CHIRANJEEVI REDDY INSTITUTE OF ENGINEERING & TECHNOLOGY

(Approved by AICTE,New Delhi & Affiliated to JNTU,ANANTAPUR)
Susheela Nagar, Bellary Road, ANANTAPUR.

DEPARTMENT OF ELECTRONICS & COMMUNICATION ENGINEERING.

ELECTRONIC CIRCUIT ANALYSIS LAB
(9A04403)

(II B.Tech II Semester)

LAB-MANUAL

Head of the Department

S.Raghavendra Swami
M-Tech.,
Assistant professor in ECE.

LIST OF EXPERIMENTS

INTRODUCTION TO SPICE

SPICE (Simulation Program with Integrated Circuit Emphasis) is a general purpose analog circuit simulator. It is a powerful program that is used in IC and board-level design to check the integrity of circuit designs and to predict circuit behavior.

 This is of particular importance for integrated circuits. The SPICE was originally developed at the Electronics Research Laboratory of the University of California, Berkeley (1975), as its name implies SPICE can do several types of circuit analyses. Here are the most important ones:

Non-linear DC analysis: calculates the DC transfer curve.

Non-linear transient analysis: calculates the voltage and current as a function of time when a large signal is applied.

Linear AC Analysis: calculates the output as a function of frequency. A bode plot is generated.

Noise analysis

Sensitivity analysis

Distortion analysis

Fourier analysis: calculates and plots the frequency spectrum.

In addition, Spice has analog and digital libraries of standard components (such as NAND, NOR, flip-flops, and other digital gates, op amps, etc). This makes it a useful tool for a wide range of analog and digital applications.

All analyses can be done at different temperatures. The default temperature is 300K.

The circuit can contain the following components:

Independent and dependent voltage and current sources

Resistors	Switches
Capacitors	Diodes
Inductors	Bipolar Transistors
Mutual inductors	MOS Transistors
Transmission lines	JFET
Operational amplifiers	MESFET
Digital gates	

About B2 Spice A/D V4

B2 Spice A/D V4' contains a mixed mode simulator is based partly on the Berkeley SPICE simulator and partly on the Georgia Tech Xspice simulator. This means that you are getting industrial strength accuracy. B2 Spice A/D V4 is a 32-bit Windows application.

B2 Spice A/D V4 is intended to help you design analog, digital, and mixed mode circuits. Rather than working on your circuit design with physical components, which require expensive test equipment and a lab, B2 Spice A/D V4 allows you to perform realistic simulations on your circuit without clipping wires or splashing solder. With B2 Spice A/D V4, editing and simulating circuits is a quick, easy, even enjoyable process.

B2 Spice A/D V4 supports the full Spice 3F5 set of commands, options, and models. This includes simulations such as DC Sweep, AC Sweep, Transient, Sensitivity, Pole-Zero, Fourier, Distortion analysis, and more. Models include no less than six distinct MOSFET models, models for switches, several transmission line models, and much more.

B2 Spice A/D V4 is an application with two separate subprograms: the Workshop, and the Database Editor. The Workshop is most frequently used. You'll use it to create and edit your circuits, to set up the simulations, to run the simulations, and to view the results. The Database Editor is used for defining new parts or modifying those already in the parts bin. Each subprogram is covered in its own chapter.

The program features a large database of devices that should be sufficient for most circuits, and can be customized to meet your design needs. The Database Editor will explain how you can add more devices into the database.

B2 Spice A/D v4 comes in three flavors, professional, standard, and student. The professional version includes features not in the standard, and the standard contains features not in the student version. These differences will be discussed in the user manual.

B2 Spice A/D V4 allows you to enter a circuit design in the schematic editor, run simulations on the circuit, and view simulation results. B2 Spice A/D V4 has two distinct and incompatible simulators. Each of the two simulators has its own schematic mode. The mixed mode simulator simulates analog and mixed analog/digital circuits. Use the mixed mode schematic and simulator if your circuit is analog or mixed mode. If your circuit is a pure digital circuit, then use a pure digital schematic and simulator.

The program can also be used to run simulations from netlists and to graph arbitrary data sets.

Schematic editing overview

The schematic editor allows you to enter your circuit design. When building a new circuit, you will add parts into the circuit window by choosing them from menus and you will draw wires to connect the devices. Also, you will set properties for the devices to customize their behavior.

How to place parts

There is a set of commonly used parts in the Devices menus. Simply choose a part from the menu. If the part you want isn't in the Devices menus, then you can choose a category that describes the part. That will open a list of parts you can choose from.

You can also use the Parts window that is part of the Workspace window on the left. Sort the list by Part name, category, or manufacturer and select the part that you want.

After you choose a part it will follow your cursor around the circuit window. To place the part, click the left mouse button.

Set model properties

Double click on a part to set its model properties. This also allows you to set the name of the device.

Set device properties

Right click on the part, and then choose Set Device Properties from the floating menu. This opens a window that allows you to name the part, edit its symbol, and choose a new behavior for the part and more.

Change a symbol

You can move the symbol's name and property fields around by simply dragging them. Edit the symbol in more detail by right clicking on the symbol and selecting Edit Symbol. This will bring up the symbol in a separate window for editing. Also, you can choose from a set of pre -defined alternate symbols by right clicking on the symbol and choosing Select alternate symbol. After changing a symbol, you have the option of saving it back to the database so that next time you choose that part, it will have the new symbol.

Wiring

Drag a wire from a pin of a device and a wire will follow it. Let go and the wire will stay in the circuit. For precise wiring, use the wire drawing tool. Left clicks lay out the wire a segment at a time, and to end the wire, use the right-click or double click. Wires can be drawn with 90 degree angles by choosing checking the Use Perpendicular Wires Only checkbox in the Edit->Options menu. Wires can also be set to snap to the grid by choosing checking that option in the Edit->Options menu.

Move, delete, duplicate parts

To move, delete or duplicate parts, you must first select it with the arrow selection tool by clicking on it. To move the part or parts, simply drag the selected parts to the new position and let go of the mouse button. To copy and paste parts, just use the appropriate Edit menu commands or Ctrl-C to copy and Ctrl-V to paste. To delete a part, press the delete key.

Undo/Redo

B2 Spice A/D v4 now has unlimited levels of undo and redo. To undo any changes, press the CTRL-Z keys simultaneously or use the Edit->Undo menu command. The redo any undone changes, press the CTRL-Y keys or use the Edit->Redo menu command.

Naming and numbering nodes

Markers can be used to name a node or explicitly set a node number. Place the marker and double click on it to access the properties. Type in a name or number for the marker and the wire will take on the marker's name or number. Or you can simply double-click on the node name or number and change its name.

Netlist environment

Beside the schematic view, you can also work with circuits via the netlist. You can create a new netlist to work with by choosing new netlist document from the File menu. You can make a netlist from the current schematic by going to the File menu and selecting Create Netlist Document.

Setting up simulations from netlist interface is just like doing it from the schematic interface. Go to the Simulations menu and choose Set Up simulations and the window will allow you to activate simulations and specify how to run them. If you are a netlist expert, then you can type the simulation commands directly into the netlist.

Running simulations from netlist interface is just like running them from the circuit schematic interface. Simply click on the go button in the toolbar or choose Run Simulations from the menu. Simulation results will appear in graphs and tables.

The netlist interface is not supported for the pure digital environment.

Running simulations

Choose Set up Simulations… under the Simulation menu or go to the project's Simulation Specs subcategory in the Workspace and select the simulation to set up. Check the boxes of the simulations you wish to run and click the buttons to set the specific simulation parameters. You can also run the simulations from here, or by pressing F5 or the green "RUN" triangle in the toolbar.

You can view and set convergence related options by selecting Set Simulation Options. For more information on convergence issues and options, please refer to the section on Convergence Options.

Mixed mode-specific options can be set in the Mixed-mode options menu item. For more information on these options, please see the section on mixed mode circuit options.

Finally, you can customize how B2 Spice runs the simulations and how frequently it collects results using Set More Simulation Options under the Simulation menu.

The most common type of simulation in the mixed mode environment is the transient simulation. Transient is a fancy word meaning time. For the transient simulation, specify the time for which the simulation is to run and the step interval during the simulation. There are ac totally two step intervals available for you to set.

The more important of the two is the step ceiling, i.e. the maximum time step that the simulator can take. This is important because if it is too large and the circuit has sharp transitions, the simulator may miss some transitions. In general, however, the simulator does a good job of tracking changes in the circuit even if the step ceiling is relatively high. The other step interval you can set is used when you request that the data results be linearized, i.e. spaced evenly, by the step interval. This is useful when you want to see the results table because each row will vary by the same time interval with linearized results.

Viewing simulation results

Simulation data is processed in one of three ways: as it is generated, at the end of the simulation run, or at every update period. You can select which method you want by going to More Simulation Options under the Simulation menu. Processing the data as it is generated will be the slowest method (because of the time it takes to update the graph) while processing at the end of the simulation will be the fastest. A good compromise is at every update period specified in the "Update Period" box.

If you run a mixed mode transient simulation with digital parts, the bottom portion of the graph window will be used to display the digital traces.

Graphs can be customized in many ways. You can set the fonts and colors for the background, text, and plot lines. You can show and hide existing plots or create one of your own using mathematical functions. Simply double click on the graph to set its properties. Double click on individual plot listings in the legend to modify their properties. You can also right click on the plot to get a menu of all available graph options. With the new Workspace window, you now also have the ability to add plots from other graphs. Simply expand the appropriate graph so that its plots are listed in the tree, and drag over plots into the graph window.

Each graph also has a table view available. If the table is not showing, go to the View menu and select the Table View. The data for all the visible plots will be in the table. You can edit the table settings by selecting "Edit Table Settings" under the Edit menu. You can also add and delete plots via the Edit menu.

Digital results from mixed mode simulations can also be viewed in pure digital graphs. This gives you more options than the digital portion of the main graph view.

And for complex results (e.g., the results of ac, noise, distortion, or network analyses) you can view the results in polar graphs or even in smith charts.
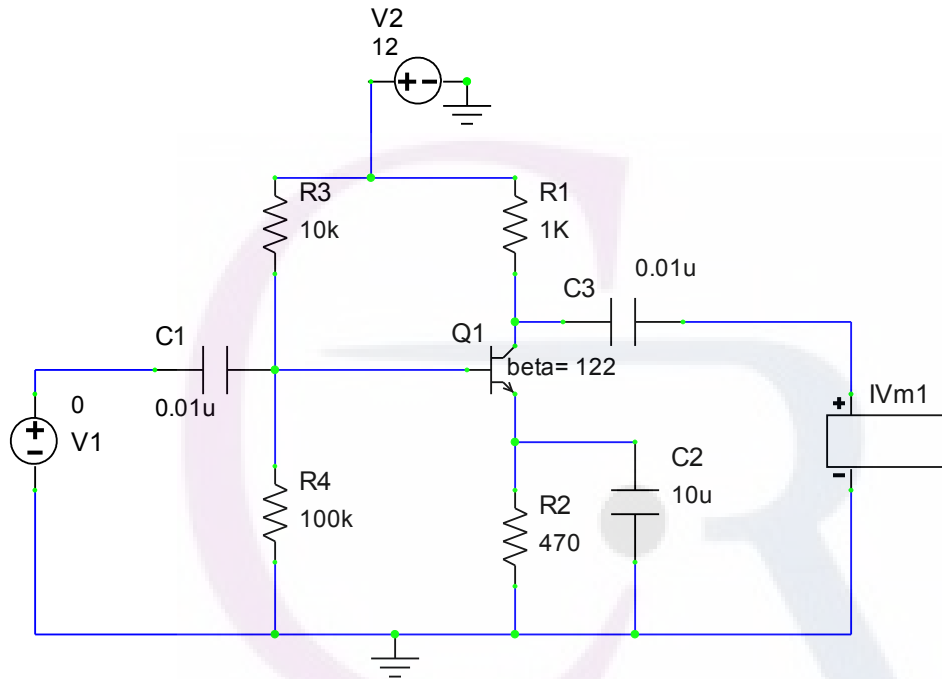
EXPERIMENT

SINGLE STAGE CE AMPLIFIER

AIM: To obtain the frequency response of single stage CE amplifier by using B2 SPICE.
APPARATUS:
    B2 SPICE software
THEORY:
CIRCUIT DIAGRAM:

TABULAR COLUMN:

S.No	Frequency in Hz	V_{in} (volts)	V_0 (volts)	Gain in dB
1	10	1	1.41	2.95
2	21.54	1	3.78	11.54
3	46.42	1	8.69	18.78
4	100	1	16.65	24.43
5	215.44	1	24.49	27.78
6	464.16	1	28.23	29.01
7	1.00k	1	29.28	29.33
8	+2.15k	1	29.52	29.4
9	+4.64k	1	29.57	29.42
10	+10.00k	1	29.58	29.42

11	+21.54k	1	29.59	29.42
12	+46.42k	1	29.59	29.42
13	+100.00k	1	29.59	29.42
14	+215.44k	1	29.59	29.42
15	+464.16k	1	29.59	29.42
16	+1.00Meg	1	29.58	29.42
17	+2.15Meg	1	29.57	29.42
18	+4.64Meg	1	29.52	29.4
19	+10.00Meg	1	29.29	29.33
20	+21.54Meg	1	28.27	29.03
21	+46.42Meg	1	24.63	27.83
22	+100.00Meg	1	16.95	24.58
23	+215.44Meg	1	9.19	19.27
24	+464.16Meg	1	4.58	13.22
25	+1.00G	1	2.41	7.63

PROCEDURE:
1. Regup the circuit as shown in figure by choosing appropriate devices from the menu titled devices
2. Choose the wire drawing tool from the tool bag and draw the lines.
3. Give the appropriate names and values for all elements present in the circuit.
4. An AC voltage source of '0' phase, 1V amplitude, variable frequency is applied as input signal by editing the voltage source.
5. Then choose set up simulation from simulation menu.
6. Choose the option of AC frequency analysis and give starting and ending frequency ranges.
7. Select the option of view table and view graph .
8. Now choose run simulation.
9. Observe the output frequency response graph and take the maximum gain and 3 dB frequencies.
10. Note down the tabular column.

Graph:

A graph should be drawn by taking frequency on x-axis and gain in dB on y-axis.
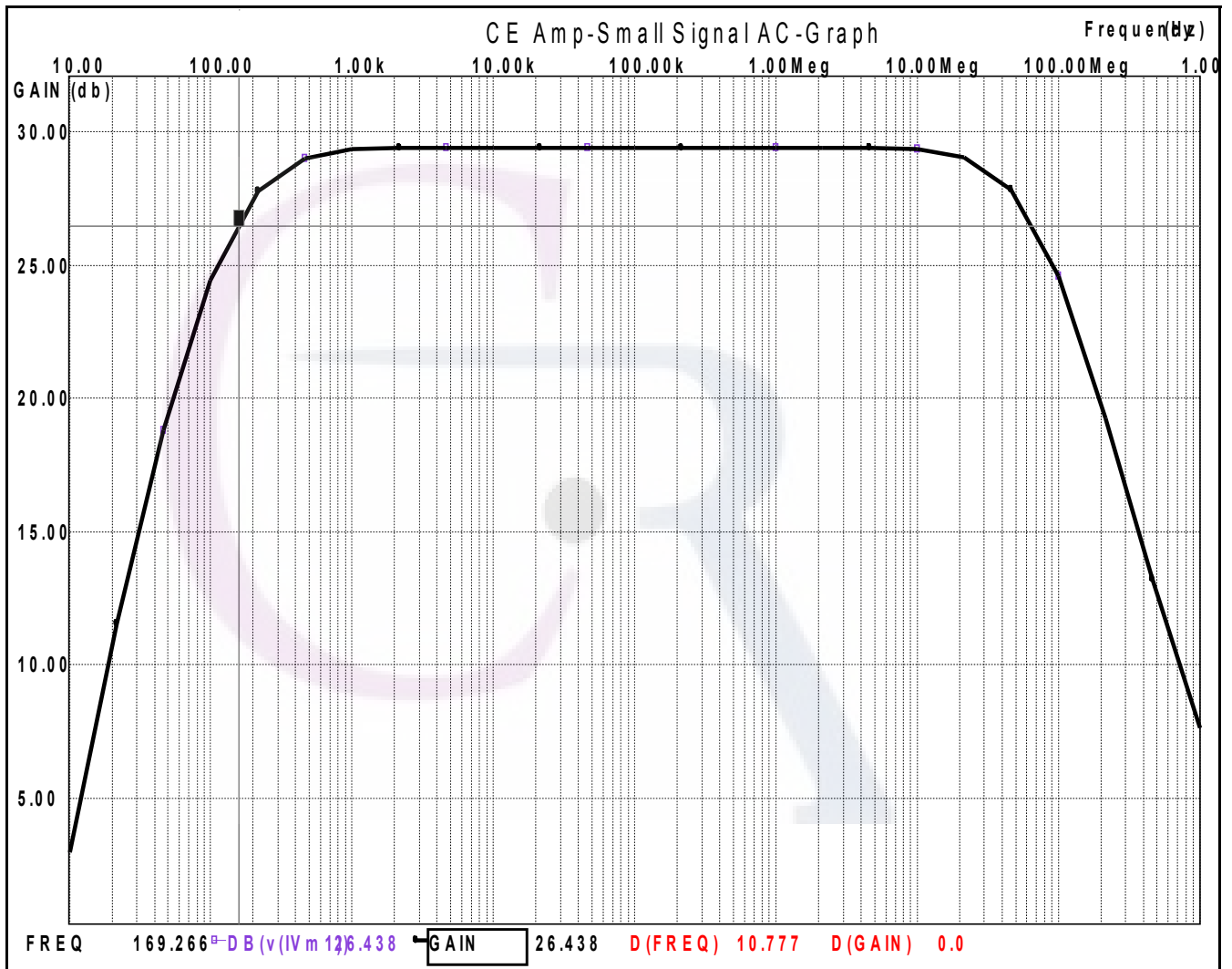
Net list:
***** main circuit
Q1 2 1 3 qbf469
R1 4 2 1K
R2 3 0 470

```
R3 4  1  100k
R4 1  0  10k
C1 5  1  1u
V1 5  0 DC 0  AC 1 0
C2 3  0  47u
C3 2  7  1u
IVm1 7  0 0
V2 4  0 DC 12
.AC Dec  3  10  1000meg
.END
```


Result: The frequency response of single stage CE amplifier is obtained by using B2
SPICE.

EXPERIMENT
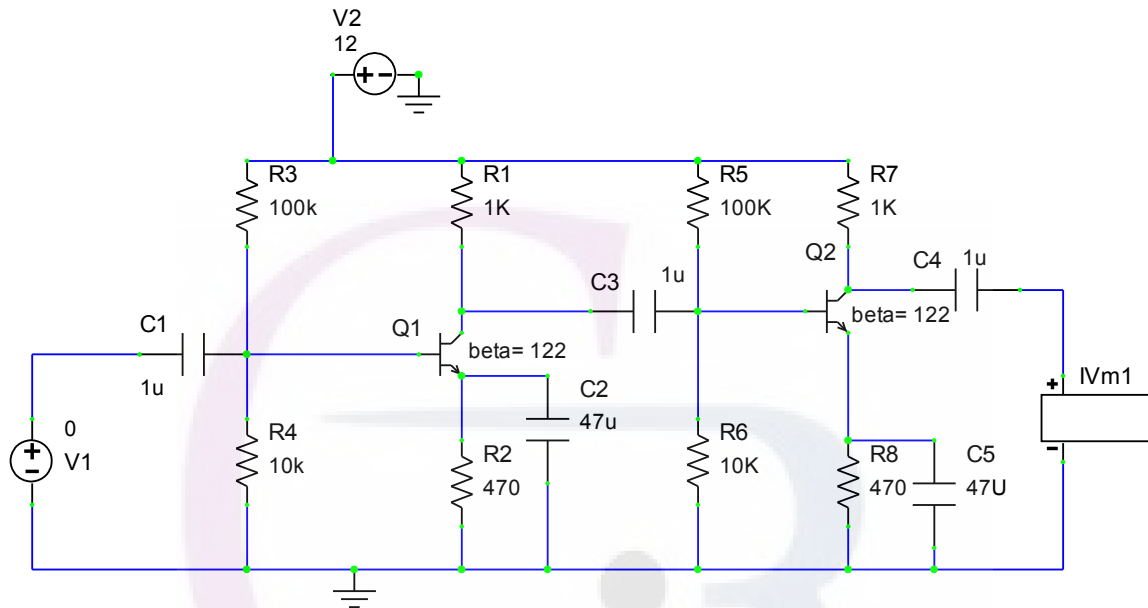
TWO STAGE RC COUPLED AMPLIFIER

AIM: To obtain the frequency response of two stage RC coupled amplifier by using B2 SPICE.

APPARATUS:

      B2 SPICE software

THEORY:

CIRCUIT DIAGRAM:

Net List:

```
***** main circuit
R5 5  11  100K
R1 5  4  1K
R2 3  0  470
Q2 15  11  16 qbf469
R4 9  0  10k
C1 10  9  1u
V1 10  0 DC 0 SIN( 0 1 1meg 0 0) AC 1 0
Q1 4  9  3 qbf469
C2 3  0  47u
R3 9  5  100k
C3 4  11  1u
V2 5  0 DC 12
R6 11  0  10K
R7 5  15  1K
R8 16  0  470
C4 15  12  1u
C5 16  0  47U
IVm1 12  0 0
.AC Dec  3  10  100meg
.END
```
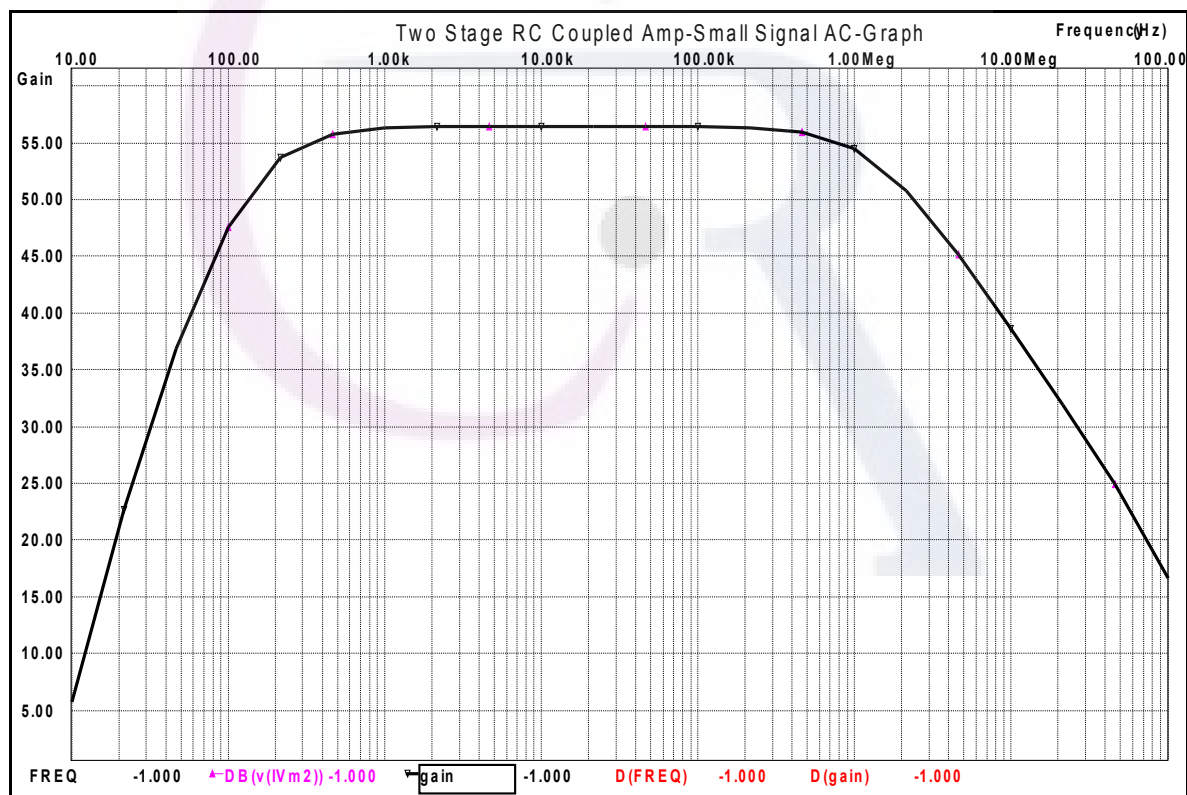
TABULAR COLUMN:

S.No	Frequency in Hz	V_{in} (volts)	V_0 (volts)	Gain in dB
1	10	1	1.93	5.71
2	21.54	1	13.55	22.64
3	46.42	1	69.38	36.82
4	100	1	240	47.6
5	215.44	1	481.3	53.65
6	464.16	1	613.92	55.76
7	+1.00k	1	652.81	56.3
8	+2.15k	1	661.85	56.42
9	+4.64k	1	663.83	56.44
10	+10.00k	1	664.25	56.45
11	+21.54k	1	664.27	56.45
12	+46.42k	1	663.96	56.44
13	+100.00k	1	662.44	56.42
14	+215.44k	1	655.5	56.33
15	+464.16k	1	625.92	55.93
16	+1.00Meg	1	527.21	54.44
17	+2.15Meg	1	343.99	50.73
18	+4.64Meg	1	179.59	45.09
19	+10.00Meg	1	85.64	38.65
20	+21.54Meg	1	39.56	31.95
21	+46.42Meg	1	17.5	24.86
22	+100.00Meg	1	6.79	16.64

PROCEDURE:

1. Regup the circuit as shown in figure by choosing appropriate devices from the menu titled devices
2. Choose the wire drawing tool from the tool bag and draw the lines.
3. Give the appropriate names and values for all elements present in the circuit.
4. An AC voltage source of '0' phase, 1V amplitude, variable frequency is applied as input signal by editing the voltage source.
5. Then choose set up simulation from simulation menu.
6. Choose the option of AC frequency analysis and give starting and ending frequency ranges.
7. Select the option of view table and view graph .
8. Now choose run simulation.
9. Observe the output frequency response graph and take the maximum gain and 3 dB frequencies.
10. Note down the tabular column.

Graph:
   A graph should be drawn by taking frequency on x-axis and gain in dB on y-axis.

Result: The frequency response of two stage RC coupled amplifier is obtained by using B2 SPICE.
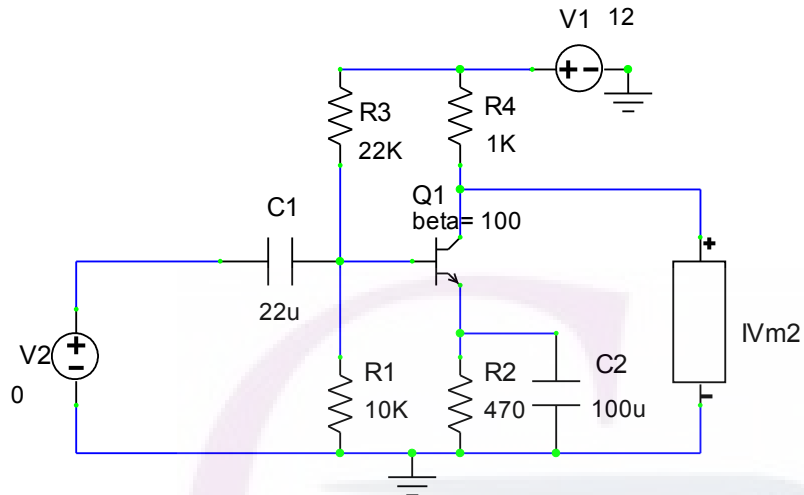
EXPERIMENT

CLASS A POWER AMPLIFIER

AIM: To obtain the frequency response of Class A power amplifier by using B2 SPICE.
APPARATUS:
   B2 SPICE software
THEORY:
CIRCUIT DIAGRAM:

Net List:

```
***** main circuit
Q1 6  1  3 q2n2219a
R1 1  0  10K
R2 3  0  470
R3 4  1  22K
R4 4  6  1K
C1 5  1  22u
C2 0  3  100u
V1 4  0 DC 12
V2 5  0 DC 0 SIN( 0) AC 1 0
IVm1 5  0 0
IVm2 6  0 0

.AC Dec  20  10  1000meg
.END
```

TABULAR COLUMN:

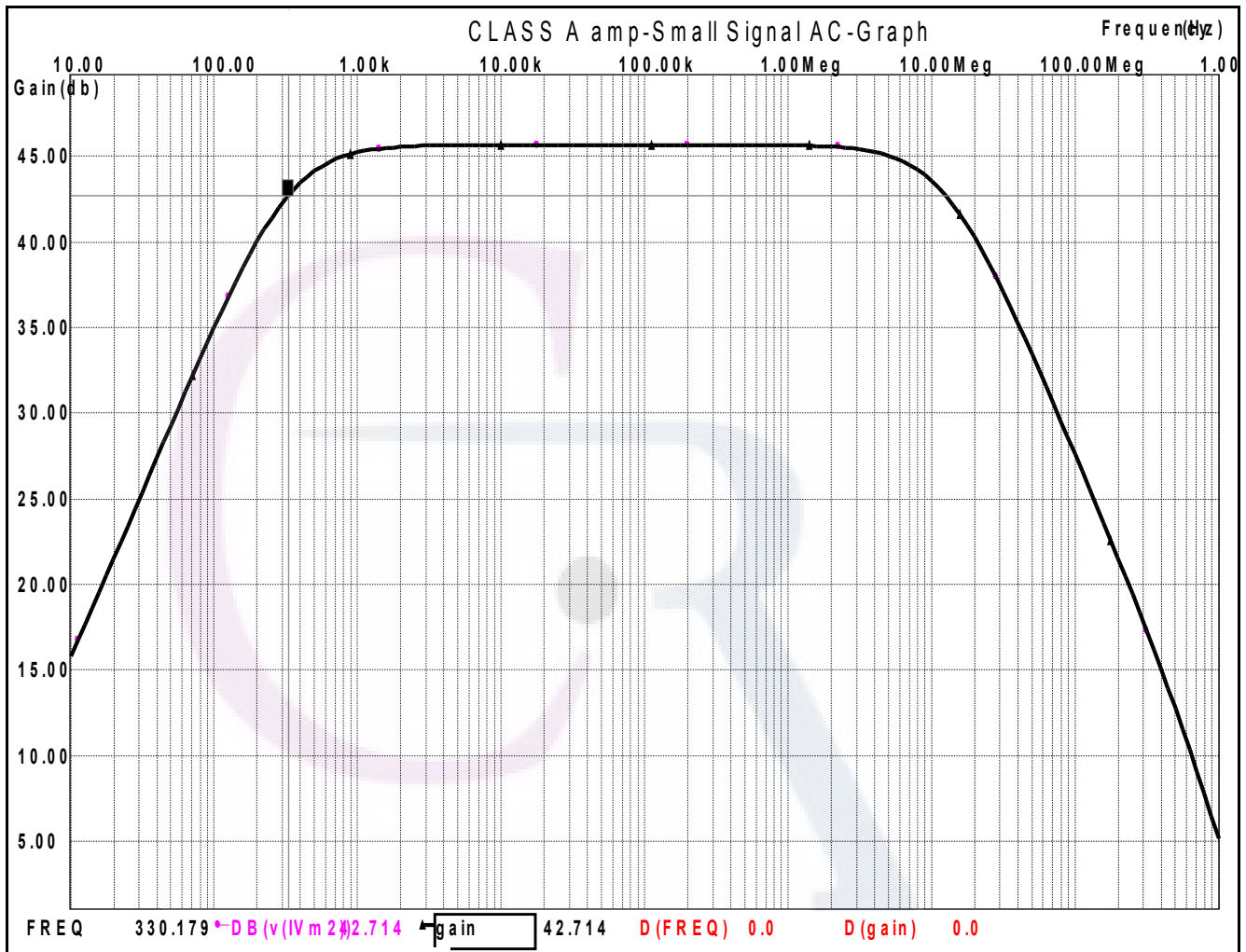| S.No.	Frequency in Hz	V_{IN} (volts)	V_0 (volts)	Gain in dB
1	10	1	6.18	15.82
2	17.78	1	10.63	20.53
3	31.62	1	18.64	25.41
4	56.23	1	32.71	30.29
5	100	1	56.37	35.02
6	177.83	1	92.04	39.28
7	316.23	1	133.84	42.53
8	562.34	1	166.34	44.42
9	+1.00k	1	182.83	45.24
10	+1.78k	1	189.16	45.54
11	+3.16k	1	191.3	45.63
12	+5.62k	1	191.99	45.67
13	+10.00k	1	192.22	45.68
14	+17.78k	1	192.29	45.68
15	+31.62k	1	192.31	45.68
16	+56.23k	1	192.31	45.68
17	+100.00k	1	192.31	45.68
18	+177.83k	1	192.3	45.68
19	+316.23k	1	192.26	45.68
20	+562.34k	1	192.13	45.67
21	+1.00Meg	1	191.72	45.65
22	+1.78Meg	1	190.45	45.6
23	+3.16Meg	1	186.6	45.42
24	+5.62Meg	1	175.8	44.9
25	+10.00Meg	1	151	43.58
26	+17.78Meg	1	111.63	40.96
27	+31.62Meg	1	71.53	37.09
28	+56.23Meg	1	42.23	32.51
29	+100.00Meg	1	24.06	27.63

PROCEDURE:
1. Regup the circuit as shown in figure by choosing appropriate devices from the menu titled devices
2. Choose the wire drawing tool from the tool bag and draw the lines.
3. Give the appropriate names and values for all elements present in the circuit.
4. An AC voltage source of '0' phase, 1V amplitude, variable frequency is applied as input signal by editing the voltage source.
5. Then choose set up simulation from simulation menu.
6. Choose the option of AC frequency analysis and give starting and ending frequency ranges.

7. Select the option of view table and view graph .
8. Now choose run simulation.
9. Observe the output frequency response graph and take the maximum gain and 3 dB frequencies.
10. Note down the tabular column.

Graph:

A graph should be drawn by taking frequency on x-axis and gain in dB on y-axis.

Result: The frequ   Class A power amplifier is obtained by using B2 SPICE.
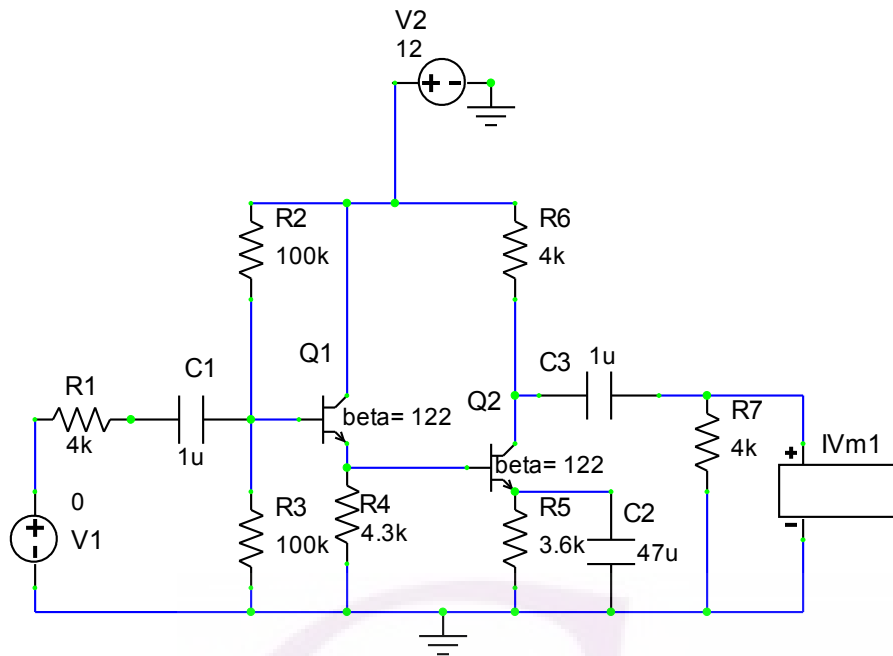
EXPERIMENT

CASCADE AMPLIFIER

AIM: To obtain the frequency response of CASCADE amplifier by using B2 SPICE.
APPARATUS:
    B2 SPICE software
THEORY:
CIRCUIT DIAGRAM:

Net List:

```
***** main circuit
Q1 22  1  3 qbf469
Q2 13  3  6 qbf469
R1 7  8  4k
R2 22  1  100k
R3 1  0  100k
R4 3  0  4.3k
R5 6  0  3.6k
R6 22  13  4k
R7 24  0  4k
C1 8  1  1u
C2 6  0  47u
C3 13  24  1u
V1 7  0 DC 0  AC 1 0
V2 22  0 DC 12
IVm1 24  0 0

.AC Dec  20  10  10meg
.END
```

TABULAR COLUMN:

S.No	Frequency in Hz	V_{IN} (volts)	V_{OUT} (volts)	Gain in dB
1	10	1	2.24	7.02
2	17.78	1	6.05	15.63
3	31.62	1	13.48	22.6

| 4	56.23	1	25.23	28.04
5	100	1	40.24	32.09
6	177.83	1	53.86	34.62
7	316.23	1	61.85	35.83
8	562.34	1	65.19	36.28
9	+1.00k	1	66.36	36.44
10	+1.78k	1	66.74	36.49
11	+3.16k	1	66.86	36.5
12	+5.62k	1	66.9	36.51
13	+10.00k	1	66.92	36.51
14	+17.78k	1	66.92	36.51
15	+31.62k	1	66.93	36.51
16	+56.23k	1	66.94	36.51
17	+100.00k	1	66.98	36.52
18	+177.83k	1	67.11	36.54
19	+316.23k	1	67.52	36.59
20	+562.34k	1	68.85	36.76
21	+1.00Meg	1	73.23	37.29
22	+1.78Meg	1	88.17	38.91
23	+3.16Meg	1	83.93	38.48
24	+5.62Meg	1	21.62	26.7
25	+10.00Meg	1	6	15.56

PROCEDURE:

11. Regup the circuit as shown in figure by choosing appropriate devices from the menu titled devices
12. Choose the wire drawing tool from the tool bag and draw the lines.
13. Give the appropriate names and values for all elements present in the circuit.
14. An AC voltage source of '0' phase, 1V amplitude, variable frequency is applied as input signal by editing the voltage source.
15. Then choose set up simulation from simulation menu.
16. Choose the option of AC frequency analysis and give starting and ending frequency ranges.

