ELECTRONIC CIRCUIT SIMULATION LAB /EC362 R19

III/IV B.TECH – II SEM-R19

LABORATORY MANUAL

PREPARED BY

MRS.B.MAHALAKSHMI, ASSISTANT PROFESSOR
&
MRS.A.V.MUTYALAMMA, ASSISTANT PROFESSOR

DEPARTMENT OF E.C.E

DEPARTMENT OF
ELECTRONICS AND COMMUNICATION ENGINEERING

BAPATLA WOMEN'S ENGINEERING COLLEGE::BAPATLA

(AFFILIATED TO ACHARYA NAGARJUNA UNIVERSITY)

BAPATLA,BAPATLA DISTRICT,ANDHRA PRADESH,INDIA PIN:522101

(A.Y:2021-22)

INSTITUTE VISION AND MISSION

VISION

Bapatla Women's Engineering College has been established with forethought to organize the growing needs of higher technical education for WOMEN, in order to provide them a competetive edge and a unique niche in the society.

MISSION

- Our mission is to impart the quality education at par with global standards to the students from all over India and in particular those from the local and rural areas.
- We continuously try to maintain high standards so as to make them technologically competent and ethically strong individuals who shall be able to improve the quality of life and economy of our country.

DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

VISION

To produce highly professional and socially responsible women engineers through world class quality technical education and research to meet industry and Society needs in the field of Electronics and Communication Engineering.

MISSION

M1- To inculcate quality education for women through innovative learning practices with the use of modern tools.
M2- To impart professional ethics and moral values through co-curricular and extra-curricular activities as individual and team member.
M3- Empowering graduates through industrial trainings/on-job training in collaboration with industries with-a-view to life-long learning.

Programme Educational Objectives (PEOs)

PEO1: To develop and implementation of Electronics and Communication systems by imparting technical knowledge, professional leadership and communication skills as an individual and team in a multi-disciplinary stream.

PEO2: To train Graduates in solving complex problems, analyzing, designing and imparting research-based knowledge by using modern scientific tools to develop least cost-effective and innovative products.

PEO3: To produce women graduates through life-long learning, who ensure ethical and moral behavior.

Programme Specific Outcomes (PSO's)

PSO1: To understand and analyze basic concepts of electronics and communication in various areas like signal processing, VLSI, embedded systems, communications, digital and analog devices.

PSO2: Ability to design and develop computing systems using concepts of mathematics, electronics engineering and other related inter disciplines to asses and resolve societal, environmental, health safety and cultural issues with professional ethics.

Programme Outcomes (POs)

- **PO1: Engineering knowledge:** Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.
- **PO2: Problem analysis:** Identify, formulate, review research literature, and analyse complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.
- **PO3: Design/development of solutions:** Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.
- **PO4: Conduct investigations of complex problems:** Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.
- **PO5: Modern tool usage:** Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools prediction and modelling to complex engineering activities with an understanding of the limitations.
- **PO6: The engineer and society:** Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice
- **PO7: Environment and sustainability:** Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and the need for sustainable development.
- **PO8: Ethics:** Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering prac tice.
- **PO9: Individual and team work:** Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.
- **PO10: Communication:** Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and receive clear instructions.
- **PO11: Project management and finance:** Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in team, to manage projects and in multidisciplinary environments.
- **PO12: Life-long learning:** Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.

ACHARYA NAGARJUNA UNIVERSITY
NAGARJUNA NAGAR, GUNTUR

COURSE STRUCTURE

Course Details			**Scheme of Instruction**			**Scheme of Examination**			
Course	Code/ Regulation	Subject	L	T	P	Maxi Internal Marks	Maxi External Marks	Total Marks	Credits
III/IV B.TECH ECE	**EC362/ R19**	**Electronic Circuit Simulation LAB**	**0**	**0**	**3**	**40**	**60**	**100**	**2**

EC 352 **ELECTRONIC CIRCUIT SIMULATION LABS**

1. Obtain the V-I characteristics of silicon and Germanium diodes.
2. Design a Zener diode voltage regulator.
3. Design and verify the operating point for a self-bias circuit.
4. Study the characteristics of a half wave and full wave rectifier.
5. Study the characteristics of a bridge rectifier.
6. Obtain the frequency response of a CE amplifier.
7. Obtain the frequency response of a two stage RC couple CE amplifier.
8. Design and simulate class A power Amplifier.
9. Simulate a differentiator and integrator using OPAMP.
10. Simulate a low pass and high pass filter using OPAMP.
11. Simulate a RC phase shift and Wein bridge oscillator using OPAMP.
12. Design and simulate a constant resistance and bridged T equalizer.
13. Simulate an Amplitude Modulator and Demodulator.
14. Simulate an Amplitude Modulator and Demodulator.
15. Simulate a Frequency Modulator and Demodulator.

NOTE: A minimum of 10(Ten) experiments have to be performed and recorded by the candidate to attain eligibility for University Practical Examination

SYLLABUS

EC362R19 ELECTRONIC CIRCUIT SIMULATION LABORATORY

L	T	P	C
0	0	3	1.5

OBJECTIVES:

- ✓ To know thebasics of PSPICE and development of program for Electronic circuits using ORCAD PSPICE A/D.
- ✓ To learn how to obtain the transient response and frequency response to various electronic circuits

LIST OF EXPERIMENTS

1. Study the characteristics of a half wave and full wave rectifier

2. Study the characteristics of a bridge rectifier.

3. Obtain the frequency response of a CE amplifier.

4. Obtain the frequency response of a two stage RC couple CE amplifier.

5. Design and simulate class A power Amplifier.

6. Simulate a differentiator and integrator using OPAMP.

7. Simulate a low pass and high pass filter using OPAMP.

8. Simulate a RC phase shift and Wein bridge oscillator using OPAMP.

9. Design and simulate a constant k type Lowpass and High pass filters.

10.Simulate an Amplitude Modulator and Demodulator.

List of Experiment(s) beyond syllabus:

1. Simulate an Astable multivibrator using OPAMP.

OUTCOMES:

Upon Completion of the course, the students will be able to:

- ✓ Study the characteristics of a half wave, full wave and bridge rectifier.
- ✓ Obtain the frequency response of a CE amplifier and a two stage RC couple CE amplifier.
- ✓ Simulate an Amplitude Modulator and Demodulator and a differentiator, integrator, low pass filter, high pass filter, RC phase shift oscillator and Wein bridge oscillator using OPAMP.
- ✓ Design a class A power Amplifier

.

Course Outcomes (COs):

Course No.	Course Outcome (Student should be able to)	Bloom's Taxonomy Level
C362.1	Study the characteristics of a half wave, full wave and bridge rectifier.	Understand
C362.2	Obtain the frequency response of a CE amplifier and a two stage RC couple CE amplifier.	Analyze
C362.3	Simulate a differentiator, integrator ,low pass filter ,high pass filter, RC phase shift oscillator and Wein bridge oscillator using OPAMP.	Analyze
C362.4	Design a class A power Amplifier	Create
C362.5	Simulate an Amplitude Modulator and Demodulator	Analyze

Mapping of Course Outcomes with PSOs and POs:

S.No.	Course out come	Program Specific Outcomes (PSOs)		Program Outcomes(Pos)											
		PSO 1	PSO 2	PO 1	PO 2	PO 3	PO 4	PO 5	PO6	PO 7	PO 8	PO 9	PO 10	PO 11	PO 12
1	C362.1	l	l	1	-	1	-	2	-	-	-	1	-	-	1
2	C362.2	l	l	1	-	1	-	2	-	-	-	1	-	-	1
3	C362.3	l	l	1	-	1	-	2	-	-	-	1	-	-	1
4	C362.4	l	l	1	-	1	-	2	-	-	-	1	-	-	1
5	C362.5	2	2	2	2	2	1	2	2	-	-	2	1	-	2
	C362	1.2	1.2	1 .2	2	1.2	1	2	2	-	-	1.2	1	-	1.2

Correlation Levels:---1: Low 2: Medium 3: Strong, '-' No Correlation

INDEX

Pgm.No.	Name of the Experiment	Page No.
I	Introduction to PSPICE &It's Basic Syntaxes	8
1	Study the characteristics of a half wave and full wave rectifier	15
2	Study the characteristics of a bridge rectifier	20
3	Obtain the frequency response of a CE amplifier	27
4	Obtain the frequency response of a two stage RC couple CE amplifier.	35
5	Design and simulate class A power Amplifier	46
6	Simulate a differentiator and integrator using OPAMP.	52
7	Simulate a low pass and high pass filter using OPAMP.	62
8	Simulate a RC phase shift and Wein bridge oscillator using OPAMP.	68
9	Design and simulate a constant k type Lowpass and High pass filters.	75
10	Simulate an Amplitude Modulator and Demodulator.	82
List of Experiment(s) beyond syllabus		
1.	Simulate an Astable multivibrator using OPAMP	85

Introduction to PSPICE &It's Basic Syntaxes

This is a brief summary of the SPICE simulation program with integratedcircuit emphasis, or its personal computer version PSPICE, electric circuit analysis program. The original SPICE computer program was developed toanalyze complex electric circuits, particularly integrated circuits. It was developed at the University of California at Berkeley in the late 1960s. Since it was developed under U.S. government funding, it is not proprietary and can be freely copied, used, and distributed. This was written for use on large mainframe computers of the time. In the early 1980s the MicroSim Corporation developed a personal computer version of SPICE called PSPICE. A number of important modifications were made, particularly in the plotting of data via the .PROBE function. Since then a number of commercial firms have modified and developed their own PCversions. But essentially the core engine is that of the original SPICE code. The MicroSim version of PSPICE, version 8, was acquired by Cadence Design Systems. Theirs is version 10.0, called OrCAD Capture. A Windows-based version is available free from www. cadence.com. The OrCAD Capture program was originally called Schematic in the MicroSim version and contains a number of enhancements.A number of books use of SPICE and PSPICE. Both the MicroSim version 8 and the OrCAD version 10 are contained in a CD at the end of other textbooks .

There are two methods of entering and executing a PSPICE program. The first method is the Direct Method, described here, where one enters the program code using an ASCII text editor (supplied with PSPICE).

Note: SPICE and PSPICE make no distinction between lowercase and uppercase letters. Then this text file is run using the PSpice A/D section of the program and the output is examined with the text editor.

The second method is the Schematic Method (now called Capture), where the user "draws" the circuit diagram directly on the screen and then executes that program. The Direct Method is generally the more rapid method of solving relatively simple problems. The Schematic (Capture), method has the advantage of visually seeing whether the circuit components are connected as intended for more complex circuits, but for most simple problems is a bit more time consuming to set up than the Direct Method since numerous windows and drop-down menus must be navigated in the Schematic or Capture method.

The Direct Method is simply to type in the PSPICE program using the TextEdit feature. To enter this and prepare the program, we select the following in this sequence.

1. Select TextEdit (lower button on the vertical toolbar on the left).
2. Type the program.
3. Save the program as XXX.cir or XXX.in and close it.
4. Select PSpice A/D (second button on the vertical toolbar on the left).
5. Click on File, Open and select the file stored previously. The program will run automatically and the output will be stored in file XXX.out.
6. Click on File, Run Probe to plot waveforms or File, Examine Output to examine the printed output.
7. Alternatively, you could recall the TextEditprogam and select File, Open, XXX.out to examine the printed output, which is selfexplanatory.

PSPICE BASICS

SUMMARY of PSPICE commands, variables, etc.

General

Title line must always be first
Use "+" to continue statement on the next line

Use "*" to comment out an entire line
Use ";" to comment out characters following semicolon

Suffixes

f	femto	10^{-15}
p	pico	10^{-12}
n	nano	10^{-9}
u	micro	10^{-6}
m	milli	10^{-3}
k	kilo	10^{3}
meg	mega	10^{6}
g	giga	10^{9}
t	tera	10^{12}

Resistors Capacitors and Inductors:

Syntax:

NAME NODE A NODE B VALUE
with NAME Rxxx = resistor, Lxxx = inductor, Cxxx = capacitor

Example:

CS 3 4 1.0uF

.MODEL statement

MODEL NAME	DESCRIPTION
D	Diode
NPN	npn Bipolar Junction Transistor
PNP	pnp Bipolar Junction Transistor
NMOS	n-Channel MOSFET
PMOS	p-Channel MOSFET
NJF	n-Channel JFET
PJF	p-Channel JFET
GASFET	n-Channel GaAs MESFET
RES	resistor
CAP	capacitor
IND	inductor

BJT Parameters

Parameter	Description	Default value

BF	Forward beta	100
CJE	Base-emitterdepletion capacitance	0
CJC	Base-collector depletion capacitance	0
VAR	Reverse Early voltage	infinite
VAF	Forward Early voltage	infinite
BR	Reverse beta	1
NR	Reverse emission coefficient	1
NF	Forward emission coefficient	1

Syntax:

Qxxx collector node base node emitter node Modelname

Example:

Q1 3 2 1 NPN
.MODEL NPN NPN(BF=200 CJC=20pf CJE=20pf)

MOSFET Parameters

Parameter	Description	Default value
VTO	Threshold voltage	0
KP	Conductance parameter	0.02 mA/V^2
LEVEL	choose zero for ideal	
CBD	Drain-to-substrate capacitance	0
CBS	Source-to-substrate capacitance	0
TOX	Oxide-layer thickness	10^{-7} m

Syntax:

Mname drain gate source bulk/substrate NFET

Example:

M1 10 20 0 0 NFET
.MODEL NFET NMOS(LEVEL=0 VTO=2 KP=0.1)

DIODE Parameters

Parameter	Description	Default value
IS	Saturation Current	10^{-14}A
CJO	Junction capacitance at VD = 0	0
VJ	Reverse-breakdown voltage	1V
BV	Reverse-breakdown voltage	infinite
IBV	Current at VD = BV	10^{-10}A
RS	Series Ohmic resistance	0
N	Emission coefficient (ideality factor)	1

Syntax:

Dxxxanode(+) cathode (-) Modelname

Example:

D1 2 1 DIODE
.MODEL DIODE D(IS=4.7E-12,N=1)

TYPES OF ANALYSES

.DC (Large-Signal Transfer Characteristic)

Syntax

.DC SOURCE START STOP INCR
SOURCE is the voltage or current source
Transfer characteristics are obtained by incrementing the SOURCE from START to STOP in steps of INCR

Example:

.DC VIN -5 5 0.1

.AC (Sinusoidal Steady-State Frequency Response)

Syntax:

.AC SCALE-TYPE NPOINTS FSTART FSTOP

SCALE-TYPE:

LIN linear increments
NPOINTS specifies the number of frequencies between FSTART and FSTOP

DEC logarithmic increments
NPOINTS defines the number of points per decade

OCT octaves of frequency
NPOINTS defines the number of points per octave

Example:

.AC DEC 50 1K 1MEG
Performs a sinusoidal steady-state analysis over 3 decades of frequency from 1kHz to 1Mhz, computing the response at 50 points within each decade

If an analysis at a single frequency is desired, NPOINTS should be set to one and FSTART = FSTOP

.TRAN (Transient Analysis)

To compute the output of the circuit as a function of time

Syntax:

.TRAN TSTEP TSTOP TSTART TINCR UIC(optional)
Analysis begins at t = 0 and ends at t = TSTOP seconds
output is printed at every increment TSTEP
if TSTART is nonzero it only plots starting at TSTART
if TINCR is omitted the default TINCR value is (TSTOP - TSTART)/50

UIC user initial condition...the initial conditions specified in capacitor and inductor element statements are used instead

NOTE: .IC control line can be used to set an initial condition for any node in the circuit

.TF (Small-Signal Transfer Function)

Syntax:

.TF OUTPUT_VARIABLE INPUT_VARIABLE

To perform a small signal analysis using small-signal linear models for nonlinear devices

INPUT_VARIABLE voltage or current source

Example:

.TF V(1,2) VIN

Finds small-signal voltage gain between nodes 1 and 2 as the output variable

.TF I(V1) VS

Finds the small-signal transconductance using the current through the source V1 as the output variable and VS as the input variable

.TEMP (Temperature Specification)

Syntax

.TEMP VAL1 VAL2 VAL3 ...

Perform the analysis at each temperature VAL in Celsius

Default is 27C (room temperature)

.IC (Initial Conditions)

Syntax:

.IC V(N) = VALUE V(J,K) = VALUE

where N, J, K are node numbers

.STEP (Parameter Step)

Syntax:

.STEP (TYPE) STEPVAR START STOP INCR

Perform simulation while stepping the parameter STEPVAR between the values START and STOP in increments of INCR

NOTE: if .AC or .DC or .TRAN is included Spice will perform these analyses for each value of STEPVAR

TYPE: is optional; LIN (linear increments), DEC (increment points per decade), OCT (incr points per octave), LIST (must be followed by a list of STEPVAR values separated by spaces). Default is LIN

BRIEF SPICE SUMMARY

Here's a brief reference of the SPICE devices and statements. Parameters enclosed by braces { } are required, while, those in brackets [ ] are optional. Parameters followed by an asterisk { }* should be repeated as necessary. Each SPICE vendor may have other parameters or commands unique to their version of SPICE.

DEVICES

C device - Capacitor.

C{name} {+node} {-node} [{model}] {value} [IC={initial}]
Examples:
CLOAD 15 0 20pF
CFDBK 3 33 CMOD 10pF IC=1.5v

D device - Diode.

D{name} {+node} {-node} {model} [area]
Examples:
DCLAMP 14 0 DMOD

I device - Current Source.

I{name} {+node} {-node} [[DC] {value}] [AC {mag} [{phase}]]
Examples:
IBIAS 13 0 2.3mA
IAC 2 3 AC .001
IPULSE 1 0 PULSE(-1mA 1mA 2ns 2ns2ns 50ns 100ns)
I3 26 77 AC 1 SIN(.002 .002 1.5MEG)
See Input Sources for PULSE, SIN and more

J device - Junction FET.

J{name} {d} {g} {s} {model} [{area]}
Examples:
JIN 100 1 0 JFAST

K device - Inductor Coupling.

K{name} L{name} { L{name} }* {coupling}
Examples:
KTUNED L3OUT L4IN .8
KXFR1 LPRIM LSEC .99

L device - Inductor.

L{name} {+node} {-node} [model] {value} [IC={initial}]
Examples:
LLOAD 15 0 20mH
L2 1 2 .2e-6
LSENSE 5 12 2uH IC=2mA

M device - MOSFET.

M{name} {d} {g} {s} {sub} {mdl} [L={value}] [W={value}]
+ [AD={value}] [AS={value}]

+ [PD={value}] [PS={value}]
+ [NRD={value}] [NRS={value}]

Examples:
M1 14 2 13 0 PNOM L=25u W=12u
M13 15 3 0 0 PSTRONG

Q device - Bipolar Transistor.

Q{name} {c} {b} {e} [{subs}] {model} [{area}]
Examples:
Q1 14 2 13 PNPNOM
Q13 15 3 0 1 NPNSTRONG 1.5

R device - Resistor.

R{name} {+node} {-node} [{model}] {value}
Examples:
RLOAD 15 0 2k

S device - Voltage-Controlled Switch.

S{name} {+node} {-node} {+control} {-control} {model}
Examples:
S12 13 17 2 0 SMOD

T device - Transmission Line.

T{name} {A+} {A-} {B+} {B-} Z0={value}
[TD={val} | F={val}[NL={val}]]
Examples:
T1 1 2 3 4 Z0=220 TD=115ns
T2 1 2 3 4 Z0=50 F=5MEG NL=0.5

V device - Voltage Source.

V{name} {+node} {-node} [[DC] {value}] [AC {mag} [{phase}]]
Examples:
VBIAS 13 0 2.3mV
VAC 2 3 AC .001
VPULSE 1 0 PULSE(-1mV 1mV 2ns 2ns2ns 50ns 100ns)
V3 26 77 AC 1 SIN(.002 .002 1.5MEG)
See Input Sources for PULSE, SIN and more.

X device - Subcircuit Call.

X{name} [{node}]* {subcircuit name}
Examples:
X12 100 101 200 201 DIFFAMP

CONTROLLED SOURCES

E device - Voltage Controlled Voltage Source VCVS.

E{name} {+node} {-node} {+cntrl} {-cntrl} {gain}
E{name} {+node} {-node} POLY({value}) {{+cntrl} {-cntrl}}* {{coeff}}*
Examples:
EBUFF 1 2 10 11 1.0
EAMP 13 0 POLY(1) 26 0 500

F device - Current Controlled Current Source CCCS.

F{name} {+node} {-node} {vsource name} {gain}
Examples:
FSENSE 1 2 VSENSE 10.0

G device - Voltage Controlled Current Source VCCS.
G{name} {+node} {-node} {+control} {-control} {gain}
Examples:
GBUFF 1 2 10 11 1.0

H device - Current Controlled Voltage Source CCVS.
H{name} {+node} {-node} {vsource name} {gain}
H{name} {+node} {-node} POLY({value}) { {vsource name} }* {{coeff}}*
Examples:
HSENSE 1 2 VSENSE 10.0
HAMP 13 0 POLY(1) VIN 500

INPUT SOURCES

EXPONENTIAL
EXP({v1} {v2} {trise_delay} {tau_rise} {tfall_delay} {tau_fall})
PULSE
PULSE({v1} {v2} {tdelay} {trise} {tfall} {width} {period})
PIECE WISE LINEAR
PWL({time1} {v1} {time2} {v2} ... {time3} {v3})
SINGLE FREQUENCY FM
SFFM({voffset} {vpeak} {fcarrier} {mod_index} {fsignal})
SINE WAVE
SIN({voffset} {vpeak} {freq} {tdelay} {damp_factor} {phase})

ANALOG BEHAVIORAL MODELING

VALUE
E|G{name} {+node} {-node} VALUE {expression}
Examples:
GMULT 1 0 VALUE = { V(3)*V(5,6)*100 }
ERES 1 3 VALUE = { I(VSENSE)*10K }

TABLE
E|G{name} {+node} {-node} TABLE {expression} = (invalue, outvalue)*
Examples:
ECOMP 3 0 TABLE {V(1,2)} = (-1MV 0V) (1MV, 10V)
LAPLACE
E|G{name} {+node} {-node} LAPLACE {expression} {s expression}
Examples:
ELOPASS 4 0 LAPLACE {V(1)} {10 / (s/6800 + 1)}
FREQ
E|G{name} {+node} {-node} FREQ {expression} (freq, gain, phase)*
Examples:
EAMP 5 0 FREQ {V(1)} (1KZ, 10DB, 0DEG) (10KHZ, 0DB, -90DEG)
POLY
E|G{name} {+node} {-node} POLY(dim) {inputs X} {coeff k0,k1,...} [IC=value]
Examples:
EAMP 3 0 POLY(1) (2,0) 0 500
EMULT2 3 0 POLY(2) (1,0) (2,0) 0 0 0 0 1
ESUM3 6 0 POLY(3) (3,0) (4,0) (5,0) 0 1.2 0.5 1.2

COEFFICIENTS

POLY(1)

y = k0 + k1·X1 + k2·X1·X1 + k3·X1·X1·X1 + ...

POLY(2)

y = k0 + k1·X1 + k2·X2 +
+ k3·X1·X1 + k4·X2·X1 + k5·X2·X2 +
+ k6·X1·X1·X1 + k7·X2·X1·X1 + k8·X2·X2·X1 +
+ k9·X2·X2·X2 + ...

POLY(3)

y = k0 + k1·X1 + k2·X2 + k3·X3 +
+ k4·X1·X1 + k5·X2·X1 + k6·X3·X1 +
+ k7·X2·X2 + k8·X2·X3 + k9·X3·X3 + ...

STATEMENTS

AC - AC Analysis.

.AC [LIN][OCT][DEC] {points} {start} {end}

Examples:

.AC LIN 101 10Hz 200Hz

.AC DEC 20 1MEG 100MEG

.DC - DC Analysis.

.DC [LIN] {varname} {start} {end} {incr}

.DC [OCT][DEC] {varname} {start} {end} {points}

Examples:

.DC VIN -.25 .25 .05

.DC LIN I2 5mA -2mA 0.1mA VCE 10V 15V 1V

.FOUR - Fourier Analysis.

.FOUR {freq} {output var}*

Examples:

.FOUR 10KHz v(5) v(6,7)

.IC - Initial Transient Conditions.

.IC { {vnode} = {value} }*

Examples:

.IC V(2)=3.4 V(102)=0

.MODEL – Device Model.

.MODEL {name} {type}

Typename	Devname	Devtype
CAP	Cxxx	capacitor
IND	Lxxx	inductor
RES	Rxxx	resistor
D	Dxxx	diode
NPN	Qxxx	NPN bipolar
PNP	Qxxx	PNP bipolar
NJF	Jxxx	N-channel JFET
PJF	Jxxx	P-channel JFET
NMOS	Mxxx	N-channel MOSFET
PMOS	Mxxx	P-channel MOSFET
VSWITCH	Sxxx	voltage controlled switch

Examples:
.MODEL RMAX RES (R=1.5 TC=.02 TC2=.005)
.MODEL QDRIV NPN (IS=1e-7 BF=30)

.NODESET – Initial bias point guess.
.NODESET { {node}={value} }*
Examples:
.NODESET V(2)=3.4 V(3)=-1V

.NOISE - Noise Analysis.
.NOISE {output variable} {name} [{print interval}]
Examples:
.NOISE V(5) VIN

.PLOT – Plot Output.
.PLOT [DC][AC][NOISE][TRAN] [[{output variable}*]
Examples:
.PLOT DC V(3) V(2,3) V(R1) I(VIN)
.PLOT AC VM(2) VP(2) VG(2)

.PRINT – Print Output.
.PRINT [DC][AC][NOISE][TRAN] [{output variable}*]
Examples:
.PRINT DC V(3) V(2,3) V(R1) IB(Q13)
.PRINT AC VM(2) VP(2) VG(5) II(7)

.PROBE – Save simulation output PSPICE COMMAND.
.PROBE [output variable]*
Examples:
.PROBE
.PROBE V(3) VM(2) I(VIN)

.SENS - Sensitivity Analysis.
.SENS {output variable}*
Examples:
.SENS V(9) V(4,3) I(VCC)

.SUBCKT - Subcircuit Definition.
.SUBCKT {name} [{node}*]
Examples:
.SUBCKT OPAMP 1 2 101 102

.TEMP – Temperature Analysis.
.TEMP {value}*
Examples:
.TEMP 0 27 125

.TF – DC Transfer Function.
.TF {output variable} {input source name}
Examples:
.TF V(5) VIN

.TRAN - Transient Analysis.
.TRAN {print step value} {final time} [{no print time}
[{step ceiling value}]] [UIC]

Examples:
.TRAN 5NS 100NS

TOPSPICE SIMULATION – STEP BY STEP RUNNING PROCEDURE:-

TopSpice from Penzar is a powerful and easy-to-use SPICE program (demo version)Here's how to run a simulation for the SPICE file named LPFILTER1.CIR, plot various results and modify the circuit.

DOWNLOADING THE CIRCUIT FILE.

1. From the **Circuit Collection** page on this site, click on a circuit such as **RC Low-Pass Filter**. You'll see a schematic along with circuit and simulation notes that you can print out.

2. Download the SPICE file LPFILTER1.CIR by clicking on **download the SPICE file** on the web page.

RUNNING A SIMULATION.

1. Start the **TopSpice** demo program.

2. To open a SPICE circuit file select **Open Saved SPICE (text) File**. Move to folder where you saved the SPICE circuit (LPFILTER.CIR), click on the filename, then click on Open. The SPICE circuit file will appear in a text editor window.

3. Select **Analysis > Run Simulation** or click the simulation icon (traffic light) along the top toolbar.

PLOTTING VOLTAGES AND CURRENTS.

1. TopSpice will automatically open a window to view your results. Select **Transient** (time) or **AC** (frequency) to view the waveforms for the simple low-pass filter.

2. TopSpice automatically plots the waveforms specified by your PRINT statements. For example, ".PRINT TRAN V(2)" plots the voltage at node 2 for transient (time) analysis.

3. You can add additional waveforms by selecting **Trace, Add**. Finally, click one of the voltages or currents in the variable list box and then click Add.

NOTE The PRINT statement also generates a numerical table of V(2) in a text file with the ".out" extension. You can view this file by selecting **View**> **Output File** from the menu. This output text file is available from most SPICE simulators and can be viewed with a simple text editor like Notepad. The PLOT statements creates a line-printer style plot in the text file using ascii text characters.

MODIFYING YOUR CIRCUIT.

1. In the text editor's window of TopSpice, modify component values or change analysis commands in your SPICE circuit file.

2. Save the file by selecting **File > Save**.

3. Now you can run a new simulation by clicking on the simulation icon (traffic light). It's easy to change a component's value and quickly see the effect on your circuit.

PSPICE SIMULATION – STEP BY STEP

PSPICE by OrCAD is one of the most popular circuit simulators. Here's how to run a simulation for the SPICE file named LPFILTER1.CIR, plot various results and modify the circuit.

DOWNLOADING THE CIRCUIT FILE.

1. From the **Circuit Collection** page on this site, click on a circuit such as **RC Low-Pass Filter**. You'll see a schematic along with circuit and simulation notes that you can print out.

2. Download the SPICE file LPFILTER1.CIR by clicking on **download the SPICE file** on the web page.

RUNNING A SIMULATION.

1. Start the **PSPICE AD LITE** demo program.

2. To open a SPICE circuit file select **Open > File**. Drop down the **Files of Type** box and select **Circuit Files (*.cir)** to make the *.cir files visible. Move to folder where you saved your SPICE circuit (LPFILTER.CIR), click on the filename, then click on **Open**. Your SPICE circuit file will appear in a text editor window.

3. Select **Simulation > Run** or click the simulation icon (blue arrow) along the top toolbar.

PLOTTING VOLTAGES AND CURRENTS.

