
Chipsmall
深圳市津利帝科技有限公司
Geelead Technology Limited
Chipsmall
采购部

Analog System Lab Kit PRO

MANUAL

Texas Instruments

TEXAS INSTRUMENTS

Table of contents

Table of contents

Experiment 5: 35

Design of a self-tuned filter

Experiment 6: 39

Design a function generator and convert it to Voltage-Controlled Oscillator/FM Generator

Experiment 7: 43

Design of a Phase Lock Loop (PLL)

Experiment 8: 47

Automatic Gain Control (AGC) Automatic Volume Control (AVC)

Experiment 9: 51

DC-DC Converter

Experiment 10: 55

Design a Low Dropout (LDO) regulator

Table of contents

Table of contents

List of figures

List of figures

List of figures

List of tables

Introduction

What you need to know before you get started

Analog System Lab

Although digital signal processing is the most common form of processing signals, analog signal processing cannot be completely avoided since the real world is analog in nature. Consider a typical signal chain (Figure below).

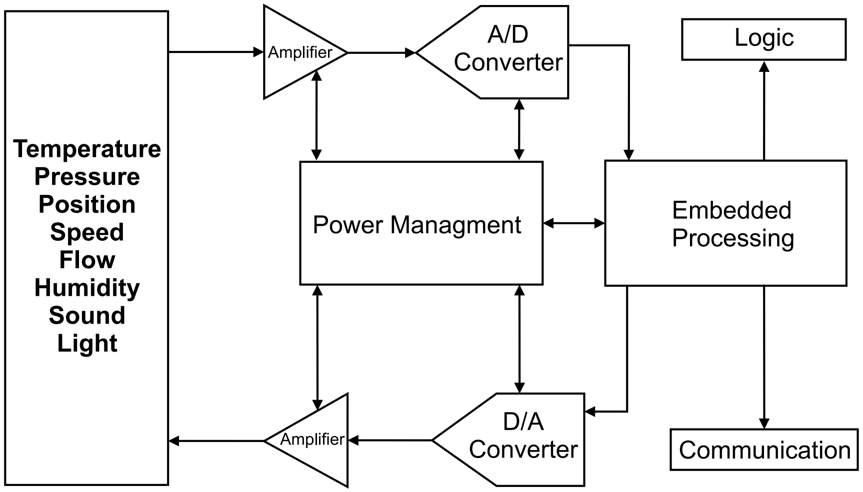


Figure: Signal Chain in an Electronic System

Typical signal chain

① A sensor converts the real-world signal into an analog electrical signal. This analog signal is often weak and noisy.

② Amplifiers are needed to strengthen the signal. Analog filtering may be necessary to remove noise from the signal. This "front end" processing improves the signal-to-noise ratio. Three of the most important building blocks used in this stage are (a) Operational Amplifiers, (b) Analog multipliers and (c) Analog Comparators.

③ An analog-to-digital converter transforms the analog signal into a stream of 0s and 1s.

④ The digital data is processed by a CPU, such as a DSP, a microprocessor, or a microcontroller. The choice of the processor depends on how intensive the computation is. A DSP may be necessary when real-time signal processing is needed and the computations are complex. Microprocessors and microcontrollers may suffice in other applications.

⑤ Digital-to-analog conversion (DAC) is necessary to convert the stream of 0s and 1s back into analog form.

⑥ The output of the DAC has to be amplified before the analog signal can drive an external actuator.

It is evident that analog circuits play a crucial role in the implementation of an electronic system.

The goal of the Analog System Lab Course is to provide students an exposure to the fascinating world of analog and mixed-signal signal processing. The course can be adapted for an undergraduate or a postgraduate curriculum. As part of the lab course, the student will build analog systems using analog ICs and study their macro models, characteristics and limitations.
Our philosophy in designing this lab course has been to focus on system design rather than circuit design. We feel that many Analog Design classes in the colleges focus on the circuit design aspect, ignoring the issues encountered in system design. In the real world, a system designer uses the analog ICs as building blocks. The focus of the system designer are to optimize system-level cost, power, and performance. IC manufacturers such as Texas Instruments offer a large number of choices of integrated circuits keeping in mind the diverse requirements of system designers. As a student, you must be aware of these diverse offerings of semiconductors and select the right IC for the right application. We have tried to emphasize this aspect in designing the experiments in this manual.

Organization of the Course

In designing the lab course, we have assumed that there are about 12 during a semester. We have designed 14 experiments which can be carried out either individually or by groups of two students. The experiments in Analog System Lab can be categorized as follows.

Part I - Learning the basics

In the first part, the student will be exposed to the operation of the basic building blocks of analog systems. Most of the experiments in the **Analog System Lab Course** are centered around the following two components.

- The OP-amp **TL082**, a general purpose JFET-input operational amplifier, made by Texas Instruments.
- Wide-bandwidth, precision analog multiplier **MPY634** from Texas Instruments.

Using these components, the student will build gain stages, buffers, instrumentation amplifiers and voltage regulators. These experiments bring out several important issues, such as measurement of gain- bandwidth product, slew-rate, and saturation limits of the operational amplifiers.

Part II - Building analog systems

Part-II concentrates on building analog systems using the blocks mentioned above.

First, we introduce **integrators** and **differentiators** which are essential for implementing filters that can band-limit a signal prior to the sampling process to avoid aliasing errors.

We then introduce the *analog comparator*, which is a mixed-mode device - its input is analog and output is digital. In a comparator, the rise time, fall time, and delay time are important apart from input offset.
A function generator is also a mixed-mode system that uses an integrator and a regenerative comparator as building blocks. The function generator is capable of producing a triangular waveform and square waveform as outputs. It is also useful in Pulse Width Modulation in DC-to-DC converters, switched-mode power supplies, and Class-D power amplifiers.

