PH 211 Electronics Laboratory I

Instruction Manual

Index

DEPARTMENT OF PHYSICS

INDIAN INSTITUTE OF TECHNOLOGY, GUWAHATI

July 2023-Nov 2023

General Instructions to Students

1. On the very first day of the lab familiarize yourself with the power supply, function generator, oscilloscope, bread board, and digital multimeter (DMM). You may request for the copies of respective manual. You may also request the Teaching Assistant or the instructor to guide you in learning these basic operations.

2. With the help of DMM learn to check the diode and transistors and to measure the value of resistance.

3. The instruction manual provides the necessary information to perform the experiments. However alternate circuits exist for most cases and students are encouraged to try out circuits other than given in this manual (with prior permission from the instructor). The procedure given is brief. Instructions given in italics are for self-study. Do try them if you want proficiency in electronic circuitry.

4. Before attending the lab **read the instruction manual THOROUGHLY** and **CAREFULLY** for analyzing the circuits to be used. You should consult any of the good text or reference books on the subject in advance. This will help you to have tentative estimates of the voltages and currents you are going to handle and enable you to set the measuring instrument without trouble.

5. Derive the relevant formula or workout the relevant waveforms expected from the experiment.

6. You should bring with you sufficient number of A4 size white papers, graph sheets, tracing paper, for compiling the report and other stationery items required for data recording and analysis.

7. The format of the report should be:

    a) Name Roll No. Date of Experiment
    b) Experiment title:
    c) Objective/Aim:
    d) Formulas, if any, with brief description
    e) Equivalent Circuit(s) if necessary
    f) Expected waveform as a function of input if applicable
    g) Observation Table(s)
    h) Input/Output waveform traces wherever necessary
    i) Graph(s) with proper labelling
    j) Calculations, if any
    k) Summary of results
    l) Brief discussion of results
    m) Suggestion(s) / New circuit idea pertaining to the experiment / Specific precautions

8. You are expected to come prepared with points (a) to (f) of above and get it signed by the instructor before starting the experiment. ~~Five~~ 10 marks are reserved for the same.

9. You have to complete the report and submit in your FOLDER FILE on the scheduled date of experiment

10. Observations should be signed by either TA or the instructor available

11. The performance in this course will be evaluated on the basis of DAY-to-DAY lab activities, a Quiz, and the final end-semester exam. (40 Marks are for lab reports, 10 marks Quiz and 50 marks for the end-semester exam).

12. Any kind of feedback on the improvement of this course is always welcome.

Course Instructors - Dr. Sovan Chakraborty and Prof. Sunil K Khijwania
Technical Staff – Mr. Hemanta Medhi and Mr. Prasenjit Dey

References:

1. P. B. Zbar and A. P. Malvino, Basic electronics: A text-lab manual, Tata McGraw Hill, 1983.
2. P. Horowitz and W. Hill, The Art of Electronics, Cambridge University Press, 1995.
3. R. A. Gayakwad, Op-Amps and Linear Integrated Circuits, Prentice Hall of India, 2002.

Expt. 01: Single Stage Amplifier

The objectives of this experiment is to observe the operating characteristics of the common emitter (CE) configuration and to learn how it can be used for small signal amplification.

Aim: To (i) determine voltage gain of the amplifier (ii) find out the clipping voltages for positive and negative polarity peaks of the output signal, (iii) study the frequency response to calculate bandwidth for sinusoidal waves, (iv) compare the input and output frequency spectrum.

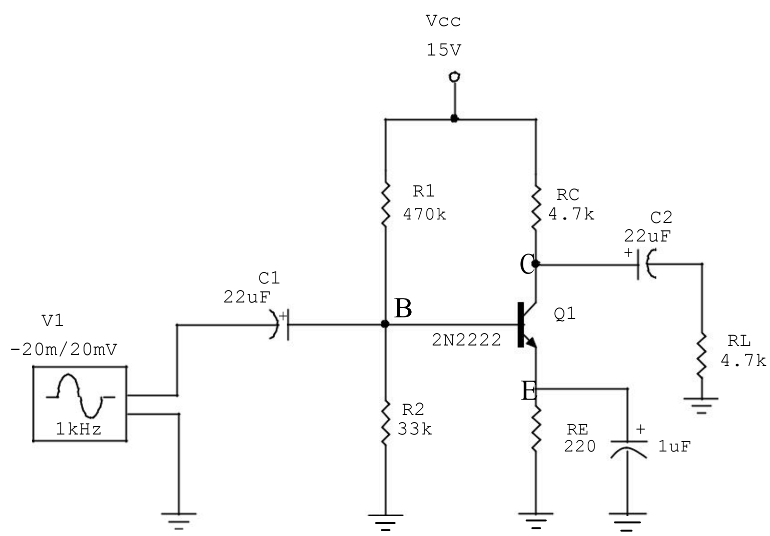


Fig. 1 Common emitter amplifier

Procedure:

1. Connect the input sinusoidal signal at point B as shown in Fig.1 using DC blocking capacitor. Connect load resistance at C with another DC blocking capacitor as shown in the circuit to take output across the load.

2. Connect input and output signal to the oscilloscope. Calculate the gain of this amplifier by taking the ratio of the output and the input voltage amplitudes. Make sure that both the input and output sine waves are not clipped or distorted in any way. If they were clipped reduce the amplitude of the input sine wave such that waves are smooth without any distortion and clipping. Also note the polarity of output sine wave relative to the applied input signal.

3. Slowly increase the amplitude of the input sine wave until the output sinusoidal wave begins to clip. Note the voltage at which clipping is on the positive or negative polarity peaks.

4. Keep increasing the input voltage until the other polarity peak of the sine wave output begins to clip and note down the voltage at which it occurs.

5. Tabulate the variation of voltage gain with frequency of the input for sinusoidal wave and plot frequency vs gain curve. Calculate the bandwidth and lower and upper cut off frequency values.

Expt. 02: Push Pull Amplifier

Aim: To obtain the maximum power output of the given push pull amplifier and its efficiency. To draw load versus power output curve and to study the cross over distortion.

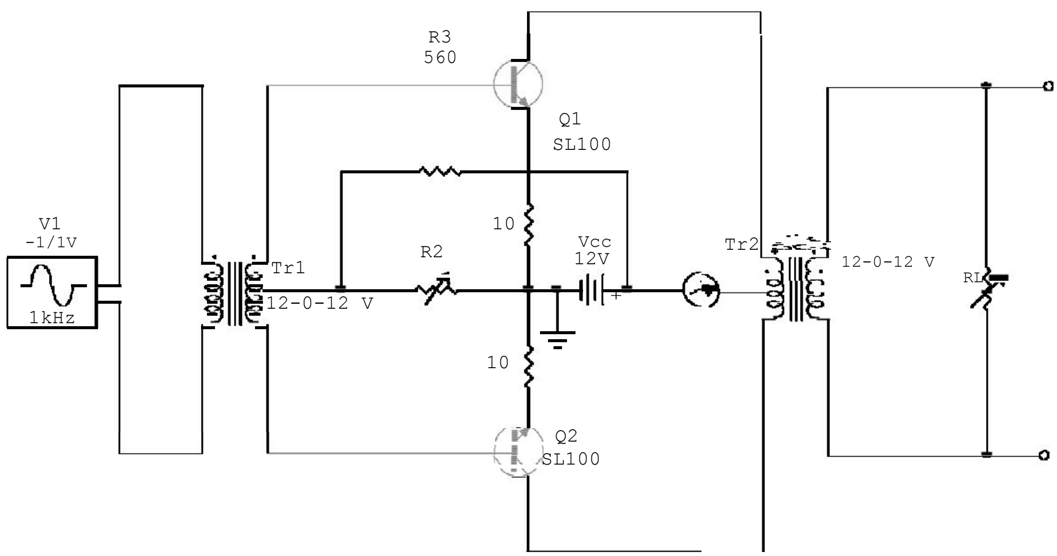


Fig.3 Push Pull Amplifier

Procedure:

1. Connect the DC supply (12 V) to the circuit form the source.
2. Apply a sinusoidal signal of frequency 1 kHz with amplitude say 5mV at the input of power amplifier.
3. Connect a loud speaker at the output across the secondary of output transformer Tr_2
4. Adjust the resistance R_2 to hear properly audible sound from the speaker and if you succeed in this, assume that the circuit is under working condition.
5. Replace the loud speaker with a load (R_L) impedance.
6. Measure the output voltage. Calculate the output power and percentage of efficiency of the circuit.
7. Repeat s.no. 6 for two other load resistors.

Exercise: Design a push pull circuit diagram to overcome the crossover distortion.

Expt. 03: Arithmetic and Amplifier Circuits using Opamps.

Aim: To construct (a) an inverting amplifier, (b) a non inverting amplifier, (c) adder circuit and (d) Subtractor circuit using 741 opamps.

(Note: Draw all the equivalent circuits and work out the expressions for the output voltages and the voltage gain of the amplifier, final values of gain can be worked out after measuring the actual resistances used).

Circuit diagrams:

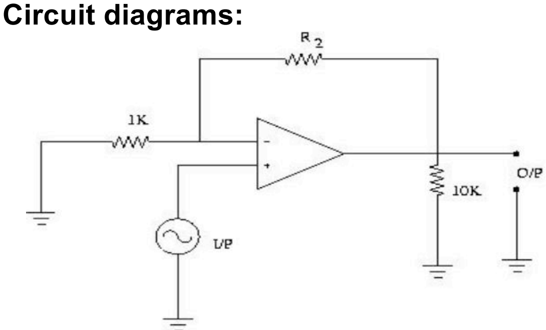


Fig: 3.1 Non inverting Amplifier

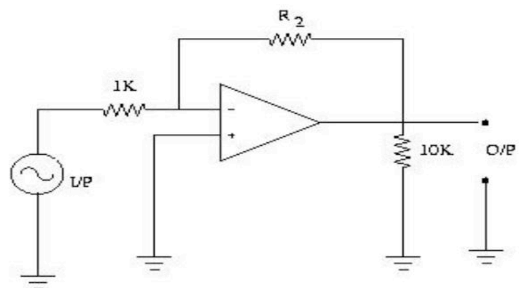


Fig: 3.2 Inverting Amplifier

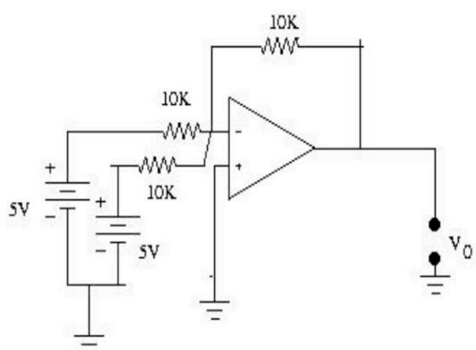


Fig: 3.3 Adder circuit

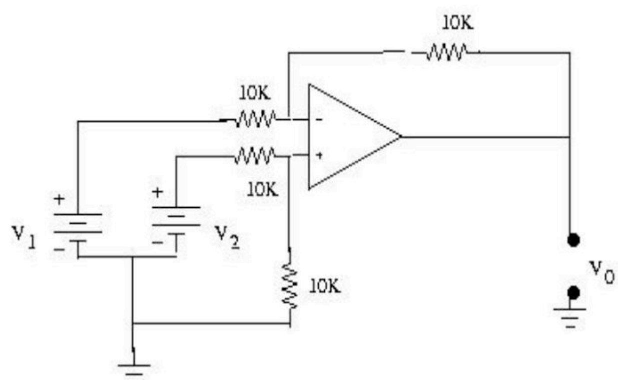


Fig: 3.4 Subtractor Circuit

Procedure:

1. Make the non-inverting amplifier circuit as shown in fig.3.1. Give a d.c. input of say 2 V and measure V_0. Repeat the above step for three different R_2 values and compare with theoretical value. Repeat the measurement by using a sinusoidal input signal V_i with frequency 1 kHz and within a peak to peak voltage of 1 V. Trace the input and output signals. Measure the peak to peak voltage of output signal V_0.

2. Make the inverting amplifier circuit as shown in fig. 3.2. Give a d.c. input of say 2 V and measure V_0. Repeat the above step for three different R_2 values and compare with theoretical value. Now give a sinusoidal input signal V_i with frequency 1 kHz and a peak to peak voltage within 1 V. Trace the input and output signals. Measure the peak to peak voltage of output signal V_0.

3. Make the adder circuit as shown in fig. 3.3. Set V_1= +1V and V_2=0. Measure the output voltage. Repeat the measurement for V_2 = 1, 2, 3 and 4V. Check the output voltage and compare it with $V_0 = -(V_1+V_2)$, the theoretical value. Tabulate the experimental output voltage and the excepted (theoretical) values. *Can you construct appropriate inverter circuit such that the output is* $V_0=V_1+V_2$

4. Make the subtractor circuit as shown in Fig. 3.4. Set V_1=0, and measure the output voltage V0 for V_2= 0, ±1, ±2, ±3, and ±4V. Tabulate the input and output voltages.

   Compare the measured output voltage with the expected (theoretical) voltage.

Exercise:

Design the circuit diagram to calculate the off-set voltages, CMRR for op-amp.

Expt. 04: Integrator and Differentiator Circuits using Opamps.

Aim: To perform (a) integration and (b) differentiation of analog signals using an operational amplifier (IC 741).

> **(Note: Draw all the equivalent circuits and work out the expressions for the output voltages and the voltage gain of the amplifier, final values of gain can be worked out after measuring the actual resistances used).**

Circuit diagrams:

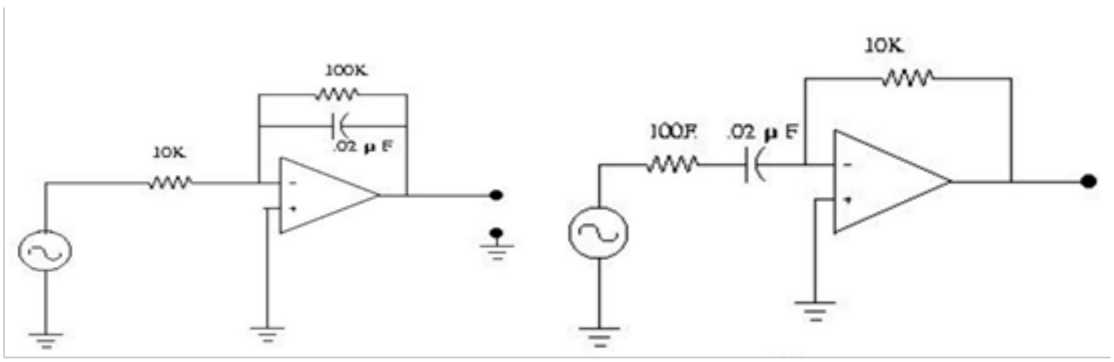


Fig: 4.1 Integrator Circuit **Fig: 4.2 Differentiator circuit**

Procedure:

1. Connect the integrator circuit as shown in Fig. 4.1. Apply a sinusoidal input signal V_i with frequency 1 kHz and peak to peak voltage 5V. Trace the input and output signals. Measure the peak to peak voltage of output signal V_0. Tabulate the readings. Repeat the experiment f or square and triangular waves . Repeat for C= 0 . 0 4 7 µF and 0.1µF. Calculate the output voltage theoretically and compare with the experimental data.

2. Connect the differentiator circuit as shown in Fig. 4.6. For sine wave, square wave and triangular wave inputs V_i (1 kHz and V_{pp} = 5V), measure the peak to peak output voltage. Trace the input and output signals. Calculate the theoretical output data.

Exercise:
Derive the expressions for the gain of the above circuits. Suggest the ways for the gain to be independent of frequency.

Expt. 05: Feedback Amplifier Circuits

Aim: The aim of the experiment is to study the effect of various feedback configurations on amplifier output.

Objective: To construct amplifier circuits with different feedback configuration and study the effect of feedback on mid-band gain, cutoff frequencies and gain-bandwidth product.

Circuit analysis (to be completed before coming to the lab)

Draw all the equivalent circuits, identify different feedback configuration and do complete circuits analysis to calculate the gain and feedback ratio whenever applicable before coming to the laboratory.

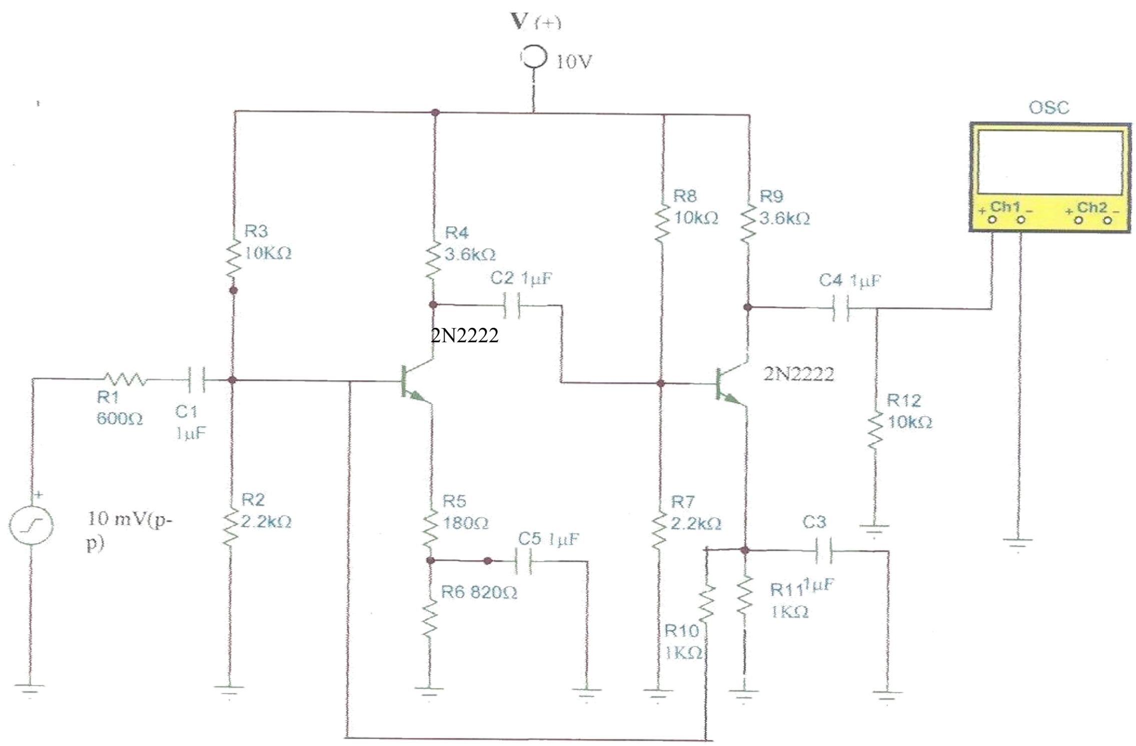


Fig: 5.1

Expt. 06: Filters

Aim: To study (a) the voltage gain as a function of frequency for low pass filter, (b) the voltage gain as a function of frequency for high pass filter and

Equipments: DC power supply, (+/-)15Volts, bread board, CRO, DMM, function generator.

Circuit diagram: **(a)**

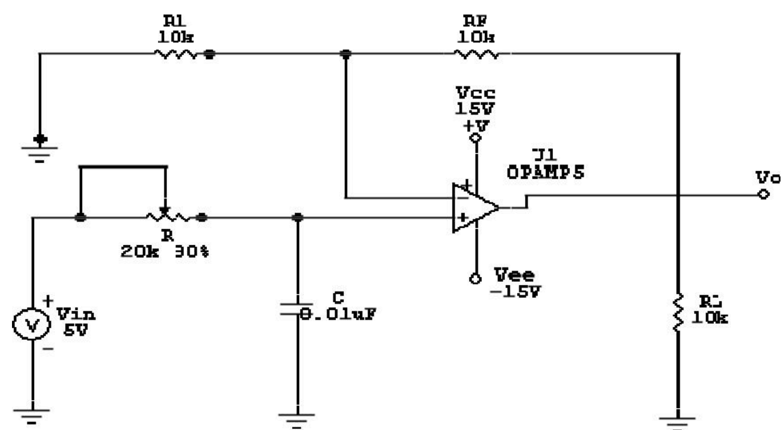


Fig: 6.1 Low Pass Filter

(b)

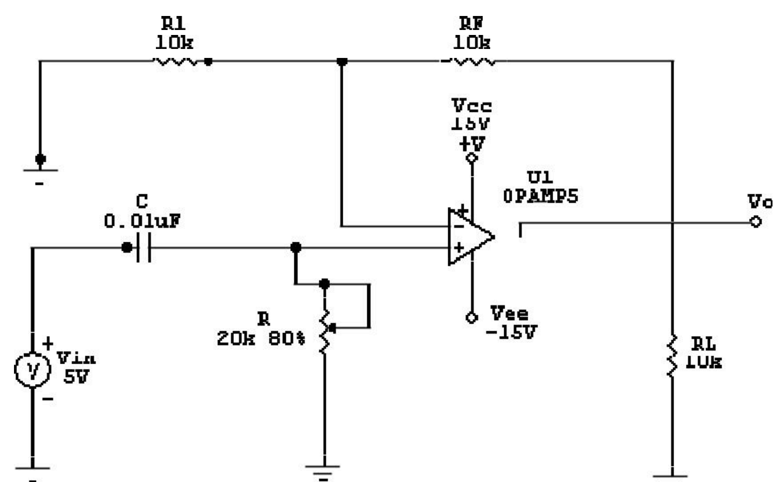


Fig: 6.2 High Pass Filter

Procedure:

1. For the low pass filter, make the circuit as shown in Fig. 6.1 Study and plot the voltage gain as a function of the frequency. Calculate the allowed frequency band for the low pass filter.
2. For the high pass filter, make the circuit as shown in Fig. 6.2 Study and plot the voltage gain as a function of the frequency. Calculate the allowed frequency band for the high pass filter.

Pin-out diagrams for IC741 chip are given in Appendix I.

Precaution: list out precautions taken by you. Write down the special techniques or simpler circuits followed by you if any.

Expt. 07: Multivibrators

Aim: To construct a (a) astable and (b) monostable multivibrator using IC 555.

Circuit Diagram: (a) Astable Multivibrator

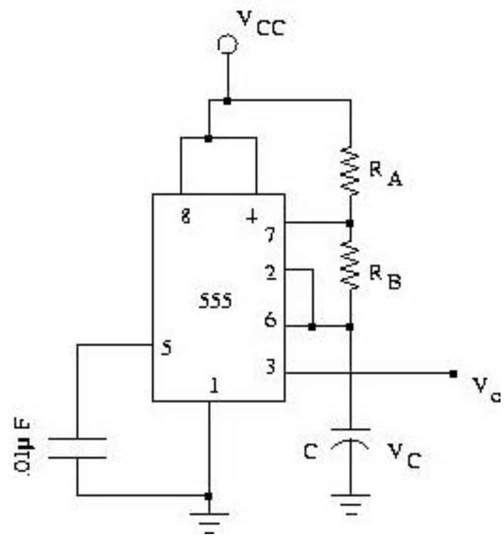


Fig: 7.1 Astable M.V. (R_A=1kΩ,R_B=3.3kΩ, 10kΩ, 18kΩ)

(b) Monostable Multivibrator

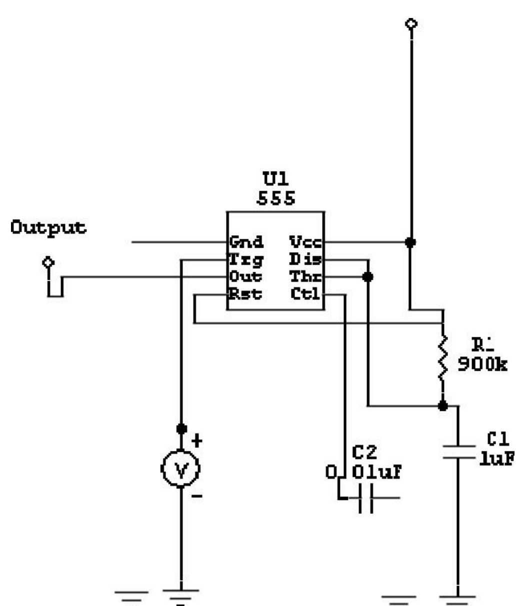


Fig: 7.2 Monostable M.V.

Procedure:

(a) Assemble the astable circuit shown in Fig. 7.1. Trace the output waveform. Try to use the control voltage terminal and vary the output pulse width and observe the output waveform. Calculate the frequency of the output waveform and the duty cycle. The square wave output will have frequency $f = 1.4/[C\,(R_A+2R_B)]$.
Compile the results and enclose the traced waveform.

(b) Construct the circuit as shown in Fig 7.2. Connect the circuit output to the CRO. Trace the output wave form. Calculate the width of the output pulse. What should be the value of R in the same monostable circuit that generates pulses with width half the value.

Precaution: List out the precaution and any special steps followed by you.

Pin-out diagram for each IC555 chip is given in Appendix I.

Expt. 08: Phase shift and Wien Bridge Oscillators

Aim: To construct phase shift and Wien Bridge Oscilltor circuits using opamp.

Circuit Diagrams:

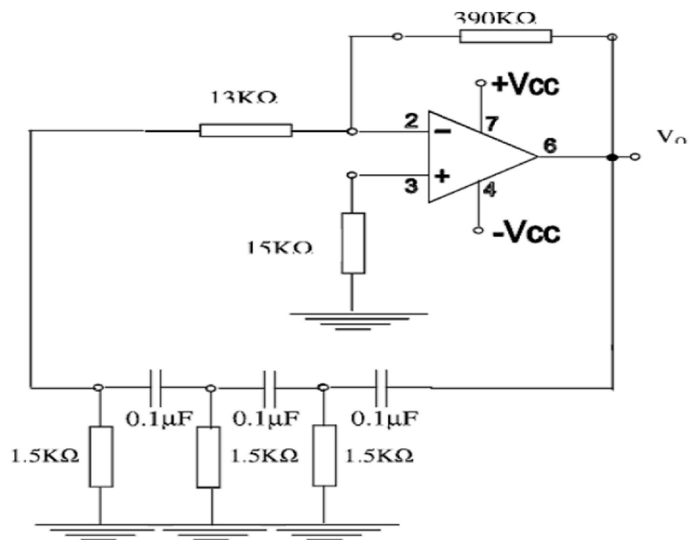


Fig: 8.1 Phase Shift Oscillator

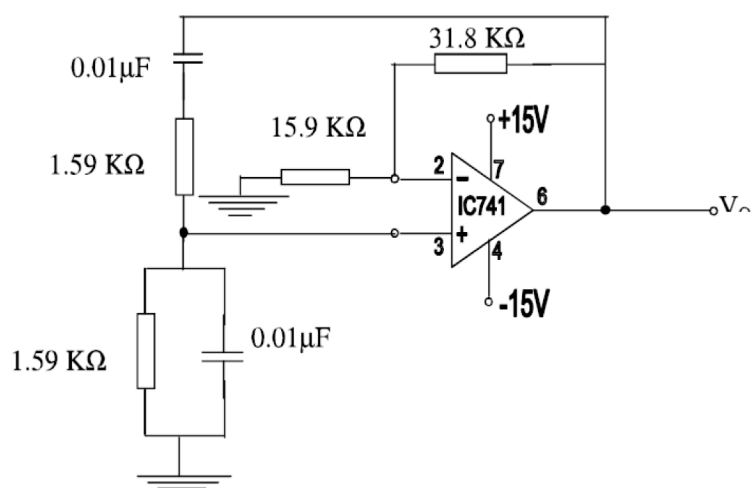


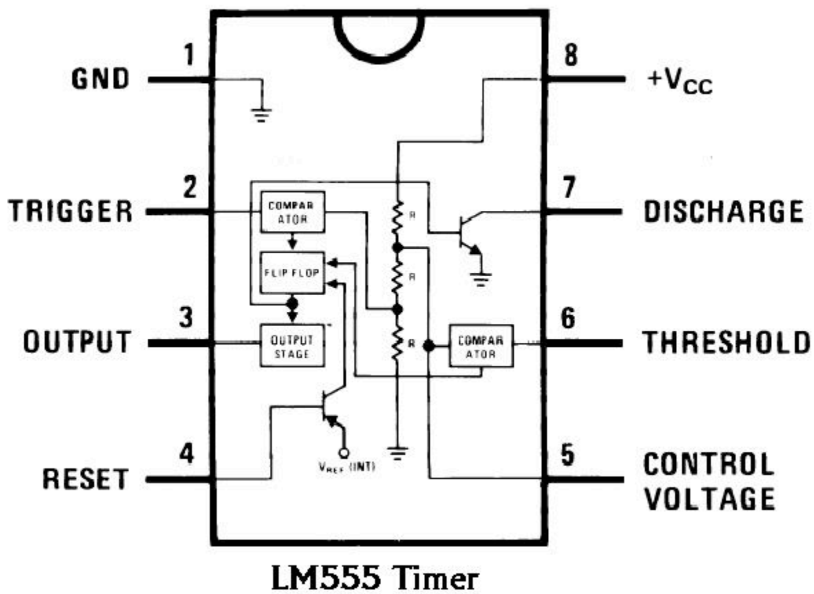
Fig: 8.2 Wien Bridge Oscillator

Procedure:

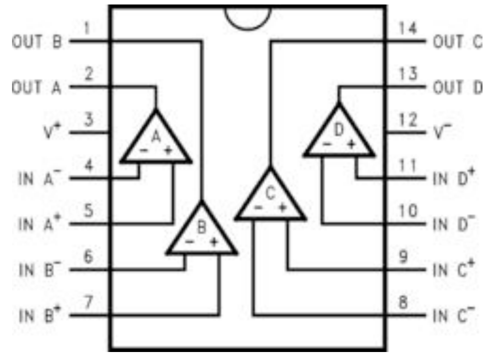
1. Make the circuits as given in diagrams. Check the output, measure the frequency of the output signal and compare with the theoretically calculated frequency.

Appendix I:

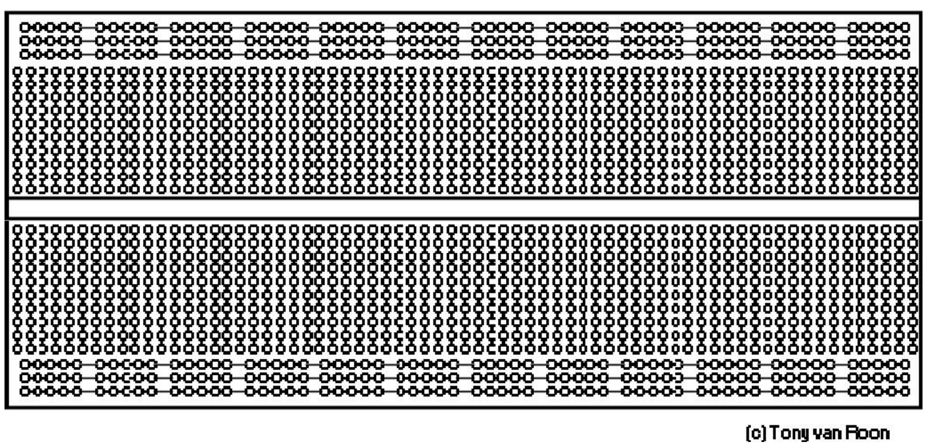
Pin diagram of IC 741



Pin diagram of IC 555



Pin diagram of LM339



Bread board layout