1. PSPICE will automatically open a window to view your results. Select **Transient** (time) or **AC** (frequency) to view the waveforms for the simple low-pass filter.

2. To plot waveforms select **Trace > Add Trace**. In the **Trace expression:**box type a voltage or current such as **V(2)** or **I(R1)** and then **OK**. Or you can click on the available output variables in the selection box and then **OK**.

NOTE The PRINT statement also generates a numerical table of V(2) in a text file with the ".out" extension. You can view this file by selecting **View**> **Output File** from the menu. This output text file is available from most SPICE simulators and can be viewed with a simple text editor like Notepad. The PLOT statements creates a line-printer style plot in the text file using ascii text characters.

MODIFYING YOUR CIRCUIT.

1. In the text editor's window of PSPICE, modify component values or change analysis commands in your SPICE circuit file.

2. Save the file by selecting **File > Save**.

3. Now you can run a new simulation by clicking on the simulation icon (blue arrow). It's easy to change a component's value and quickly see the effect on your circuit.

SwitcherCAD III SIMULATION – STEP BY STEP

SwitcherCAD III from Linear Technology is the only free version with no component limit. Here's how to run a simulation for the SPICE file named LPFILTER1.CIR, plot various results and modify the circuit.

DOWNLOADING THE CIRCUIT FILE.

1. From the **Circuit Collection** page on this site, click on a circuit such as **RC Low-Pass Filter**. You'll see a schematic along with circuit and simulation notes that you can print out.

2. Download the SPICE file LPFILTER1.CIR by clicking on **download the SPICE file** on the web page.

RUNNING A SIMULATION.

1. Start the **SwictherCAD III** program.

2. To open a SPICE circuit file select **Open > File**. Drop down the **Files of Type** box and select **Files of Type (*.cir)** to make the **.cir* files visible. Move to folder where you saved your SPICE circuit (LPFILTER.CIR), click on the filename, then click on **Open**. Your SPICE circuit file will appear in a text editor window.

3. Select **Simulate > Run** or click the simulation icon (running man) on the top toolbar.

NOTE You can specify an **.AC or .TRAN Analysis** in the netlist but **not both simulatenously**. If both exist in your netlist, just place a "*" before one of the statements to tell SPICE to ignore the line.

PLOTTING VOLTAGES AND CURRENTS.

1. SwitcherCAD III automatically opens a window for you to select which waveforms to plot. The PRINT statement tells SPICE which waveforms to make available. Excluding the PRINT statement makes all of the voltages and currents available for selection.

2. To add waveforms select **Plot Settings > Add Traces**. Click on one or more of the voltages or currents available and then click **OK**.

MODIFYING YOUR CIRCUIT.

1. In the text editor's window, modify component values or change analysis commands in your SPICE circuit file.

2. Save the file by selecting **File > Save** or can click the simulation icon (running man) to save and run a new simulation.

PGM:01	Obtain the V-I characteristics of silicon and Germanium diodes.

AIM :-To Obtain the V-I characteristics of silicon and Germanium diodes using PSPICE software.

Apparatus:-PC, PSPICE A/D software

Program:-

Forward Bias:

```
Rps 1 0 0 30v
D1 1 2 da
V 1 2 dc (0 30v)
I 2 3 dc (0 20ma)
R 3 0 1k
.model da d
.tran 0.01ms 50ms
.plot tran v(1) v(2) I(2) I(3)
.plot tran V(2) I(3)
.probe
.end
```

Reverse Bias:

```
Rps 1 0 0 30v
D1 2 1  da
V 2 1  dc (0 30v)
I 3 2  dc (0 20ma)
R 3 0 1k
.model da d
.tran 0.01ms 50ms
```

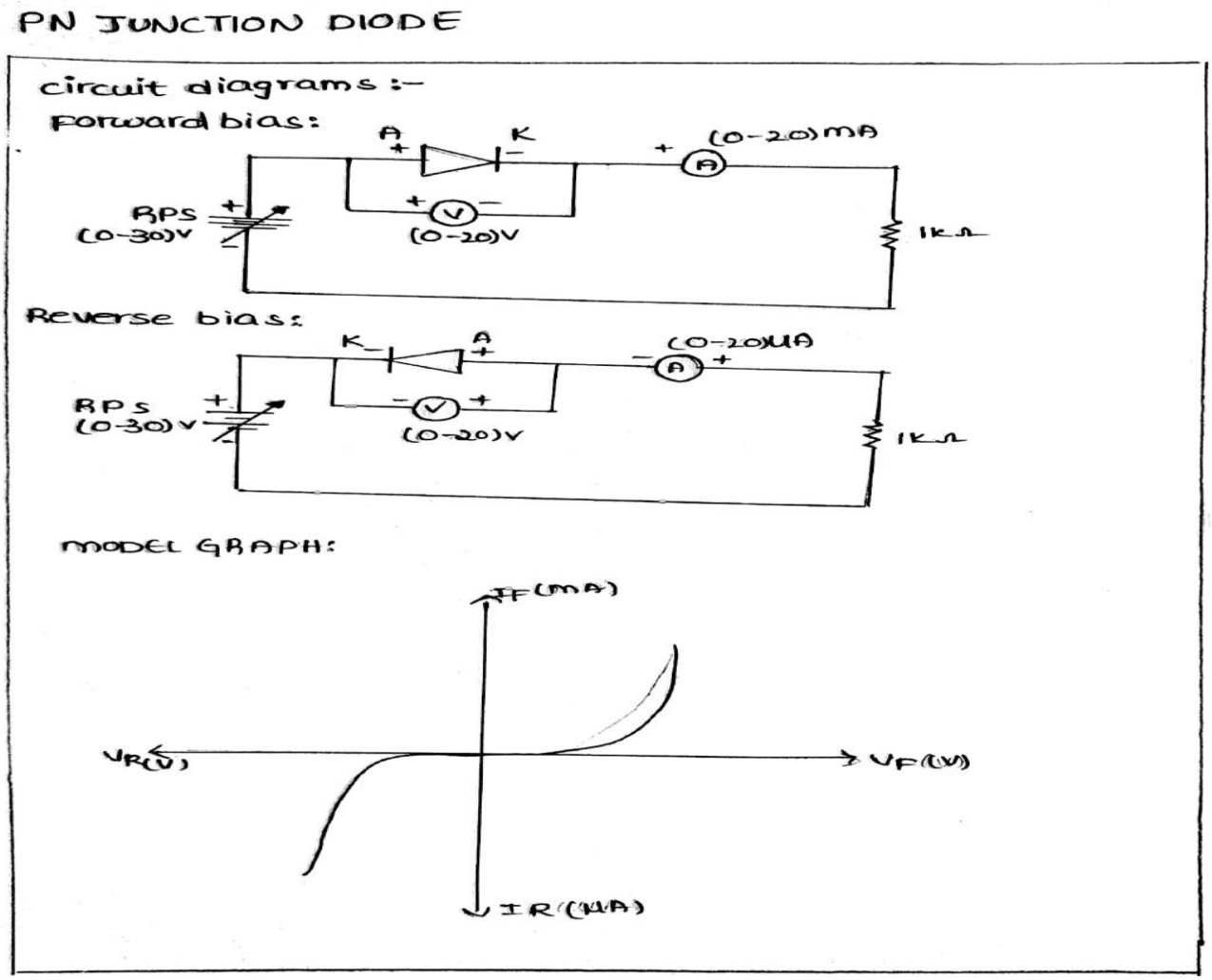
.plot tran v(2) v(1) I(3) I(1)

.plot tran V(1) I(1)

.probe

.end

Circuit Diagram & Model graphs:-



Result:- Obtained the V-I characteristics of silicon and Germanium diodes using PSPICE software and drawn the graphs.

Viva-Voce Questions:-

1) What is meant by PN junction diode?
2) Define Forward bias and reverse bias of diode?
3) What is meant by cut-in voltage ?
4) What is the cut-in voltages for Ge and Si diodes?
5) Draw the equivalent circuit of diode in forward bias and reverse bias conditions?

PGM:02	Design a Zener diode voltage regulator.

AIM :-To Design a Zener diode voltage regulator using PSPICE software.

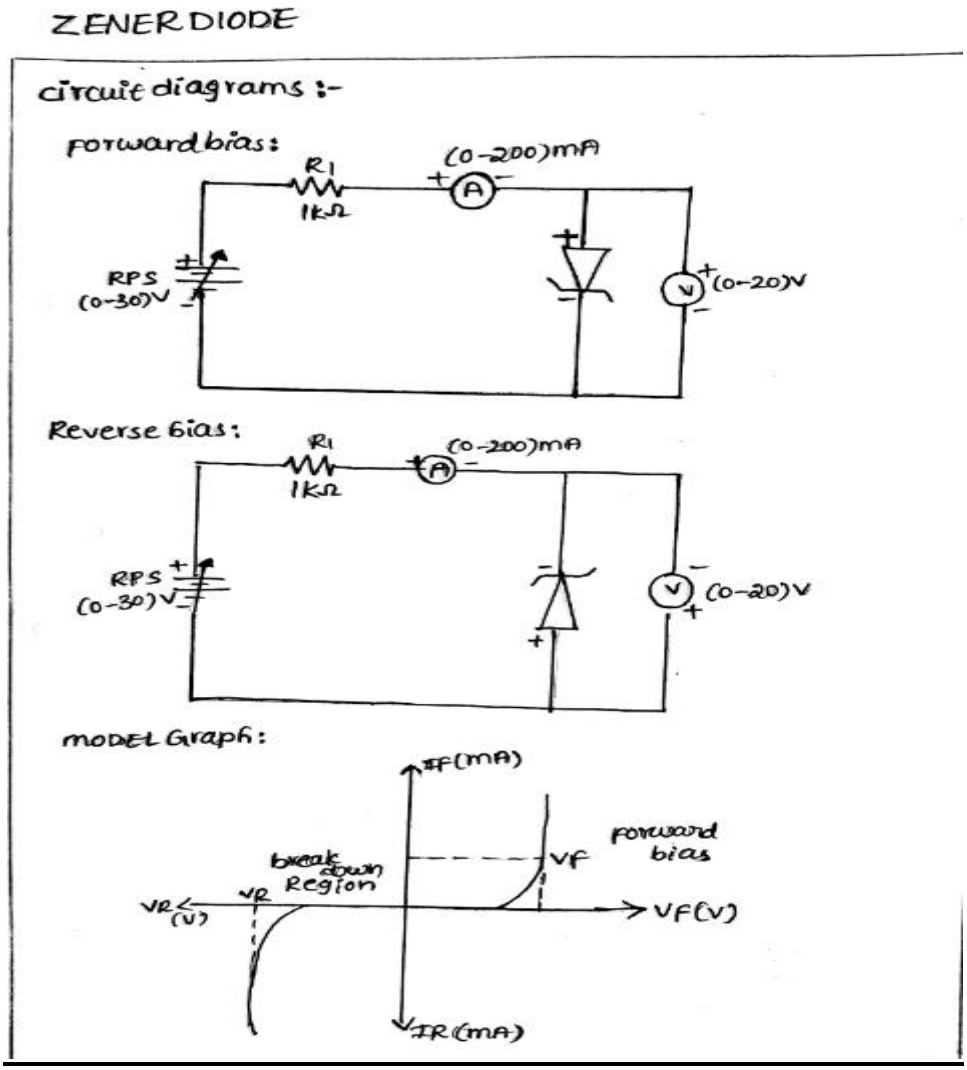
Apparatus:-PC, PSPICE A/D software

Program:-

```
*SRC=UDZ5V1B; UDZ5V1B; Diodes; Zener <=10V; 5.10V 0.200W   DIODES INC Zener
*SYM=HZEN
.SUBCKT UDZ5V1B 1 2
*       Terminals    A   K
D1 1 2 DF
DZ 3 1 DR
VZ 2 3 4.20
.MODEL DF D ( IS=16.2p RS=25.3 N=1.10
+ CJO=28.0p VJ=0.750 M=0.330 TT=50.1n )
.MODEL DR D ( IS=3.23f RS=1.89 N=1.22 )
.ENDS


*SRC=UDZ5V1BF; UDZ5V1BF; Diodes; Zener <=10V; 5.09V  0.500W   DIODES Zener
*SYM=HZEN
.SUBCKT UDZ5V1BF 1 2
*       Terminals    A   K
D1 1 2 DF
DZ 3 1 DR
VZ 2 3 2.66
.MODEL DF D ( IS=40.5p RS=1.92 N=1.10
+ CJO=30.0p VJ=0.750 M=0.330 TT=50.1n)
.MODEL DR D (IS=8.09
```

Circuit Diagram & Expected Model graphs:-



Result:- Designed aZener diode voltage regulator using PSPICE software.

Viva-Voce Questions:-

1) What is differences between PN diode and zenor diode?
2) What is the basic principle of operation of zenor diode?
3) What are the applications of zenor diode?

PGM:03	Design and verify the operating point for a self-bias circuit.

AIM :-To Design and verify the operating point for a self-bias circuitusing PSPICE software.

Apparatus:-PC, PSPICE A/D software

Program:-

Vbb 1 0 1.58v

R1 1 2 4.7k

R2 1 0 2.2k

Vcc 3 0 dc 10v

Rc 3 4 1k

Q1 4 1 5 bc108

Re 5 0 220

.model q1 bc108 npn

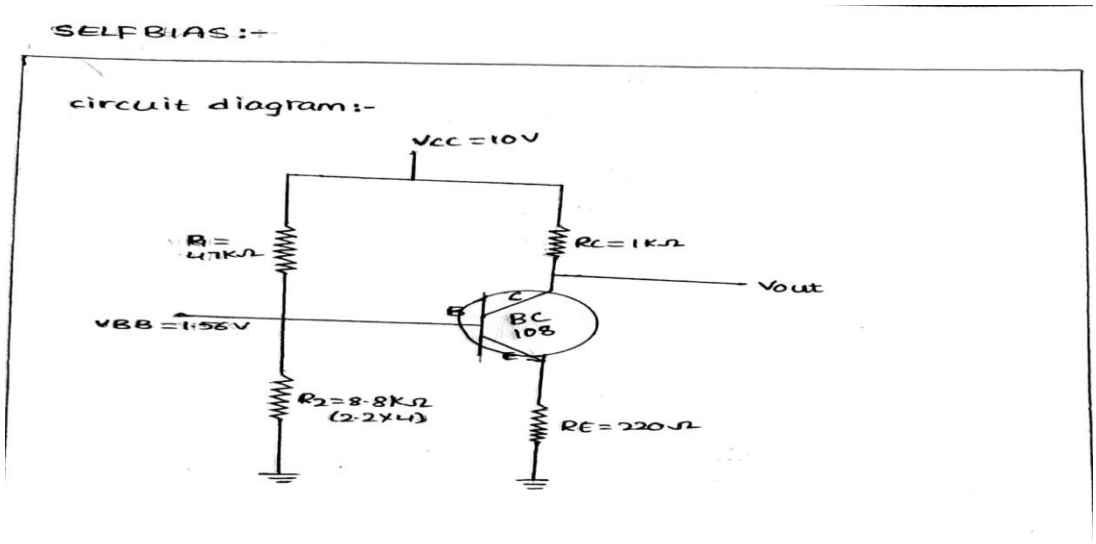
.tran 0.1ms 1ms

.plot tran v(4)

.probe

.end

Circuit Diagram& Expected Model graphs:-



Result:- Designed and verified the operating point for a self-bias circuit using PSPICE software.

