ANALOG SYSTEM LAB MANUAL

Second Edition

Analog System Lab Manual

Second Edition

Learning to Design Analog Systems
using
Analog System Lab Starter Kit

Dr. K.R.K. Rao and Dr. C.P. Ravikumar
Texas Instruments, India
Bagmane Tech Park
CV Raman Nagar
Bangalore 560093
India

Texas Instruments
Wiley

Analog System Lab Manual **Second Edition**

Learning to Design Analog Systems using Analog System Lab Starter Kit

Other Wiley Editorial Offices:

John Wiley & Sons, Inc. 111 River Street, Hoboken, NJ 07030, USA

Wiley-VCH Verlag GmbH, Pappellaee 3, D-69469 Weinheim, Germany

John Wiley & Sons Australia Ltd, 42 McDougall Street, Milton, Queensland 4064, Australia

John Wiley & Sons (Asia) Pte Ltd, 1 Fusionopolis Walk #07-01 Solaris, South Tower, Singapore 138628

John Wiley & Sons Canada Ltd, 22 Worcester Road, Etobicoke, Ontario, Canada, M9W 1L1

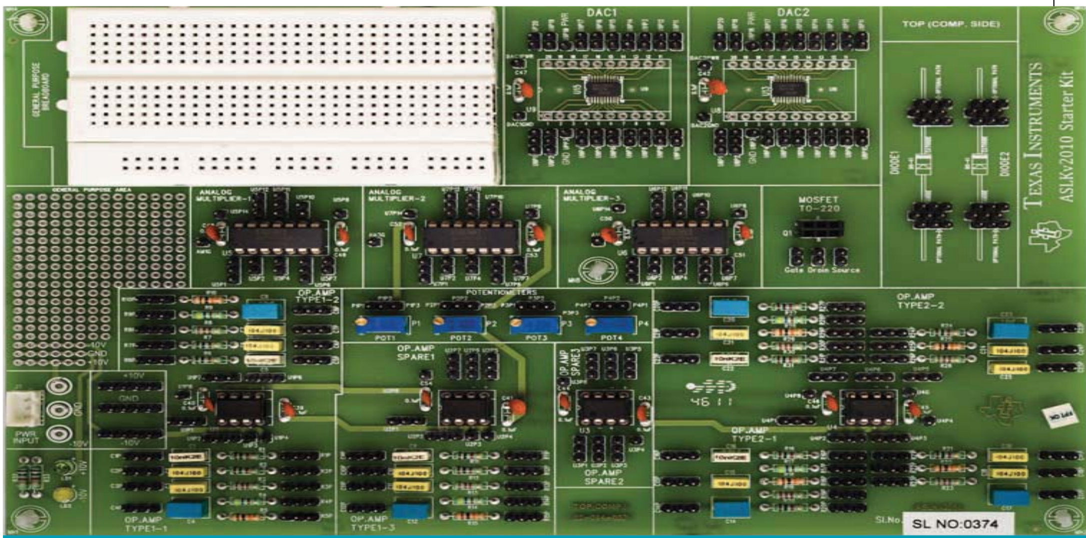
First Edition: 2011

Second Edition: 2012

ISBN: 978-81-265-3742-6

www.wileyindia.com

Printed at: *Paras Printers, Delhi*



Foreword to the Second Edition

The first version of this manual, released in 2011, has been received with great enthusiasm by teachers and students. We thank everyone for this warm reception.

We are happy to place in your hands the revised version of the Analog System Lab Manual. The Analog System Lab Kit and the associated manual were created to help colleges in India in updating their curriculum for courses related to analog. Analog electronics occupies a very special and significant place in modern-day systems. In the past decade, India has seen the emergence of a number of system design companies. Manufacturing of electronic products has also received a significant boost. These companies look for system-level design skills in both analog and digital domains. Unfortunately, analog system design is not emphasized in the conventional way of teaching analog. Our attempt is to help bridge this gap at an early stage in undergraduate coursework. We believe that the ASLK can be adopted by both undergraduate and postgraduate students.

Since ASLK Starter kit was introduced, close to 100 colleges in India have introduced it in their teaching curriculum. We have interacted with hundreds of Indian teachers in the faculty development programs on Analog System Design that were conducted in the last year. Several colleges have independently conducted such hands-on workshops for teachers as well as students. We are encouraged by the acceptance of the kit as an educational tool that is easy to use. We are grateful to all the comments and feedback we have received from academia. Several teachers have told us that they have designed new experiments in the areas of communications and controls. Several companies used ASLK in their in-house training programs. To see the reaction from students, we made it mandatory for participants of TI India Analog Design Contest to carry out a challenging experiment on the ASLK. We could not have expected a more positive response! Some students even surprised us by using ASLK for new experiments such as motor control and simulation of chaos in oscillators! We encourage students and teachers to explore such innovative applications of ASLK. TI has a vast portfolio of analog ICs to select from. Please make use of TI's "free samples" program to carry out additional experiments on ASLK.

Based on the feedback we received, we have introduced a few changes in this version of the manual. We are sure you will like the aesthetic improvements to the manual, including the colored illustrations. A pin diagram of the ASLK is included with the kit to facilitate the connections. We have added several additional exercises in almost all chapters. These additional exercises are marked with a star; we have provided the component values and typical simulation results in these exercises. The starred exercises can be good starting points when one begins to use ASLK. However, we emphasize the importance of the other exercises which involve design. We suggest that a teacher assigns a mix of starred and the other exercises in the lab work.

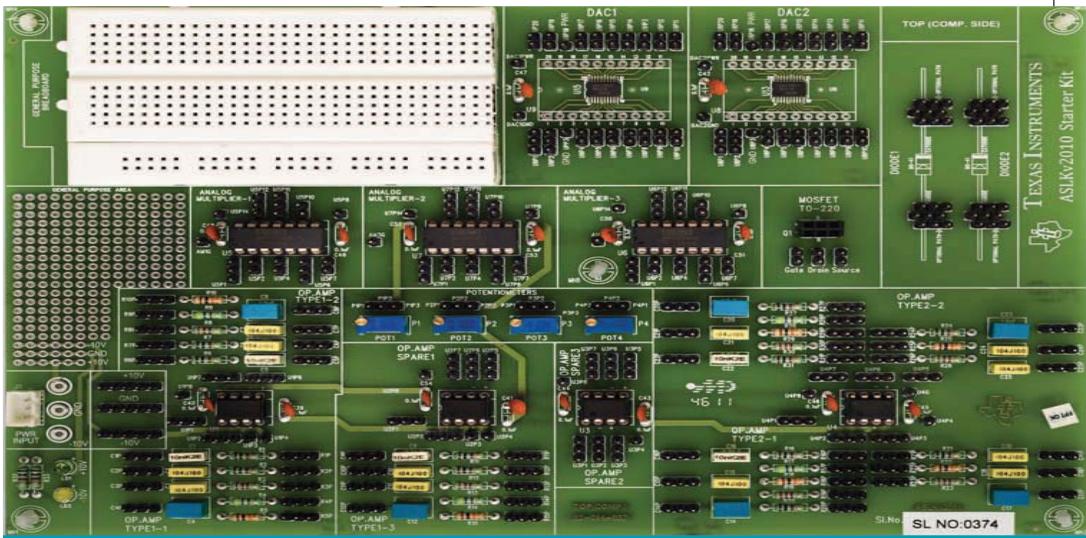
We are pleased to acknowledge the help from several persons in preparing this manual. Ms Meenakshi Sehrawat of Wiley-India has done a creditable job of editing. We thank Praveen Settigere of Wiley-India for his continued support. Joyan Gratian Sanctis of TI India has taken the excellent picture of the ASLK included in this manual. We are indebted to Sagar Juneja for his constant help in all aspects of ASLK promotion.

We thank Cranes Software for their manufacturing and promotion of ASLK. We thank all our colleagues in TI India for their constant support and encouragement.

As always, we are eager to know your feedback!

K.R.K. Rao
C.P. Ravikumar
July 2012

Texas Instruments, India
Bagmane Tech Park
CV Raman Nagar
Bangalore 560093
India



Foreword to the First Edition

Although digital signal processing is the most common way to process signals, analog signal processing cannot be completely avoided since the real world is analog in nature. The goal of the *Analog System Lab* 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 is 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. A designer 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.

We believe that there is a need to make a significant change to the way analog design is taught in the engineering colleges today. The conventional way of starting with device physics and moving on to the design and analysis of analog circuits at the transistor-level needs rethinking. What is proposed is a two-tier approach to teaching analog design – start with analog systems and then move to circuits. Analog systems are part of every electronic system today and we believe they must be taught in the "building block" spirit that has worked well for digital design. There are many system design companies today looking for engineers who can design using analog ICs; they hardly ever design or even use a common emitter amplifier or a Wien-bridge oscillator!

There are 10 experiments in the Analog System Lab, which can be carried out either individually or by groups of two or three students. In the first phase of experiments, two basic analog building blocks are introduced, namely, Operational Amplifiers and Analog Multipliers. In the second phase, we explain how larger analog systems such as integrators, differentiators, filters, function generators, VCO, PLL, DC-DC converters and regulators can be constructed using the basic building blocks. The emphasis is on learning by paper design, simulation through SPICE, hardware construction, and analysis of results. With each experiment, we provide brief theoretical background, references to literature (mostly online and easy to access), the specification of the design experiment, measurements to be taken, and the documents to be submitted at the end of the experiment. A teacher's manual can be made available on request.

This manual is the result of almost a year's effort. We have received support from a number of individuals when we were working on this manual. It is our pleasure to acknowledge their contribution. We acknowledge the encouragement and support from Syd Coppersmith (WW Manager, TI Analog University Program) throughout this endeavor. A number of colleagues at Texas Instruments, India have helped us and encouraged us at different stages of the development of the kit and the manual. Our sincere thanks are due to all of them. Mr Krishnamurthy Bhat of Basaveshwara

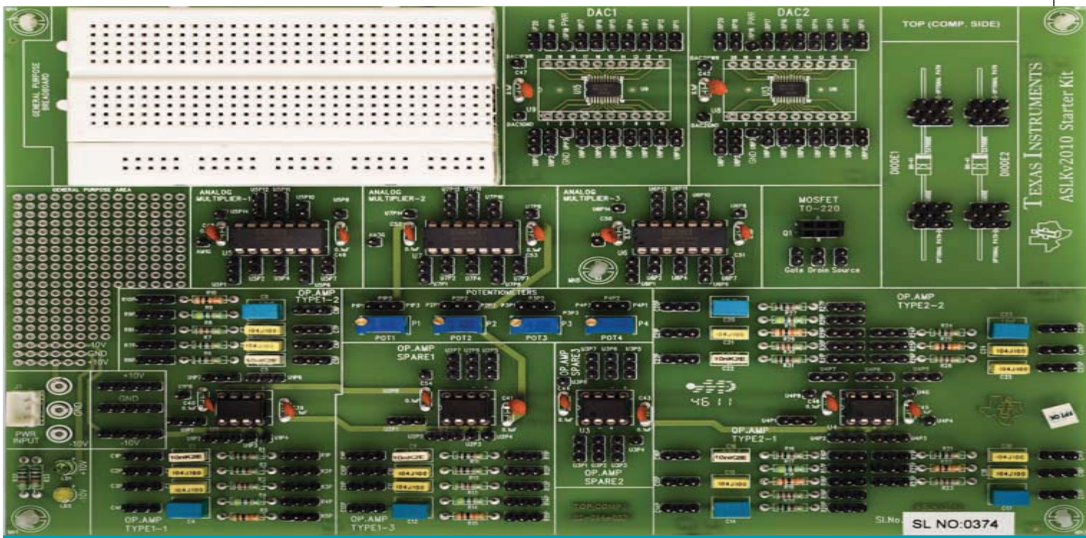
Engineering College (Bagalkot, Karnataka, India) spent several months with us, helping us realize the kit as a product. He was ably helped by Sagar Juneja, then a student intern at TI India. Sagar has also read various drafts of this manual and provided helpful comments. Ullas Taneja, another student intern, helped in recording the video lectures that provide more information on these experiments. Pulkit Jain, also an intern, helped us by drawing many of the diagrams in this manual. We thank the faculty members who attended the faculty development programs where initial drafts of this manual and the Analog System Lab Kit were used; their feedback has been useful in improving the kit as well as the manual.

We thank Mr. E.S. Kannan of YEE YES and Mr. Ashfaq Ibrahim of Cranes Software for their support. We thank Mr. Praveen Settigere of Wiley India for his interest in this project and for all the help he provided in publishing the manual.

We hope you and your students will find the Analog Systems Lab Kit and the experiments in this manual rewarding. We intend to continue to develop more experiments and learning materials in the future; we will share them on the TI India University Program website (**www.uniti.in**). We are eager to know your critique of the kit as well as the manual. Do write to us!

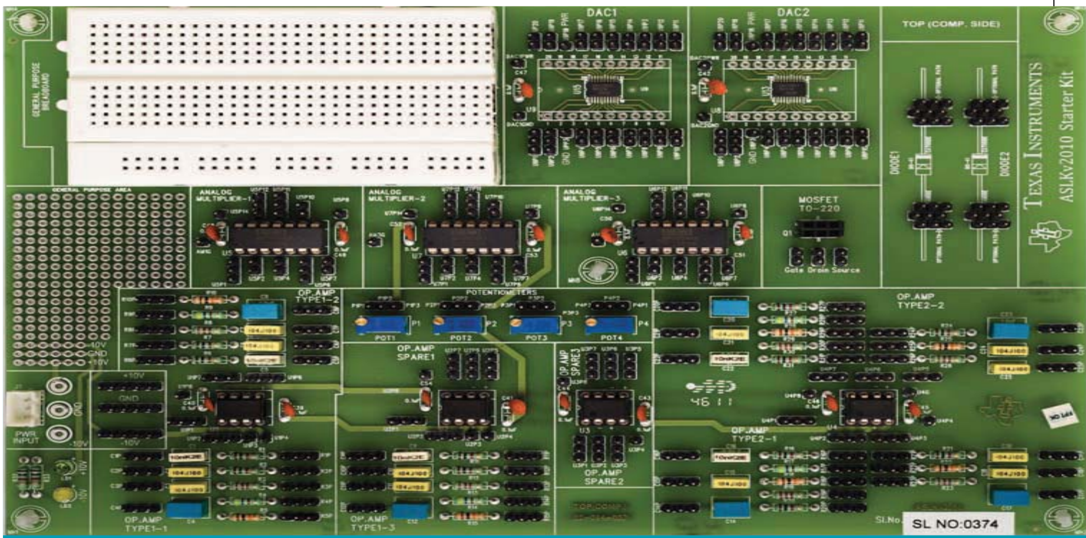
K.R.K. Rao
C.P. Ravikumar
November, 2010

Texas Instruments, India
Bagmane Tech Park
CV Raman Nagar
Bangalore 560093
India

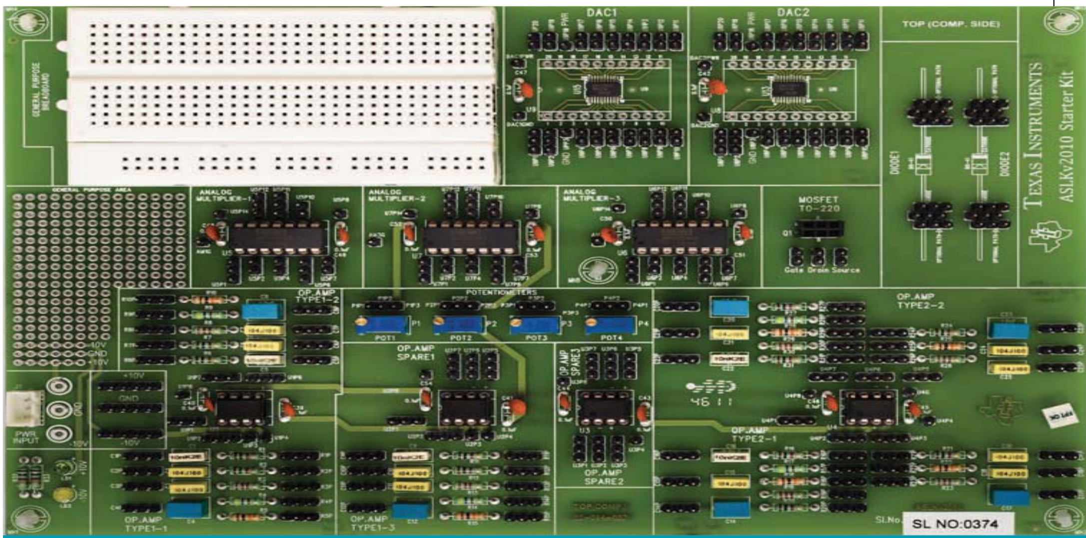


Contents

Contents

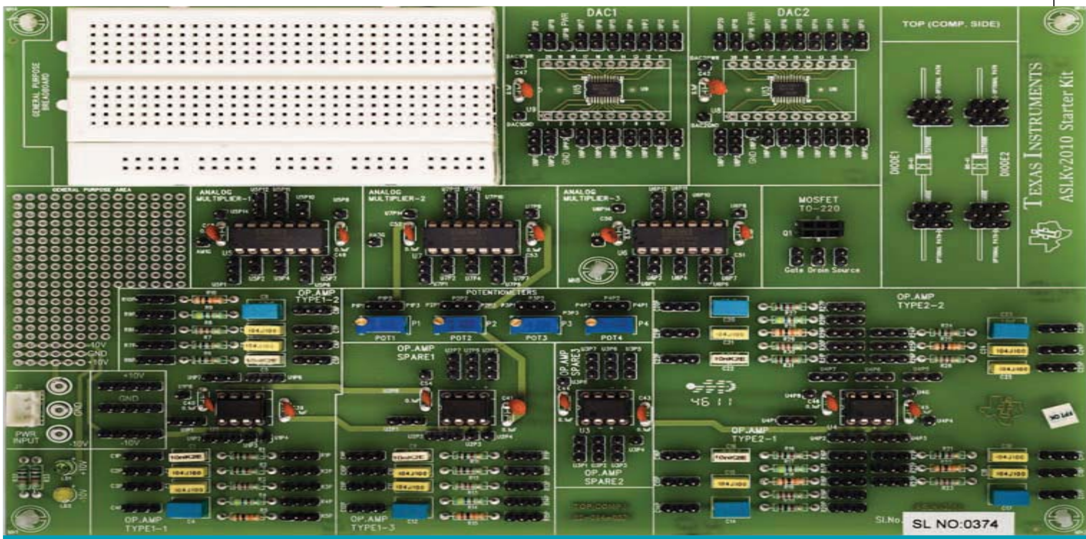


List of Figures



List of Tables

Chapter Zero



Introduction

0.1 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 0-1).

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.

Texas Lab Manual: “tlm” - 2012/8/14 - 14:55 - page 1 - #25

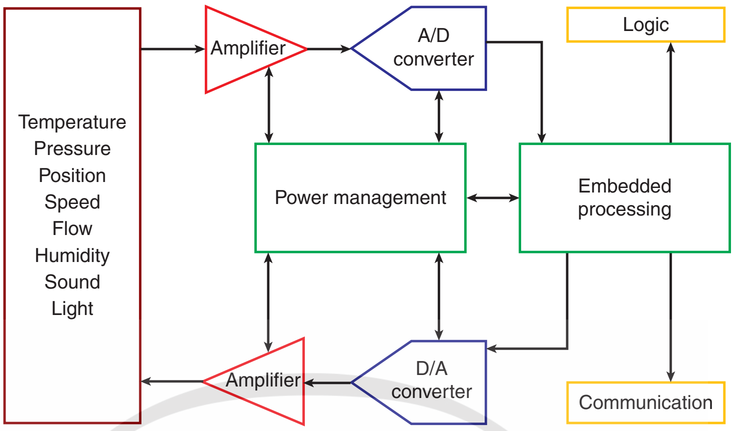


Figure 0-1 Signal chain in an electronic system

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.
7. A *Power Management* block is needed to provide power to the various blocks. In modern-day VLSI chips, power dissipation is a major consideration so that we can keep the power density under control. Since the source of power can be a battery, it is important to ensure long battery life through techniques such as clock gating, power gating, etc. The *Power Management* block is responsible for these functions.

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 can *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 is 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.

0.2 Organization of the Analog System Lab Course

In designing the lab course, we have assumed that there are about 12 lab sessions during a semester. We have designed 10 experiments that can be carried out either individually or by groups of two or three students. The experiments in **Analog System Lab** can be categorized as follows:

1. **Part I:** 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.

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.

2. **Part-II:** The second part concentrates on building analog systems using the blocks mentioned in the previous point.

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

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* (VCO), *Frequency Modulated Waveform Generators*, or *Frequency Shift Key Generators* in modems, *Automatic Gain Controllers*, *Amplitude Stabilized Oscillators*, *Self-tuned Filters* and *Frequency Locked Loop*. Voltage controlled phase generators and VCOs that use multiplier as a phase detector are built and their lock range and capture range estimated and verified.

In the **Analog System Lab**, the frequency range of all applications has been restricted to 1–10 kHz, with the following in mind: (a) Simple macromodels can be used for active devices 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`.

Figure 0-2 shows the dependence among the experiments included in **Analog System Lab**. The sequence in which the experiments are carried out can be altered using this dependence graph. We believe that the students must carry out all the experiments.

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 characteristics and sensitivity characteristics

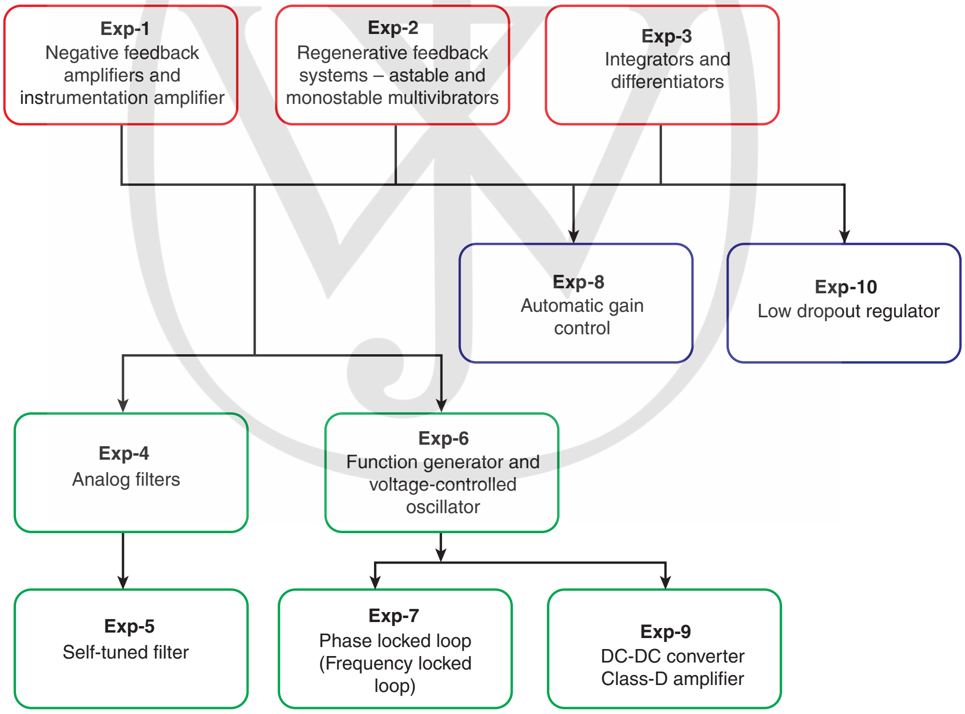


Figure 0-2 Dependence among experiments on the `ASLKv2010 Starter`

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

0.3 Lab Setup

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

1. `ASLKv2010 Starter` kit and the associated Lab Manual from Texas Instruments, India – the lab kit comes with required connectors. Refer to Section 0.4 for an overview of the kit.
2. A low frequency operation oscilloscope which can operate in the frequency range of 1 to 10 MHz. Texas Instruments also offers an oscilloscope card which can be plugged into laptops so that the laptop can work as an oscilloscope (See [30]). Alternately, we also provide an experiment that helps you build a circuit to directly interface analog outputs to a PC (See Appendix C).
3. Dual power supply with the operating voltages of ± 10 V.
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 simulation software such as TINA [9] or PSPICE [32] and design software from Texas Instruments (`FilterPro` and `SwitcherPro`) installed on it.