17. Select the option of view table and view graph .
18. Now choose run simulation.
19. Observe the output frequency response graph and take the maximum gain and 3 dB frequencies.
20. Note down the tabular column.

Graph:

A graph should be drawn by taking frequency on x-axis and gain in dB on y-axis.

CASCADE amp-Small Signal AC-Graph
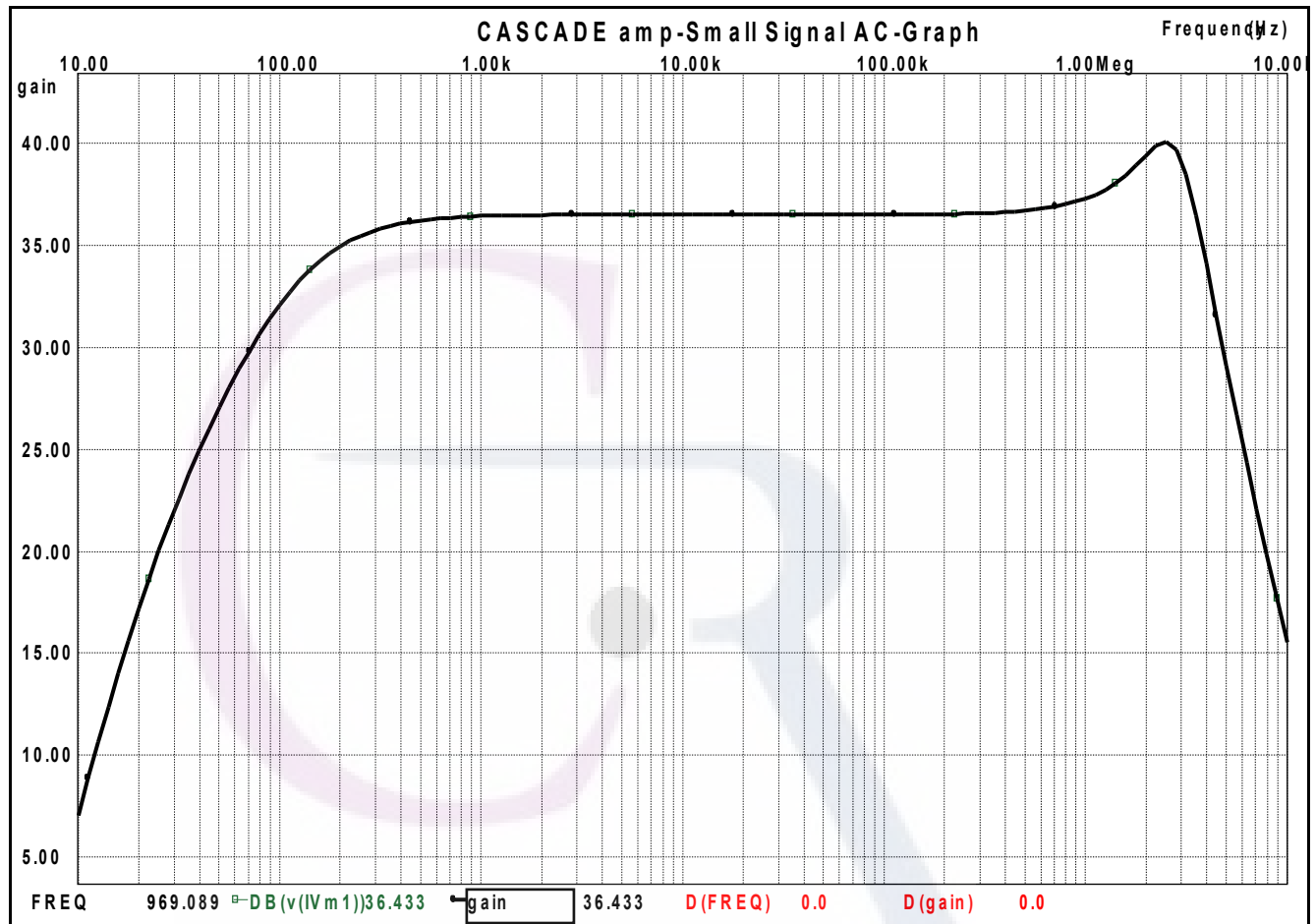
Result: The frequency response of CASCADE amplifier is obtained by using B2 SPICE.
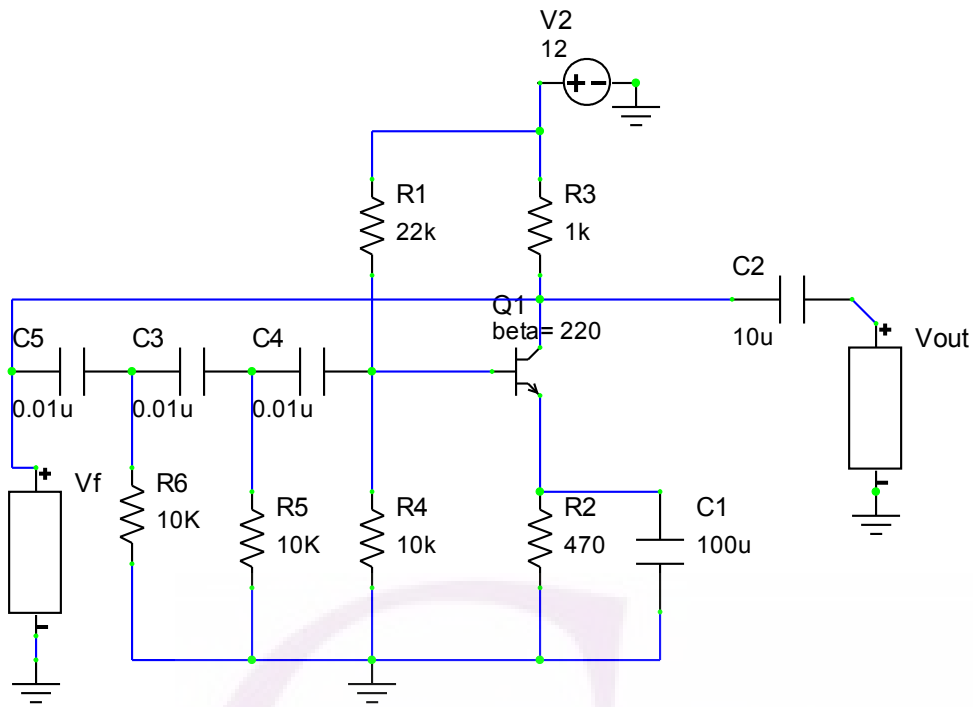
EXPERIMENT

RC PHASE SHIFT OSCILLATOR

AIM: To study the operation of RC phase shift oscillator by using B2 SPICE.
APPARATUS:
B2 SPICE software
THEORY:
CIRCUIT DIAGRAM:

Net List:
```
***** main circuit
Q1 3  5  6 q2n2222a
R1 4  5  22k
R2 6  0  470
R3 4  3  1k
R4 5  0  10k
C1 6  0  100u
C2 3  7  10u
C3 10  9  0.01u
R6 10  0  10K
V2 4  0 DC 12
R5 9  0  10K
IVout 7  0 0
C4 9  5  0.01u
C5 3  10  0.01u
IVf 3  0 0

.TRAN  100m 80m 45m 0.01u uic
.IC
.END
```
TABULAR COLUMN:

PROCEDURE:

S.No	Time(Sec)	V REF	V OUT		S.No	Time(Sec)	V_{REF}	V_{OUT}
1	+50.00m	6.14	6.14		41	+50.04m	6.24	6.24
2	+50.00m	6.14	6.14		42	+50.04m	6.25	6.25
3	+50.00m	6.14	6.14		43	+50.04m	6.25	6.25
4	+50.00m	6.15	6.15		44	+50.04m	6.25	6.25
5	+50.00m	6.15	6.15		45	+50.04m	6.25	6.25
6	+50.01m	6.15	6.15		46	+50.05m	6.26	6.26
7	+50.01m	6.15	6.15		47	+50.05m	6.26	6.26
8	+50.01m	6.16	6.16		48	+50.05m	6.26	6.26
9	+50.01m	6.16	6.16		49	+50.05m	6.26	6.26
10	+50.01m	6.16	6.16		50	+50.05m	6.27	6.27
11	+50.01m	6.17	6.17		51	+50.05m	6.27	6.27
12	+50.01m	6.17	6.17		52	+50.05m	6.27	6.27
13	+50.01m	6.17	6.17		53	+50.05m	6.27	6.27
14	+50.01m	6.17	6.17		54	+50.05m	6.28	6.28
15	+50.01m	6.18	6.18		55	+50.05m	6.28	6.28
16	+50.02m	6.18	6.18		56	+50.06m	6.28	6.28
17	+50.02m	6.18	6.18		57	+50.06m	6.28	6.28
18	+50.02m	6.19	6.19		58	+50.06m	6.28	6.28
19	+50.02m	6.19	6.19		59	+50.06m	6.29	6.29
20	+50.02m	6.19	6.19		60	+50.06m	6.29	6.29
21	+50.02m	6.19	6.19		61	+50.06m	6.29	6.29
22	+50.02m	6.2	6.2		62	+50.06m	6.29	6.29
23	+50.02m	6.2	6.2		63	+50.06m	6.3	6.3
24	+50.02m	6.2	6.2		64	+50.06m	6.3	6.3
25	+50.02m	6.2	6.2		65	+50.06m	6.3	6.3
26	+50.03m	6.21	6.21		66	+50.07m	6.3	6.3
27	+50.03m	6.21	6.21		67	+50.07m	6.3	6.3
28	+50.03m	6.21	6.21		68	+50.07m	6.31	6.31
29	+50.03m	6.21	6.21		69	+50.07m	6.31	6.31
30	+50.03m	6.22	6.22		70	+50.07m	6.31	6.31
31	+50.03m	6.22	6.22		71	+50.07m	6.31	6.31
32	+50.03m	6.22	6.22		72	+50.07m	6.31	6.31
33	+50.03m	6.22	6.22		73	+50.07m	6.32	6.32
34	+50.03m	6.23	6.23		74	+50.07m	6.32	6.32
35	+50.03m	6.23	6.23		75	+50.07m	6.32	6.32
36	+50.04m	6.23	6.23		76	+50.08m	6.32	6.32
37	+50.04m	6.23	6.23		77	+50.08m	6.32	6.32
38	+50.04m	6.24	6.24		78	+50.08m	6.33	6.33
39	+50.04m	6.24	6.24		79	+50.08m	6.33	6.33
40	+50.04m	6.24	6.24		80	+50.08m	6.33	6.33

1. connect the circuit as per the circuit diagram & give the specified values for all devices
2. Then click on SIMULATION menu & choose setup simulation
3. Then a window is displayed from that choose TRANSIENT option & set the values as given
(a) start value
(b) stop time
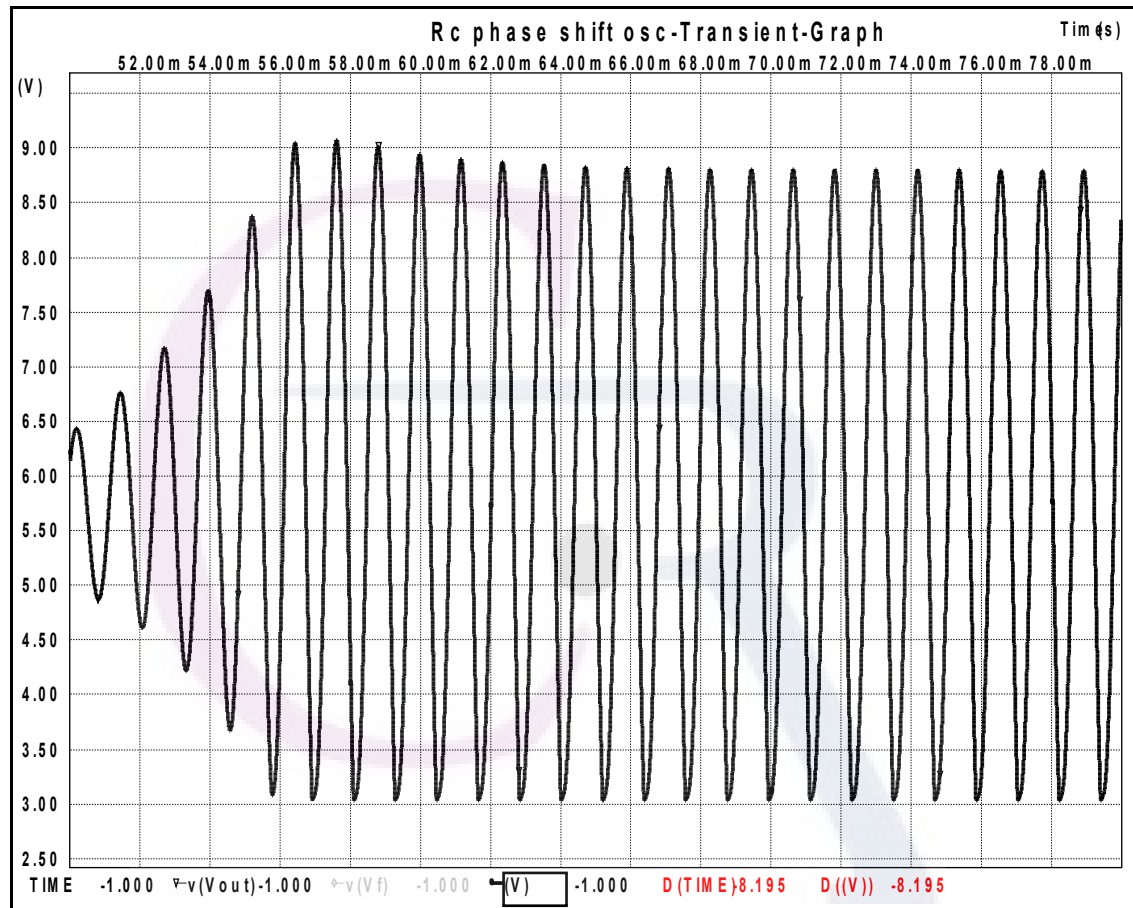(c) linearization setup

(d) step ceiling

4.select linearise result & then click ok.
5.After click on ok button & then choose RUNNOW option from SIMULATION window & hen graph will be displayed
6.Then choose NET LIST option from the file menu & note down the net list
7.Then note down time period for one cycle & calculate the frequency theoretical as well as practical.

