PH 210 Electronics Laboratory I
Instruction Manual

Index

		Page No
General Instructions		2
Experiment 1	**Common Emitter (CE) Amplifier**	4
Experiment 2	**Multistage amplifier: Cascade of two CE stages**	7
Experiment 3	**Push pull amplifier**	9
Experiment 4	**OP Amp circuits**	11
Experiment 5	**Filter circuits Feed back circuit**	13
Experiment 6	**Feed back amplifiers**	15
Experiment 7	**Oscillator & Multivibrator Circuits**	17
Experiment 8.	**Modulation**	20
Experiment 9	**Controller circuit**	22
Appendix		24

Department of Physics

Indian Institute of Technology Guwahati

January_April 2011

Instructors: Dr. A.P. Vajpeyi and Dr. S. Ravi

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 these 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 analyses.

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 with proper labeling
   (i) Graph(s) with proper labeling
   (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. Ten 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 the instructor.

11. **The performance in this course will be evaluated on the basis of DAY-to-DAY lab activities, Quiz, and the final end-semester exam. (40 Marks are for lab reports, 20%marks for Quiz and 40% marks for the end-semester Lab exam).**

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

Experiment 1 Single Stage CE Amplifier

The aim of this experiment is to study the single stage amplifiers; using BJT in common emitter (CE) configuration and to learn its application as a small signal amplification.

Objective: To (i) measure the operating points, (ii) measure Gain of the amplifier as a function of frequency and determine the band width for sinusoidal, square and triangular waves, (iii) compare the input and output frequency spectrum and (iv) find out the clipping voltages for positive and negative polarity peaks of the output signal,

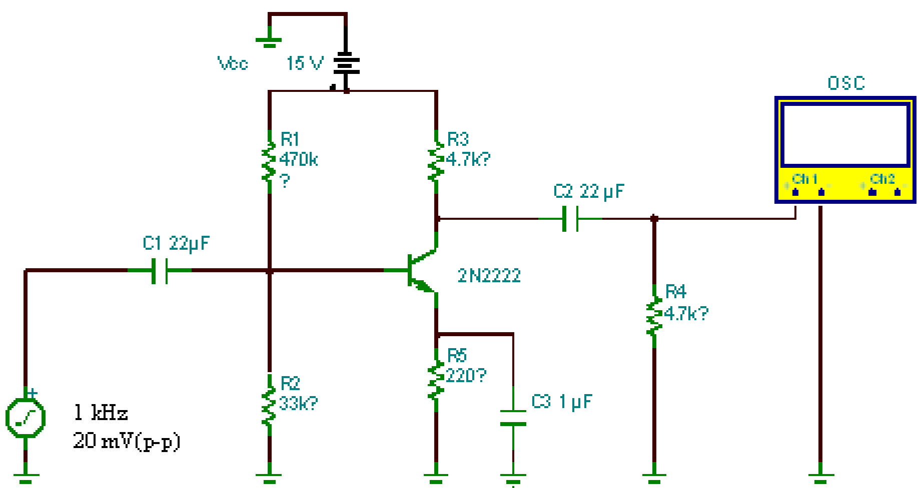


Fig 1.1 Common Emitter amplifier

Circuit analyses (to be completed before coming to the Lab):

Draw the DC and AC equivalent circuit for Fig 1.1. Work out the equation for the load line and the operating point. Draw the load line and mark the operating point on it. Analyse the ac circuit and calculate the expression for voltage gain.

Procedure:

A. To record the operating points of the transistor and confirm that it lies on the load line.

Assemble the circuit as shown in Fig 2. and record the base current I_B, collector current I_c and collector to emitter voltage V_{CE} or V_C and base to emitter voltage V_{BE}. Verify that operating point lies around the center of the load line.

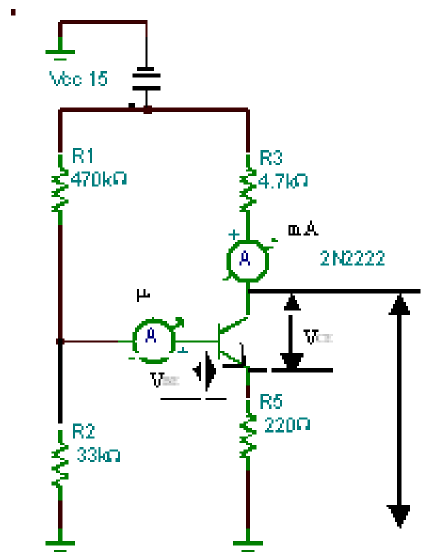


Fig.1. 2. Biasing circuit

B. To measure the gain of the amplifier as a function of frequency

1. Assemble the circuit of Fig 1.1

2. Set the function generator for sinusoidal out put. With the help of a Tee connector and BNC cable display the function generator output (input to the amplifier circuit) on one of the channel of the scope. Set the function generator for ~ 1kHz and 20mV peak to peak signal

3. Display the out of the amplifier on second channel of scope.

4. Make sure that both the input and output sine waves are not clipped or distorted in any way. If it is so then reduce the input voltage from the function generator. until nice clean looking sine waves are displayed at both input and output terminals. Also note the polarity of output sine wave relative to the applied input signal.

5. Record the output signal as a function of frequency maintaining the input fixed at 20mV.

6. Trace the input and output signals for atleast two frequencies (the scale should be well labeled)

7. Calculate the gain of this amplifier by taking the ratio of the output and the input amplitude.

8. Plot the gain as a function of frequency and determine the band width of the amplifier. Mark the cut off frequencies on the graph.

9. Slowly increase the amplitude of the input sine wave until the output sinusoidal wave begins to clip. Note the voltage at which clipping is observed for the positive as well negative polarity. Trace the distorted signals for both the polarity.

Repeat the experiment for square and triangular waves and plot frequency vs gain curve. Calculate the bandwidth and lower and upper cut off frequency. Observe the difference in the bandwidth if any for these two signals compared to sinusoidal signal. Explain your results.

Experiment2. Multistage amplifier: Cascade of two CE stages

The aim of this experiment is to study a two stage RC coupled CE amplifier.

Objective: To assemble a two stage common emitter RC coupled amplifier and to measure the gain as a function of frequency and hence find the gain band width.

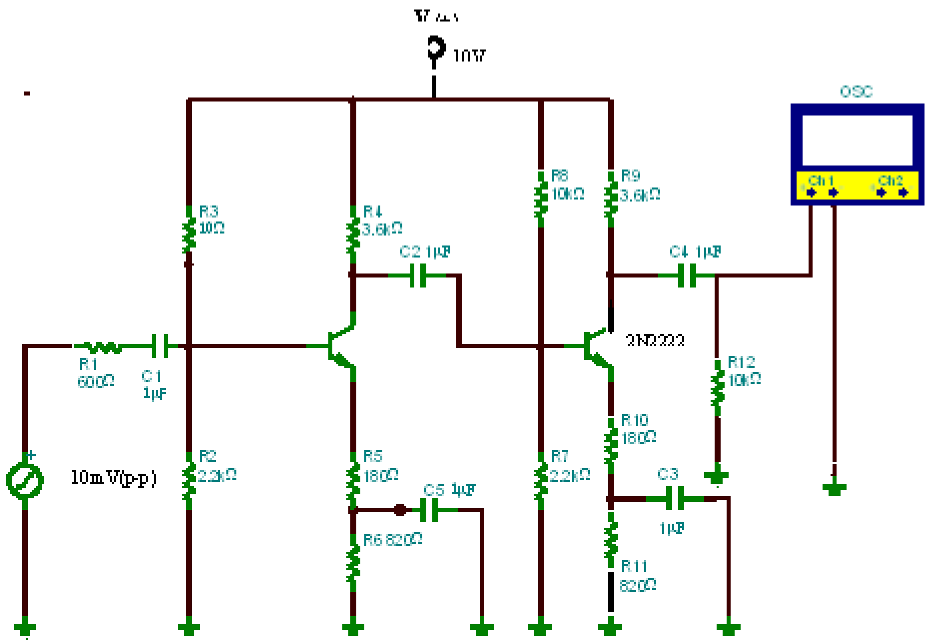


Fig. 2.1 Two-stage RC coupled amplifieramplifier

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

Draw the ac equivalent circuit and work out the expression for voltage gain for this amplifier.

Procedure:

1. Assemble the circuit shown in Fig.2.1.
2. Apply sinusoidal signal having peak to peak voltage~ 10mV to the input of the amplifier.
3. Display input as well as out put signal on to the CRO on two different channels simultaneously.

4. Measure the output voltage as a function of frequency keeping the input voltage fixed ~10mV(**why?**).
5. Plot the gain as a function of frequency.
6. Determine the lower and higher frequency cut off and hence the bandwidth of the amplifier.
7. Trace input and out put signals at two-three frequencies. (The traces should be well labeled).
8. Compare the frequency response of single stage amplifier of experiment 1 and the RC coupled amplifier.
9. Record the band width for triangular as well as square waves also.
10. Compare the frequency response of sinusoidal, triangular and square pulse.

Question:1. What are the advantages of RC coupled amplifier

Experiment 3.Push Pull Amplifier

The aim this experiment is to study a push pull amplifier mainly for audible range.

Objective: To assemble a push-pull amplifier and study the effect of biasing on cross over distortion. To study the variation of output power as a function of load resistnace and find the the maximum power output of the given push pull amplifier, find its efficiency. Find the range of the audible frequency for this amplifier.

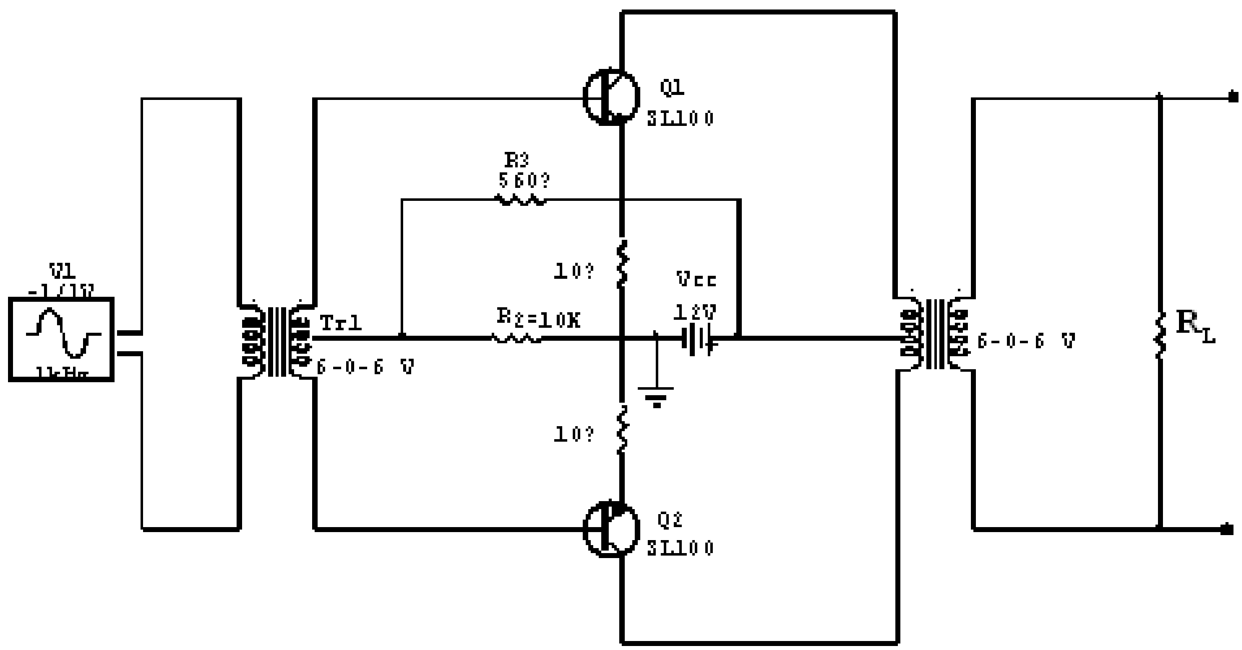


Fig.3.1 Push Pull Amplifier

Circuit analyses to be completed before coming to the lab.

Procedure:

1. Connect the DC supply (12 V) to the circuit (fig 3.1) from the source.
2. Apply sinusoidal signal of frequency ~1 kHz, amplitude 1V peak to peak to the input of power amplifier.
3. Turn variable resistance R_2 to observe the cross-over distortion. Trace it.
4. Tune R_2 in such a way so that the cross over distortion is removed (or brought to minimum. Trace it. **Henceforth do not disturb R_2.** (Why?)

5. Record output voltage as a function of R_L in the range of few ohms to 200kΩ (e.g 50Ω, 100Ω, 200Ω,---1.0kΩ,1.2kΩ , 1.3kΩ,---- 2kΩ, 3kΩ, 4kΩ---- 10kΩ, 50kΩ, 100kΩ and 200kΩ.
6. Plot the graph between load impedance (R_L) and output power (P_O).
7. Find the maximum power for the optimum load. Calculate the % efficiency.
8. Replace R_L with a speaker at the output, across the secondary of output transformer Tr_2
9. Try to hear the sound as a function of frequency and note down the frequency range for which the sound is audible.

Experiment No. 4 Operational Amplifier Circuits

Aim: The aim of the experiment is understand the working of operational amplifier and its use in analog computation

Objective: To construct the inverting and non inverting amplifier circuits using operational amplifier and perform various mathematical operations on analog signals

Circuits:

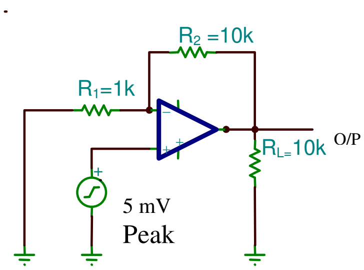


Fig. 4a Non-Inverting amplifier

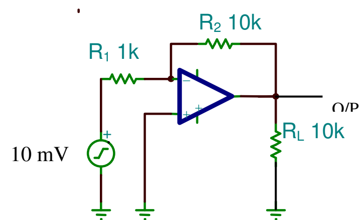


Fig. 4b Inverting amplifier

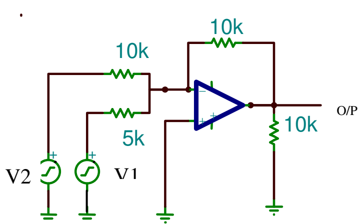


Fig. 4c Adder circuit (two ac signal)

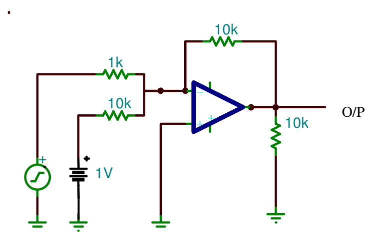


Fig. 4d Adder circuit (one ac and one dc signal)

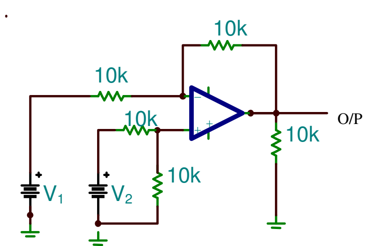


Fig. 4e Subtraction circuit (two dc signal)

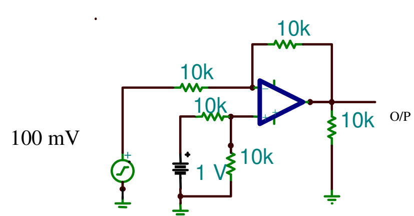


Fig. 4f Subtraction circuit (one ac and one dc signal)

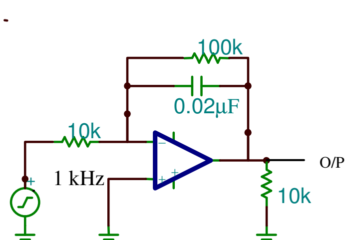


Fig. 4g Integrator circuit

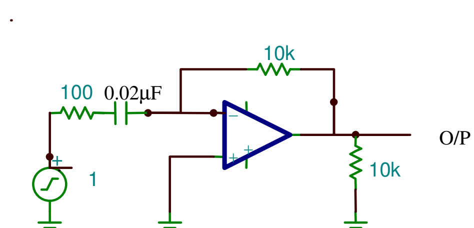
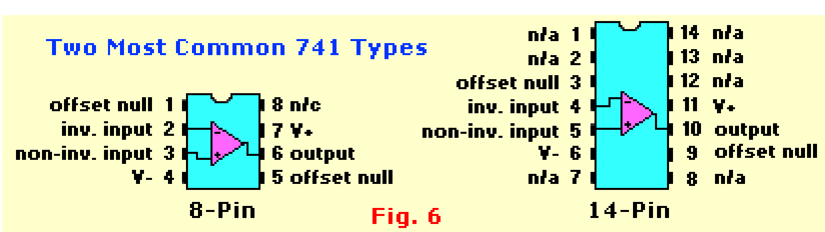


Fig. 4h Differentiator circuit

Note: **Circuit analyses should be done before coming to the laboratory.**
Trace the input and output waveforms from CRO screen. Do the necessary calculations and compare your results with the theoretical value.

Procedure:

1. Make a non-inverting amplifier circuit as shown in Fig 4a. Give an ac signal (frequency1KHz, 10 mV peak). Measure output signal amplitude and phase. Repeat the measurements for ac signals of 500mV peak, 1 KHz and 100mV 100KHz. Compare the gain with the theoretical value for all inputs.
2. Repeat the non inverting amplifier circuit for ac input 10mV, vary the frequency of the input signal from 100Hz to 1MHz in steps. Record the output and calculate gain as a function of frequency. Determine the bandwidth of the op-amp.
3. Make the inverting amplifier circuit as shown in Fig 4b. Feed the ac signal 10mv peak, 1KHz and record the output signal amplitude and phase. Repeat the measurements for Triangular waveform of 10mV peak,1KHz. Change the value of R_1 to 10K and repeat the measurements for Sinusoidal wave.
4. Make the circuit as in Fig 4c. Give two equal ac signals (derived from same function generator) of 10mV peak, 10KHz. record the output signal amplitude and phase.
5. Make the circuit as in Fig 4d. Now instead of two equal ac signals (derived from same function generator), take one ac signal of 10mV peak, 10KHz and other as 1V dc signal, record the output signal amplitude and phase. Repeat the measurement for 100mV peak ac signal keeping the dc signal same as 1V.
6. Make the circuit as in Fig 4e. Choose both input signals as dc signal. Let V1 be 1V. Choose V2 as ±1V. Verify the subtraction operation.
7. Make the circuit as in Fig 4f.. Choose one input signal as ac signal (sinusoidal ~100mV, ~1KHz) and other as 1V dc. Record the output signal amplitude (**take care of base line!**)
8. Make the circuit as in Fig 4g. Choose the input as 10mV, 1KHz sinusoidal ac signal. Calculate the phase difference between the input and output signal. Repeat the experiment for square and triangular wave ac inputs.
9. Make the circuit as in Fig 4h. Choose the input as 10mV, 5KHz sinusoidal ac signal. Calculate the phase difference between the input and output signal. Repeat the experiment for square and triangular wave ac inputs.



Pin diagram for IC741 op-am

Experiment No. 5: Filters

Aim: the aim of the experiment is to use the operational amplifier as filers of different frequency range.

Objective: To construct and study low, high and band pass filters using op-amp, determine the cut-off frequency (ies) and measure the voltage gain and phase shift as a function of frequency.

Circuit diagram:

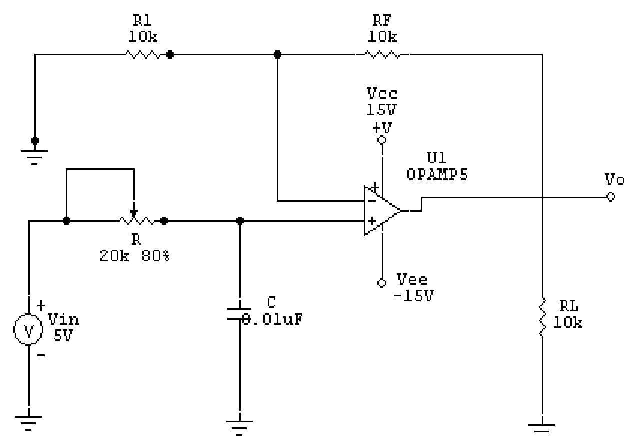


Fig: 5a Low Pass Filter

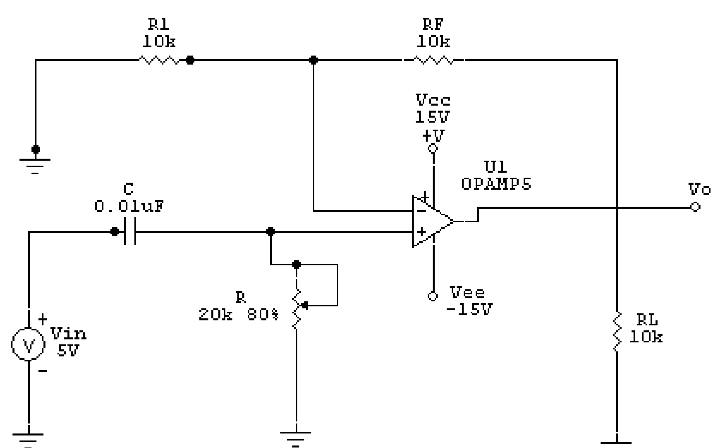


Fig: 5b High Pass Filter

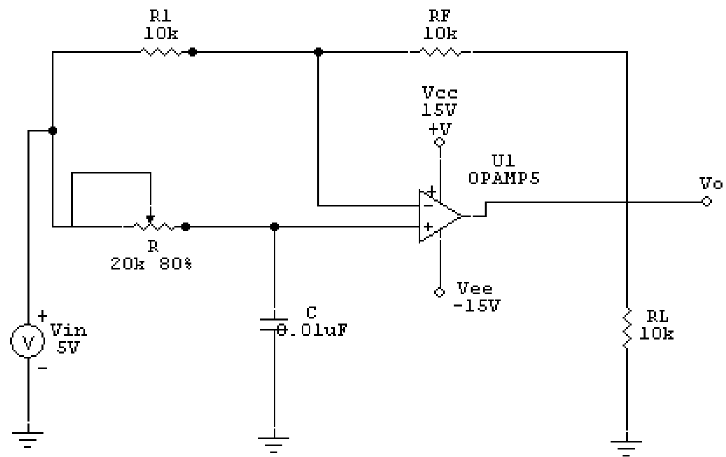
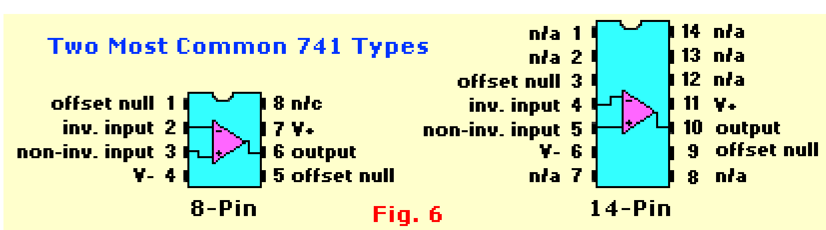


Fig: 5c All Pass Filter

Circuit analysis has to be done before coming to the laboratory.

Procedure:

1. For the low pass filter, make the circuit as shown in Fig. 5a. Feed ac signal of 5V peak to peak and measure the output voltage by varying the frequency of the input signal from 100Hz-1MHz. Study and plot the voltage gain as a function of the frequency. Determine the cut off frequency and calculate the allowed frequency band for the low pass filter. Check if there is any change in phase shift of the signal with frequency.
2. For the high pass filter, make the circuit as shown in Fig. 5b. Repeat the experiment as for low pass filter.
3. For the all pass filter, make the circuit as shown in Fig. 5c. Repeat the experiment as in previous circuits. Measure and plot the phase shift as a function of the frequency.



Pin-out diagrams for IC741

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

6. Oscillator & Multivibrators Circuit

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

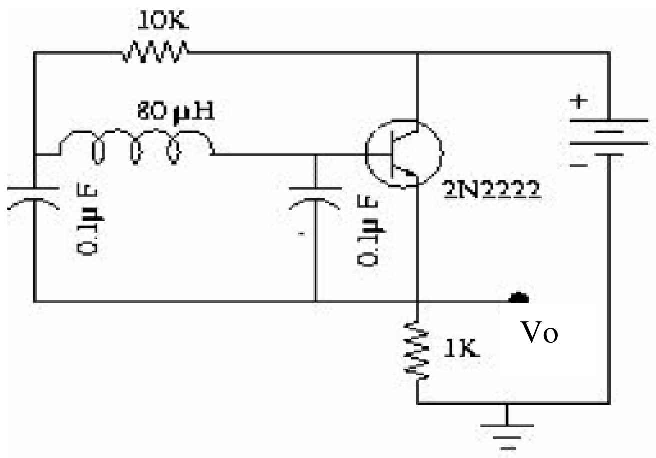


Fig: 6.1. Colpitt Oscillator circuit

Procedure:

1. Assemble the Colpitt oscillator circuit shown in Fig. 6.1. The frequency of oscillation is given by the expression, $f=1/(2\pi\sqrt{LC_T})$, where C_T is the total capacitance. Trace the Oscillator output. Repeat for different values of C_T (0.01μF, 0.02μF and 0.1μF). Measure the frequency of the oscillator using a CRO. Compare the experimental frequencies with the theoretical values. Compile the results and enclose the traced waveform.

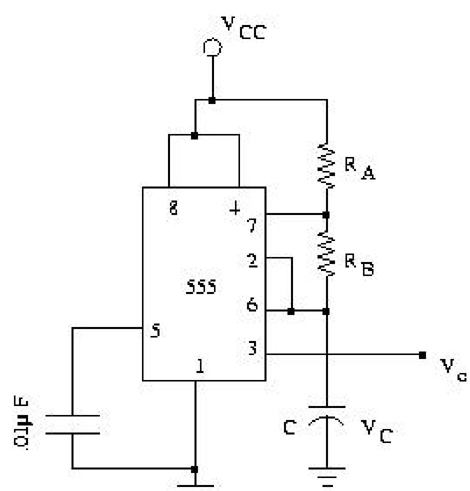


Fig: 6.2. Astable M.V. (R_A=1kΩ, R_B=3.3kΩ, 10kΩ, 18kΩ)

2. Assemble the astable circuit shown in Fig.6.2. 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)].Repeat for different R_A, R_B as listed in Fig 2 and C (0.01μF, 0.02μF and 0.1μF).
Compile the results and enclose the traced waveform.
Try-out: Redesign the circuit for operation at approximately 1 kHz with an approximate duty cycle of 50%. Show the calculations for R_A, R_B and C.

3. Construct the circuit as shown in Fig 6.3. Connect the circuit output to the CRO. Apply square wave from the Fs at V_s in the range of 1kHz –2kHz Trace the output wave form. Calculate the width of the output pulse. What should be the value of R for the in the same monostable circuit that generates pulses with width half the value.

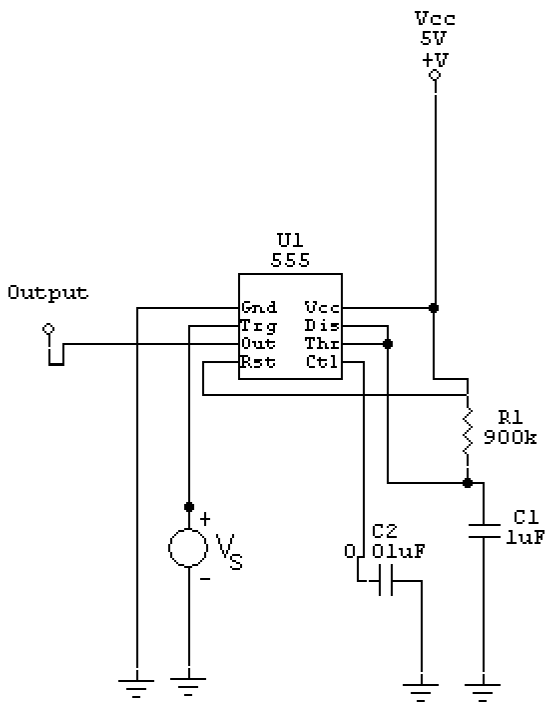


Fig: 6.3 Monostable multivibrator

Pin-out diagram for each IC555 chip is given below:

Experiment No.7: Feed Back Amplifier Circuits

Aim: The aim of the experiment is to study the effect of various feed back configuration 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 analyses (to be completed before coming to the Lab)
Draw the equivalent circuits, Identify different feed back configurations and do Complete Circuit analysis to calculate the gain and feed back ratio wherever applicable before coming to the laboratory.

Circuit diagram:

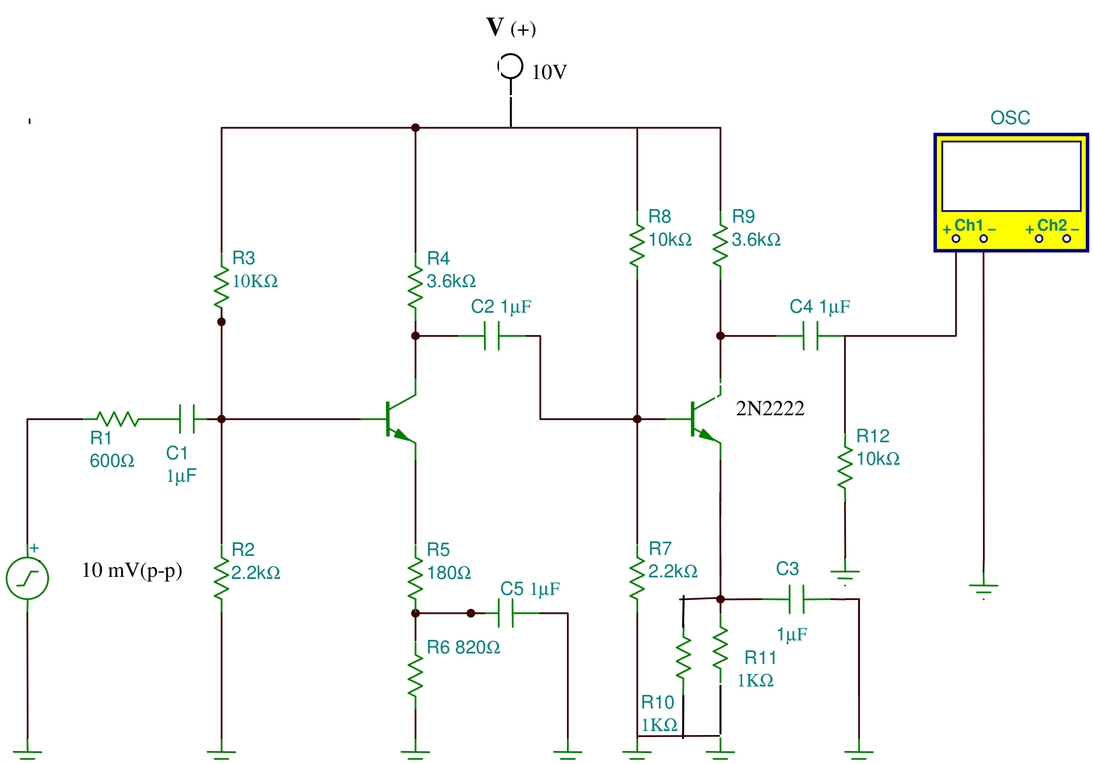


Fig. 7.1

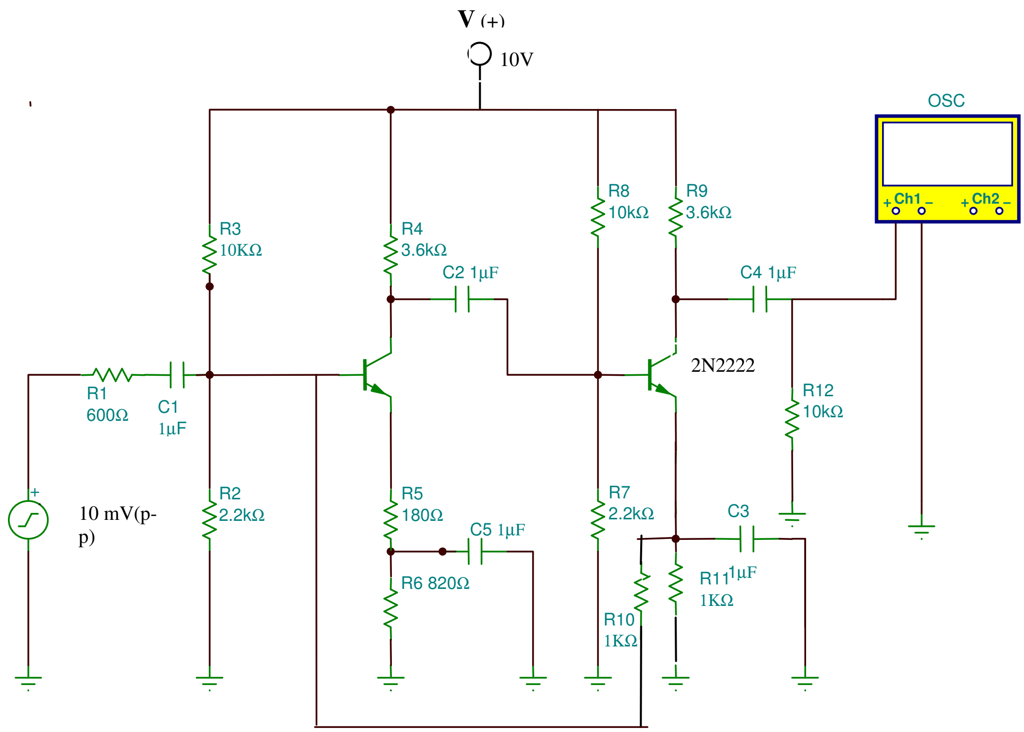

V (+)
10V
OSC
Ch1
Ch2
R3
10KΩ
R4
3.6kΩ
C2 1µF
R8
10kΩ
R9
3.6kΩ
C4 1µF
2N2222
R1
600Ω
C1
1µF
R12
10kΩ
10 mV(p-p)
R2
2.2kΩ
R5
180Ω
C5 1µF
R7
2.2kΩ
C3
R6 820Ω
R11
1µF
1KΩ
R10
1KΩ

Fig 7.2

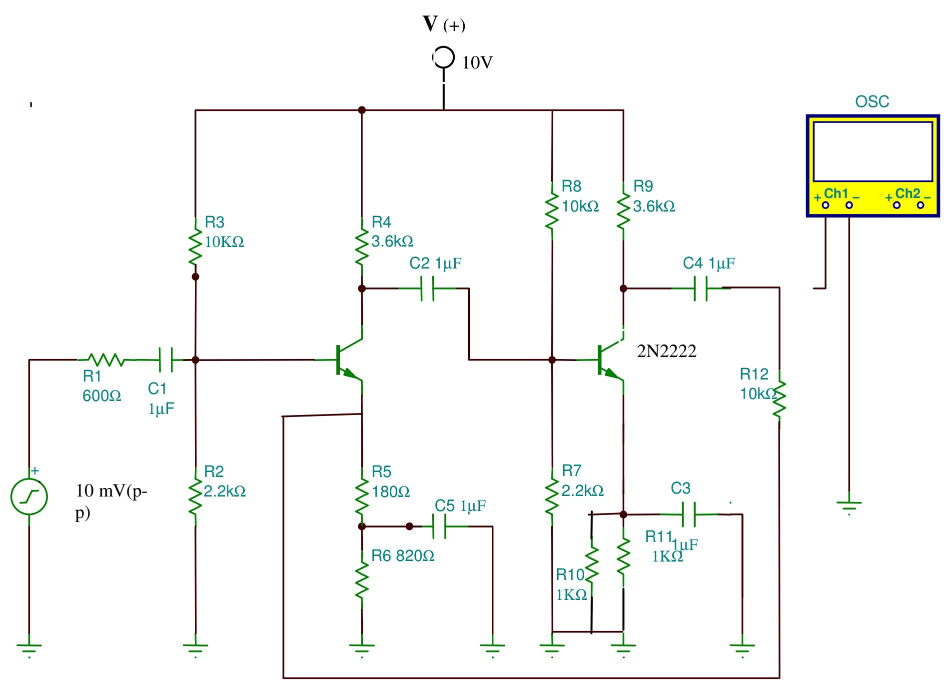

V (+)
10V
OSC
Ch1
Ch2
R8
10kΩ
R9
3.6kΩ
R3
10KΩ
R4
3.6kΩ
C2 1µF
C4 1µF
2N2222
R1
600Ω
C1
1µF
R12
10kΩ
10 mV(p-p)
R2
2.2kΩ
R5
180Ω
C5 1µF
R7
2.2kΩ
C3
R11
1µF
1KΩ
R6 820Ω
R10
1KΩ

Fig. 7.3

Procedure:

1. Assemble the RC coupled amplifier circuit as in Fig. 7.1. Give an input ac signal and see the output on the CRO screen. Increase the frequency till you get the maximum signal. Trace, input and output waveforms. Increase the frequency further, till the gain is equivalent to that corresponding to upper cutoff frequency. Trace the input and output signal at this frequency. Note down the cut off frequency. Now reduce the frequency, till you get the lower cut off frequency. Note down the lower cut off. Trace the input and output again at this frequency. Determine the mid band gain and band width for the amplifier circuit.
2. Now assemble the circuit as in Fig. 7.2 (for current feedback). Check a part of the output is being fed to the input. Once again, check the mid band gain (it is lower than that obtained for Fig. 7.1, why?). Trace the input and output at mid band gain frequency. Note down the and upper and lower cut off frequencies. Determine the mid band gain and band width for the amplifier circuit.
3. Now assemble the circuit as in Fig. 7.3(voltage feedback). Check a part of the output is being fed to the input. The feedback configuration is different than that in Fig. 7.2. Once again, check the mid band gain (it is lower than that obtained for Fig. 7.1, why?). Trace the input and output at mid band gain frequency, and note down upper and lower cutoff frequencies. Determine the mid band gain and band width for the amplifier circuit.
4. If time permits, record, frequency vs gain for each configuration and plot the graph.
5. Determine gain band width product in each case and compare it for different feedback configuration.

Question:

1. *What are the advantages and disadvantages of the feedback amplifiers. How will you choose the suitable feedback configuration for different applications.*
2. *What is the difference between the circuit of Fig 7.1 and that of the circuit used in second experiment?*

8. Modulation

Aim: To study the (a) Amplitude Modulation and demodulation of a sine wave signal (b) Pulse Width Modulation and demodulation of a square wave signal.

Circuits:

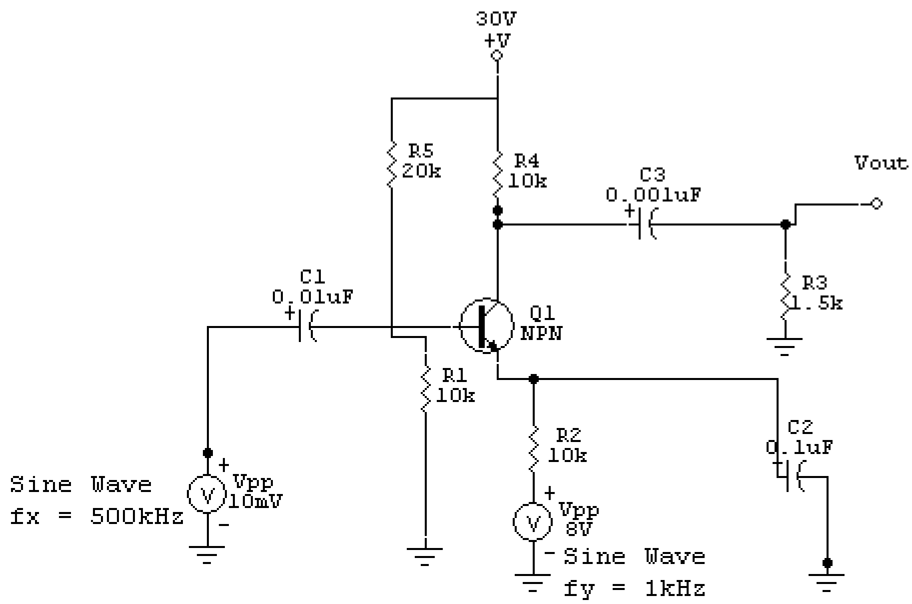


Fig.8.1. Amplitude Modulation

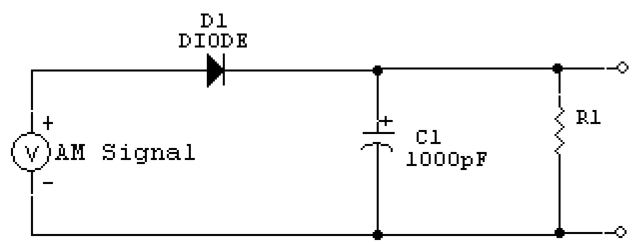


Fig.8.2. Amplitude Demodulation

Circuit Analyses (to be completed before coming to the lab.
Analyse all the circuits and work out the frequency spectrum (or possible frequencies for amplitude modulation and pulse width modulation) before coming to the lab.
Procedure:

1) Assemble the amplitude modulation circuit as shown in Fig. 8.1. Apply sinusoidal the carrier wave ~500kHz or more from one of the function generator (~20mVp-p) and the signal wave ~ in the range of 100Hz-500Hz. To get the stable display on CRO, trigger it with out put channel. Record the modulated

output and calculate the percent of modulation for various signal frequencies. Trace the modulated wave for two-three frequencies.

2) Connect the demodulating ckt of Fig 8.2 at the output of modulating circuit of Fig 8. 1. Record the demodulated signal. Check the frequency of demodulated signal and compare it with the original signal frequency. Trcae the modulated signal (trace should be of the same frequencies as above in step 1 for easy comparison.

3) For the pulse width modulation (PWM), assemble the circuit as shown in Fig. 8.3. Adjust the carrier (triangular wave) and signal inputs (sinusoidal) to generate a pulse width modulated signal of frequency 1kHz, 20kHz and 50 kHz and duty cycle of 40%. For the best result the ratio of carrier frequency to the signal frequency for the PWM should be around 25:1Trace the modulated signal, note the frequencies of the input signals.

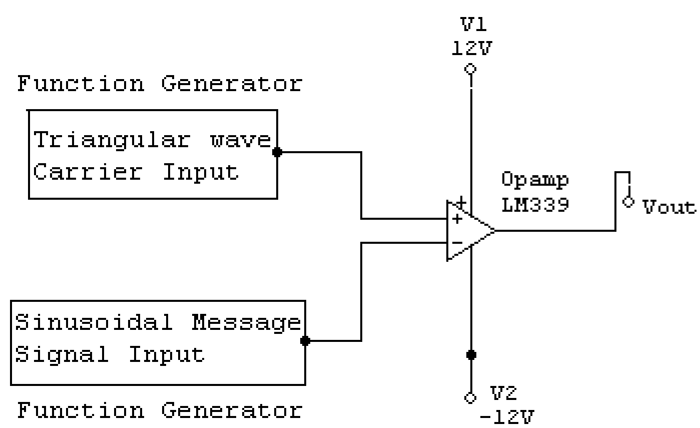


Fig. 8.3 Pulse Width Modulation

Questions:

1. What is the need of modulation. Give one example for each, amplitude modulation, frequency modulation and pulse width modulation.
2. What is the frequency spectrum of amplitude modulation, PWM modulation and frequency modulation?
3. What kind of filter is required for PWM?

Try Out:

1) *Design the demodulation circuit for pulse width modulation.*
2) *Design the circuit for frequency modulation.*

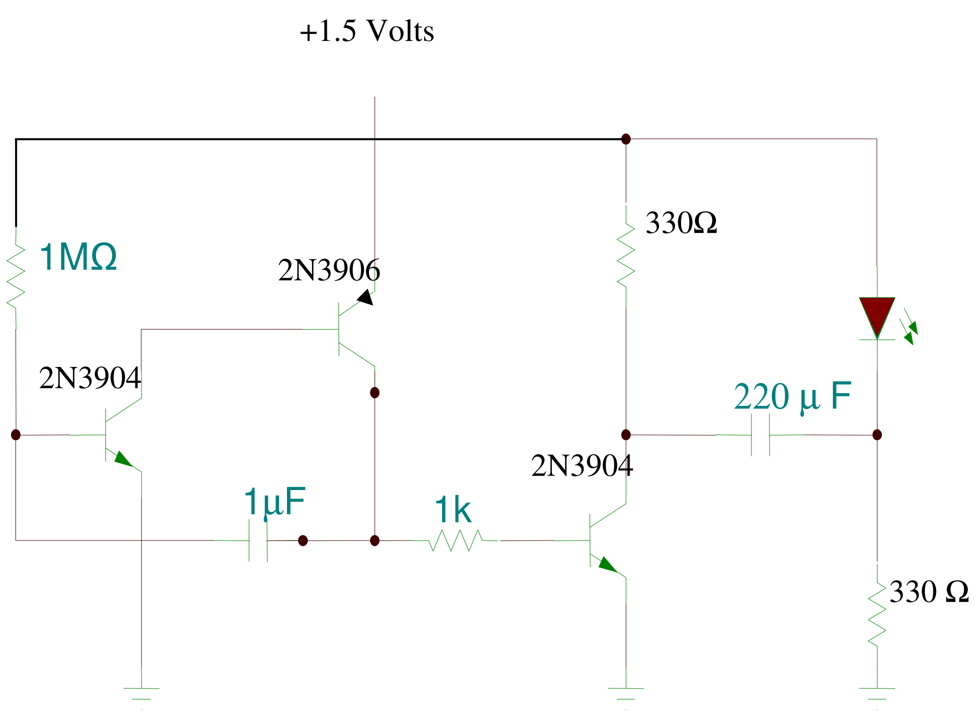
Experiment No. 9: Control circuits

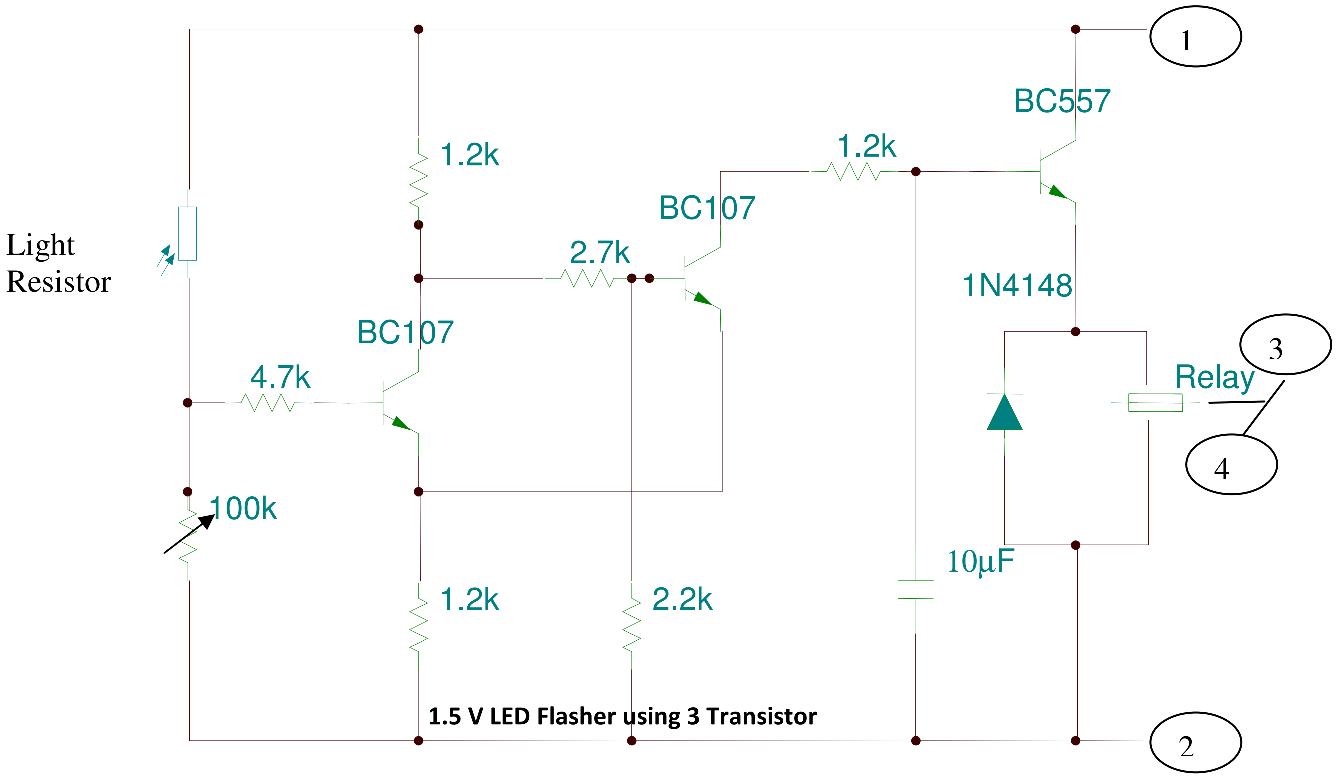
Aim: The aim of the experiment is to design a few control circuits.
(You may try your hands on any one of the control circuits)
You are given two control circuits in this experiment. The first circuit is a LED flasher circuit, which will flash a LED using only 1.5Volts. Normally to make any LED lamp work, one needs more than 2 Volts power supply as LED works only when the supply voltage is higher than the forward bias voltage. The second circuit is a Light switch that is activated by light falling on a sensor.
You can read more about these circuits on web, some links are given below:
http://freecircuitdiagram.com/2009/05/30/15v-led-flasher-using-3-transistors/
http://www.electronics-lab.com/projects/sensors/001/
Circuit analysis has to be done before coming to the laboratory.
Procedure:

1. Choose any of the circuits below and do the circuit analysis. Understand how the circuit works.

2. Assemble the circuit and see if it works. Record your observations.

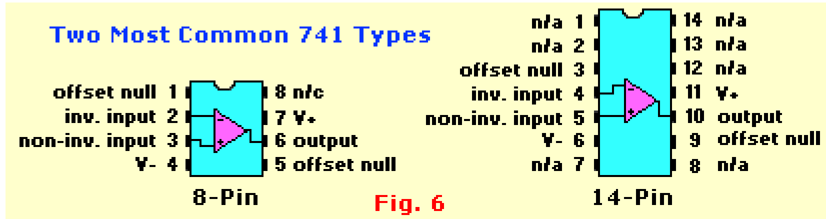
Circuit diagram:



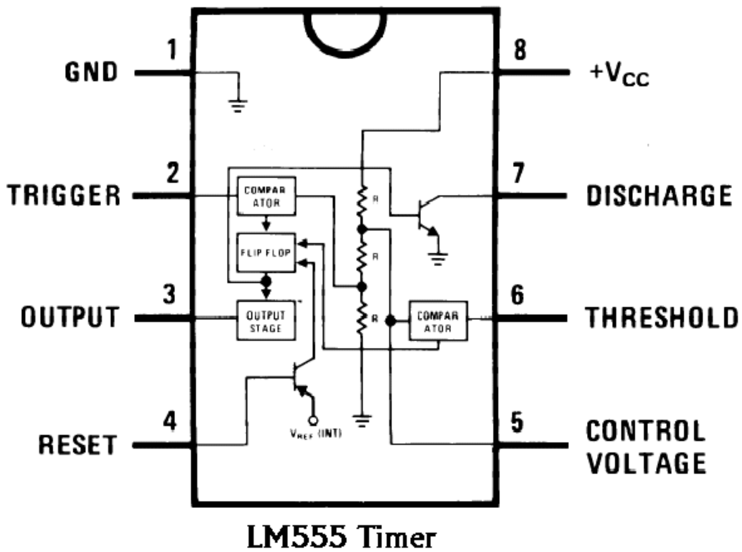


Light switch

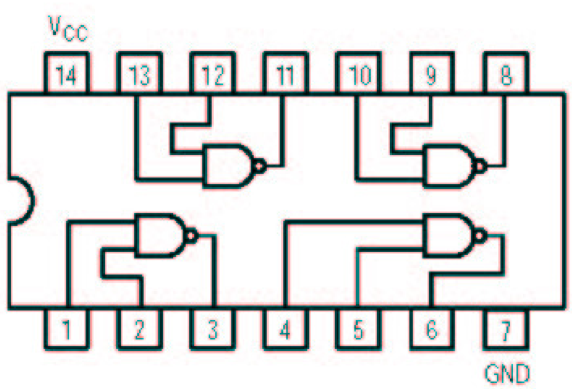
Appendix I:



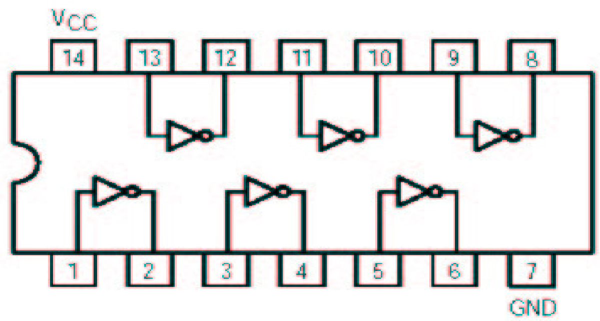
Pin diagram of IC 741



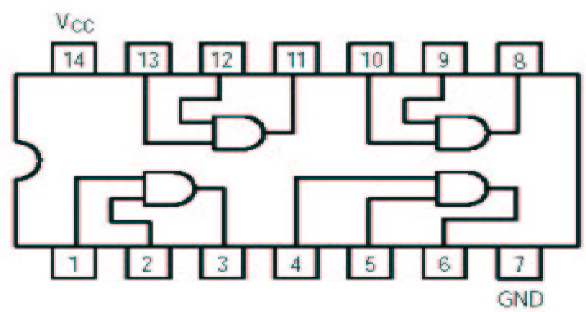
Pin diagram of IC 555



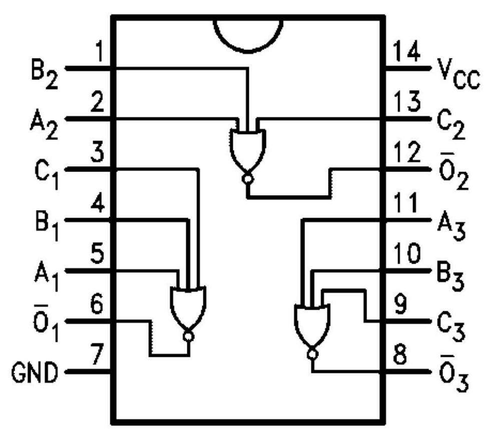
Pin diagram of IC 7400



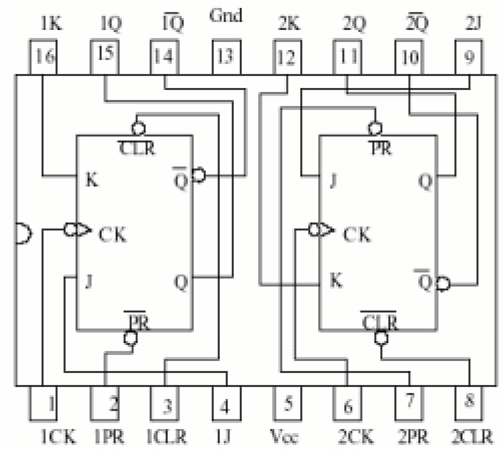
Pin diagram of IC 7404



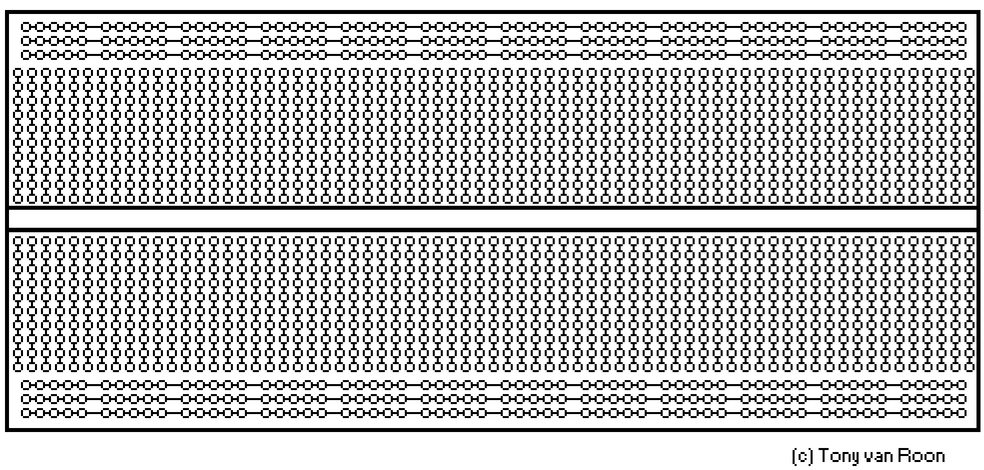
Pin diagram of IC 7408



Pin diagram of IC 7427



Pin diagram of IC 7476



Bread board layout