Viva-Voce Questions:-

1) What is meant by biasing?
2) What are the different types of transistor biasing models?
3) What is meant by self biasing?
4) What is the stability factor for self bias?

PGM:04	Study the characteristics of a half wave and full wave rectifier.

AIM :-To Study the characteristics of a half wave and full wave rectifier using PSPICE software.

Apparatus:-PC, PSPICE A/D software

Program:-

(i)

***HalfWave Rectifier**

```
vin 1 0 sin(0 230v 50hz)
rs 1 2 10
l1 2 0 2000mh
l2 3 0 20uh
k1 l1 l2 0.9999
d1 3 4 da
rl 4 0 500k
.model da d
.tran 0.1ms 100ms
.plot tran v(1) v(4)
.probe
.end
```

(ii)

***FullWave Rectifier**

```
vin 1 0 sin(0 230v 50hz)
r1 5 0 1000
rs 1 2 10
l1 2 0 2000uh
l2 3 0 10uh
```

l3 0 4 10uh

k1 l1 l2 l3 0.9999

d1 3 5 da

d2 4 5 da

.model da d

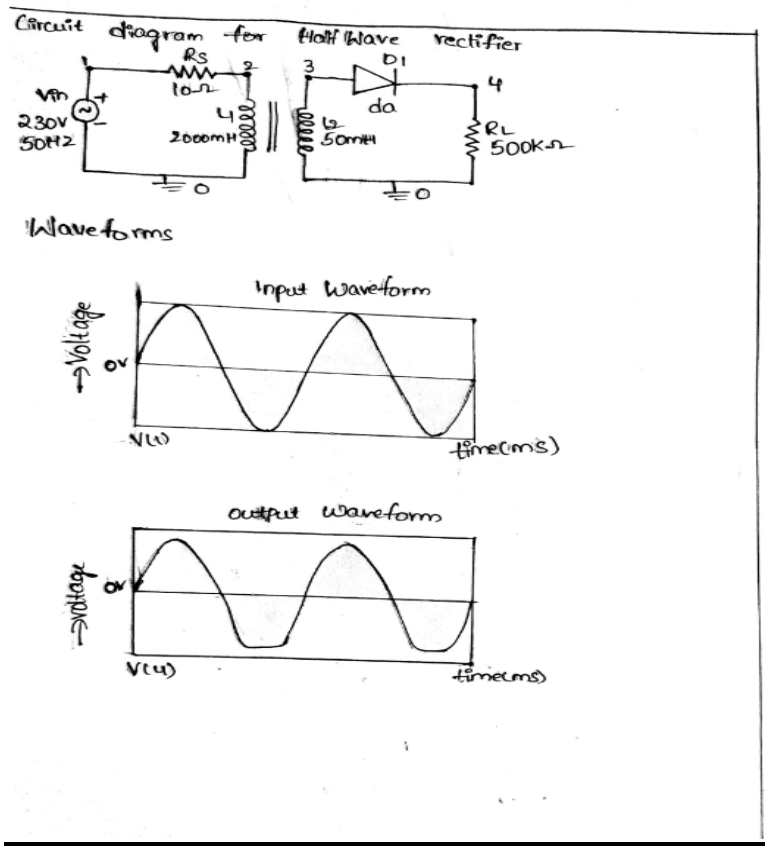
.tran 0.01ms 50ms

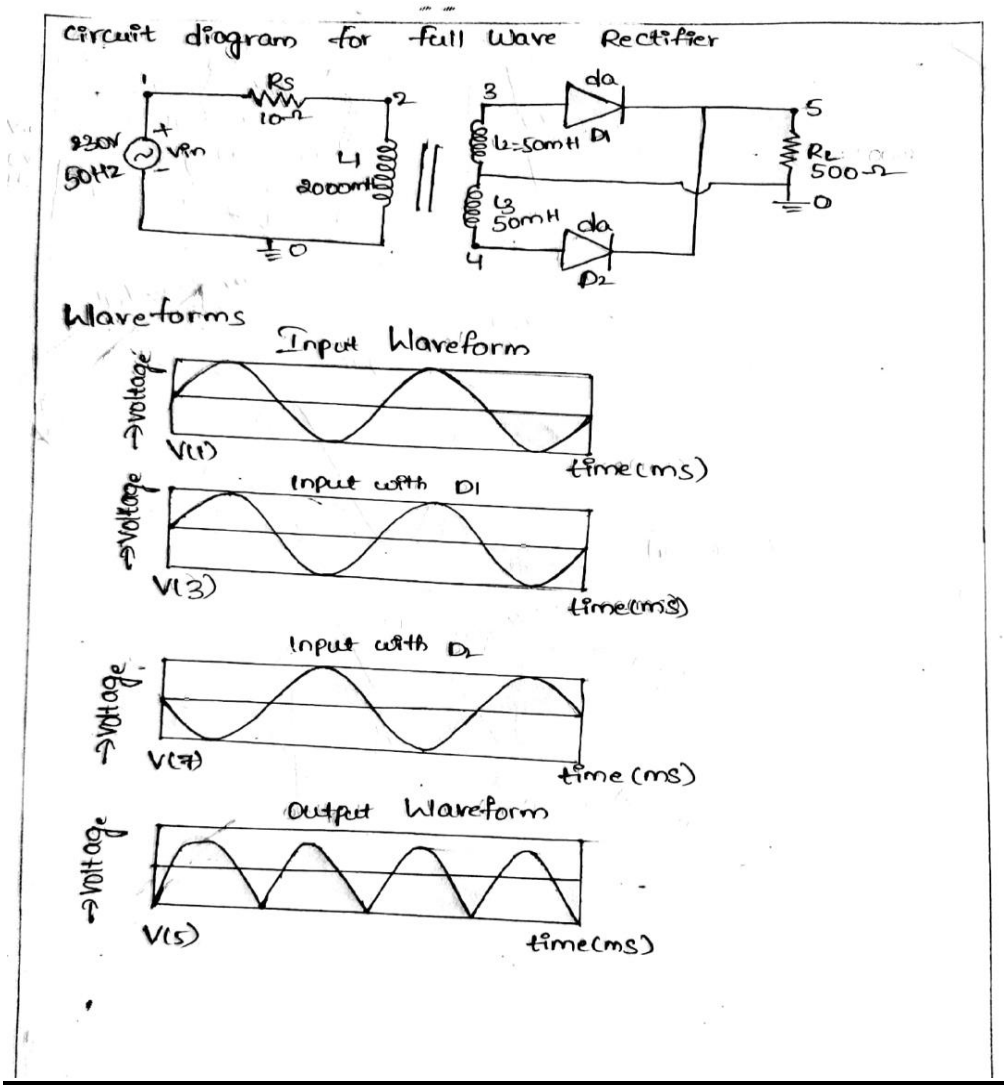
.plot tranv(1) v(3) v(4) v(5)

.probe

.end

Circuit Diagram & Expected Model graphs:-





Result:-Studied the characteristics of a half wave and full wave rectifier using PSPICE Software and plotted the graphs.

Viva-Voce Questions:-

1)What is meant by rectifier?

2) What are the advantages of FWR over HWR?

3) What is the basic element used in rectification?

4) List any differences between FWR and HWR?

PGM:05	Study the characteristics of a bridge rectifier.

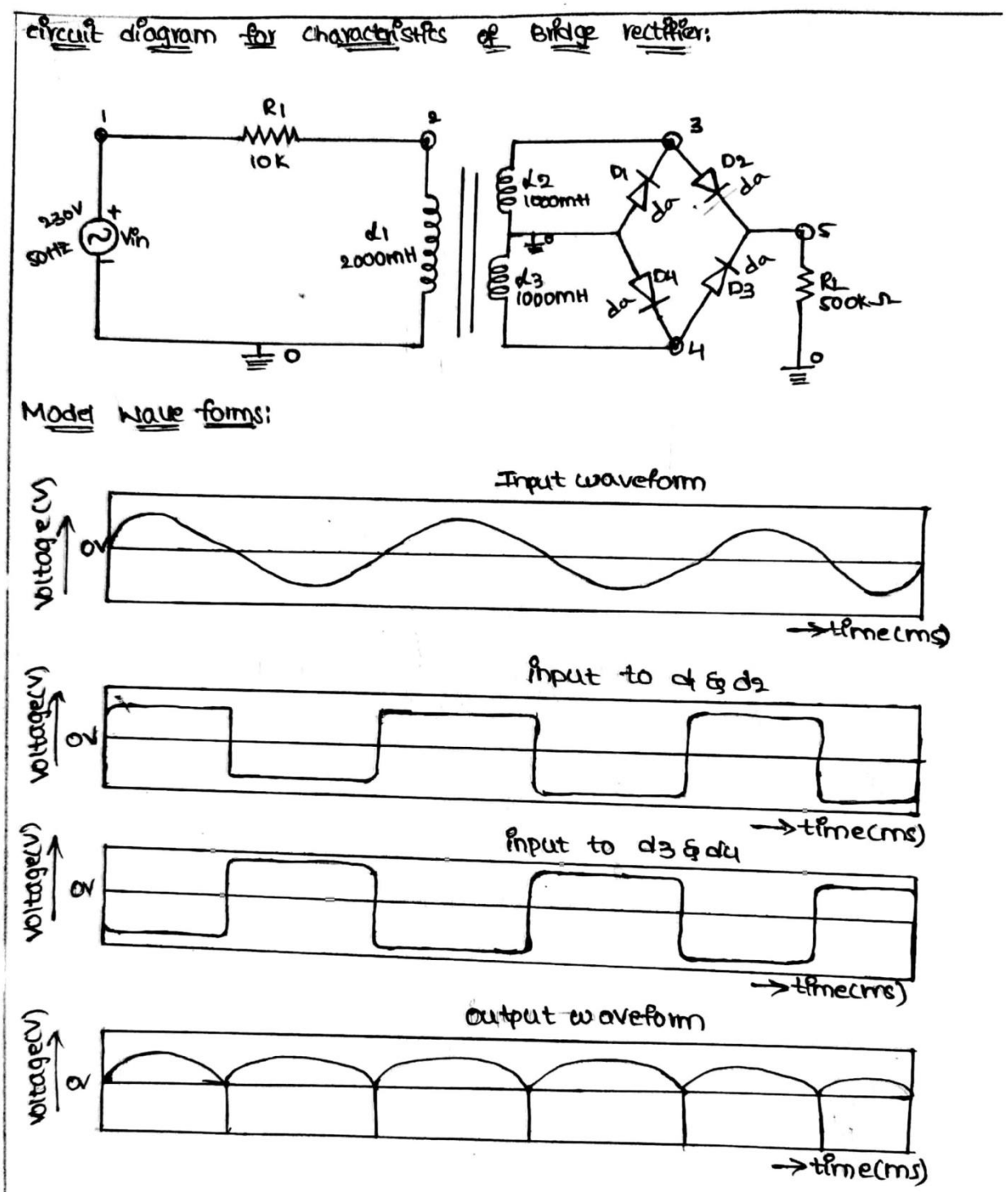
AIM :-To Study the characteristics of a half wave and full wave rectifier using PSPICE software.

Apparatus:-PC, PSPICE A/D software

Program:-

```
*bridge rectifier
vin 1 0 sin(0 230v 50hz)
r1 1 2 10k
l1 2 0 2000mh
l2 3 4 20mh
k1 l1 l2 0.9999
d1 0 3 mod1
d2 3 5 mod1
d4 0 4 mod1
d3 4 5 mod1
rl 5 0 500k
.model mod1 d
.tran 0.1ms 50ms
.plot tranv(1) v(5)
.probe
.end
```

Circuit Diagram & Expected Model graphs:-



Result:-Studied the characteristics of a half wave and full wave rectifier using PSPICE Software and graphs are plotted.

Viva-Voce Questions:-

1)What is the differences between bridge rectifier ,FWR and HWR?

2) Which is most widely used rectifier?

3) List the applications of Bridge rectifier.

PGM:06	Obtain the frequency response of a CE amplifier.

AIM :-To Obtain the frequency response of a CE amplifierusing PSPICE software.

Apparatus:-PC, PSPICE A/D software

Program:-

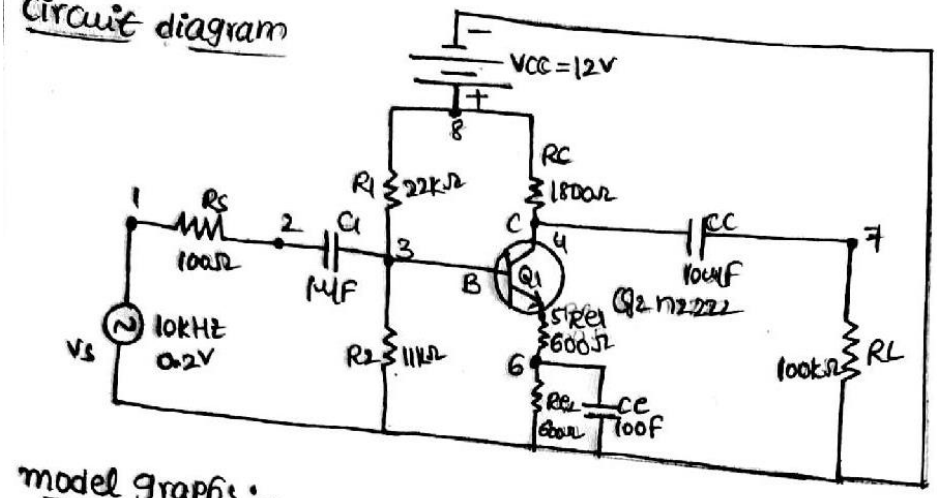
***frequency response of ce amplifier**

```
vs 1 0 ac 1 sin(0 0.2v 10khz)
vcc 8 0 dc 12v
rs 1 2 100
c1 2 3 1uf
r2 3 0 11k
r1 3 8 22k
q1 4 3 5 q2n2222
rc 8 4 1800
re1 5 6 600
re2 6 0 600
ce 6 0 10uf
cc 4 7 10uf
rl 7 0 100k
.model q2n2222 npn
.ac dec 5 10 10meg
.tran 0.001ms 0.2ms
.plot ac v(7)
.plot tranv(2) v(5) v(7)
.probe
.end
```