The analog multiplier, which is a voltage or current controlled amplifier, finds applications in communication circuits in the form of mixer, modulator, demodulator and phase detector. We use the multiplier in building Voltage Controlled Oscillators, Frequency Modulated waveform generators, or Frequency Shift Key waveform generators in modems, Automatic Gain Controllers, Amplitude Stabilized Oscillators, Self-tuned Filters and Frequency Locked Loop using voltage controlled phase generators and VCOs and multiplier as phase detector are built and their lock range and capture range.

In the Analog System Lab, the frequency range of all applications has been restricted to 1-10 kHz, with the following in mind - (a) The macromodels for the ideal device can be used in simulation, (b) A PC can be used in place of an oscilloscope. We have also included an experiment that can help the student use a PC as an oscilloscope. We also suggest an experiment on the development of macromodels for an OP-Amp.

What is our goal?

At the end of Analog System Lab, we believe you will have the following know-how about analog system design.

1. You will learn about the characteristics and specification of analog ICs used in electronic systems.
2. You will learn how to develop a macromodel for an IC based on its terminal characteristics, I/O characteristics, DC-transfer characteristics, frequency response, stability characteristic and sensitivity characteristic.
3. You will be able to make the right choice for an IC for a given application.
4. You will be able to perform basic fault diagnosis of an electronic system.

Lab Setup

The setup for the Analog System Lab is very simple and requires the following.

1. ASLK PRO and the associated Lab Manual from Texas Instruments India - the lab kit comes with required connectors. Refer to *Chapter 1.4* for an overview of the kit.
2. Oscilloscope. We provide an experiment that helps you build a circuit to directly interface analog outputs to an oscilloscope (See *Chapter C*).
3. Dual power supply with the operating voltages of ±10V.
4. Function generators which can operate in the range on 1 to 10 MHz and capable of generating sine, square and triangular waves.
5. A computer with installed circuit simulation software.

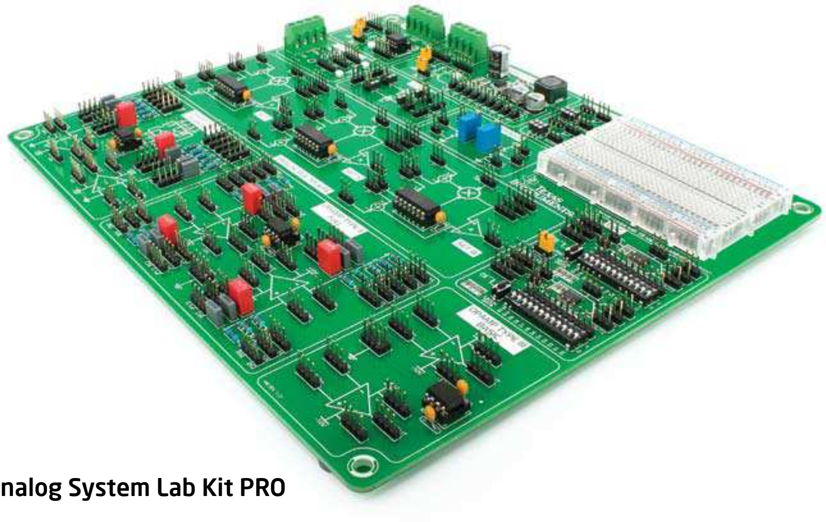
In all the experiments of Analog System Lab, please note the following.

1. When we do not explicitly mention the magnitude and frequency of the input waveform, please use 0 to 1V as the amplitude of the input and 1 kHz as the frequency.
2. Always use sinusoidal input when you plot the frequency response and use square wave input when you plot the transient response.
3. Precaution! Please note that **TL082** is a dual OP-Amp. This means that the IC has two OP-Amp circuits. If your experiment requires only one of the two ICs, do not leave the inputs and output of the other OP- Amp open; instead, place the second OP-Amp in unity-gain mode and ground the inputs.
4. Advisory to Students and Instructors. We strongly advise that the student performs the simulation experiments outside the lab hours. The student must bring a copy of the simulation results to the class and show it to the instructor at the beginning of the class. The lab hours must be utilized only for the hardware experiment and comparing the actual outputs with simulation results.

System Lab Kit overview

Hardware

ASLK PRO has been developed at Texas Instruments India. This kit is designed for undergraduate engineering students to perform analog lab experiments. The main idea behind ASLK PRO is to provide a cost efficient platform or test bed for students to realize almost any analog system using general purpose ICs such as OP-Amps and analog multipliers.



Analog System Lab Kit PRO

ASLK PRO comes with three general-purpose operational amplifiers (**TL082**) and three wide-bandwidth precision analog multipliers (**MPY634**) from Texas Instruments. We have also included two 12-bit parallel-input multiplying digital-to-analog converters **DAC7821**, a wide-input non-synchronous buck-type DC/DC controller **TPS40200**, and a low dropout regulator **TPS7250** from Texas Instruments. A portion of ASLK PRO is left for general-purpose prototyping which can be used for carrying out mini-projects.

The kit has a provision to connect ±10V DC power supply. The kit comes with the necessary short and long connectors.

This comprehensive user manual included with the kit gives complete insight of how to use ASLK PRO. The manual covers exercises of analog system design along with brief theory and simulation results.

Refer to *Appendix A* for the details of the integrated circuits that are included in ASLK PRO. Refer to *Appendix D* for additional details of ASLK PRO.

Software

The following software is necessary to carry out the experiments suggested in this manual.

1. **TINA** or **PSpice** or any powerful simulator based on the SPICE Simulation Engine
2. **FilterPro** - A software program for designing analog filters
3. **SwitcherPro** - A software program for designing power supplies

We will assume that you are familiar with the concept of simulation and are able to simulate a given circuit.

FilterPro is a program for designing active filters. At the time of writing this manual, **FilterPro** Version 3.1 is the latest. It supports the design of different types of filters, namely *Bessel*, *Butterworth*, *Chebychev*, *Gaussian*, and linear-phase filters. The software can be used to design low-pass filters, high-pass filters, band-stop filters, and band-pass filters with up to 10 poles. The software can be downloaded from **[9]**.

Getting to know ASLK PRO

The Analog System Lab kit ASLK PRO is divided into many sections. Refer to the photo of ASLK PRO when you read the following description.

1. **There are three TL082 OP-Amp** ICs labelled 1, 2, 3 on ASLK PRO. Each of these ICs has two amplifiers, which are labelled A and B. Thus 1A and 1B are the two OP-AMps on OP-AMP IC 1, etc. The six OP-amps are categorized as below.

OP-Amp	Type	Purpose
1A	TYPE I	Inverting Configuration only
1B	TYPE I	Inverting Configuration only
2A	TYPE II	Full Configuration
2B	TYPE II	Full Configuration
3A	TYPE III	Basic Configuration
3B	TYPE III	Basic Configuration

Thus, the OP-amps are marked TYPE I, TYPE II and TYPE III on the board. The OP-Amps marked TYPE I can be connected in the inverting configuration only. With the help of connectors, either resistors or capacitors can be used in the feedback loop of the amplifier. There are two such TYPE I amplifiers. There are two TYPE II amplifiers which can be configured to act as inverting or non-inverting. Finally, we have two TYPE III amplifiers which can be used as voltage buffers.

2. **Three analog multipliers** are included in the kit. These are wide-bandwidth precision analog multipliers from Texas Instruments (**MPY634**). Each multiplier is a 14-pin IC and operates on internally provided ±10V supply.

3. **There are two digital-to-analog converters (DAC)** provided in the kit, labeled **DAC I** and **DAC II**. Both the DACs are **DAC7821** from Texas Instruments. They are 12-bit, parallel-input multiplying DACs which can be used in place of analog multipliers in circuits like AGC/AVC. Ground and power supplies are provided internally to the DAC. **DAC Logic Supply Jumper** can be used to connect logic power supplies of both **DAC I** and **DAC II** to either **LDO** or **DC/DC** converter located on the board. Using **Tri-state switches** you can set 12-bits of input data for each DAC to desired value. Click the **Latch Data button** to trigger Digital-to-analog conversion.

4. We have included a **wide-input non-synchronous DC/DC buck converter TPS40200** from Texas Instruments on ASLK PRO. The converter provides an output of 3.3V over a wide input range of 5.5-15V at output currents ranging from 0.125A to 2.5A. Using **Vout SEL jumper** you can select output voltage to be either 5V or 3.3V. Another jumper allows you to select whether input voltage is provided from the board (+10V), or externally using screw terminals.

5. We have included **two transistor sockets** on the board, which are needed in designing an LDO regulator (*Experiment 10*), or custom experiments.

6. **A specialized LDO regulator IC** (**TPS7250**) has been included on the board, which can provide a constant output voltage for input voltage ranging from 5.5V to 11V. Ground connection is internally provided to the IC. Using **ON/OFF jumper** you can enable or disable LDO IC. Another jumper allows you to select whether input voltage is provided from the board (+10V), or externally using screw terminals.

7. **There are two 1kΩ trimmers** (potentiometer) in the kit to enable the designer to obtain a variable voltage if needed for a circuit. The potentiometers are labeled **P1** and **P2**. These operate respectively in the range 0V to +10V, and -10V to 0V.

8. The kit has a **screw terminals to connect ±10V power supply**. All the ICs on the board are internally connected to power supply. Please refer to *Appendix D* for schematics of ASLK PRO.

9. We have included **two diode sockets** on the board, which can be used as rectifiers in custom laboratory experiments.

10. The top right portion of the kit is a **general-purpose area which can be used as a proto-board**. ± 10V points and GND are provided for this area.

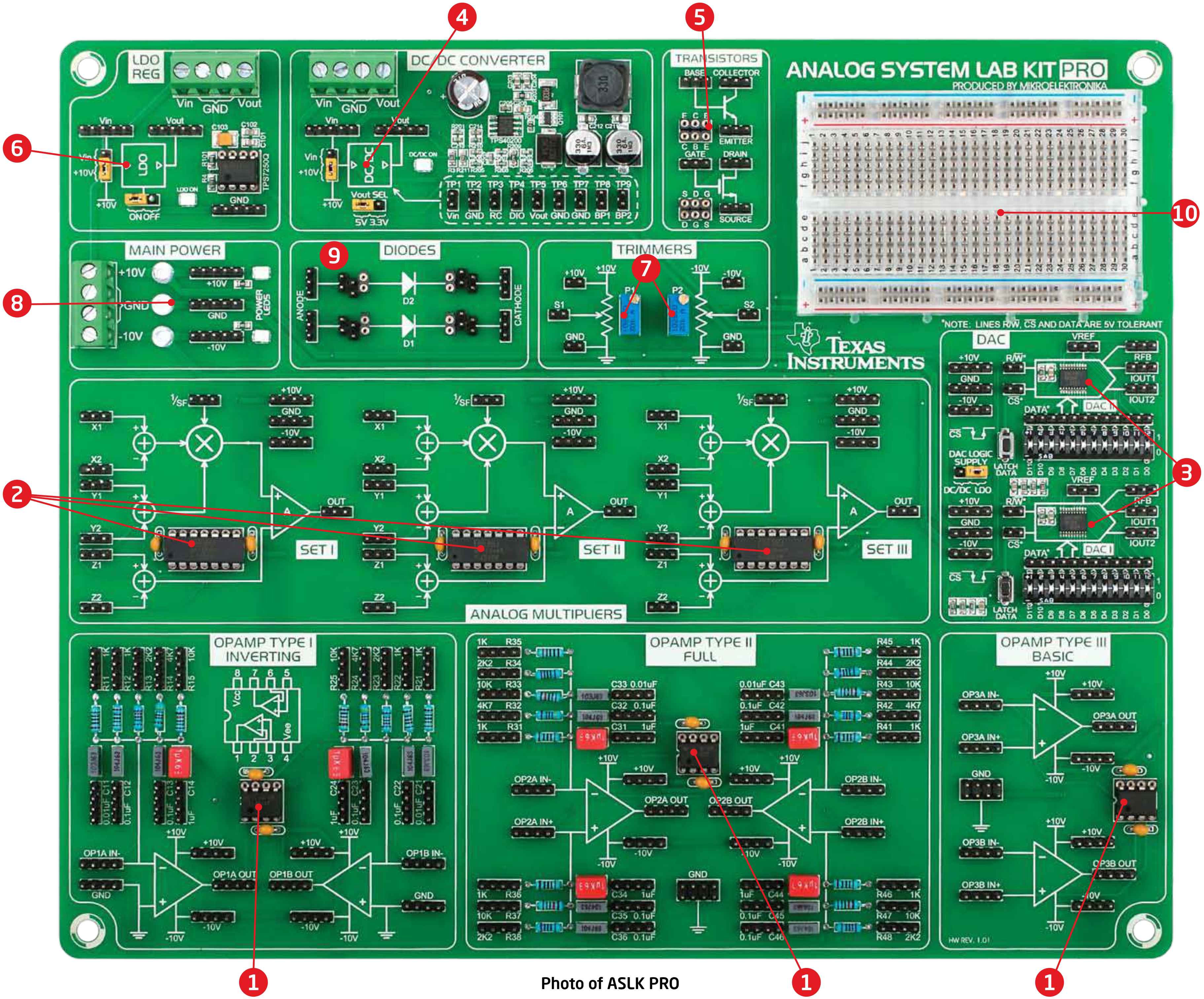


Photo of ASLK PRO

Organization of the Manual

There are *14 experiments* in this manual and the next *14 chapters* are devoted to them, We recommend that in the first cycle of experiments, the instructor introduces the ASLK PRO and ensure that all the students are familiar with a simulation software. A warm-up exercise can be included, where the students are asked to use the simulation software. For each of the experiments, we have clarified the goal of the experiment and provided the theoretical background. The Analog System Lab can be conducted parallel to a theory course on Analog Design or as a separate lab that follows a theory course.

The student should have the following skills to pursue Analog System Lab:

1. Basic understanding of electronic circuits
2. Basic computer skills required to run the simulation tools
3. Ability to use the oscilloscope
4. Concepts of gain, bandwidth, transfer function, filters, regulators and wave shaping

Chapter 1

Experiment 1

Study the characteristics of negative feedback amplifiers and design of an instrumentation amplifier

Goal of the experiment

The goal of this experiment is two-fold. In the first part, we will understand the application of negative feedback in designing amplifiers. In the second part, we will build an instrumentation amplifier.

1.1 Brief theory and motivation

1.1.1 Unity Gain Amplifier

An OP-Amp **[8]** can be used in negative feedback mode to build unity gain amplifiers, non-inverting amplifiers and inverting amplifiers. While an ideal OP-Amp is assumed to have infinite open-loop gain and infinite bandwidth, real OP-Amps have finite numbers for these parameters. Therefore, it is important to understand some limitations of real OP-Amps, such as finite Gain-Bandwidth Product (GB). Similarly, the slew rate and saturation limits of an operational amplifier are equally important. Given an OP-amp, how do we measure these parameters?

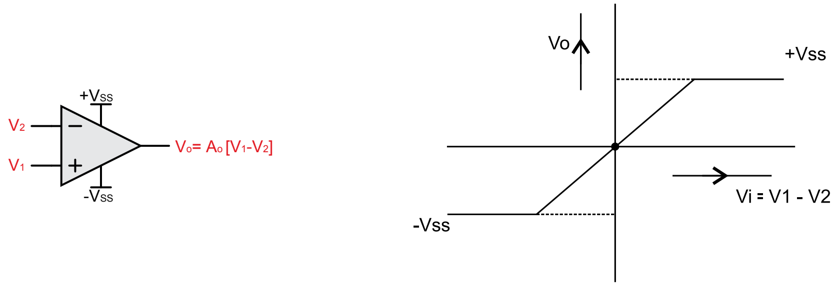


Figure 1.1: An ideal Dual-Input, Single-Output OP-Amp and its I-O characteristic

Since the frequency and transient response of an amplifier are impacted by these parameters, we can measure the parameters if we have the frequency and transient response of the amplifier; you can obtain these response characteristics by applying sinusoidal and square wave inputs respectively. We invite the reader to view the recorded lecture **[16]**.

An OP-Amp can be considered as a Voltage Controlled Voltage Source (VCVS) with the voltage gain tending towards infinity. For finite output voltage, the input voltage is practically zero. This is the basic theory of OP-Amp in the negative feedback configuration. Figure 1.1 shows a differential-input, single-ended-output OP-Amp which uses dual supply $\pm Vss$ for biasing.

$$V_0 = A_0 \cdot (V_1 - V_2) \tag{1.1}$$

$$V_1 - V_2 = \frac{V_0}{A_0} \tag{1.2}$$

In the above equations, A_0 is the open-loop gain; for real amplifiers, A_0 is in the range 10^5 to 10^6 and hence $V_1 \approx V_2$. A unity feedback circuit is shown in the Figure 1.2. It is easy to see that,

$$\frac{V_0}{V_s} = \frac{A_0}{1 + A_0} \tag{1.3}$$

$$\frac{V_0}{V_s} \rightarrow 1 \; as \; A_0 \rightarrow \infty \tag{1.4}$$

In OP-amps, closed loop gain A is frequency dependent, as shown in the equation below, where ω_{d1} and ω_{d2} are called the dominant poles of the OP-amp. This transfer function is typical OP-Amp that has *internal frequency compensation*. Please view the recorded lecture **[17]** to get to know more about frequency compensation.

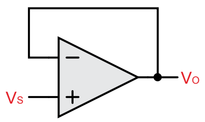


Figure 1.2: A Unity Gain System

$$A = \frac{A_0}{(1 + s/\omega_{d1})(1 + s/\omega_{d2})} \tag{1.5}$$

We can now write the transfer function T for a unity-gain amplifier as,

$$T = \frac{1}{1 + 1/A} \tag{1.6}$$

$$= \frac{1}{(1 + 1/A_0 + s/A_0\omega_{d1} + s/A_0\omega_{d2} + s^2/A_0\omega_{d1}\omega_{d2})}$$

$$= \frac{1}{(1 + (s/GB + s/A_0\omega_{d2} + s^2/GB \cdot \omega_{d2}))} \tag{1.7}$$

The term $GB = A_0\omega_{d1}$, also known as the gain bandwidth product of the operational amplifier, is one of the most important parameters in OP-Amp negative feedback circuit. The above transfer function can be rewritten as

$$T = \frac{1}{1 + s/\omega_0 Q + s^2/\omega_0^2}$$

where

$$Q = \frac{1}{\sqrt{\frac{\omega_{d2}}{GB}} + \frac{1}{A_0}\sqrt{\frac{GB}{\omega_{d2}}}}$$

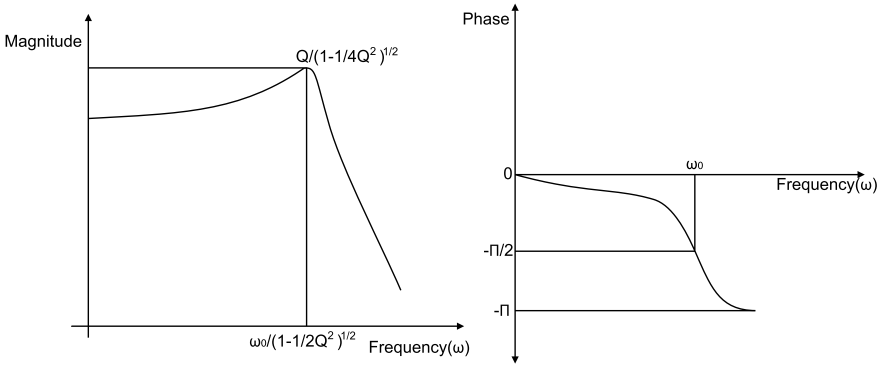


Figure 1.3: Magnitude and Phase response of a Unity Gain System

and

$$\omega_0 = \sqrt{GB \cdot \omega_{d2}}$$

Q is the quality factor and $\xi = \frac{1}{2Q}$ is the damping factor, and ω_0 is the natural frequency of the system. When the frequency response is plotted with magnitude vs ω/ω_0 and phase vs ω/ω_0, it appears as shown in Figure 1.3.

If one applies a step of peak voltage V_p to the unity gain amplifier, and if $V_p \cdot GB <$ slew rate, then the output appears as shown in Figure 2.4 if $Q > \frac{1}{2}$ or $\xi < 1$.

Q is approximately equal to the total number of visible peaks in the step response and the frequency of ringing is $\frac{\omega_0}{(1-1/4Q^2)}$.

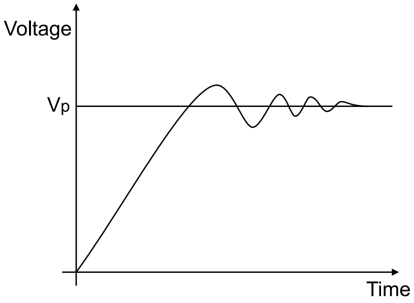


Figure 1.4: Time Response of an Amplifier for a step input of size Vp

Slew-rate is known as the maximum rate at which the output of the OP-Amps is capable of rising; in other words, slew rate is the maximum value that dVo/dt can attain. In this experiment, as we go on increasing the amplitude of the step input, at some amplitude the rate at which the output starts rising remains constant and no longer increases with the peak voltage of input; this rate is called slew rate. It can therefore be determined by applying a square wave of Vp at certain high frequency and increasing the magnitude of the input.

Figure 1.5: (a) Non-inverting amplifier of gain 2, (b) Inverting amplifier of gain 2

1.1.2 Non-inverting Amplifier

A non-inverting amplifier with a gain of 2 is shown in Figure 1.5 (a).

1.1.3 Inverting Amplifier

An inverting amplifier with a gain of 2 is shown in Figure 1.5 (b).

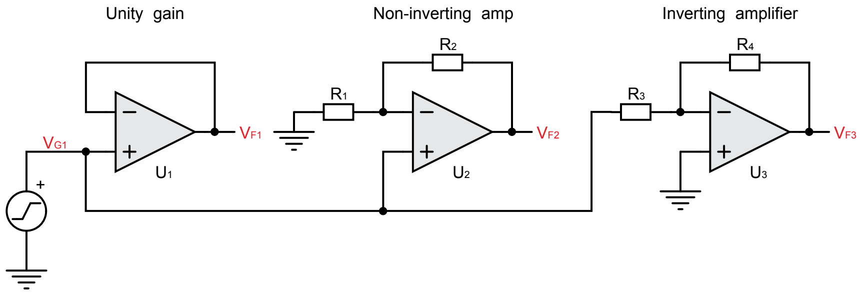


Figure 1.6: Negative Feedback Amplifiers

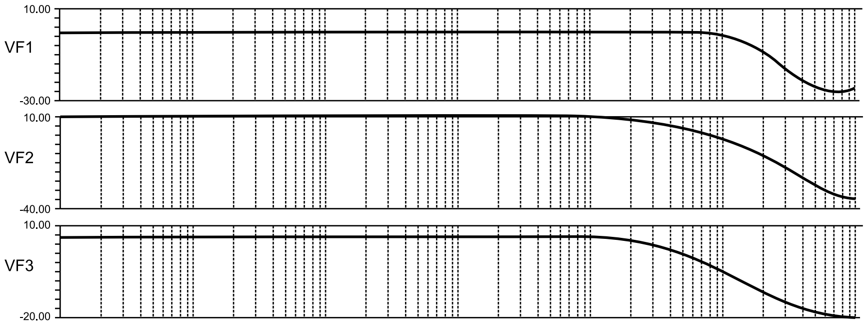
Figure 1.6 shows all the three negative feedback amplifier configurations. Figure 1.7 illustrates the frequency response (magnitude and phase) of the three different negative feedback amplifier topologies. Figure 1.8 shows the output of the three types of amplifiers for a square-wave input, illustrating the limitations due to slew-rate.

1.2 Exercise Set 1

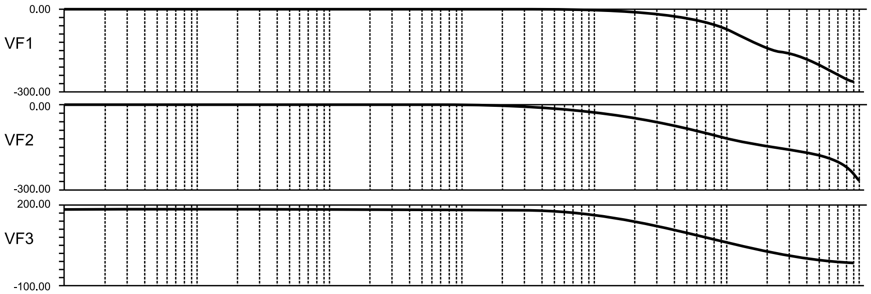
1. Design the following amplifiers - (a) a unity gain amplifier, (b) a non-inverting amplifier with a gain of 2 (Figure 1.5(a)) and an inverting amplifier with the gain of 2.2 (Figure 1.5(b)).

2. Design an instrumentation amplifier using three OP-Amps with a controllable differential-mode gain of 3. Refer to Figure 1.9(a) for the circuit diagram. Assume that the resistors have 1% tolerance and determine the Common Mode Rejection Ratio (CMRR) of the setup and estimate its bandwidth. We invite the reader to view the recorded lecture **[18]**.

3. Design an instrumentation amplifier using two OP-Amps with a controllable differential-mode gain of 5. Refer to Figure 1.9 for the circuit diagrams of the instrumentation amplifiers and determine the values of the resistors. Assume that the resistors have 1% tolerance and determine the CMRR of the setup and estimate its bandwidth.



(A) MAGNITUDE RESPONSE



(B) PHASE RESPONSE

Figure 1.7: Frequency Response of Negative Feedback Amplifiers

1.3 Measurements to be taken

1. Transient response - Apply a square wave of fixed magnitude and study the effect of slew rate on unity gain, inverting and non-inverting amplifiers.

2. Frequency Response - Obtain the gain bandwidth product of the unity gain amplifier, the inverting amplifier and the non-inverting amplifier from the frequency response.

3. DC Transfer Characteristics - Study the saturation limits for an OP-Amp.

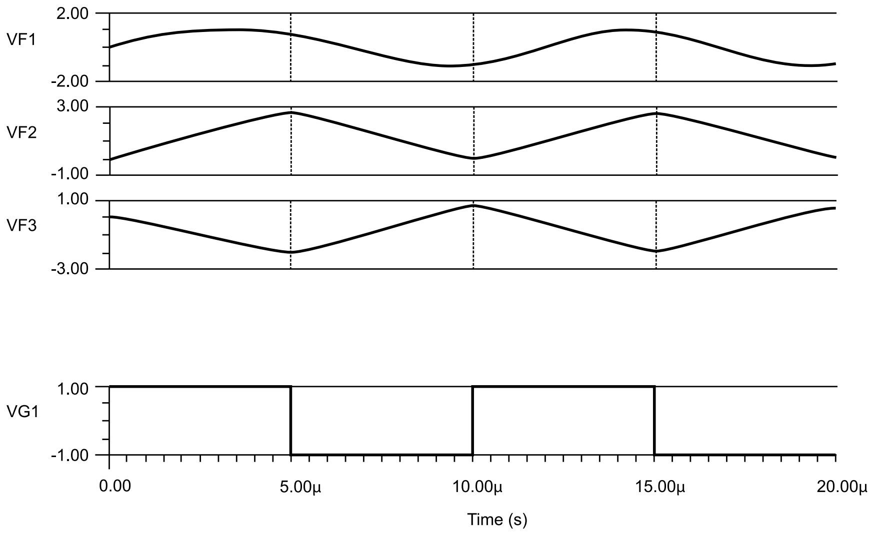


Figure 1.8: Outputs VF1, VF2 and VF3 of Negative Feedback Amplifiers of Figure 1.6 for Square-wave Input VG1

4. Determine the second pole of an OP-Amp and develop the macromodel for the given OP-Amp IC TL082. See Appendix B for an introduction to the topic of analog macromodels.

1.4 What should you submit

1. Submit the simulation results for Transient response, Frequency response and DC transfer characteristics.
2. Take the plots of Transient response, Frequency response and DC transfer characteristics from the oscilloscope and compare it with your simulation results.
3. Apply square wave of amplitude 1V at the input. Change the input frequency and study the peak to peak amplitude of the output. Take the readings in Table 1.1 and compute the slew-rate.

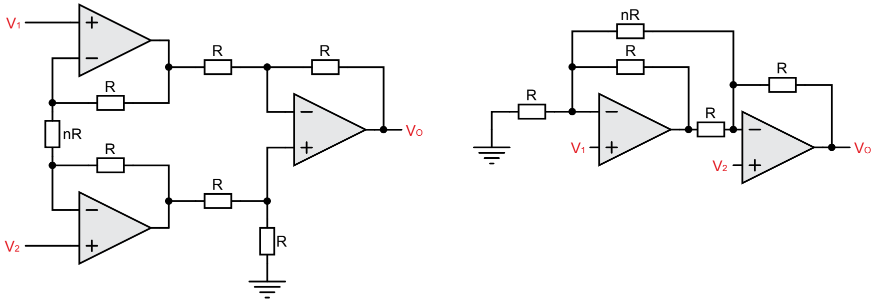


Figure 1.9: Instrumentation Amplifiers with (a) three and (b) two operational amplifiers

S. No.	Input Frequency	Peak to Peak Amplitude of output (Vpp)
1		
2		
3		
4		

Table 1.1: Plot of Peak to Peak amplitude of output Vpp w.r.t. Input frequency

4. Frequency Response - Apply sine wave input to the system and study the magnitude and phase response. Take your readings in Table 1.2.
5. DC transfer Characteristics - Vary the DC input voltage and study its effect on the output voltage. Take your readings in Table 1.3.

1.5 Other related ICs

Specific ICs from Texas Instruments which can be used as instrumentation Amplifiers are INA114, INA118 and INA128. Additional ICs from Texas Instruments which can be used as general purpose OP-Amps are OPA703, OPA357, etc. See CHAPTER 2, EXPERIMENT 1.

S. No.	Input Frequency	Magnitude Variation	Phase Variation
1			
2			
3			
4			

Table 1.2: Plot of Magnitude and Phase variation w.r.t. Input Frequency

S. No.	DC Input Voltage	DC Output Voltage	Phase Variation
1			
2			
3			
4			

Table 1.3: Plot of DC output voltage and phase variation w.r.t. DC input voltage

Further Reading

Datasheets of all these ICs are available at http://www.ti.com.
An excellent reference about operational amplifiers is the "Handbook of Operational Amplifier Applications" by Carter and Brown **[5]**.

Notes on Experiment 1:

Chapter 2

Experiment 2

Study the characteristics of regenerative feedback system with extension to design an astable and monostable multivibrator

Goal of the experiment

The goal of this experiment is to understand the basics of hysteresis and the need of hysteresis in the switching circuits.

2.1 Brief theory and motivation

2.1.1 Inverting Regenerative Comparator

In the earlier experiment we had discussed the use of only negative feedback. Let us now introduce the case of regenerative positive feedback as shown in Figure 2.1. The reader will benefit by listening to the recorded lecture at **[20]**.

$$V_0 = -A_0\cdot\left(V_i - \beta V_0\right) \tag{2.1}$$

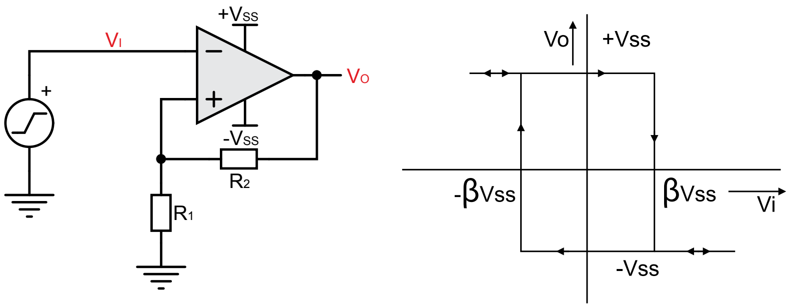


Figure 2.1: Inverting Schmitt-Trigger and its Hysteresis Characteristic

$$\frac{V_0}{V_i} = -A_0\cdot\frac{1}{1 - A_0\cdot\beta} \tag{2.2}$$

$$\beta = \frac{R_1}{R_1 + R_2} \tag{2.3}$$

However, when $\left|A_0\cdot\beta\right| = 1$, it becomes unstable as amplifier as output saturates. When $\left|A_0\cdot\beta\right| \gg 1$ the region of operation of this circuit is regenerative comparator. This is the mixed-mode circuit. Output is stable only in two stages $+V_{ss}$ and $-V_{ss}$. When the input is large negative value output saturates at $+V_{ss}$ as input in increased output remain at $+V_{ss}$ until input reaches $\beta\cdot V_{ss}$ at this point it changes to stable state V_{ss}. Now when the input is decreased it can change state only at V_{ss}. Thus hysteresis of $2\cdot\beta\cdot V_{ss}$ is seen around 0. This kind of comparator is a must while driving a MOSFET as a switch in ON-OFF controllers SMPS (Switched Mode Power Supply), pulse width modulators and class-D audio power amplifiers. The symbol for this inverting type Schmitt trigger is shown in Figure 2.2. The non-inverting Schmitt trigger is as shown in Figure 2.3.

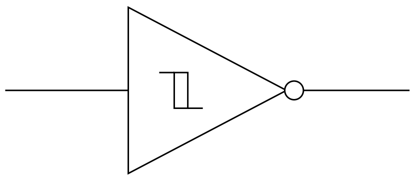


Figure 2.2: Symbol for an Inverting Schmitt Trigger

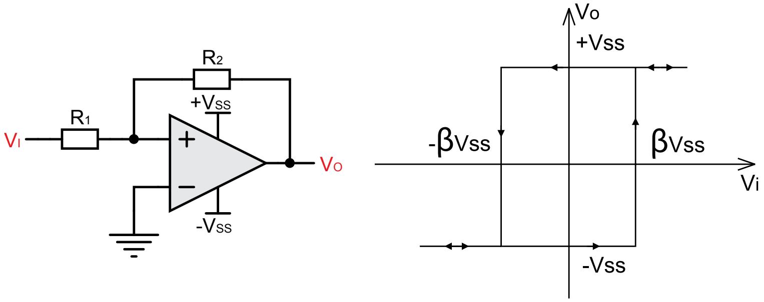


Figure 2.3: Non-inverting Schmitt Trigger and its Hysteresis Curve

2.1.2 Astable Multivibrator

An astable multivibrator is shown in Figure 2.4. The square and the triangular waveforms shown in the figure are both generated using the astable multivibrator. We refer to β as the regenerative feedback. The time period of the multivibrator is given by

$$T = 2\cdot RC\cdot\ln\left(\frac{1+\beta}{1-\beta}\right) \tag{2.4}$$

2.1.3 Monostable Multivibrator (Timer)

The circuit diagram for a monostable multivibrator is shown in 2.6. The trigger waveform shown in Figure 2.5 is applied to the monostable. The negative edge triggers the monostable, which produces the square waveform shown in Figure 2.6.

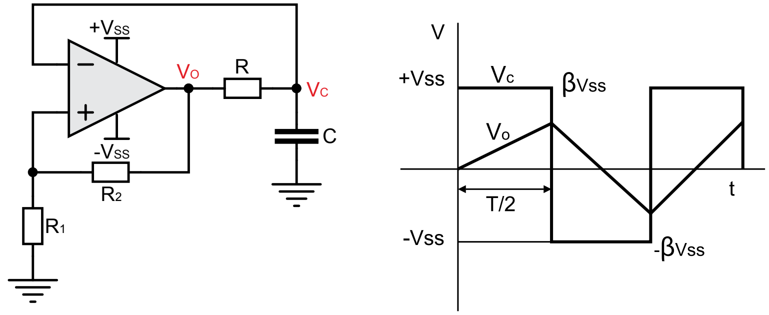


Figure 2.4: Astable Multivibrator and its characteristics

The monostable remains in the "on" state until it is triggered; at this time, the circuit switches to the "off" state for a period equal to τ. The equation for τ is shown below.

$$\tau = RC \cdot \ln\left(\frac{1}{1-\beta}\right) \tag{2.5}$$

After triggering the monostable at time t, the next trigger pulse must be applied after $t+\tau'$. The formula for τ' is given below.

$$\tau' = RC \cdot \ln\left(\frac{1+\beta}{\beta}\right)$$

S. No.	Regenerative Feedback	Hysteresis
1		
2		
3		
4		

Table 2.1: Plot of Hysteresis w.r.t. Regenerative Feedback

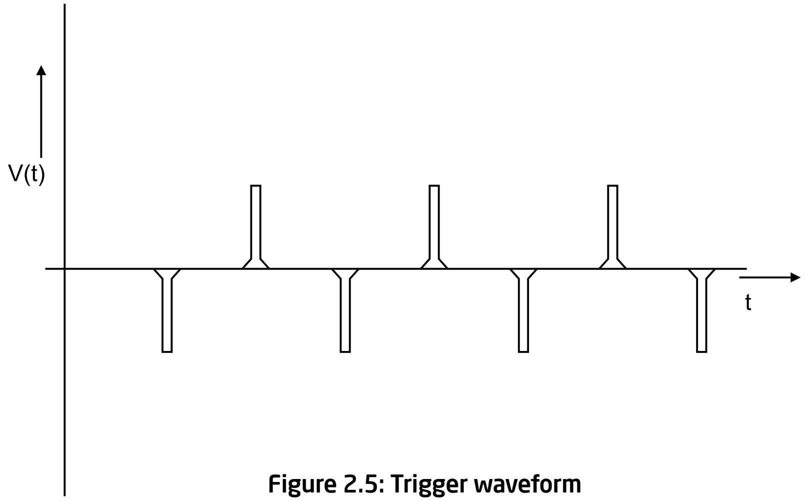


Figure 2.5: Trigger waveform

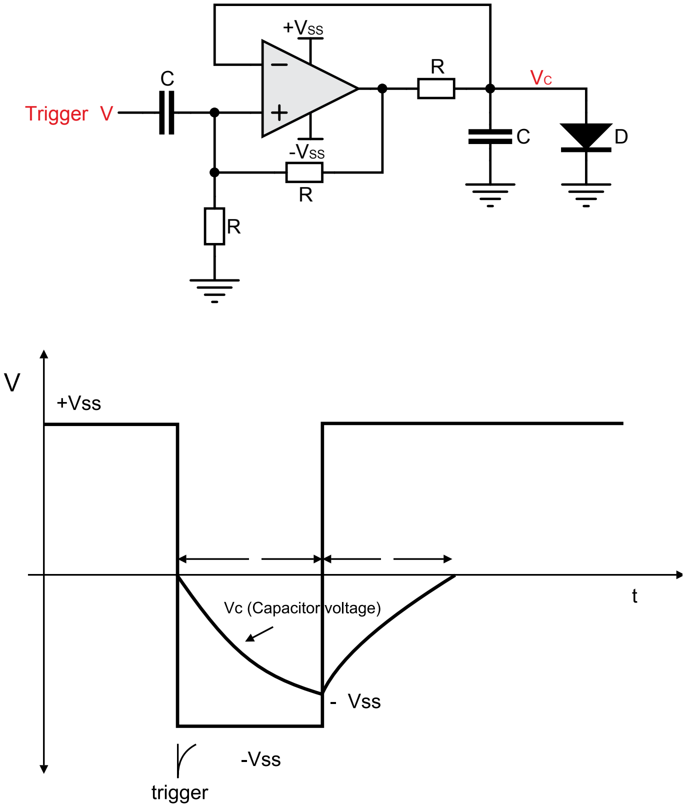


Figure 2.6: Monostable Multivibrator and its outputs