EXPERIMENT-1

Common Emitter Amplifier.

AIM: To design and determine the frequency of operation of a Common-Emitter Amplifier using simulation software Tina Ver. 7 or Ver 9 or PSpice Version 9.1

APPARATUS

1. Tina or PSpice software.
2. Personal computer.

COMPONENTS AND EQUIPMENTS

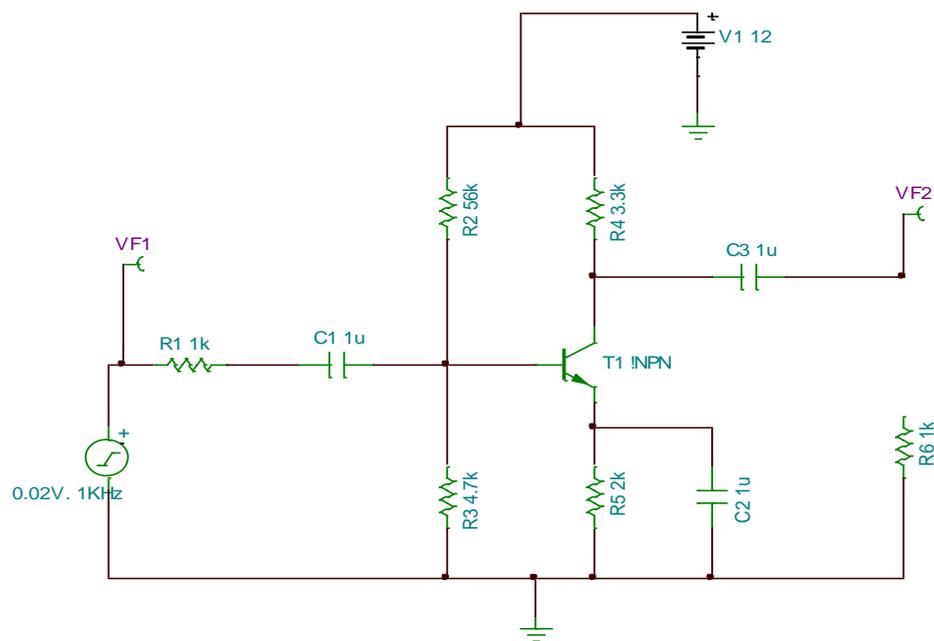| S. No.	Description	Part No./Value	Quantity
1	DC voltage Source	12V	1
2	Function Generator	0.02V, 1KHz	1
3	Transistor	BC107	1
	Resistors	56KΩ, 4.7 KΩ, 3.3 KΩ, 2KΩ, 1 KΩ-2	5
4	Capacitors	1 µF(3)	3

CIRCUIT DIAGRAM

Fig-1: CE Amplifier

PROCEDURE

Drawing the circuit schematic

1. Open the Tina Schematics Editor page.
2. Click the Semiconductors tab and pick and place an NPN transistor on the centre of the page. Double click the transistor and set the part no. as BC107.
3. Next, click the Basic tab and pick and place resistors, capacitors, battery and the signal generator in convenient locations following the schematic of Fig. 1. (To rid the cursor of the component after placing it on the editor page, click anywhere on the page else, right click and select "Cancel Mode"). Then, place the ground symbol from the Basic tab and a Voltage pin from the Meters tab.
4. Double-click the resistors, capacitors and transistors in succession and set the parameter value of each component in the component dialog box. For transistors ensure that the right part no. is selected. **Note:** For megohms, use "M", for milliohms use "m", for microfarad use "u" and for nanofarad use "n". Connect the components using the Wire symbol in the menu bar, according to the schematic in Fig. 1. Then, place a single pole single throw (SPST or ON-OFF) switch from the Switches tab. Set the switch in ON position.
5. Double-click the power supply symbol and set to 25V DC. Next, double-click the signal generator symbol and enter DC = 0V, Signal (select Sinusoidal) AC = 20millivolts by entering 20m and set the frequency to 10kHz by entering 10k.
    Check the node voltages
6. The Electric Rules Check (ERC) is performed by clicking Analysis in the menubar and then clicking ERC. This will show up errors and warning messages if the schematic is not proper. Correct the schematic based on the summary displayed by the ERC.
7. Next, record the DC node voltages V_{CE}, across R_c and across R_e and current I_C by clicking Analysis →DC Analysis →Table of DC Results.

    Frequency Response with the switch S1 in ON position.
8. **In PSpice:** Click the "Analysis" in the menu bar and then click "Setup" and select "AC Sweep" and set the starting frequency to 10 Hz and end frequency to 1 GHz and select radio button "Decade" and then close the box.
    Then, click "Analysis: and click "Simulate" and when the Analysis dialog box opens, select "AC". Select "Traces" button in the taskbar and click "Delete all traces" and then select "Adding Traces" and then select [B].

9. **In TINA :** Click "Analysis" in the menu bar and then click "AC Analysis" and then "AC Transfer Characteristic" and set the starting frequency to 10 Hz and end frequency to 1 GHz. Set the Sweep type to "Logarithmic" and click OK.
    A graph of gain in dB vs. Frequency in Hz will be displayed. Sketch the graph obtained on graph paper of *only* the response with the switch in ON position.

10. Click "Analysis" in the menu bar and then click "AC Analysis" and then "Table of AC Results" and note the voltage of the generator v_i, the voltage at the amplifier output v_o, and calculate the gain in dB using the expression $20 \log v_o / v_i$ and verify that it tallies with the value in the graph at 10kHz.
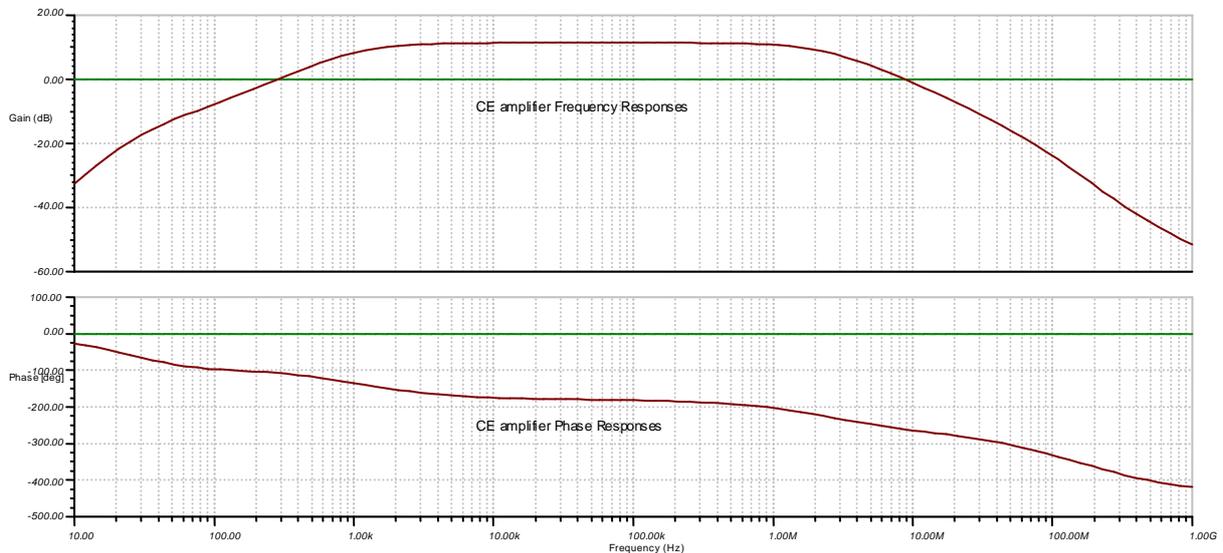
    Sketch the output characteristics and draw the DC load line and mark the quiescent point of operation indicating I_C and I_B at this point from the values recorded in the previous step in DC Analysis.

11. Next, set the switch to OFF position i.e with the bypass capacitor disconnected from across the emitter resistor and run step 6. Note down the gain (in dB) at 10 KHz.

    Additional exercise

12. Set the switch again to ON position.

13. From the menu, double click the signal generator and set the input sinusoidal frequency to 10kHz. On the menu bar click Test & Measurement (T&M) and click "Oscilloscope".

14. Click Run and then Auto (bar at the bottom of the virtual oscilloscope, and set connection to AC) and set the V/div and Time/div till a clean sinusoidal trace appears. Then press the "stop" button to freeze the oscilloscope trace and using the B button in the Cursor box and with the ON button pressed in the Cursor box, measure the amplitude of the traces taking care with the V/div mentioned at the top of the screen.

15. Next, change the amplitude of the signal generator to say, 500mV and again obtain a trace and record your observations of the trace.

MODEL GRAPH

RESULT

1. The frequency response of the Common-emitter amplifier is measured and the graph plotted.

2. From the graph.

  i.   The gain at $10\,kHz$ = ……..dB.

  ii.   The lower cutoff frequency corresponding to a gain $20\log\left[\dfrac{v_o}{v_i}\right]$ dB -3dB = ……Hz.

  iii. The upper cutoff frequency corresponding to a gain $20\log\left[\dfrac{v_o}{v_i}\right]$ dB -3dB = ……Hz.

  iv.   iv). A sketch of the oscilloscope display of the response of the amplifier at 10 KHz frequency is made.

EXPERIMENT-2

Common Base Amplifier.

AIM: To design and determine the frequency of operation of a Common-Base Amplifier using simulation software Tina Ver. 7 or Ver 9 or PSpice Version 9.1

APPARATUS

1. Tina or PSpice software.
2. Personal computer.

COMPONENTS AND EQUIPMENTS

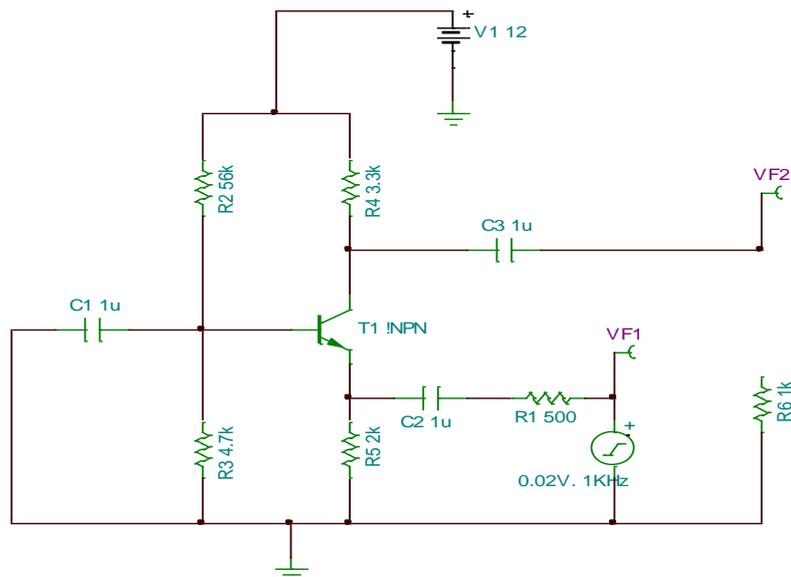| S. No.	Description	Part No./Value	Quantity
1	DC voltage Source	12V	1
2	Function Generator	0.02V, 1KHz	1
3	Transistor	BC107	1
	Resistors	56KΩ, 4.7 KΩ, 3.3 KΩ, 2KΩ, 1 KΩ-2	5
4	Capacitors	1 µF(3)	3

CIRCUIT DIAGRAM

Fig-2: CB Amplifier

PROCEDURE

Drawing the circuit schematic

1. Open the Tina Schematics Editor page.
2. Click the Semiconductors tab and pick and place an NPN transistor on the centre of the page. Double click the transistor and set the part no. as BC107.
3. Next, click the Basic tab and pick and place resistors, capacitors, battery and the signal generator in convenient locations following the schematic of Fig. 1. (To rid the cursor of the component after placing it on the editor page, click anywhere on the page else, right click and select "Cancel Mode"). Then, place the ground symbol from the Basic tab and a Voltage pin from the Meters tab.
4. Double-click the resistors, capacitors and transistors in succession and set the parameter value of each component in the component dialog box. For transistors ensure that the right part no. is selected. **Note:** For megohms, use "M", for milliohms use "m", for microfarad use "u" and for nanofarad use "n". Connect the components using the Wire symbol in the menu bar, according to the schematic in Fig. 1. Then, place a single pole single throw (SPST or ON-OFF) switch from the Switches tab. Set the switch in ON position.
5. Double-click the power supply symbol and set to 25V DC. Next, double-click the signal generator symbol and enter DC = 0V, Signal (select Sinusoidal) AC = 20millivolts by entering 20m and set the frequency to 10kHz by entering 10k.
   Check the node voltages
6. The Electric Rules Check (ERC) is performed by clicking Analysis in the menubar and then clicking ERC. This will show up errors and warning messages if the schematic is not proper. Correct the schematic based on the summary displayed by the ERC.
7. Next, record the DC node voltages V_{CE}, across R_c and across R_e and current I_C by clicking Analysis →DC Analysis →Table of DC Results.

   Frequency Response with the switch S1 in ON position.
8. **In PSpice:** Click the "Analysis" in the menu bar and then click "Setup" and select "AC Sweep" and set the starting frequency to 10 Hz and end frequency to 1 GHz and select radio button "Decade" and then close the box.
   Then, click "Analysis: and click "Simulate" and when the Analysis dialog box opens, select "AC". Select "Traces" button in the taskbar and click "Delete all traces" and then select "Adding Traces" and then select [B].

9. **In TINA :** Click "Analysis" in the menu bar and then click "AC Analysis" and then "AC Transfer Characteristic" and set the starting frequency to 10 Hz and end frequency to 1 GHz. Set the Sweep type to "Logarithmic" and click OK.
   A graph of gain in dB vs. Frequency in Hz will be displayed. Sketch the graph obtained on graph paper of *only* the response with the switch in ON position.