Circuit Diagram & Expected Model graphs:-

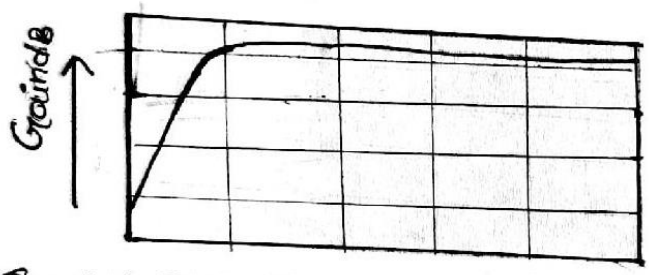
FREQUENCY RESPONSE OF CE AMPLIFIER

Circuit diagram



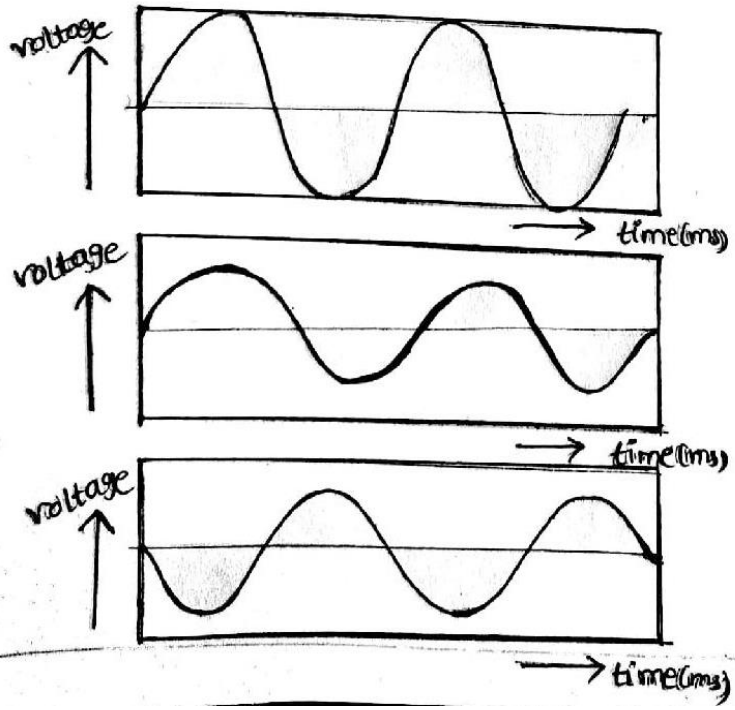
model graphs :-

Frequency response :-



Transient Response :-

waveforms :-



Result:-Obtained the frequency response of a CE amplifierusing PSPICE

Software and graphs are plotted and found out the bandwidth of CE amplifier.

Viva-Voce Questions:-

1) What is the phase shift given by CE amplifier?
2) What are the applications of CE amplifier?
3) Specify the characteristics of CE amplifier?
4) What is the frequency range used in this program.

PGM:07	Obtain the frequency response of a two stage RC couple CE amplifier.

AIM :-To Obtain the frequency response of a two stage RC couple CE amplifierusing PSPICE software.

Apparatus:-PC, PSPICE A/D software

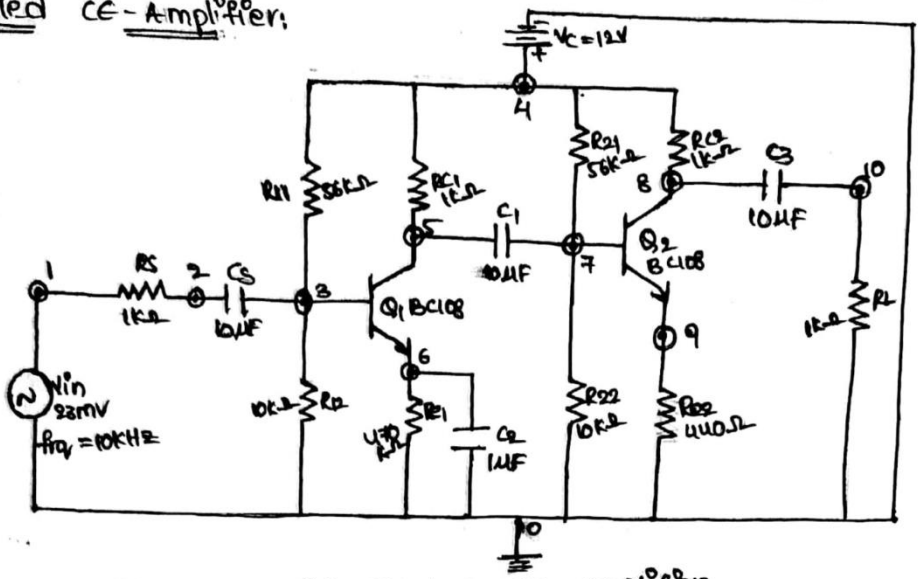
Program:-

***twostagerccoupledce amplifier**

```
vin 1 0 ac 1 sin(0 23mv 100hz)
rs 1 2 1k
cs 2 3 10uf
r11 3 4 56k
r12 3 0 10k
q1 5 3 6 bc108
rc1 5 4 1k
re1 6 0 470
c1 5 7 10uf
r21 7 4 56k
r22 7 0 10k
q2 8 7 9 bc108
rc2 8 4 1k
re2 9 0 440
c2 6 0 1uf
c3 8 10 10uf
rl 10 0 1k
vcc 4 0 12v
.model bc108 npn
.ac dec 5 10 10meg
.plot ac v(10)
.probe
.end
```

Circuit Diagram & Expected Model graphs:-

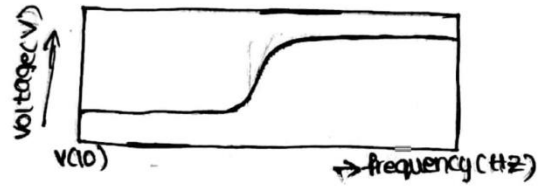
Circuit diagram for frequency response of a two stage RC coupled CE-Amplifier:



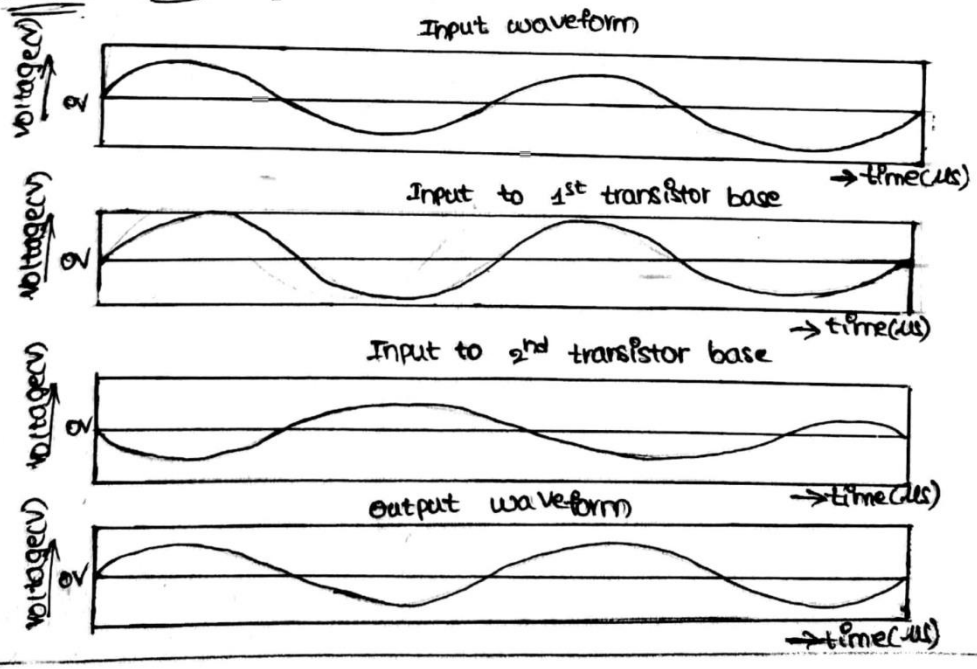
Two stage RC coupled CE-amplifier.

Model Graphs:

AC analysis:



Transpent Response:



Result:-Obtained the frequency response of a two stage CE amplifierusing PSPICE Software and graphs are plotted and found out the bandwidth of two stageCE amplifier.

Viva-Voce Questions:-

1)What is the differences between single stage and two stage CE amplifiers?

2) What is the phase shift offered by two stage CE amplifier?

3) What is the R_i and R_O of 2-stage CE amplifier ?

4) What is A_V and $A_I$of 2-stage CE amplifier ?

PGM:08	Design and simulate class A power Amplifier.

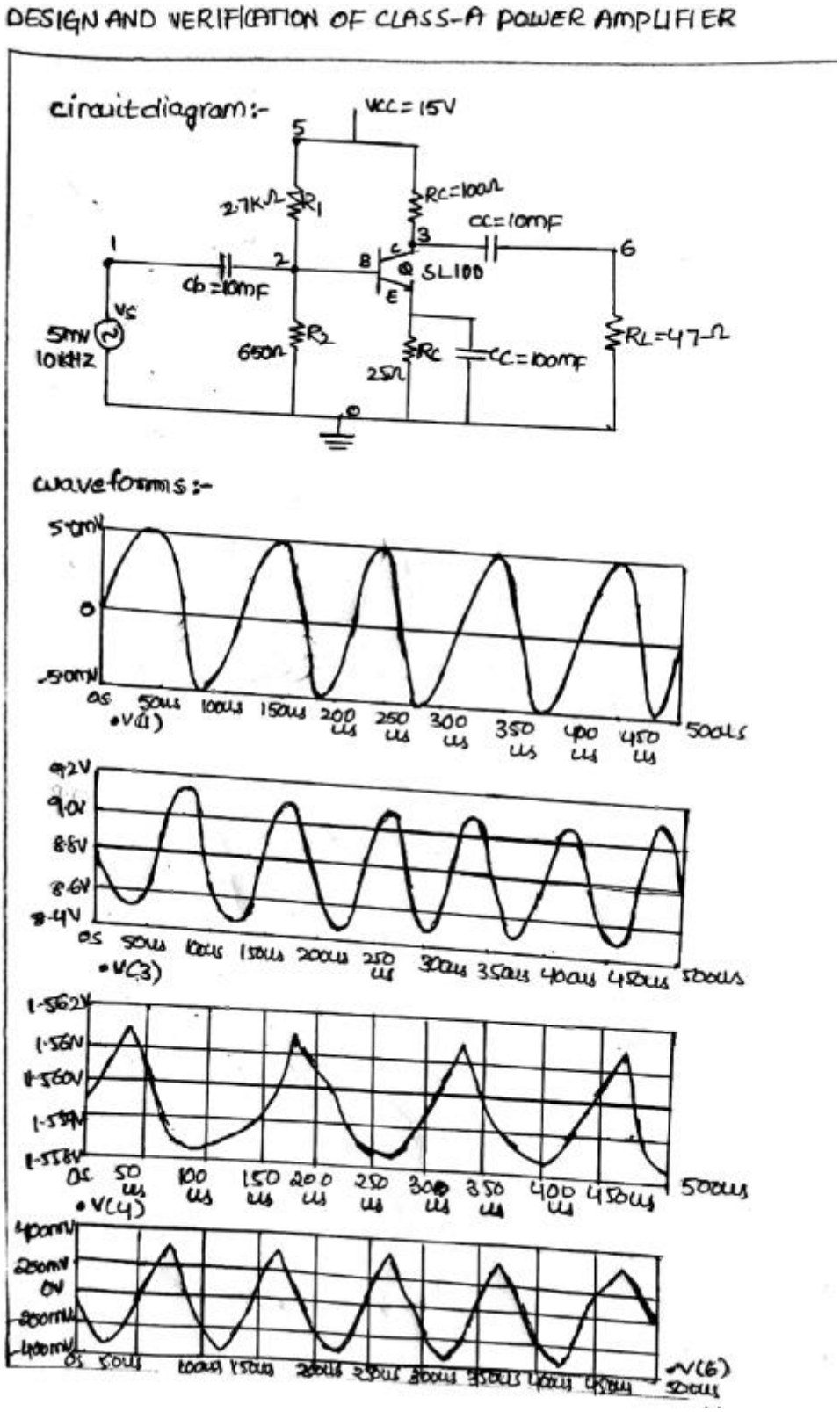
AIM :-To Design and simulate class A power Amplifierusing PSPICE software.

Apparatus:-PC, PSPICE A/D software

Program:-

```
*classa power amplifier
vs 1 0 sin(0 5mv 10khz)
vcc 5 0 15v
cb 1 2 10uf
r1 2 5 2.7k
r2 2 0 605
q1 3 2 4 sl100
rc 5 3 100
re 4 0 25
rl 6 0 47
ce 4 0 100uf
cc 3 6 10uf
.model sl100 npn
.tran 0.1ms 0.5ms
.plot tranv(1) v(3) v(4) v(6)
.probe
.end
```

Circuit Diagram & Expected Model graphs:-



Result:-Designed and simulated class A power Amplifierusing PSPICE Software and graphs are plotted.

Viva-Voce Questions:-

1) What is meant by Amplifier?
2) What is the differences between amplifier and power amplifier?
3) List the classification of power amplifiers?
4) What is the phase shift offered by class-A power amplifier?
5) Specify the applications of class-A power amplifier?

\

PGM:09	Simulate a differentiator and integrator using OPAMP.

AIM :-To simulate a differentiator and integrator using OPAMPusing PSPICE software.