RESULT:
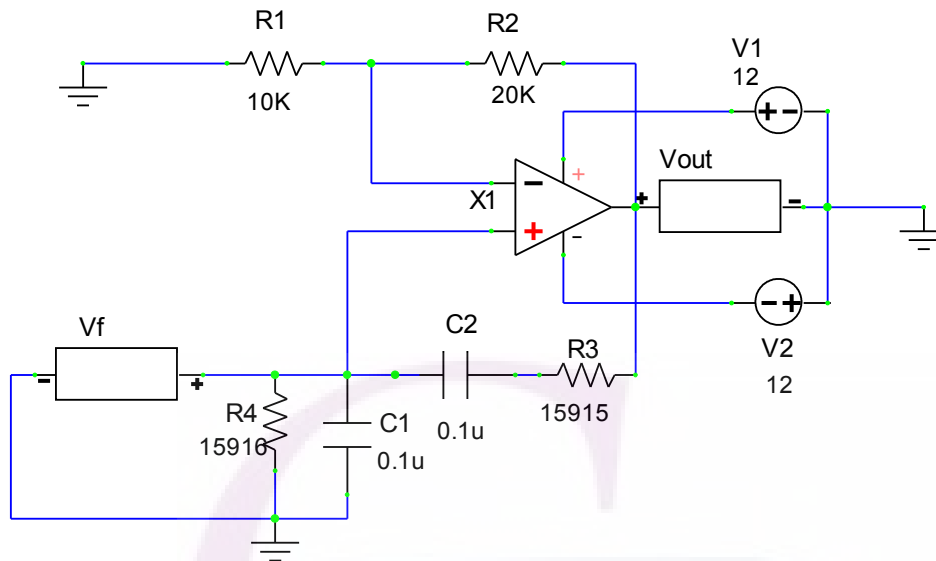
EXPERIMENT

WEIN BRIDGE AMPLIFIER

AIM: To study the operation of WEIN BRIDGE oscillator by using B2 SPICE.
APPARATUS:

      B2 SPICE software

THEORY:

CIRCUIT DIAGRAM:

Net List:

****

* B2 Spice

****

* B2 Spice default format (same as Berkeley Spice 3F format)

***** subcircuit definitions

* Op-Amp Macromodel
* based on op-amp macromodelling discussion located in
* 'Macromodelling with Spice',
* by Connelly & Choi, Prentice Hall publisher.
* Pin #   Pin Name   Pin description
* 1   +IN   Input Node
* 5   -IN   Input Node
* 14 OUT   Output Node
* 9   VCC+ + Power Supply
* 11 VCC- - Power Supply
.SubCkt OpAmp 1 5 14 9 11
R1 3 0 2.000000e+009
R2 3 4 2.000000e+006
R3 4 0 2.000000e+009
R4 6 0 1e3
R5 12 0 7.500000e+001
R6 13 0 1e3
R7 17 18 10e3
R8 18 0 -5.000000e+003
R9 19 0 1e3
I1 3 0 9.000000e-008
I2 4 0 7.000000e-008

```
C1 7 0 3.183099e-005
C2 13 0 6.241370e-011
C3 3 4 1.400000e-012
C4 17 18 -5.305165e-013
C5 19 0 2.273642e-015
G1 0 6 19 0 1.995262e+002
G2a 0 6 3 0 3.154787e-003
G2b 0 6 4 0 3.154787e-003
G3 0 12 7 0 1.333333e-002
G4 0 13 3 4 0.001
G5 0 19 18 0 0.001
VA 12 14 DC 0
VB 6 7 DC 0
BF1 8 9  I = -1.591549e+001+ I(VB) * 1
BF2 11 10  I = -1.591549e+001+ I(VB) * (-1)
BF3 15 9 I = -2.500000e-002 + I(VA) * 1
BF4 11 16 I= -2.500000e-002 + I(VA) * (-1)
E1 17 0 13 0 -1.000000e+000
VC 2 3 1.000000e-003
* This is for a more accurate model of an npn input:
D1 1 2 DX
D2 5 4 DX
D3 7 8 DX
D4 8 9 DX
D5 10 7 DX
D6 11 10 DX
D7 7 9 DX
D8 11 7 DX
D9 14 15 DX
D10 15 9 DX
D11 16 14 DX
D12 11 16 DX
.MODEL DX D(N=.001)
.ends


***** main circuit
XX1 3  12  5  2  7 OpAmp
V1 2  0 DC 12
V2 0  7 DC 12
IVout 5  0 0
R1 0  12  10K
R2 12  5  20K
R3 15  5  15915
R4 3  0  15916
C1 3  0  0.1u
C2 3  15  0.1u
IVf 3  0 0
.OPTIONS  gmin = 1E-12  reltol = 1E-4  itl1 = 500  itl4 = 500
+  rshunt = 1G
.TRAN  100u 100m 50u 100u uic
.IC
.END
```

TABULAR COLUMN:

S.No.	Time	V REF	V OUT	S.No.	Time	V REF	V OUT
1	+1.05m	-2.17m	-8.14m	41	+5.05m	-2.77m	-9.93m
2	+1.15m	-2.38m	-8.76m	42	+5.15m	-2.57m	-9.34m
3	+1.25m	-2.58m	-9.36m	43	+5.25m	-2.37m	-8.74m
4	+1.35m	-2.78m	-9.95m	44	+5.35m	-2.17m	-8.12m
5	+1.45m	-2.97m	-10.52m	45	+5.45m	-1.96m	-7.50m
6	+1.55m	-3.15m	-11.08m	46	+5.55m	-1.75m	-6.87m
7	+1.65m	-3.33m	-11.61m	47	+5.65m	-1.54m	-6.24m
8	+1.75m	-3.50m	-12.13m	48	+5.75m	-1.33m	-5.60m
9	+1.85m	-3.67m	-12.62m	49	+5.85m	-1.12m	-4.97m
10	+1.95m	-3.82m	-13.08m	50	+5.95m	-905.35u	-4.34m
11	+2.05m	-3.96m	-13.51m	51	+6.05m	-697.49u	-3.72m
12	+2.15m	-4.10m	-13.92m	52	+6.15m	-492.53u	-3.10m
13	+2.25m	-4.22m	-14.29m	53	+6.25m	-291.27u	-2.50m
14	+2.35m	-4.34m	-14.63m	54	+6.35m	-94.50u	-1.91m
15	+2.45m	-4.44m	-14.93m	55	+6.45m	+97.00u	-1.33m
16	+2.55m	-4.53m	-15.20m	56	+6.55m	+282.47u	-774.87u
17	+2.65m	-4.61m	-15.43m	57	+6.65m	+461.18u	-238.61u
18	+2.75m	-4.67m	-15.63m	58	+6.75m	+632.44u	+275.28u
19	+2.85m	-4.72m	-15.79m	59	+6.85m	+795.55u	+764.77u
20	+2.95m	-4.76m	-15.91m	60	+6.95m	+949.89u	+1.23m
21	+3.05m	-4.79m	-15.98m	61	+7.05m	+1.09m	+1.66m
22	+3.15m	-4.80m	-16.02m	62	+7.15m	+1.23m	+2.07m
23	+3.25m	-4.80m	-16.02m	63	+7.25m	+1.35m	+2.44m
24	+3.35m	-4.79m	-15.98m	64	+7.35m	+1.47m	+2.78m
25	+3.45m	-4.76m	-15.90m	65	+7.45m	+1.57m	+3.09m
26	+3.55m	-4.72m	-15.78m	66	+7.55m	+1.66m	+3.36m
27	+3.65m	-4.67m	-15.63m	67	+7.65m	+1.74m	+3.59m
28	+3.75m	-4.60m	-15.43m	68	+7.75m	+1.80m	+3.79m
29	+3.85m	-4.52m	-15.20m	69	+7.85m	+1.86m	+3.95m
30	+3.95m	-4.43m	-14.92m	70	+7.95m	+1.90m	+4.07m
31	+4.05m	-4.33m	-14.62m	71	+8.05m	+1.92m	+4.15m
32	+4.15m	-4.22m	-14.28m	72	+8.15m	+1.94m	+4.19m
33	+4.25m	-4.09m	-13.90m	73	+8.25m	+1.94m	+4.19m
34	+4.35m	-3.96m	-13.50m	74	+8.35m	+1.92m	+4.15m
35	+4.45m	-3.81m	-13.07m	75	+8.45m	+1.90m	+4.07m
36	+4.55m	-3.66m	-12.60m	76	+8.55m	+1.86m	+3.96m
37	+4.65m	-3.50m	-12.11m	77	+8.65m	+1.81m	+3.80m
38	+4.75m	-3.33m	-11.60m	78	+8.75m	+1.74m	+3.60m
39	+4.85m	-3.15m	-11.06m	79	+8.85m	+1.66m	+3.37m
40	+4.95m	-2.96m	-10.51m	80	+8.95m	+1.57m	+3.10m

PROCEDURE:

4. connect the circuit as per the circuit diagram & give the specified values for all devices
5. Then click on SIMULATION menu & choose setup simulation

6. Then a window is displayed from that choose TRANSIENT option & set the values as given
(e) start value
(f) stop time
(g) linearization setup
(h) step ceiling

4.select linearise result & then click ok.
5.After click on ok button & then choose RUNNOW option from SIMULATION window & hen graph will be displayed
6.Then choose NET LIST option from the file menu & note down the net list
7.Then note down time period for one cycle & calculate the frequency theoretical as well as practical.
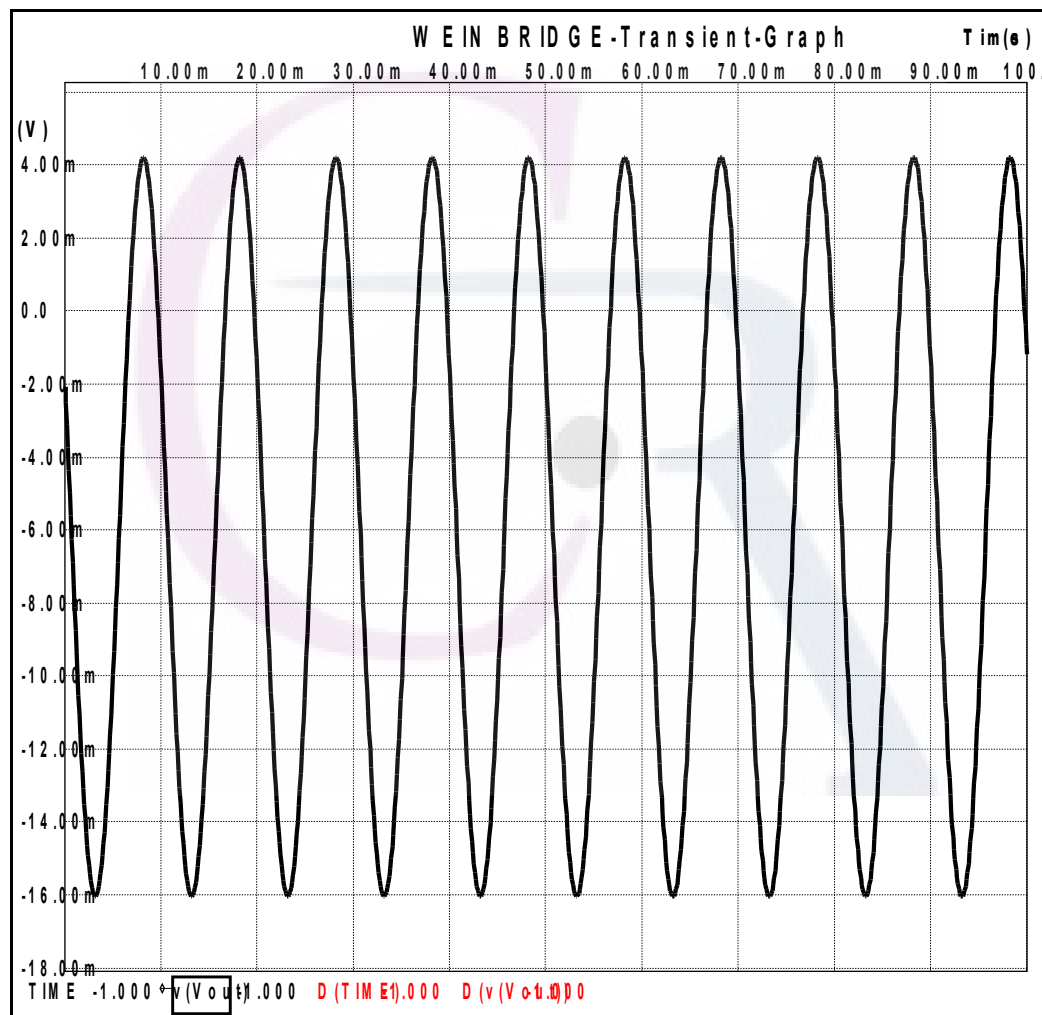
RESULT:

TWO STAGE RC COUPLED AMPLIFIER

AIM:

1. To study the Two-stage RC coupled amplifier.
2. To measure the voltage gain of the amplifier at 1KHz.
3. To obtain the frequency response characteristic and the band width of the amplifier.