10. Click "Analysis" in the menu bar and then click "AC Analysis" and then "Table of AC Results" and note the voltage of the generator v_i, the voltage at the amplifier output v_o, and calculate the gain in dB using the expression 20 log $v_o/$ v_i and verify that it tallies with the value in the graph at 10kHz.
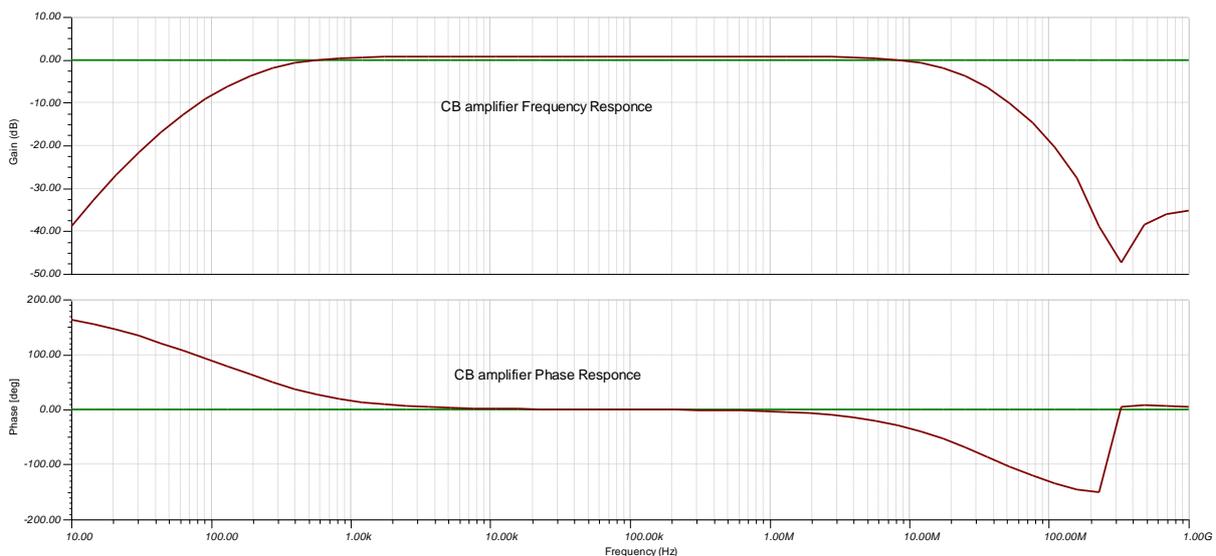
Sketch the output characteristics and draw the DC load line and mark the quiescent point of operation indicating I_C and I_B at this point from the values recorded in the previous step in DC Analysis.

11. Next, set the switch to OFF position i.e with the bypass capacitor disconnected from across the emitter resistor and run step 6. Note down the gain (in dB) at 10 KHz.

Additional exercise

12. Set the switch again to ON position.
13. From the menu, double click the signal generator and set the input sinusoidal frequency to 10kHz. On the menu bar click Test & Measurement (T&M) and click "Oscilloscope".
14. Click Run and then Auto (bar at the bottom of the virtual oscilloscope, and set connection to AC) and set the V/div and Time/div till a clean sinusoidal trace appears. Then press the "stop" button to freeze the oscilloscope trace and using the B button in the Cursor box and with the ON button pressed in the Cursor box, measure the amplitude of the traces taking care with the V/div mentioned at the top of the screen.
15. Next, change the amplitude of the signal generator to say, 500mV and again obtain a trace and record your observations of the trace.

MODEL GRAPH

RESULT

1. The frequency response of the Common-Base amplifier is measured and the graph plotted.

2. From the graph.

i) The gain at $10kHz = $ …….dB.

ii) The lower cutoff frequency corresponding to a gain $20\log\left[\dfrac{v_o}{v_i}\right]$ dB -3dB = ……Hz.

iii) The upper cutoff frequency corresponding to a gain $20\log\left[\dfrac{v_o}{v_i}\right]$ dB -3dB = ……Hz.

iv). A sketch of the oscilloscope display of the response of the amplifier at 10 KHz frequency is made.

EXPERIMENT-3

Common Source Amplifier.

AIM: To design and determine the frequency of operation of a Common-Source Amplifier using simulation software Tina Ver. 7 or Ver 9 or PSpice Ver. 9.1.
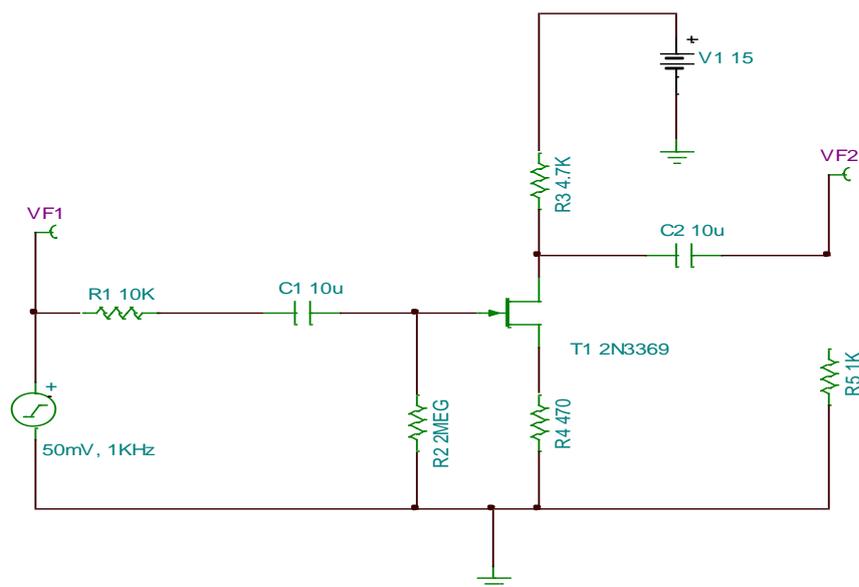
APPARATUS

1. Tina or PSpice software.
2. Personal computer.

COMPONENTS AND EQUIPMENTS

S. No.	Description	Part No./Value	Quantity
1	DC voltage Source	15V	1
2	Function Generator	0.05V, 1KHz	1
3	Transistor	2N3369	1
	Resistors	10KΩ, 4.7 KΩ, 470Ω, 2MΩ, 1 KΩ	5
4	Capacitors	10 μF(2)	3

CIRCUIT DIAGRAM

Fig-3: CS Amplifier

PROCEDURE

Drawing the circuit schematic

1. Open the Tina Schematics Editor page.
2. Click the Semiconductors tab and pick and place a "Junction FET N-Channel" transistor on the centre of the page. Double click the transistor and set the part no. as 2N4393.
3. Next, click the Basic tab and pick and place resistors, capacitors, battery and the signal generator in convenient locations following the schematic of Fig. 1. (To rid the cursor of the component after placing it on the editor page, click anywhere on the page else, right click and select "Cancel Mode"). Then, place the ground symbol from the Basic tab and a Voltage pin from the Meters tab.
4. Double-click the resistors, capacitors and transistors in succession and set the parameter value of each component in the component dialog box. For transistors ensure that the right part no. is selected. **Note:** For megohms, use "M", for milliohms use "m", for microfarad use "u" and for nanofarad use "n". Connect the components using the Wire symbol in the menu bar, according to the schematic in Fig. 1. Then, place a single pole single throw (SPST or ON-OFF) switch from the Switches tab. Set the switch in ON position.
5. Double-click the power supply symbol and set to 25V DC. Next, double-click the signal generator symbol and enter DC = 0V, Signal (select Sinusoidal) AC = 20millivolts by entering 20m and set the frequency to 10kHz by entering 10k.
   Check the node voltages
6. The Electric Rules Check (ERC) is performed by clicking Analysis in the menubar and then clicking ERC. This will show up errors and warning messages if the schematic is not proper. Correct the schematic based on the summary displayed by the ERC.
7. Next, record the DC node voltages V_{CE}, across R_c and across R_e and current I_C by clicking Analysis →DC Analysis →Table of DC Results.

   Frequency Response with the switch S1 in ON position.
8. **In PSpice:** Click the "Analysis" in the menu bar and then click "Setup" and select "AC Sweep" and set the starting frequency to 10 Hz and end frequency to 1 GHz and select radio button "Decade" and then close the box.
   Then, click "Analysis: and click "Simulate" and when the Analysis dialog box opens, select "AC". Select "Traces" button in the taskbar and click "Delete all traces" and then select "Adding Traces" and then select [dB].

9. **In TINA :** Click "Analysis" in the menu bar and then click "AC Analysis" and then "AC Transfer Characteristic" and set the starting frequency to 10 Hz and end frequency to 1 GHz. Set the Sweep type to "Logarithmic" and click OK.
   A graph of gain in dB vs. Frequency in Hz will be displayed. Sketch the graph obtained on graph paper of *only* the response with the switch in ON position.

10. Click "Analysis" in the menu bar and then click "AC Analysis" and then "Table of AC Results" and note the voltage of the generator v_i, the voltage at the amplifier output v_o, and calculate the gain in dB using the expression $20 \log v_o/ v_i$ and verify that it tallies with the value in the graph at 10kHz.
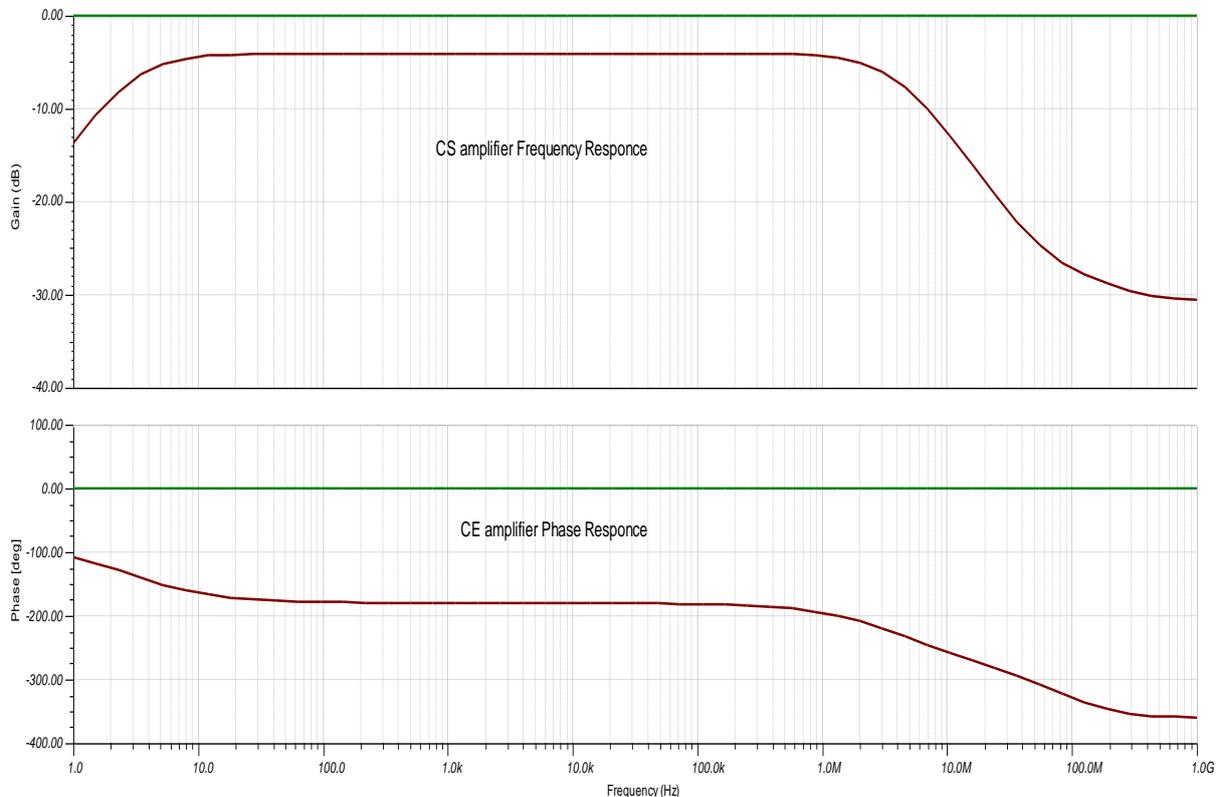
Sketch the output characteristics and draw the DC load line and mark the quiescent point of operation indicating I_C and I_B at this point from the values recorded in the previous step in DC Analysis.

11. Next, set the switch to OFF position i.e with the bypass capacitor disconnected from across the emitter resistor and run step 6. Note down the gain (in dB) at 10 KHz.

Additional exercise

12. Set the switch again to ON position.

13. From the menu, double click the signal generator and set the input sinusoidal frequency to 10kHz. On the menu bar click Test & Measurement (T&M) and click "Oscilloscope".

14. Click Run and then Auto (bar at the bottom of the virtual oscilloscope, and set connection to AC) and set the V/div and Time/div till a clean sinusoidal trace appears. Then press the "stop" button to freeze the oscilloscope trace and using the B button in the Cursor box and with the ON button pressed in the Cursor box, measure the amplitude of the traces taking care with the V/div mentioned at the top of the screen.

15. Next, change the amplitude of the signal generator to say, 500mV and again obtain a trace and record your observations of the trace.

MODEL GRAPH

RESULT

1.The frequency response of the FET common-source amplifier is simulated and the graph plotted.

2. From the graph

i) The gain of the amplifier at 10 kHz = ………..dB

ii) The lower cutoff frequency corresponding to a gain $20\log\left[\dfrac{v_o}{v_i}\right]$ dB -3dB =……Hz.

iii) The upper cutoff frequency corresponding to a gain $20\log\left[\dfrac{v_o}{v_i}\right]$ dB -3dB =……Hz

EXPERIMENT-4

Two Stage RC Coupled Amplifier.

AIM: To design and determine the gain of the first stage and overall gain of a two-stage RC coupled amplifier using simulation software Tina Ver. 7 or Ver 9 or PSpice Ver. 9.1.
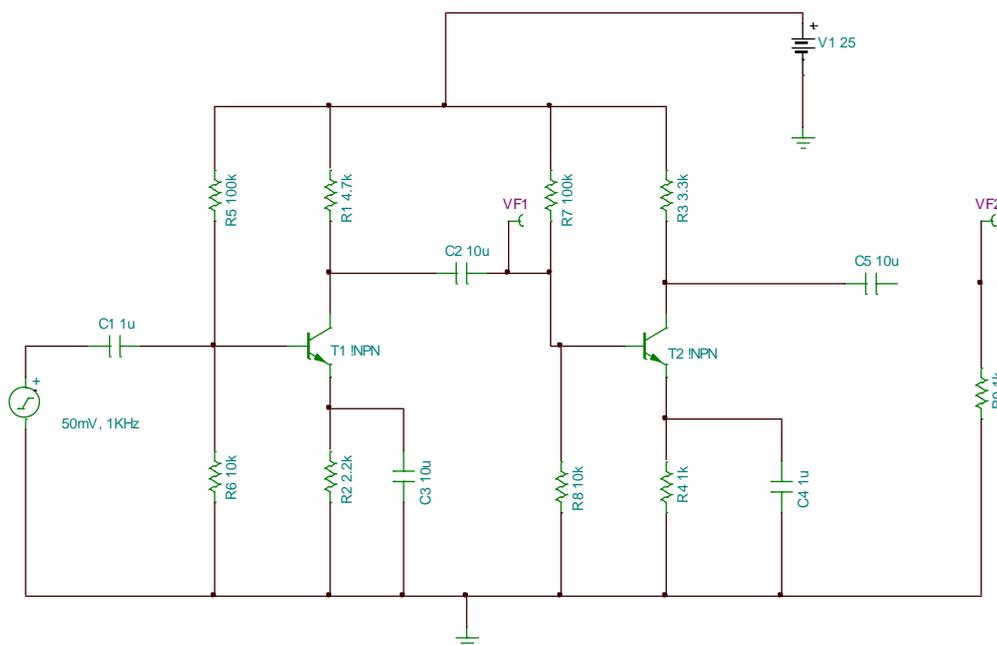
APPARATUS

1. Tina or PSpice software.
2. Personal computer.

COMPONENTS AND EQUIPMENTS

S. No.	Description	Part No./Value	Quantity
1	DC voltage Source	12V	1
2	Function Generator	0.05V, 1KHz	1
3	Transistor	2N3369	2
	Resistors (Ω)	33K, 10K(2),6.8 K(2), 2.2K(2), 1 K(2)	9
4	Capacitors	10 μF	5

CIRCUIT DIAGRAM

Fig-4: Two stage RC- Coupled Amplifier

PROCEDURE

Drawing the circuit schematic

1. Open the Tina Schematics Editor page.
2. Click the Semiconductors tab and pick and place two NPN transistors on the centre of the page. Double click the transistors and set their part nos. as BC107.
3. Next, click the Basic tab and pick and place resistors, capacitors, battery and the signal generator in convenient locations following the schematic of Fig. 1. (To rid the cursor of the component after placing it on the editor page, click anywhere on the page else, right click and select "Cancel Mode"). Then, place the ground symbol from the Basic tab and a Voltage pin from the Meters tab.
4. Double-click the resistors, capacitors and transistors in succession and set the parameter value of each component in the component dialog box. For transistors ensure that the right part no. is selected. **Note:** For megohms, use "M", for milliohms use "m", for microfarad use "u" and for nanofarad use "n". Connect the components using the Wire symbol in the menu bar, according to the schematic in Fig. 1. Then, place a single pole single throw (SPST or ON-OFF) switch from the Switches tab. Set the switch in ON position.
5. Double-click the power supply symbol and set to 25V DC. Next, double-click the signal generator symbol and enter DC = 0V, Signal (select Sinusoidal) AC = 20millivolts by entering 20m and set the frequency to 10kHz by entering 10k.
   Check the node voltages
6. The Electric Rules Check (ERC) is performed by clicking Analysis in the menubar and then clicking ERC. This will show up errors and warning messages if the schematic is not proper. Correct the schematic based on the summary displayed by the ERC.
7. Next, record the DC node voltages V_{CE}, across R_c and across R_e and current I_C by clicking Analysis →DC Analysis →Table of DC Results.

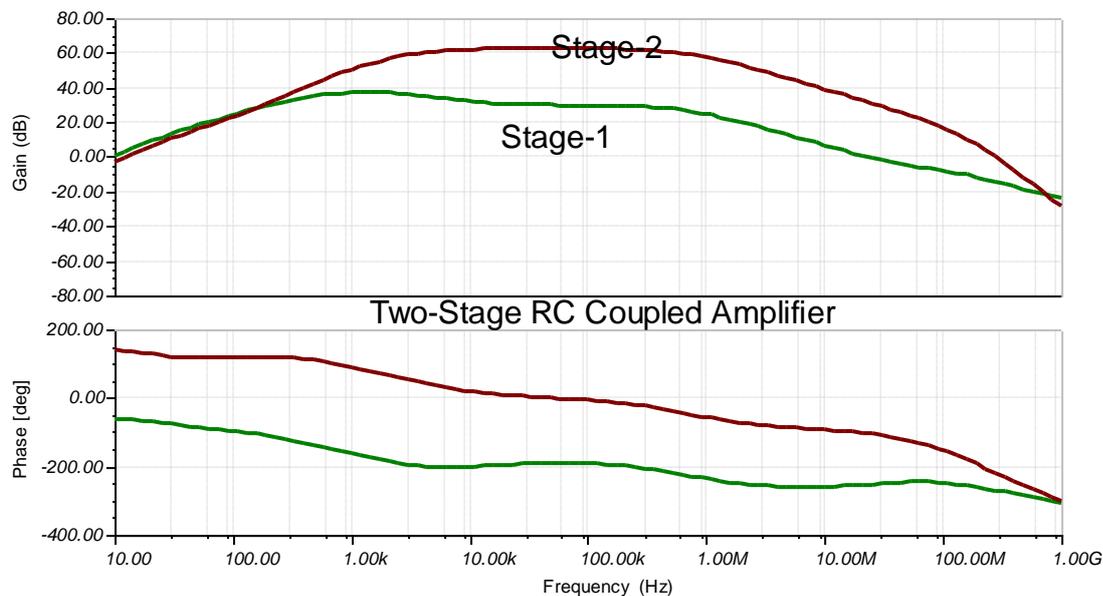   **Frequency Response with the switch S1 in ON position.**
8. **In PSpice:** Click the "Analysis" in the menu bar and then click "Setup" and select "AC Sweep" and set the starting frequency to 10 Hz and end frequency to 1 GHz and select radio button "Decade" and then close the box.
   Then, click "Analysis: and click "Simulate" and when the Analysis dialog box opens, select "AC". Select "Traces" button in the taskbar and click "Delete all traces" and then select "Adding Traces" and then select [dB].

9. **In TINA :** Click "Analysis" in the menu bar and then click "AC Analysis" and then "AC Transfer Characteristic" and set the starting frequency to 10 Hz and end frequency to 1 GHz. Set the Sweep type to "Logarithmic" and click OK.
   A graph of gain in dB vs. Frequency in Hz will be displayed. Sketch the graph obtained on graph paper of *only* the response with the switch in ON position.

10. Click "Analysis" in the menu bar and then click "AC Analysis" and then "Table of AC Results" and note the voltage of the generator v_i, the voltage at the amplifier output v_o, and calculate the gain in dB using the expression $20 \log v_o/ v_i$ and verify that it tallies with the value in the graph at 10kHz.

Sketch the output characteristics and draw the DC load line and mark the quiescent point of operation indicating I_C and I_B at this point from the values recorded in the previous step in DC Analysis.

11. Next, set the switch to OFF position i.e with the bypass capacitor disconnected from across the emitter resistor and run step 6. Note down the gain (in dB) at 10 KHz.

MODEL GRAPH

RESULT

1. The frequency response of the Two stage RC coupled amplifier is measured and the graph plotted.

2. From the plot of overall gain vs. frequency

i) The gain of the first stage of the amplifier at 10 kHz = ………..dB

ii) The gain of the second stage of the amplifier at 10 kHz = ………..dB

iii) The overall gain of the amplifier $20\log \left[\dfrac{v_{o2}}{v_{i1}} \right]$ at 10 kHz = ……….dB

iv) The lower cutoff frequency corresponding to a gain $20\log \left[\dfrac{v_{o2}}{v_{i1}} \right]$ dB -3dB = ……Hz.

v) The upper cutoff frequency corresponding to a gain $20\log \left[\dfrac{v_{o2}}{v_{i1}} \right]$ dB -3dB = ……Hz.

EXPERIMENT-5

Current Shunt Feedback Amplifier.

AIM: To design and determine the frequency of operation of a Current Shunt Feedback Amplifier using simulation software Tina Ver. 7 or Ver 9 or PSpice Version 9.1

APPARATUS

1. Tina or PSpice software.
2. **Personal computer.**

COMPONENTS AND EQUIPMENTS

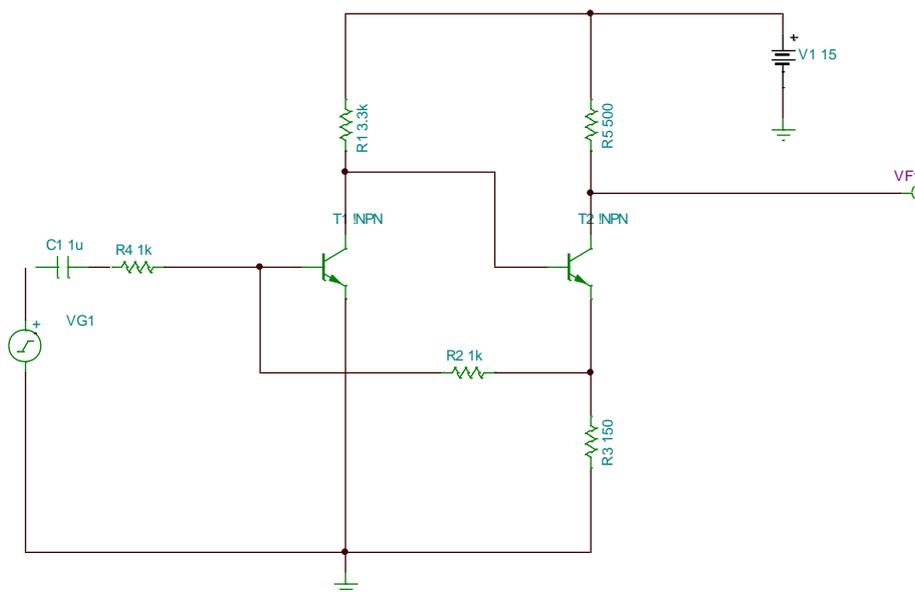| S. No.	Description	Part No./Value	Quantity
1	DC voltage Source	15V	1
2	Function Generator	0.05V, 1KHz	1
3	Transistor	2N3369	2
	Resistors	10KΩ, 500Ω, 470Ω, 3.3KΩ, 1KΩ(2)	5
4	Capacitors	10 µF(2)	3

CIRCUIT DIAGRAM

Fig-5: Current Shunt Feedback Amplifier

PROCEDURE

Drawing the circuit schematic

1. Open the Tina Schematics Editor page.
2. Click the Semiconductors tab and pick and place an NPN transistor on the centre of the page. Double click the transistor and set the part no. as BC107.
3. Next, click the Basic tab and pick and place resistors, capacitors, battery and the signal generator in convenient locations following the schematic of Fig. 1. (To rid the cursor of the component after placing it on the editor page, click anywhere on the page else, right click and select "Cancel Mode"). Then, place the ground symbol from the Basic tab and a Voltage pin from the Meters tab.
4. Double-click the resistors, capacitors and transistors in succession and set the parameter value of each component in the component dialog box. For transistors ensure that the right part no. is selected. **Note:** For megohms, use "M", for milliohms use "m", for microfarad use "u" and for nanofarad use "n". Connect the components using the Wire symbol in the menu bar, according to the schematic in Fig. 1. Then, place a single pole single throw (SPST or ON-OFF) switch from the Switches tab. Set the switch in ON position.
5. Double-click the power supply symbol and set to 25V DC. Next, double-click the signal generator symbol and enter DC = 0V, Signal (select Sinusoidal) AC = 20millivolts by entering 20m and set the frequency to 10kHz by entering 10k.

   Check the node voltages
6. The Electric Rules Check (ERC) is performed by clicking Analysis in the menubar and then clicking ERC. This will show up errors and warning messages if the schematic is not proper. Correct the schematic based on the summary displayed by the ERC.
7. Next, record the DC node voltages V_{CE}, across R_c and across R_e and current I_C by clicking Analysis →DC Analysis →Table of DC Results.

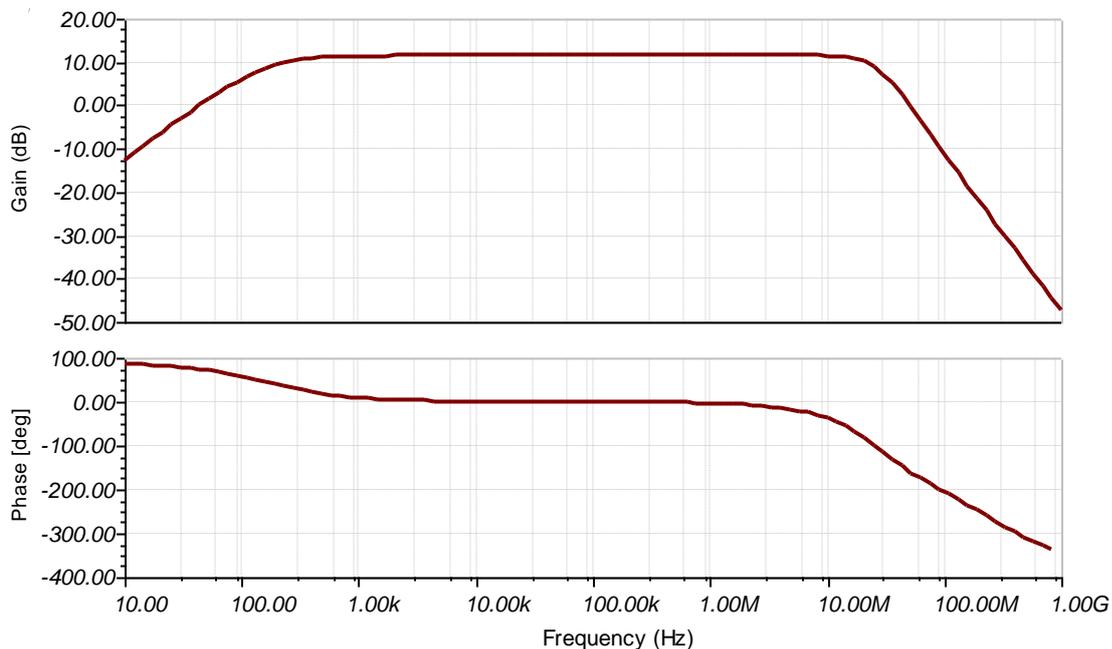   **Frequency Response with the switch S1 in ON position.**
8. **In PSpice:** Click the "Analysis" in the menu bar and then click "Setup" and select "AC Sweep" and set the starting frequency to 10 Hz and end frequency to 1 GHz and select radio button "Decade" and then close the box.
   Then, click "Analysis: and click "Simulate" and when the Analysis dialog box opens, select "AC". Select "Traces" button in the taskbar and click "Delete all traces" and then select "Adding Traces" and then select [B].

9. **In TINA :** Click "Analysis" in the menu bar and then click "AC Analysis" and then "AC Transfer Characteristic" and set the starting frequency to 10 Hz and end frequency to 1 GHz. Set the Sweep type to "Logarithmic" and click OK.
   A graph of gain in dB vs. Frequency in Hz will be displayed. Sketch the graph obtained on graph paper of *only* the response with the switch in ON position.

10. Click "Analysis" in the menu bar and then click "AC Analysis" and then "Table of AC Results" and note the voltage of the generator v_i, the voltage at the amplifier output v_o, and calculate the gain in dB using the expression 20 log v_o/ v_i and verify that it tallies with the value in the graph at 10kHz.

Sketch the output characteristics and draw the DC load line and mark the quiescent point of operation indicating I_C and I_B at this point from the values recorded in the previous step in DC Analysis.

11. Next, set the switch to OFF position i.e with the bypass capacitor disconnected from across the emitter resistor and run step 6. Note down the gain (in dB) at 10 KHz.

MODEL GRAPH

RESULT

1. The frequency response of the Common-emitter amplifier is measured and the graph plotted.

2. From the graph.

i) The gain at 10kHz = ……..dB.

ii) The lower cutoff frequency corresponding to a gain $20\log\left[\dfrac{v_o}{v_i}\right]$ dB -3dB = ……Hz.

iii) The upper cutoff frequency corresponding to a gain $20\log\left[\dfrac{v_o}{v_i}\right]$ dB -3dB = ……Hz.

iv). A sketch of the oscilloscope display of the response of the amplifier at 10 KHz frequency is made.

EXPERIMENT-6

Voltage Series Feedback Amplifier.

AIM: To design and determine the frequency of operation of a Voltage Series Feedback Amplifier using simulation software Tina Ver. 7 or Ver 9 or PSpice Version 9.1

APPARATUS

1. Tina or PSpice software.
2. Personal computer.

COMPONENTS AND EQUIPMENTS

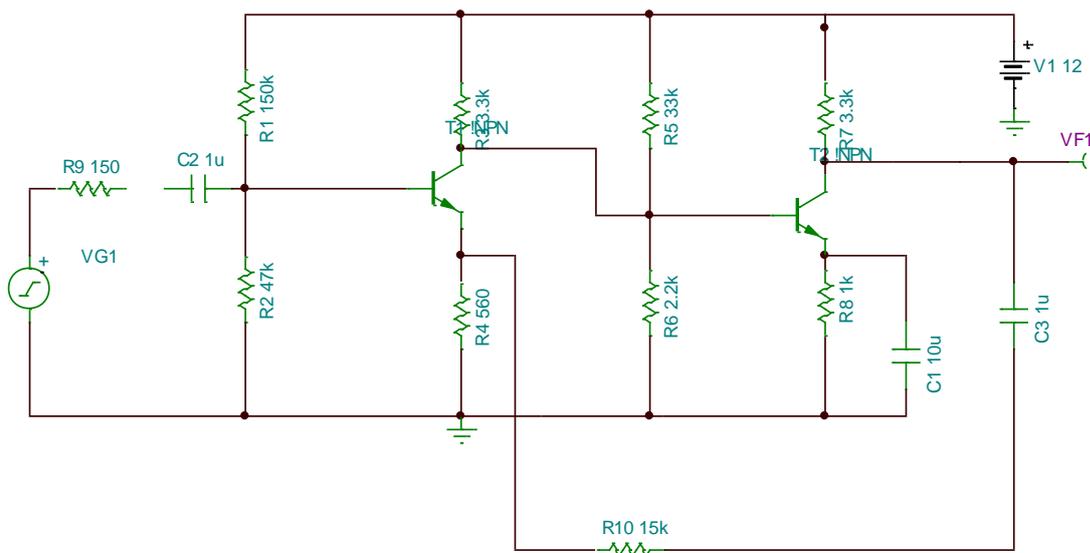| S. No.	Description	Part No./Value	Quantity
1	DC voltage Source	12V	1
2	Function Generator	0.05V, 1KHz	1
3	Transistor	2N3369	2
	Resistors(Ω)	150K,47K, 3.3K, 33K, 2.2K, 1K,150	9
4	Capacitors	1 μF	3

CIRCUIT DIAGRAM

Fig-6 : Voltage Series Feedback Amplifier

PROCEDURE

Drawing the circuit schematic

1. Open the Tina Schematics Editor page.
2. Click the Semiconductors tab and pick and place an NPN transistor on the centre of the page. Double click the transistor and set the part no. as BC107.
3. Next, click the Basic tab and pick and place resistors, capacitors, battery and the signal generator in convenient locations following the schematic of Fig. 1. (To rid the cursor of the component after placing it on the editor page, click anywhere on the page else, right click and select "Cancel Mode"). Then, place the ground symbol from the Basic tab and a Voltage pin from the Meters tab.
4. Double-click the resistors, capacitors and transistors in succession and set the parameter value of each component in the component dialog box. For transistors ensure that the right part no. is selected. **Note:** For megohms, use "M", for milliohms use "m", for microfarad use "u" and for nanofarad use "n". Connect the components using the Wire symbol in the menu bar, according to the schematic in Fig. 1. Then, place a single pole single throw (SPST or ON-OFF) switch from the Switches tab. Set the switch in ON position.
5. Double-click the power supply symbol and set to 25V DC. Next, double-click the signal generator symbol and enter DC = 0V, Signal (select Sinusoidal) AC = 20millivolts by entering 20m and set the frequency to 10kHz by entering 10k.
   Check the node voltages
6. The Electric Rules Check (ERC) is performed by clicking Analysis in the menubar and then clicking ERC. This will show up errors and warning messages if the schematic is not proper. Correct the schematic based on the summary displayed by the ERC.
7. Next, record the DC node voltages V_{CE}, across R_c and across R_e and current I_C by clicking Analysis →DC Analysis →Table of DC Results.

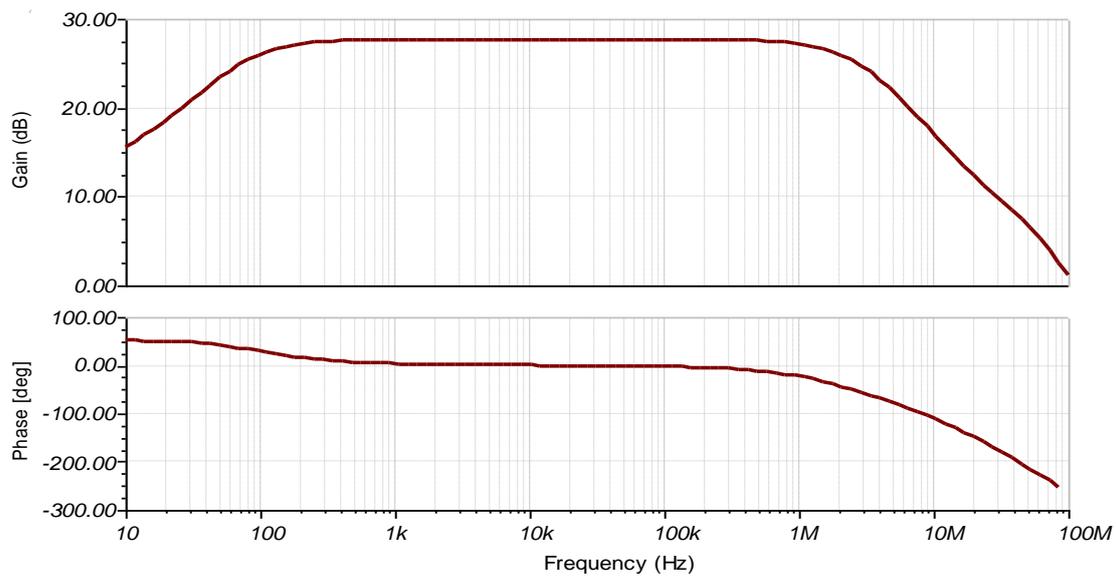   **Frequency Response with the switch S1 in ON position.**
8. **In PSpice:** Click the "Analysis" in the menu bar and then click "Setup" and select "AC Sweep" and set the starting frequency to 10 Hz and end frequency to 1 GHz and select radio button "Decade" and then close the box.
   Then, click "Analysis: and click "Simulate" and when the Analysis dialog box opens, select "AC". Select "Traces" button in the taskbar and click "Delete all traces" and then select "Adding Traces" and then select [B].

9. **In TINA :** Click "Analysis" in the menu bar and then click "AC Analysis" and then "AC Transfer Characteristic" and set the starting frequency to 10 Hz and end frequency to 1 GHz. Set the Sweep type to "Logarithmic" and click OK.
   A graph of gain in dB vs. Frequency in Hz will be displayed. Sketch the graph obtained on graph paper of *only* the response with the switch in ON position.

10. Click "Analysis" in the menu bar and then click "AC Analysis" and then "Table of AC Results" and note the voltage of the generator v_i, the voltage at the amplifier output v_o, and calculate the gain in dB using the expression 20 log v_o/ v_i and verify that it tallies with the value in the graph at 10kHz.

Sketch the output characteristics and draw the DC load line and mark the quiescent point of operation indicating I_C and I_B at this point from the values recorded in the previous step in DC Analysis.

11. Next, set the switch to OFF position i.e with the bypass capacitor disconnected from across the emitter resistor and run step 6. Note down the gain (in dB) at 10 KHz.

MODEL GRAPH

RESULT

1. The frequency response of the Common-emitter amplifier is measured and the graph plotted.

2. From the graph.

i) The gain at 10kHz = ……..dB.

ii) The lower cutoff frequency corresponding to a gain $20 \log \left[\dfrac{v_o}{v_i} \right]$ dB -3dB = ……Hz.

iii) The upper cutoff frequency corresponding to a gain $20 \log \left[\dfrac{v_o}{v_i} \right]$ dB -3dB = ……Hz.

iv). A sketch of the oscilloscope display of the response of the amplifier at 10 KHz frequency is made.

EXPERIMENT-7

Cascode Amplifier.

AIM: To design and determine the frequency of operation of a Cascode Amplifier using simulation software Tina Ver. 7 or Ver 9 or PSpice Version 9.1

APPARATUS

1. Tina or PSpice software.
2. **Personal** computer.

COMPONENTS AND EQUIPMENTS

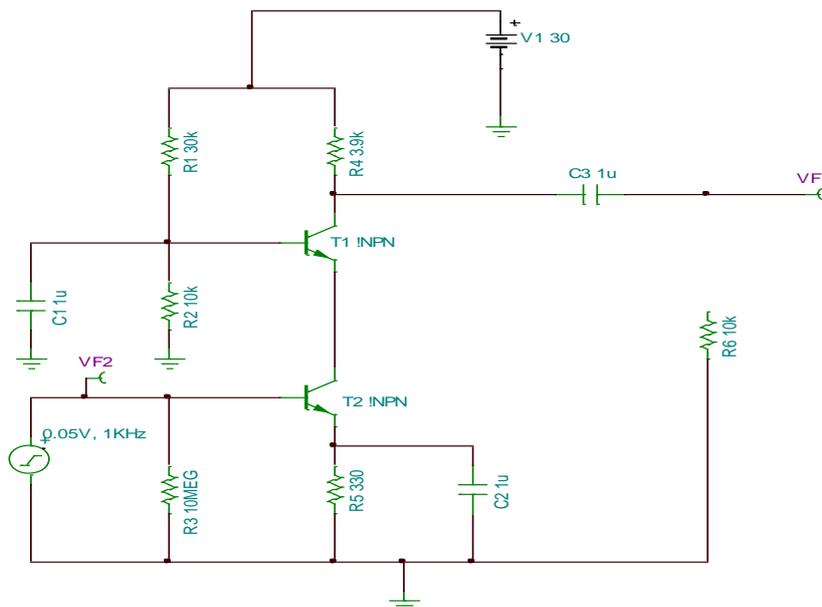| S. No.	Description	Part No./Value	Quantity
1	DC voltage Source	30V	1
2	Function Generator	0.05V, 1KHz	1
3	Transistor	2N3369	1
	Resistors	30K,3.3Ω, 470Ω, 10M, 10K, 330	6
4	Capacitors	10 µF(2)	3

CIRCUIT DIAGRAM

Fig-7 : Cascode Amplifier

PROCEDURE

Drawing the circuit schematic

1. Open the Tina Schematics Editor page.
2. Click the Semiconductors tab and pick and place an NPN transistor on the centre of the page. Double click the transistor and set the part no. as BC107.
3. Next, click the Basic tab and pick and place resistors, capacitors, battery and the signal generator in convenient locations following the schematic of Fig. 1. (To rid the cursor of the component after placing it on the editor page, click anywhere on the page else, right click and select "Cancel Mode"). Then, place the ground symbol from the Basic tab and a Voltage pin from the Meters tab.
4. Double-click the resistors, capacitors and transistors in succession and set the parameter value of each component in the component dialog box. For transistors ensure that the right part no. is selected. **Note:** For megohms, use "M", for milliohms use "m", for microfarad use "u" and for nanofarad use "n". Connect the components using the Wire symbol in the menu bar, according to the schematic in Fig. 1. Then, place a single pole single throw (SPST or ON-OFF) switch from the Switches tab. Set the switch in ON position.
5. Double-click the power supply symbol and set to 25V DC. Next, double-click the signal generator symbol and enter DC = 0V, Signal (select Sinusoidal) AC = 20millivolts by entering 20m and set the frequency to 10kHz by entering 10k.
   Check the node voltages
6. The Electric Rules Check (ERC) is performed by clicking Analysis in the menubar and then clicking ERC. This will show up errors and warning messages if the schematic is not proper. Correct the schematic based on the summary displayed by the ERC.
7. Next, record the DC node voltages V_{CE}, across R_c and across R_e and current I_C by clicking Analysis →DC Analysis →Table of DC Results.

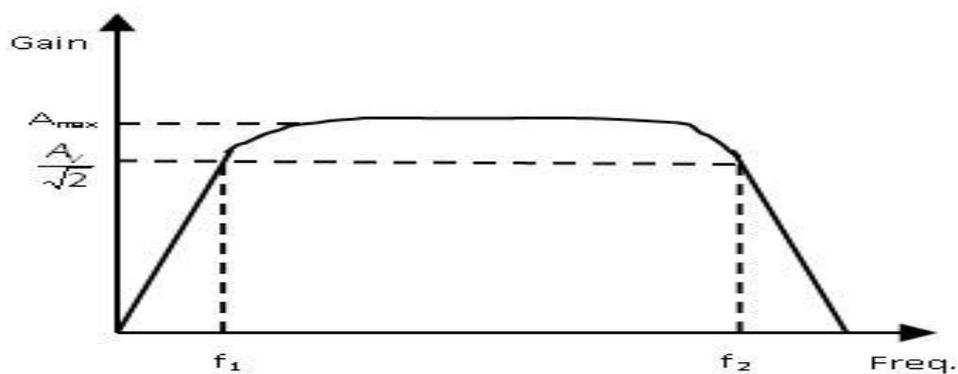   **Frequency Response with the switch S1 in ON position.**
8. **In PSpice:** Click the "Analysis" in the menu bar and then click "Setup" and select "AC Sweep" and set the starting frequency to 10 Hz and end frequency to 1 GHz and select radio button "Decade" and then close the box.
   Then, click "Analysis: and click "Simulate" and when the Analysis dialog box opens, select "AC". Select "Traces" button in the taskbar and click "Delete all traces" and then select "Adding Traces" and then select [B].

9. **In TINA :** Click "Analysis" in the menu bar and then click "AC Analysis" and then "AC Transfer Characteristic" and set the starting frequency to 10 Hz and end frequency to 1 GHz. Set the Sweep type to "Logarithmic" and click OK.
   A graph of gain in dB vs. Frequency in Hz will be displayed. Sketch the graph obtained on graph paper of *only* the response with the switch in ON position.

10. Click "Analysis" in the menu bar and then click "AC Analysis" and then "Table of AC Results" and note the voltage of the generator v_i, the voltage at the amplifier output v_o, and calculate the gain in dB using the expression $20 \log v_o / v_i$ and verify that it tallies with the value in the graph at 10kHz.

Sketch the output characteristics and draw the DC load line and mark the quiescent point of operation indicating I_C and I_B at this point from the values recorded in the previous step in DC Analysis.

11. Next, set the switch to OFF position i.e with the bypass capacitor disconnected from across the emitter resistor and run step 6. Note down the gain (in dB) at 10 KHz.

MODEL GRAPH

RESULT

1. The frequency response of the Cascode amplifier is measured and the graph plotted.

2. From the graph.

i) The gain at 10kHz = ……..dB.

ii) The lower cutoff frequency corresponding to a gain $20 \log \left[\dfrac{v_o}{v_i} \right]$ dB -3dB = ……Hz.

iii) The upper cutoff frequency corresponding to a gain $20 \log \left[\dfrac{v_o}{v_i} \right]$ dB -3dB = ……Hz.

iv). A sketch of the oscilloscope display of the response of the amplifier at 10 KHz frequency is made.

EXPERIMENT-8

Wien Bridge Oscillator using Transistors.

AIM: To design and determine the frequency of Wien Bridge Oscillator using simulation software Tina Ver. 7 or Ver 9 or PSpice Version 9.1

APPARATUS

1. Tina or PSpice software.
2. **Personal computer.**

COMPONENTS AND EQUIPMENTS

S. No.	Description	Part No./Value	Quantity
1	DC voltage Source	30V	1
2	Transistor	2N3369	1
3	Resistors	22K, 4.7 K, 10K, 2.2K, 1 KΩ	11
4	Capacitors	10 μF, 1nF	5

CIRCUIT DIAGRAM

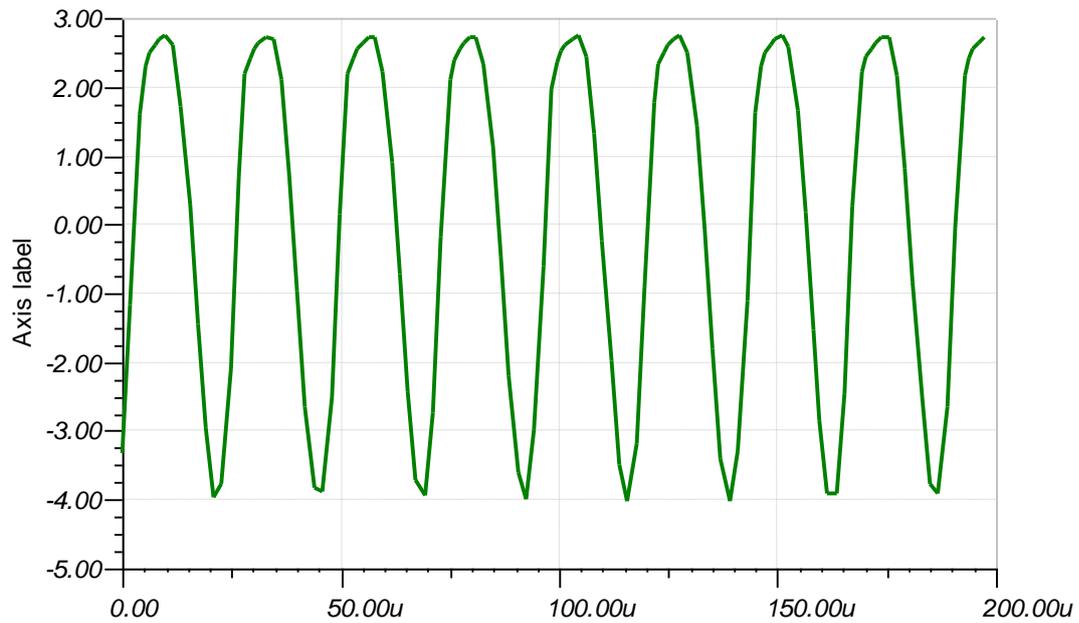**Fig-8: Wein Bridge Oscillator**

PROCEDURE

Drawing the circuit schematic

1. Open the Tina Schematics Editor page.
2. Click the Semiconductors tab and pick and place an NPN transistor on the centre of the page. Double click the transistor and set the part no. as BC107.
3. Next, click the Basic tab and pick and place resistors, capacitors, battery and the signal generator in convenient locations following the schematic of Fig. 1. (To rid the cursor of the component after placing it on the editor page, click anywhere on the page else, right click and select "Cancel Mode"). Then, place the ground symbol from the Basic tab and a Voltage pin from the Meters tab.
4. Double-click the resistors, capacitors and transistors in succession and set the parameter value of each component in the component dialog box. For transistors ensure that the right part no. is selected. **Note:** For megohms, use "M", for milliohms use "m", for microfarad use "u" and for nanofarad use "n". Connect the components using the Wire symbol in the menu bar, according to the schematic in Fig. 1. Then, place a single pole single throw (SPST or ON-OFF) switch from the Switches tab. Set the switch in ON position.
5. Double-click the power supply symbol and set to 25V DC. Next, double-click the signal generator symbol and enter DC = 0V, Signal (select Sinusoidal) AC = 20millivolts by entering 20m and set the frequency to 10kHz by entering 10k.
   ### Check the node voltages
6. The Electric Rules Check (ERC) is performed by clicking Analysis in the menubar and then clicking ERC. This will show up errors and warning messages if the schematic is not proper. Correct the schematic based on the summary displayed by the ERC.
7. Next, record the DC node voltages V_{CE}, across R_c and across R_e and current I_C by clicking Analysis →DC Analysis →Table of DC Results.
8. Next, select "Oscilloscope" from the T&M Menu. Then click Run and then Auto in the bottom of the virtual oscilloscope dialog. Select AC coupling and observe the steady-state sinusoidal waveform. To synchronize screen waveform set Trigger mode to Normal and then using the B cursor ON, measure the time period of the waveform. Then, use the horizontal cursor to measure the peak to peak voltage of the waveform. A screen shot of the output waveform is shown in Fig. 2.
9. Sketch the steady-state sinusoidal output waveform on a graph paper.

MODEL GRAPH

RESULT

1. The frequency of the wein bridge oscillator was observed and the graph is plotted.

   The frequency of oscillation is $\dfrac{1}{T}$ =…….Hz and v_o(p-p) is ………..volts.

2. Caculte the theoretical frequency $f = 1/2\pi RC$

EXPERIMENT-9

RC Phase Shift Oscillator using Transistors.

AIM: To design and determine the frequency of RC Phase Shift Oscillator using simulation software Tina Ver. 7 or Ver 9 or PSpice Version 9.1

APPARATUS

1. Tina or PSpice software.
2. Personal computer.

COMPONENTS AND EQUIPMENTS

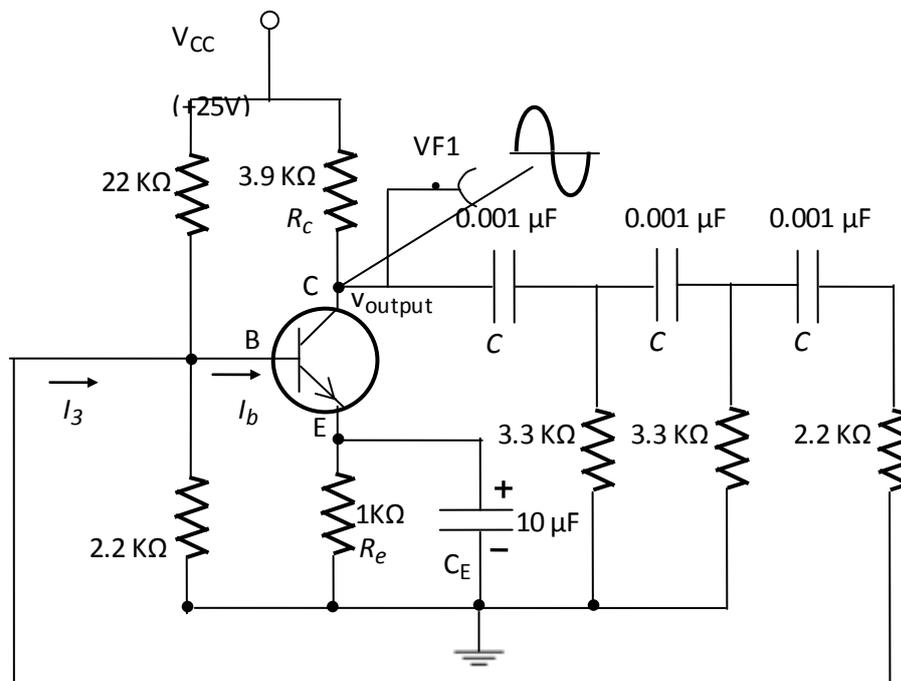| S. No.	Description	Part No./Value	Quantity
1	DC voltage Source	25V	1
2	Transistor	2N3369	1
3	Resistors	22KΩ, 2.2KΩ, 3.3KΩ, 1 KΩ	7
4	Capacitors	0.001µF, 10uF	4

CIRCUIT DIAGRAM
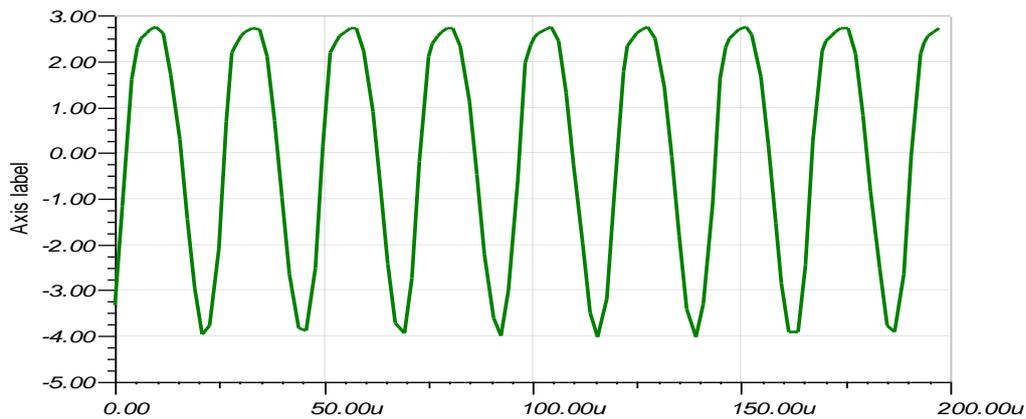

Fig-9: RC- Phase Shift Oscillator

PROCEDURE

Drawing the circuit schematic

1. Open the Tina Schematics Editor page.
2. Click the Semiconductors tab and pick and place an NPN transistor on the centre of the page. Double click the transistor and set the part no. as BC107.
3. Next, click the Basic tab and pick and place resistors, capacitors, battery and the signal generator in convenient locations following the schematic of Fig. 1. (To rid the cursor of the component after placing it on the editor page, click anywhere on the page else, right click and select "Cancel Mode"). Then, place the ground symbol from the Basic tab and a Voltage pin from the Meters tab.
4. Double-click the resistors, capacitors and transistors in succession and set the parameter value of each component in the component dialog box. For transistors ensure that the right part no. is selected. **Note:** For megohms, use "M", for milliohms use "m", for microfarad use "u" and for nanofarad use "n". Connect the components using the Wire symbol in the menu bar, according to the schematic in Fig. 1. Then, place a single pole single throw (SPST or ON-OFF) switch from the Switches tab. Set the switch in ON position.
5. Double-click the power supply symbol and set to 25V DC. Next, double-click the signal generator symbol and enter DC = 0V, Signal (select Sinusoidal) AC = 20millivolts by entering 20m and set the frequency to 10kHz by entering 10k.

   Check the node voltages

6. The Electric Rules Check (ERC) is performed by clicking Analysis in the menubar and then clicking ERC. This will show up errors and warning messages if the schematic is not proper. Correct the schematic based on the summary displayed by the ERC.
7. Next, record the DC node voltages V_{CE}, across R_c and across R_e and current I_C by clicking Analysis →DC Analysis →Table of DC Results.
8. Next, select "Oscilloscope" from the T&M Menu. Then click Run and then Auto in the bottom of the virtual oscilloscope dialog. Select AC coupling and observe the steady-state sinusoidal waveform. To synchronize screen waveform set Trigger mode to Normal and then using the B cursor ON, measure the time period of the waveform. Then, use the horizontal cursor to measure the peak to peak voltage of the waveform. A screen shot of the output waveform is shown in Fig. 2.
9. Sketch the steady-state sinusoidal output waveform on a graph paper.

MODEL GRAPH

RESULT

1. The frequency of the RC phase shift oscillator was observed and the graph for 2 cycles plotted. The frequency of oscillation is $\dfrac{1}{T}$ =…….Hz and v_o(p-p) is ………..volts.

2. The dc voltages i) V_{CE} =……volts   ii) across R_c = ………volts, and

iii) across R_e. = ………volts

On the output characteristics, the dc load line is drawn and the quiescent point of operation indicating I_C and I_B at this point is marked.

EXPERIMENT-10

Class-A Power Amplifier (Transformer less).

AIM: To design and determine the frequency of operation of Class-A Power Amplifier using simulation software Tina Ver. 7 or Ver 9 or PSpice Version 9.1
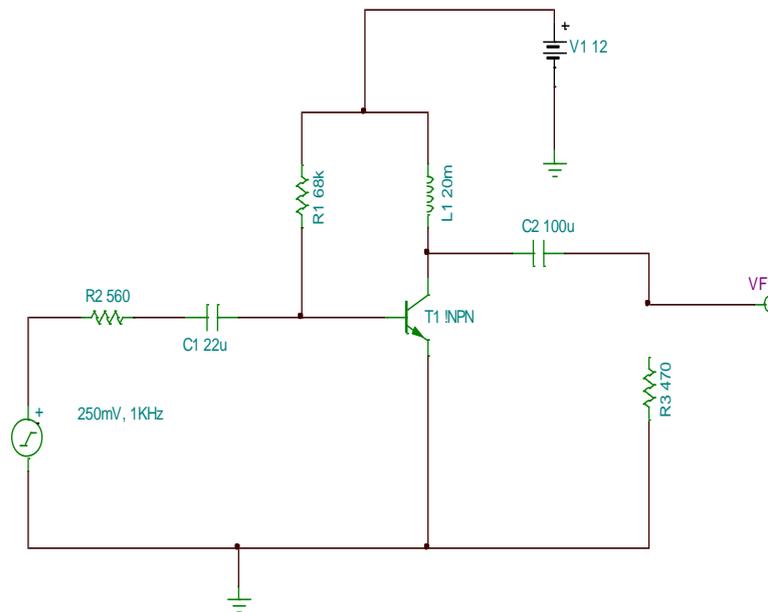
APPARATUS

1. Tina or PSpice software.
2. Personal computer.

COMPONENTS AND EQUIPMENTS

S. No.	Description	Part No./Value	Quantity
1	DC voltage Source	15V	1
2	Transistor	2N3369	1
3	Resistors	10KΩ, 4.7 KΩ, 470Ω, 2MΩ, 1 KΩ	5
4	Capacitors	10 µF(2)	3

CIRCUIT DIAGRAM

Fig-10: Class-A Power Amplifier

PROCEDURE

Drawing the circuit schematic

1. Open the Tina Schematics Editor page.
2. Click the Semiconductors tab and pick and place an NPN transistor on the centre of the page. Double click the transistor and set the part no. as BC107.
3. Next, click the Basic tab and pick and place resistors, capacitors, battery and the signal generator in convenient locations following the schematic of Fig. 1. (To rid the cursor of the component after placing it on the editor page, click anywhere on the page else, right click and select "Cancel Mode"). Then, place the ground symbol from the Basic tab and a Voltage pin from the Meters tab.
4. Double-click the resistors, capacitors and transistors in succession and set the parameter value of each component in the component dialog box. For transistors ensure that the right part no. is selected. **Note:** For megohms, use "M", for milliohms use "m", for microfarad use "u" and for nanofarad use "n". Connect the components using the Wire symbol in the menu bar, according to the schematic in Fig. 1. Then, place a single pole single throw (SPST or ON-OFF) switch from the Switches tab. Set the switch in ON position.
5. Double-click the power supply symbol and set to 25V DC. Next, double-click the signal generator symbol and enter DC = 0V, Signal (select Sinusoidal) AC = 20millivolts by entering 20m and set the frequency to 10kHz by entering 10k.
    Check the node voltages
6. The Electric Rules Check (ERC) is performed by clicking Analysis in the menubar and then clicking ERC. This will show up errors and warning messages if the schematic is not proper. Correct the schematic based on the summary displayed by the ERC.
7. Next, record the DC node voltages V_{CE}, across R_c and across R_e and current I_C by clicking Analysis →DC Analysis →Table of DC Results.

    Frequency Response with the switch S1 in ON position.
8. **In PSpice:** Click the "Analysis" in the menu bar and then click "Setup" and select "AC Sweep" and set the starting frequency to 10 Hz and end frequency to 1 GHz and select radio button "Decade" and then close the box.
    Then, click "Analysis: and click "Simulate" and when the Analysis dialog box opens, select "AC". Select "Traces" button in the taskbar and click "Delete all traces" and then select "Adding Traces" and then select [B].

9. **In TINA :** Click "Analysis" in the menu bar and then click "AC Analysis" and then "AC Transfer Characteristic" and set the starting frequency to 10 Hz and end frequency to 1 GHz. Set the Sweep type to "Logarithmic" and click OK.
    A graph of gain in dB vs. Frequency in Hz will be displayed. Sketch the graph obtained on graph paper of *only* the response with the switch in ON position.

10. Click "Analysis" in the menu bar and then click "AC Analysis" and then "Table of AC Results" and note the voltage of the generator v_i, the voltage at the amplifier output v_o, and calculate the gain in dB using the expression $20 \log v_o / v_i$ and verify that it tallies with the value in the graph at 10kHz.
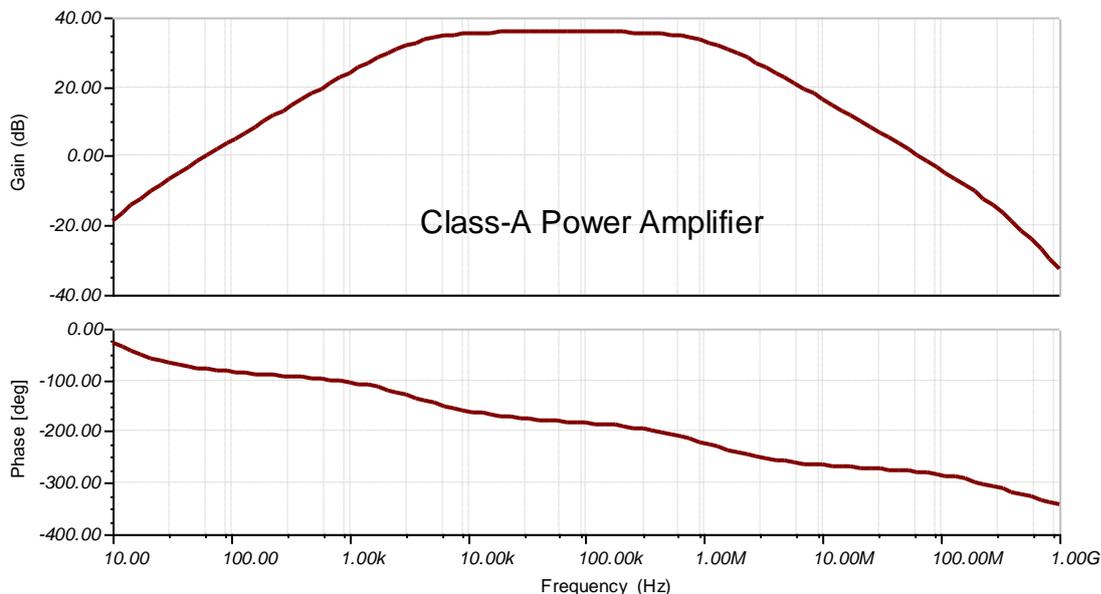
Sketch the output characteristics and draw the DC load line and mark the quiescent point of operation indicating I_C and I_B at this point from the values recorded in the previous step in DC Analysis.

11. Next, set the switch to OFF position i.e with the bypass capacitor disconnected from across the emitter resistor and run step 6. Note down the gain (in dB) at 10 KHz.

Additional exercise

12. Set the switch again to ON position.

13. From the menu, double click the signal generator and set the input sinusoidal frequency to 10kHz. On the menu bar click Test & Measurement (T&M) and click "Oscilloscope".

14. Click Run and then Auto (bar at the bottom of the virtual oscilloscope, and set connection to AC) and set the V/div and Time/div till a clean sinusoidal trace appears. Then press the "stop" button to freeze the oscilloscope trace and using the B button in the Cursor box and with the ON button pressed in the Cursor box, measure the amplitude of the traces taking care with the V/div mentioned at the top of the screen.

15. Next, change the amplitude of the signal generator to say, 500mV and again obtain a trace and record your observations of the trace.

MODEL GRAPH

Class-A Power Amplifier

RESULT

1. The frequency response of the Class-A Power amplifier is measured and the graph plotted.

2. From the graph.

i) The gain at 10kHz = ……..dB.

ii) The lower cutoff frequency corresponding to a gain $20\log\left[\dfrac{v_o}{v_i}\right]$ dB -3dB = ……Hz.

iii) The upper cutoff frequency corresponding to a gain $20\log\left[\dfrac{v_o}{v_i}\right]$ dB -3dB = ……Hz.

iv). A sketch of the oscilloscope display of the response of the amplifier at 10 KHz frequency is made.

EXPERIMENT-11

Class-B Complementary Symmetry Amplifier.

AIM: To design and determine the frequency of operation of a Class-B Complementary Symmentry Amplifier using simulation software Tina Ver. 7 or Ver 9 or PSpice Version 9.1

APPARATUS

1. Tina or PSpice software.
2. Personal computer.

COMPONENTS AND EQUIPMENTS

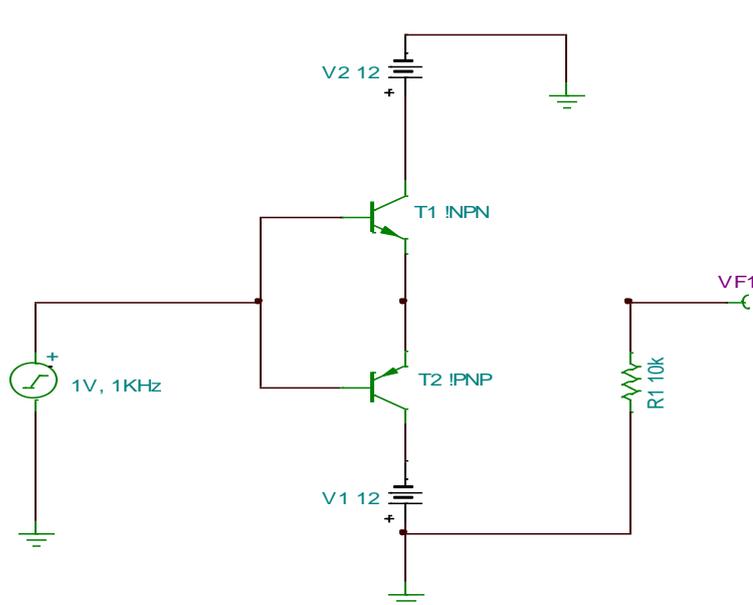| S. No.	Description	Part No./Value	Quantity
1	DC voltage Source	15V	1
2	Function Generator	0.05V, 1KHz	1
3	Transistor	2N3369	1
	Resistors	10KΩ, 4.7 KΩ, 470Ω, 2MΩ, 1 KΩ	5
4	Capacitors	10 µF(2)	3

CIRCUIT DIAGRAM

Fig-11: Class-B Complementary Symmentry Amplifier

PROCEDURE

Drawing the circuit schematic

1. Open the Tina Schematics Editor page.
2. Click the Semiconductors tab and pick and place an NPN transistor on the centre of the page. Double click the transistor and set the part no. as BC107.
3. Next, click the Basic tab and pick and place resistors, capacitors, battery and the signal generator in convenient locations following the schematic of Fig. 1. (To rid the cursor of the component after placing it on the editor page, click anywhere on the page else, right click and select "Cancel Mode"). Then, place the ground symbol from the Basic tab and a Voltage pin from the Meters tab.
4. Double-click the resistors, capacitors and transistors in succession and set the parameter value of each component in the component dialog box. For transistors ensure that the right part no. is selected. **Note:** For megohms, use "M", for milliohms use "m", for microfarad use "u" and for nanofarad use "n". Connect the components using the Wire symbol in the menu bar, according to the schematic in Fig. 1. Then, place a single pole single throw (SPST or ON-OFF) switch from the Switches tab. Set the switch in ON position.
5. Double-click the power supply symbol and set to 25V DC. Next, double-click the signal generator symbol and enter DC = 0V, Signal (select Sinusoidal) AC = 20millivolts by entering 20m and set the frequency to 10kHz by entering 10k.
    Check the node voltages
6. The Electric Rules Check (ERC) is performed by clicking Analysis in the menubar and then clicking ERC. This will show up errors and warning messages if the schematic is not proper. Correct the schematic based on the summary displayed by the ERC.
7. Next, record the DC node voltages V_{CE}, across R_c and across R_e and current I_C by clicking Analysis →DC Analysis →Table of DC Results.

    Frequency Response with the switch S1 in ON position.
8. **In PSpice:** Click the "Analysis" in the menu bar and then click "Setup" and select "AC Sweep" and set the starting frequency to 10 Hz and end frequency to 1 GHz and select radio button "Decade" and then close the box.
    Then, click "Analysis: and click "Simulate" and when the Analysis dialog box opens, select "AC". Select "Traces" button in the taskbar and click "Delete all traces" and then select "Adding Traces" and then select [B].

9. **In TINA :** Click "Analysis" in the menu bar and then click "AC Analysis" and then "AC Transfer Characteristic" and set the starting frequency to 10 Hz and end frequency to 1 GHz. Set the Sweep type to "Logarithmic" and click OK.
    A graph of gain in dB vs. Frequency in Hz will be displayed. Sketch the graph obtained on graph paper of *only* the response with the switch in ON position.

10. Click "Analysis" in the menu bar and then click "AC Analysis" and then "Table of AC Results" and note the voltage of the generator v_i, the voltage at the amplifier output v_o, and calculate the gain in dB using the expression $20 \log v_o/ v_i$ and verify that it tallies with the value in the graph at 10kHz.
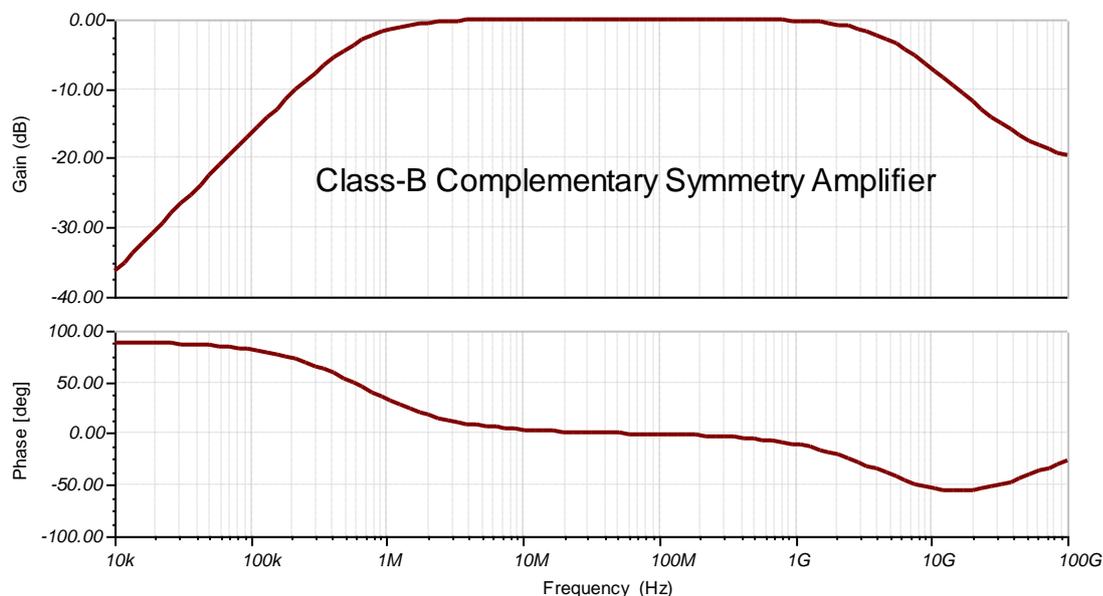
    Sketch the output characteristics and draw the DC load line and mark the quiescent point of operation indicating I_C and I_B at this point from the values recorded in the previous step in DC Analysis.

11. Next, set the switch to OFF position i.e with the bypass capacitor disconnected from across the emitter resistor and run step 6. Note down the gain (in dB) at 10 KHz.

    Additional exercise

12. Set the switch again to ON position.

13. From the menu, double click the signal generator and set the input sinusoidal frequency to 10kHz. On the menu bar click Test & Measurement (T&M) and click "Oscilloscope".

14. Click Run and then Auto (bar at the bottom of the virtual oscilloscope, and set connection to AC) and set the V/div and Time/div till a clean sinusoidal trace appears. Then press the "stop" button to freeze the oscilloscope trace and using the B button in the Cursor box and with the ON button pressed in the Cursor box, measure the amplitude of the traces taking care with the V/div mentioned at the top of the screen.

15. Next, change the amplitude of the signal generator to say, 500mV and again obtain a trace and record your observations of the trace.

MODEL GRAPH

RESULT

1. The frequency response of the Class-B Complementary Symmentry Amplifier is measured and the graph plotted.

2. From the graph.

i) The gain at 10kHz = ……..dB.

ii) The lower cutoff frequency corresponding to a gain $20\log\left[\dfrac{v_o}{v_i}\right]$ dB -3dB = ……Hz.

iii) The upper cutoff frequency corresponding to a gain $20\log\left[\dfrac{v_o}{v_i}\right]$ dB -3dB = ……Hz.

iv). A sketch of the oscilloscope display of the response of the amplifier at 10 KHz frequency is made.

EXPERIMENT-12

<u>Hartley Oscillator.</u>

<u>AIM</u>: To design and determine the frequency of Hartley Oscillator using simulation software Tina Ver. 7 or Ver 9 or PSpice Version 9.1

<u>APPARATUS</u>

1. Tina or PSpice software.

2. Personal computer.

<u>COMPONENTS AND EQUIPMENTS</u>

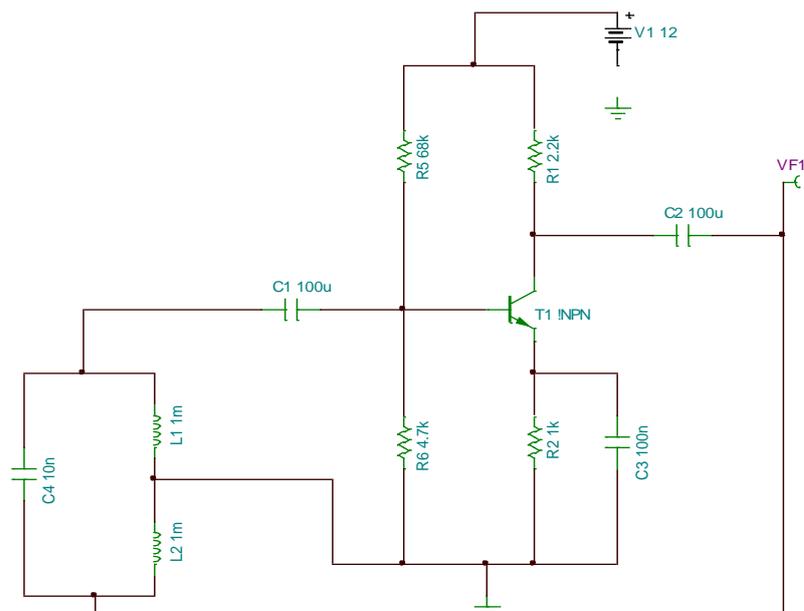| S. No.	Description	Part No./Value	Quantity
1	DC voltage Source	12V	1
2	Transistor	2N3369	1
3	Resistors	100KΩ, 5KΩ, 470Ω, 1KΩ, 10KΩ	5
4	Capacitors	0.001uF, 0.01μF, 0.1uF	3
5	Inductor	1mH	2

<u>CIRCUIT DIAGRAM</u>
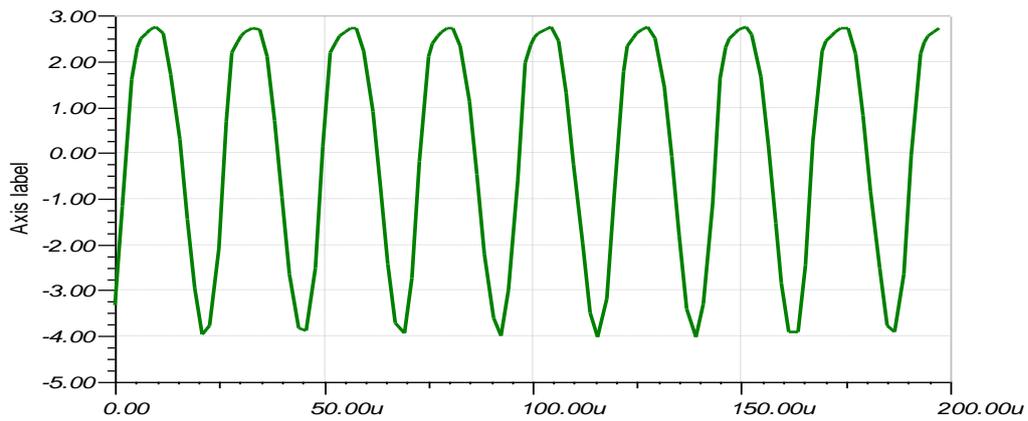

Fig-12: Hartley Oscillator

PROCEDURE

Drawing the circuit schematic

1. Open the Tina Schematics Editor page.
2. Click the Semiconductors tab and pick and place an NPN transistor on the centre of the page. Double click the transistor and set the part no. as BC107.
3. Next, click the Basic tab and pick and place resistors, capacitors, battery and the signal generator in convenient locations following the schematic of Fig. 1. (To rid the cursor of the component after placing it on the editor page, click anywhere on the page else, right click and select "Cancel Mode"). Then, place the ground symbol from the Basic tab and a Voltage pin from the Meters tab.
4. Double-click the resistors, capacitors and transistors in succession and set the parameter value of each component in the component dialog box. For transistors ensure that the right part no. is selected. **Note:** For megohms, use "M", for milliohms use "m", for microfarad use "u" and for nanofarad use "n". Connect the components using the Wire symbol in the menu bar, according to the schematic in Fig. 1. Then, place a single pole single throw (SPST or ON-OFF) switch from the Switches tab. Set the switch in ON position.
5. Double-click the power supply symbol and set to 25V DC. Next, double-click the signal generator symbol and enter DC = 0V, Signal (select Sinusoidal) AC = 20millivolts by entering 20m and set the frequency to 10kHz by entering 10k.
   Check the node voltages
6. The Electric Rules Check (ERC) is performed by clicking Analysis in the menubar and then clicking ERC. This will show up errors and warning messages if the schematic is not proper. Correct the schematic based on the summary displayed by the ERC.
7. Next, record the DC node voltages V_{CE}, across R_c and across R_e and current I_C by clicking Analysis →DC Analysis →Table of DC Results.
8. Next, select "Oscilloscope" from the T&M Menu. Then click Run and then Auto in the bottom of the virtual oscilloscope dialog. Select AC coupling and observe the steady-state sinusoidal waveform. To synchronize screen waveform set Trigger mode to Normal and then using the B cursor ON, measure the time period of the waveform. Then, use the horizontal cursor to measure the peak to peak voltage of the waveform. A screen shot of the output waveform is shown in Fig. 2.
9. Sketch the steady-state sinusoidal output waveform on a graph paper.

MODEL GRAPH

RESULT

1. The frequency of the Hartley Oscillator was observed and the graph for 2 cycles plotted.

The frequency of oscillation is $\dfrac{1}{T}$ =......Hz and v_{o}(p-p) is ....volts.

EXPERIMENT-13

Colpitt's Oscillator.

AIM: To design and determine the frequency of Colpitt's Oscillator using simulation software Tina Ver. 7 or Ver 9 or PSpice Version 9.1
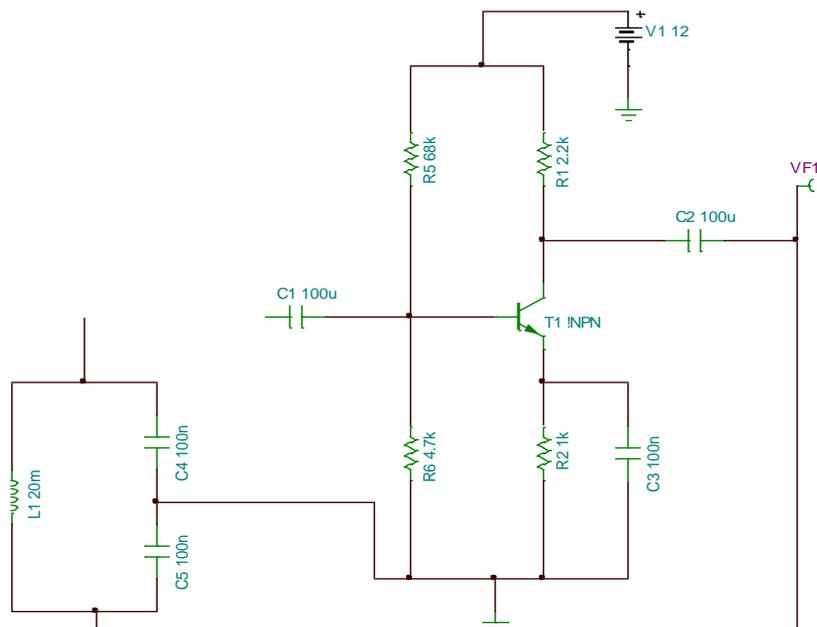
APPARATUS

1. Tina or PSpice software.
2. Personal computer.

COMPONENTS AND EQUIPMENTS

S. No.	Description	Part No./Value	Quantity
1	DC voltage Source	12V	1
2	Transistor	BC107	1
3	Resistors	100KΩ, 5 KΩ, 10KΩ, 1 KΩ	4
4	Capacitors	10 μF, 100nF, 0.1uF	5
5	Inductor	100mH	1

CIRCUIT DIAGRAM
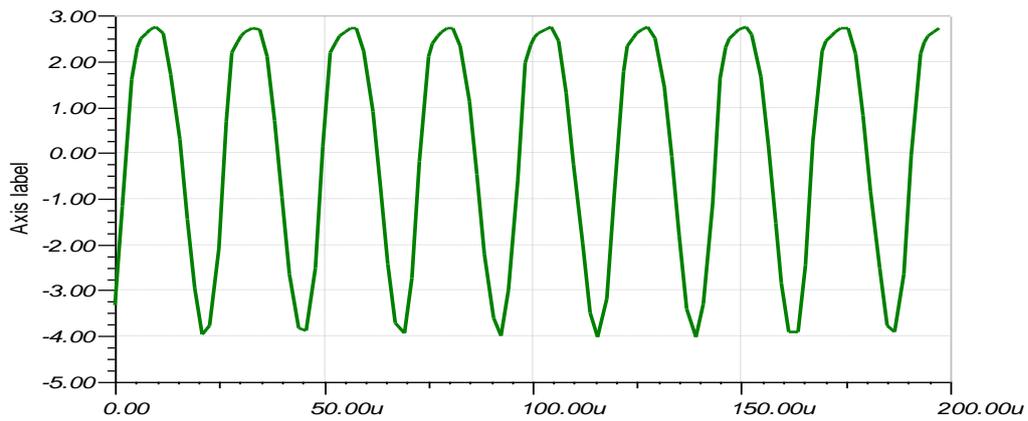

Fig-13: Colpitt's Oscillator

PROCEDURE

Drawing the circuit schematic

1. Open the Tina Schematics Editor page.
2. Click the Semiconductors tab and pick and place an NPN transistor on the centre of the page. Double click the transistor and set the part no. as BC107.
3. Next, click the Basic tab and pick and place resistors, capacitors, battery and the signal generator in convenient locations following the schematic of Fig. 1. (To rid the cursor of the component after placing it on the editor page, click anywhere on the page else, right click and select "Cancel Mode"). Then, place the ground symbol from the Basic tab and a Voltage pin from the Meters tab.
4. Double-click the resistors, capacitors and transistors in succession and set the parameter value of each component in the component dialog box. For transistors ensure that the right part no. is selected. **Note:** For megohms, use "M", for milliohms use "m", for microfarad use "u" and for nanofarad use "n". Connect the components using the Wire symbol in the menu bar, according to the schematic in Fig. 1. Then, place a single pole single throw (SPST or ON-OFF) switch from the Switches tab. Set the switch in ON position.
5. Double-click the power supply symbol and set to 25V DC. Next, double-click the signal generator symbol and enter DC = 0V, Signal (select Sinusoidal) AC = 20millivolts by entering 20m and set the frequency to 10kHz by entering 10k.
    Check the node voltages
6. The Electric Rules Check (ERC) is performed by clicking Analysis in the menubar and then clicking ERC. This will show up errors and warning messages if the schematic is not proper. Correct the schematic based on the summary displayed by the ERC.
7. Next, record the DC node voltages V_{CE}, across R_c and across R_e and current I_C by clicking Analysis →DC Analysis →Table of DC Results.
8. Next, select "Oscilloscope" from the T&M Menu. Then click Run and then Auto in the bottom of the virtual oscilloscope dialog. Select AC coupling and observe the steady-state sinusoidal waveform. To synchronize screen waveform set Trigger mode to Normal and then using the B cursor ON, measure the time period of the waveform. Then, use the horizontal cursor to measure the peak to peak voltage of the waveform. A screen shot of the output waveform is shown in Fig. 2.
9. Sketch the steady-state sinusoidal output waveform on a graph paper.

MODEL GRAPH

RESULT

1. The frequency of the Colpitt's Oscillator was observed and the graph for 2 cycles plotted.

The frequency of oscillation is $\dfrac{1}{T}$ =……Hz and v_o(p-p) is ………..volts.

EXPERIMET-14

Single Tuned Voltage Amplifier.

AIM: To design and determine the frequency of Single Tuned Voltage Amplifier using simulation software Tina Ver. 7 or Ver 9 or PSpice Version 9.1
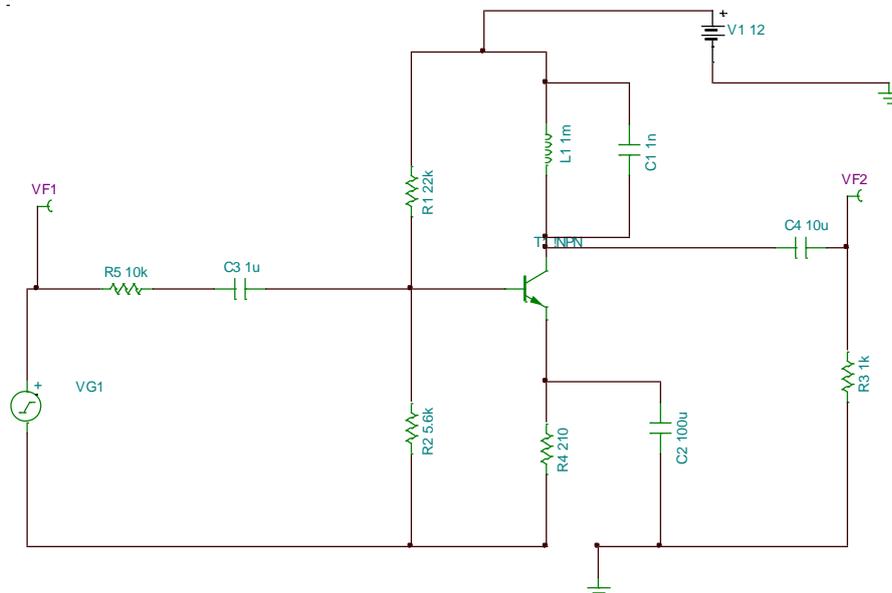
APPARATUS

1. Tina or PSpice software.
2. Personal computer.

COMPONENTS AND EQUIPMENTS

S. No.	Description	Part No./Value	Quantity
1	DC voltage Source	12V	1
2	Transistor	2N3369	1
3	Resistors	22KΩ, 5.6KΩ, 210Ω, 10KΩ, 1K	5
4	Capacitors	10 µF, 1uF, 100uF,1nF	4

CIRCUIT DIAGRAM

Fig-14: Single Tuned Voltage Amplifier

PROCEDURE

Drawing the circuit schematic

1. Open the Tina Schematics Editor page.
2. Click the Semiconductors tab and pick and place an NPN transistor on the centre of the page. Double click the transistor and set the part no. as BC107.
3. Next, click the Basic tab and pick and place resistors, capacitors, battery and the signal generator in convenient locations following the schematic of Fig. 1. (To rid the cursor of the component after placing it on the editor page, click anywhere on the page else, right click and select "Cancel Mode"). Then, place the ground symbol from the Basic tab and a Voltage pin from the Meters tab.
4. Double-click the resistors, capacitors and transistors in succession and set the parameter value of each component in the component dialog box. For transistors ensure that the right part no. is selected. **Note:** For megohms, use "M", for milliohms use "m", for microfarad use "u" and for nanofarad use "n". Connect the components using the Wire symbol in the menu bar, according to the schematic in Fig. 1. Then, place a single pole single throw (SPST or ON-OFF) switch from the Switches tab. Set the switch in ON position.
5. Double-click the power supply symbol and set to 25V DC. Next, double-click the signal generator symbol and enter DC = 0V, Signal (select Sinusoidal) AC = 20millivolts by entering 20m and set the frequency to 10kHz by entering 10k.

Check the node voltages

6. The Electric Rules Check (ERC) is performed by clicking Analysis in the menubar and then clicking ERC. This will show up errors and warning messages if the schematic is not proper. Correct the schematic based on the summary displayed by the ERC.
7. Next, record the DC node voltages V_{CE}, across R_c and across R_e and current I_C by clicking Analysis →DC Analysis →Table of DC Results.

Frequency Response with the switch S1 in ON position.

8. **In PSpice:** Click the "Analysis" in the menu bar and then click "Setup" and select "AC Sweep" and set the starting frequency to 10 Hz and end frequency to 1 GHz and select radio button "Decade" and then close the box.
   Then, click "Analysis: and click "Simulate" and when the Analysis dialog box opens, select "AC". Select "Traces" button in the taskbar and click "Delete all traces" and then select "Adding Traces" and then select [B].

9. **In TINA :** Click "Analysis" in the menu bar and then click "AC Analysis" and then "AC Transfer Characteristic" and set the starting frequency to 10 Hz and end frequency to 1 GHz. Set the Sweep type to "Logarithmic" and click OK.
   A graph of gain in dB vs. Frequency in Hz will be displayed. Sketch the graph obtained on graph paper of *only* the response with the switch in ON position.
10. Click "Analysis" in the menu bar and then click "AC Analysis" and then "Table of AC Results" and note the voltage of the generator v_i, the voltage at the amplifier

output v_o, and calculate the gain in dB using the expression 20 log v_o/ v_i and verify that it tallies with the value in the graph at 10kHz.
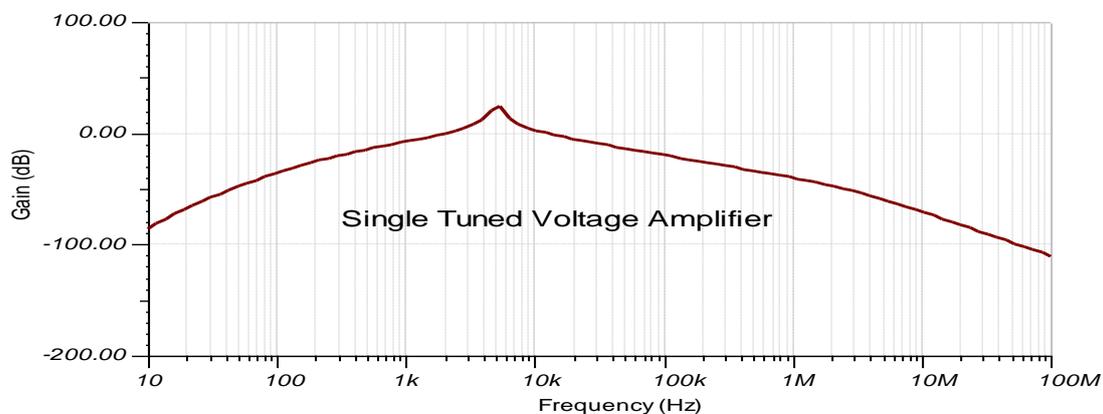
Sketch the output characteristics and draw the DC load line and mark the quiescent point of operation indicating I_C and I_B at this point from the values recorded in the previous step in DC Analysis.

11. Next, set the switch to OFF position i.e with the bypass capacitor disconnected from across the emitter resistor and run step 6. Note down the gain (in dB) at 10 KHz.

Additional exercise

12. Set the switch again to ON position.

13. From the menu, double click the signal generator and set the input sinusoidal frequency to 10kHz. On the menu bar click Test & Measurement (T&M) and click "Oscilloscope".

14. Click Run and then Auto (bar at the bottom of the virtual oscilloscope, and set connection to AC) and set the V/div and Time/div till a clean sinusoidal trace appears. Then press the "stop" button to freeze the oscilloscope trace and using the B button in the Cursor box and with the ON button pressed in the Cursor box, measure the amplitude of the traces taking care with the V/div mentioned at the top of the screen.

15. Next, change the amplitude of the signal generator to say, 500 mV and again obtain a trace and record your observations of the trace.

MODEL GRAPH

RESULT

1. The frequency response of the Single Tuned Voltage Amplifier is measured and the graph plotted.

2. From the graph.

i) The gain at 10kHz = ……..dB.