Apparatus:-PC, PSPICE A/D software

Program:-

***opamp as differentiator**

```
vin 1 0 sin(0 2v 100hz)
r1 1 2 10
c1 2 3 0.1uf
rf 3 5 1.5k
cf 3 5 0.005uf
rom 4 0 1.5k
e 5 0 4 3 2e5
ri 4 3 1e6
rl 5 0 10k
.tran 0.1ms 100ms
.plot tranv(1) v(5)
.probe
.end
```

***opamp as intergrator**

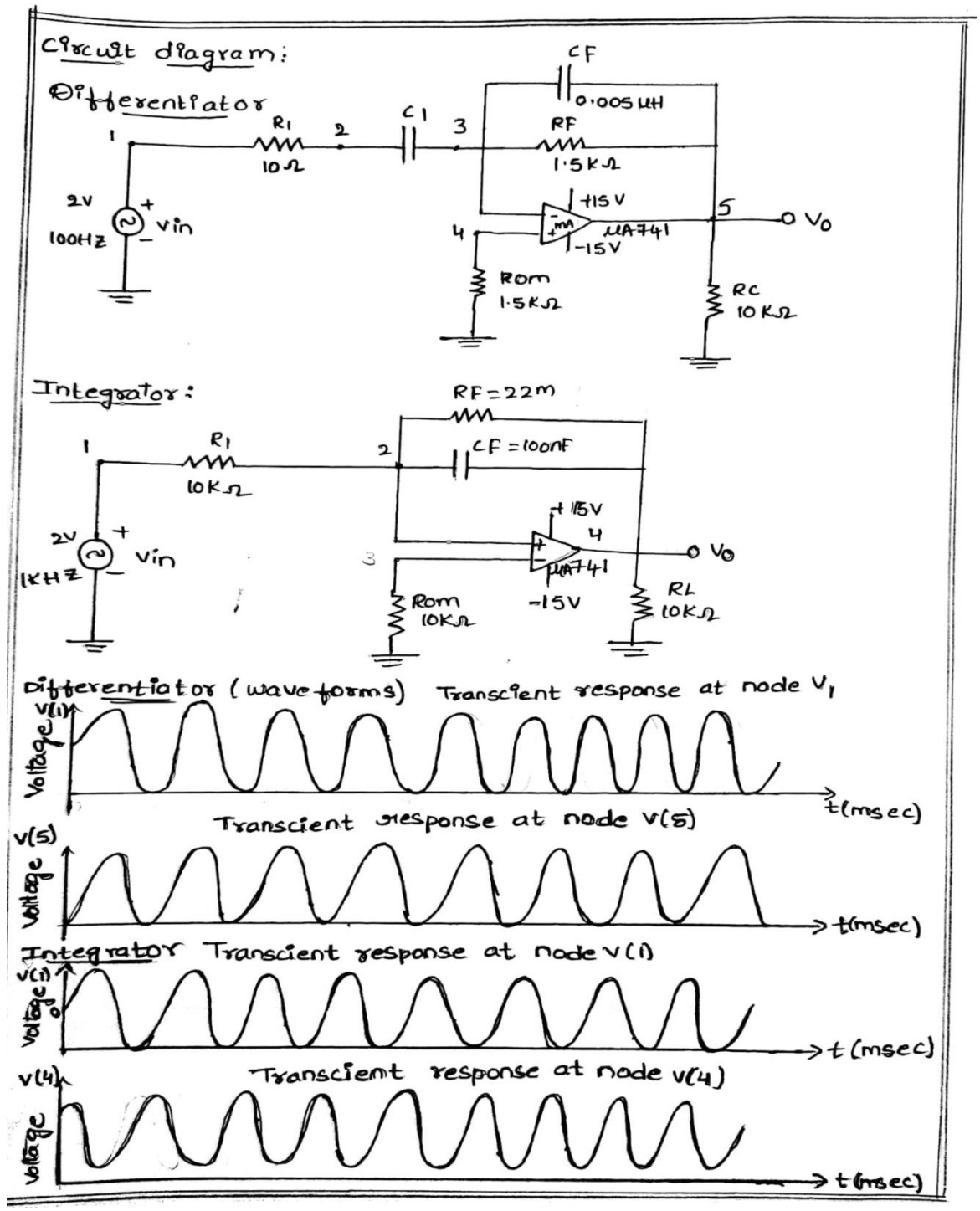
```
vin 1 0 sin(0 2 100)
r1 1 2 10k
cf 2 4 100nf
rom 3 0 10k
rf 2 4 22meg
e 4 0 3 2 2e5
ri 3 2 1e6
rl 4 0 10k
.tran 0.1ms 100ms
```

.plot tranv(1) v(4)

.probe

.end

Circuit Diagram & Expected Model graphs:-



Result:-Simulateda differentiator and integrator using OPAMPusing PSPICE Software and graphs are plotted.

Viva-Voce Questions:-

1) What is the condition for high pass circuit acts as differentiator?
2) What is the condition for lowpass circuit acts as integrator?
3) What is the input impedance of Op-Amp?
4) What is the bandwidth of Op-Amp?
5) List the applications of differentiator and integrator.

PGM:10	Simulate a low pass and high pass filter using OPAMP.

AIM :-ToSimulate a low pass and high pass filter using OPAMPusing PSPICE software.

Apparatus:-PC, PSPICE A/D software

Program:-

***opamp as lowpass filter**

```
vin 1 0 ac 1 sin(0 20 100)
r 1 2 15.9k
c1 2 0 0.01uf
r2 0 3 10k
rf 3 4 10k
rl 4 0 10k
e 4 0 3 2 2e5
ri 2 3 1e6
.ac dec 5 10 10meg
.plot ac v(4)
.probe
.end
```

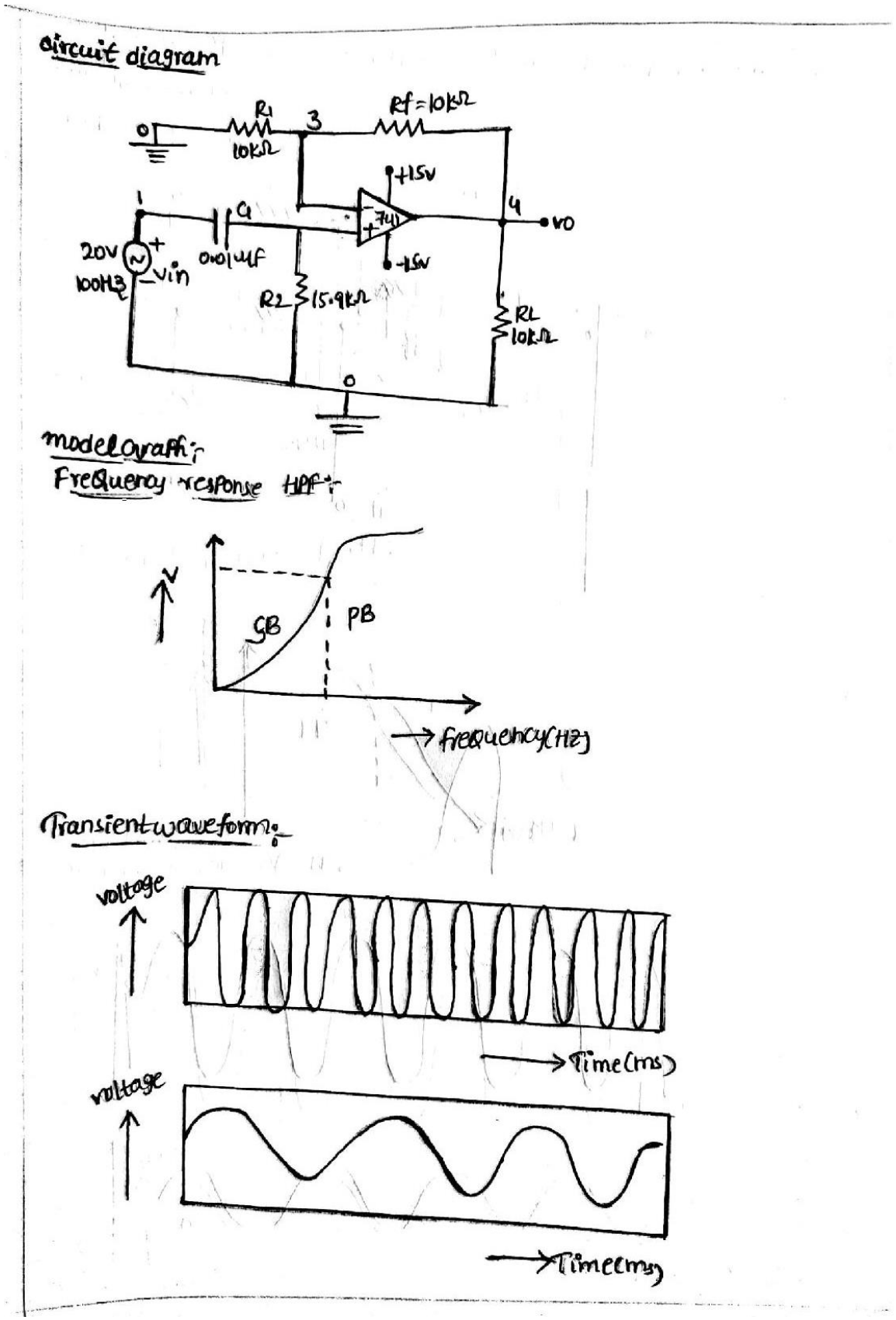
***opamphighpass filter**

```
vin 1 0 ac 1 sin(0 20 100)
c 1 2 0.01uf
r2 0 2 15.9k
r1 0 3 10k
rf 3 4 10k
rl 4 0 10k
e 4 0 3 2 2e5
ri 2 3 1e6
.ac dec 5 10 10meg
.plot ac v(4)
```

.probe

.end

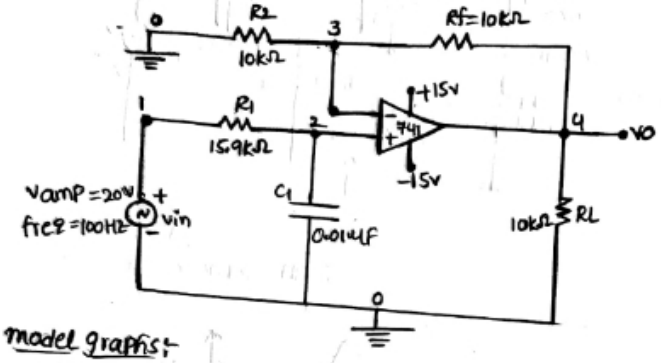
Circuit Diagram & Expected Model graphs:-



Simulation of Low pass filter and High pass filter using op-amp

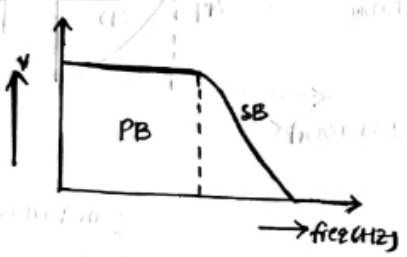
Circuit diagram

LPF using OP-AMP :-



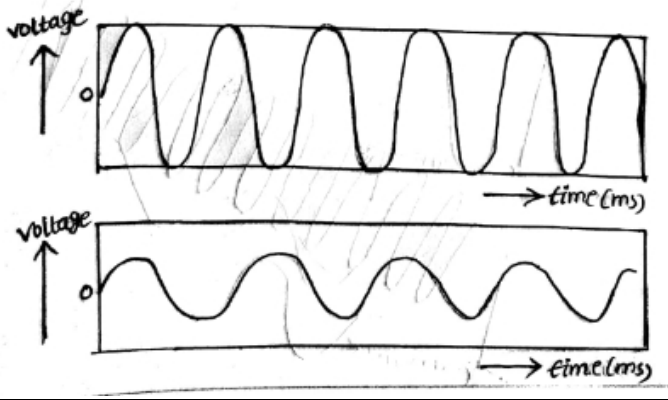
model graphs:-

frequency response of LPF:-



waveforms:-

Transient Response of LPF:-



Result:-Simulated a Low pass and High pass filter using OPAMPusing PSPICE Software and plotted the frequency response graph and indicated the pass band and stop band.

Viva-Voce Questions:-

1) What is meant by lowpass filter?
2) What is meant by high pass filter?
3) What is the relation shipbetween capacitive reactance and frequency?
4) What are the applications of lowpass and high pas filter?
5) What are the different bands used in frequency response of lowpass and highpass filters?

PGM:11	Simulate a RC phase shift and Wein bridge oscillator using OPAMP.

AIM :-ToSimulate a RC phase shift and Wein bridge oscillator using OPAMPusing PSPICE software.

Apparatus:-PC, PSPICE A/D software

Program:-

***rcphaseshift oscillator**

r 6 2 33k

rf 2 3 1meg

rom 1 0 33k

r1 6 0 3.3k

r2 5 0 3.3k

r3 4 0 3.3k

c1 6 5 0.1uf ic=2

c2 5 4 0.1uf ic=2

c3 4 3 0.1uf ic=2

e 3 0 1 2 2e5

ri 1 2 1e6

.tran 0.1ms 100ms uic

.plot tranv(6) v(3)

.probe

.end

***weinbridge oscillator**

R 0 1 10k

rf 1 4 20k

r 1 3 6366

c 13 4 1nfic=2

r2 2 0 6366

c2 2 0 1nfic=2

e 4 0 2 1 2e5

ri 2 1 1e6

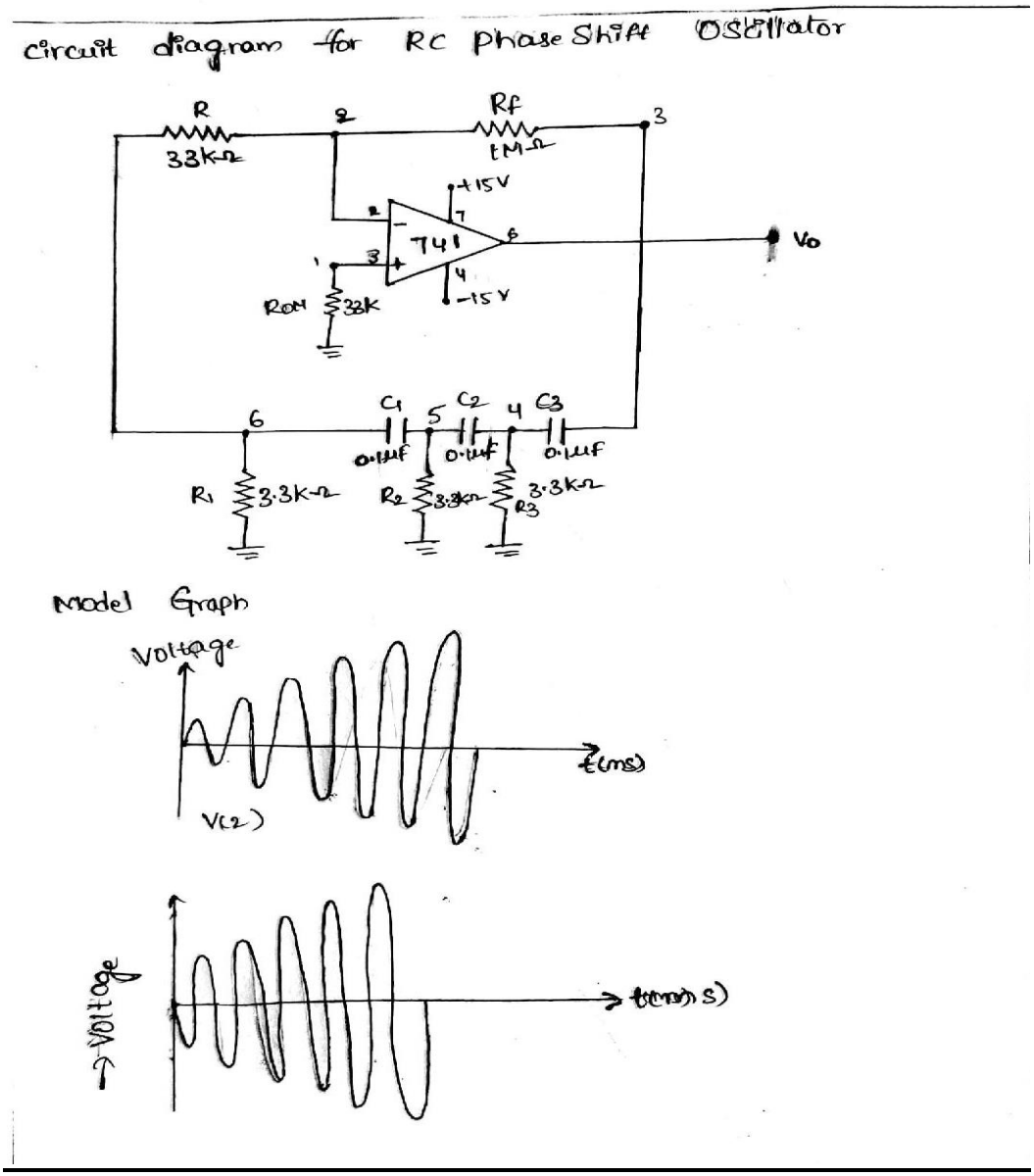
.tran 0.1us 100us uic

.plot tranv(2) v(4)

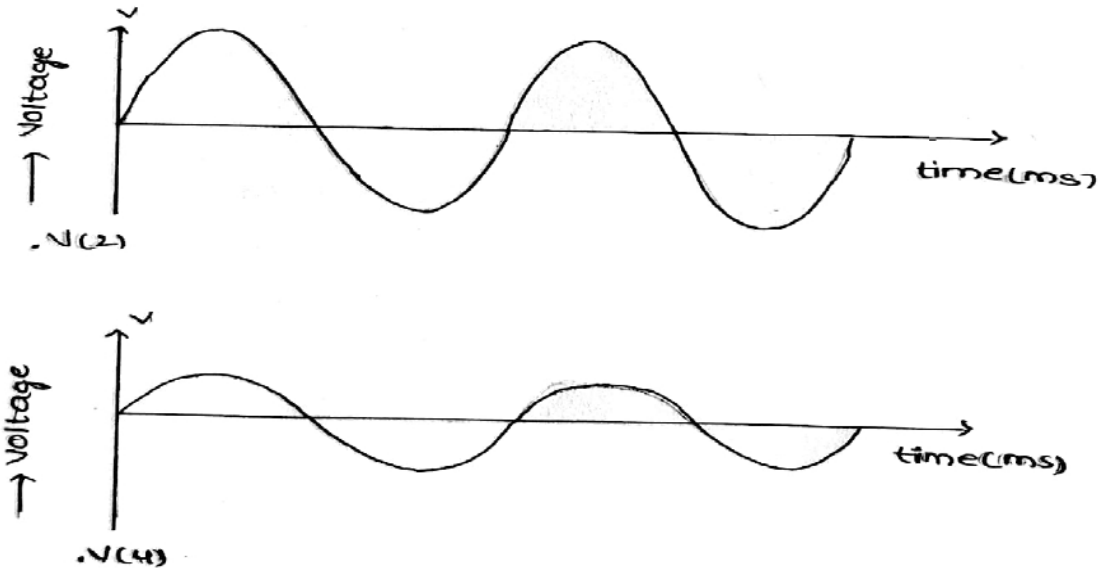
.probe

.end

Circuit Diagram & Expected Model graphs:-

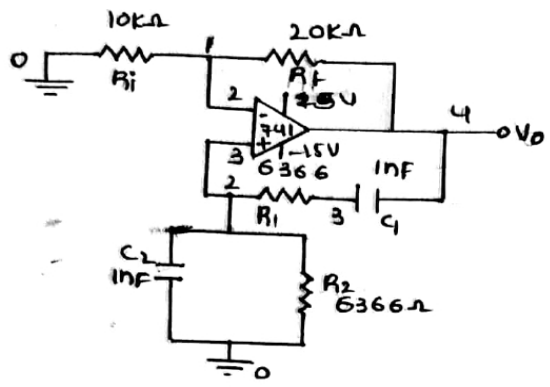


Wave forms:

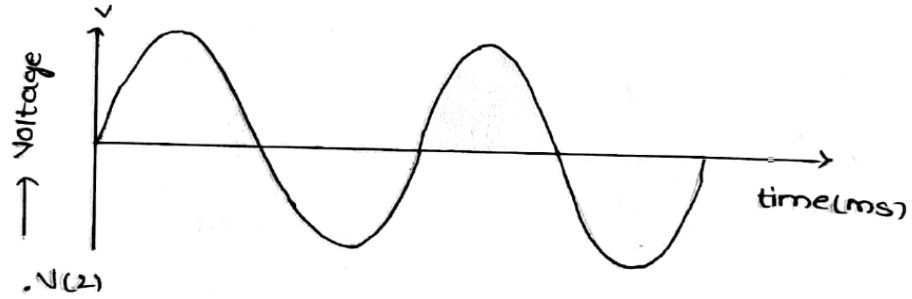


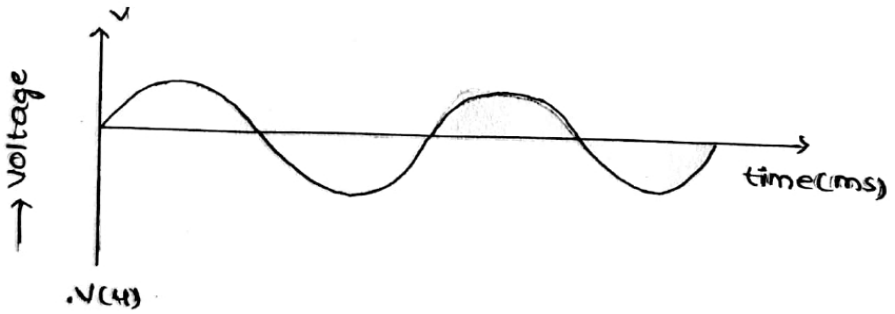
Wein bridge oscillator

circuit diagram:



Wave forms:





Result:-Simulated a RC phase shift and Wein bridge oscillatorusing OPAMPusing PSPICE Software and plotted the frequency response graph and verified the practical and theoretical frequency response of oscillations.

Viva-Voce Questions:-

1) What is meant by bark hausen criterion for frequency of oscillations?
2) What is the feed back gain of RC phase shift and Wein bridge oscillator?
3) Which type of feedback is used in oscillators and amplifiers?
4) What is the phase shift offered by RC phase shift wein bridge oscillators

PGM:12	Design and simulate a constant resistance and bridged T equalizer.

AIM :-ToDesign and simulate a constant resistance and bridged T equalizerusing PSPICE software.

Apparatus:-PC, PSPICE A/D software

Program:-

***constant k lowpassfilter**

vin 1 0 ac 1 sin(0 2 100)

rs 1 2 680

c1 2 0 0.1uf

l1 2 3 110mh

c2 3 0 0.1uf

r1 3 0 680

.ac dec 5 10 10meg

.plot ac v(3)

.probe

.end

***constant k highpassfilter**

vin 1 0 ac 1 sin(0 2 1khz)

r1 1 2 680

c1 2 3 0.2uf

l 3 0 55mh

c2 3 4 0.2uf

r2 4 0 680

.ac dec 5 10 10meg

.plot ac v(4)

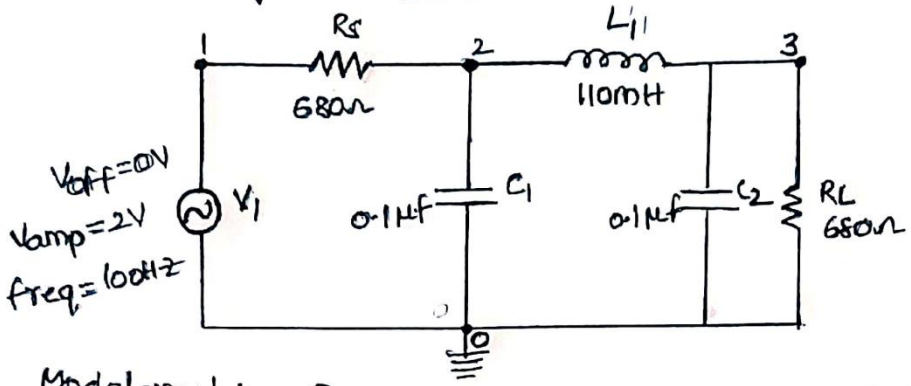
.probe

.end

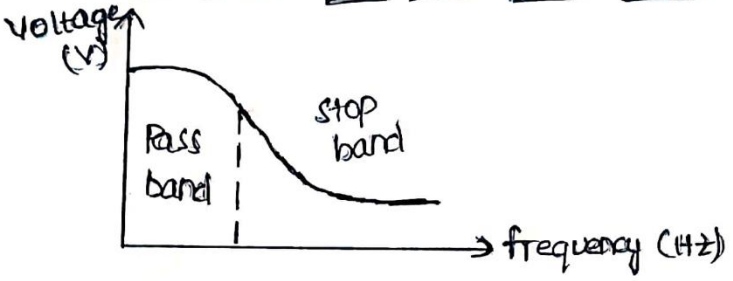
Circuit Diagram & Expected Model graphs:-

Simulation of a Constant k- low Pass filter and High Pass filter

Circuit diagram :- k - Lowpass filter :-



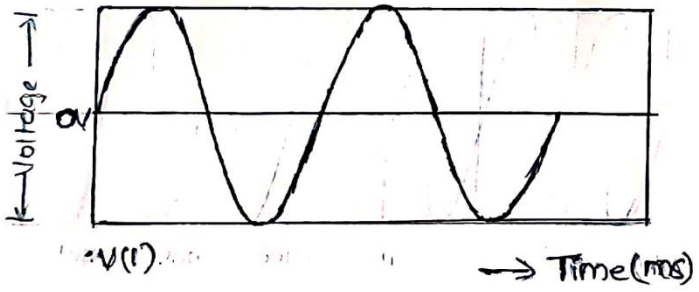
Modelgraph:- Frequency response of constant k - lowpass filter



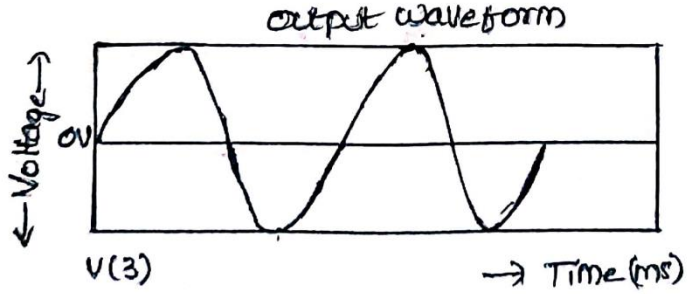
Model Waveforms :-

Transient Response :-

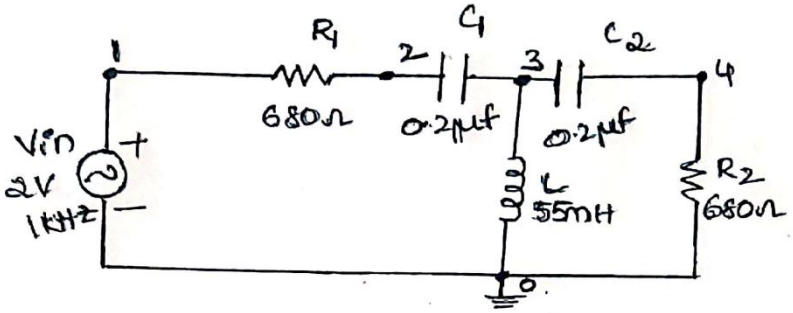
Input waveform



output waveform

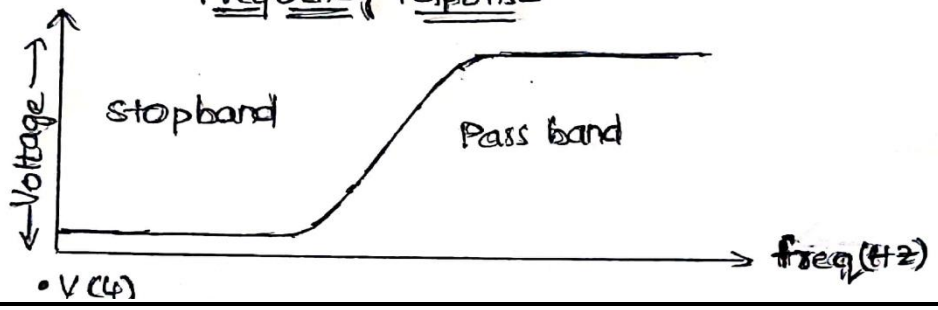


Circuit diagram :- K- High pass filter



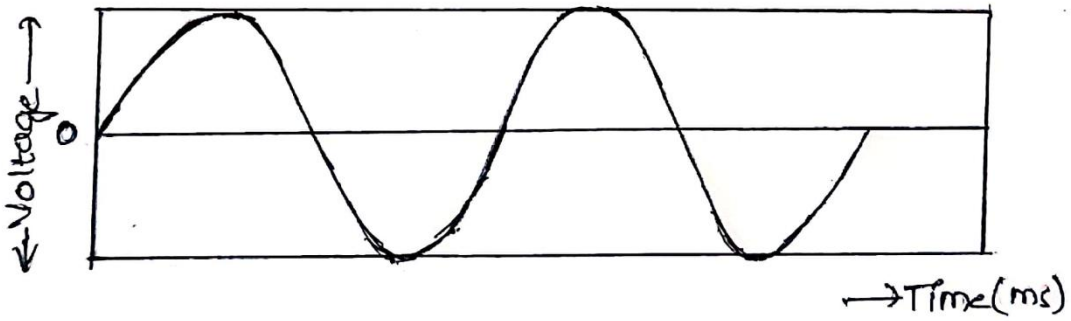
Model graphs :- K- High pass filter

Frequency response

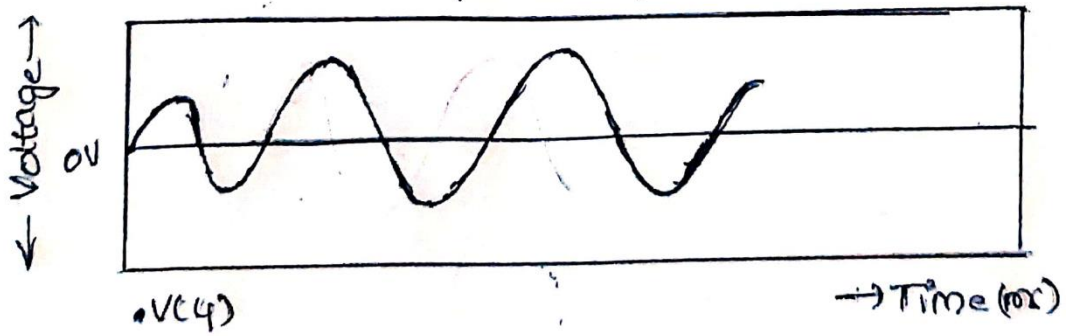


Transient Response

Input Waveform



Output Waveform



Result:-Designed and Simulated a constant k resistance Lowpass and high pass filters using PSPICE Software and plotted the frequency response graphs.

Viva-Voce Questions:-

1) What is meant by equalizer?
2) What is the differences between constant resistance type and T type equalizers?
3) List out the applications of equalizers?
4) What is the differences between attenuators and equalizers?

PGM:13 & 14	Simulate an Amplitude Modulator and Demodulator.

AIM :-To simulate an Amplitude Modulator and Demodulatorusing PSPICE software.

Apparatus:-PC, PSPICE A/D software

Program:-

***amplitude modulation**

vs 1 0 sin(0 10mv 5khz)

vcc 3 0 15v

cb 1 2 0.01uf

r1 2 3 20k

r2 2 0 10k

q1 4 2 5 bc108

rc 3 4 10k

re 5 7 10k

rl 6 0 20k

ce 5 0 0.1uf

cc 4 6 0.01uf

v1 7 0 sin(0 20mv 100khz)

.model bc108 npn

.tran 0.01ms 1ms

.plot tranv(1) v(7) v(6)

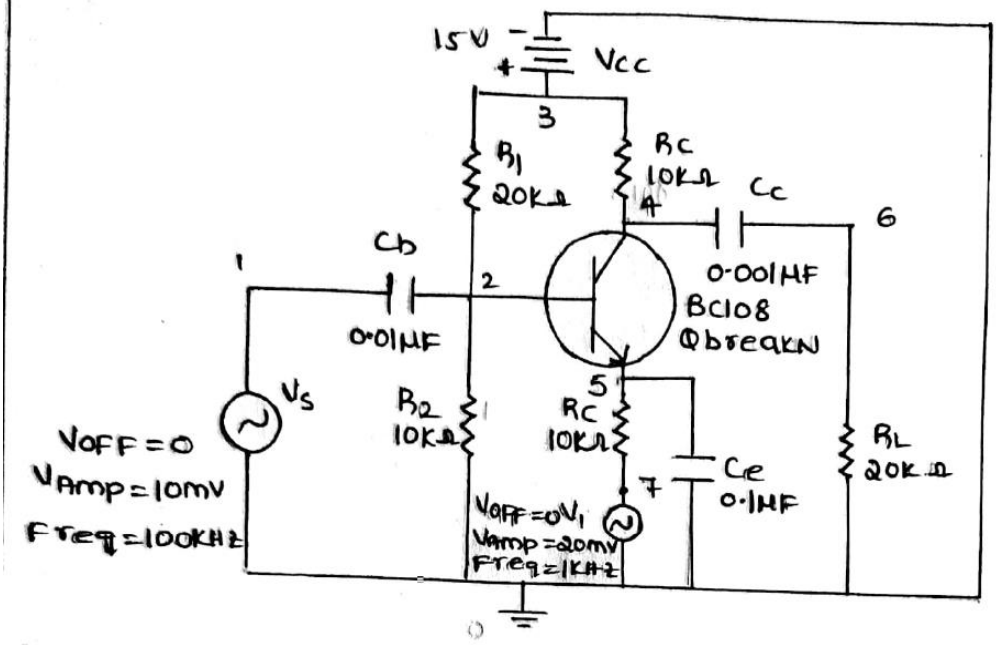
.probe

.end

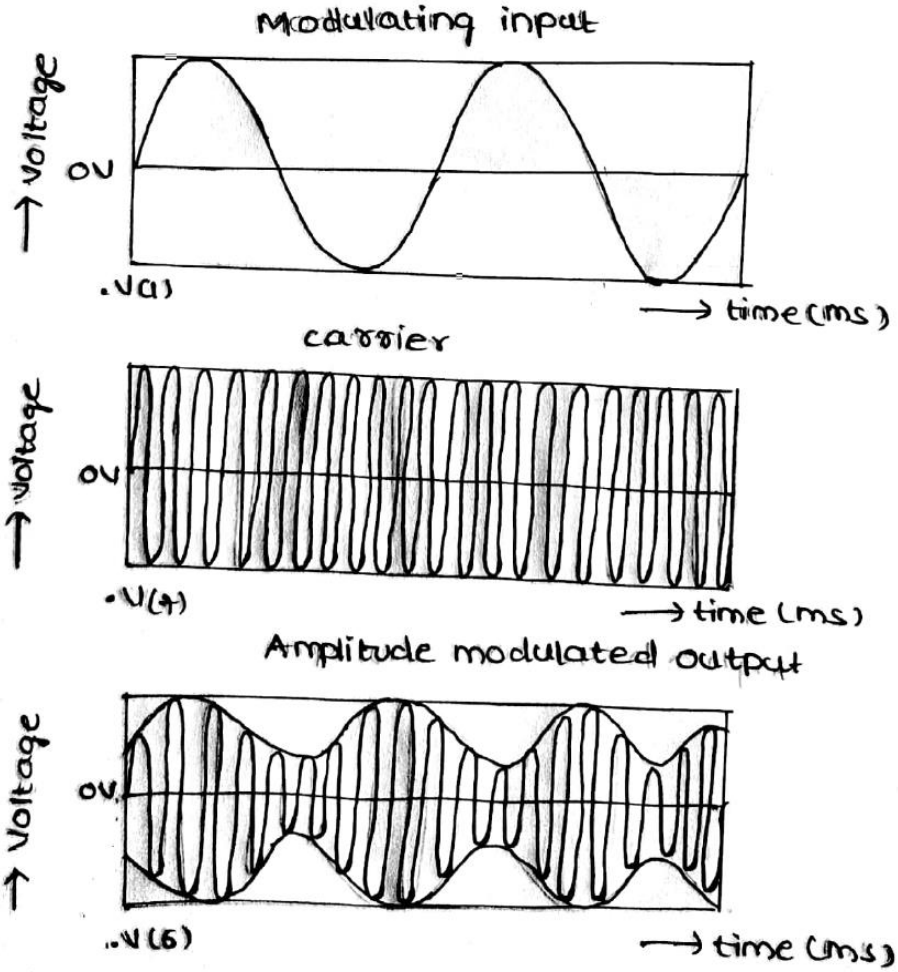
Circuit Diagram & Expected Model graphs:-

Verification of Amplitude Modulator

Circuit diagram:



Wave forms:



Result:-Simulated an Amplitude Modulator and Demodulatorusing PSPICE Software and plotted the transient response graphs.

Viva-Voce Questions:-

1)What is meant by modulation?

2) What are the different types of analog modulation techniques?

3) What is meant by amplitude modulation?

4) what are the applications of amplitude modulation?

5) List the various demodulation techniques of amplitude modulation?

PGM:15	Simulate a Frequency Modulator and Demodulator.

AIM :-To simulate a Frequency Modulator and Demodulatorusing PSPICE software.

Apparatus:-PC, PSPICE A/D software

Program:-

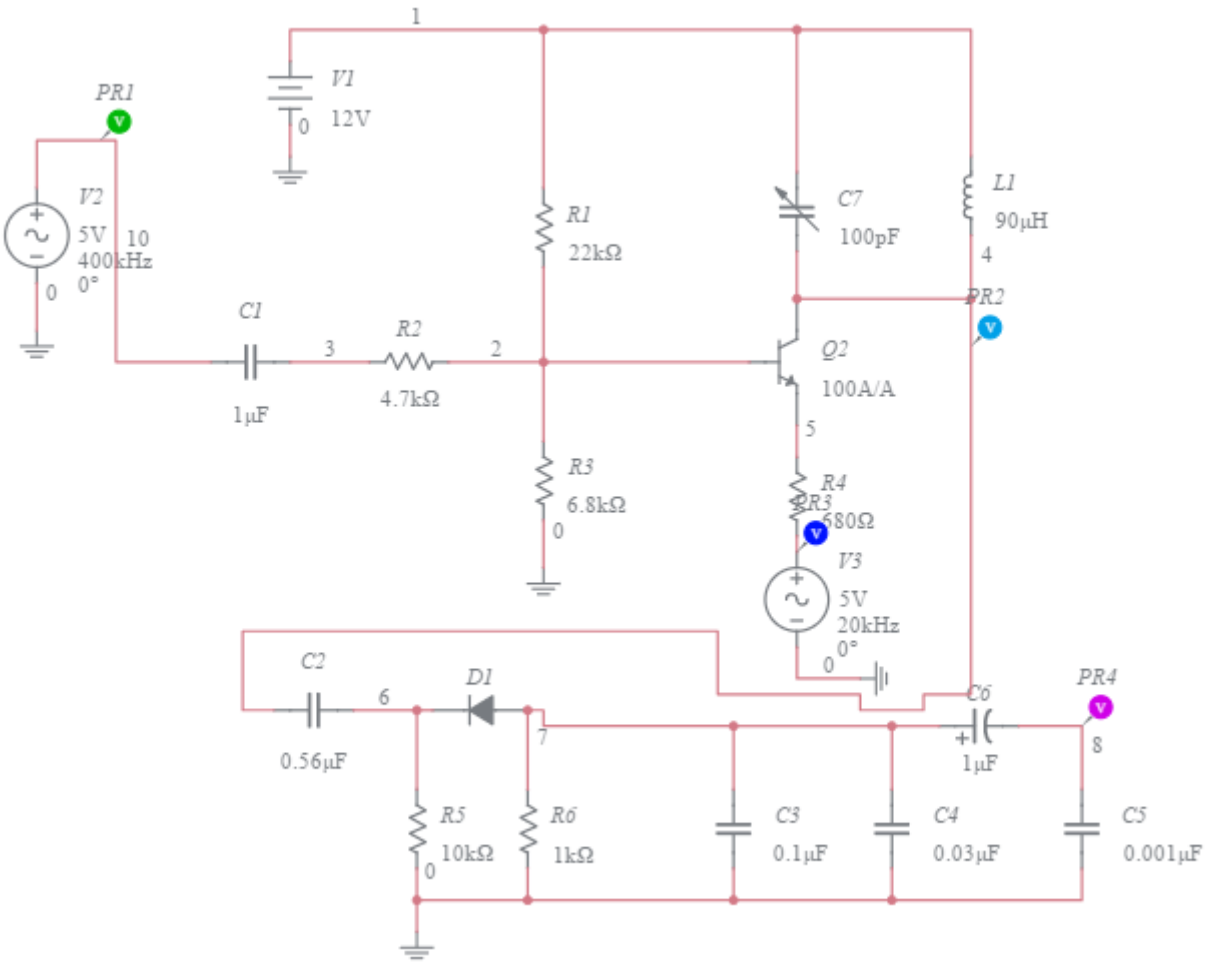
```
V1 dc 1 0 12v
R1 1 2 22k
R3 2 0 6.8k
R2 2 3 4.7k
C1  3 9 1uf
V2  ac 9 0  sin(5v, 400khz)
C7 1 10 100pf
Q2 10 2 5 100A/A
R4 5 11 80
V3 ac 11 0 sin(5v,20khz)
L1 1 4 90uh
C2 12 6 0.56uf
D1 7 6 da
R5 6 0 10k
R6 7 0 1k
C3 7 0 0.1uf
C4 7 0 0.03uf
C5 7 8 1uf
C6 8 0 0.001uf
.model 100A/A  npn
.tran 0.001ms 1ms
```

.plot tran v(9) v(4) v(5) v(8)

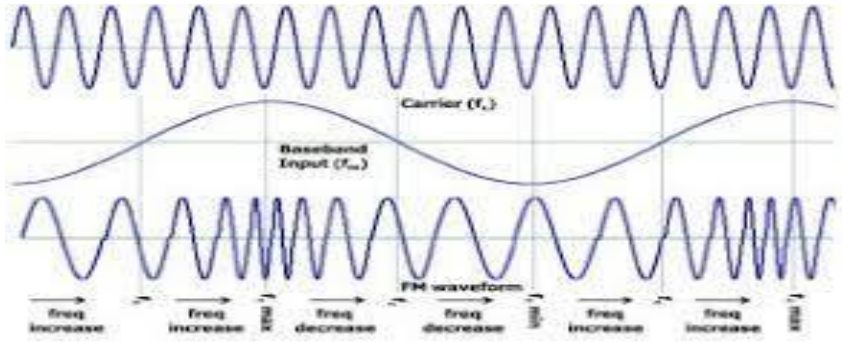
.probe

.end

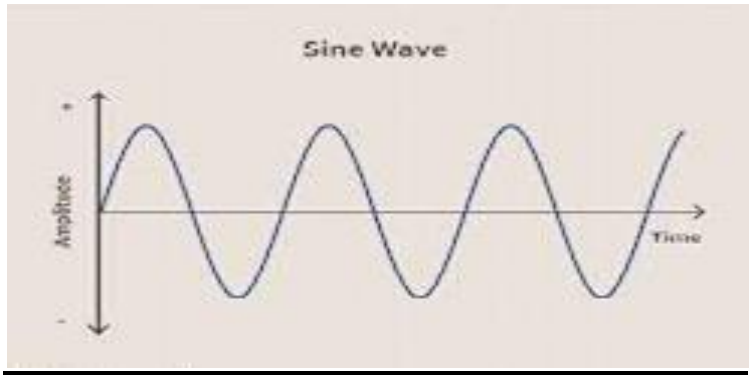
Circuit Diagram:-



Expected Model graphs:-



Carrier, Modulating and Modulated signals of FM



FM Demodulated signal

Result:-Simulated a Frequency Modulator and Demodulatorusing PSPICE Software and plotted the transient response graphs.

Viva-Voce Questions:-

1)What is meant by Frequency Modulation(FM)?

2) What are the different modulation and demodulations techniques used in FM?

3) What is bandwidth of FM used?

4) List the applications of FM?

PGM:1	Simulate an Astable multivibrator using an OP-Amp.

AIM :-To Simulate an Astable multivibrator using an OP-Amp. In PSPICEsoftware.

Apparatus:-PC, PSPICE A/D software

Program:-

r 3 2 50k

c 3 0 0.01uf

r1 4 1 35k

r2 1 0 30k

e 4 0 3 2 2e5

ri 2 3 1e6

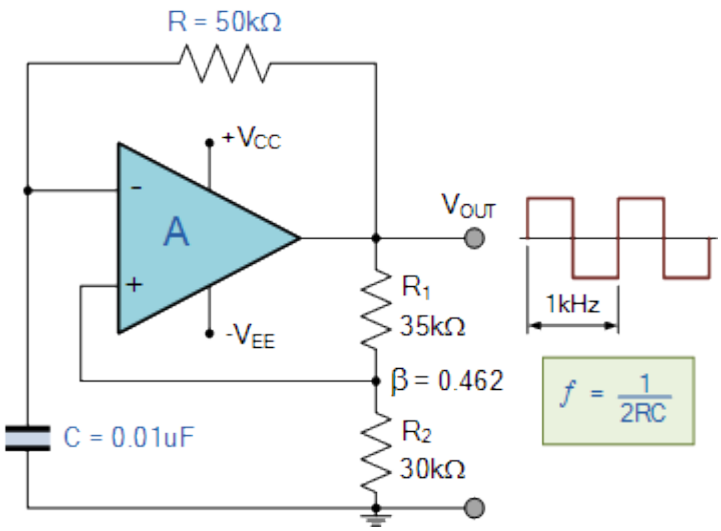
.ac dec 5 10 10meg

.plot ac v(4)

.probe

.end

Circuit Diagram



Result:-Simulated an Astable multivibrator using an OP-Amp in PSPICE Software and plotted the transient response graphs.

Viva-Voce Questions

1.What is meant by a multivibrator?

2.What are the other names of an astable multivibrator?

3.What type of output waveforms are generated by an astable multivibrator?