EQUIPMENT:

Two stage RC coupled amplifier, trainer.

1. Signal Generator.
2. C.R.O
3. Connecting patch cords.
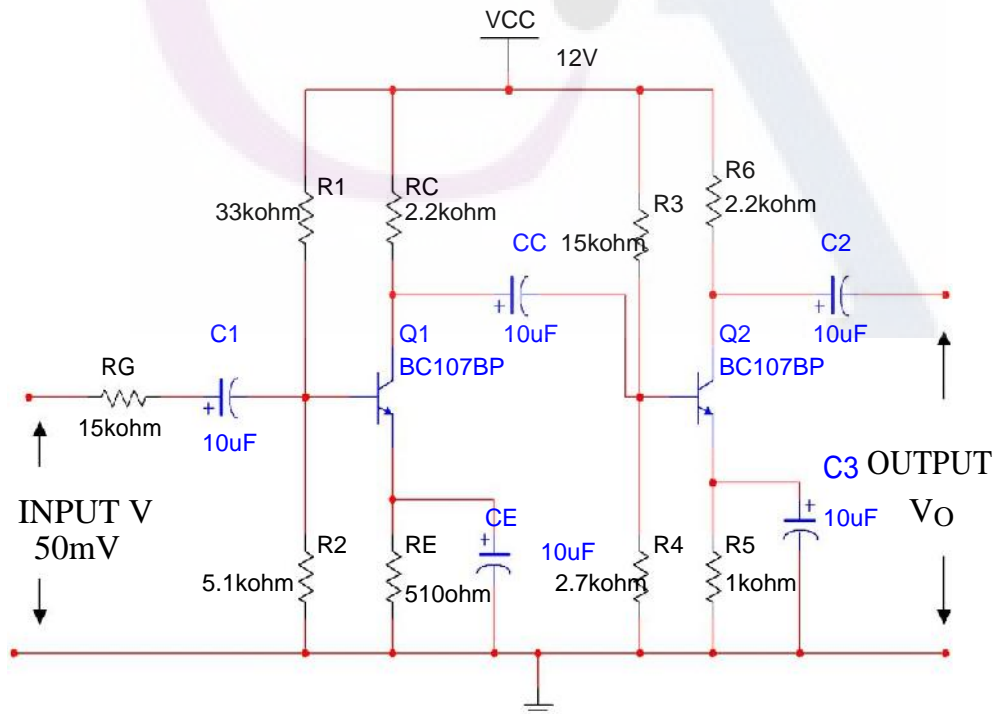
CIRCUIT DIAGRAM:

PROCEDURE:

1. Switch ON the power supply.

2. Connect the signal generator with sine wave output 50mV p-p at the input terminals.

3. Connect the C.R.O at output terminals of the module.

4. Measure the voltage at the second stage of amplifier.

5. Now vary the input frequency from 10Hz to 1MHz in steps,and for every value of input frequency note the output voltage keeping the input amplitude at constant value.

6. Calculate the gain magnitude of the amplifier using the formula

$$\text{Gain} = V_o/V_i$$

$$\text{Gain in dB} = 20 \log (V_o / V_i)$$

7. Plot a graph of frequency versus gain (dB) of the amplifier. Sample frequency response graph is as shown in fig. Below.

OBSERVATION:

$V_i = 50mV(p\text{-}p)$

Frequency	V_O	Gain $=20 \log (V_o/V_i)$dB

FREQUENCY RESPONSE:

$0.707\ V_O/V_I$

Gain V_O/V_I
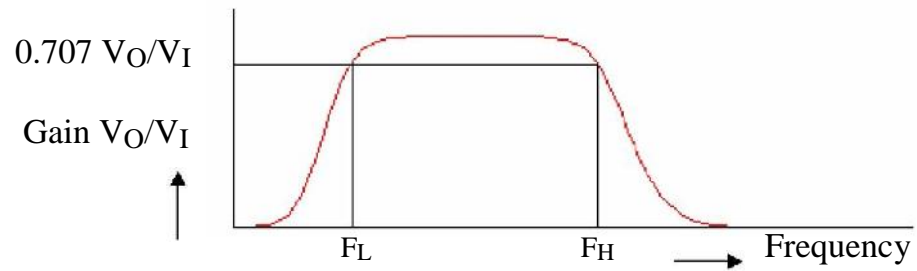
F_L F_H Frequency

RESULT:

The gain of the amplifier at 1 KHz is ------

The BW of the amplifier is -------

.CURRENT SHUNT FEEDBACK AMPLIFIER

AIM:

1. To study the current shunt feedback amplifier
2. To measure the voltage gain of the amplifier at 1KHz.

3. To obtain the frequency response characteristic and the band width

   of the amplifier.

EQUIPMENT:

Current shunt feed back amplifier trainer.

4. Signal Generator.

5. C.R.O

6. Connecting patch cords.
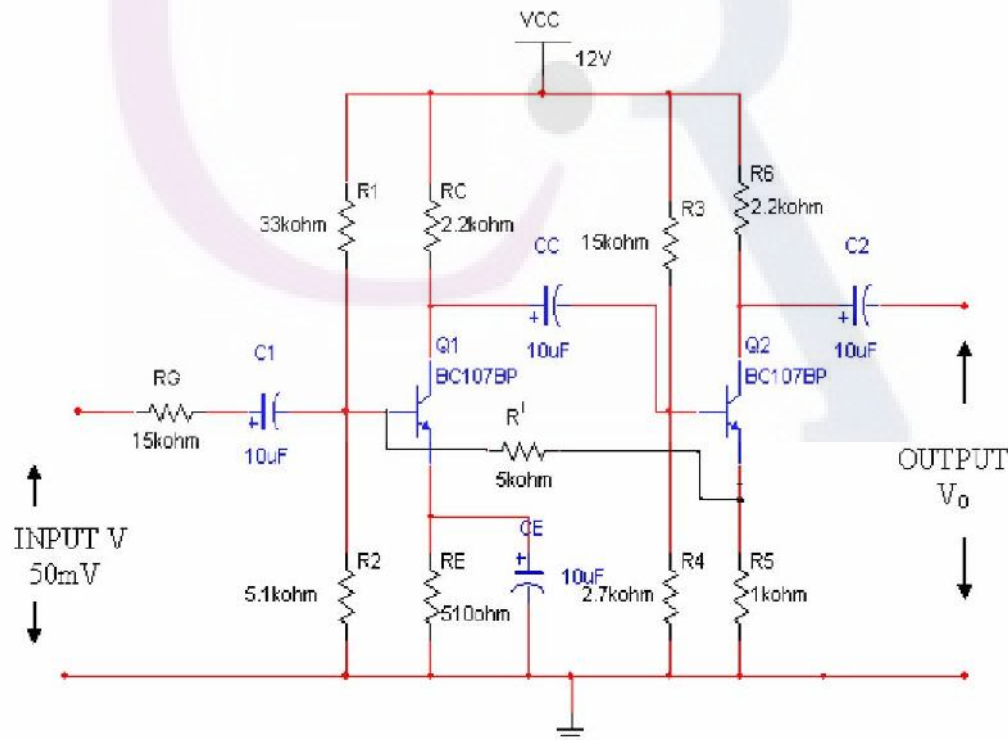
CIRCUIT DIAGRAM:

PROCEDURE:

1. Switch ON the power supply.

2. Connect the signal generator with sine wave output 50mV p-p at the input terminals.

3. Connect the C.R.O at output terminals of the module.

4. Measure the voltage at the second stage of amplifier.

5. Now vary the input frequency from 10Hz to 1MHz in steps, and for each value of input frequency note the output voltage keeping the input amplitude at constant value.

6. Calculate the gain magnitude of the amplifier using the formula

$$Gain = V_o/V_i$$

Gain in dB= $20 \log (V_o / V_i)$

7. Plot a graph of frequency versus gain (dB) of the amplifier. Sample frequency response graph is as shown in fig. Below.

OBSERVATIONS :

$V_i = 50mV(p-p)$

Frequency	V_O	Gain =20 log (V_o/V_i)dB

FREQUENCY RESPONSE:

RESULT:

The gain of the amplifier at 1 KHz is ------

The BW of the amplifier is -------

CLASS A/B/C/AB POWER AMPLIFIER

AIM: To study the operation of Class A, Class B, Class AB and Class C power amplifiers.

EQUIPMENT:

1.Class/A/B/C/AB amplifier trainer

2.Function generator.

3.C.R.O

4. Connecting patch cords.
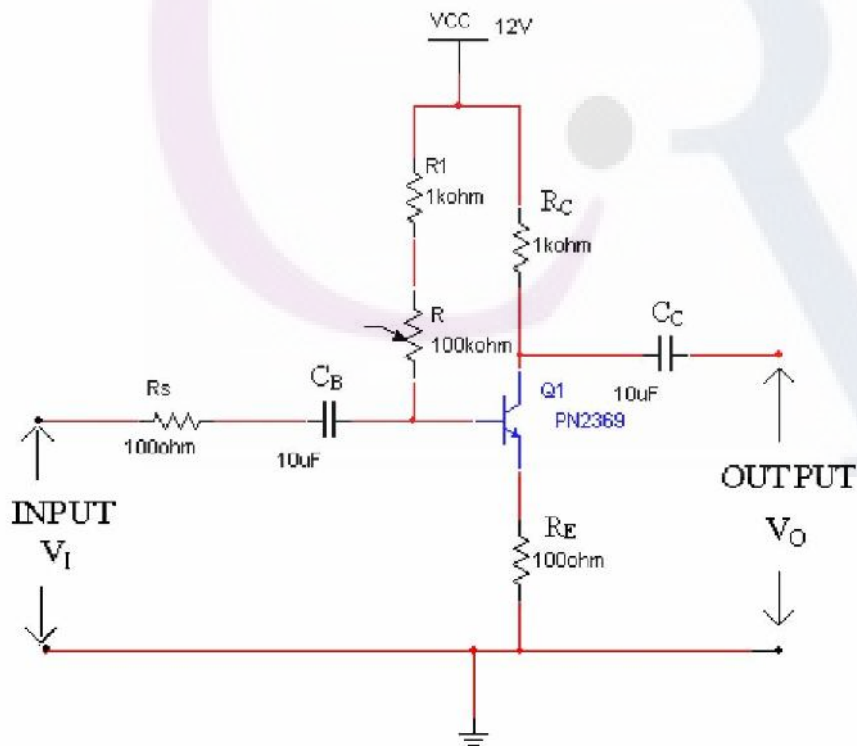
CIRCUIT DIAGRAM:

PROCEDURE:

1. Connect the circuit as shown in the circuit diagram, and get the circuit verified by your Instructor.

2. Connect the signal generator with sine wave at 1KHz and keep the amplitude at .5V (peak-to-peak)

3. Connect the C.R.O across the output terminals.

4. Now switch ON the trainer and see that the supply LED glows.

5. Keep the potentiometer at minimum position, observe and record the waveform from the C.R.O.

6. Slowly varying the potentiometer, observe the outputs for the Class A/B/AB/C amplifiers as shown in fig.

CLASS A:
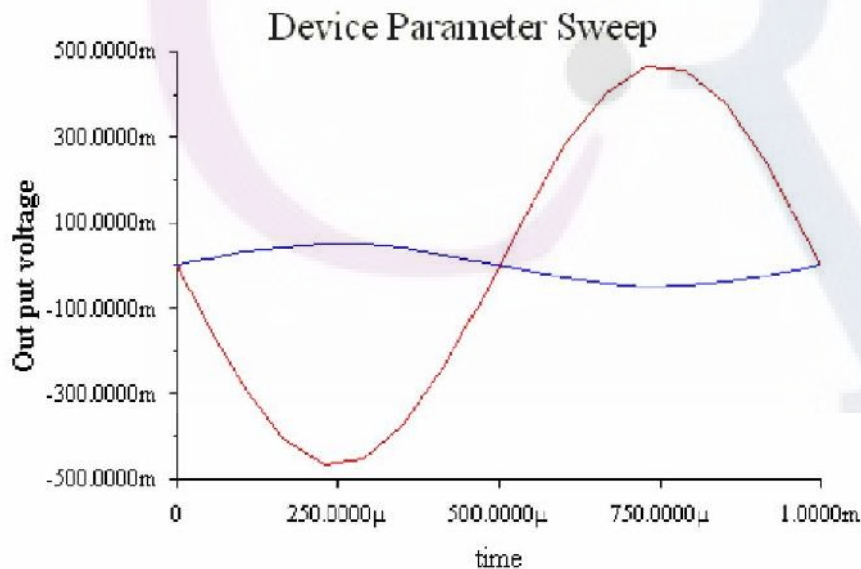
CLASS B:

CLASS AB:

CLASS C :

Device Parameter Sweep
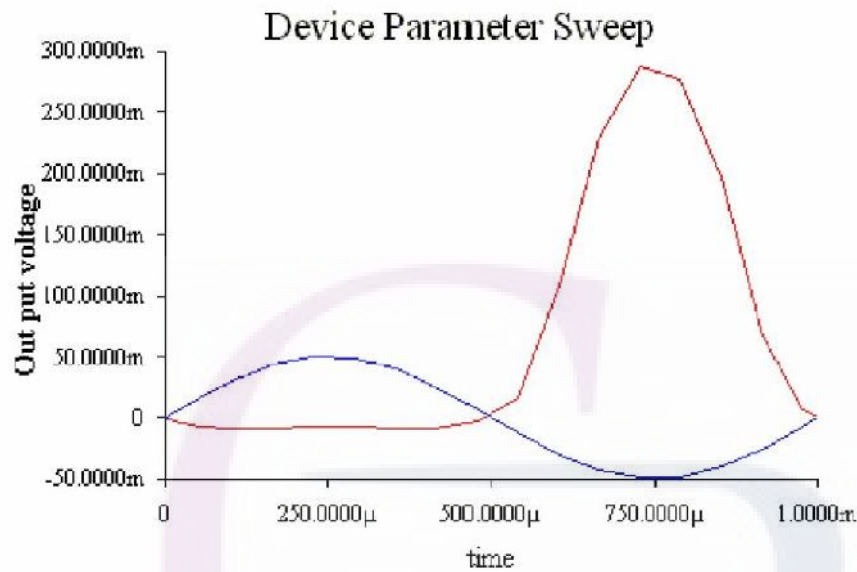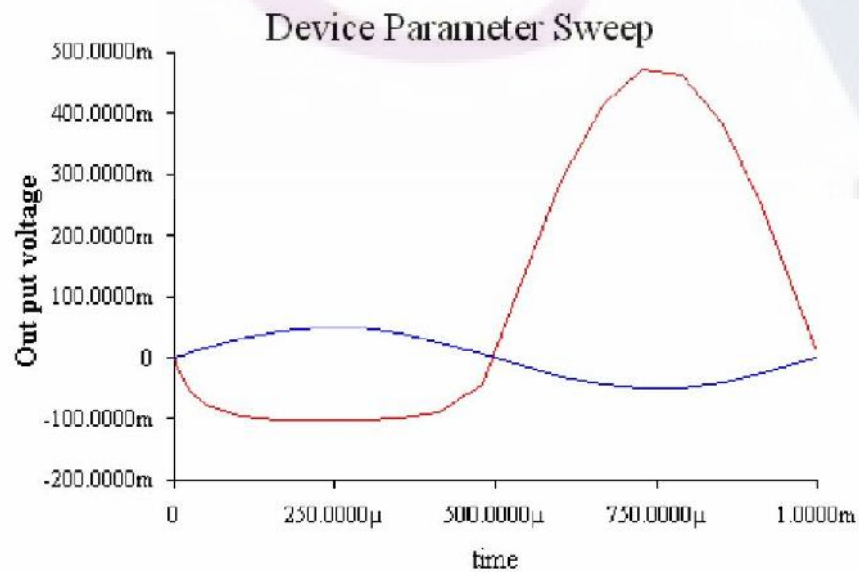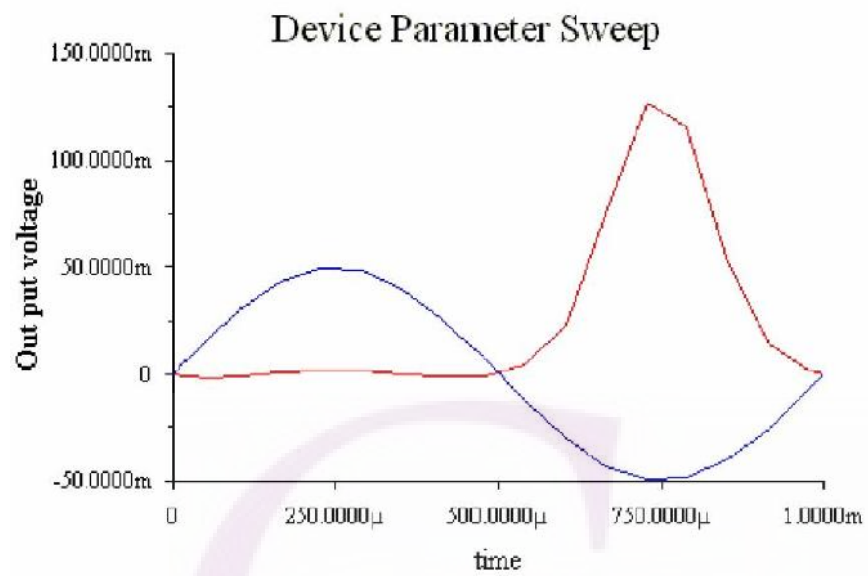
RESULT:

.SINGLE TUNED VOLTAGE AMPLIFIER

AIM: 1.To calculate the resonant frequency of tank circuit.

2. To plot the frequency response of the tuned amplifier.

EQUIPMENT:
1. Tuned voltage amplifier trainer.
2. Function generator.
3. C.R.O.
4. Connecting patch cords.

CIRCUIT DIAGRAM:
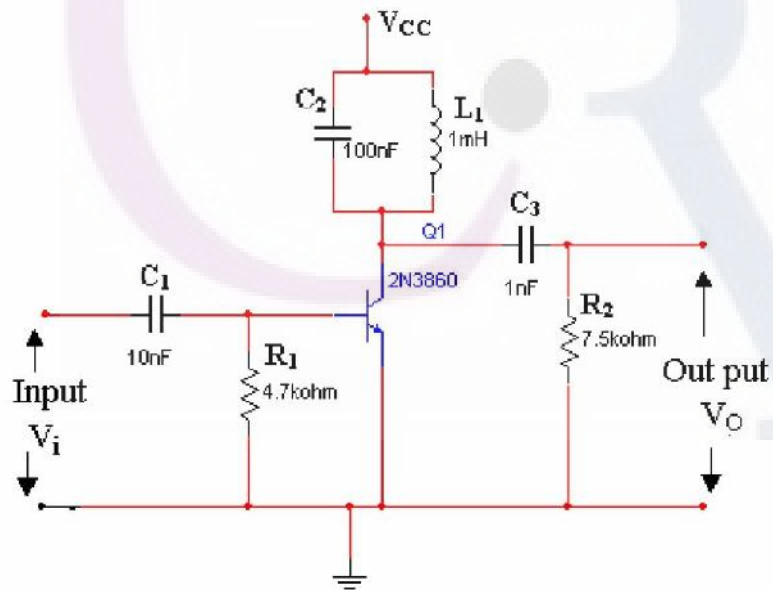
PROCEDURE:

1. Connect the circuit as shown in fig and get the circuit verified by your Instructor.

2. Connect the signal generator with sine wave at the input and keep the amplitude to minimum position, and connect a C.R.O at output terminals of the circuit.

3. Apply the amplitude between 1.6v to 4.4v to get the distortion less output sine wave.

4. Now, vary the input frequency in steps and observe and record The output voltage.

5. Calculate the gain of the tuned RF amplifier using the formula Gain = out put voltage/ input voltage.

6. plot a graph with input frequency versus gain (in dBs) Gain (in dBs) = $20 \log (V_o/V_i)$

Graph :-
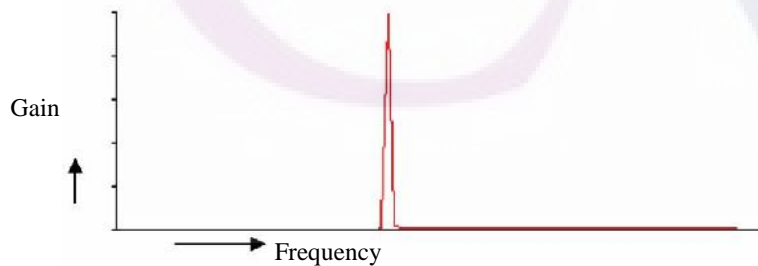
RESULT:

.HARTLEY AND COLPITT'S OSCILLATORS

AIM: To design Hartley and Colpitt's Oscillators to have resonant frequency of 1KHz.

APPARATUS:

BJT(BC107),Resistors(2.2kΩ,100kΩ,10kΩ,1kΩ),

Capacitors(10µf,100µf,0.33 µf), Decade inductance box ,RPS.

EQUIPMENT:
1. SDC kit.
2. Function generator.
3. C.R.O.

DESIGN PROCEDURE:

Hartley Oscillator
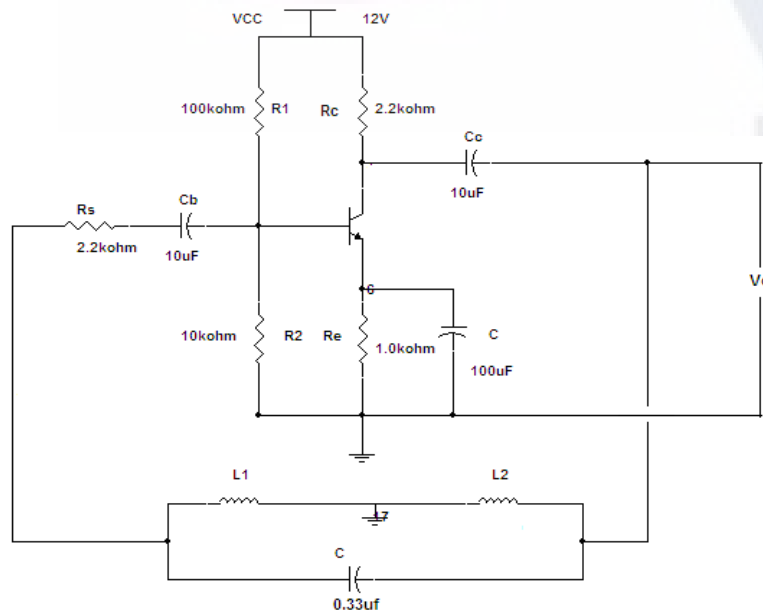$$F = 1 / (2\pi\sqrt{LeqC})$$
Where Leq=L1+L2
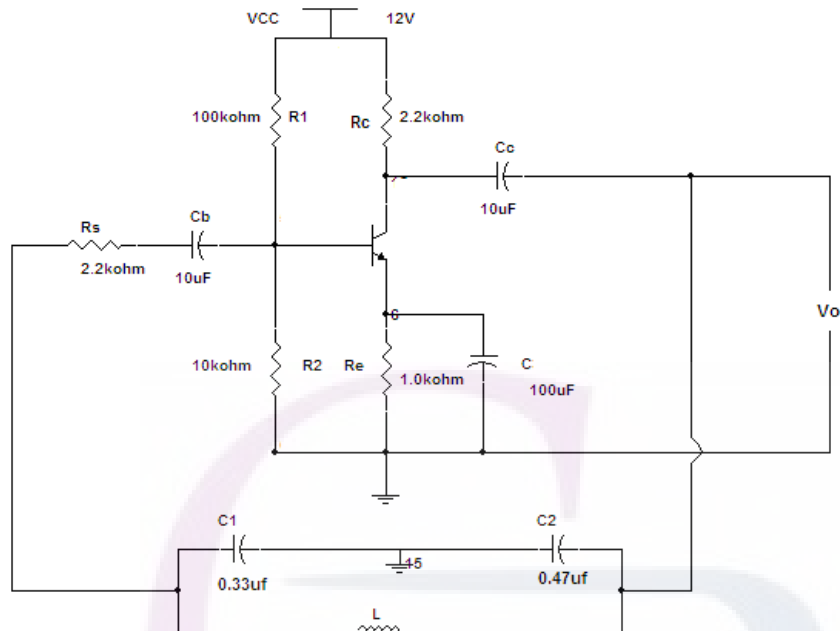
Colpitt's Oscillator
$$F = 1 / (2\pi\sqrt{LCeq})$$
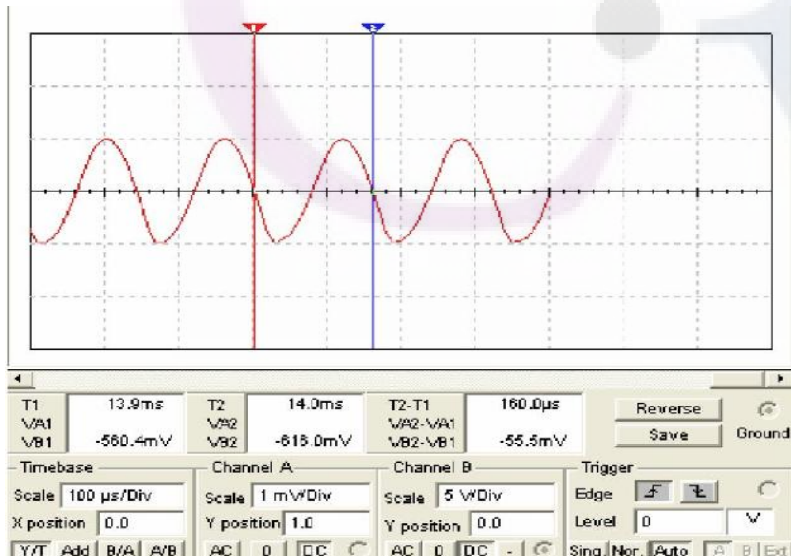Where Ceq= (c1*c2) / (c1+c2)

CIRCUIT DIAGRAMS:
HARTLEY OSCILLATOR:

COLPITT'S OSCILLATOR:

EXPECTED WAVEFORM:

RESULT:

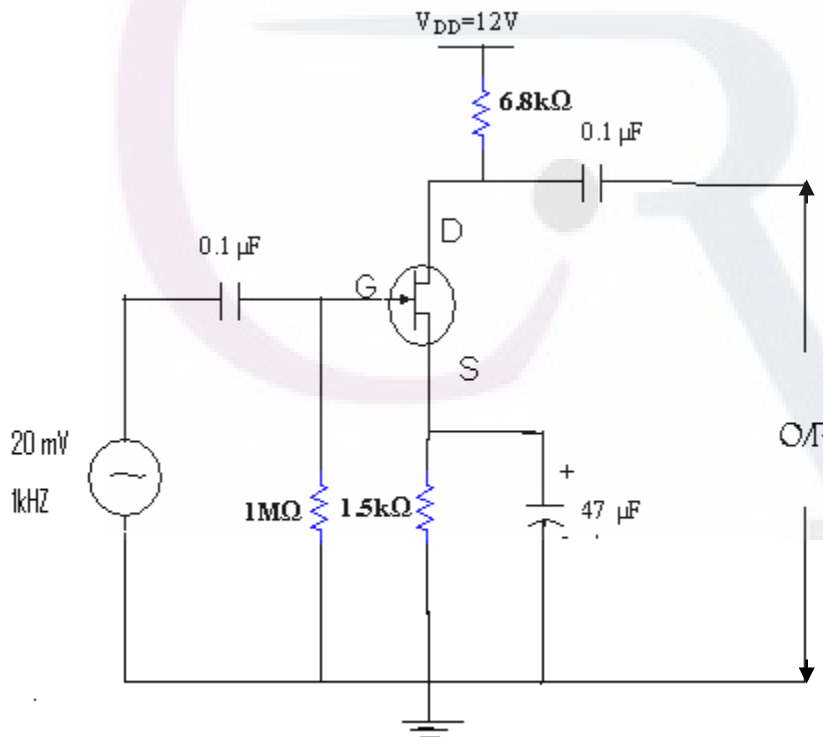# .COMMON SOURCE FET AMPLIFIER

AIM: 1. To obtain the frequency response of the common source FET Amplifier

     2. To find the Bandwidth.

APPRATUS:

N-channel FET (BFW11),Resistors (6.8KΩ, 1MΩ, 1.5KΩ),Capacitors (0.1µF, 47µF)

Regulated power Supply (0-30V), Function generator, CRO,CRO probes,Bread board, Connecting wires

CIRCUIT DIAGRAM:

PROCEDURE:

1. Connections are made as per the circuit diagram.

2. A signal of 1 KHz frequency and 50mV peak-to-peak is applied at the
   Input of amplifier.

3. Output is taken at drain and gain is calculated by using the expression,

$$A_v = V_0/V_i$$

4. Voltage gain in dB is calculated by using the expression,

$$A_v = 20\log 10(V0/V_i)$$

5. Repeat the above steps for various input voltages.

6. Plot A_v vs. Frequency

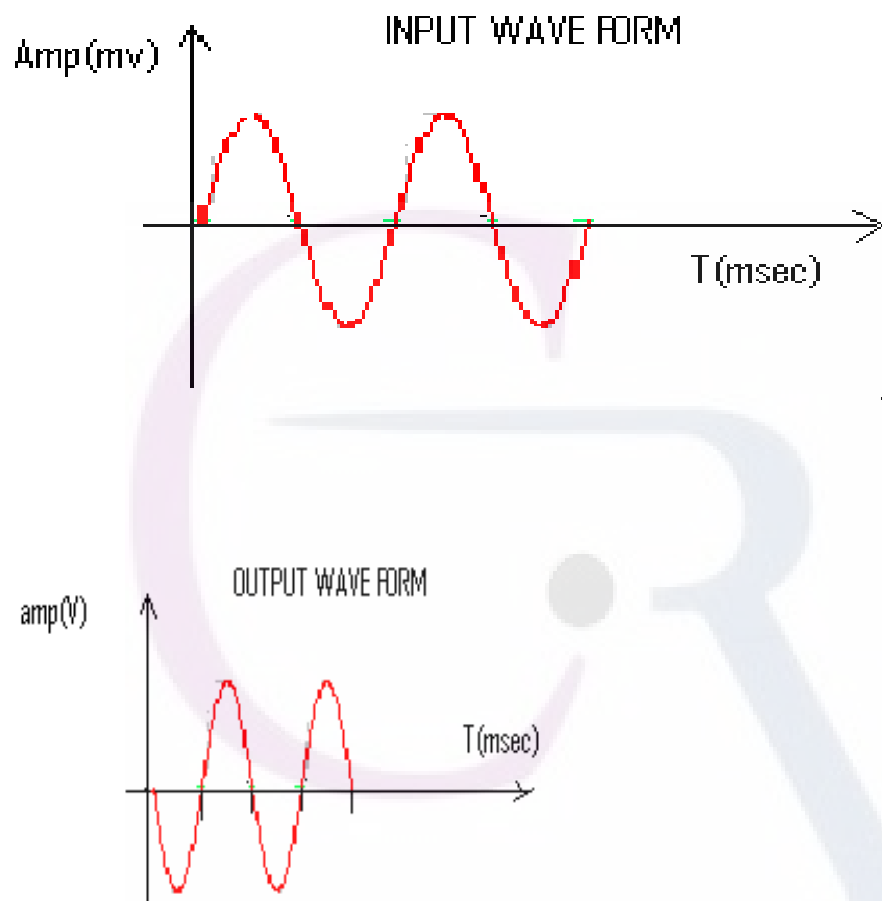7. The Bandwidth of the amplifier is calculated from the graph using the
   Expression,

$$\text{Bandwidth } BW = f_2 - f_1$$

Where f_1 is lower 3 dB frequency, f_2 is upper 3 dB frequency

OBSERVATIONS:

S.NO	INPUT VOLTAGE(V_i)	OUTPUT VOLTAGE(V_0)	VOLTAGE GAIN $A_v = (V0/V_i)$

MODEL GRAPH:

INPUT WAVE FORM

Amp(mv)

T(msec)

OUTPUT WAVE FORM

amp(V)
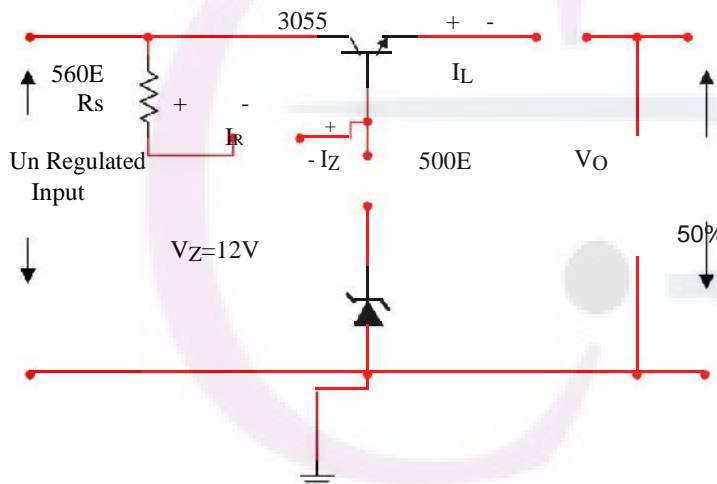
T(msec)

.

RESULT:

SERIES VOLTAGE REGULATOR

AIM : To study and design a Series voltage regulator and to observe the load regulation feature.

EQUIPMENT :

1. Series voltage regulated power supply trainer.
2. Multimeter.
3. Patch chords

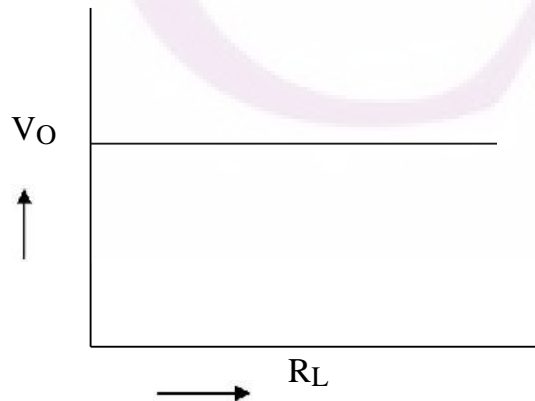## CIRCUIT DIAGRAM:

PROCEDURE:

1. Switch ON the power supply.
2. Observe the Unregulated voltage at the output of rectifier.
3. Connect this voltage to the input of series voltage regulator circuit.
4. Keep the load resistance 1K at constant.
5. Observe the output voltage $V_O = V_Z - V_{BE}$

6. And also observe the voltage across R_S, and values of I_R, I_L and I_Z.

7. Compare the practical values with theoretical values.

8. By changing the load resistance, observe the output voltage and various currents.

OBSERVATIONS:

R_L	V_O	I_R	I_Z	I_L

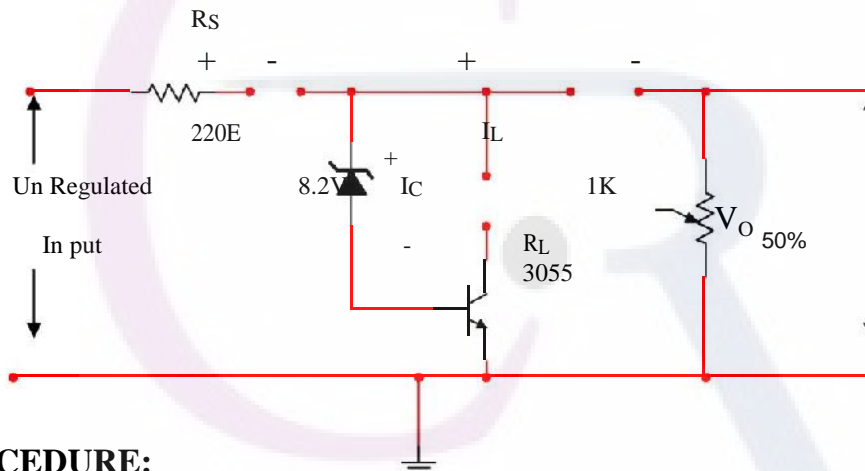**LOAD REGULATION :**

RESULT:

SHUNT VOLTAGE REGULATOR

AIM : To study and design a Shunt Regulator and to observe the load regulation feature.

EQUIPMENT :

1. Shunt regulated power supply trainer.
2. Multimeter.
3. Patch chords.

CIRCUIT DIAGRAM:

PROCEDURE:

1. Switch On the main power supply.

2. Observe the unregulated voltage at the output of rectifier.

3. Connect this voltage to the input of shunt Regulator circuit

4 .Keep the load resistance 1K constant.

5. Observe the output voltage across the load resistor $V_0 = V_Z + V_{BE}$