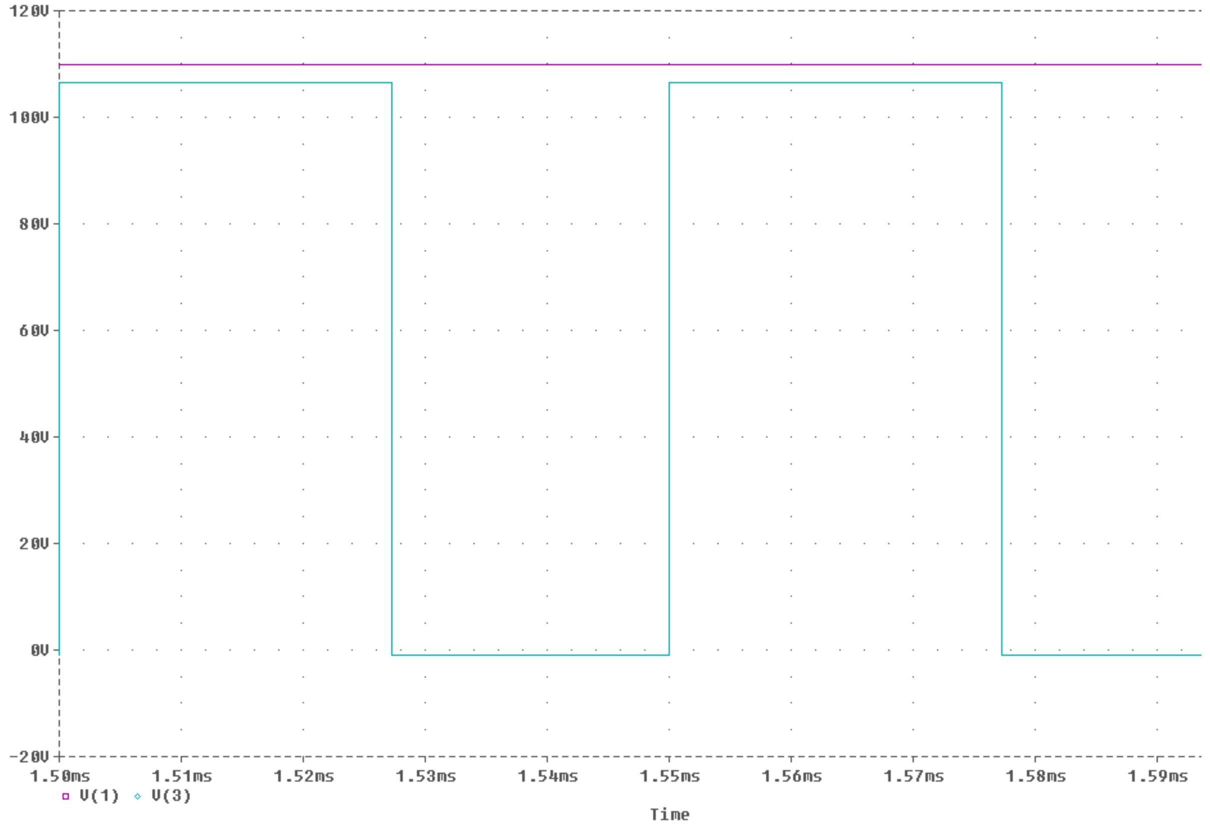


FIG: WAVE FORM OF BUCK CHOPPE

Expt No:10

Simulation of a Single-Phase Inverter with a PWM Control

AIM: To study the characteristics of single phase inverter with PWM control

Circuit Diagram:

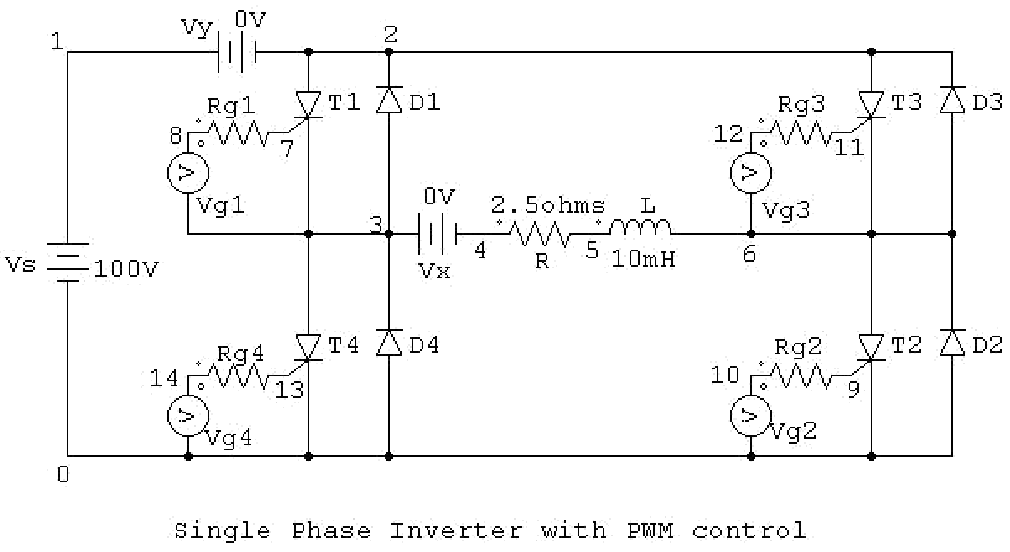
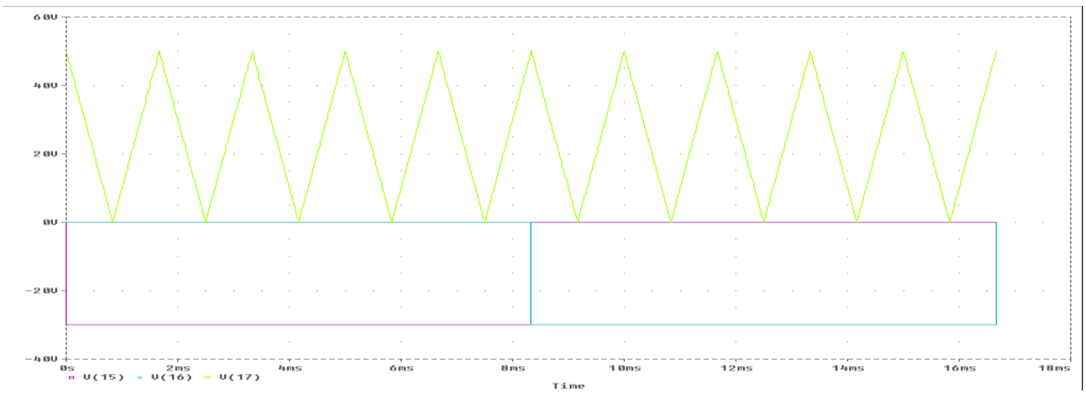


FIG: Simulation of a Single-Phase Inverter with a PWM Control CIRCUIT

MODEL GRAPH:

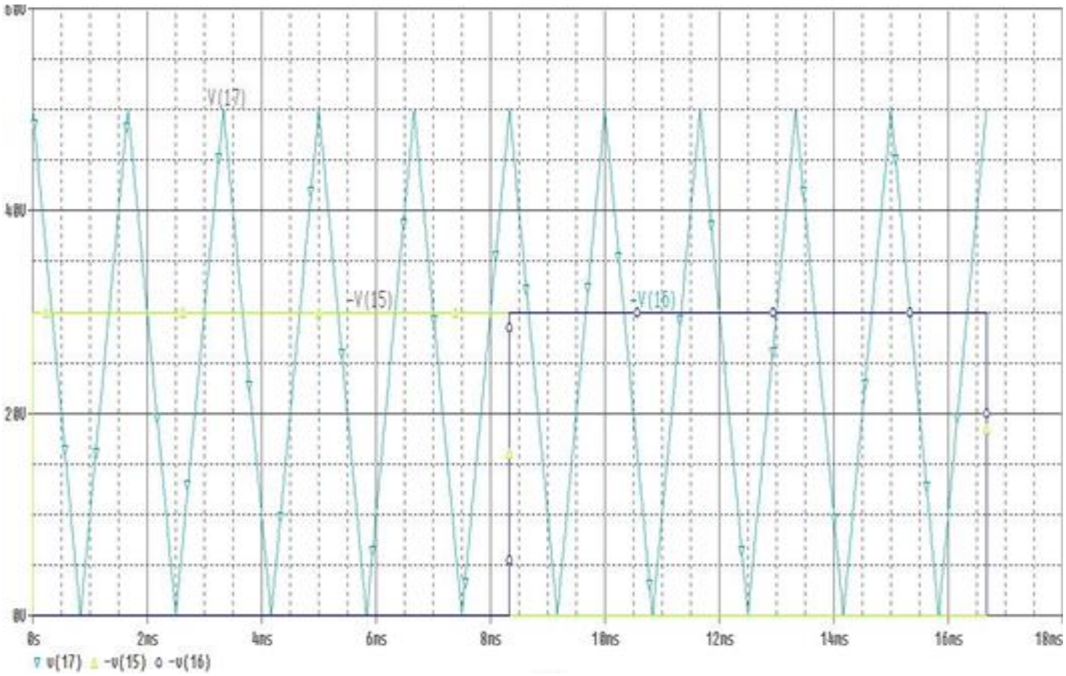


PROGRAM:

```
vs 1 0 dc 100
 vr 17 0 pulse (50 0 0 833.33u 833.33u 1n 1666.67u)
 rr 17 0 2meg
 vc1 15 0 pulse (0 -30 0 1n 1n 8333.33u 16666.67u)
 rc1 15 0 2meg
 vc3 16 0 pulse (0 -30 8333.33u 1 n 1n 8333.33u 16666.67u)
 rc3 16 0 2meg
 *L 5 6 10m
 vx 3 4 dc 0
 vx 3 4 dc 0
 vy 1 2 dc 0
d2 0 6 dmod
d3 6 2 dmod
d4 0 3 dmod
 .model dmod d (is=2.2e-15 bv=1800 tt=0)
q1 2 7 3 qmod
q2 6 9 0 qmod
q3 2 11 6qmod
q4 3 13 0qmod
 .model qmod npn (is=6.734f bf=416.4 cjc=3.638p cje=4.493p)
 rg1 8 7 100
 rg2 10 9 100
 rg3 12 11 100
 rg4 14 13 100
*sub circuit call for PWM control
xpw1 17 15 8 3 pwm
 xpw2 17 15 10 0 pwm
 xpw3 17 16 12 6 pwm
xpw4 17 16 14 0 pwm
 *sub circuit for PWM control

.subckt PWM  1   2   3    4
r1 1 5 1k
 r2 2 5 1k
 rin 5 0 2meg
 rf 5 3 100k
 ro 6 3 75
 co 3 4 10p
 e1 6 4 0 5 2e+5
 .ends pwm
 .tran 10u 16.67m 0 10u
 .probe
 .end
```

RESULTANT WAVE FORM:



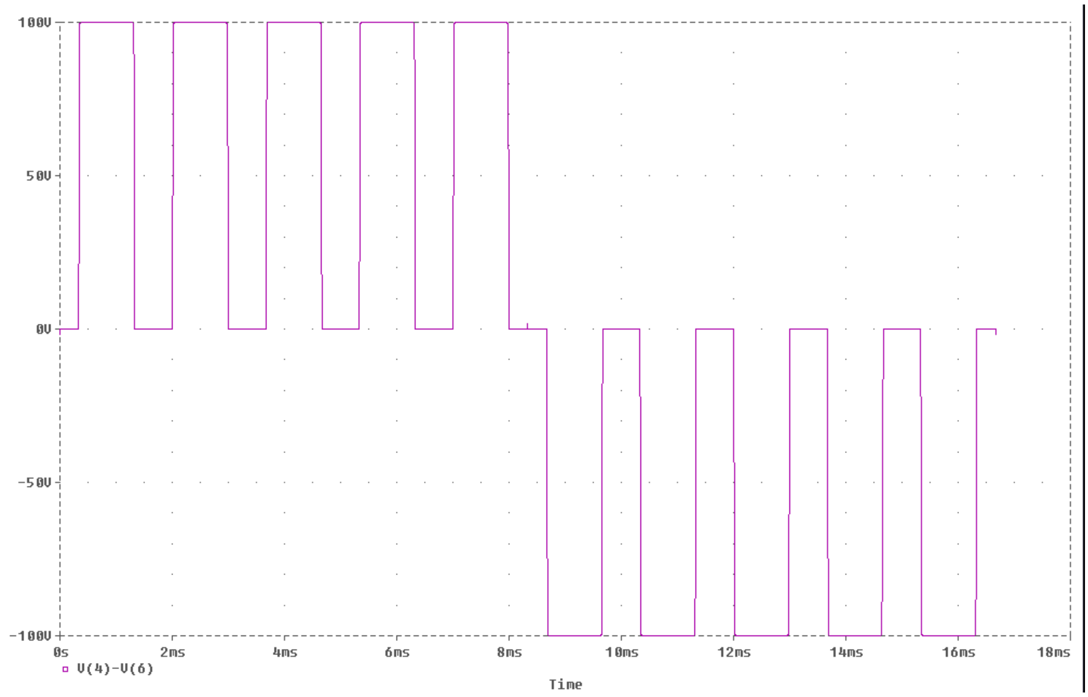


FIG: WAVE FORM OF Simulation of a Single-Phase Inverter with a PWM Control

RESULT: