INSTITUTE OF AERONAUTICAL ENGINEERING
(Autonomous)
Dundigal, Hyderabad - 500 043

Lab Manual:

IC APPLICATIONS LABORATORY (AECC15)

Prepared by

S Swathi (IARE10689)

Electronics and Communication Engineering
Institute of Aeronautical Engineering

March 31, 2022

Contents

INTRODUCTION

1.1 Introduction

The main aim of this lab is to teach the linear and non-linear applications of operational amplifiers (741). Students are made familiar with theory and applications of 555 timers and design of combinational logic circuits using digital ICs. Integrated Circuits design can be divided into the broad categories of digital and analog IC design. The physical world is inherently analog indicating that there is always need for analog circuitry. Today the growth of any industry is dependent upon electronics to a great extent. Integrated circuit is electronics and this course IC application acquaints the students with general analog principles and design methodologies using practical devices and applications. It focuses on process of learning about signal condition, signal generation, instrumentation, timing and control using various IC circuitry. With modern digitization advantages we need to work with digital data and hence digital ICs play a crucial role in connecting physical world to the more sophisticated digital world. This course focuses on analysis, design and applications of modern digital integrated circuits.

1.1.1 Student Responsibilities

The student is expected tocome prepared for each lab.Lab preparation includes understanding the labexperiment from the lab manual and reading the related textbook material.
Students have to write the allotted experiment for that particular week in the work sheets given and carry them to the Lab. In case of any questions or problems with the preparation, students can contact the Faculty Teaching the Lab course, but in a timely manner.
Students have to be in formal dress code, wear shoes and lab coat for the Laboratory Class.
After the demonstration of experiment by the faculty, student has to perform the experiment individually. They have to note down the observations in the observation Tables drawn in work sheets, do the calculations and analyze the results.
Active participation by each student in lab activities is expected. The student is expected to ask the Faculty any questions they may have related to the experiment.
The student should remain alert and use commonsense while performing the lab experiment.They are also responsible for keeping a professional and accurate record of the labexperiments in the files provided.

1.1.2 Responsibilities of Faculty Teaching the Lab Course

The Faculty shall be completely familiar with each labprior to the laboratory. He/She shall provide the students with details regarding the syllabus and safety review during the first week.Lab experiments should be checked in advance to make sure that everything is in working order.The Faculty should demonstrate and explain the experiment and answer any questions posed by the students.Faculty have to supervise the students while they perform the lab experiments. The Faculty is expected to evaluate the lab worksheets and grade them based on their practical skills and understanding of the experiment by taking Viva Voce. Evaluation of work sheets has to be done in a fair and timely manner to enable the students, for uploading them online through their CMS login within the stipulated time.

1.1.3 Laboratory In-charge Responsibilities

The Laboratory In-charge should ensure that the laboratory is properly equipped, i.e., the Faculty teaching the lab receive any equipment/components necessary to perform the experiments.He/She is responsible for ensuring that all the necessary equipment for the lab is available and in working condition. The Laboratory In-charge is responsible for resolving any problems that are identified by the teaching Faculty or the students.

1.1.4 Course Coordinator Responsibilities

The course coordinator is responsible fo rmaking any necessary corrections in Course Description and lab manual. He/She has to ensure that it is continually updated and available to the students in the CMS learning Portal.

1.2 Lab Policy and Grading

The student should understand the following policy:

ATTENDANCE: Attendance is mandatory as per the academic regulations.

LAB RECORD's: The student must:

1. Write the work sheets for the allotted experiment and keep them ready before the beginning of eachlab.
2. Keep all work in preparation of and obtained during lab.
3. Perform the experiment and record the observations in the worksheets.
4. Analyze the resultsand get the work sheets evaluated by the Faculty.
5. Upload the evaluated reports online from CMS LOGIN within the stipulated time.

Grading Policy:

The final grade of this course is awarded using the criterion detailed in the academic regulations. A large portion of the student's grade is determined in the comprehensive final exam of the Laboratory course (SEE PRACTICALS),resulting in a requirement of understanding the concepts and procedure of each lab experiment for successful completion of the lab course.

Pre-Requistes and Co-Requisties:

The lab course is to be taken during the same semester as AECC12(ICA), but receives a separate grade. If AECC12(ICA), is dropped then AECC15(ICA LAB) must be dropped as well. Students are required to have completed both AECC12 and AECC15 with better grade in each.

1.3 Course Goals and Objectives

Linear and digital IC Applications Lab enables to learn design, testing and describing of circuit performance with digital and analog integrated circuits. It focuses on applications of special ICs and apply the techniques for the design of 741 ICs, applications of 555 timers and data

converters. This course provides practical hands-on experiments to analyze characteristics of commercially available digital integrated circuits.
The experiments are designed to complement the concepts introduced in AECC12. In addition, the student should learn how to record experimental results effectively and present these results in a written report.

More explicitly, the class objectives are:

- The experiments on design of Linear and Digital Integrated circuits using operational amplifier and digital ICs.
- The design and implementation of analog circuits and gain the hands-on experience on the various building blocks of digital circuits.
- The IC based real-time applications in the fields of communication systems and home-based automation systems.

1.4 Use of Laboratory Instruments

One of the major goals of this lab is to familiarize the student with the proper equipment and techniques for making electronic measurements. Some understanding of the lab instruments is necessary to avoid personal or equipment damage. By understanding the device's purpose and following a few simple rules, costly mistakes can be avoided. Most of the instrumentation used in this laboratory is implemented through breadboards, dual trace cathode ray oscilloscope, function generators, dual power supply and trainer kits.
In general, all devices have physical limits. These limits are specified by the device manufacturer and are referred to as the device rating. The ratings are usually expressed in terms of voltage limits, current limits, or power limits. It is up to the engineer to make sure that in device operation, these ratings (limit values) are not exceeded. The following rules provide a guideline for instrument protection.
The following rules provide a guideline for instrument protection.

1.4.1 Instrument Protection Rules

- Disconnect a plug by pulling on the connector body not the cable.
- Disconnect any device from the circuit before service.
- Never bypass fuses or circuit breakers.
- Keep electrical service and breaker panels accessible at all times.
- Electrical equipment and connections should not be handled with wet hands.
- Set multirange meters to highest range before connecting to an unknown source.
- Do not move equipment around the room except under the supervision of an instructor.

1.5 Data Recording and Reports

1.5.1 The Laboratory Worksheets

Students must record their experimental values in the provided tables in this laboratory manual and reproduce them in the lab reports. Reports are integral to recording the methodology and results of an experiment. In engineering practice, the laboratory notebook serves as an invaluable reference to the technique used in the lab and is essential when trying to duplicate a result or write a report. Therefore, it is important to learn to keep accurate data. Make plots of data and sketches when these are appropriate in the recording and analysis of observations. Note that the data collected will be an accurate and permanent record of the data obtained during the experiment and the analysis of the results. You will need this record when you are ready to prepare a lab report.

1.5.2 The Lab Report

Reports are the primary means of communicating your experiment and conclusions to other profes- sionals. In this course you will use the lab report to inform your LTA about what you did and what you have learned from the experiment. Engineering results are meaningless unless they can be communicated to others. You will be directed by your LTA to prepare a lab report on a few selected lab experiments during the semester. Your assignment might be different from your lab partner's assignment.
Your laboratory report should be clear and concise. The lab report shall be typed on a word processor. As a guide, use the format on the next page. Use tables, diagrams, sketches and plots as necessary to show what you did, what was observed and what conclusions you can draw from this. Even though you will work with one or more lab partners, your report will be the result of your individual effort in order to provide you with practice in technical communication.

1.6 Formatting Style

- The lab report shall be typed in a word format.
- All page margins must be 1.25 inches. All content (including text, figures, tables, etc.) must fit within the margins.
- Body text should be double-spaced.
- Basic text should be in 12-point size in a commonly used text font.
- Set your main text justified (with even left/right margins).
- The first line of each paragraph should have a left indent.
- All the tables should have titles and should be numbered. Tables should be labelled numerically as Table 1, Table 2, etc.
- Table captions appear above the table. The column headings should be labeled with the units specified.
- Graphs should be presented as figures. All the figures should have titles and should be numbered. Figure captions appear below the figure. Graphs should have labeled axes and clearly show the scales and units of the axes.
- All the figures and tables must be centered on the page. All the figures and tables in your report must be referenced in your discussion. References to figures in the main body of the text are usually written in abbreviated form (e.g. 'see Fig. 1").
- Use MS-Word equation (under Insert Equation menu), Math Type, or a similar tool to type formulas.
- If you need to copy a schematic or figure from the lab manual to your report, use Copy and Paste function or take a screenshot by using Snipping Tool in MS-Windows.
- Do not place screenshots of your lab notebook in the report! Diagrams, tables, calculations, etc. must be generated using the existing tools in the word processor.

1.7 Order of Lab Report Components

Cover Page
Cover page must include lab name and number, your name, your lab partner's name and the date the lab was performed.
Objective
Clearly state the experiment objective in your own words.
Equipment Used
Indicate which equipment was used in performing the experiment.
For Each Part of the Lab
- Write the lab's part number and title in bold font.
- Firstly, describe the problem that you studied in this part, give an introduction of the theory, and explain why you did this experiment. Do not lift the text from the lab manual; use your own words.
- Secondly, describe the experimental setup and procedures. Do not follow the lab manual in listing out individual pieces of equipment and assembly instructions. That is not relevant information in a lab report! Instead, describe the circuit as a whole (preferably with diagram), and explain how it works. Your description should take the form of a narrative, and include information not present in the manual, such as descriptions of what happened during intermediate steps of the experiment.
- Thirdly, explain your findings. This is the most important part of your report, because here, you show that you understand the experiment beyond the simple level of completing it. Explain (compare expected results with those obtained). Analyze (analyze experimental error). Interpret (explain your results in terms of theoretical issues and relate to your experimental objectives). This part includes tables, graphs, and sample calculations. When showing calculations, it is usual to show the general equation, and one worked example. All the results should be presented even if there is any inconsistency with the theory. It should be possible to understand what is going on by just reading through the text paragraphs, without looking at the figures. Every figure/table must be referenced and discussed somewhere in the text.
- Finally, provide a summary of what was learned from this part of the laboratory experiment. If the results seem unexpected or unreliable, discuss them and give possible explanations.

1.8 Conclusions

The conclusion section should provide a take-home message summing up what has been learned from the experiment:
- Briefly restate the purpose of the experiment (the question it was seeking to answer)
- Identify the main findings (answer to the research question)
- Note the main limitations that are relevant to the interpretation of the results.
- Summarize what the experiment has contributed to your understanding of the problem.

1.9 Probing Further Experiments

Questions pertaining to this lab must be answered at the end of laboratory report.

LAB-1 ORIENTATION

2.1 Introduction to OP-AMPS

To study the pin configurations, specifications and functioning of different integrated circuits used in the practical applications.

2.2 Objective

To familiarize the students with the lab facilities, equipment, standard operating procedures, lab safety and the course requirements.

2.3 Equipment needed

2.3.1 Hardware Requirements:

a) IC µA 741 OP-Am
b) NE ISE 555/SE 555C
c) VCO IC 566
d) Phase Locked Loop NE/SE 565
e) IC 723 Voltage Regulator
f) Three Terminal Voltage Regulators

2.4 IC 741 op-amp

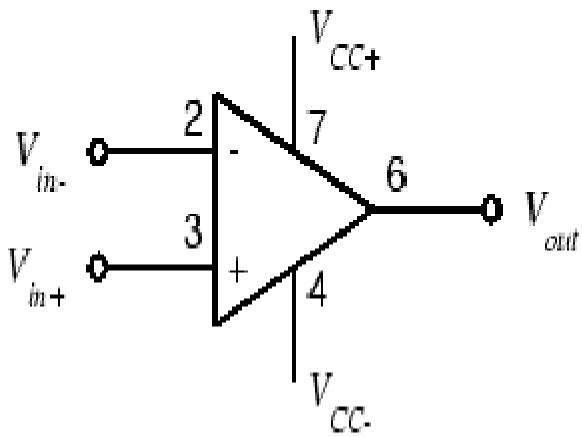

Figure 2.1: Symbol of IC 741

2.5 IC 741 op-amp pin configuration

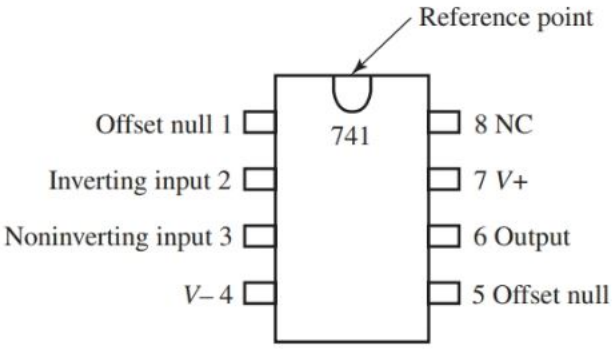

Figure 2.2: IC 741 pin configuration

The operational amplifier (op-amp) is a voltage-controlled voltage source with very high gain. It is a five terminal four port active element. The symbol of the op-amp with the associated terminals and ports is shown in Figures. The power supply voltages VCC and VEE power the operational amplifier and in general define the output voltage range of the amplifier. The terminals labeled with the "+" and the "-" signs are called non-inverting and inverting respectively. The input voltage Vp and Vn and the output voltage Vo are referenced to ground.

Specifications:

1. Supply voltage:
µA 741A, µA741, µA741E ±22v
µA 741C ±18v
2. Internal power dissipation
Dip package 310mw
Differential input voltage ±30v
3.Operating temperature range
Military (µA 741A, µA741) -550 to +1250 C.
4.Commercial (µA 741E, µA 741C) 00 C to +700 C.
5.Input offset voltage 1.0 mV.
6.Input Bias current 80 nA.
7.PSRR 30µV/V.
8.Input resistance 2MΩ.
9.CMRR 90dB.
10.Output resistance 75Ω.
11.Bandwidth 1.0 MHz.
12.Slew rate 0.5 V/ sec.

2.6 IC 555 timer

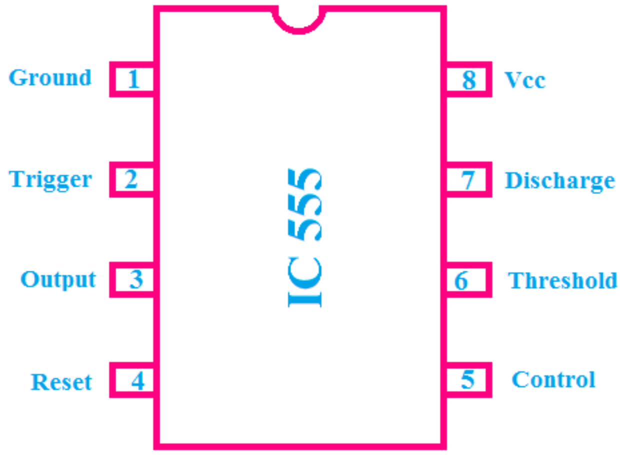

Figure 2.3: pin diagram of IC 555

One of the most versatile linear ICs is the 555 timer which was first introduced in early 1970 by Signetic Corporation giving the name as SE/NE 555 timer. This IC is a monolithic timing circuit that can produce accurate and highly stable time delays or oscillation. Like other commonly used op-amps, this IC is also very much reliable, easy to use and cheaper in cost. It has a variety of applications including monostable and astable multivibrators, dc-dc converters, digital logic probes, waveform generators, analog frequency meters and tachometers, temperature measurement and control devices, voltage regulators etc. The timer basically operates in one of the two modes either as a monostable (one-shot) multivibrator or as an astable (free-running) multivibrator. The SE 555 is designed for the operating temperature range from – 55°C to 125° while the NE 555 operates over a temperature range of 0° to 70°C.

Specifications:

1. Supply voltage 4.5V to 18V
2. Supply current 3mA
3. Output voltage (low) 0.1V
4. Output voltage (high) 12.5V 3.3V
5. Maximum operating frequency 500 kHz
6. Timing µsec to hours

2.7 IC 565 pin configuration

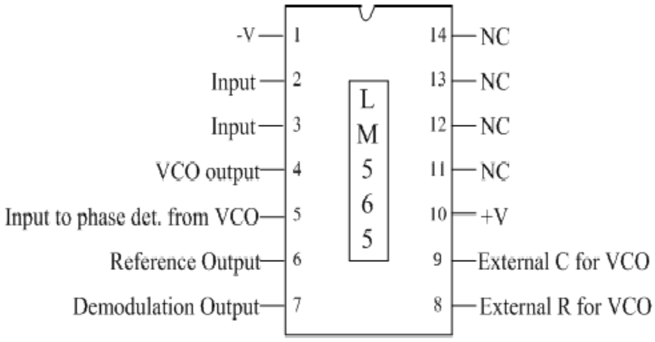

Figure 2.4: pin diagram of IC 565

The NE/SE566 Function Generator is a voltage-controlled oscillator of exceptional linearity with buffered square wave and triangle wave outputs. The frequency of oscillation is determined by an external resistor and capacitor and the voltage applied to the control terminal. The oscillator can be programmed over a ten-to-one frequency range by proper selection of an external resistance and modulated over a ten-to-one range by the control voltage, with exceptional linearity.

Specifications:
1. Operating supply voltage 12V to 24V
2. Operating supply current 12.5mA
3. Input voltage 3Vp-p
4. Operating temperature 0 to 70oC
5. Power dissipation 30mw

2.8 IC 566 pin configuration

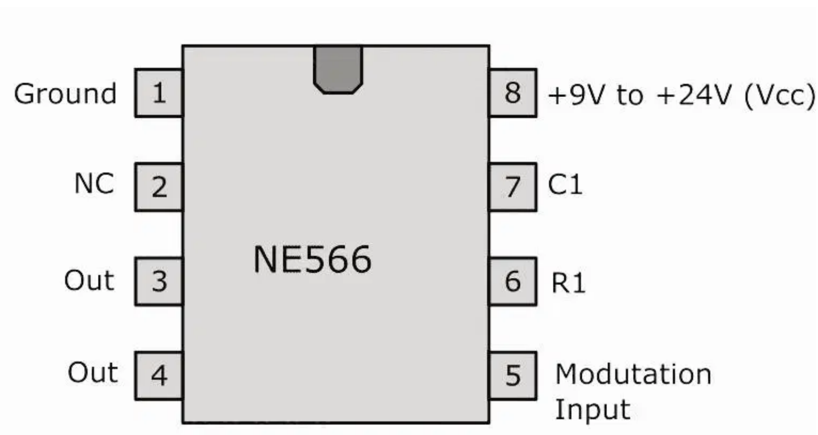

Figure 2.5: pin diagram of IC 566

Specifications:
1. Maximum supply voltage 26v
2. Input voltage 3v (p-p)
3. Power dissipation 300mw

4. Operating temperature range
NE565-00 to 700C,
SE 565-55to 1250C
5. Supply voltage 12v
6. Supply current 8mA
7. Output current sink 1mA
8. Output current source 10mA

2.9 IC 723 pin configuration

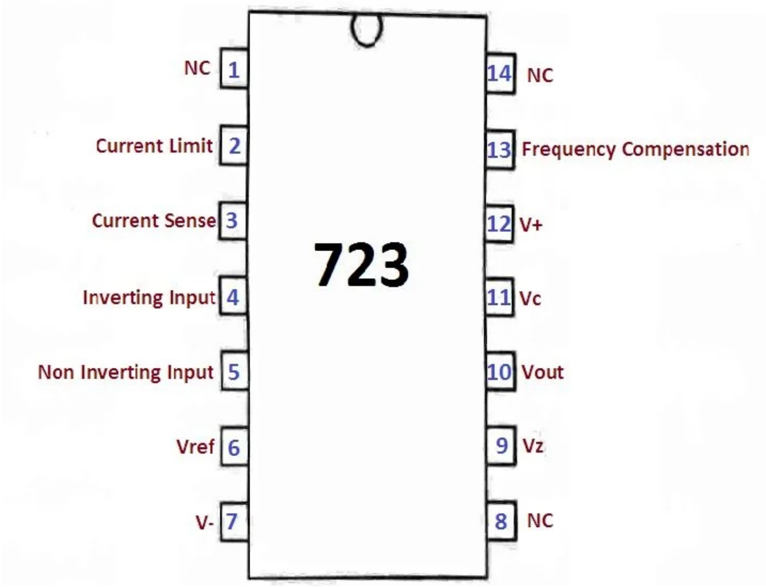

Figure 2.6: pin diagram of IC 723

The 723 voltage regulator is commonly used for series voltage regulator applications. It can be used as both positive and negative voltage regulator. It has an ability to provide up to 150 mA of current to the load, but this can be increased more than 10A by using power transistors. It also comes with comparatively low standby current drain, and provision is made for either linear or fold-back current limiting. LM723 IC can also be used as a temperature controller, current regulator or shunt regulator and it is available in both Dual-In-Line and Metal Can packages. The input voltage ranges from 9.5 to 40V and it can regulate voltage from 2V to 37V.

Specifications:
1. Input voltage 40v max
2. Output voltage 2v to 37v
3. Output current 150mA
4. Input regulation 0.025. Load regulation 0.036. Operating temperature 550C to1250C

2.10 Three terminal voltage regulators

2.10.1 78XX series

Specifications:
1. Input voltage
For 5V to 18V regulated output 35V.
Up to 24V regulated output 40V.

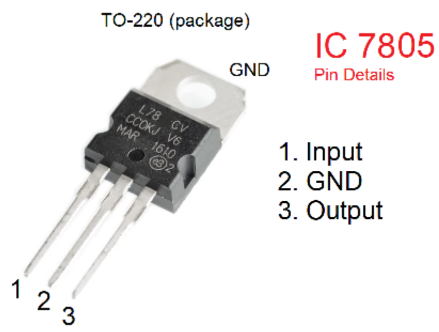

Figure 2.7: pin diagram of IC 78xx series

2. Internal power dissipation internally limited.
3. Storage temperature range -650 C to 1500 C.
4. Operating junction Temperature range
µA7800 -550 C to 1500 C.
µA7800C 00 C to 1250 C.

2.10.2 79XX series

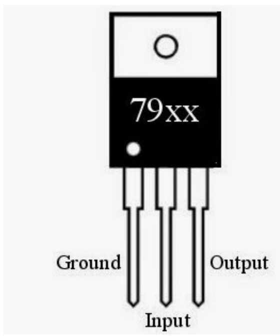

Figure 2.8: pin diagram of IC 79xx series

Specifications:
1. Input voltage
For -5v to -18v regulated output -35V
For -24v regulated output -40V
2. Internal power dissipation internally limited
3. Storage temperature range -65o C to 150o C
4. Operating junction temperature range
µA7800 -55o C to 150o C
µA7800C 0o C to 125o C

LAB-2 Inverting, Non-Inverting and Differential Amplifiers

3.1 Introduction

In an inverting amplifier circuit, the operational amplifier inverting input receives feedback from the output of the amplifier. Assume the concept of virtual short at the input terminals of op-amp, the voltage at the inverting terminal is equal to non-inverting terminal. The non-inverting amplifier is one in which the output is in phase with respect to the input. The feedback is applied at the inverting input. However, the input is now applied at the non-inverting input. The output is a non-Inverted amplified version of input. A difference amplifier is a special purpose amplifier designed to measure differential signals, otherwise known as a subtractor.

3.2 Objectives

3.2.1 Educational

- Learn to measure output voltage.
- Learn to apply Kirchoff's voltage and current laws.
- Learn to measure voltage gains of inverting and non-inverting amplifers.
- Gain experience in the construction of IC741 operational amplifiers.

3.2.2 Experimental

- Determine the output voltage.
- Measure voltage gains of inverting and non-inverting amplifers.
- Compare theoretical and practical voltage gains of inverting and non-inverting amplifers.
- Draw the input and output waveforms.

3.3 Prelab Preparation:

3.3.1 Reading

- Read and study the Background section of this Laboratory.
- Read Appendix B, Voltage Measurement of an oscilloscope with two vertical inputs.

3.3.2 Written

- Prior to coming to lab class, complete Part 0 of the Procedure.

3.4 Equipment needed

3.4.1 Hardware Requirements:

1.Bread Board
2.IC741

3.Regulated power supply(0-30v)
4.Resistors
5.Digital Multimeter
6.Probes, Connecting Wires

3.5 Circuit diagrams:

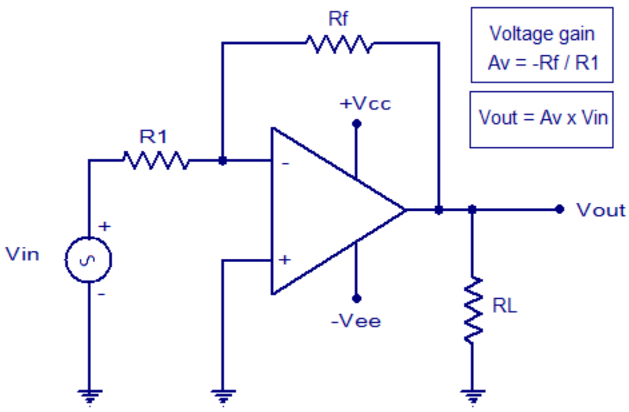

Figure 3.1: Inverting amplifier

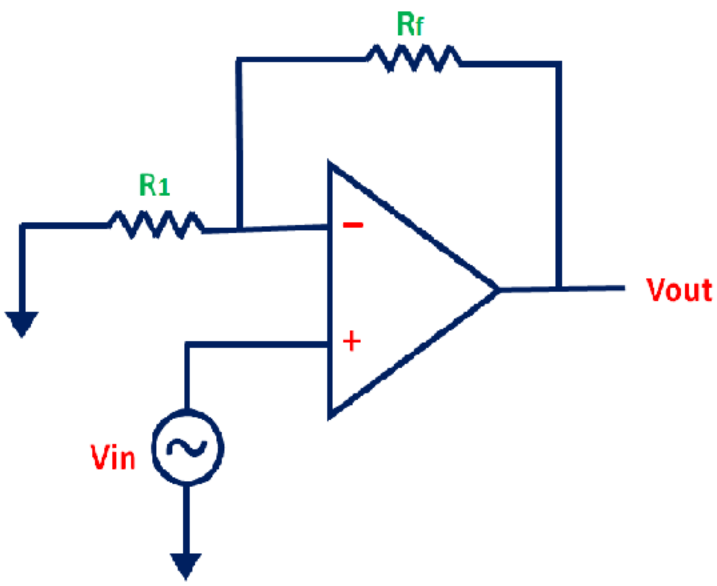

Figure 3.2: Non-Inverting amplifier

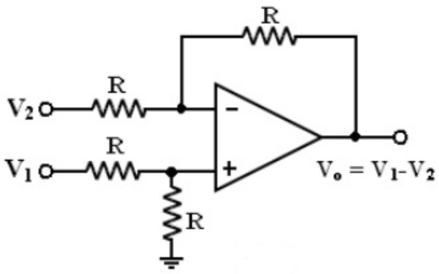

Figure 3.3: Difference amplifier

3.6 Background

Inverting and Non-Inverting Amplifier: In this configuration, the input voltage signal, (Vin) is applied directly to the non-inverting (+) input terminal which means that the output gain of the amplifier becomes "Positive" in value in contrast to the "Inverting Amplifier" circuit whose output gain is negative(-).
Differential Amplifier: A Differential Amplifier is a circuit that gives the difference of the two inputs,Vo =V2-V1, Where V1 and V2 are the inputs. By connecting one input voltage V1 to inverting terminal and another input voltage V2 to the non – inverting terminal, we get the resulting circuit as the Differential Amplifier. This is also called as Subtractor using op-amps. Output of a differential amplifier (subtractor) is given as

$$Vo = (-Rf/R1)\ (V1-V2)$$

If all external resistors are equal in value, then the gain of the amplifier is equal to -1. The output voltage of the differential amplifier with a gain of -1 is Vo = (V2-V1) Thus, the output voltage Vo is equal to the voltage V2 applied to the non – inverting terminal minus the voltage V1 applied to the inverting terminal. Hence the circuit also called a Subtractor.

3.7 Procedure

Inverting and Non inverting Amplifier:
1. Connect the components/equipment as shown in the circuit diagram.
2. Switch ON the power supply.
3. Apply dc voltages at each input terminal for V1 and V2 from the dc supply and check the output voltage Vo at the output terminal.
4. Tabulate 3 different sets of readings by repeating the above step.
5. Compare practical Vo with the theoretical output voltage Vo =V1+V2.

Differential Amplifier:
1. Connect the components/equipment as shown in the circuit diagram.
2. Switch ON the power supply.
3. Apply dc voltages at each input terminal for V1 and V2 from the dc supply and check the output voltage Vo at the output terminal.
4. Tabulate 3 different sets of readings by repeating the above step.
5. Compare practical Vo with the theoretical output voltage Vo =V2-V1.

Table 3.1: **Tabular Column For Inverting amplifier**

S No	**V1 Volts**	**Theoretical V0**	**Practical V0**

Table 3.2: **Tabular Column For Non-Inverting amplifier**

SNo	**V1 Volts**	**Theoretical Vo**	**Practical Vo**

Table 3.3: **Tabular Column For difference amplifier**

SNo	**V1 Volts**	**V2 Volts**	**Theoretical Vo**	**Practical Vo**

3.8 Result

3.9 Further Probing Experiments

1. Draw the circuit diagram and the output waveform of a Zero Crossing Detector if the input is sinusoidal?

2. Calculate the voltage gain of inverting and non-inverting amplifiers with the following: R1=5k, Rf= 20k.

LAB-3 Integrator and Differentiator

4.1 Introduction

An op-amp based differentiator produces an output, which is equal to the differential of input voltage that is applied to its inverting terminal. An op-amp based integrator produces an output, which is an integral of the input voltage applied to its inverting terminal.

4.2 Objectives

4.2.1 Educational

- Learn to measure output voltage.
- Learn to apply Kirchoff's voltage and current laws.
- Learn to measure time period and frequency of integrator and differentiator circuits.
- Gain experience in the construction of IC741 operational amplifiers.

4.2.2 Experimental

- Determine the output voltage.
- Measure time period and frequency of integrator and differentiator circuits.
- Compare theoretical and practical time period and frequency of integrator and differentiator circuits.
- Draw the input and output waveforms for different time constants.

4.3 Prelab Preparation

4.3.1 Reading

- Read and study the Background section of this Laboratory.
- Read Appendix B, time and frequency Measurement of an oscilloscope.

4.3.2 Written

- Prior to coming to lab class, complete Part 0 of the Procedure.

4.4 Equipment needed

4.4.1 Hardware Requirements

1. Bread Board / CDS Board.
2. Function Generator (1MHz).
3. Cathode Ray Oscilloscope (20MHz/30 MHz)
4. Regulated Power Supply (Dual Channel).

5. Connecting Wires.

4.5 circuit diagrams

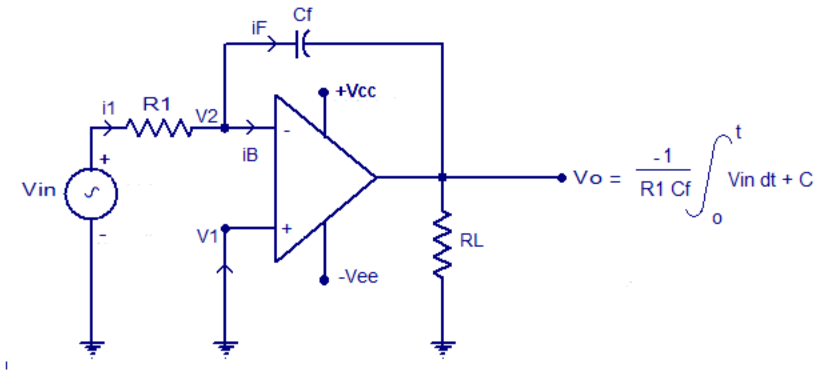

Figure 4.1: Integrator

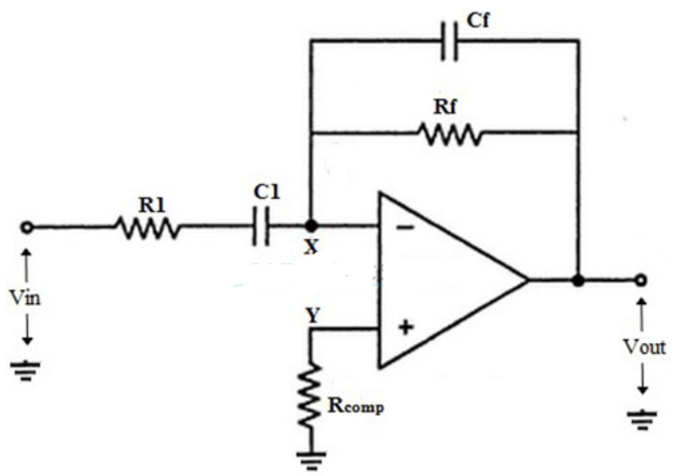

Figure 4.2: Differentiator

4.6 Background

Integrator

A circuit in which the output voltage waveform is the integration of the input is called integrator.

V0=-1/R4C1

The above equation indicates that the output voltage is directly proportional to the negative integral of the input voltage and inversely proportional to the time constant R1CF. For Example, if the input is a sine wave, the output will be a cosine wave or if the input is a square wave, the output will be a triangular wave.

1. When the input signal frequency is ZERO, the integrator works as an open – loop amplifier. This is because of the capacitor CF acts as an open circuit (XCF =1/ CF = infinite for f=0).

2. Therefore, the ideal integrator becomes unstable and suffers with low frequency noise. To overcome this problem RF is connected across the feedback capacitor CF. Thus, RF limits the

low-frequency gain and hence minimizes the variations in the output voltage.
3. Frequency fb at which the gain of the integrator is 0 dB, is given by

fb =1/2 R1CF

4. Both the stability and the low – frequency roll-off problems can be corrected by the addition of a resistors RF in the feedback path.
NOTE: The input signal will be integrated properly if the time period T of the input signal is greater than or equal to RFCF.
Differentiator The differentiator circuit performs the mathematical operation of differentiation. That is the output waveform is the derivative of the input waveform. Therefore

VO = Rf C1 dVin / dt

1. The above equation indicates that the output voltage is directly proportional to the derivative of the input voltage and also proportional to the time constant RFC1.
2. For Example, if the input is a sine wave, the output will be a cosine wave or if the input is a square wave, the output will be spikes.
3. The reactance of the circuit increases with increase in frequency at a rate of 20dB/ decade. This makes the circuit unstable. In other words, the gain of an ideal differentiator circuit is direct dependent on input signal frequency. Therefore, at high frequencies ($f=\infty$), the gain of the circuit becomes infinite making the system unstable.
4. The input impedance XC1 decreases with increase in frequency, which makes the circuit very susceptible to high frequency noise.
5. The frequency response of the basic differentiator is shown in fig 3.4 In this fig fa is the frequency at which the gain is 0 dB.

fa =1/2 RFC1

6. Both the stability and the high – frequency noise problem can be corrected by the addition of two components R1 and CF as shown in fig . The frequency response of which is shown in fig. From f to fa the gain decreases at 40dB/decade. This 40 dB/decade change in gain is caused by the R1C1 and RFCF combinations.
NOTE: The input signal will be differentiated properly if the time period T of the input signal is greater than or equal to RF C1.

4.7 Procedure

Integrator
1. Connect the circuit as shown in fig 3.1 on the breadboard.
2. Switch „ON the power supply and apply + 15V to pin no.7 and -15V to pin no.4 of the IC741.
3. Apply a sine wave input signal of 2V peak-to-peak amplitude at 1 kHz frequency from the function generator (at pin no.2 of the IC741).
4. Connect the C.R.O at (pin no.6) the output terminals.
5. Observe and plot the input and output voltage waveforms.
6. Measure the output voltage (Vo) from the experimental results.
Differentiator
1. Connect the circuit as shown in fig 3.2 on the breadboard.
2. Switch „ON the power supply and apply + 15V to pin no.7 and -15V to pin no.4 of the IC741.
3. Apply a sine wave input signal of 2V peak-to-peak amplitude at 1 kHz frequency from the function generator (at pin no.2 of the IC741).
4. Connect the C.R.O at (pin no.6) the output terminals.
5. Observe and plot the input and output voltage waveforms.

6. Measure the output voltage (Vo) from the experimental results.

4.8 Model waveforms

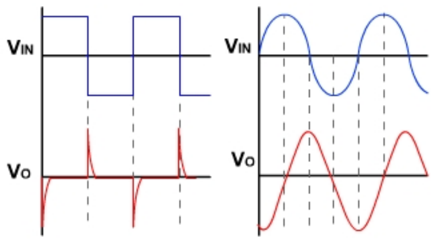

Figure 4.3: Differentiator waveforms

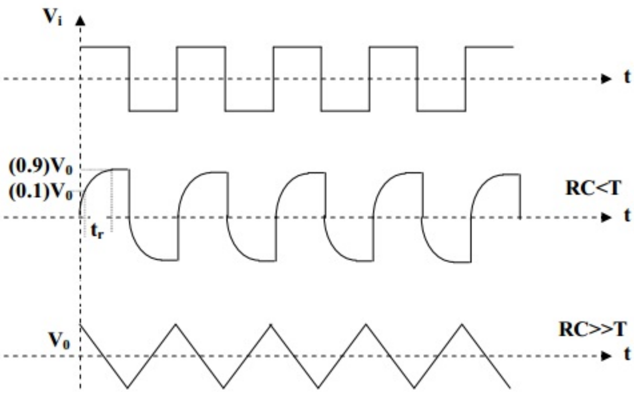

Figure 4.4: Integrator waveforms

4.8.1 Obsevations for integrator:

Input signal
Amplitude = Time period =
Output signal
Amplitude = Time period =

4.8.2 Obsevations for differentiator:

Input signal
Amplitude = Time period =
Output signal
Amplitude = Time period =

4.9 Applications

4.9.1 Applications of Integrator:

1. In the analog computers
2. In solving differential equations
3. In analog to digital converters
4. In various signal wave shaping circuits
5. In ramp generators

4.9.2 Applications of differentiator:

1. In wave shaping circuits
2. as a rate of change detector in FM demodulators

4.10 Result

The Integrator and Differentiator circuits were constructed using IC 741 and verified their response for sine and square wave inputs.

4.11 Further Probing Experiments

1. A low frequency differentiator is desired for a particular application to Perform the operation Vo (t) =-0.001 dvi(t)/dt . Determine the suitable design of differentiator circuit for the periodic signal with a frequency of 1 kHz.

2. Determine the suitable design of integrator circuit for the periodic signal with a frequency of 5 kHz.

LAB-4 Frequency Response of Lowpass and Highpass Active Filters

5.1 Introduction

The second order low pass filter circuit has two RC networks, R1 – C1 and R2 – C2 which give the filter its frequency response properties. The filter design is based around a non-inverting op-amp configuration so the filters gain, A will always be greater than 1. The basic operation of an active high pass filter is exactly the same as that for its equivalent RC passive high pass filter circuit except this time the circuit has an operational amplifier included within its filter design providing amplification and gain control.

5.2 Objectives

5.2.1 Educational

- Learn to measure output voltage.
- Learn to measure output voltage for different frequencies of low-pass and high-pass filters.
- Gain experience in the construction of IC741 operational amplifiers.

5.2.2 Experimental

- Determine the output voltage.
- Measure output voltage for different frequencies of low-pass and high-pass filters.
- Compare theoretical and practical cutoff frequencies of low-pass and high-pass filters.
- Draw the frequency response of filters with semi-log graphs.

5.3 Prelab Preparation:

5.3.1 Reading

- Read and study the Background section of this Laboratory.
- Read Appendix B, output voltage and frequency Measurement of an oscilloscope.

5.3.2 Written

- Prior to coming to lab class, complete Part 0 of the Procedure.

5.4 Equipment needed

5.4.1 Hardware Requirements:

1. Bread Board / CDC Board.
2. Function Generator (1MHz).

3. Cathode Ray Oscilloscope (20MHz/30 MHz)
4. Regulated Power Supply (Dual Channel).
5. Connecting Wires.
6. IC 741

5.5 Circuit diagrams

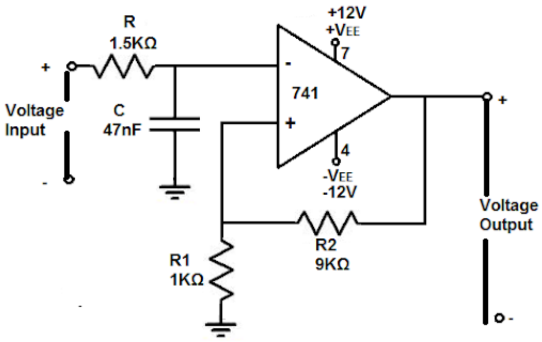

Figure 5.1: Lowpass filter

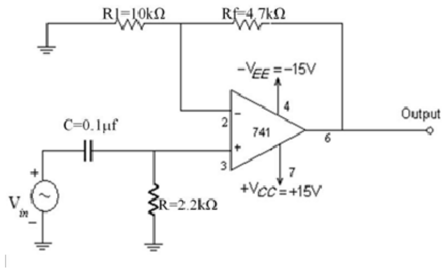

Figure 5.2: Highpass filter

5.6 Background

A first order filter consists of a single RC network connected to the non-inverting input terminal of the op-Amp as shown in the figure. Resistors R1 and Rf determine the gain of the filter in the pass band. Components R and C determine the cutoff frequency of the filter. Low-Pass filter: The circuit of 1st order low-pas filter is shown in fig.1 and its frequency response is as shown in the fig3. The dashed curve in the fig 4.3 indicates the ideal response and solid curve indicates practical filter response. It is not possible to achieve ideal characteristics. However, with special design techniques (Higher order filters) it is possible to closely approximate the ideal response. Active filters are typically specified by the voltage transfer function, H(s) = V0 (s)/ Vi(s)………………. (1) (under steady state conditions) High Pass Filter: The circuit of 1st order high pass filter is shown in fig.2 and its frequency response is as shown in the fig4.4 the dashed curve in the fig 4.4 indicates the ideal response and solid curve indicates practical filter response. When an input signal is applied to High pass filter, the signals at high frequencies are passed through circuit and signals at low frequencies are rejected. That is the signal which are having frequencies less than the lower cutoff frequency fL are rejected and the signal with frequency greater the lower cut off frequency fL are passed through the circuit. That is

1. For f $> fL, Vo(s)/Vi(s) = Maximumandiscalledaspassband.$

2.$Forf < fL, Vo(s)/vi(s) = 0andiscalledasthestopband$

5.7 Procedure

1. Connections are made as per the circuit diagram.
2. Apply sine wave of amplitude 4Vp-p to the non-inverting input terminal.
3. Values the input signal frequency.
4. Note down the corresponding output voltage.
5. Calculate gain in db.
6. Tabulate the values.
7. Plot a graph between frequency and gain.
8. Identify stop band and pass band from the graph.

5.8 Observation table

Table 5.1: **Tabular Column For Lowpass filter**

SNo	**Frequency(Hz)**	**Vo(V)**	**Gain= (V0/ Vi)**	**Gain in dB=20 log(V0/Vi)**

Table 5.2: **Tabular Column For Highpass filter**

SNo	Frequency(Hz)	Vo(V)	Gain= (V0/ Vi)	Gain in dB=20 log(V0/Vi)

5.9 Model waveforms

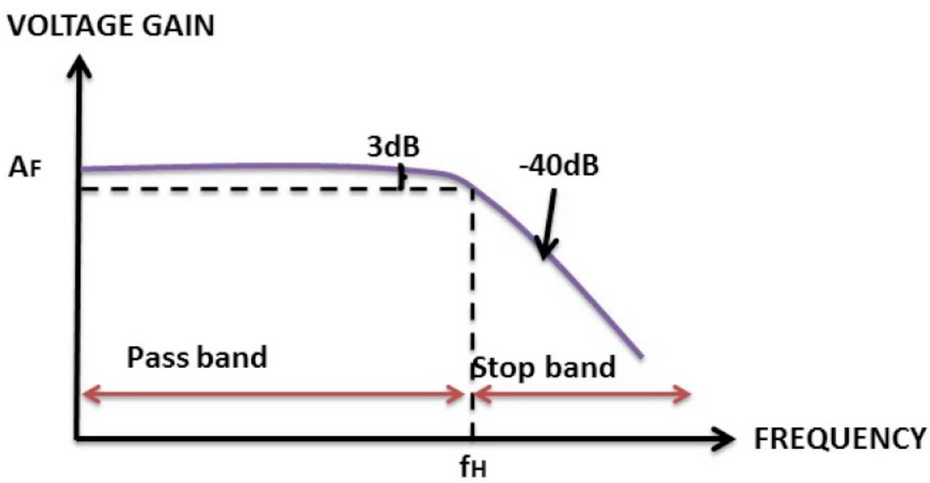

Figure 5.3: Frequency response of lowpass filter

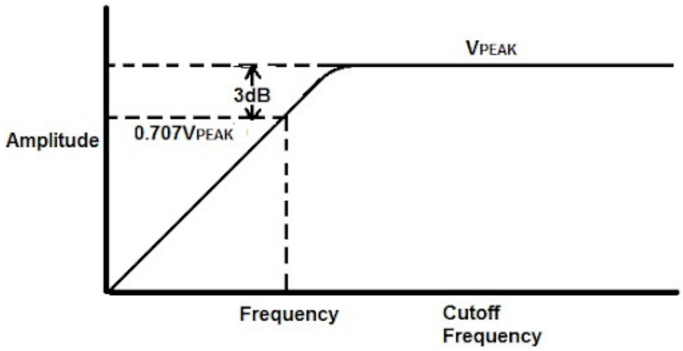

Figure 5.4: Frequency response of highpass filter

5.10 Applications

1. In communications systems, use filters to suppress noise, to isolate a single communication from many channels, to prevent spillover of adjacent bands, and to recover the original message signal from modulated signals.
2. In instrumentation systems, engineers use filters to select desired frequency components or eliminate undesired ones. In addition, we can use these filters to limit the bandwidth of analog signals before converting them to digital signals. You also need these filters to convert the digital signals back to analog representations.
3. In audio systems, engineers use filters in crossover networks to send different frequencies to different speakers. In the music industry, record and playback applications require fine control of frequency components.
4. In biomedical systems, filters are used to interface physiological sensors with data logging and diagnostic equipment.

5.11 Result

The first order LPF and HPF are designed for a chosen cutoff frequency and the frequency response curves were plotted between voltage gain (dB) and frequency (Hz).

5.12 Further Probing Experiments

Q1. Design a lowpass filter with the cutoff frequency of 1kHz and draw its frequency response.

Q2. Design a first order high-pass filter with the cutoff frequency of 2kHz and draw its frequency response.

LAB-5 Frequency Response of Bandpass and Band Reject Active Filters

6.1 Introduction

The Active Band Pass Filter is a frequency selective filter circuit used in electronic systems to separate a signal at one particular frequency, or a range of signals that lie within a certain “band” of frequencies from signals at all other frequencies. The Band Stop Filter is another type of frequency selective circuit that functions in exactly the opposite way to the Band Pass Filter we looked at before. The band stop filter, also known as a band reject filter, passes all frequencies with the exception of those within a specified stop band which are greatly attenuated.

6.2 Objectives

6.2.1 Educational

- Learn to measure output voltage.
- Learn to measure output voltage for different frequencies of band-pass and band-reject filters.
- Gain experience in the construction of IC741 operational amplifiers.

6.2.2 Experimental

- Determine the output voltage.
- Measure output voltage for different frequencies of band-pass and band-reject filters.
- Compare theoretical and practical cutoff frequencies of band-pass and band-reject filters.
- Draw the frequency response of filters with semi-log graphs.

6.3 Prelab Preparation

6.3.1 Reading

- Read and study the Background section of this Laboratory.
- Read Appendix B, output voltage and frequency Measurement of an oscilloscope.

6.3.2 Written

- Prior to coming to lab class, complete Part 0 of the Procedure.

6.4 Equipment needed

6.4.1 Hardware Requirements:

1. Bread Board / CDS Board.
2. Function Generator

3. Cathode Ray Oscilloscope
4. Regulated Power Supply (Dual Channel)
5. Connecting Wires.
6. IC 741

6.5 Circuit diagrams

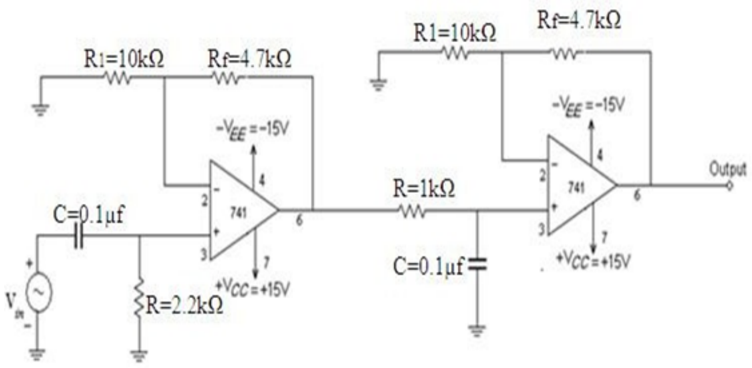

Figure 6.1: Bandpass filter

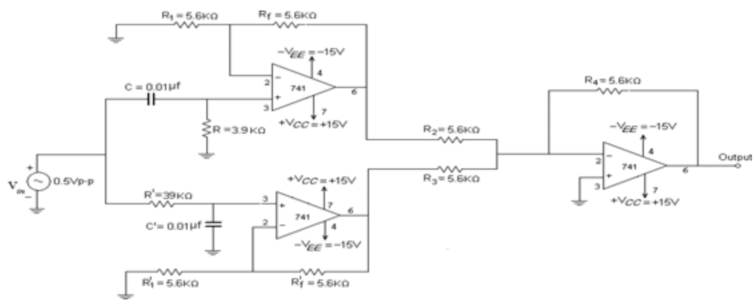

Figure 6.2: Band reject filter

6.6 Back ground

BAND PASS FILTER:

A Band Pass Filter (BPF) has a pass band between the lower cut-off frequency, fL and the higher cut-off frequency fH, such that fH greater than fL. When the input frequency is zero, the gain of the filter will be zero. As the input signal frequency increases from zero to fL, the gain will increase at a rate 20dB/decade up to 3dB less than its maximum value. If the input signal frequency increases beyond fL, the gain will reach its maximum value and remains constant up to high frequencies as shown in the Fig 5.3. When the input signal frequency reaches the higher cut-off frequency, fH, the gain will fall 3dB less from its maximum value. If the input signal frequency increases beyond fH, the gain will decreases to zero at rate of 20dB/decade. After reaching the total pass band region, the gain of the filter is constant up to its designed fH (high cut off frequency). There is a phase shift between input and output voltages of BPF as a function of frequency in its Pass Band region. This filter passes all frequencies equally well i.e. the output and input voltages are equal in amplitude for all frequencies. This highest frequency up to which the input and output amplitudes remain equal is dependent of the unity gain bandwidth of Op – Amp. At this frequency, the phase shift between input and output

becomes maximum.
BAND REJECT FILTER:
A Band Reject Filter (BRF) has a stop band between the cutoff frequencies fH fL such that fH less than fL. When the input signal frequency is zero, the gain of the BPF will be maximum and will remains constant as the input signal frequency increases. At the higher cut off frequency fH, the gain becomes 3dB less than its maximum value. As the input signal frequency increases beyond fH, the gain of the filter decreases becomes zero at the central (fC) or operating frequency (fO). After this center frequency fC, the gain increases to 3dB less than its maximum value at the lower cut-off frequency, fL. As the input signal frequency increases beyond fL the gain increases to the maximum value and becomes constant. There is a phase shift between input and output voltages of BPF in its "Pass band region". This filter passes all the frequencies equally well i.e. output and input voltages are equal in (magnitude) amplitude for all frequencies. This highest frequency up to which the input and output amplitude remains equal is dependent on the unity gain bandwidth of the Op- Amp. However, at this frequency, the phase shift between the input and output is maximum.

6.7 Procedure

1. Make the circuit connection as shown in figure.
2. Connect the signal generator to input terminals. And connect the C.R.O at output terminals of the trainer switch on the trainer.
3. Apply the input signal frequency from 100Hz to 10 KHz.
4. Record the input frequency, Input voltage and Output voltage. Find the gain of the B.P.F using the formula. The gain magnitude in dB is equal to 20 Log (Vo/Vi).

6.8 Observation Tables

Table 6.1: **Tabular Column For Bandpass filter**

SNo	Frequency(Hz)	Vo(V)	Gain= (V0/ Vi)	Gain in dB=20 log(V0/Vi)

Expected Waveforms

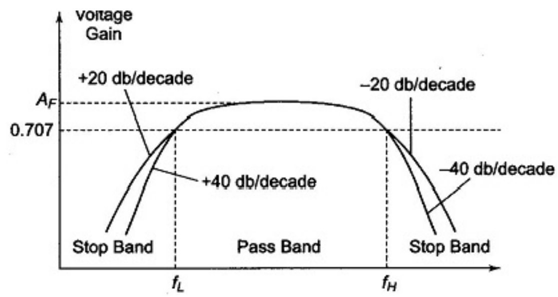

Figure 6.3: Frequency response of Bandpass filter

Band reject filter response

Table 6.2: **Tabular Column For Band reject filter**

SNo	Frequency(Hz)	Vo(V)	Gain= (V0/ Vi)	Gain in dB=20 log(V0/Vi)

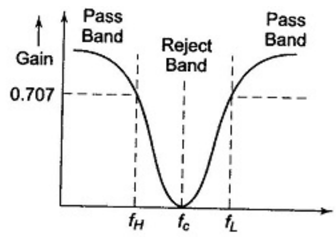

Figure 6.4: Frequency response of Band reject filter

6.9 Result

The band pass and band reject filters have been designed for chosen fL, fH and frequency responses were plotted between voltage gain (in dB) and input frequency.

6.10 Further Probing Experiments

Q1. Design a band pass filter with an upper cutoff frequency of 3kHz and a lower cut off frequency of 1kHz.

Q2. Design a band reject filter with a frequency of 50 Hz.

LAB-6 Monostable Multivibrator Using 555 Timer

7.1 Introduction

A monostable multivibrator (MMV) often called a one-shot multivibrator, is a pulse generator circuit in which the duration of the pulse is determined by the R-C network, connected externally to the 555 timers. In such a vibrator, one state of output is stable while the other is quasi-stable (unstable).

7.2 Objectives

7.2.1 Educational

- Learn to measure output voltage and voltage across the capacitor.
- Learn to measure frequency of oscillations of monostable multivibrator.
- Learn to measure pulse width of monostable multivibrator.
- Gain experience in the construction of IC555 timers.

7.2.2 Experimental

- Determine the output voltage and voltage across the capacitor.
- Measure frequency of oscillations of monostable multivibrator.
- Compare theoretical and practical values of pulse width of monostable multivibrator.
- Draw the output voltage and voltage across the capacitor waveforms.

7.3 Prelab Preparation:

7.3.1 Reading

- Read and study the Background section of this Laboratory.
- Read Appendix B, output voltage and frequency Measurement of an oscilloscope.

7.3.2 Written

- Prior to coming to lab class, complete Part 0 of the Procedure.

7.4 Equipment needed

7.4.1 Hardware Requirements:

1. C.R.O
2. Regulated DC power Supply
3. Function generator
4. CDS Board/ Bread Board.
5. Connecting patch chords.

7.5 Circuit diagram

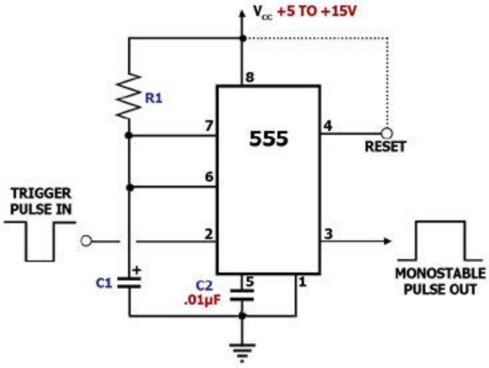

Figure 7.1: Monostable Multivibrator

7.6 Back ground

The 555 Timer is used in number of applications; it can be used as monostable, astable multivibrators, DC to DC converters, digital logic probes, analog frequency meters, voltage regulators and time delay circuits. The IC 555 timer is 8-pin IC and it can operate in free- running (Astable) mode or in one- shot (Monostable) mode. It can produce accurate and highly stable time delays or oscillations. Monostable can also be called as One-shot Multivibrator. When the output is low, the circuit is in stable state, Transistor Q1 is ON and capacitor C is shorted out to ground. However, upon application of a negative trigger pulse to pin-2, transistor Q1 is turned OFF, which releases short circuit across the external capacitor and drives the output High. The capacitor C now starts charging up toward Vcc through R. However, when the voltage across the external capacitor equals 2/3 Vcc, the output of comparator1 switches from low to high, which in turn drives the output to its low state. The output, Q of the flip flop turns transistor Q1 ON, and hence, capacitor C rapidly discharges through the transistor. The output of the Monostable remains low until a trigger pulse is again applied. Then the cycle repeats. Fig shows the trigger circuit and Fig shows trigger input, output voltage and capacitor voltage waveforms. Pulse width of the trigger input must be smaller than the expected pulse width of the output waveforms. Trigger pulse must be a negative going input signal with amplitude larger than 1/3 Vcc. The time during which the output remains high is given by tp =1.1RC Once triggered, the circuit's output will remain in the high state until the set time tp elapses. The output will not change its state even if an input trigger is applied again during this time interval tp.
DESIGN: Step 1: Choose C=1µF.
Step 2: Since in monostable multivibrator, tp=1.1RC. Therefore R= tp / 1.1C
Step 3: Using equations, design the value of R.

7.7 Procedure

1. Connect the 555 timer in Monostable mode as shown in fig.
2. Connect the C.R.O at the output terminals and observe the output.
3. Apply external trigger at the trigger input terminal (PIN 2) and observe the output of Monostable Multivibrator.
4. Record the trigger input, voltage across the capacitor and output waveforms and measure the output pulse width.
5. Verify results with the sample output waveforms as shown in fig.

6. Calculate the time period of pulse (tp =1.1RC) theoretically and compare it with practical values.

7.8 Observation Table

Table 7.1: **Tabular Column For Monostable multivibrator**

S No	**Theoretical output pulse width(msec)**	**Practical output pulse width(msec)**

7.9 Model waveforms

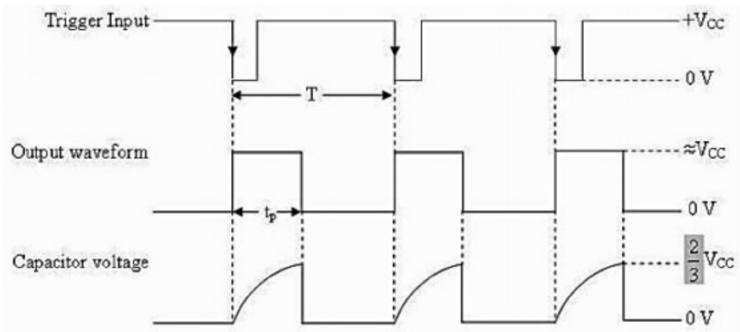

Figure 7.2: Monostable Multivibrator

Observations:
Trigger input
Amplitude = Time period =
Square wave Output signal
Amplitude = Time period =
Triangle wave Output signal
Amplitude = Time period =

7.10 Applications

1. Frequency divider
2. Pulse width modulation
3. Linear ramp generator
4. Missing pulse detector

7.11 Result

Hence designed and studied 555 timer as a Monostable multivibrator and also theoretical and Practical of time period values of the output waveform are compared.

7.12 Further Probing Experiments

Q1. Consider the Monostable multivibrator with R=3KΩ and C=0.0068μF. Determine the pulse width.

Q2. Design a Monostable Multivibrator using 555 timers to get 10msec.

LAB-7 Astable Multivibrator Using IC 555 Timer

8.1 Introduction

Astable multivibrator is also called as Free Running Multivibrator. It has no stable states and continuously switches between the two states without application of any external trigger. The IC 555 can be made to work as an astable multivibrator with the addition of three external components: two resistors (R1 and R2) and a capacitor (C).

8.2 Objectives

8.2.1 Educational

- Learn to measure output voltage and voltage across the capacitor.
- Learn to measure frequency of oscillations of astable multivibrator.
- Learn to measure pulse width of astable multivibrator.
- Gain experience in the construction of IC555 timers.

8.2.2 Experimental

- Determine the output voltage and voltage across the capacitor.
- Measure frequency of oscillations of astable multivibrator.
- Compare theoretical and practical values of pulse width of astable multivibrator.
- Draw the output voltage and voltage across the capacitor waveforms.

8.3 Prelab Preparation:

8.3.1 Reading

- Read and study the Background section of this Laboratory.
- Read Appendix B, output voltage and frequency Measurement of an oscilloscope.

8.3.2 Written

- Prior to coming to lab class, complete Part 0 of the Procedure.

8.4 Equipment needed

8.4.1 Hardware Requirements:

1. C.R.O
2. Function generator
3. Regulated DC power Supply
4. CDS Board/ Bread Board

5. Connecting patch chords
6. IC 555 timer

8.5 Circuit diagram

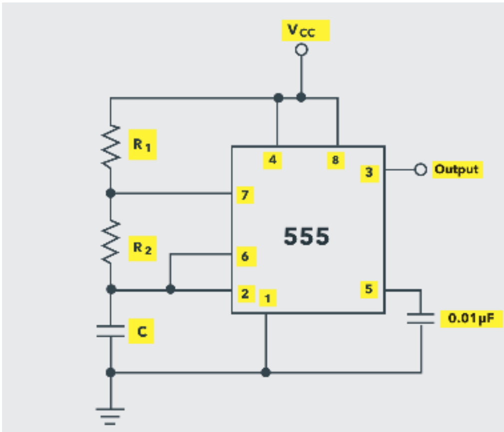

Figure 8.1: Astable Multivibrator

8.6 Background

The 555 Timer is used in number of applications; it can be used as monostable, astable multivibrators, DC to DC converters, digital logic probes, analogy frequency meters, voltage regulators and time delay circuits. The IC 555 timer is 8-pin IC and it can operate in free-running (Astable) mode or in one-shot (Monostable) mode. The pin configuration of NE 555 Timer is as shown figure. It can produce accurate and highly stable time delays or oscillations. Astable Multivibrator often called a free-running Multivibrator. External Trigger input is not required to operate the 555 as an Astable Configuration. However, the time during which the output is either high or low is determined by two external components Resistor and Capacitor. Fig shows the 555 as Astable Multivibrator. Initially, when the output is high, capacitor C starts charging towards Vcc through resistor Ra and Rb. As soon as voltage across the capacitor equals to 2/3 Vcc, comparator-1 triggers the flip-flop, and the output is low. Now capacitor discharges through Rb and transistor Q1. When the voltage across capacitor C equals to 1/3Vcc, comparator- 2 s output triggers the flip-flop, and the output goes high. Then the cycle repeats. The output voltage waveforms are as shown in fig (3). In this way capacitor periodically charges and discharges between 2/3Vcc and 1/3Vcc respectively.
The time during which the capacitor charges from 1/3Vcc to 2/3 Vcc is equal to the ON time of the timer (i.e. the output is HIGH) and is given by tc =0.69(R1+R2)C
The time during which the capacitor discharges from 2/3 Vcc to 1/3Vcc is equal to the OFF time of the timer, during which the output is LOW and is given by td =0.69(R2)C
The total time period of the output is the sum of charging time(tc)and discharging time(td) and is given by T = tc + td = 0.69(R1 + 2R2) C
Therefore, the frequency of oscillations of Astable multivibrator is given by F = 1/T = 1.45/ (R1 + 2R2) C
DUTY CYCLE:
This term is in conjunction with Astable Multivibrator. The duty cycle is the ratio of the ON time, tc during which the output is high to the total time period T. It is generally expressed as a percentage.

Duty cycle, D = (TON /TON+ TOFF) = tc /T = (R1 + R2) / (R1 + 2R2)
DESIGN:
Step1: Choose C=0.01 µF
Step2: using the formula, F = 1.45/ (R1 + 2R2) C, Get a relation between R1 and R2.
Step3: Consider the expression for duty cycle,
D= (TON /TON+ TOFF) = (R1 + R2) / (R1 +2R2) and obtain a relation between R1 and R2.
Step4: Using the relations between R1 and R2., obtained in step2 and step3, solve for R1 and R2.

8.7 PROCEDURE

1. Connect the IC 555 timer in Astable mode as shown in fig.
2. Connect the C.R.O at the output terminal (pin 3) and observe the output.
3. Record the waveforms at pin3, across the capacitor and compare them with the sample output waveforms as shown in figure.
4. Measure the charging time (tc), discharging time (td) and total time period/ Frequency from the output waveform.
5. Calculate tc, td, time period (T), frequency (f) of the square wave output and percentage duty cycle theoretically.
6. Compare the theoretical values charging time (tc), discharging time (td) total time period/ Frequency and Duty cycle with the practical values.

8.8 Observation table

Table 8.1: **Tabular Column For Astable multivibrator**

S No	**Theoretical values**	**Practical values**

8.9 Model waveforms

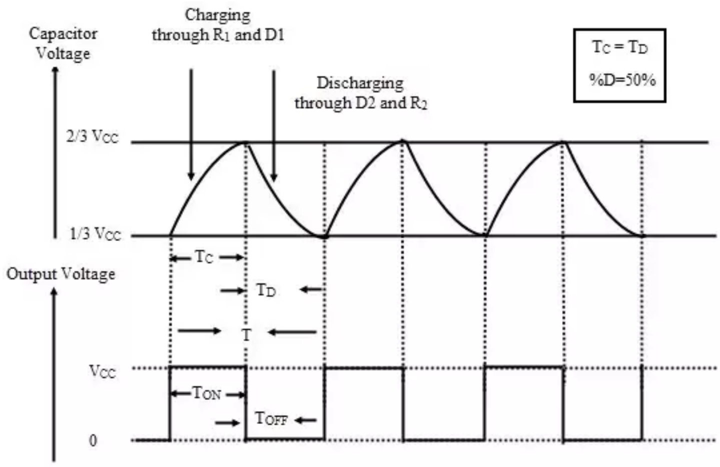

Figure 8.2: Astable Multivibrator waveforms

Observations:
Square wave Output signal
Amplitude = Time period =
Triangle wave Output signal
Amplitude = Time period =

8.10 Applicatios

1. square wave generator
2. Voltage controlled oscillator
3. FSK (frequency shift keying) generator

8.11 Result

Hence designed and studied IC 555 timer as an Astable multivibrator and also calculated the frequency of oscillations and time period of output waveform.

8.12 Further Probing Experiments

Q1. Consider the Astable multivibrator with R1=10KΩ, R2=200KΩ and C=0.1µF. Determine a) Frequency b) Duty cycle.

Q2. Design an Astable 555 timer circuit to produce a 2kHz square wave with a duty cycle of 70

LAB-8 Schmitt Trigger Circuits using IC 741 and IC 555 Timer

9.1 Introduction

A Schmitt trigger circuit is also called a regenerative comparator circuit. The circuit is designed with a positive feedback and hence will have a regenerative action which will make the output switch levels. The use of positive voltage feedback instead of a negative feedback, aids the feedback voltage to the input voltage, instead of opposing it.

9.2 Objectives

9.2.1 Educational

- Learn to measure output voltage and threshold voltage levels.
- Learn to measure hysteresis voltage of schmitt trigger circuit.
- Gain experience in the construction of IC741 and IC555 timers.

9.2.2 Experimental

- Determine the output voltage and threshold voltage levels.
- Compare theoretical and practical values of hysteresis voltage of schmitt trigger circuit.
- Draw the output voltage and threshold voltage levels.

9.3 Prelab Preparation:

9.3.1 Reading

- Read and study the Background section of this Laboratory.
- Read Appendix B, output voltage and threshold voltage levels Measurement of an oscilloscope.

9.3.2 Written

- Prior to coming to lab class, complete Part 0 of the Procedure.

9.4 Equipment needed

9.4.1 Hardware Requirements:

1. Function Generator
2. Regulated DC power Supply
3. Dual Channel Oscilloscope (CRO)
4. Digital Multimeter
5. CDS Board / Bread Board
6. Connecting wires

9.5 Circuit diagrams

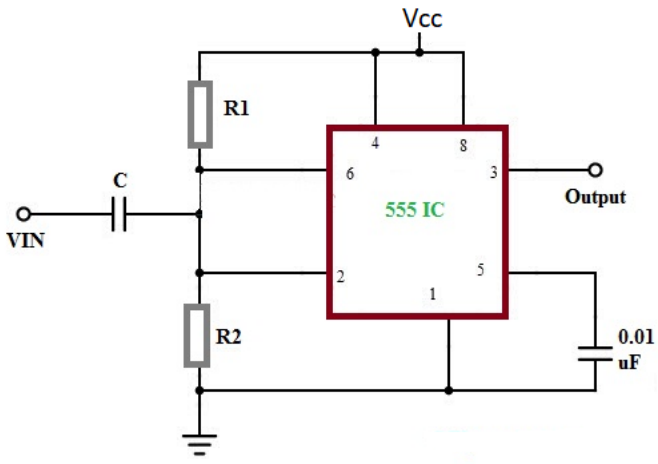

Figure 9.1: Schmitt trigger using IC 555 timer

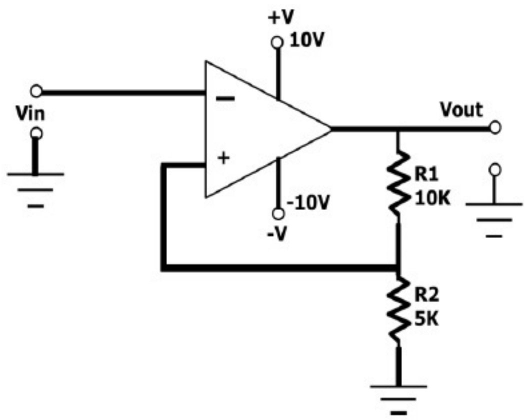

Figure 9.2: Schmitt trigger using IC 741

9.6 Background

In Schmitt Trigger two internal comparators are tied together and externally biased at VCC/2 through R1 and R2. Since the upper comparator will trip at (2/3) VCC and lower comparator at (2/3) VCC the bias provided by R1 and R2 is centered within these two thresholds. Thus, a sine wave of sufficient amplitude (greater than VCC/6 = 2/3 VCC – VCC/2) to exceed the reference levels causes the internal flip –flop to alternately set and reset providing a square wave output.

9.7 Procedure

For IC 741

1. Connect the circuit as shown Fig.
2. Set Function Generator output for sine wave signal of Amplitude at 1V(p-p) and frequency 1kHz.
3. Set R1 and R2 values at fixed positions and note down the values in tabular column. Calculate

theoretical values of Vut and Vlt and note down the values in tabular column. (+Vsat = 14V,-Vsat = -14V).
4. Apply Function Generator output at input terminals Vi, connect C.R.O- CH2 at output terminals Vo, C.R.O-CH1 at input terminals Vi.
5. Observe square wave output on C.R.O for the given input sine wave and compare them with the sample waveform as shown in fig.2.
6. Note down the practical Vut , Vlt and VH values in tabular column.
7. Compare the theoretical and practical values of Vut, Vlt and VH.
For IC 555 timer
1. Connect the circuit as shown in figure.
2. Apply the input sine wave 5V (P-P) using function generator at 1KHZ frequency.
3. Observe the output waveform at Pin No: 3.

9.8 Model waveforms

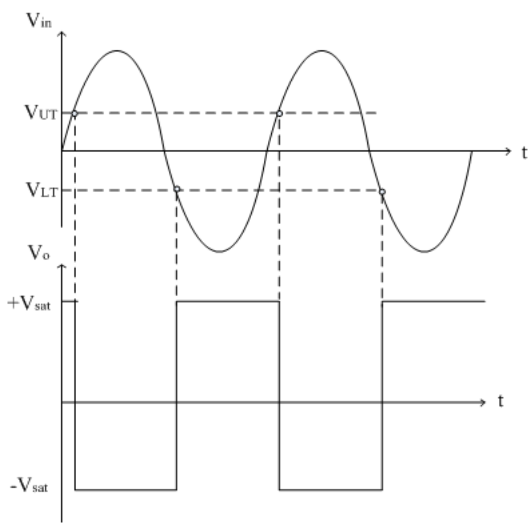

Figure 9.3: Schmitt trigger waveforms

Observations for 555 timer:
Input signal
Amplitude = Time period =
Output signal
Amplitude = Time period =
Observations for IC 741:
Input signal
Amplitude = Time period =
Output signal
Amplitude = Time period =

9.9 Applications

1. on/off controllers
2. Used as a comparator

9.10 Result

Hence constructed and studied Schmitt trigger using IC 741 and IC 555 timer.

9.11 Further Probing Experiments

Q1. Design a Schmitt trigger with an UTP =3V and LTP=5V and an input voltage of 10v.

Q2. Design the schmitt trigger with the given specification: R2 =56kΩ, R1 = 100Ω, Vref = 0v and Vsat = ±14v.

LAB-9 Phase Locked Loop (PLL) Using IC 565

10.1 Introduction

A phase-locked loop or phase lock loop (PLL) is a control system that generates an output signal whose phase is related to the phase of an input signal. There are several different types; the simplest is an electronic circuit consisting of a variable frequency oscillator and a phase detector in a feedback loop. The oscillator generates a periodic signal, and the phase detector compares the phase of that signal with the phase of the input periodic signal, adjusting the oscillator to keep the phases matched.

10.2 Objectives

10.2.1 Educational

- Learn to measure capture range and lock-in range of phase locked loop.
- Learn to measure theoretical frequency of PLL.
- Gain experience in the construction of IC 565.

10.2.2 Experimental

- Determine the capture range and lock-in range of phase locked loop.
- Compare theoretical and practical frequency of PLL.
- Draw the different lock-in ranges on the graph.

10.3 Prelab Preparation

10.3.1 Reading

- Read and study the Background section of this Laboratory.
- Read Appendix B, different lock-in ranges Measurement of an oscilloscope.

10.3.2 Written

- Prior to coming to lab class, complete Part 0 of the Procedure.

10.4 Equipment needed

10.4.1 Hardware Requirements

1. C.R.O
2. Function Generator
3. DC power supply

4. CDS board / Bread Board
5. Connecting wires

10.5 Circuit diagram

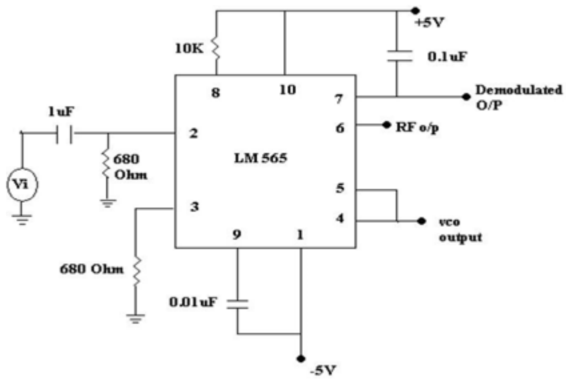

Figure 10.1: PLL using IC 565

10.6 Back ground

The PLL consists of i) a phase detector ii) a low pass filter and iii) a voltage-controlled oscillator. The phase detector, or comparator compares the input frequency fIN with the feedback frequency fOUT. The output of the phase detector is proportional to the phase difference between fIN and fout. The output voltage of a phase detector is a dc voltage and therefore is often referred to as the error voltage. The output of the phase detector is then applied to the low-pass filter, which removes the high-frequency noise and produces a dc level. This dc level, in turn, is the input to the voltage-controlled oscillator (VCO). The filter also helps in establishing the dynamic characteristics of the PLL circuit. The output frequency of the VCO is directly proportional to the input dc level. The VCO frequency is compared with the input frequencies and adjusted until it is equal to the input frequencies. In short, the phase-locked loop goes through three states: free running, capture, and phase lock. Before the input is applied, the phase-locked loop is in the free-running state. Once the input frequency is applied, the VCO frequency starts to change and the phase-locked loop is said to be in the capture mode. The VCO frequency continues to change until it equals the input frequency, and the phase-locked state. When phase locked, the loop tracks any change in the input frequency through its repetitive action.
Lock Range: The range of frequencies over which the PLL can maintain lock with incoming signal is called the “ Lock Range” or “Track Range”
FL= 8f0/V where V= + V −(−V), where f0 is free running frequency.
Capture range: The range of frequencies over which the PLL can acquire lock with an input signal is called the capture range.

10.7 Procedure

1. Apply +5v to pin 10 and –5v to pin 1 of LM565
2. Connect R1= 10KΩ resistor from pin 8 to10 and C1 =0.01µ F capacitor from pin 9 to 1.
3. Connect 680Ω resistor from pin 2 and pin 3 to ground.
4. Connect pin 4(VCO o/p) to CRO and measure its frequency. This frequency is called the free running frequency, fo.
5. Calculate f0 theoretically using the formula f0 = 1.2 /4R1C1 and compare it with practical value.
6. Apply square wave at the input with amplitude of 2Vpp and also connect it to channel 1 of CRO.
7. Connect pin 4(VCO o/p) to channel 2 of CRO.
8. Vary the input signal frequency in steps and measure its corresponding o/p frequency.
9. Find the lock range and capture range from the obtained data.

10.8 Observation Table

Table 10.1: **Tabular Column For PLL**

S No	**Input frequency, Hz**	**Output frequency, Hz**	**Fc, Hz**	**Fl, Hz**

10.9 Applications

1. Frequency multiplier
2. Frequency synthesizer
3. FM demodulator
4. FSK demodulator
5. AM detection
6. Frequency translation

10.10 Result

Free running frequency, lock range and capture range of PLL are measured practically and compared with theoretical values.

10.11 Further Probing Experiments

Q1. Design the phase locked loop circuit with the basic operating frequency of 2kHz and cut off frequency of 1kHz?

LAB-10 Voltage Regulator using IC 723

11.1 Introduction

The function of a voltage regulator is to maintain a constant DC voltage at the output irrespective of voltage fluctuations at the input and (or) variations in the load current. In other words, voltage regulator produces a regulated DC output voltage. Voltage regulators are also available in Integrated Circuits (IC) forms. These are called as voltage regulator ICs.

11.2 Objectives

11.2.1 Educational

- Learn to measure input voltage and output voltage.
- Learn to measure the load and line regulation of voltage regulator.
- Learn to calculate the percentage regulation.
- Gain experience in the construction of IC 723.

11.2.2 Experimental

- Determine the load and line regulation of voltage regulator.
- Calculate the percentage regulation.
- Draw the line and load regulation on the graph.

11.3 Prelab Preparation

11.3.1 Reading

- Read and study the Background section of this Laboratory.
- Read Appendix B, different voltage Measurement in regulated power supply.

11.3.2 Written

- Prior to coming to lab class, complete Part 0 of the Procedure.

11.4 Equipment needed

11.4.1 Hardware Requirements

1. Digital Multimeter
2. Connecting patch chords
3. CDS Board / Bread Board
4. Regulated Power Supply

11.5 Circuit diagrams

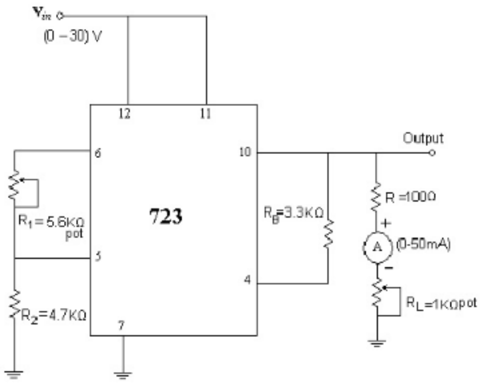

Figure 11.1: High voltage regulator using IC 723

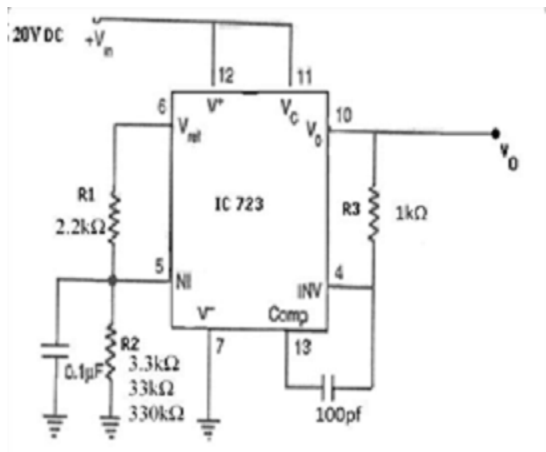

Figure 11.2: Low voltage regulator using IC 723

11.6 Back ground

The IC 723 is a monolithic integrated circuit voltage regulator featuring high ripple rejection, excellent input and load regulation and excellent temperature stability etc. It consists of a temperature compensating reference voltage amplifier, an error amplifier, 150mA output transistor and an adjustable output current limiter. The basic low voltage regulator type 723 circuit is shown in figure. The unregulated input voltage is 24V and the regulated output voltage is varied from 0.2V to 7.5V by varying the value of R2. A stabilizing capacitor (C1) of 100pF is connected between frequency compensation terminal and inverting (INV) terminal. External NPN pass transistor is added to the basic 723-regulator circuit to increase its load current capability. For intermediate output voltages the following formula can be used.
Vout = (R2/R1+R2) Vref
The basic high voltage regulator type 723 circuit is shown in figure. The output voltage can be regulated from 7 to 37Volts for an input voltage range from 9.5 to 40Volts.

11.7 Procedure

Low voltage regulator
1. Connect the circuit diagram as shown in figure.
2. Apply the unregulated voltage to the 723 IC and note down the regulator output voltage.
3. Calculate the line regulation of the regulator using the formula Line Regulation = ΔVO / ΔVi.
4. By varying 10K potentiometer at the load section and note down the regulator output voltage.
5. Calculate the Load regulation of the regulator using the formula
Load Regulation ==ΔVO / ΔIL
High voltage regulator
1. Connect the circuit diagram as shown in figure.
2. Apply the unregulated voltage to the 723 IC and note down the regulator output voltage.
3. Calculate the line regulation of the regulator using the formula Line Regulation = ΔVO / ΔVi
5. By varying 10K potentiometer at the load section and note down the regulator output Voltage.
6. Calculate the Load regulation of the regulator using the formula Load Regulation = ΔVO / ΔIL

11.8 Observation table

Low voltage regulator
Line regulation, RL is constant

Table 11.1: **Tabular Column For low voltage regulator**

S No	Vi, (V)	Regulated DC output voltage		
		3.3k	**33k**	**330k**

Load regulation:(Vi constant)

Table 11.2: **Tabular Column For low voltage regulator**

S No	**Load regulator, RL**	**Regulated DC output voltage**		
		3.3k	**33k**	**330k**

High voltage regulator
Line regulation, RL is constant

Table 11.3: **Tabular Column For high voltage regulator**

S No	Vi, (V)	Regulated DC output voltage		
		3.3k	**33k**	**330k**

Load regulation:(Vi constant)

Table 11.4: **Tabular Column For high voltage regulator**

S No	Load regulator, RL	Regulated DC output voltage		
		3.3k	33k	330k

11.9 Model waveforms

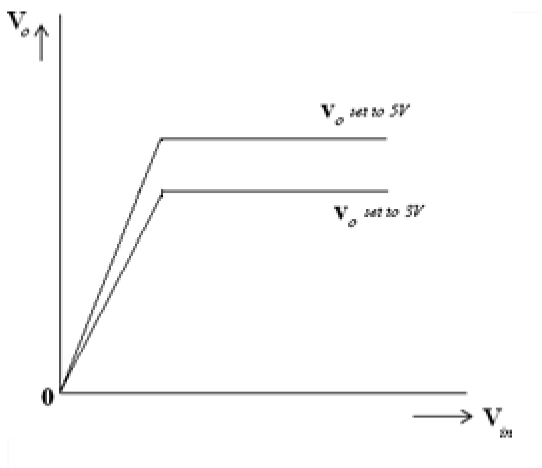

Figure 11.3: Line regulation

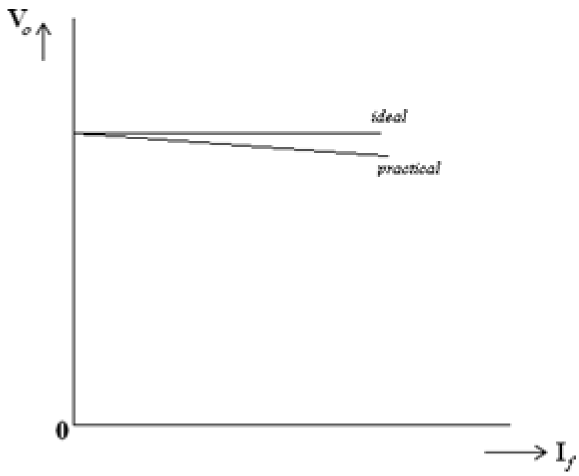

Figure 11.4: Load regulation

11.10 Applications

1. Low current consumption
2. Overvoltage/ Short-circuit protection
3. Reverse polarity protection
4. Over temperature protection
5. Load protection

11.11 Result

Low and high voltage regulators using IC 723 were constructed and studied. Also, the line and load regulations of the low and high voltage regulators are verified.

11.12 Further Probing Experiments

Q1. Design a high voltage and low voltage regulator using IC 723.

LAB-11 Digital to Analog (D/A) Converter

12.1 Introduction

A digital-to-analog converter (DAC or D-to-A) is a device for converting a digital (usually binary) code to an analog signal (current, voltage or charges). Digital-to-Analog Converters are the interface between the abstract digital world and the analog real life. Simple switches, a network of resistors, current sources or capacitors may implement this conversion.

12.2 Objectives

12.2.1 Educational

- Learn to measure different analog output voltages.
- Gain experience in the construction of IC 741.

12.2.2 Experimental

- Determine the different analog output voltages.
- Draw the output voltage on the graph.

12.3 Prelab Preparation

12.3.1 Reading

- Read and study the Background section of this Laboratory.
- Read Appendix B, different voltage Measurement in digital multimeter.

12.3.2 Written

- Prior to coming to lab class, complete Part 0 of the Procedure.

12.4 Equipment needed

12.4.1 Hardware Requirements

1. 4 – Bit D/A converter (R-2R) Trainer Kit.
2. Multimeter.
3. Connecting wires.

12.5 Circuit diagram

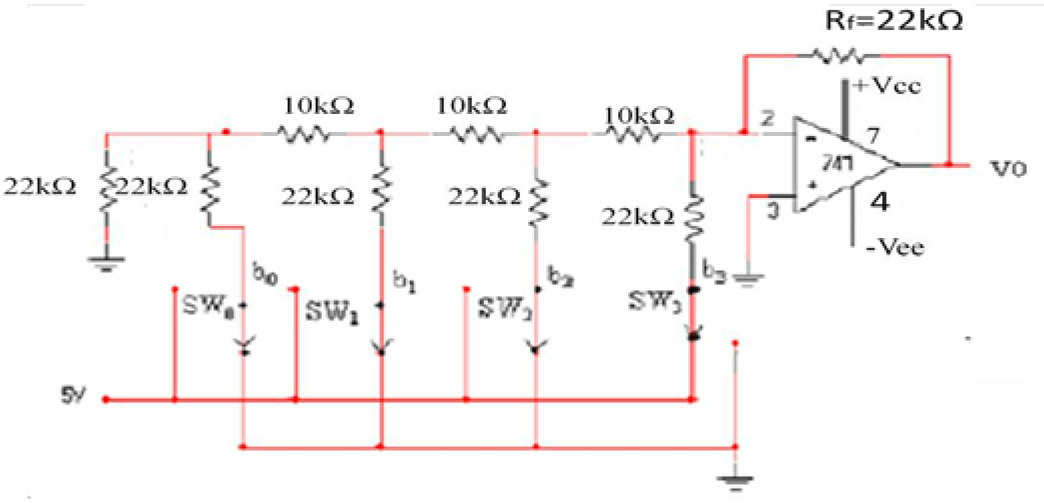

Figure 12.1: R-2R ladder DAC

12.6 Back ground

This 4-bit resistive ladder circuit may look complicated, but it is all about connecting resistors together in parallel and series combinations and working back to the input source using simple circuit laws to find the proportional value of the output. Let us assume all the binary inputs are grounded at 0 volts, that is: VA = VB = VC = VD = 0V (LOW). The binary code corresponding to these four inputs will therefore be: 0000. The output voltage for an inverting operational amplifier is given as: (RF/RIN)*VIN. If we make RF equal to R, that is RF = R = 1, and as R is terminated to ground (0V), then there is no VIN voltage value, (VIN = 0) so the output voltage would be: (1/1)*0 = 0 volts. So for a 4-bit R-2R DAC with four grounded inputs (LOW), the output voltage will be "zero" volts, thus a 4-bit digital input of 0000 produces an analogue output of 0 volts.

12.7 Procedure

1. Connect the trainer to the mains and switch on the power supply.
2. Measure the supply voltages of the circuit as +12V and -12V.
3. Calculate theoretically Vo for all digital Input data using formula.
V0= -Rf[b0/R + 2b1/R + 4b2/R + 8b3/R]
4. In this experiment Rf = 22kΩ and R= 10kΩ.
5. Note down Output voltages for different combinations of digital inputs and compare it with theoretical values.

12.8 Observation table

Table 12.1: **Tabular Column For R-2R ladder DAC**

Digital input				**Theoretical output voltage**	**Practical output voltage**
b0	**b1**	**b2**	**b3**		
0	0	0	0		
0	0	0	1		
0	0	1	0		
0	0	1	1		
0	1	0	0		
0	1	0	1		
0	1	1	0		
0	1	1	1		
1	0	0	0		
1	0	0	1		
1	0	1	0		
1	0	1	1		
1	1	0	0		
1	1	0	1		
1	1	1	0		
1	1	1	1		

12.9 Model waveform

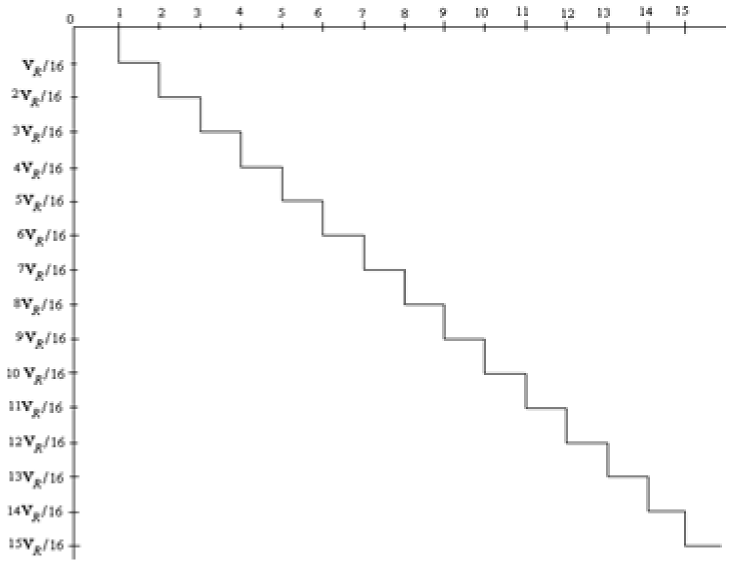

Figure 12.2: R-2R ladder DAC waveform

12.10 Applications

1. Audio
Most modern audio signals are stored in digital form (for example MP3s and CDs) and in Order to be heard through speakers they must be converted into an analog signal.
2. Video
Video signals from a digital source, such as a computer, must be converted to analog form If they are to be displayed on an analog monitor.

12.11 Result

Obtained analog output voltages for the given digital input data using 4- bit R-2R ladder network D/A converter.

12.12 Probing Further Experiments

Q1. Design the 4-bit digital to analog converter with the following input digital words, when 4-bit D/A converter with Vr=10V, Rf=10KΩ is considered i) 0001 ii) 0110 iii) 1010

LAB-12 Resistor-Transistor Logic(RTL)

13.1 Introduction

Resistor–transistor logic (RTL) (sometimes also transistor–resistor logic (TRL)) is a class of digital circuits built using resistors as the input network and bipolar junction transistor (BJTs) as switching devices. RTL is the earliest class of transistorized digital logic circuit used; other classes include diode-transistor logic (DTL) and transistor -transistor logic(TTL).

13.2 Objectives

13.2.1 Educational

- Learn to verify truth tables of NAND and NOR logic.
- Learn to measure the logic high and logic low level voltages.
- Gain experience in the construction of logic families.

13.2.2 Experimental

- Verify truth tables of NAND and NOR logic.
- Measure the logic high and logic low level voltages.

13.3 Prelab Preparation

13.3.1 Reading

- Read and study the Background section of this Laboratory.
- Read Appendix B, different voltage level Measurement in digital multimeter.

13.3.2 Written

- Prior to coming to lab class, complete Part 0 of the Procedure.

13.4 Equipment needed

13.4.1 Hardware Requirements

1. Transistors (BC107)
2. Resistors
3. Connecting wires

13.5 Circuit diagram

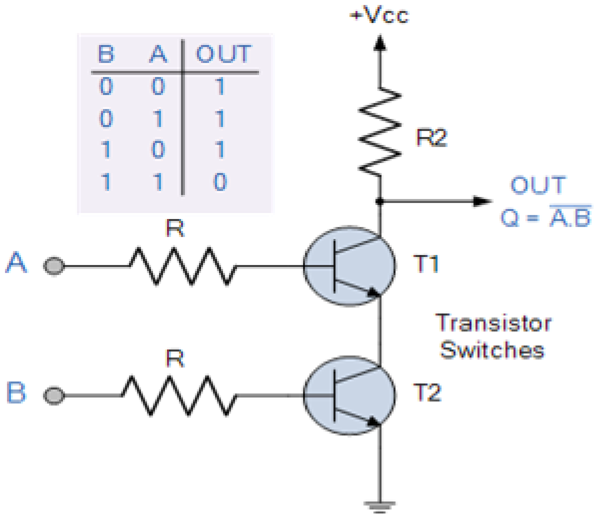

Figure 13.1: NAND-RTL

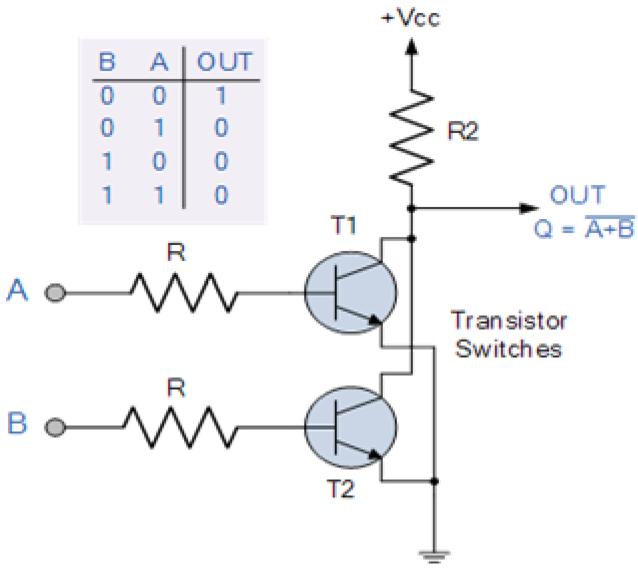

Figure 13.2: NOR-RTL

13.6 Back ground

NOR gate:
When both the inputs A and B are at 0V or logic 0, it is not enough to turn on the gates of both the transistor. So, the transistors will not conduct. Due to this, the voltage +VCC will appear at the output Y. Hence the output is logic 1 or logic HIGH at terminal Y. When any one of the inputs, either A or B is given HIGH voltage or logic 1, then the transistor with HIGH gate input will be turned on. This will make a path for the supply voltage to go to the ground through the resistor RC and transistor. Thus, there will be 0 v at the output terminal Y. When both the inputs are HIGH, it will drive both the transistor to turn on. It will make a path for

the supply voltage to flow to the ground through resistor RC and transistor. Therefore, there will be 0 v at the output terminal Y.

NAND gate:

The NAND gate implemented using resistor-transistor logic, the earliest form of logic implemented with transistors. Click on the inputs on the left to toggle their state. When all of the inputs are high, the output is low; otherwise, the output is high. When all the inputs are high (3.6 V), a current flow from the base to the emitter of all the transistors. Each transistor wants its collector-emitter current to be 100 times the base current, but it can't, because the collector is connected to the same voltage through a larger resistor. So, the transistors are in saturation mode; they maximize the current to bring the output voltage down as low as possible. When any of the inputs are low (at ground), no current flows through the base of the corresponding transistor, so it switches off. With no path to ground, the output stays at 3.6 V.

13.7 Procedure

1. Connect the circuit as shown in figure.
2. Assemble the circuit on your breadboard for NOR/NAND operation.
3. Apply all four possible combinations of inputs at A and B from the power supply using dip switch.
4. For each input combination, note the logic state of the output, Q, as indicated by the LED (ON = 1; OFF = 0), and record that result in the table.
5. Compare your results with the truth table of a logic "NOR"/ "NAND" operation.
6. Measure the logic HIGH- and LOW- level voltages in digital multimeter.

13.8 Observation table

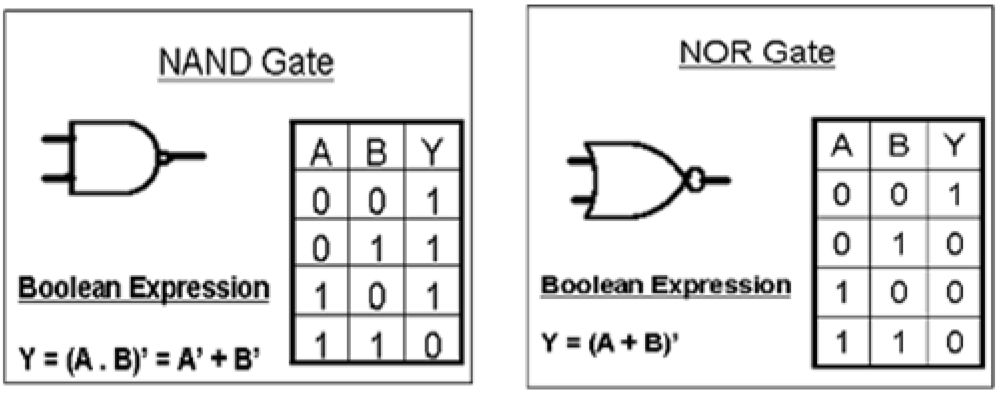

Figure 13.3: RTL truth table

13.9 Result

Obtained logic Low- and high- level voltages using RTL logic through NAND and NOR implementation.

13.10 Further Probing Experiments

Q1. Design a two input positive logic RTL NAND gate due to the following specifications: RL = 1 KΩ and maximum collector current ICmax = 10 mA.

LAB-13 Diode-Transistor Logic(DTL)

14.1 Introduction

Diode–transistor logic (DTL) is a class of digital circuits that is the direct ancestor of transistor-transistor logic. It is called so because the logic gating function (e.g., AND) is performed by a diode network and the amplifying function is performed by a transistor (in contrast with RTL and TTL).

14.2 Objectives

14.2.1 Educational

- Learn to verify truth tables of NAND and NOR logic.
- Learn to measure the logic high and logic low level voltages.
- Gain experience in the construction of logic families.

14.2.2 Experimental

- Verify truth tables of NAND and NOR logic.
- Measure the logic high and logic low level voltages.

14.3 Prelab Preparation

14.3.1 Reading

- Read and study the Background section of this Laboratory.
- Read Appendix B, different voltage level Measurement in digital multimeter.

14.3.2 Written

- Prior to coming to lab class, complete Part 0 of the Procedure.

14.4 Equipment needed

14.4.1 Hardware Requirements

1. Transistors (BC107)
2. Resistors
3. Diodes
4. Connecting wires

14.5 Circuit diagram

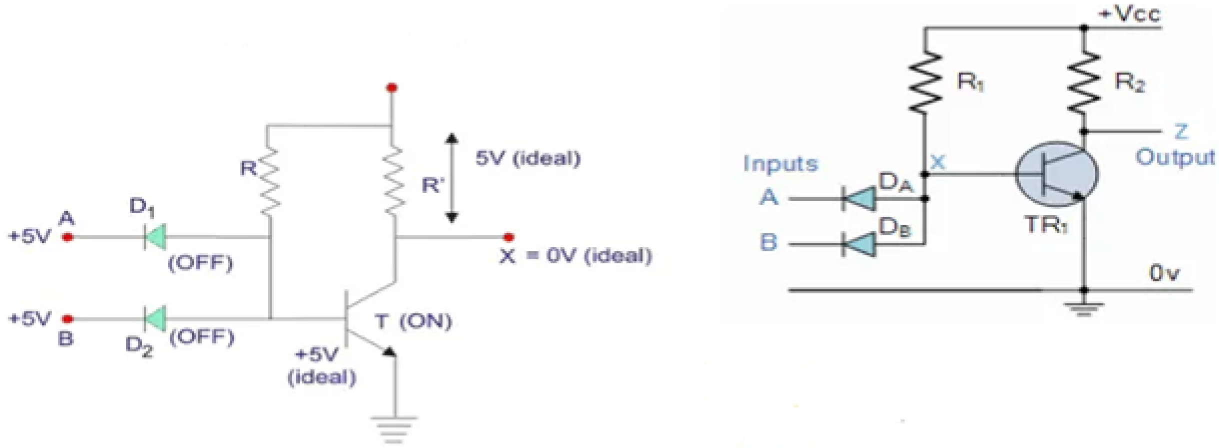

Figure 14.1: DTL circuit diagram

14.6 Background

NAND gate
The NAND gate is called DTL NAND gate or Diode Transistor Logical NAND Gate. When both input A and B are given with 0 V, both of the diodes are in forward biased condition that is in ON condition. Supply voltage will get path to the ground through diode D1 and D2. Entire supply voltage +5 V will ideally drop across resistor R and hence base terminal of transistor T will not get enough potential to turn ON the transistor and hence the transistor will be in OFF condition. As a result supply voltage +5 V will appear at output terminal X and hence output X will become high or logical 1.
NOR gate
This is a Diode-Transistor Logic (DTL) NOR Gate circuit using a general-purpose bipolar junction transistor (BC547), and general-purpose diodes (1N4148). This page shows how to make this circuit, and the implementation is on a breadboard using discrete components. The working logic is very simple. The emitter-collector junctions of the transistor connect in parallel to the light emitting diode (LED). When the transistor conducts (ON), it bypasses the current because almost all the current passes through the emitter-collector junctions, and therefore the LED goes OFF. The transistor will conduct when either one or both of the inputs receive a logic 1 signal. When both inputs are logic 0, the transistor stops conducting (OFF) and all the current passes through the LED instead, hence the LED lights up.

14.7 Procedure

1. Connect the circuit as shown in figure.
2. Assemble the circuit on your breadboard for NOR/NAND operation.
3. Apply all four possible combinations of inputs at A and B from the power supply using dip switch.
4. For each input combination, note the logic state of the output, Q, as indicated by the LED (ON = 1; OFF = 0), and record that result in the table.
5. Compare your results with the truth table of a logic “NOR”/ “NAND” operation.
6. Measure the logic HIGH- and LOW- level voltages in digital multimeter.

14.8 Observation table

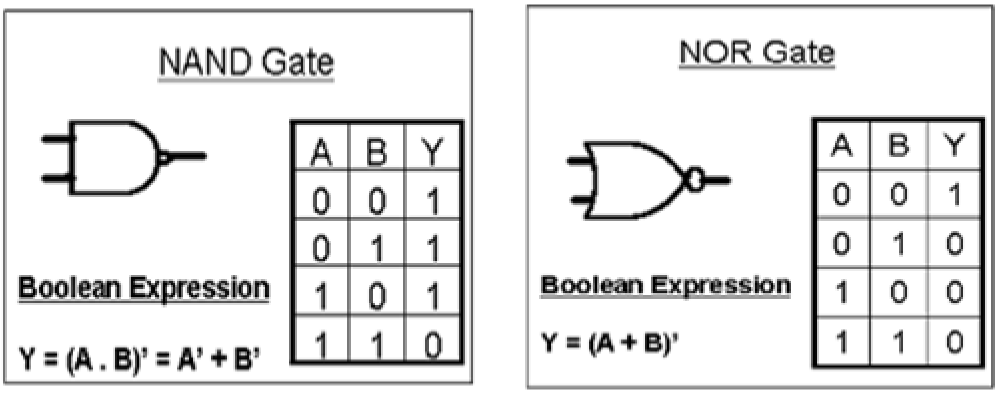

Figure 14.2: DTL truth table

14.9 Result

Obtained logic Low- and high- level voltages using DTL logic through NAND and NOR implementation.

14.10 Further Probing Experiments

Q1. Design a two input positive logic DTL NAND gate due to the following specifications: RL = 1.5 KΩ and maximum collector current ICmax = 10 mA.

LAB-13 Instrumentation Amplifier using IC 741

15.1 Introduction

An instrumentation amplifier (sometimes shorthanded as In-Amp or InAmp) is a type of differential amplifier that has been outfitted with input buffer amplifiers, which eliminate the need for input impedance matching and thus make the amplifier particularly suitable for use in measurement and test equipment. Additional characteristics include very low DC offset, low drift, low noise, very high open-loop gain, very high common-mode rejection ratio, and very high input impedances. Instrumentation amplifiers are used where great accuracy and stability of the circuit both short and long-term are required.

15.2 Objectives

15.2.1 Educational

- Learn to verify the output voltage of instrumentation amplifier.
- Learn to measure the voltage gain of instrumentation amplifier.
- Gain experience in the construction of IC 741.

15.2.2 Experimental

- Determine the output voltage of instrumentation amplifier.
- Measure the voltage gain of instrumentation amplifier.

15.3 Prelab Preparation

15.3.1 Reading

- Read and study the Background section of this Laboratory.
- Read Appendix B, different voltage level Measurement in digital multimeter.

15.3.2 Written

- Prior to coming to lab class, complete Part 0 of the Procedure.

15.4 Equipment needed

15.4.1 Hardware Requirements:

1. IC741
2. Resistors
3. Connecting wires
4. Digital multimeter

15.5 Circuit diagram

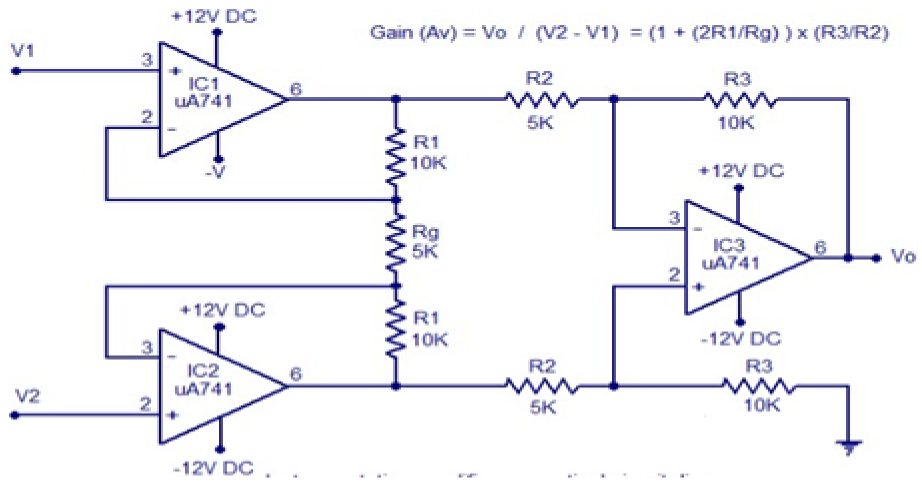

Figure 15.1: Instrumentation amplifier

15.6 Background

Instrumentation amplifier is a kind of differential amplifier with additional input buffer stages. The addition of input buffer stages makes it easy to match (impedance matching) the amplifier with the preceding stage. Instrumentation are commonly used in industrial test and measurement application. The instrumentation amplifier also has some useful features like low offset voltage, high CMRR (Common mode rejection ratio), high input resistance, high gain etc. In the circuit diagram, opamps labelled A1 and A2 are the input buffers. Anyway the gain of these buffer stages are not unity because of the presence of R1 and Rg. Op amp labelled A3 is wired as a standard differential amplifier. R3 connected from the output of A3 to its non inverting input is the feedback resistor. R2 is the input resistor. The voltage gain of the instrumentation amplifier can be expressed by using the equation below. Voltage gain (Av) = Vo/(V2-V1) = (1 + 2R1/Rg) x R3/R2

15.7 Procedure

Connect the circuit as shown in figure.
The amplifier operates from +/-12V DC and has a gain 10.
If you need a variable gain, then replace Rg with a 5K POT.
Measure the Voltage gain by using given formulae.

15.8 Result

Verified the theoretical and practical output voltage of instrumentation amplifier.

15.9 Further Probing Experiments

Q1. Design an instrumentation amplifier with following specifications:
R1 = 15 KΩ, R2 = 5 KΩ, R3 = 10 KΩ and Rg = 10 KΩ.

Appendix A - Safety

Electricity, when improperly used, is very dangerous to people and to equipment. This is especially true in an industrial environment where large amounts of power is used, and where high voltages are present [1]; in environments where people are especially susceptible to electric shock such as maintenance of a high voltage system (while in operation) or in hospitals where electrical equipment is used to test or control physiological functions [2, 3]; and in an experimental or teaching laboratory where inexperienced personnel may use electrical equipment in experimental or nonstandard configuration.

Engineers play a vital role in eliminating or alleviating the danger in all three types of environments mentioned above. For conditions where standard equipment is used in standard configurations, govern- mental agencies and insurance underwriters impose strict laws and regulations on the operation and use of electrical equipment including switchgear, power lines, safety devices, etc. As a result, corporations and other organizations in turn impose strict rules and methods of operation on their employees and contractors. Engineers who are involved in using electrical equipment, in supervising others who use it, and in designing such systems, have a great responsibility to learn safety rules and practices, to observe them, and to see that a safe environment is maintained for those they supervise. In any working environment there is always pressure to "get the job done" and take short cuts. The engineer, as one who is capable of recognizing hazardous conditions, is in a responsible position both as an engineer and as a supervisor or manager and must maintain conditions to protect personnel and avoid damage to equipment.

Because of their non-standard activities, experimental laboratories are exempt from many of these rules and regulations. This puts more responsibility on the engineer in this environment to know and enforce the safest working procedures.

The knowledge and habit-forming experience to work safely around electrical equipment and the ability to design safe electrical equipment begins with the first student laboratory experience and con- tinues through life. This includes learning the types of electrical injuries and damage, how they can be prevented, the physiology of electrical injuries, and steps to take when accidents.

Physiology of Electrical Injuries

There are three main types of electrical injuries: electrical shock, electrical burns, and falls caused by electrical shock. A fourth type, 'sunburned' eyes from looking at electric arcs, such as arc-welding, is very painful and may cause loss of work time but is usually of a temporary nature. Other injuries may be indirectly caused by electrical accidents, e.g., burns from exploding oil-immersed switch gear or transformers.

Although electric shock is normally associated with high-voltage AC contact, under some circum- stances death can occur from voltages from substantially less than the nominal 120 Volts AC found in residential systems. Electric shock is caused by an electric current passing through a part of the human body. The human body normally has a high resistance to electric currents so that a high voltage is usually required to cause lethal currents. This resistance is almost all in the skin, but when the skin is wet its resistance is much lower. When a person is hot and sweaty or is standing in water, contact with 120 Volts or less is likely to cause a fatal shock.

Electric shock is not a single phenomenon but is a disturbance of the nerves that is caused by electric current. A current through a part of the body such as the arm or leg will cause pain and muscle contraction. If a victim receives an electric shock from grasping a live conductor, a current of greater than 15 to 30 mA through the arm will cause muscle contractions so severe that the victim cannot let go. Similar currents through leg muscles may cause sudden contractions causing the victim to jump or fall, resulting in possible injuries or death. It is also possible for a prolonged period of contact of more than a minute or so to cause chest muscles to be contracted, preventing breathing and resulting in suffocation or brain damage from lack of oxygen.

The predominant cause of death by electric shock is generally attributed to ventricular fibrillation, which is an uncontrolled twitching or beating of the heart that produces no pumping action and therefore no blood circulation. Unless corrective action is taken, death follows quickly from lack of oxygen to the brain. While the amount of current that will cause fibrillation depends on several variables, 0.5 to 5A through the body will normally cause the very small current through the heart that causes fibrillation in most people. Larger currents than this through the heart causes contraction or clamping of the heart muscle and resulting death unless corrective action is taken. Prolonged contact of more than a minute or so may cause chest muscles to contract, preventing breathing and resulting in suffocation or brain damage from lack of oxygen.

Death by electric shock is most often attributed to ventricular fibrillation, which is an uncontrolled twitching or beating of the heart that produces no pumping action and therefore no blood circulation. Unless corrective action is taken, death follows quickly from lack of oxygen to the brain. While the amount of current that will cause fibrillation depends on several variables, 0.5 to 5 amperes through the body will normally cause the very small current (approximately 1 mA) through the heart that is sufficient to cause fibrillation in most people. Larger currents than this through the heart cause contraction or clamping of the heart muscle, resulting in death unless corrective action is taken.

Electric burns may be caused by electric currents flowing in or near parts of the body. Such burns are similar to burns from ordinary heat sources, except that those caused by high-frequency currents are generally deeper and take longer to heal the other burns. Electrocution will often leave severe burns at the points where the current entered and left the body.

Source of Electric Shock

Since electric shock is caused by an electric current through a part of the body, it is prevented by not allowing the body to become part of any electric circuit. From this viewpoint, electric circuits may be classified as either grounded or ungrounded.

Electric circuits may be classified as either grounded or ungrounded. Grounded circuits are safer for most conditions, since they result in known voltages at other points in the circuit and provide easier and better protection against faulty conditions in the circuit. The disadvantage is that a person standing on a non-insulated floor can receive a shock by touching only one conductor.

Almost all electric power generation, transmission, and distribution systems are grounded to pro- tect people and equipment against fall conditions caused by windstorms, lightning, etc. Residential, commercial, and industrial systems such as lighting and heating are always grounded for greater safety. Communication, computer, and similar systems are grounded for safety reasons and to prevent or reduce noise, crosstalk, static, etc. Many electronic equipment or instruments are grounded for safety and noise prevention, also. Common examples are DC power supplies, oscilloscopes, oscillators, and analog and digital multimeters.

Ungrounded circuits are used in systems where isolation from other systems is necessary, where low voltages and low power are used, and in other instances where obtaining a ground connection is difficult or impractical. In the ungrounded circuit, contact with two points in the circuit that are at different potentials is required to produce an electrical shock. The hazard is that with no known ground, a hidden fault can occur, causing some unknown point to be grounded, in which case, touching a supposedly safe conductor while standing on the ground could result in an electric shock.

Protecting People and Equipment in the Laboratory

Prevention of electric shock to individuals and damage to equipment in the laboratory can be done by strict adherence to several common-sense rules summarized below:

Protecting People

1. When hooking up a circuit, connect to the power source last, while power is off.
2. Before making changes in a circuit, turn off or disconnect the power first, if possible.
3. Never work alone where the potential of electric shock exists.
4. When changing an energized connection, use only one hand. Never touch two points in the circuit that are at different potentials.
5. Know that the circuit and connections are correct before applying power to the circuit.
6. Avoid touching capacitors that may have a residual charge. The stored energy can cause a severe shock even after a long period of time.
7. Insulate yourself from ground by standing on an insulating mat where available.

The above rules and the additional rules given below also serve to protect instruments and other circuits from damage.

Protecting Equipment

1. Set the scales of measurement instrument to the highest range before applying power.
2. Before making changes in a circuit, turn off or disconnect the power first, if possible.
3. When using an oscilloscope, do not leave a bright spot or trace on the screen for long periods of time. Doing so can burn the image into the screen.
4. Be sure instrument grounds are connected properly. Avoid ground loops and accidental grounding of “hot” leads.
5. Check polarity markings and connections of instruments carefully before connecting power.
6. Never connect an ammeter across a voltage source, but only in series with a load.
7. Do not exceed the voltage or current ratings of circuit elements or instruments. This particularly applies to wattmeters, since the current or voltage rating may be exceeded with the needle still reading on the scale.
8. Be sure any fuses and circuit breakers are of suitable value.

When connecting electrical elements to make up a network in the laboratory, it easy to lose track of various points in the network and accidentally connect a wire to the wrong place. One procedure to help avoid this problem is to connect first the main series loop of the circuit, then go back and add the elements in parallel.

Types of Equipment Damage Excessive currents and voltages can damage instruments and other circuit elements. A large over- current for a short time or a smaller over-current for a longer time will cause overheating, resulting in insulation scorching and equipment failure.

Blown fuses are the most common equipment failure mode in this laboratory. The principal causes for these failures include:

- incorrectly wired circuits;
- accidental shorts;
- switching resistance settings while power is applied to the circuit;
- changing the circuit while power is applied;
- using the wrong scale on ammeter;
- connecting an ammeter across a voltage source;
- using a low-power resistor box (limit 1/2 amp) when high power is required;
- turning on an auto-transformer at too high a setting.

All of these causes are the result of carelessness by the experimenter.

Some type of insulating material, such as paper, cloth, plastic, or ceramic, separates conductors that are at different potentials in electrical devices. The voltage difference that this material can withstand is determined by design (type, thickness, moisture content, temperature, etc.). Exceeding the voltage rating of a device by an appreciable amount can cause arcing or corona, resulting insulation breakdown, and failure.

Some electrical devices can also be damaged mechanically by excessive currents. An example is the D’Arsonval meter, the indicator in most analog metering instruments. A large pulse of over current will provide mechanical torque that can cause the needle to wrap around the pin at the top of the scale, thereby causing permanent damage even though the current may not have been on long enough to cause failure due to overheating.

After Accident Action

Since accidents do happen despite all efforts to prevent them, plans for appropriate reaction to an accident can save time and lives. Such a plan should include immediate availability of first aid material suitable for minor injuries or for injuries that are likely because of the nature of the work. Knowledge of how to obtain trained assistance such as Emergency Medical Services (EMS) should be readily available for everyone.

Treating victims for electrical shock includes four basic steps that should be taken immediately. Step two requires qualification in CPR and step three requires knowledge of mouth-to-mouth resuscitation. Everyone who works around voltages that can cause dangerous electrical shock should take advantage of the many opportunities available to become qualified in CPR and artificial respiration.

Immediate Steps After Electric Shock

1. Shut off all power and remove victim from the electric circuit. If the power cannot be shut off immediately,use an insulator of some sort, such as a wooden pole, to remove victim from the circuit. Attempts to pull the victim from the circuit with your hands will almost always result in your joining the victim in the electric shock.

2. If you are qualified in CPR, check for ventricular fibrillation or cardiac arrest. If either is detected, external cardiac massage should be started at once. Whether you are qualified in CPR or not, notify EMS and the ECE Department at once, using the telephone numbers listed below.

3. Check for respiratory failure and take appropriate action. This may have resulted from physical paralysis of respiratory muscles or from a head injury. Sometimes many hours pass before normal respiration returns. Artificial respiration should be continued until trained EMS assistance arrives.

4. Check for and treat other injuries such as fractures from a fall or burns from current entry and exit sites. Investigations are always after accidents. As an engineer you will be involved as a part of the investigating team or in providing information to an investigator. Information obtained and notes written immediately after the emergency will aid this investigation and assist in preventing future accidents of a similar nature.

Investigations are always made after accidents. As an engineer, you will be involved as a part of the investigating team or in providing information to an investigator. Information obtained and notes written immediately after the emergency will aid the investigation and assist in preventing future accidents of a similar nature.

Emergency Numbers

Fire / EMS: 911 or (864) 656-2222

Student Health Center: (864) 656-2233

ECE Department Office: (864) 656-5650

Appendix A References

1. 1. D. Roy Chowdhury, “Linear Integrated Circuits”, New age international (p) Ltd, 2nd Edition,2003

2. Ramakanth A. Gayakwad, “Op-Amps linear ICs”, PHI, 3rdEdition,2003.

3. John F. Wakerly, “Digital Design Principles and Practices”, Prentice Hall, 3rdEdition,2005.

4. Salivahanan, “Linear Integrated Circuits and Applications”, TMH, 1st Edition, 2008.

Appendix B - Instruments for Electrical Measurements

Electrical engineers measure and use a wide variety of electrical circuit variables, such as voltage, current, frequency, power, and energy, as well as electrical circuit parameters, such as resistance, capacitance, and inductance. Many instruments can be used to make such measurements, but the proper use of the instruments and interpretation of the measurements depend on a fundamental understanding of how the instruments work, their capabilities, and their limitations.

This appendix provides a brief overview of the fundamentals of the electrical equipment and instru- ments that you will use in this and other laboratory courses. As you encounter more and varied types of electrical equipment and instruments in this and subsequent courses, you will find several books, in addition to your textbook, useful in developing your understanding and measurement skills. In addition, many commercial instrument manufacturers publish handbooks and application notes that provide more information on specific measurement techniques.

Digital Multimeter

A multimeter is an electronic device that measures a multitude of electrical values, usually including at least AC and DC voltage and current, as well as resistance. Analog multimeters have an analog display and digital multimeter (DMM) have a digital display. The DMM in this laboratory is used to measure voltage (DC and AC), current (DC and AC), resistance, capacitance. Additionally it may be used for diode tests and audible continuity tests. For capacitance and inductance measurements you must make connections to the DMM/Impedance Analyzer on the prototyping board. For all other measurements make connections to the DMM banana jacks on the workstation.

Dual-Beam Oscilloscope

The oscilloscope is a tool to allow engineers to look at the shape of an electrical voltage versus time or versus a second signal. Until relatively recently, oscilloscopes used a cathode ray tube (CRT) to draw the waveforms onto a screen, just like an image on a television. Televisions and computer monitors also used cathode ray tubes until the advent of the new flat screens. Most engineers refer to the instrument as a “scope”.

The Dual-Beam Oscilloscope has two vertical (“y”) input channels and one horizontal (“x”) channel. The horizontal channel can be connected either to an external AC voltage signal or to an internal time- base generator (x = time). You should become familiar with the scale options on the y input channels, the x input channel, and the time base, since you will be using these to obtain values for voltage and time. The y inputs can be either direct (1X) or through a 10X probe. Figure shows the equivalent input circuit for a direct input and Figure shows an equivalent circuit for the 10X probe input. Note that in both configurations one side of the input is grounded, which means that care must be used in connecting the ground clip of the probe or connector used to assure that these are not connected to a “hot” ($V > 0$) part of the circuit. (See the section “Oscilloscope Grounding Errors”.) The calibrated time base is useful when measuring the phase difference between two waveforms (on the y1 and y2 inputs) by carefully lining up the zero levels for both y inputs and then using in the ac-couple mode to observe the time difference between zero crossings of the two waveforms.

textbfDigital Storage Oscilloscope The digital storage oscilloscope (DSO) is now the preferred type of oscilloscope for most industrial applications, although analog oscilloscopes are still widely used. The DSO uses digital memory to store data as long as required without degradation. The digital storage allows use of an enormous array of sophisticated digital signal processing tools for the analysis of complex waveforms in today's circuitry.

The digital storage oscilloscopes of the Analog Discovery 2 are dual-beam oscilloscopes with two vertical inputs, as described above. The vertical input on the oscilloscope, instead of driving a vertical amplifier, is digitized by an analog-to-digital (A-to-D) converter to create a data set that is stored in the memory of a microprocessor. The data set is processed and then sent to the display. The data set can be written to a flash drive or sent over a LAN or a WAN for processing or archiving. The screen image can be directly recorded on paper by means of an attached printer or plotter, without the need for an oscilloscope camera. The scope's own signal analysis software can extract many useful time- domain features (e.g. rise time, pulse width, amplitude), frequency spectra, histograms and statistics, persistence maps, and a large number of parameters meaningful to engineers in specialized fields such as telecommunications, disk drive analysis, and power electronics.

Digital oscilloscopes are limited principally by the performance of the analog input circuitry and the sampling frequency. In general, the sampling frequency should be at least the Nyquist rate – double the frequency of the highest-frequency component of the observed signal – to avoid aliasing.

Appendix C - Operating Instructions for a Typical Oscilloscope

The oscilloscope is an instrument for the analysis of electrical circuits by observation of voltage and current waves. It may be used to study frequency, phase angle, and time, and to compare the relation between two variables directly on the display screen. Perhaps the greatest advantage of the oscilloscope is its ability to display the periodic waveforms being studied.

Until recently, oscilloscopes used a cathode-ray tube to display the signals of interest. A cathode ray tube (CRT) contains an electron gun that directs a high-velocity beam of electrons onto a fluorescent screen. The beam is controlled by a pair of horizontal and a pair of vertical deflecting plates. When the voltage on the deflection plates is equal to zero, the beam produces a spot of light in the center of the screen. Any potential applied to the plates creates an electric field that deflects the electron beam proportionally to the applied voltage.

The basic components of the traditional oscilloscope are cathode-ray tube, amplifiers, sweep or tim ing oscillator, and power supply. A voltage to be observed is applied to the vertical deflection plates. This signal may be amplified by the vertical amplifier in order to obtain a satisfactory vertical deflection. Meanwhile, a sweep oscillator moves the beam horizontally at a uniform rate. The simultaneous horizontal and vertical sweep of the beam across the CRT screen displays the waveform of the voltage applied to the vertical plates. The sweep oscillator blanks the CRT electron gun during its reverse sweep across the screen to switch off the electron beam.

If several voltage waveforms are to be studied and must maintain their relative phase positions, the sweep generator must be synchronized to the same voltage during the entire test. In this case, one voltage is applied to the oscilloscope as an external trigger.

An independent voltage may be applied to the horizontal input in place of the sweep oscillator voltage. In this case, two independent input voltages are displayed against one another. If the horizontal frequency is a submultiple of the vertical frequency, the trace will form a stationary pattern on the screen.

The traditional CRT oscilloscopes are rapidly being replaced with digital oscilloscopes that have flat-panel liquid crystal displays (LCDs), some with color displays. Instead of directly applying the incoming voltages to deflection plates, the digital oscilloscopes capture the voltage information and store it in computer memory as digital signals, which are then analyzed and displayed on the LCD. While the new digital scopes handle the incoming signal differently from the CRT-based scopes, the basic purpose and many of the operational controls remain the same. Therefore, the discussion that follows applies, for the most part, to both types of oscilloscopes. Avoiding Grounding Errors with Oscilloscope The shield or ground wire on the oscilloscope's signal input is connected to the oscilloscope's chassis ground, and therefore to the ground lead on the instrument's electrical power connection. This fact creates the possibility of grounding errors when making connections to circuits, especially when trying to measure voltages across ungrounded components.

The problem is illustrated in the following diagram. Suppose one wants to measure the voltage across resistor R1. Because the circuit and the oscilloscope both have the same ground, connecting the oscilloscope's input leads directly across R1, as indicated, will create a short across R2. Besides giving an incorrect reading, such a connection might damage either the oscilloscope or the circuit under test.

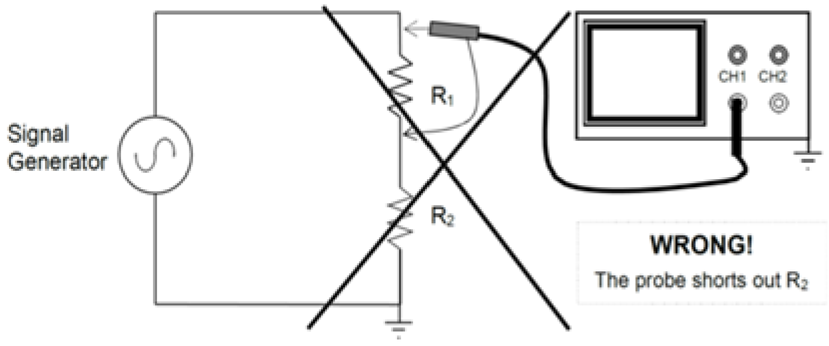

Figure C.1: Grounding error

Waveform Observation

After the initial adjustments are made, the oscilloscope is ready for operation. To observe the waveform of any periodic signal, apply the signal to the vertical input terminals. Use DC coupling if the input is DC or very low in frequency or if you want to capture any DC offset. Now the signal from an oscillator, signal generator, or some component of an electrical circuit may be observed on the screen. The best resolution of the waveform is obtained when the time scale is adjusted so one or two cycles appear on the screen and when the vertical scale is adjusted so the amplitude occupies most of the graticule. If the waveform will not stabilize, adjust the SYNC or TRIGGER just enough to cause the pattern to stop. Whenever possible, connect the oscilloscope ground to the common ground of the circuit. Exercise great care when making measurements with both terminals above ground potential, as there may be a difference in potential between two instrument cases, causing ground loop currents, faulty readings, and damaged equipment.

Voltage Measurement (AC and DC)

The oscilloscope has advantages as a voltmeter: a very high input impedance compared to an analog voltmeter; the ability to measure voltages over a very wide frequency range; and the ability to indicate magnitude regardless of waveform. Also, scopes measure peak-to-peak values of AC voltages, whereas standard AC voltmeters measure rms values of sine wave voltages.

To use the oscilloscope as an AC voltmeter, apply the signal to the vertical input terminals, and adjust the calibrated VERTICAL SENSITIVITY (or VOLTS/DIV) so the amplitude is of suitable magnitude on the graticule. The peak-to-peak value is then the distance indicated multiplied by the vertical calibration. For example, assume that a sine wave generator is set to 1000 Hz and adjusted for maximum output voltage. A peak-to-peak value of 60V is observed on the oscilloscope. The output of the generator at 1000 Hz, therefore, is approximately 60V peak-to-peak, and 21.2 Vrms.

For DC measurements, apply the voltage to the vertical input terminals, again suitably adjusting the VERTICAL SENSITIVITY. A straight line is produced with the horizontal sweep functioning. With no horizontal voltage applied, a spot will appear on the screen. In measuring DC voltages, it is necessary to remember where the trace was with 0V applied to the vertical input.

Frequency Measurement

The frequency of an unknown signal may be calculated from the oscilloscope very easily. The period of the waveform is the product of the distance along the x-axis covered by one cycle and the horizontal sweep setting. As an example, a sine-wave generator is set to 1000 Hz with the voltage applied to the oscilloscope vertical. One cycle covers 9.95 cm, with a sweep speed of

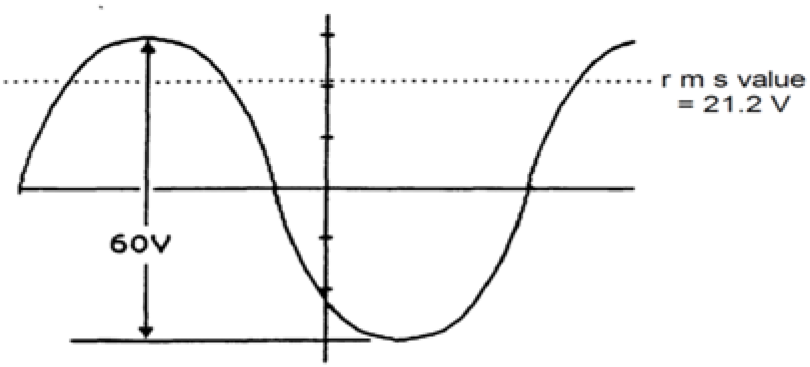

Figure C.2: Sinusoidal waveform

100 µsec/cm. The period is T = (9.95) · (100x10−6)sec. The measured frequency is f = 1/T = 1005 Hz.

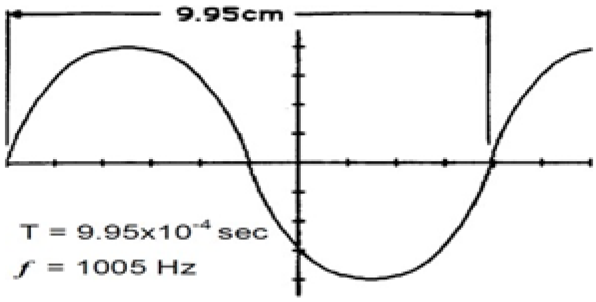

Figure C.3: Frequency measurement

Phase-Angle Measurement
The difference in phase angle between two waveforms may be measured directly on the oscilloscope with little difficulty. For an oscilloscope with only one vertical input: One wave is chosen as the reverence and applied to the vertical input terminals. This same wave is applied to the external trigger input of the scope. Next, a convenient point on the wave is selected as a time reference, such as where the wave is zero and about to swing positive. Then, this waveform is removed from the vertical input and a second waveform is applied. The voltage of this wave at the time reference is observed. The ratio of voltage at the time reference to the maximum voltage is equal to the sine of the phase difference between the two waves.
For an oscilloscope with two vertical inputs:
Connect the reference voltage to Channel 1 of the oscilloscope and connect the second voltage to Channel 2. Adjust the amplitudes so the overlapping signals look something like the figure below, where the solid curve is Channel 1 (the reference) and the dashed curve is Channel 2. (In this figure, the dashed curve is lagging the solid curve; if the dashed curve were shifted to the left so it "started" before the solid curve, then the dashed curve would be leading the solid curve.) The phase shift in degrees can be calculated by the following formula: where is the phase shift, f is the frequency, and t is the time difference between the two waveforms. Many new digital scopes have cursors that allow directly marking, calculating, and displaying the time difference and perhaps even the phase shift.

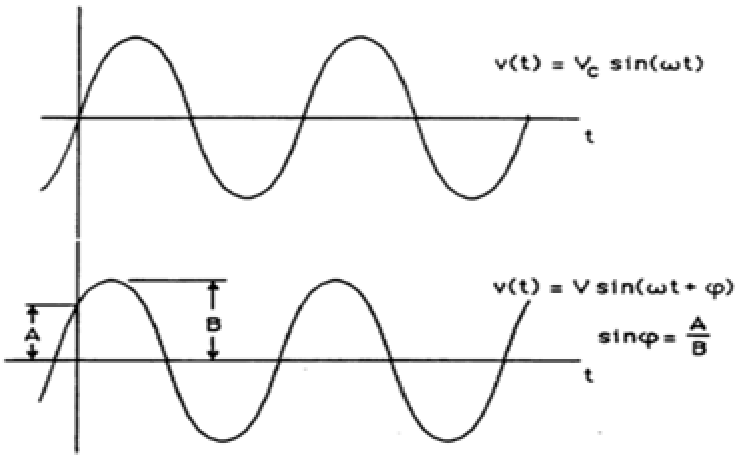

Figure C.4: Phase measurement

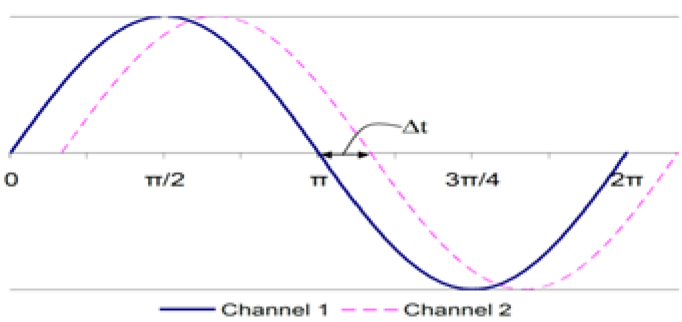

Figure C.5: Frequency measurement