0.3.1 Important Notes

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 1 V 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. **Note to students:** With every experiment, we have included tables that can be used to record the experimental data that you collect during the experiment. We have shown four blank entries in every table to illustrate the type of data the student must collect; the student must actually record many more data points.
4. **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 `Op-Amp` circuits, 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 input.
5. **Precaution!** Never connect any point from the board to the oscilloscope. Instead, use a probe that is connected to the oscilloscope to investigate different points on the board!
6. **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 from SPICE simulation 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.

0.4 System Lab Kit ASLKv2010 Starter: An Overview

0.4.1 Hardware

`ASLKv2010 Starter` kit (see Figure 0-3; Pin diagram is shown in Figure 0-4) has been developed at Texas Instruments, India. This kit is designed for undergraduate

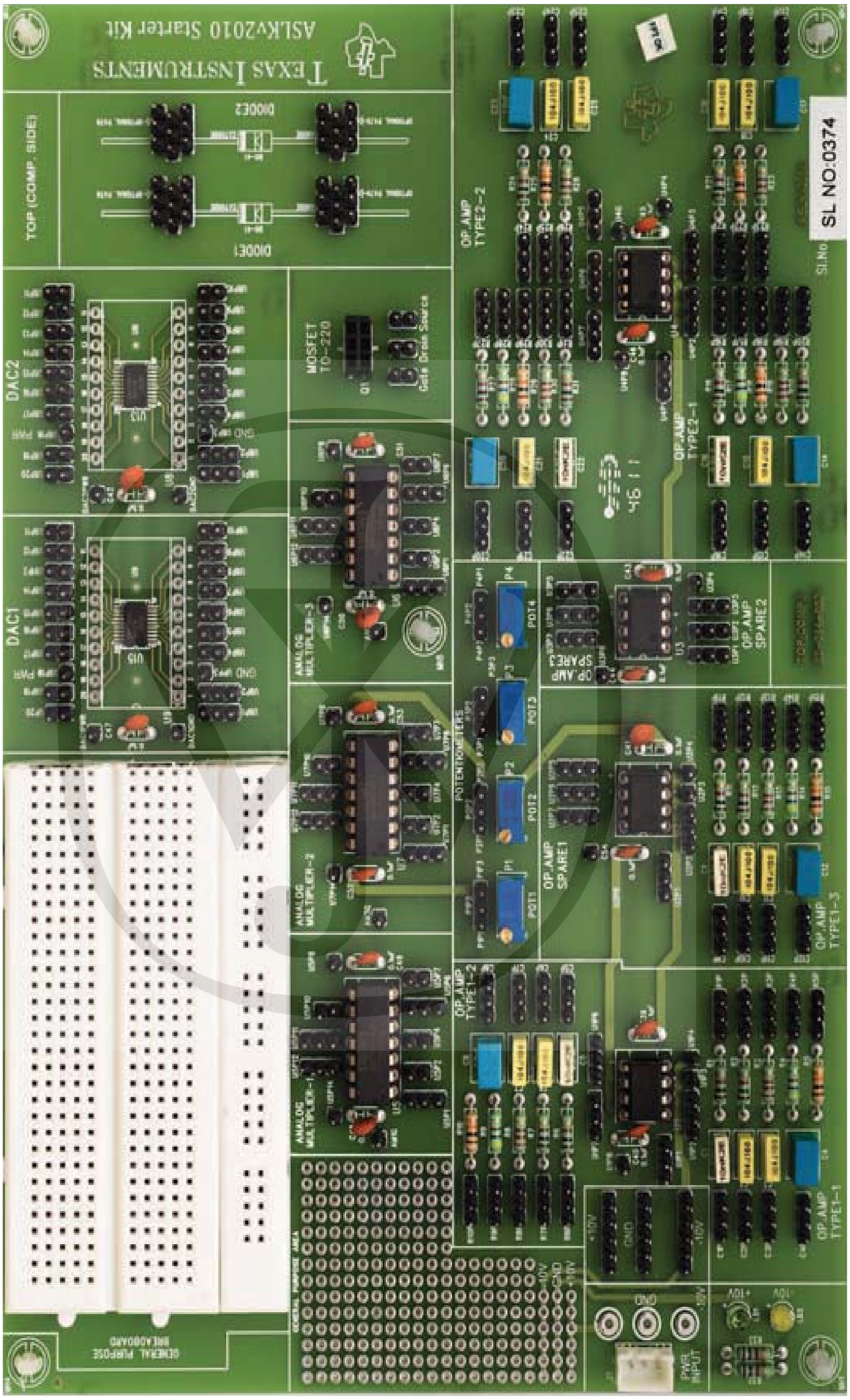


Figure 0-3 Picture of ASLKv2010 Starter kit

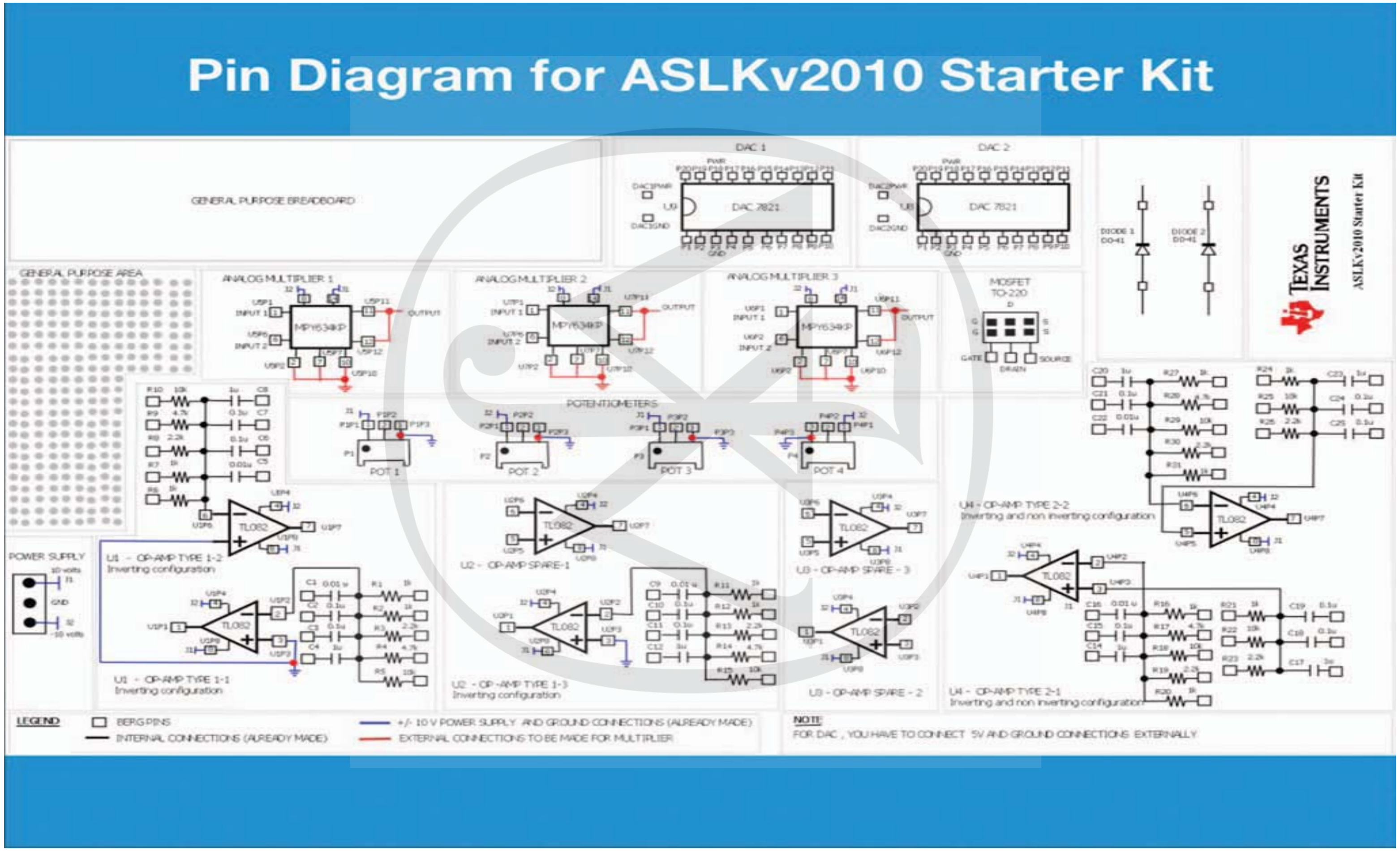


Figure 0-4 Pin diagram for `ASLKv2010 Starter` kit

engineering students to perform analog lab experiments. The main idea behind ASLKv2010 Starter kit 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.

ASLKv2010 Starter kit comes with four general-purpose operational amplifiers (TL082) and three wide-bandwidth precision analog multipliers (MPY634) from Texas Instruments. There is also a provision to include a 12-bit parallel-input multiplying digital-to-analog converter DAC7821. A portion of ASLKv2010 Starter kit is left for general-purpose prototyping and can be used for carrying out mini-projects.

The kit has a provision to connect $\pm$10 V DC power supplies. The kit comes with the necessary short and long connectors, as well as connectors for power supplies. Figure D-2 (Appendix D) shows the way power supply connections are made on ASLKv2010 Starter kit. The $\pm$10 V supplies are connected internally to all the ICs that require $\pm$10 V supplies, namely, the operational amplifiers and the multipliers. Since the DAC requires 5 V supply, the student has three options – to use the output from one of the potmeters to provide 5 V supply, to generate a 5 V supply (see the DC-DC converter experiment in this manual) or to use an external 5 V supply.

This comprehensive user manual included with the kit gives complete insight of how to use ASLKv2010 Starter kit. The manual covers exercises of analog system design along with brief theory and simulation results obtained using simulation software.

Refer to Appendix A for the details of the integrated circuits that are included in ASLKv2010 Starter kit. Refer to Appendix D for additional details of ASLKv2010 Starter kit.

0.4.2 Software

The following softwares are necessary to carry out the experiments suggested in this manual:

1. A SPICE-based simulation software, such as TINA [9], Multisim [15] or PSPICE [32].
2. FilterPro – a software program for designing analog filters.

3. `SwitcherPro` – a software program for designing switched-mode power supplies.
4. `MDACBufferPro` – a software for designing multiplying D/A converters.
5. `ADCPro` – a software for designing A/D converters.
6. `ClockPro` – a software for synthesizing clock generators.

Several SPICE-based simulation software [9, 32] are available today to verify the behavior of circuits before they are implemented. These are powerful and easy-to-use simulators for electronic circuits. It allows the simulation of circuits with passive components such as resistors, capacitors and inductors, as well as active components like transistors and analog integrated circuits. Texas Instruments makes *macromodels* of integrated circuits available for the users of the simulation programs. (Appendix B explains what macromodels are.) We will assume that you are familiar with the concept of simulation, and are able to simulate a given circuit in TINA or PSPICE.

`FilterPro` is a program for designing active filters. At the time of writing this manual, `FilterPro` Version 3.0 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 [10].

0.5 Familiarizing with `ASLKv2010 Starter` Kit

The **Analog System Lab** `ASLKv2010 Starter` kit is divided into many sections. Refer to the picture in Figure 0-5 when you read the following description. Please also refer to the floorplan of the chip shown in Figure D-1 (Appendix D).

Figure D-1 shows the overall floorplan of the `ASLKv2010 Starter` kit. We have shown the power connections in `ASLKv2010 Starter` in Figure D-2. Note that the $\pm$10 V power and ground connections have to be connected to the power inlets at the side of the kit; the power and ground are internally connected to the `Op-Amps` and analog multipliers. If you wish to carry out an experiment using the DAC integrated circuits on the board, you must use an external 5 V supply and ground connection, as shown in Figure D-2. There are three potmeters included in the kit, each of which is connected across 10 V and ground. The output of the potmeter can be used to derive a voltage in the range of 0 to 10 V; this can be useful in generating a reference voltage or even in generating a 5 V power supply for the DAC.

❶ There are four TL082 Op-Amp ICs labeled 1, 2, 3 and 4 on ASLKv2010 Starter kit. Each of these ICs has two amplifiers, which are labeled A and B. Thus, 1A and 1B are the two Op-Amps in the Op-Amp IC 1, etc. The eight Op-Amps are categorized as shown in the following table:

Op-Amp **IC**	Op-Amp	**Label on Kit**	**Type**	**Purpose**
1	1A	TYPE 1-1	TYPE-1	Inverting Configuration only
	1B	TYPE 1-2	TYPE-1	Inverting Configuration only
2	2A	TYPE 1-3	TYPE-1	Inverting Configuration only
	2B	TYPE SPARE-1	SPARE	Spare
3	3A	TYPE SPARE-2	SPARE	Spare
	3B	TYPE SPARE-3	SPARE	Spare
4	4A	TYPE 2-1	TYPE-2	Inverting or Non-inverting
	4B	TYPE 2-2	TYPE-2	Inverting or Non-inverting

Refer to the floorplan of the kit and identify the Op-Amp ICs (Figure D-1). It will also be helpful to refer to the power connections shown in Figure D-2. Please see connection diagrams shown in Figures D-3, D-4, D-5 and D-6. The Op-Amps are marked TYPE 1, TYPE 2 or SPARE on the board. The Op-Amps marked TYPE 1 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 three TYPE 1 amplifiers. There are three spare Op-Amps and two TYPE-2 amplifiers. TYPE-2 amplifiers can be connected in inverting or non-inverting configurations. All the Op-Amps ICs operate on ±10 V; the power supply and ground connections are internally provided and the user need not worry about these.

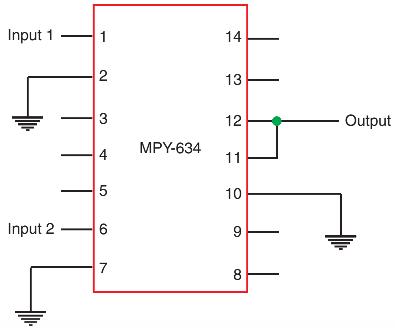


Figure 0-5 External connections needed for using the analog multiplier

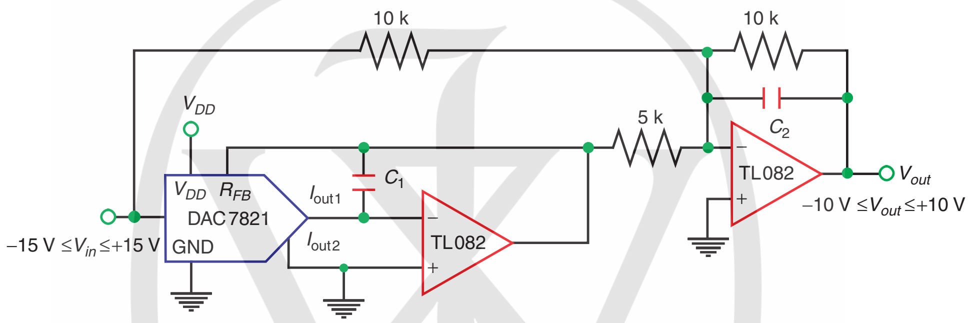


Figure 0-6 External connections needed for using the DAC

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 $\pm$10 V supply. The power supply connections for the multipliers are provided internally. In order to use the analog multiplier IC on the `ASLKv2010 Starter` kit, the external connections shown in Figure 0-5 are required.

3. Two digital-to-analog converters (DAC), labeled DAC1 and DAC2 are provided in the kit. Both the DACs are DAC7821 from Texas Instruments. They are 12-bit, parallel-input multiplying DACs that can be used in place of analog multipliers in circuits like AGC/AVC. Ground and power supplies are provided internally to the DAC. Pins U9P3 and U8P3 of DAC1 and DAC2 are to be grounded and U9P18 and

U8P18 of DAC1 and DAC2 are to be connected to +5 V. In order to use the DAC integrated circuit on the `ASLKv2010 Starter` kit, the external connections shown in Figure 0-6 are required.

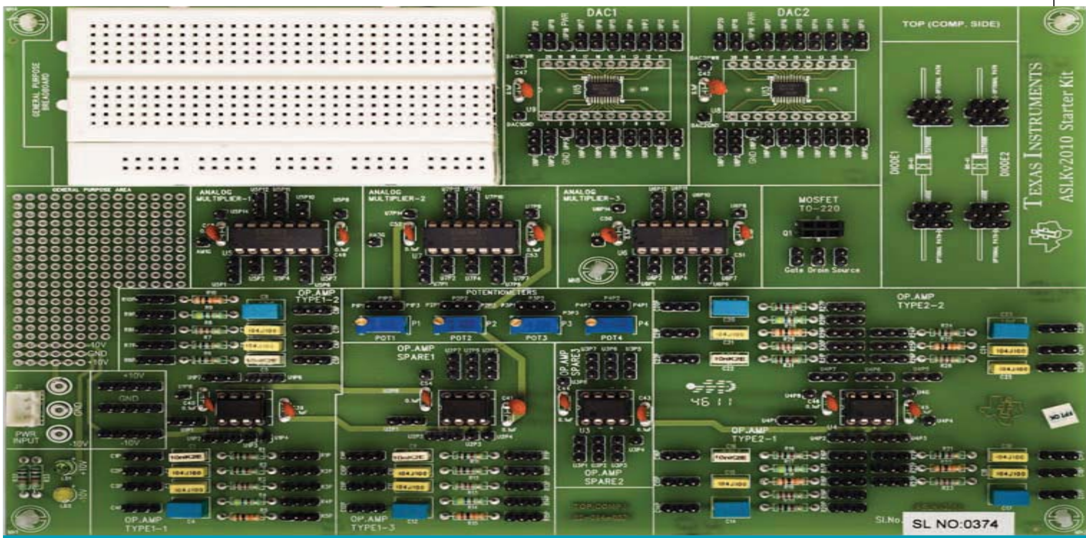
4. The kit has a provision to connect ±10 V power supplies. In addition, using the potentiometers, variable voltage can be obtained if needed for any circuit or IC. All the ICs on the board except DAC are internally connected to power supply, but in case external connection is required, it can be taken easily from Power Distribution Pins. Please refer to Appendix D for experimental configurations of `ASLKv2010 Starter` kit.
5. The top left portion of the kit is a general-purpose area which uses a proto-board.

0.6 Organization of the Manual

There are 10 experiments in this manual and the next 10 chapters are devoted to them, We recommend that in the first cycle of experiments, the instructor introduces the `ASLKv2010 Starter` kit and ensure that all the students are familiar with a SPICE-based simulation program. A warm-up exercise can be included, where the students are asked to use such a simulation program. 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 tools such as TINA, PSPICE, `FilterPro` and `SwitcherPro`.
3. Ability to use the oscilloscope.
4. Concepts of gain, bandwidth, transfer function, filters, regulators and wave shaping.

Chapter One



Experiment 1 Negative Feedback Amplifiers and Instrumentation Amplifier

1.1 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.

1.2 Brief Theory and Motivation

1.2.1 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 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? Since the frequency and transient response of an amplifier are affected 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 input, respectively. We invite the reader to view the recorded lecture [18].

An `Op-Amp` may be considered as a Voltage-Controlled Voltage Source (VCVS) with the voltage gain tending toward ∞, or a Current-Controlled Current Source (CCCS) with current gain tending toward ∞. 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 $\pm V_{ss}$ for biasing, so that the output offset voltage can be made zero when the input offset voltage is zero.

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

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

In the above equations, A_0 is the open-loop gain; for real amplifiers, A_0 is in the range of 10^3 to 10^6 and hence $V_1 \approx V_2$.

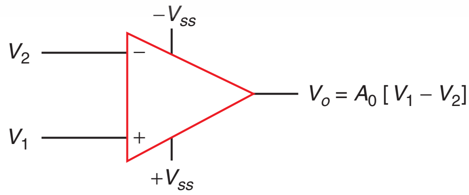


Figure 1-1 An ideal dual-input, single-output `Op-Amp`

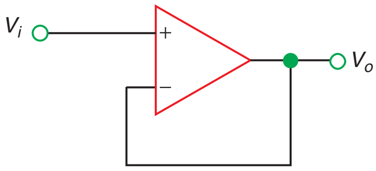


Figure 1-2 A unity gain system

A unity feedback circuit is shown in Figure 1-2. It is easy to see that

$$\frac{V_o}{V_i} = \frac{A_0}{(1 + A_0)} \tag{1.3}$$

$$\frac{V_o}{V_i} \rightarrow 1 \text{ as } A_0 \rightarrow \infty \tag{1.4}$$

In `Op-Amps`, closed loop gain A is frequency-dependent, as shown in Equation 1.5, where ω_{d1} and ω_{d2} ($\omega_{d1} < \omega_{d2}$) are known as the **dominant poles** of the operational amplifier. This transfer function is typical in an `Op-Amp` that has *internal frequency compensation*. Please view the recorded lecture [19] to get to know more about frequency compensation.

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

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

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

$$= \frac{1}{1 + \left[(1/A_0) + (s/A_0 \cdot \omega_{d1}) + (s/A_0 \cdot \omega_{d2}) + (s^2/A_0 \cdot \omega_{d1} \cdot \omega_{d2})\right]}$$

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

The term $GB = A_0\omega_{d1}$, known as the *gain bandwidth product* of the operational amplifier, is one of the most important parameters in `Op-Amp` negative feedback circuits.

The transfer function in Equation 1.7 can be rewritten as

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

where

$$Q = \frac{1}{\sqrt{\omega_{d2}/GB} + (1/A)\sqrt{GB/\omega_{d2}}}$$

We can approximate Q as

$$Q \approx \frac{1}{\sqrt{\omega_{d2}/GB}} = \sqrt{\frac{GB}{\omega_{d2}}}$$

Also,

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

Q is the Quality Factor, $\zeta = 1/2Q$ is the Damping Factor, and ω_0 is the natural frequency of the second-order system. Figure 1-3 shows the frequency response (magnitude vs ω/ω_0) of a unity gain amplifier.

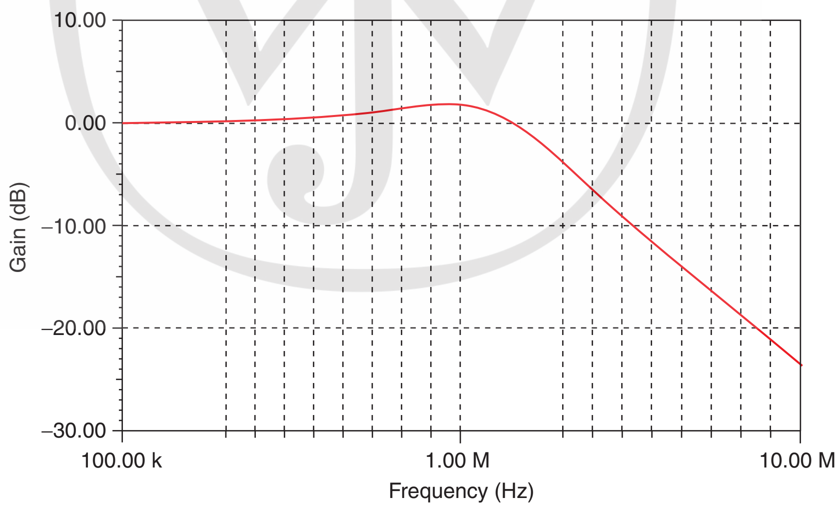


Figure 1-3 Magnitude response of a unity gain system

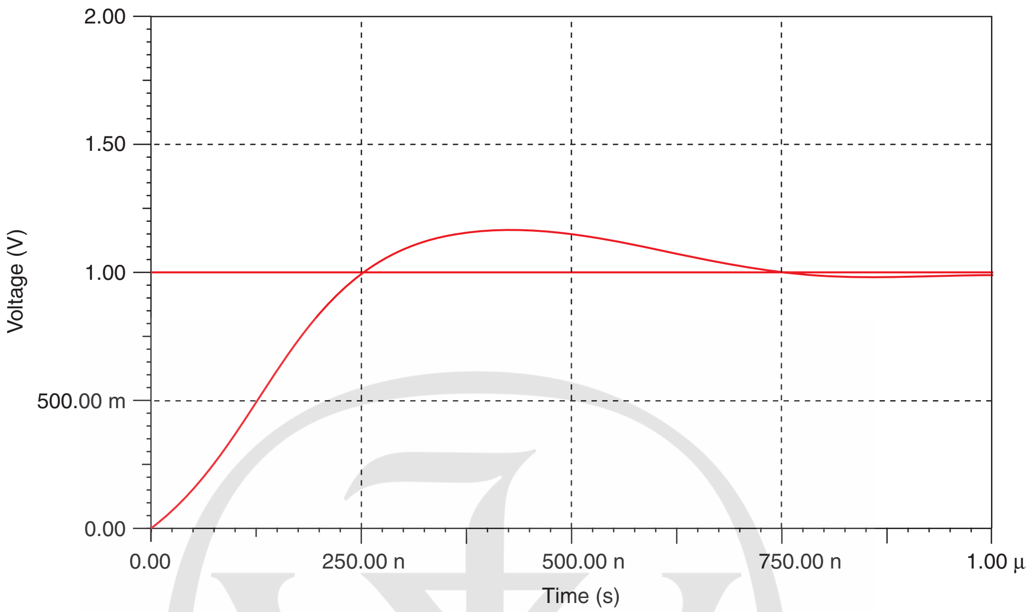


Figure 1-4 Time response of an amplifier for a step input of size V_p

If we apply a step voltage of amplitude V_p to the unity gain amplifier, and if $V_p \cdot GB <$ slew rate, then the output appears as shown in Figure 1-4 if $Q > 1/2$ or $\zeta < 1$.

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

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 increase the amplitude V_p of the step input, at some value of V_p, the rate at which the output starts rising remains constant and no longer increases with V_p; this rate is called **slew rate**. The slew rate can, therefore, be determined by applying a square wave of amplitude V_p at certain high frequency (close to gain bandwidth product) and increasing the magnitude of the input.

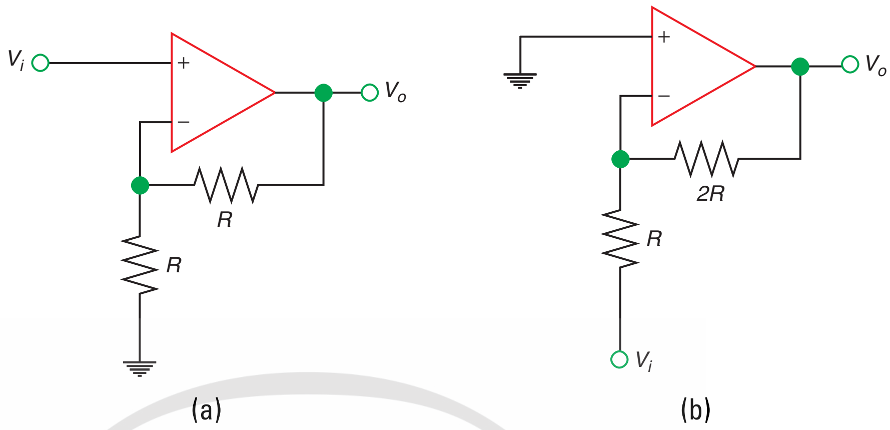


Figure 1-5 (a) Non-inverting amplifier of gain 2; (b) inverting amplifier of gain 2

A non-inverting amplifier with a gain of 2 is shown in Figure 1-5(a). An inverting amplifier with a gain of 2 is shown in Figure 1-5(b). Figure 1-6 illustrates the frequency response (magnitude and phase) of the three different negative feedback amplifier topologies.

1. V_{o1} is the frequency response of the unity-gain amplifier.
2. V_{o2} is the frequency response of the non-inverting amplifier.
3. V_{o3} is the frequency response of the inverting amplifier.

The figure also shows the time-domain response of the amplifier. Figure 1-7 shows the output of the three types of amplifiers for a square-wave input, illustrating the limitations due to slew rate.

1.3 Specifications

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 [Figure 1-5(b)].

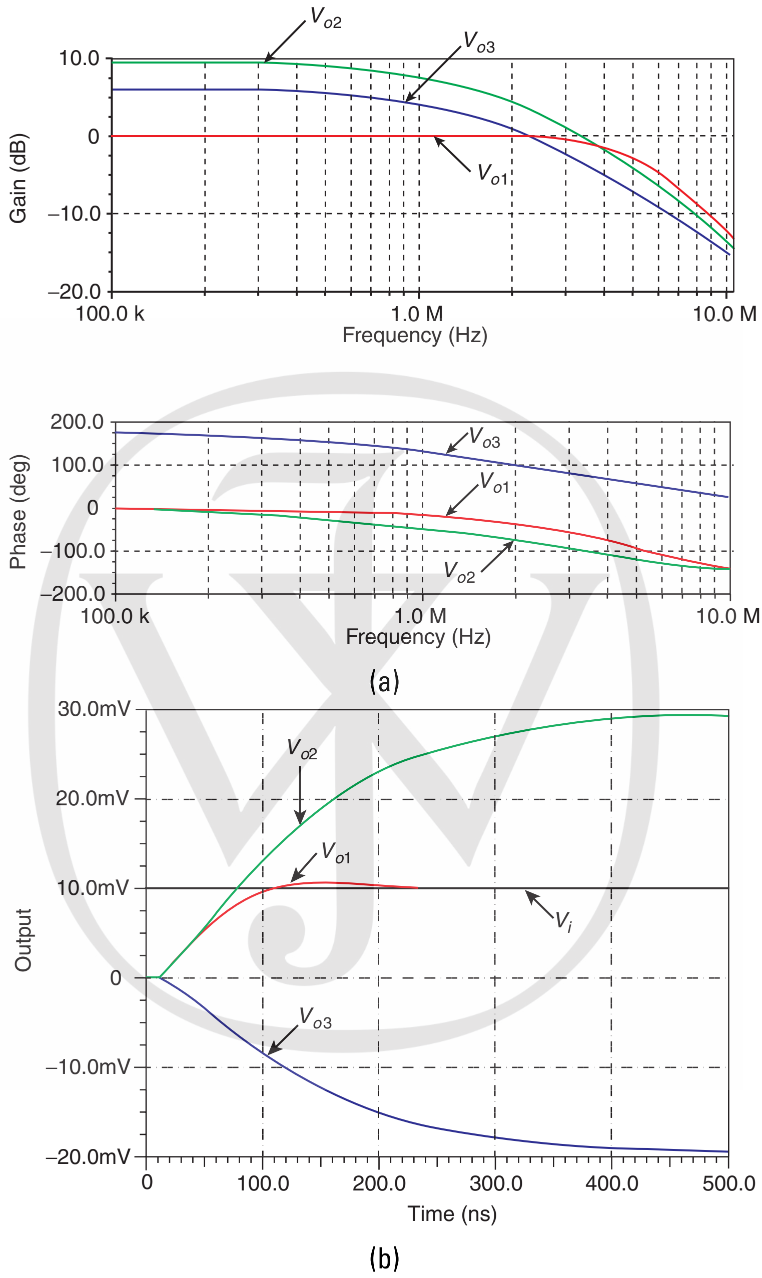


Figure 1-6 (a) Frequency response of negative feedback amplifiers; (b) time response of negative feedback amplifiers

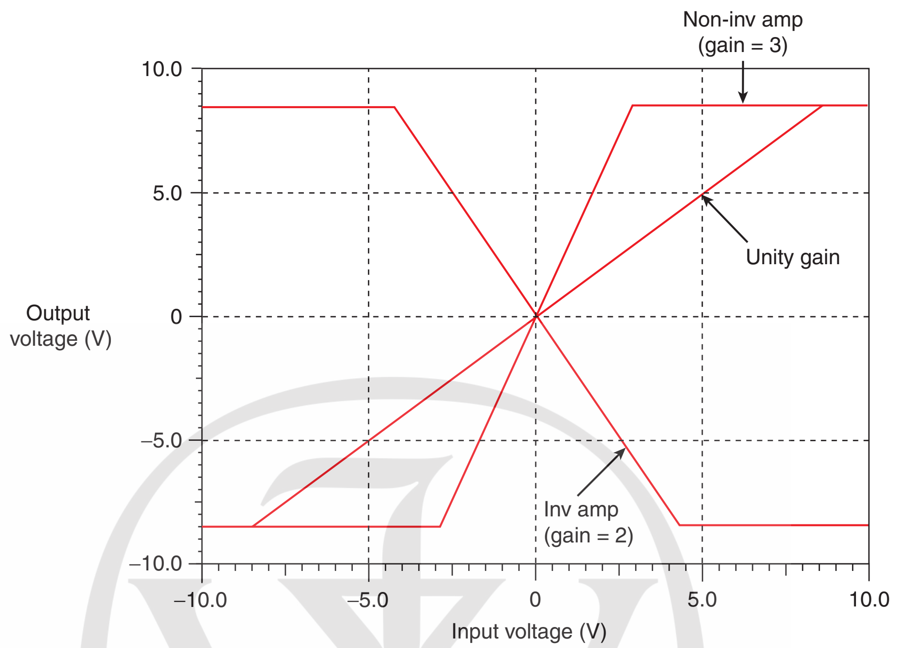


Figure 1-7 Transfer characteristics of unity-gain, non-inverting, and inverting amplifiers

1.4 Measurements to be Taken

1. Time response: Apply a square wave of fixed magnitude and study the effect of slew rate on the three types of feedback amplifiers, namely, unity-gain, inverting and non-inverting amplifiers.
2. Frequency response: Obtain the gain bandwidth product of the three types of feedback amplifiers, namely, unity-gain, inverting and non-inverting amplifiers, from the frequency response. If we refer to the gain of the feedback amplifier as A and the bandwidth of the feedback amplifier as ω, notice that

$$A \cdot \omega = GB$$

This illustrates the tradeoff between gain and bandwidth in a feedback amplifier.

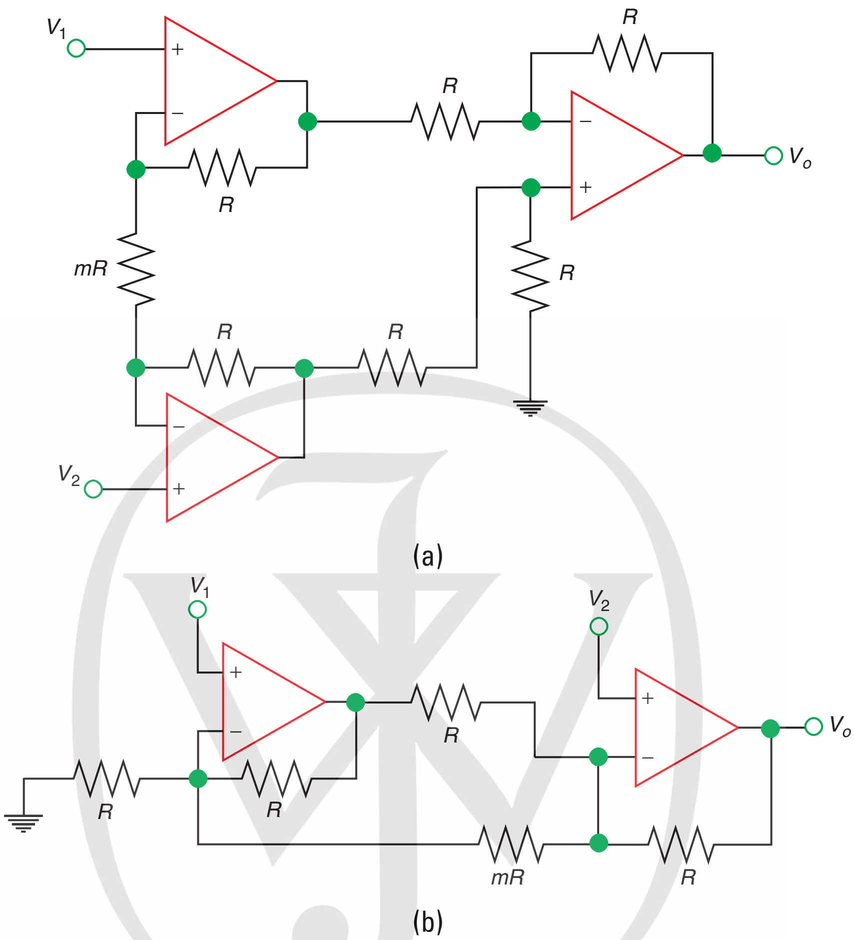


Figure 1-8 Instrumentation amplifier configurations with (a) **three**; (b) **two** operational amplifiers

3. DC transfer characteristics: When we increase the gain of the feedback amplifier, the *input range* over which the output of the amplifier remains *linear* with respect to input voltage will begin to reduce. In fact, this range is given by $2 \cdot V_{ss}/A$. From the DC-transfer characteristic of Table 1-4, determine the input range of the amplifier where the output remains linear with respect to the input voltage.

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.

1.5 What Should you Submit

1. Submit the simulation results for time response, frequency response and DC transfer characteristics.
2. Take the plots of time response, frequency response and DC transfer characteristics from the oscilloscope and compare them with your simulation results.
3. Apply a square wave of amplitude 1 V 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. (Hint for calculating the slew rate: After the slew rate has been achieved, the peak-to-peak amplitude of output starts falling.)
4. Apply a high-frequency square wave and increase the peak-to-peak amplitude of the input. Compute the slew rate. Take the readings in Table 1-2.
5. **Frequency response:** Apply sine wave input to the system and study the magnitude and phase response. Take your readings in Table 1-3.
6. **DC transfer characteristics:** Vary the DC input voltage and study its effect on the output voltage. Take your readings in Table 1-4.

Table 1-1 Measurement of slew rate: Method 1

S. No.	Input Frequency	Peak-to-Peak Amplitude of Output (V_{pp})
1		
2		
3		
4		

Table 1-2 Measurement of slew rate: Method 2

S. No.	Input Voltage	Peak-to-Peak Amplitude of Output (V_{pp})
1		
2		
3		
4		

Table 1-3 Plot of magnitude and phase variation w.r.t. input frequency

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

Table 1-4 DC transfer characteristic

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

1.6 Exercises

1. Design an instrumentation amplifier of a differential mode gain A_d of 3 using *three* `Op-Amps`. Refer to Figure 1-8(a) for the circuit diagram and determine the values of the resistors. Assume that the resistors have tolerance δ of 1% and determine the Common Mode Rejection Ratio (CMRR) of the setup using the following equation.

$$\text{CMRR} = \frac{A_d}{2 \cdot \delta}$$

Estimate the bandwidth of the instrumentation amplifier. We invite the reader to view the recorded lecture [20].

2. Design an instrumentation amplifier with a differential-mode gain A_d of 5 using *two* `Op-Amps`. Refer to Figure 1-8(b) for the circuit diagram and determine the values of the resistors. Assume that the resistors have 1% tolerance and determine the CMRR of the setup. Estimate the bandwidth of the instrumentation amplifier.

*3. Figure 1-9(a) shows an inverting amplifier whose gain is inversely proportional to the control voltage V_3. Show that the gain of the amplifier is $10/V_3$. Remember that the multiplier has a scaling factor of 0.1/V. Measure the gain and bandwidth of the amplifier when $V_3 = 1$ V. Repeat this experiment for $V_3 = 2$ V and 5 V. How is this amplifier topology better than that of Figure 1-5(a)? Can you think of an application for this amplifier?

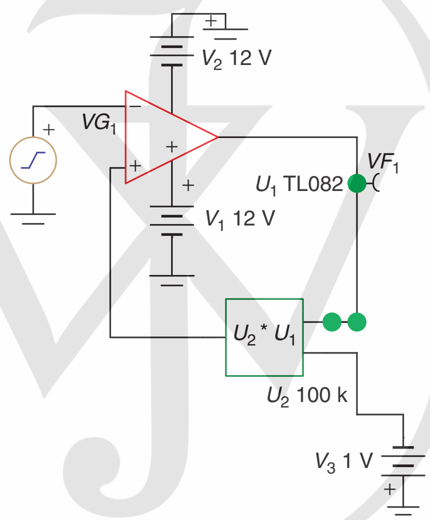


Figure 1-9(a) An inverting amplifier with analog gain control

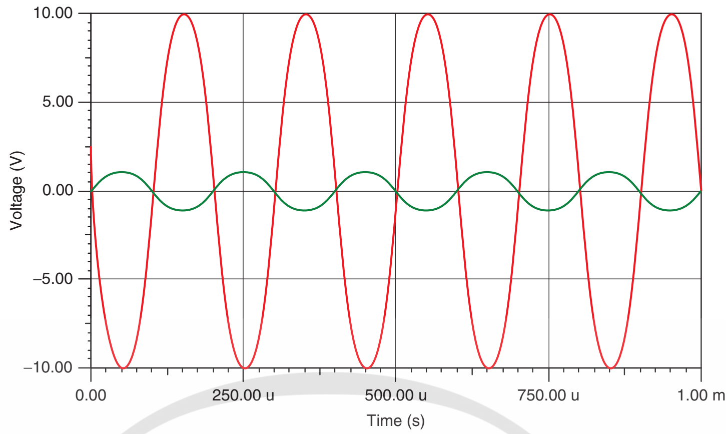


Figure 1-9(b) Simulation of the circuit for Figure 1-9(a) when $V_3 = 1\,\text{V}$

*4 A digitally controlled (programmable) amplifier is shown in Figure 1-10. It is an inverting amplifier whose gain (magnitude) G is given by

$$\frac{V_{out}}{V_{in}} = \frac{R_2}{R_1} \cdot \frac{4096}{\sum_{0}^{11} A_0 \cdot 2^n}$$

Determine the maximum and minimum limits of the gain G. Note that the input to the DAC is a 12-bit binary word ($A_{11}, \ldots, A_0$). Can you think of an application for such an amplifier? Compare the circuits of Figure 1-9(a) and Figure 1-10.

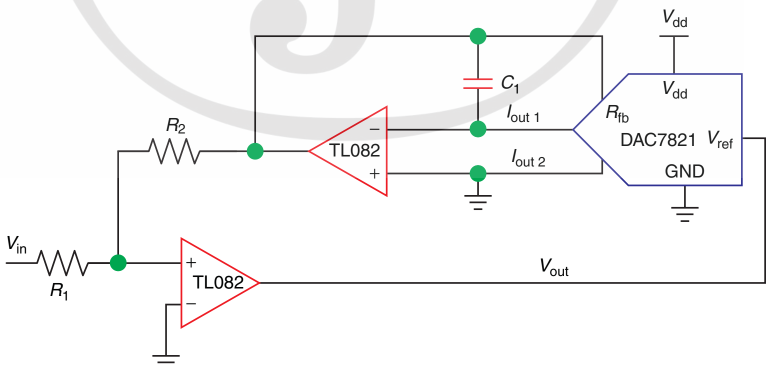


Figure 1-10 Digitally controlled amplifier

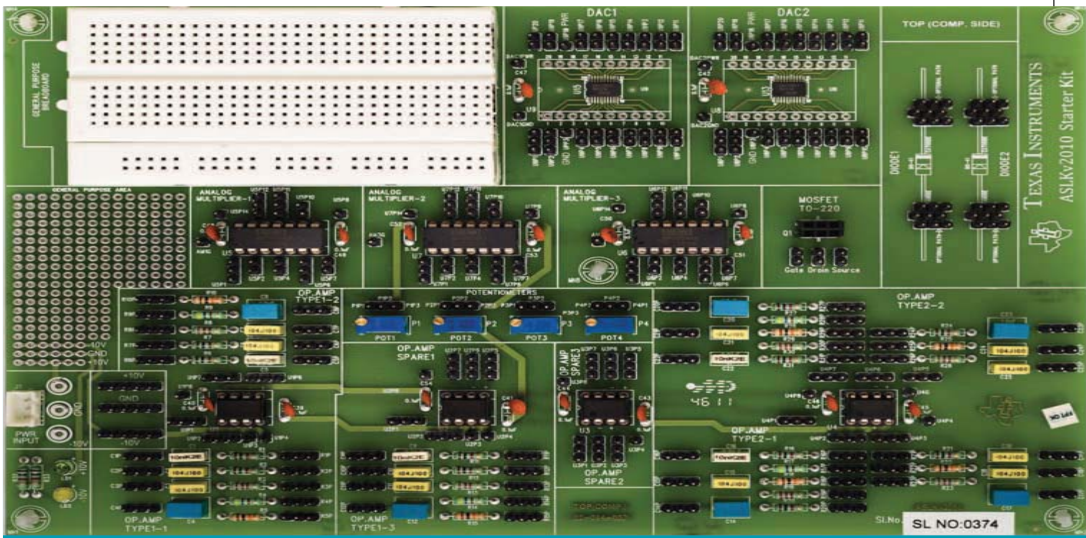
1.7 Other Related ICs

Specific ICs from Texas Instruments that can be used as instrumentation amplifiers are INA114, INA118 and INA128. Additional ICs from Texas Instruments that can be used as general purpose `Op-Amps` are OPA703, OPA357, etc. See [3].

1.8 Related Reading

Datasheets of all these ICs are available at www.ti.com. An excellent reference about operational amplifiers is the "Handbook of Operational Amplifier Applications" by Carter and Brown [5]. The book "OPAMPS For Everyone" by Carter and Mancini is also an excellent resource [8].

Chapter Two



Experiment 2 Regenerative Feedback System, Astable and Monostable Multivibrator

2.1 Goal of the Experiment

This experiment illustrates the use of positive regenerative feedback used in all ON–OFF control systems such as temperature controllers, pulse width modulators and Class-D amplifiers. The goal of this experiment is to understand the basics of hysteresis and the need of hysteresis in switching circuits.

2.2 Brief Theory and Motivation

2.2.1 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 the Figure 2-1.

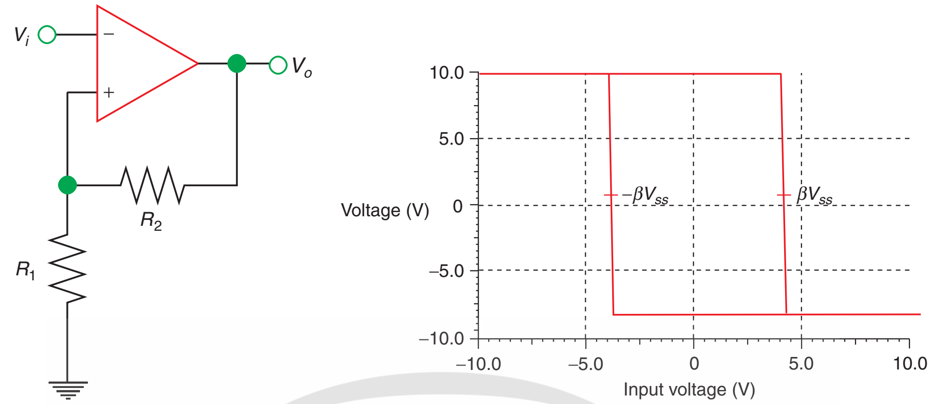


Figure 2-1 Inverting Schmitt trigger and its hysteresis characteristic

The reader will benefit by listening to the recorded lecture at [22]. The relation between the input voltage V_i and output voltage V_o is given by Equation 2.3, where $\beta = \frac{R_1}{R_1+R_2}$.

$$V_o = -A_0(V_i - \beta V_o) \tag{2.1}$$

$$V_o/V_i = -A_0\frac{1}{1 - A_0\beta} \tag{2.2}$$

$$= \frac{\frac{1}{\beta}}{1 - \frac{1}{A_0\beta}} \tag{2.3}$$

There are three cases to be considered.

1. Case 1 – $|A_0 \cdot \beta| < 1$: In this case, the circuit behaves as an amplifier and the output voltage has a linear relation to the input voltage. However, the gain is very sensitive to variations in $|A_0\beta|$
2. Case 2 – $|A_0 \cdot \beta| = 1$: In this case, the amplifier becomes unstable and its output saturates.
3. Case 3 – $|A_0 \cdot \beta| >> 1$: The output voltage is no longer related linearly to input voltage. This configuration is useful in interface circuits, where the output voltage behaves in a “digital” way and shows two stable states, namely, $+V_{ss}$ and $-V_{ss}$.

When the input is a large negative value, the output saturates at $+V_{ss}$. As the input is increased, the output remains constant at $+V_{ss}$, and when the input reaches $\beta \cdot V_{ss}$, the device enters into the regenerative feedback mode and the output changes from $+V_{ss}$ to $-V_{ss}$. Now when the input is decreased, the circuit can change state only when the input becomes $-\beta V_{ss}$. See Figure 2-1. Thus there is a hysteresis of $\pm\beta V_{ss}$ on either side of origin and there is a total hysteresis of $2 \cdot \beta \cdot V_{ss}$. This kind of comparator is required when 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(a). One can similarly construct a non-inverting Schmitt trigger, for which the symbol is shown in Figure 2-2(b). The non-inverting Schmitt trigger circuit is shown in Figure 2-3.

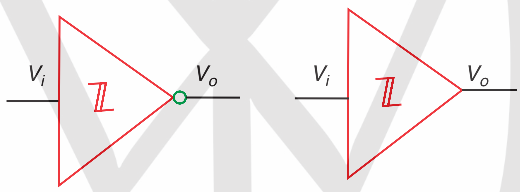


Figure 2-2 Symbols for (a) inverting; (b) non-inverting Schmitt trigger circuits

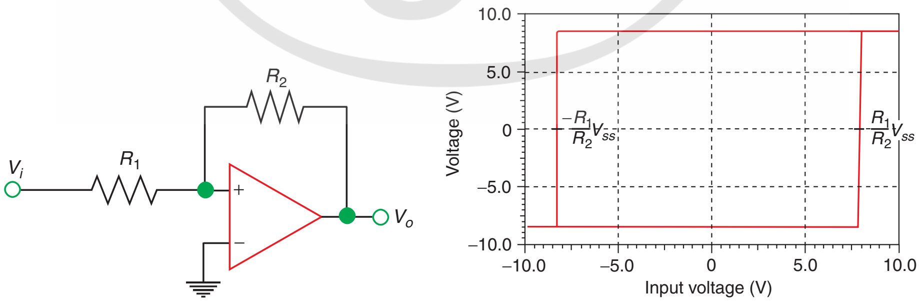


Figure 2-3 Non-inverting Schmitt trigger and its hysteresis characteristic

2.2.2 Astable Multivibrator

An astable multivibrator is shown in Figure 2-4. The first two waveforms V_{o1} and V_{o2} shown in Figure 2-5, namely, square and the triangular waveforms, are generated using the astable multivibrator. We refer to β as the regenerative feedback factor. The time period of the square waveform generated by the multivibrator is given by

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

βV_{ss} refers to the peak amplitude of the triangular waveform.

2.2.3 Monostable Multivibrator (Timer)

The circuit diagram for a monostable multivibrator is shown in Figure 2-6. The trigger waveform is applied to the monostable multivibrator at the positive terminal, which produces the outputs V_{o3} and V_{o4} at the output, as shown in Figure 2-5. The monostable remains in the "ON" state until it is triggered; at this time, the circuit switches to the

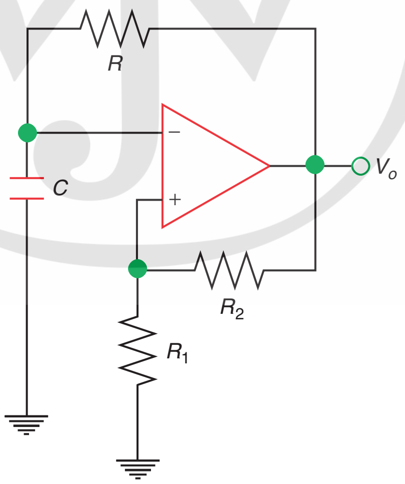


Figure 2-4 Astable multivibrator

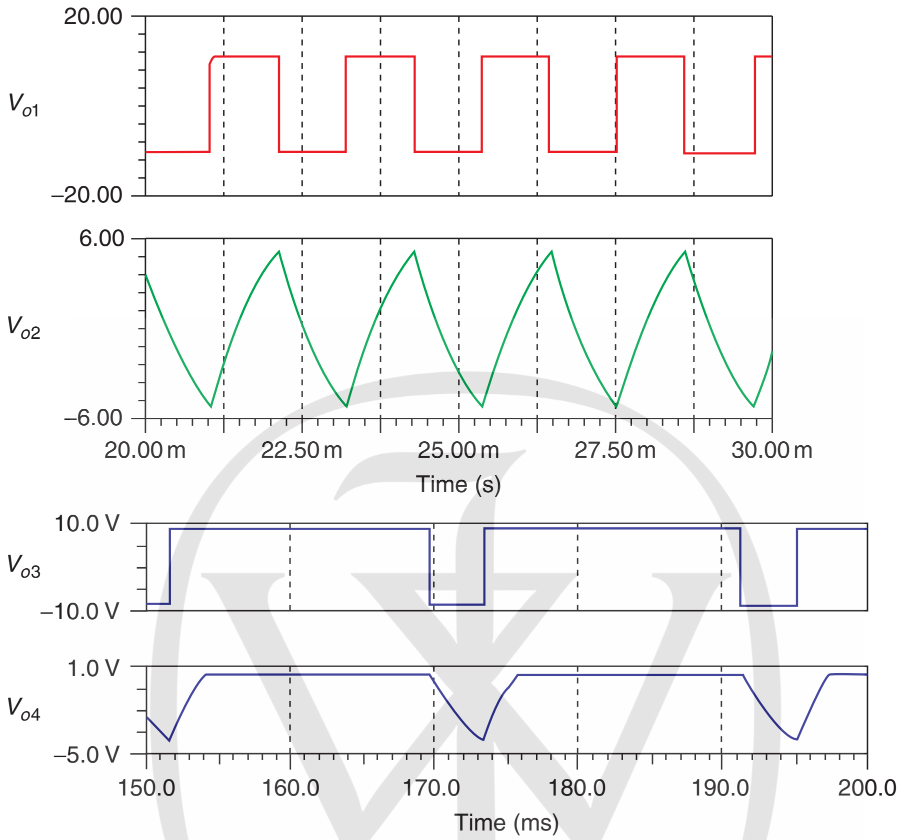


Figure 2-5 Simulation results for (a) astable multivibrator; (b) monostable multivibrator

"OFF" state for a period equal to τ. The equation for τ is shown below.

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

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

$$\tau' = RC \ln \left(\frac{1 + \beta}{\beta} \right) \tag{2.6}$$

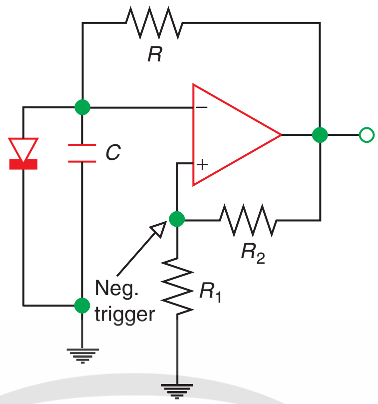


Figure 2-6 Monostable multivibrator

2.3 Specifications

Design a regenerative feedback circuit with a hysteresis of ± 1 V. Refer to Figure 2-3 for the circuit diagram.

2.4 Measurements to be Taken

Obtain the DC transfer characteristics of the system. Estimate the hysteresis and see how it can be controlled by varying the regenerative feedback factor. Vary either R_1 or R_2 in order to vary β.

Apply the triangular waveform with the peak voltage of 10 V at a given frequency and observe the output waveform. Observe the delay between the input and the output waveforms at the zero-crossover point. Enter the reading of the delay in the rightmost column of the Table 2-1. As you vary the hysteresis, the delay must also vary in direct proportion to the hysteresis.

Table 2-1 Plot of hysteresis w.r.t. regenerative feedback

S. No.	Regenerative Feedback Factor β	Hysteresis (Width)	Delay
1			
2			
3			
4			

2.5 What Should you Submit

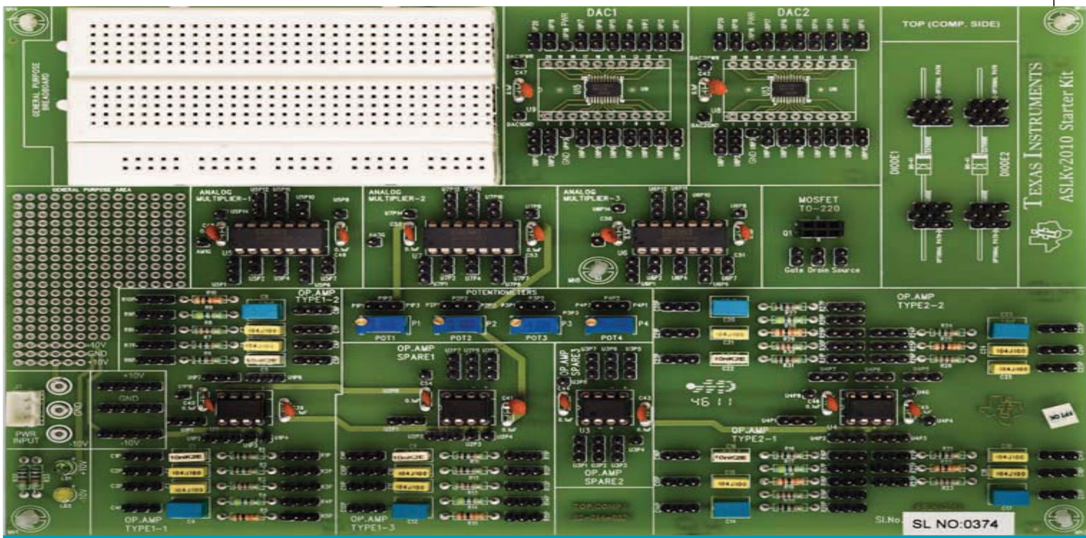
Use Table 2-1 to note down your readings.

1. Submit the DC transfer characteristics obtained using simulation.
2. Take the plot of DC transfer characteristics from the oscilloscope and compare it with simulation result.
3. Vary the regenerative feedback and observe the variation in the hysteresis. Is the hysteresis directly proportional to regenerative feedback factor β?

2.6 Exercises

1. Design an astable multivibrator using charging and discharging of capacitor C through resistance R between input and output of the Schmitt trigger. See Figure 2-4. Assume that frequency $f = 1/T = 1$ kHz.
2. Design a monostable multivibrator (Timer) for $\tau = 10$ ms and estimate RC using Equation 2.5.

Chapter Three



Experiment 3 Integrators and Differentiators

3.1 Goal of the Experiment

The goal of the experiment is to understand the advantages and disadvantages of using integrators or differentiators as building blocks in building N^{th} order filters.

3.2 Brief Theory and Motivation

Integrators and differentiators can be used as building blocks for filters. Filters are essential blocks in analog signal processing to improve *signal to noise ratio*. An `Op-Amp` can be used to construct an integrator or a differentiator. This experiment is to understand the advantage of using integrators instead of differentiators as building blocks. Differentiators are rejected because of their good response to noise.

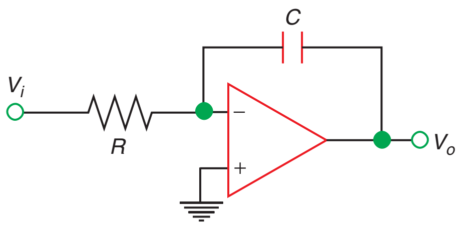


Figure 3-1 Integrator

3.2.1 Integrators

An integrator circuit that uses an Op-Amp is shown in Figure 3-1.

Assuming $A = GB/s$, the transfer function of the integrator is given by

$$\frac{V_o}{V_i} = -\frac{\dfrac{1}{sCR}}{\left(1 + \dfrac{1}{GB \cdot RC} + \dfrac{s}{GB}\right)}$$

The output goes to saturation in practice. To make it work, a high valued resistance across C must be added in order to bring the Op-Amp to the active region where it can act as an integrator.

3.2.2 Differentiators

A differentiator circuit that uses an Op-Amp is shown in Figure 3-2.

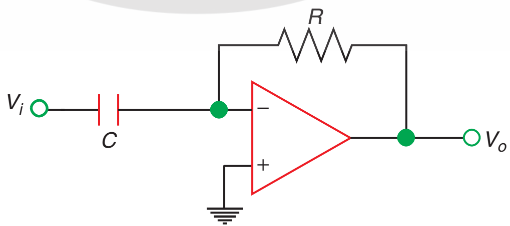


Figure 3-2 Differentiator

Again, assuming that $A = GB/s$, the transfer function of the differentiator is given by

$$\frac{V_o}{V_i} = \frac{-sRC}{\left(1 + \frac{s}{GB} + s^2 \cdot \frac{RC}{GB}\right)} \tag{3.1}$$

$$= \frac{-sRC}{\left(1 + \frac{s}{\omega_0 Q} + \frac{s^2}{\omega_0^2}\right)} \tag{3.2}$$

The output of the differentiator remains at input offset (approximately 0). However, any sudden disturbance at the input causes it to go to ringing at natural frequency ω_0.

3.3 Specifications

Fix the RC time constant of the integrator or differentiator so that the phase shift and magnitude variation of the ideal block remains unaffected by the active device parameters.

3.4 Measurements to be Taken

1. Time response: Apply a step input and a square-wave input to the integrator and study the output response. Apply a triangular and square-wave input to the differentiator and study the output response.
2. Frequency response: Apply the sine-wave input and study the phase error and magnitude error for integrator and differentiator.

3.5 What Should you Submit

1. Simulate the integrator and differentiator using a simulator software and obtain the transient response.
2. Take the plots of transient response on an oscilloscope and compare them with simulation results.
3. Frequency response: Apply a sine wave to the integrator (similarly to the differentiator) and vary the input frequency to obtain phase and magnitude errors. Prepare a table of the form Table 3-1. Figure 3-3 shows the typical frequency response for integrators and differentiators. The first two plots, VF_1 and VF_2, are the magnitude responses of the integrator and differentiator, respectively. The next two plots VF_1 and VF_2 are the phase responses of the integrator and differentiator, respectively. For an integrator, the plot shows a phase lag proportional to ω/GB. The magnitude decreases with increasing frequency. For the differentiator, the phase will change rapidly at natural frequency in direct proportion to Quality Factor. The magnitude peaks at natural frequency and is directly proportional to the Quality Factor.
4. Time response: Apply a square-wave input of amplitude V_p to the integrator. Vary the peak amplitude of the square wave and obtain the peak-to-peak value V_{pp} of the output. V_{pp} is directly proportional to V_p and is given by $V_{pp} = V_p T/2RC$, where $T = 1/f$, f being the input frequency. Figure 3-4 shows sample output waveforms obtained through simulation. In Figure 3-4(a), the input waveform is a square wave; the triangular waveform is the output of the integrator and the ringing waveform is

Table 3-1 Plot of magnitude and phase w.r.t. input frequency for the integrator

S. No.	Input Frequency	Magnitude	Phase
1			
2			
3			
4			
5			

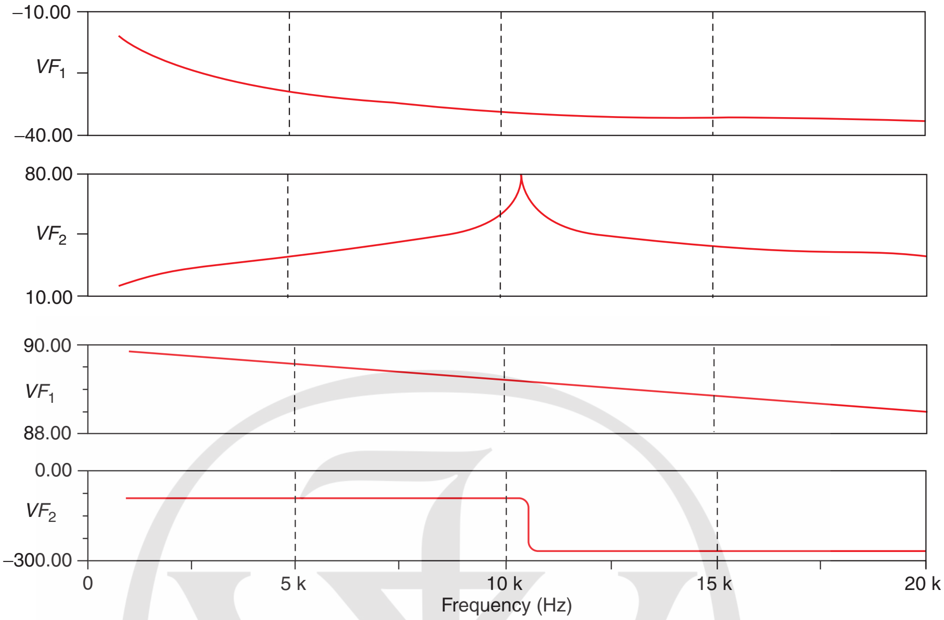


Figure 3-3 Frequency response of integrator and differentiator

Table 3-2 Plot of magnitude and phase w.r.t. input frequency for the differentiator

S. No.	Input Frequency	Magnitude	Phase
1			
2			
3			
4			
5			

the output of the differentiator. We leave it as an exercise for the student to figure out which are the outputs of the integrator and differentiator in Figure 3-4(b).

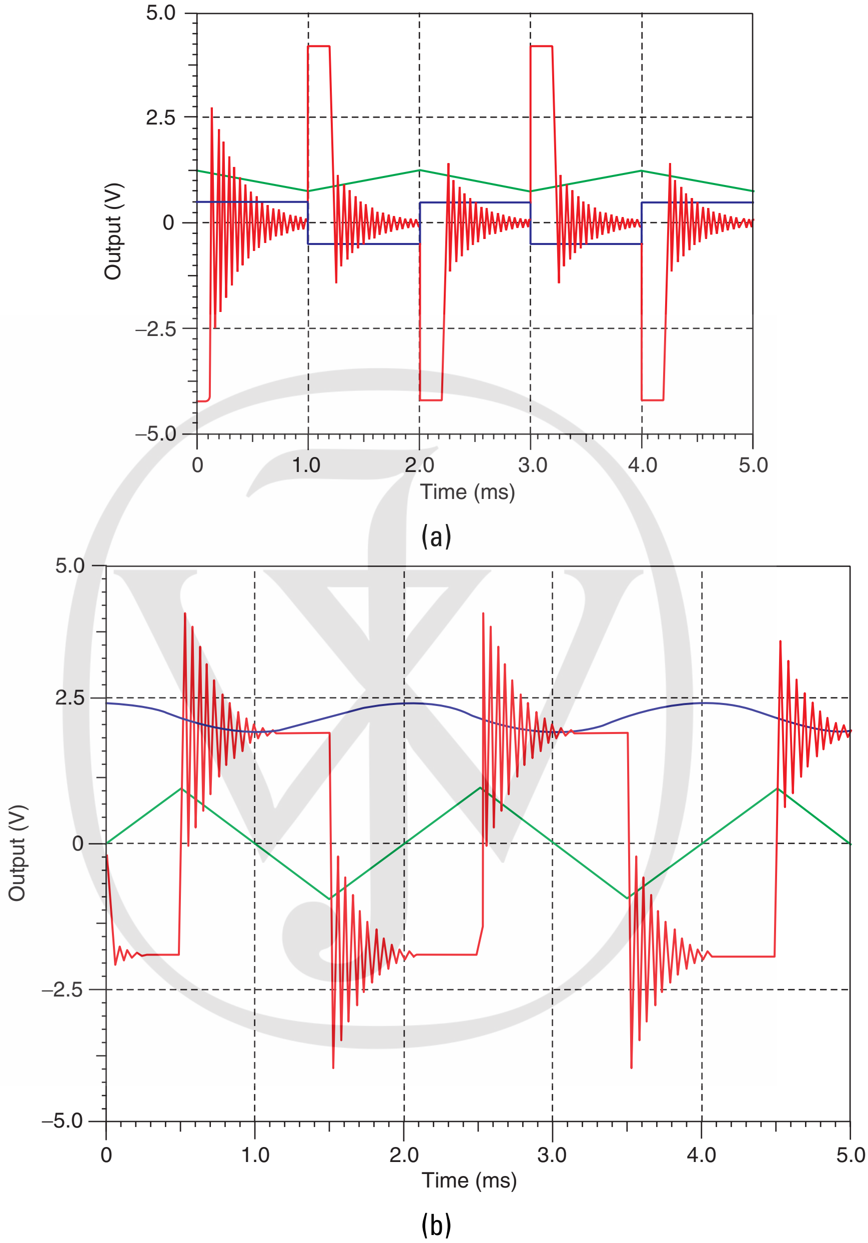


Figure 3-4 Outputs of integrator and differentiator for (a) square-wave; (b) triangular-wave inputs

Table 3-3 Variation of peak-to-peak value of output w.r.t. peak value of input

S. No.	Peak Value of Input V_p	Peak-to-Peak Value of Output
1		
2		
3		
4		

3.6 Exercise: Grounded Capacitor Topologies of Integrator and Differentiator

Determine the function of the circuits shown in Figure 3-5. What are the advantages and disadvantages of these circuits when compared to their conventional counterparts?

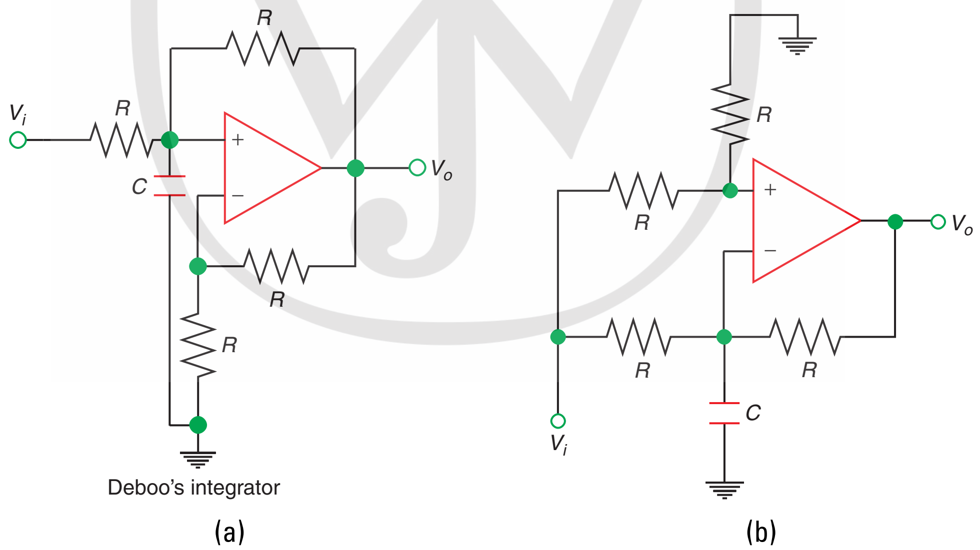
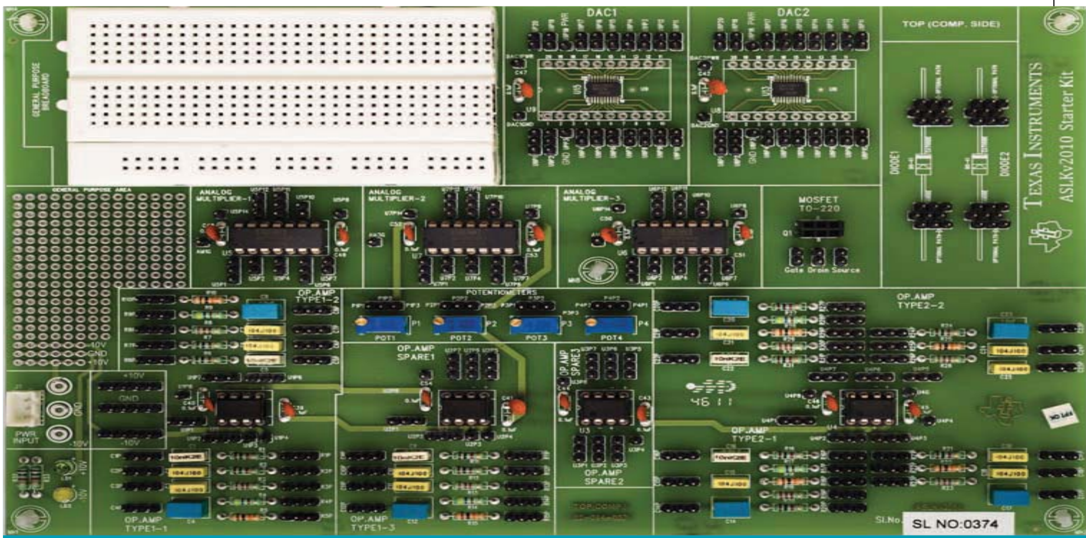


Figure 3-5 Circuits for Exercise

Chapter Four



Experiment 4 Analog Filters

4.1 Goal of the Experiment

To understand the working of four types of *second-order filters,* namely, **Low Pass**, **High Pass**, **Band Pass**, and **Band Stop** filters, and study their frequency characteristics (phase and magnitude).

4.2 Brief Theory and Motivation

Second-order filters (or biquard filters) are important since they are the building blocks in the construction of N^{th}-order filters, for $N > 2$. When N is odd, the N^{th}-order filter can be realized using $(N-1)/2$ second-order filters and one first-order filter. When N

is even, we need $N/2$ second-order filters. Please listen to the recorded lecture at [21] for a detailed explanation of active filters.

Second-order filter can be used to construct four different types of filters. The transfer functions for the different filter types are shown in Table 4-1, where $\omega_0 = 1/RC$ and H_0 is the low frequency gain of the transfer function. The filter names are often abbreviated as LPF (Low Pass Filter), HPF (High Pass Filter), BPF (Band Pass Filter), and BSF (Band Stop Filter). In this experiment, we will describe a universal active filter that provides all four filter functionalities. Figure 4-5(b) shows a second-order universal filter

Table 4-1 Transfer functions of active filters

Low Pass Filter	$\frac{V_{o3}}{V_i} = \frac{+H_0}{\left(1 + \frac{s}{\omega_0 Q} + \frac{s^2}{\omega_0^2}\right)}$
High Pass Filter	$\frac{V_{o1}}{V_i} = \frac{\left(H_0 \cdot \frac{s^2}{\omega_{o2}}\right)}{\left(1 + \frac{s}{\omega_0 Q} + \frac{s^2}{\omega_0^2}\right)}$
Band Pass Filter	$\frac{V_{o2}}{V_i} = \frac{\left(-H_0 \cdot \frac{s}{\omega_0}\right)}{\left(1 + \frac{s}{\omega_0 Q} + \frac{s^2}{\omega_0^2}\right)}$
Band Stop Filter	$\frac{V_{o4}}{V_i} = \frac{-\left(1 + \frac{s^2}{\omega_0^2}\right) \cdot H_0}{\left(1 + \frac{s}{\omega_0 Q} + \frac{s^2}{\omega_0^2}\right)}$

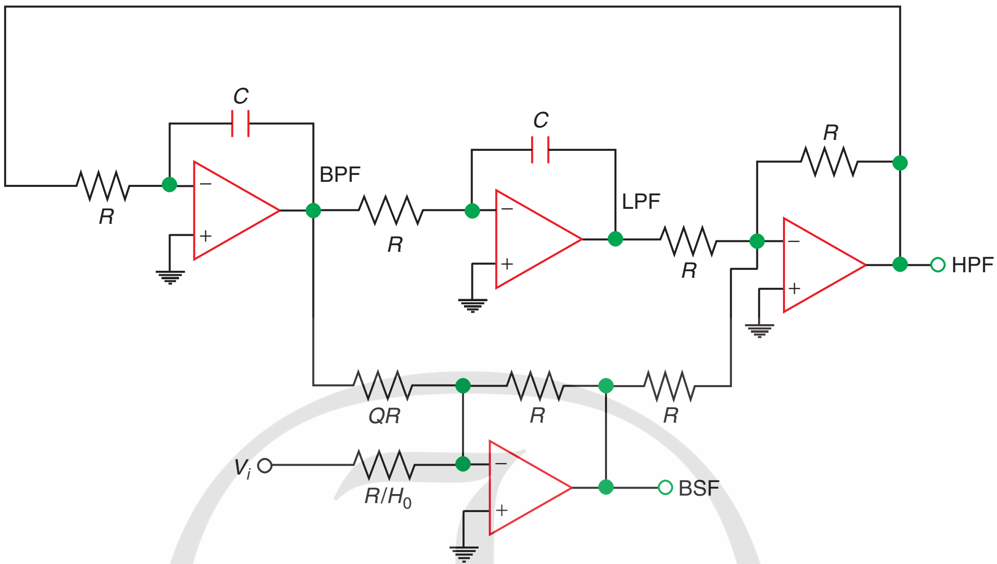


Figure 4-1 A second-order universal active filter

realized using two integrators. Note that there are different outputs of the circuit that realize LPF, HPF, BPF and BSF functions.

4.2.1 | Frequency Response of Filters

The magnitude response of two of the filters, BPF and BSF, are shown in Figure 4-3. The phase sensitivity $\delta\phi/\delta\omega$ is maximum at $\omega = \omega_0$ and is given by $-2Q/\omega_0$. This information about phase variation can be used to tune the filter to a desired frequency ω_0. This is demonstrated in the next experiment. For the BPF, the magnitude response peaks at $\omega = \omega_0$ and is given by $H_0 Q$. The BSF shows a null magnitude response at $\omega = \omega_0$.

4.3 Specifications

Design a Band Pass and a Band Stop filter. For the BPF, assume F_0 = 1 kHz and $Q = 1$. For the BSF, assume F_0 = 10 kHz and $Q = 10$.

4.4 Measurements to be Taken

1. Steady-state response: Apply a square-wave input (try $f = 1$ kHz and $f = 10$ kHz) to both BPF and BSF circuits and observe the outputs. A sample output is shown in Figure 4-2.

   - Band Pass output will output the fundamental frequency of the square wave multiplied by the gain at the center frequency. The amplitude at this frequency is given by $\dfrac{4 \cdot V_p}{\pi \cdot H_0 \cdot Q}$, where V_p is the peak amplitude of the input square wave.

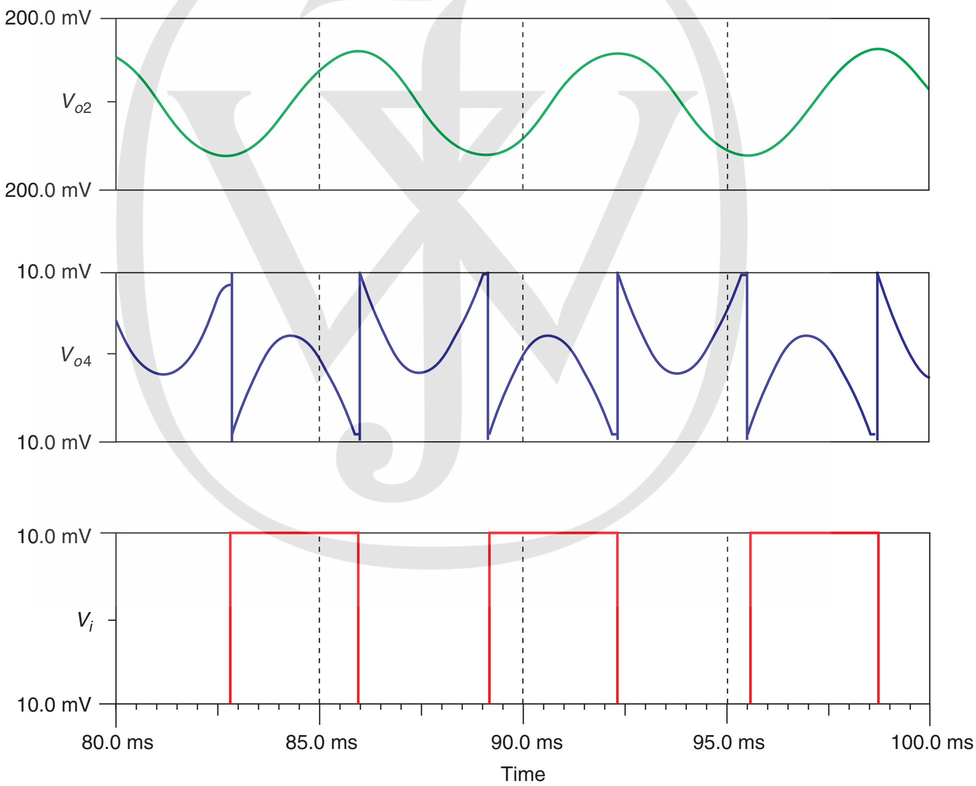


Figure 4-2 Simulation waveform for a universal active filter

- The BSF output will carry all the harmonics of the square wave, other than the fundamental frequency. This illustrates the application of BSF as a distortion analyzer.

2. Frequency response: Apply a sine-wave input and obtain the magnitude and the phase response.

4.5 What Should you Submit

1. Simulate the circuits in using a simulator software and obtain the steady-state response and frequency response for both the filters.
2. Take the plots of the steady-state response and frequency response from the oscilloscope for both the filters and compare the results with simulation results.
3. Frequency response: Apply a sine-wave input and vary its input frequency to obtain the phase and magnitude error. Use Tables 4-2 and 4-3 to note your readings. The nature of graphs should be as shown in Figure 4-3.

Table 4-2 Frequency response of a BPF with $F_0 = 1$ kHz, $Q = 1$

		Band Pass		Band Stop	
S. No.	**Input frequency**	**Phase**	**Magnitude**	**Phase**	**Magnitude**
1					
2					
3					
4					

Table 4-3 Frequency response of a BSF with $F_0 = 10$ kHz, $Q = 10$

		Band Pass		Band Stop	
S. No.	**Input frequency**	**Phase**	**Magnitude**	**Phase**	**Magnitude**
1					
2					
3					
4					

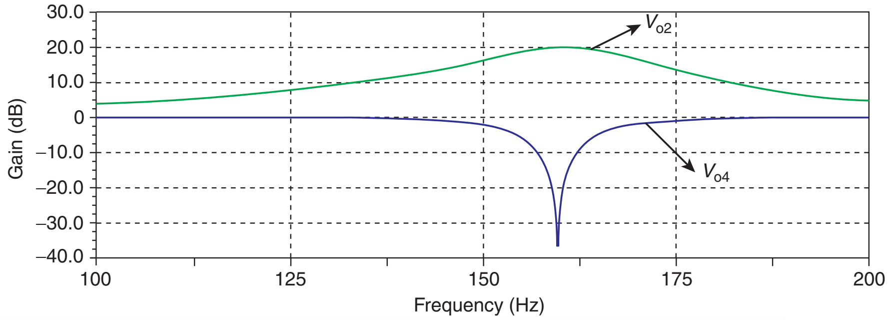


Figure 4-3 Magnitude response of BPF, BSF

4.6 Exercises

1. Higher-order filters are normally designed by cascading second-order filters and, if needed, one first-order filter. Design a third-order Butterworth Low Pass Filter using `FilterPro` and obtain the frequency response as well as the transient response of the filter. The specifications are: bandwidth of the filter $\omega_0 = 2 \cdot \pi \cdot 10^4$ rad/s and $H_0 = 10$.

2. Design a notch filter (Band Stop filter) to eliminate the 50 Hz power line frequency. In order to test this circuit, synthesize a waveform $v(t) = \sin(100\pi t) + 0.1 \sin(200\pi t)$ Volts and use it as the input to the filter. What output did you obtain?

*3. A third-order Butterworth filter is designed as shown. It realizes a transfer function,

$$\frac{1}{1 + 2(sCR)^1 + 2(sCR)^2 + (sCR)^3}$$

with $C = 1\,\mu$F and $R = 1$ k. Determine its bandwidth. The desired transfer function is realized using cascading of a second-order filter with Q of 1 and a first-order filter.

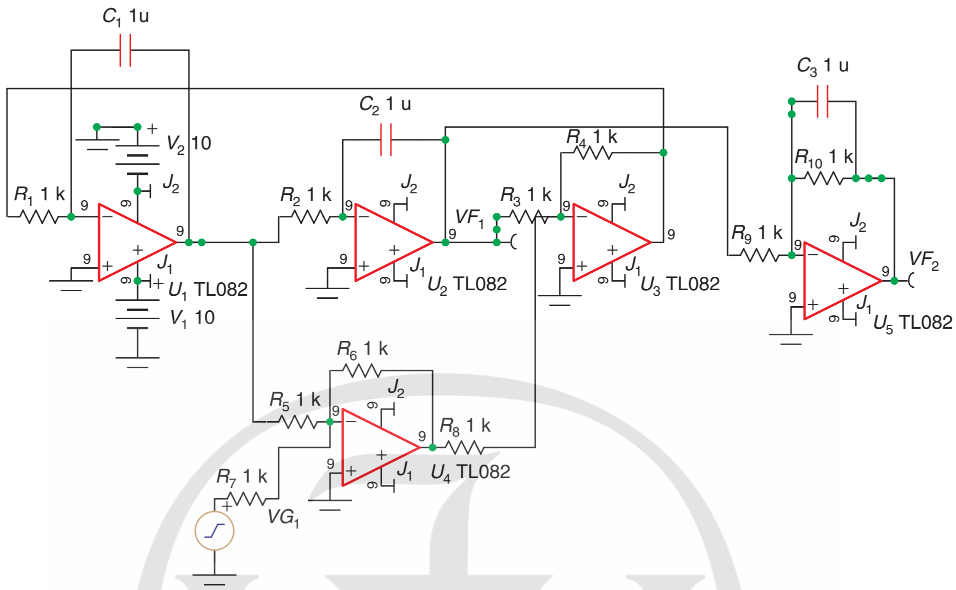


Figure 4-4(a) Third-order Butterworth filter

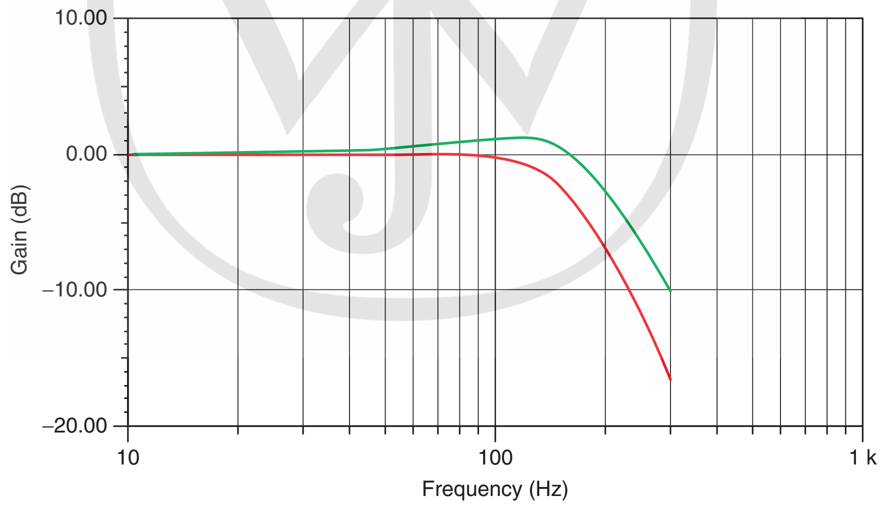


Figure 4-4(b) Frequency response of the Butterworth filter

*4 The filter of Figure 4-5(a) is known as Tow-Thomas Biquad Filter. It is designed for a pole *Q* of 10 and pole frequency of 10 krad/sec. Obtain its frequency response VF_1/VG_1 and VF_2/VG_2.

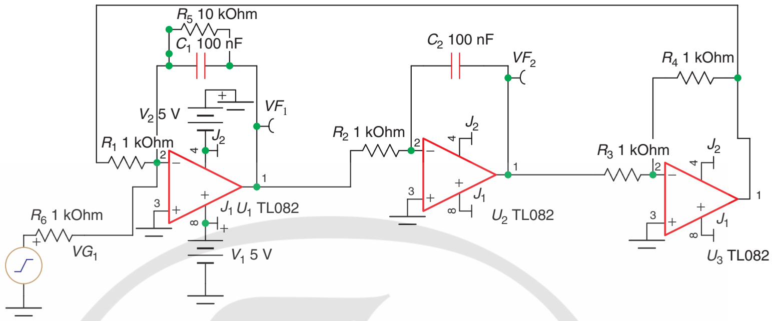


Figure 4-5(a) Tow-Thomas biquad filter

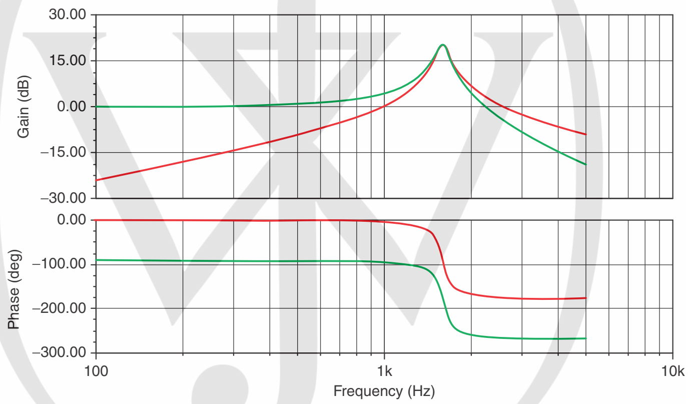
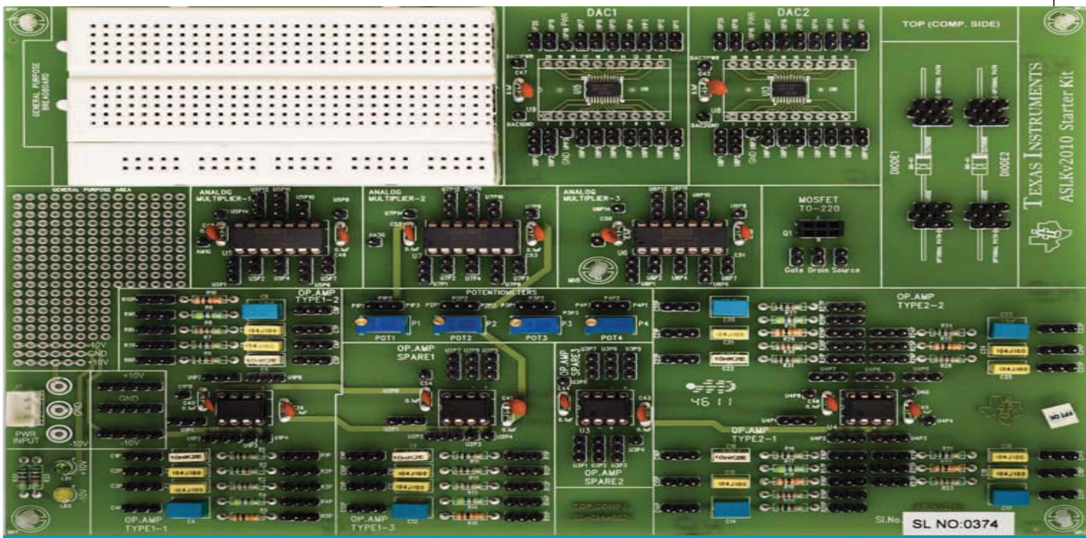


Figure 4-5(b) Frequency response of the filter

4.7 Related Circuits

The circuit described in Figure 4-5(b) is a *universal active filter circuit*. While this circuit can be built with `Op-Amps`, a specialized IC called **UAF42** from Texas Instruments provides the functionality of the universal active filter. We encourage you to use this circuit and understand its function. Datasheet of UAF42 is available from www.ti.com. Also refer to the application notes [7], [12], and [13].

Chapter Five



Experiment 5 Self-tuned Filter

5.1 Goal of the Experiment

The goal of this experiment is to learn the concept of *tuning* a filter. The idea is to adjust the *RC* time constants of the filter so that given in-phase response of a Low Pass filter, the output phase w.r.t. input is exactly 90° at the incoming frequency. This principle is utilized in distortion analyzers and spectrum analyzers. Such self-tuned filters are used to lock on to the fundamental frequency and harmonics of the input.

5.2 Brief Theory and Motivation

In order to design self-tuned filters and other analog systems in subsequent experiments, we need to introduce one more building block, the analog multiplier. The reader will benefit from viewing the recorded lecture at [23]. In the `ASLKv2010 Starter` kit, we have used the MPY634 analog multiplier from Texas Instruments. Figure 5-1 shows the symbol of an analog multiplier. In our experiments, we will use $V_r = 10$ V. We also show the output of the multiplier when two sinusoidal waveforms are multiplied; note that the output of the multiplier depends on the phase difference between the two inputs and can, therefore, be used as a measure of the phase difference.

$$V_o = V_{\text{offset}} + K_x \times V_x + K_y \times V_y + K_o \times V_x \times V_y + \xi \tag{5.1}$$

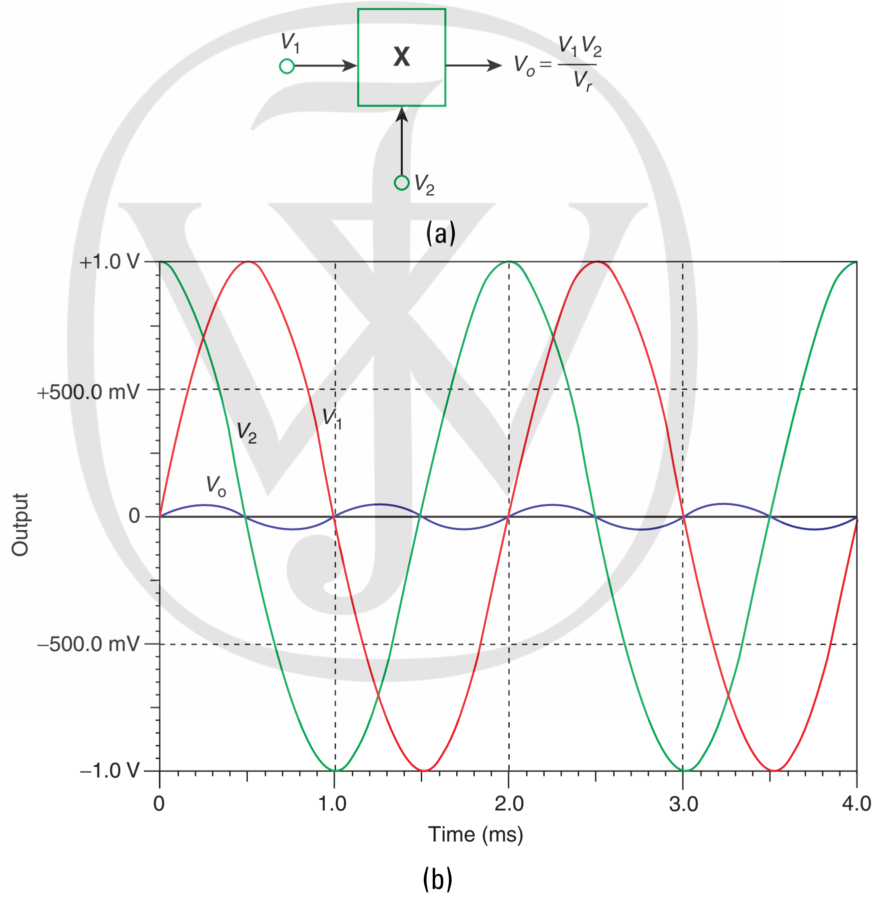


Figure 5-1 (a) Symbol of an analog multiplier; (b) multiplier as a phase detector

where ξ is a non-linear term in V_x and V_y. K_x and K_y are called *feedthrough components* and K_o is called the *normalizing component*. We define

$$V_r = \frac{1}{K_o}$$

For a precision multiplier, $V_r \le V_x$ and $V_y \le V_r$, where V_r is the parameter defined above. Hence, for precision amplifiers, $V_o = V_x \times V_y / V_r$.

In Experiment 4, if we replace the integrator with a multiplier followed by integrator, then the circuit becomes a *Voltage-Controlled Filter* (or a *Voltage-Controlled Phase Generator*) shown in Figure 5-2. This forms the basic circuit for self tuned filter. See Figure 5-3(a). The output of the self-tuned filter for a square-wave input, including the control voltage waveform, is shown in Figure 5-4. The figure brings out the aspect of automatic control and self-tuning. A simpler version of the voltage-controlled phase generator, which can be part of a self-tuned filter, is shown in Figure 5-3(b). You may use this simpler circuit which uses only two `Op-Amps`; note that the circuit of Figure 5-3(a) uses four `Op-Amps`. In the simpler circuit, you can study the variation of the phase in direct proportion to V_{ref} for a given sine-wave input frequency.

5.2.1 Multiplier as a Phase Detector

In the circuit of Figure 5-1, assume that

$$V_x = V_p \sin(\omega t) \tag{5.2}$$

$$V_y = V_p' \sin(\omega t + \phi) \tag{5.3}$$

Then the output of the multiplier is

$$V_o = \frac{V_p V_p'}{2V_r} \cdot [\cos\phi - \cos(\omega t + \phi)] \tag{5.4}$$

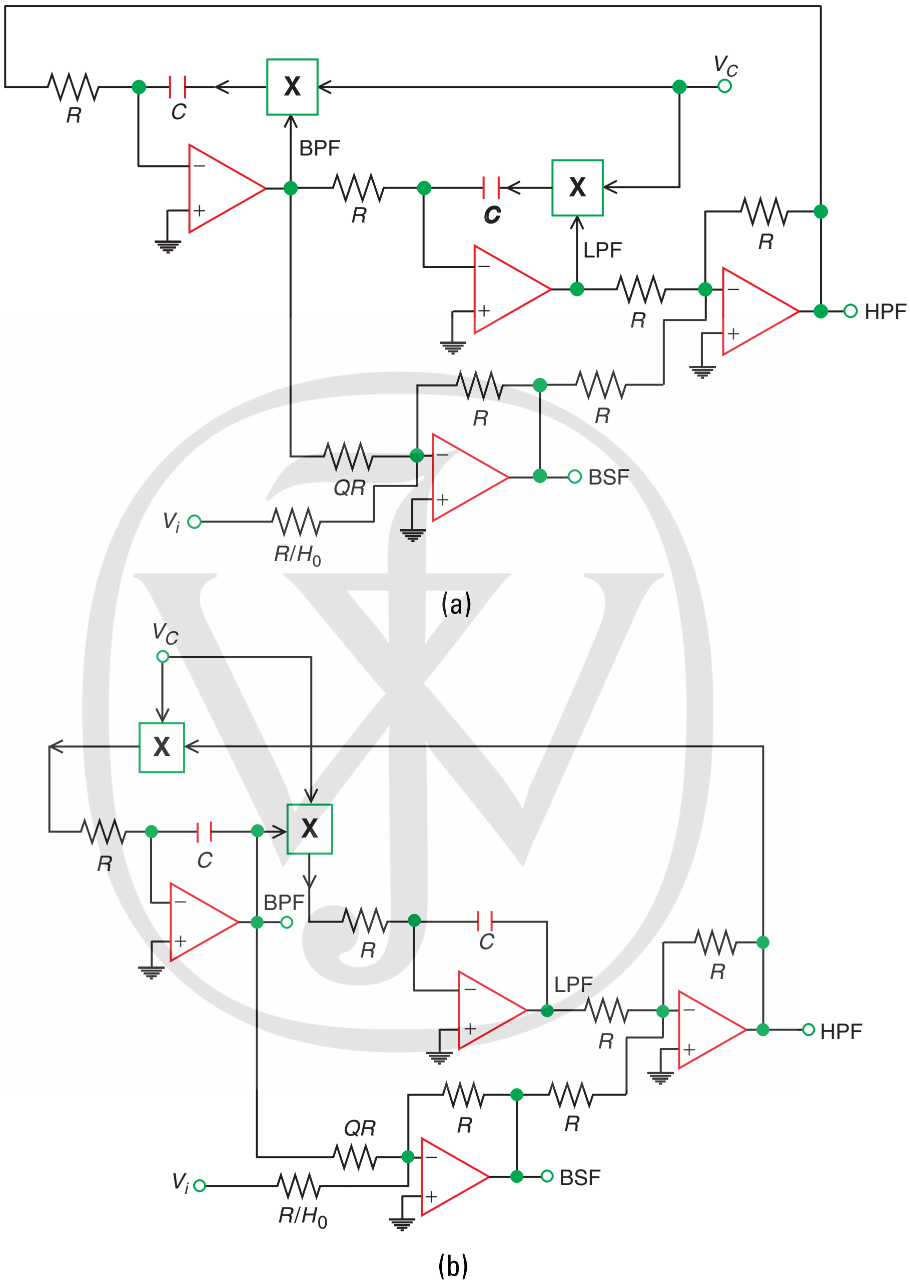
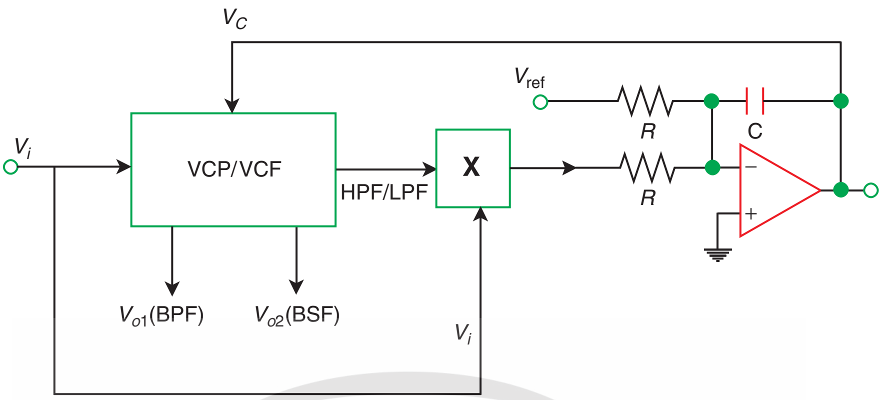


Figure 5-2 Voltage controlled filter with frequency (a) $\propto \frac{1}{V_c}$; (b) $\propto V_c$



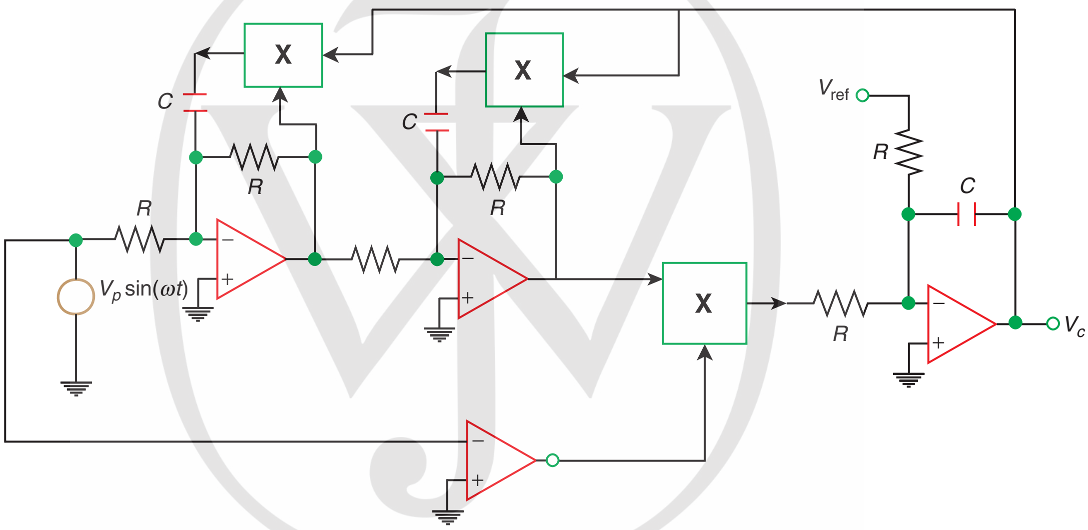


Figure 5-3 (a) A self-tuned filter based on a voltage-controlled filter or voltage-controlled phase generator; (b) a simple voltage-controlled phase generator that can become part of a self-tuned filter

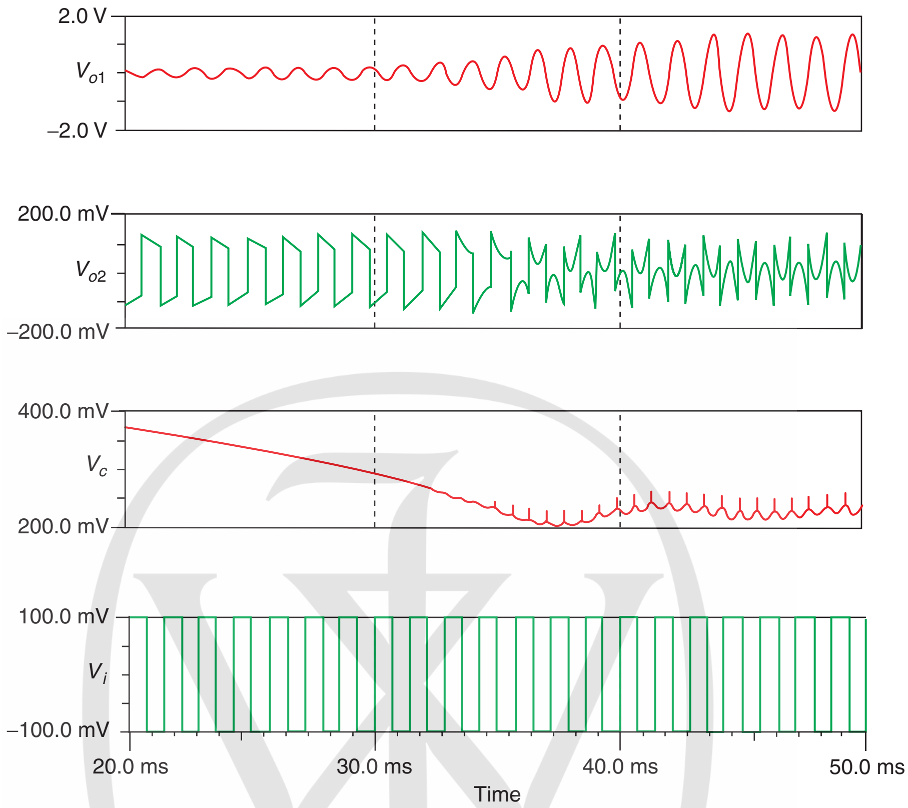


Figure 5-4 Output of the self-tuned filter based on simulation. V_{o1} corresponds to BPF, V_{o2} corresponds to BSF, V_c is the control voltage and V_i is the input voltage

After passing through the LPF, the high frequency component gets filtered out and only the average value of output V_{av} remains.

$$V_{av} = \frac{V_p V_p'}{2V_r} \cos\phi \tag{5.5}$$

$$K_{pd} = \frac{dV_{av}}{d\phi} \tag{5.6}$$

K_{pd} is called the phase detector sensitivity and is measured in Volts/radians.

Texas Lab Manual: "tlm" - 2012/8/14 - 14:55 - page 58 - #82

For ϕ = 90°, V_{av} becomes 0. This information is used to tune the voltage-controlled filter (VCF) automatically. ω_0 of the VCF is given by

$$\omega_0 = \frac{V_r}{V_c \cdot RC}$$

Therefore,

$$\frac{d\omega_0}{dV_c} = \frac{-V_r}{{V_c}^2 \cdot RC} = \omega_0 / V_c$$

The sensitivity of VCF is $\dfrac{d\phi}{dV_c}$ radians/sec/Volts. Now,

$$\frac{d\phi}{dV_c} = \frac{d\phi}{d\omega_0} \cdot \frac{d\omega_0}{dV_c}$$

If we consider the low-pass output, then

$$\frac{V_o}{V_i} = \frac{+H_0}{\left(1 + \dfrac{s}{\omega_0 Q} + \dfrac{s^2}{\omega_0^2}\right)}$$

$$\phi = \tan^{-1}\left[\frac{\left(\dfrac{\omega_r}{\omega_0 Q}\right)}{\left(1 - \dfrac{\omega_r}{\omega_0}\right)^2}\right]$$

$$\frac{d\phi}{d\omega_0} = \frac{2Q}{\omega_0}$$

Hence, sensitivity of VCF (K_{VCF}) is equal to $\dfrac{d\phi}{dV_c} = -2Q/V_c$.

For varying input frequency the output phase will always lock to the input phase with 90° phase difference between the two if $V_{av} = 0$.

Table 5-1 Variation of output amplitude with input frequency

S. No.	Input Voltage =	
	Input Frequency	Output Amplitude
1		
2		
3		
4		

5.3 Specification

Assuming that the input frequency is 1 kHz, design a high-Q BPF whose center frequency gets tuned to 1 kHz.

5.4 Measurements to be Taken

Apply a square-wave input and observe the amplitude of the Band Pass output for fundamental and its harmonics.

5.5 What Should you Submit

1. Simulate the circuits and obtain the transient response of the system.
2. Take the plots of transient response from oscilloscope and compare them with simulation results.
3. Measure the output amplitude of the fundamental (Band Pass output) at varying input frequency at fixed input amplitude.

Output amplitude should remain constant for varying input frequency within the lock range of the system.

5.6 Exercises

1. Determine the lock range of the self-tuned filter you designed. The lock range is defined as the range of input frequencies where the amplitude of the output voltage remains constant at $H_0 \times Q \times V_p$.

2. Repeat the experiment above with other periodic input waveforms such as the triangular waveform.

*3. A self-tuned filter is shown in Figure 5-5(b). Determine its lock range. Estimate the output at VF_1 and the control voltage VF_3 for a square wave input VG_1 of 0.1 V magnitude. Repeat for $VG_1 = 0.2$ V.

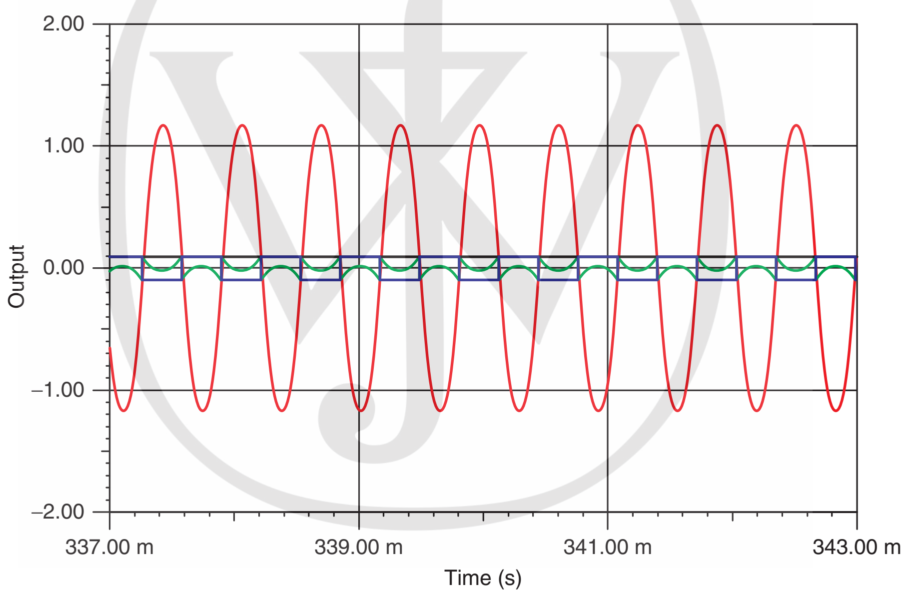


Figure 5-5(a) Simulation of the self-tuned filter shown in Figure 5-5(b) when $VG_1 = 0.1$ V

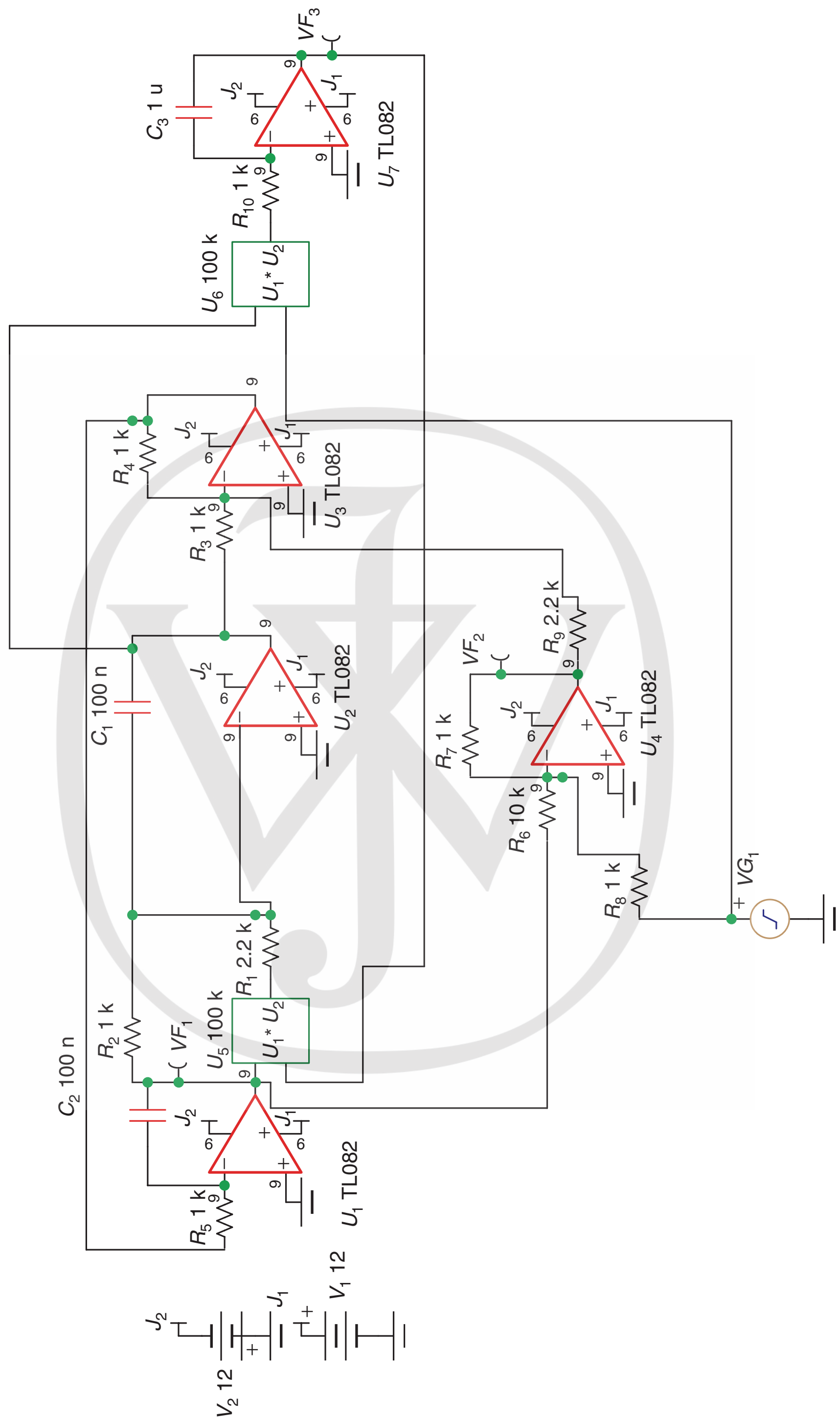
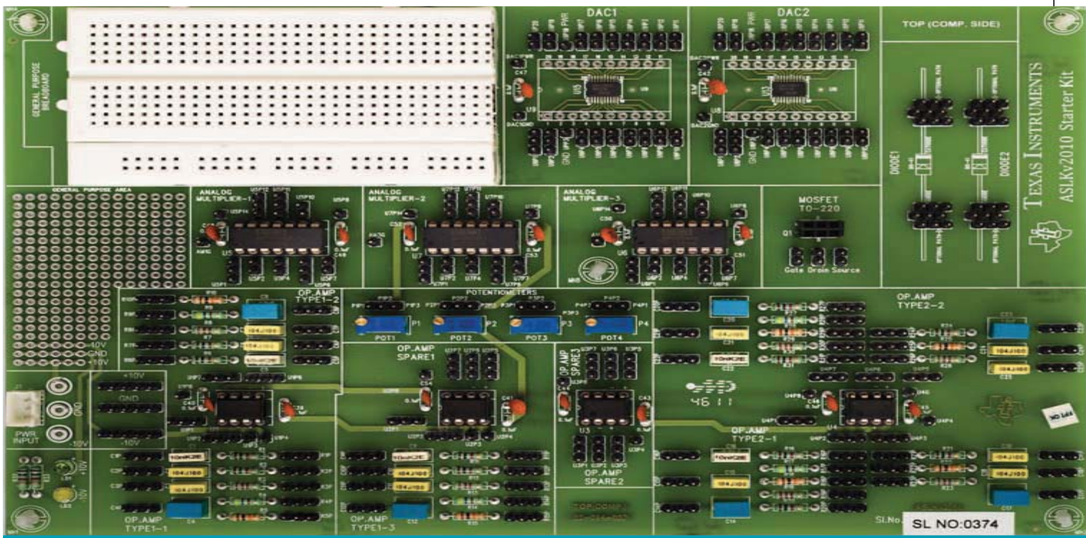


Figure 5-5(b) Self-tuned filter

5.7 Related ICs

Texas Instruments also manufactures the following related ICs – Voltage-controlled amplifiers (e.g. VCA820) and multiplying DAC (e.g. DAC7821) that can be used in place of analog multiplier. Refer to www.ti.com for application notes.

Chapter Six



Experiment 6 Function Generator and Voltage-Controlled Oscillator

6.1 Goal of the Experiment

The goal of this experiment is to design and build a function generator capable of generating a square wave and a triangular wave of a known frequency f. We will also convert a function generator to a *Voltage-Controlled Oscillator* which is a versatile building block that finds numerous applications.

6.2 Brief Theory and Motivation

The function generator circuit consists of a feedback loop, which includes a Schmitt trigger and an integrator. Recall that the Schmitt trigger is a two-bit A/D converter (at $\pm V_{ss}$ levels). If the integrator in a function generator is replaced by a combination of a

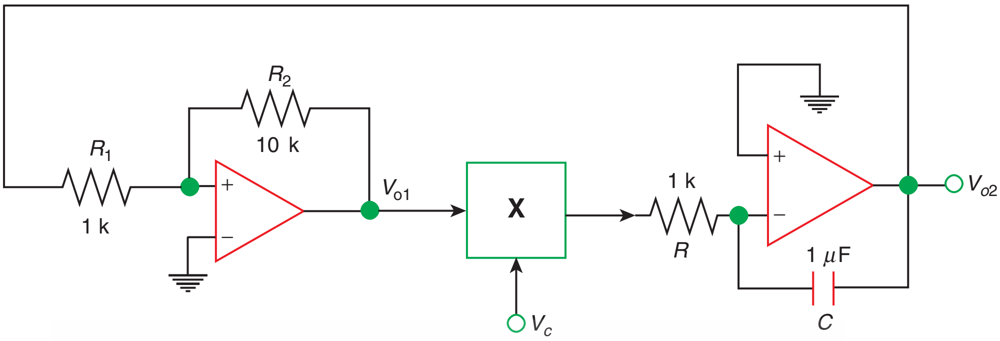


Figure 6-1 Voltage-Controlled Oscillator (VCO)

multiplier and an integrator, we get a *Voltage-Controlled Oscillator* (VCO) as shown in Figure 6-1. You will benefit from listening to the recorded video lectures from [28]. The output of the VCO is shown in Figure 6-2.

The function generator produces a square wave at the Schmitt trigger output and a triangular wave at the integrator output with the frequency of oscillation equal to $f = (1/4RC) \cdot (R_2/R_1)$. The function generator circuit can be converted as a linear VCO by using the multiplier integrator combination as shown in Figure 6-1.

The frequency of oscillation of the VCO becomes

$$f' = \frac{V_c \cdot R_2}{4 \cdot RC \cdot V_r \cdot R_1}$$

K_{VCO}, the sensitivity of the VCO, is an important parameter and is given by

$$K_{VCO} = \frac{df'}{dV_c} = \frac{R_2}{4RC \cdot V_r R_1} = \frac{f}{V_c} \text{Hz/Volts} \tag{6.1}$$

VCO is an important analog circuit and finds many applications. It is used in the generation of FSK/FM waveforms and constitutes the “modulator” part of the MODEM. In this role, the VCO is also called “mod of modem”. As a VCO, it finds use in the Phase Locked Loop (PLL) which we will study in Chapter 7. The VCO can also be used as a reference oscillator for a Class-D amplifier and the Switched Mode Power Supply (SMPS).

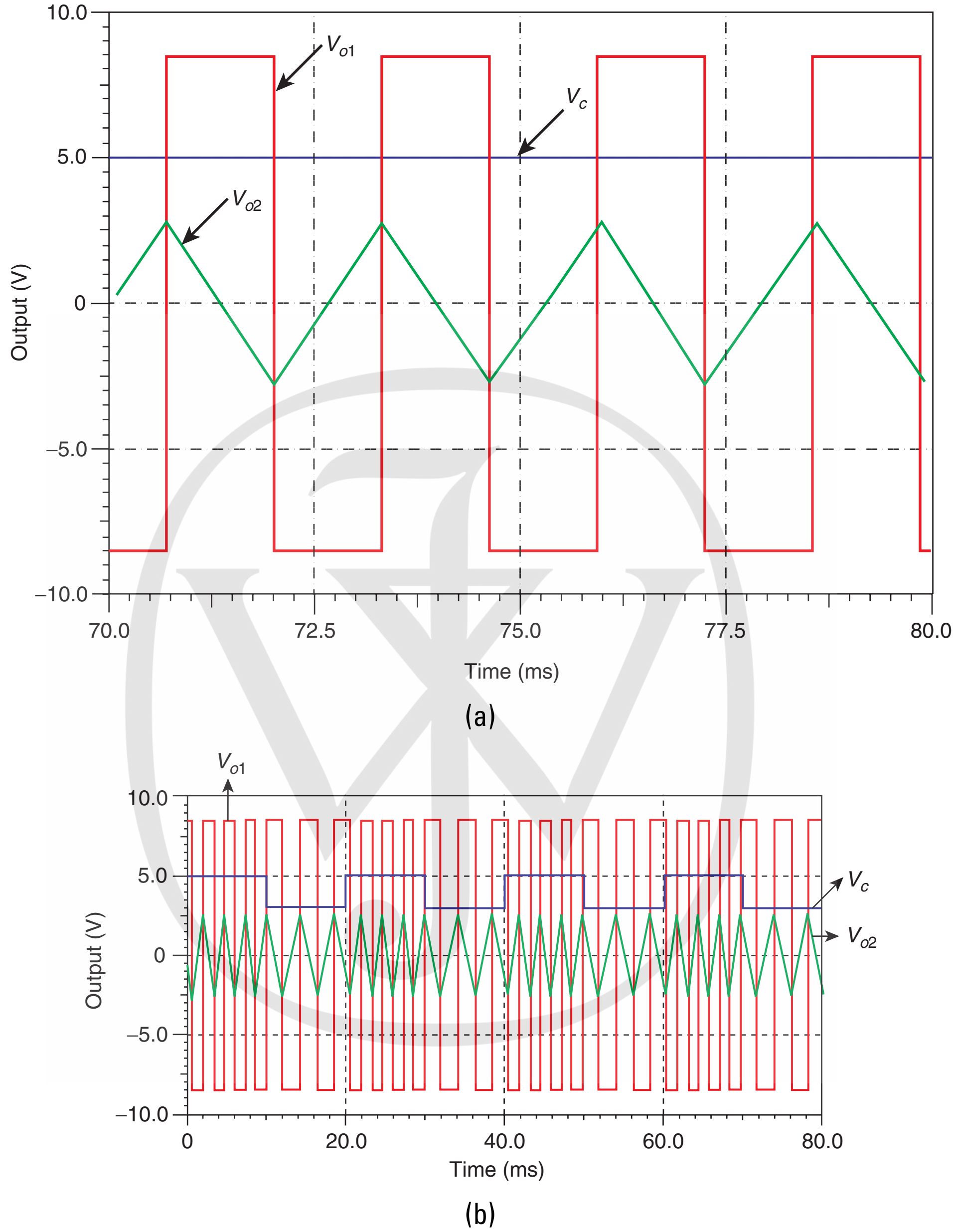


Figure 6-2 Simulation outputs for (a) function generator; (b) FSK generator

6.3 Specifications

Design a function generator to generate both square and triangular waveforms for a frequency of 1 kHz.

6.4 Measurements to be Taken

Determine the frequency of oscillations of square and triangular waves. Theoretically, the frequency of oscillation should be

$$f = \frac{1}{4RC} \times (R_2/R_1)$$

Convert the function generator into a VCO. Measure the sensitivity of the VCO, defined as $\frac{df}{dV_c}$.

6.5 What Should you Submit

1. Simulate the circuits and obtain the print-out of the waveforms generated by the function generator.
2. Build the function generator in `ASLKv2010 Starter` kit and observe the waveforms generated by the circuit on an oscilloscope. Compare the results with simulation results.
3. Vary the control voltage of the VCO and see its effect on the frequency of the output waveform. Measure the sensitivity (K_{VCO}) of the VCO. Use Table 6-1 to note your readings and compute the sensitivity.

Table 6-1 Change in frequency as a function of control voltage

S. No.	Control Voltage (V_c)	Change in Frequency
1		
2		
3		
4		

6.6 Exercises

1. Design a function generator that can generate square wave and triangular wave outputs of 10 kHz frequency.
2. Apply 1 V, 1 kHz square wave over 2 V DC and observe the FSK for a VCO designed for 10 kHz frequency.
3. *For the function generator of Figure 6-3(a) which uses Deboo's integrator and an inverting Schmitt trigger, determine the frequency of oscillation. Transform this circuit into a VCO using an analog multiplier.

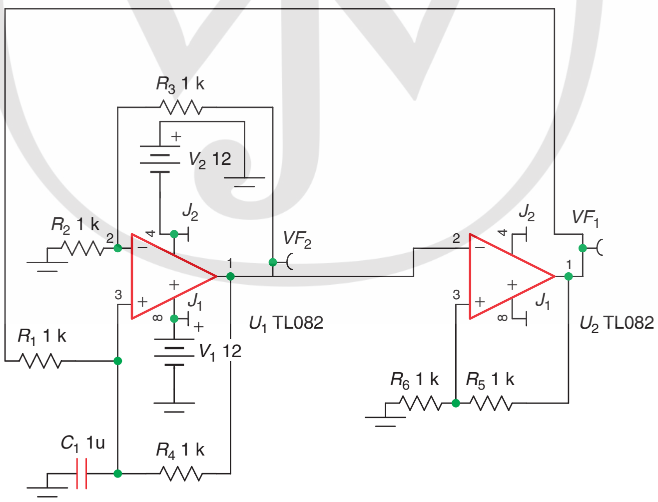


Figure 6-3(a) Function generator

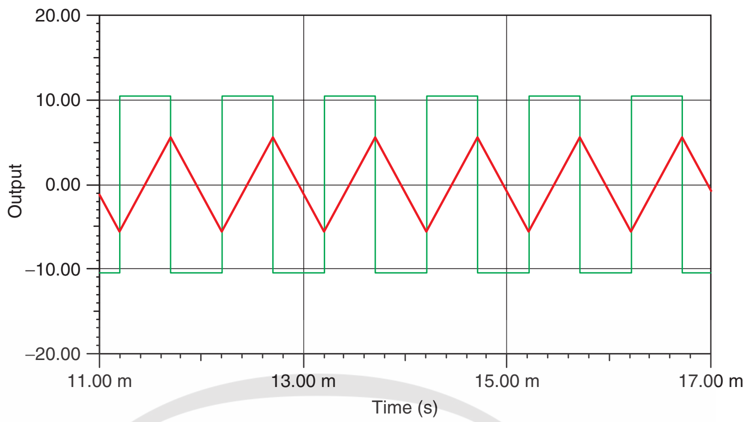


Figure 6-3(b) Simulation of the function generator of Figure 6-3(a)

*4 (a) A digitally controlled oscillator (DCO) is shown in Figure 6-4. The frequency of oscillation is given by

$$f = \frac{1}{4RC}\left(1 + \frac{R_1}{R_2}\right) \cdot \frac{\sum_{0}^{11} A_0 2^n}{4096}$$

$R = 1$ k and $C = 1\ \mu$F. $R_1 = R_2 = 1$ k. Determine the maximum and minimum frequency of oscillation in the linear range.

(b) Design a digitally controlled Band Pass filter with $Q = 10$ using the same integrator with multiplying DAC.

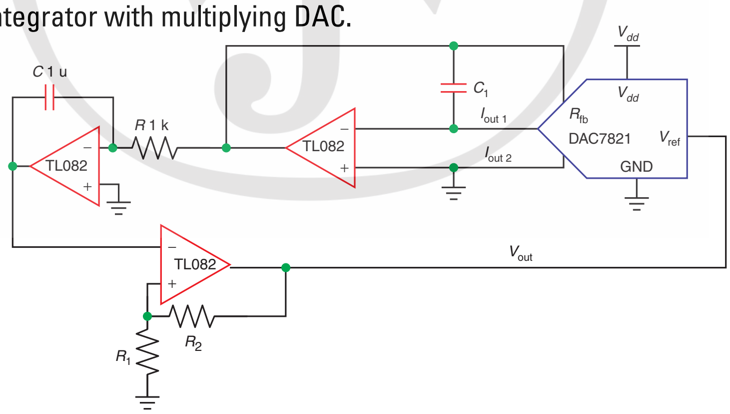
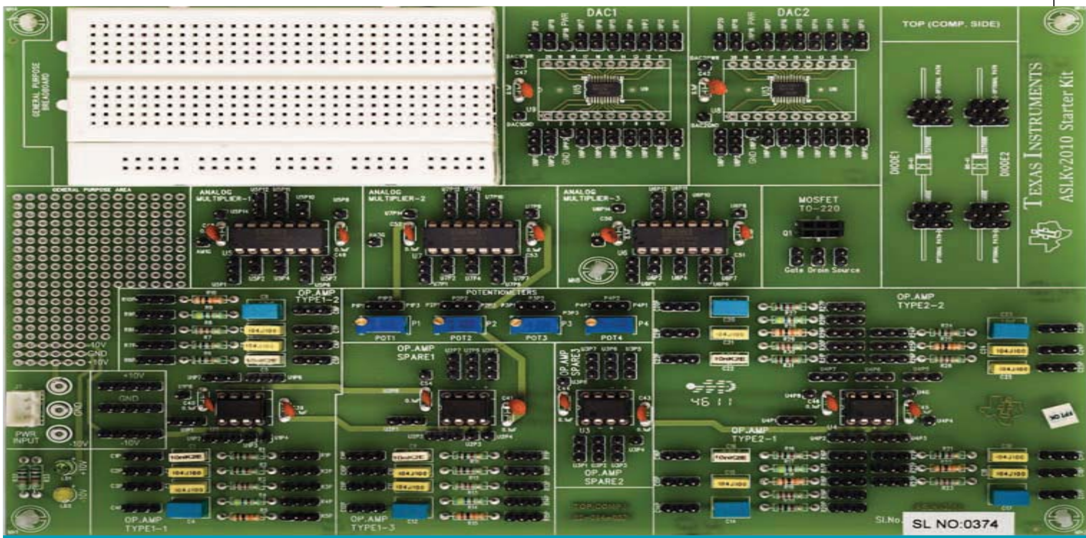


Figure 6-4 Digitally Controlled Oscillator (DCO)

Chapter Seven



Experiment 7 Phase Locked Loop

7.1 Goal of the Experiment

The goal of this experiment is to make you aware of the functionality of the *Phase Locked Loop,* commonly referred to as PLL. The PLL is mainly used for generating stable, high-frequency clocks in the 100 MHz – GHz range.

7.2 Brief Theory and Motivation

Crystals can be used to generate stable clocks in the range of a few hundreds of kilohertz to a few megahertz. If we need stable clocks of much larger frequency, we can use the clock waveform from the crystal source as a *reference clock* and additional analog circuits to multiply the frequency of the reference clock. Such a circuit is called a Phase Locked Loop. The reader will benefit from viewing the recorded lecture at [24]. The PLL uses the same concept that was introduced earlier in this lab, namely, self-tuned filter (Experiment 5). If we replace the voltage-controlled phase generator, *voltage-controlled filter* with a VCO, we obtain a PLL. This is shown in Figure 7-1.

The sensitivity of the PLL is given by K_{VCO}:

$$K_{\mathrm{VCO}} = \frac{d\omega}{dV_c} \tag{7.1}$$

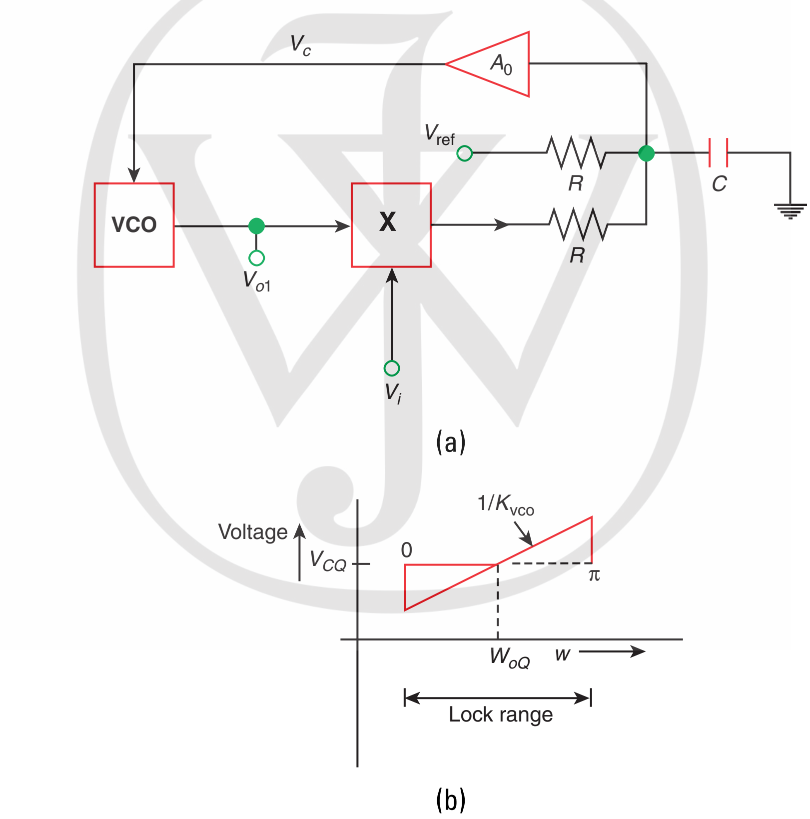


Figure 7-1 (a) Phase Locked Loop (PLL) circuit; (b) characteristics of the PLL

Here, $\omega = V_c/4V_r \cdot RC$ is the frequency of oscillation of the VCO. Therefore,

$$K_{VCO} = \frac{d\omega}{dV_c} = \frac{1}{4V_r \cdot RC} = \omega/V_c$$

When no input voltage is applied to the system, the system oscillates at the free-running frequency of the VCO, given by ω_{0Q} with corresponding control voltage of V_{CQ}. If an input voltage V_i with the frequency same as ω_{0Q} is applied, the PLL will continue to run at the free-running frequency and the phase difference between the two signals V_0 and V_i gets adjusted to 90° since V_c is 0. This was explained in Chapter 5. As the frequency of input signal is changed, the control voltage will change correspondingly, so as to lock the frequency of the output to the input frequency. As a result, there the phase difference between the input and output signals shifts away from 90°. The range of input frequencies for which the output frequency gets locked to the input frequency is called the **lock range** of the system. If K_{pd} denotes the sensitivity of the phase detector, the lock range is given by

$$\text{Lock Range} = K_{pd} \times \frac{\pi}{2} \times A_0 \times K_{VCO} \tag{7.2}$$

on either side of ω_{0Q}.

7.3 Specifications

Design a PLL to get locked to frequency of 1.59 kHz when the free-running frequency is 1.59 kHz.

7.4 Measurements to be Taken

1. Measure the lock range of the system.
2. Measure the change in the phase of the output signal as input frequency is varied within the lock range.

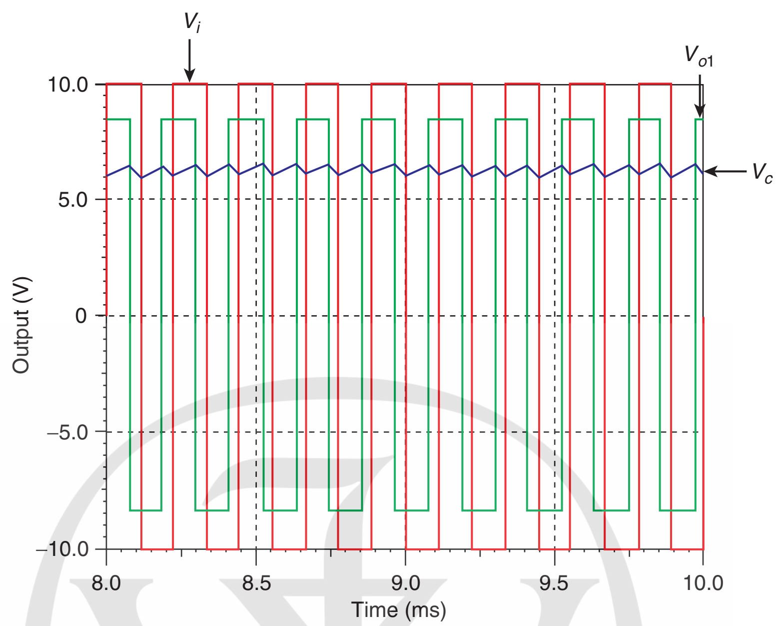


Figure 7-2 Sample output waveform for the Phase Locked Loop (PLL) for a square-wave input waveform

3. Vary the input frequency and obtain the change in the control voltage and plot the output. A sample output characteristic of the PLL is shown in Figure 7-2. In the diagram, the square wave of ± 10 V is the input waveform, the square wave of ± 8 V is the output waveform. The slow-varying waveform in thick line is the control voltage.

7.5 What Should you Submit

1. Simulate the system and predict the output waveforms of the PLL.
2. Build the PLL system using `ASLKv2010 Starter` kit and take the plots of the output waveform on the oscilloscope. Compare the simulation results with

Table 7-1 Output phase as a function of input frequency

S. No.	Input Frequency	Output Phase
1		
2		
3		
4		

Table 7-2 Control voltage as a function of input frequency

S. No.	Input Frequency	Control Voltage
1		
2		
3		
4		

the actual waveforms. Observe what happens to the output frequency when the system is not locked. What do you see as the control voltage waveform then?

3. Measure the change in the phase of the output signal as input frequency is varied within the lock range.
4. Vary the input frequency and obtain the change in the control voltage. Use Table 7-2 to record your readings.

7.6 Exercises

*1. For the PLL/FLL shown in Figure 7-3(a), determine the free-running frequency. Determine the lock ranges when the input is a square-wave of amplitude 0.5 V. Repeat the experiment when the input amplitude is 1 V.

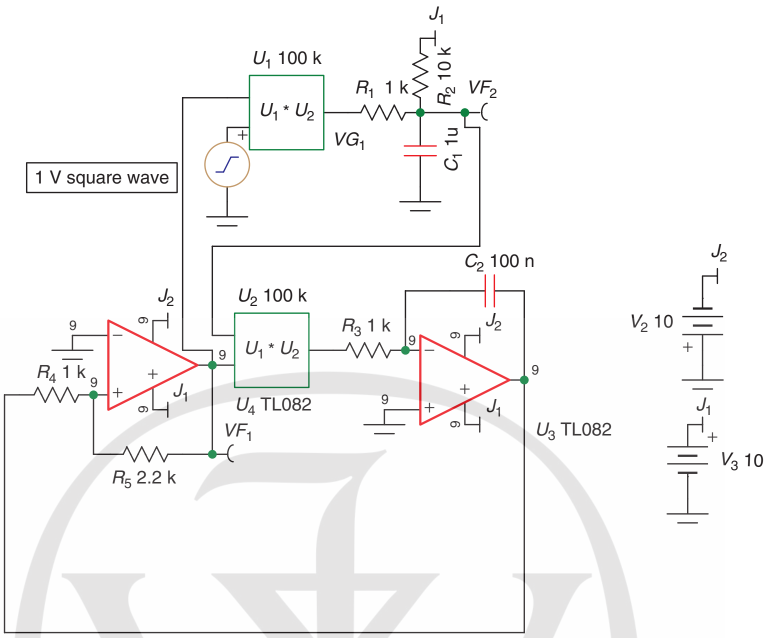


Figure 7-3(a) Phase locked loop

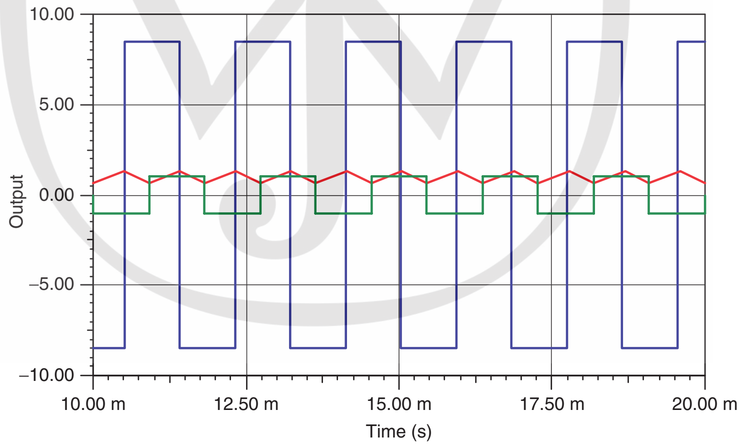


Figure 7-3(b) Simulation of the PLL at free-running frequency, when input amplitude is 1 V

Texas Lab Manual: "tlm" - 2012/8/14 - 14:55 - page 76 - #100

❷ Design a frequency synthesizer to generate a waveform of 1 MHz frequency from a 100 kHz crystal as shown in Figure 7-4.

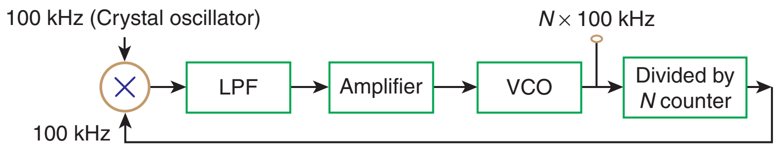
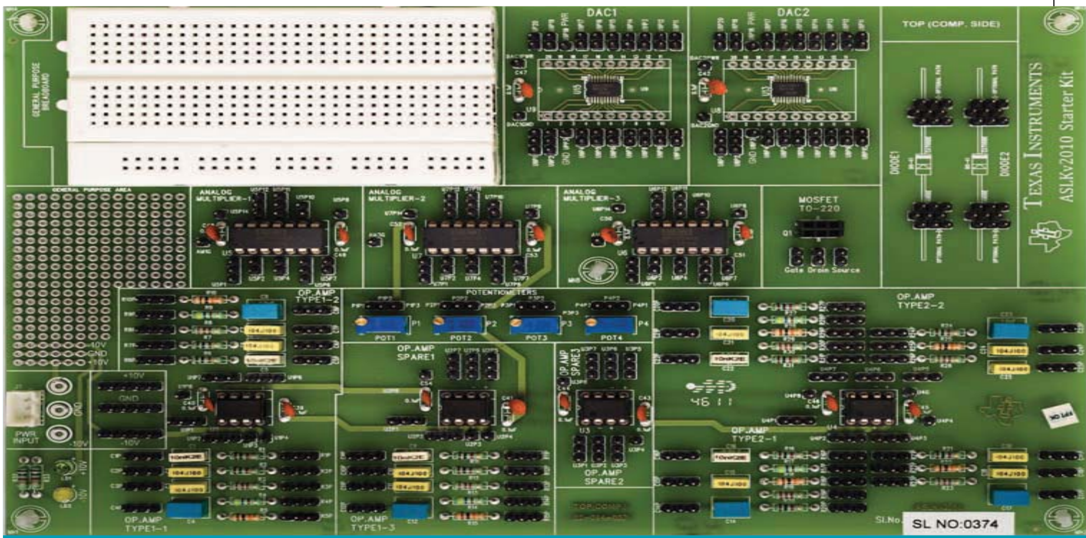


Figure 7-4 Block diagram of frequency optimizer



Chapter Eight

Experiment 8 Automatic Gain Control (AGC)/Automatic Volume Control (AVC)

8.1 Goal of the Experiment

In the front-end electronics of a system, we may require that the gain of the amplifier is adjustable, since the amplitude of the input keeps varying. Such a system can be designed using feedback. This experiment demonstrates one such system.

8.2 Brief Theory and Motivation

The reader will benefit from the recorded lectures at [27]. Another useful reference is the application note on *Automatic Level Controller for Speech Signals using PID Controllers* [2].

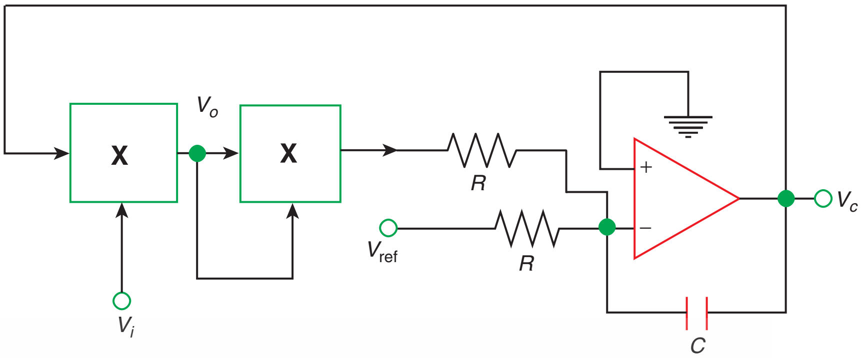


Figure 8-1 Automatic Gain Control (AGC)/Automatic Volume Control (AVC)

In the signal chain of an electronic system, the output of the sensor can vary depending on the strength of the input. To adapt to wide variations in the magnitude of the input, we can design the amplifier such that its gain can be adjusted dynamically. This is possible when the input signal has a narrow bandwidth and the control system is called *Automatic Gain Control* (AGC). Since we may wish to maintain the output voltage of the amplifier at a constant level, we also use the term *Automatic Volume Control* (AVC). Figure 8-1 shows an AGC system. The typical I/O characteristic of AGC/AVC system is shown in Figure 8-2. As shown in Figure 8-2, the output value of the system remains constant at $\sqrt{2V_r V_{ref}}$ beyond input voltage $V_{pi} = \sqrt{2V_r V_{ref}}$.

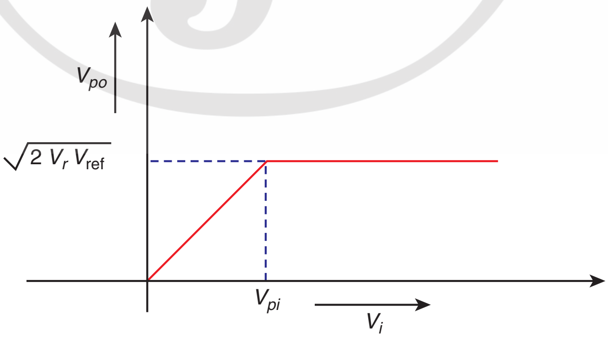


Figure 8-2 Input–output characteristics of AGC/AVC

8.3 Specification

Design an AGC/AVC system to maintain a peak amplitude of sine-wave output at 2 V.

8.4 Measurements to be Taken

Transfer Characteristics: Plot the input versus output characteristics for the AGC/AVC.

8.5 What Should you Submit

1. Simulate the system of Figure 8-1 and plot the output of the AGC system. Assume that the input comes from a function generator; use a sine-wave input of a single frequency.
2. Build the AGC system of Figure 8-1 using `ASLKv2010 Starter` kit. Note the output of the AGC system from the oscilloscope and compare the output with simulation result.
3. Plot the output as a function of input voltage. Take sufficient number of readings. Does the output remain constant as the magnitude of the input is increased? Beyond what value of the input voltage does the gain begin to stabilize? Use Table 8-1 to record your readings. We have included sample output waveform for the AGC system in Figure 8-3.

Table 8-1 Transfer characteristic of the AGC system

S. No.	Input Voltage	Output Voltage	Control Voltage
1			
2			
3			
4			

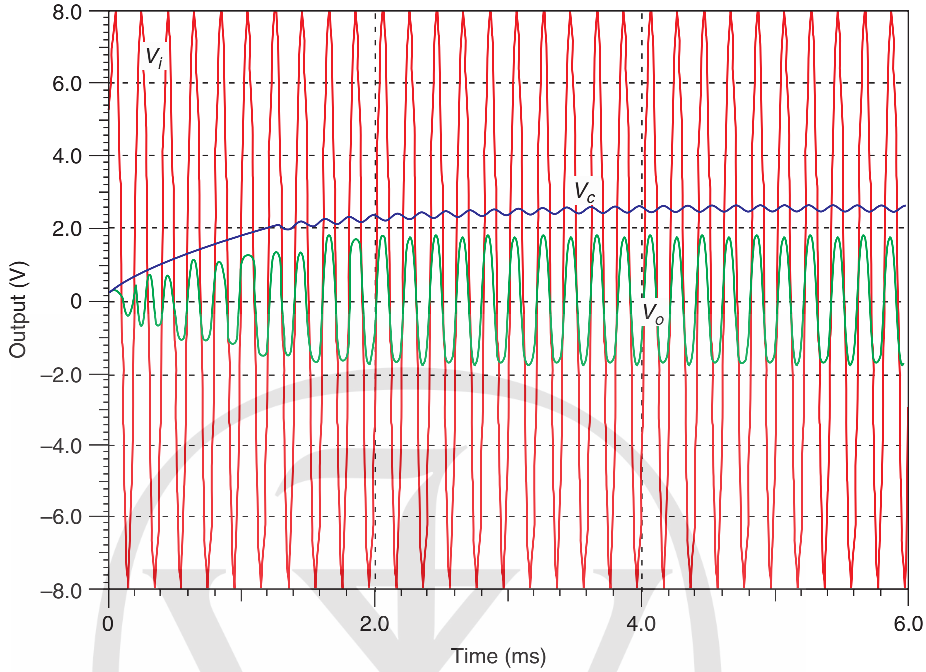


Figure 8-3 Output of AGC circuit

8.6 Exercises

1. Determine the lock range for the AGC that was built as part of the experiment. The lock range is defined as the range of input values for which output voltage remains constant.

*2. The AGC/AVC circuit of Figure 8-4(a) is designed for $V_1 = 0.2$ V. Determine the peak amplitude of the output VF_3 and control voltage VF_2 when the input $VG_1 = 0.1$ V. Repeat for $VG_1 = 1$ V, 2 V, 4 V, and 8 V.

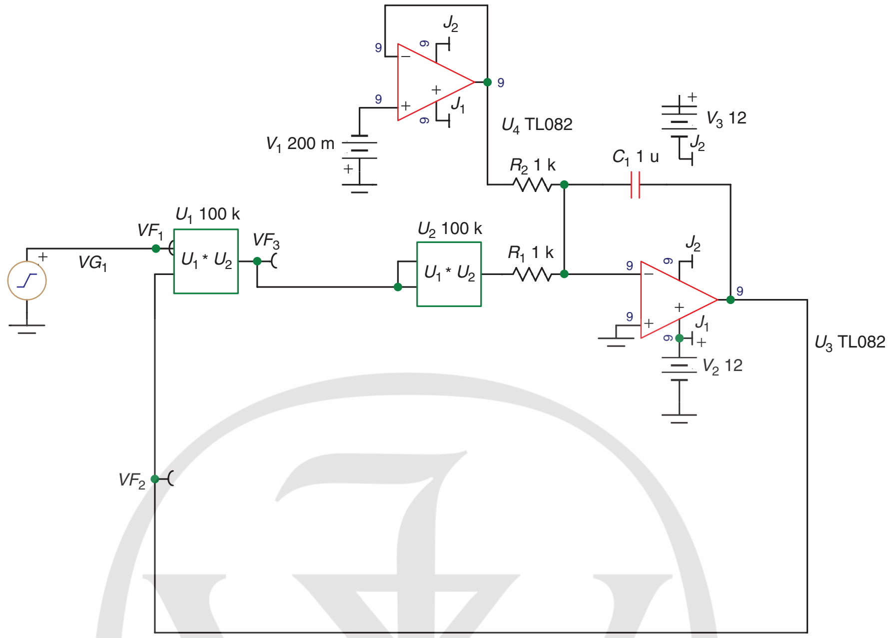


Figure 8-4(a) AGC circuit

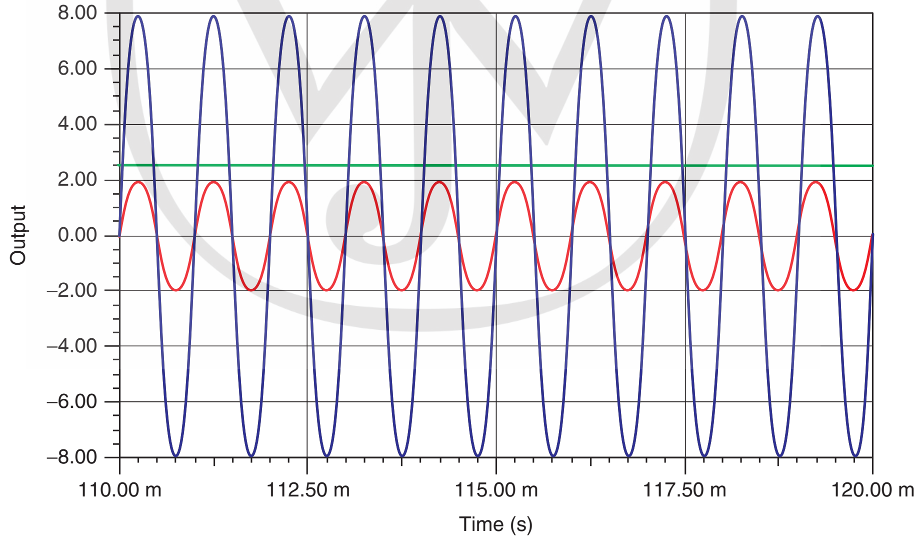
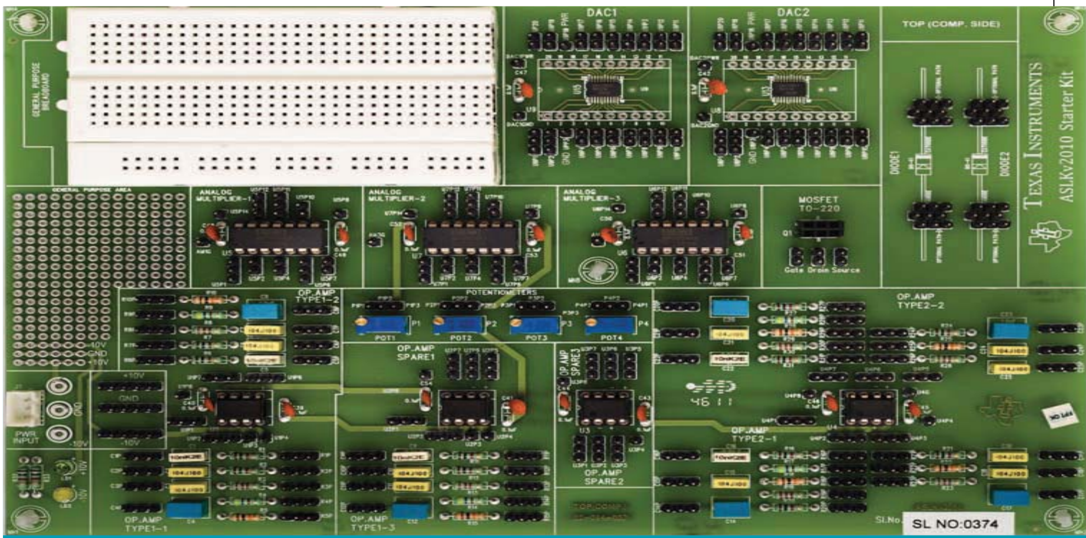


Figure 8-4(b) Simulation of the AGC circuit for output voltage = 2 V peak

Chapter Nine



Experiment 9 DC–DC Converter

9.1 Goal of the Experiment

The goal of this experiment is to design a DC–DC converter using a general-purpose `Op-Amp` and a comparator, and to study its characteristics. We also aim to study the characteristics of a DC–DC converter *integrated circuit*; we select the wide-input non-synchronous buck DC–DC converter TPS40200 from Texas Instruments. Our aim is to design a DC–DC converter with high efficiency using a general purpose `Op-Amp` for a variety of applications like Switched Mode Power Supply (SMPS), audio amplifier (Class-D Power Amplifier), etc.

9.2 Brief Theory and Motivation

The reader will benefit from viewing the recorded lecture at [26]. Also refer to the application note, *Design Considerations for Class-D Audio Power Amplifiers* [17].

The function generator, which we studied in Chapter 6, is a basic building block in a DC–DC converter. The triangular output of the function generator with peak amplitude V_p and frequency f is fed as an input to a comparator, whose second input comes from a reference voltage V_{ref}. The output of the comparator is a Pulse Width Modulation (PWM) waveform whose duty cycle is given by

$$\frac{\tau}{T} = \frac{1}{2}(1 - V_{\text{ref}}/V_p)$$

where $T = 1/f$ is the time period of the triangular waveform. The duty cycle is directly proportional to reference voltage V_{ref}. If we connect a lossless Low Pass filter (LC filter) at the output of the comparator, as shown in Figure 9-1, it is possible to get a stable DC voltage V_{av} given by

$$V_{av} = -V_{\text{ref}} \cdot V_{ss}/V_p \qquad (9.1)$$

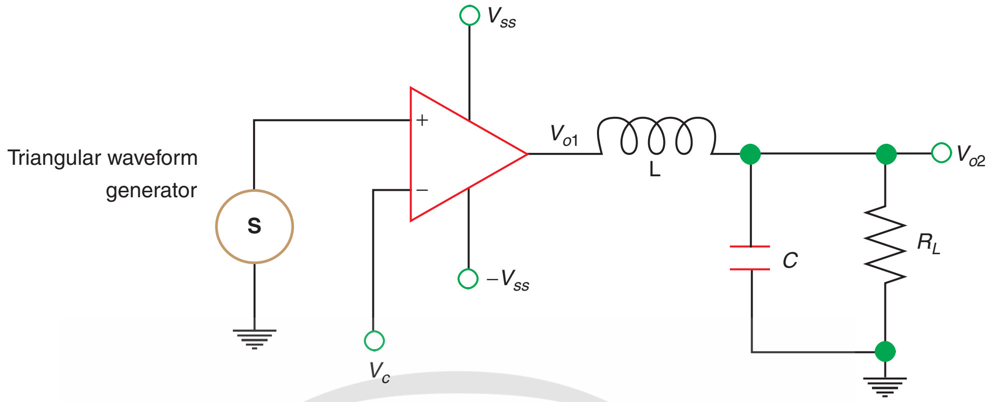
We thus get a converter with high conversion efficiency. We can also insert a PMOS switch in between the comparator and the LC filter to achieve Class-D operation.

9.3 Specification

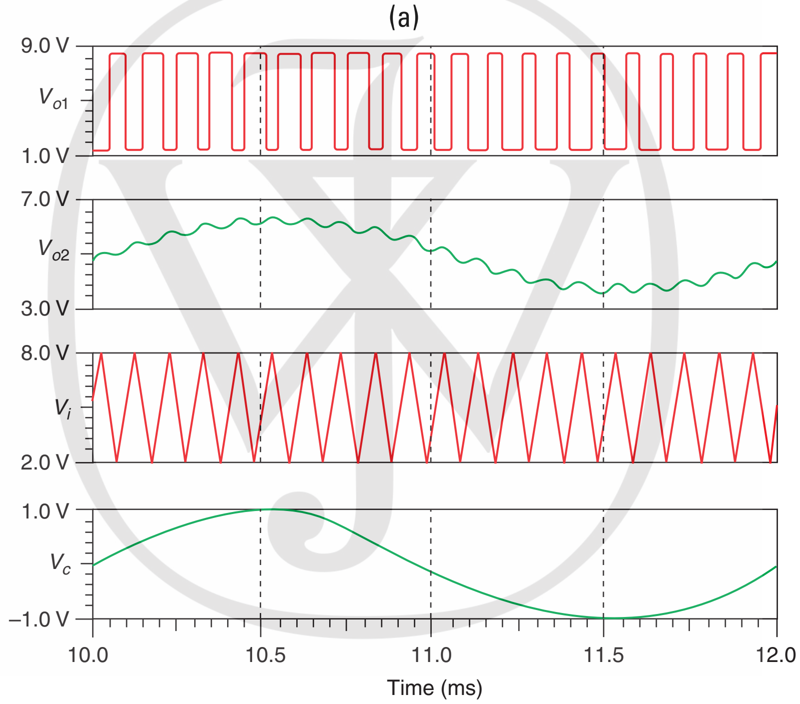
Design a DC–DC converter using a switching frequency of 10 kHz and 100 kHz using an available reference voltage, for an output voltage of 5 V.

9.4 What Should you Submit

1. Simulate the system and plot the output waveforms of the comparator and the Low Pass filter output as shown in Figure 9-1. We have included a DC–DC converter and typical simulation results in Figure 9-2. V_{ss} in the system is the unregulated input. V_o is the converted output.



(a)



(b)

Figure 9-1 (a) DC–DC converter; (b) waveforms from simulation

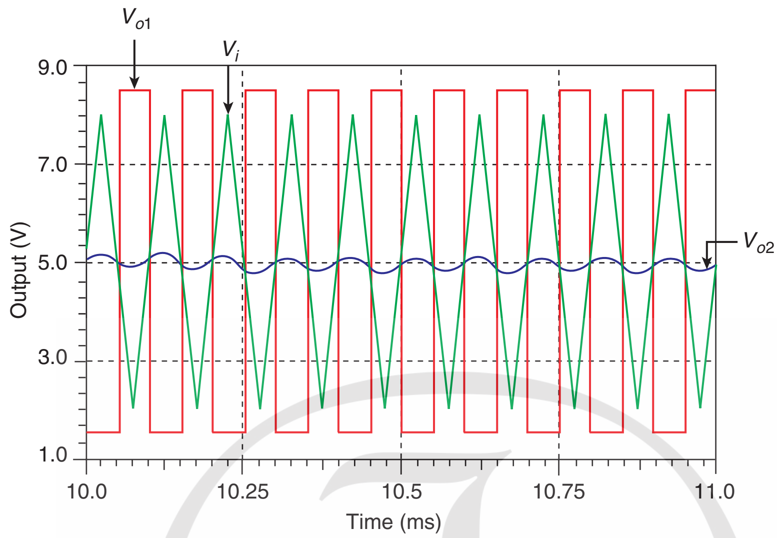


Figure 9-2 PWM and Class-D output waveforms

2. Build the DC–DC converter using `ASLKv2010 Starter` kit and observe the waveforms mentioned above. Compare with simulation results.
3. Plot the average output voltage V_{av} as a function of control voltage V_c and obtain the plot. Use a table similar to Table 9-1 to take your readings. Is the plot linear? Determine the peak-to-peak ripple at the output of the LPF.
4. Plot the duty cycle τ/T as a function of control voltage V_c. Use a table similar to Table 9-2 to take your readings. Is the plot linear?

Table 9-1 Variation of output voltage with control voltage in a DC–DC converter

S. No.	Control Voltage	Controlled Voltage
1		
2		
3		
4		

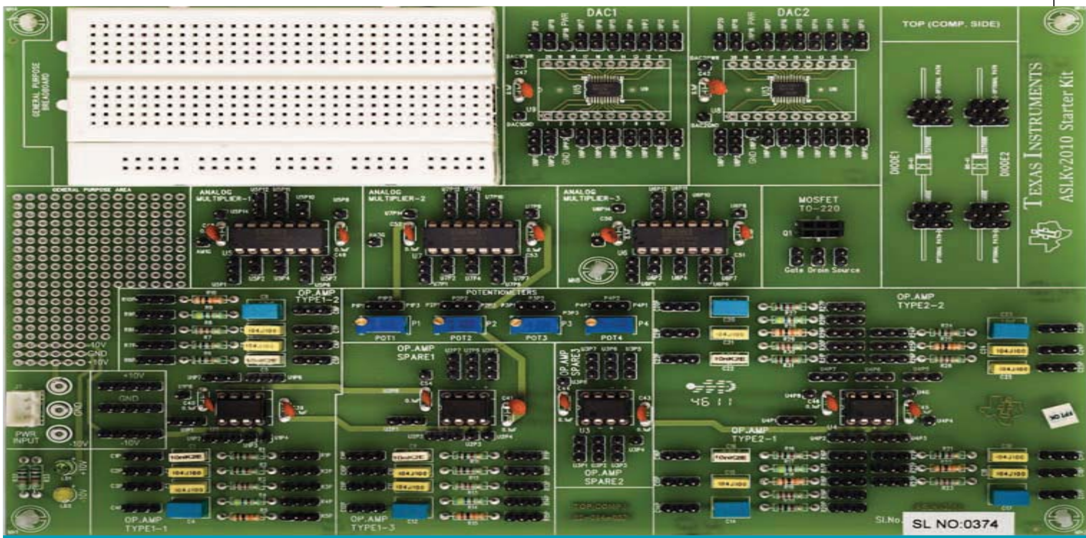
Table 9-2 Variation of duty cycle with control voltage in a DC–DC converter

S. No.	Control Voltage	Duty Cycle τ/T
1		
2		
3		
4		

9.5 Exercises

1. Explain how a PMOS switch can be used to achieve Class-D operation for the DC–DC converter system we studied. Show the block diagram. You may need a diode in your system – why?
2. Perform the same experiment with the specialized IC for DC–DC converter from Texas Instrument TPS40200 and compare the characteristics of both the systems.

Chapter Ten



Experiment 10 Low Dropout (LDO)/Linear Regulator

10.1 Goal of the Experiment

The goal of this experiment is to design a Low Dropout/Linear regulator using a general purpose `Op-Amp` and study its characteristics. We will also see that an integrated circuit family of regulators, called TLV700xx, is available for the purpose and study their characteristics. Our aim is to design a linear voltage regulator with high efficiency, used in *low noise*, *high efficiency* applications.

10.2 Brief Theory and Motivation

Please view the recorded lectures at [25] for a detailed description of voltage regulators. In the case of the DC–DC converter studied in the previous experiment, the

Texas Lab Manual: “tlm” - 2012/8/14 - 14:55 - page 91 - #115

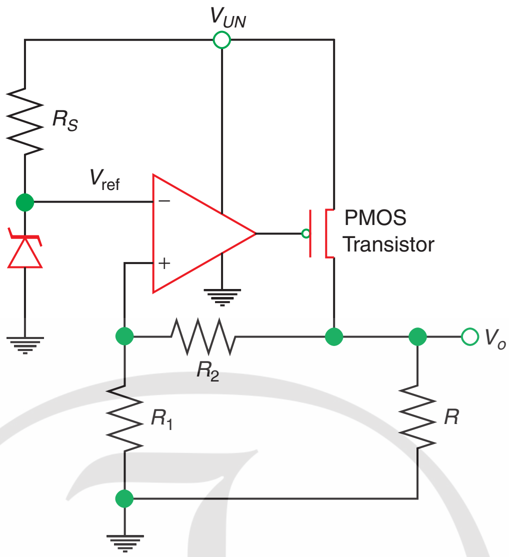


Figure 10-1 Low Dropout Regulator (LDO)

switching activity exemplified by the PWM waveform is a source of noise. As a result, DC–DC converter of the previous chapter is not suitable for *low noise* applications. An LDO is better suited in such cases. An LDO system is shown in Figure 10-1. It uses a PMOS current amplifier along with an `Op-Amp` so that power dissipation in `Op-Amp` and PMOS combination is minimal. The efficiency of the LDO, defined as the ratio of the output voltage to input voltage, is high. The regulated output voltage is given by

$$V_o = V_{\text{ref}}\left(1 + \frac{R_2}{R_1}\right) \tag{10.1}$$

10.3 Specifications

Generate a 3 V output when input voltage is varying from 4 V to 5 V.

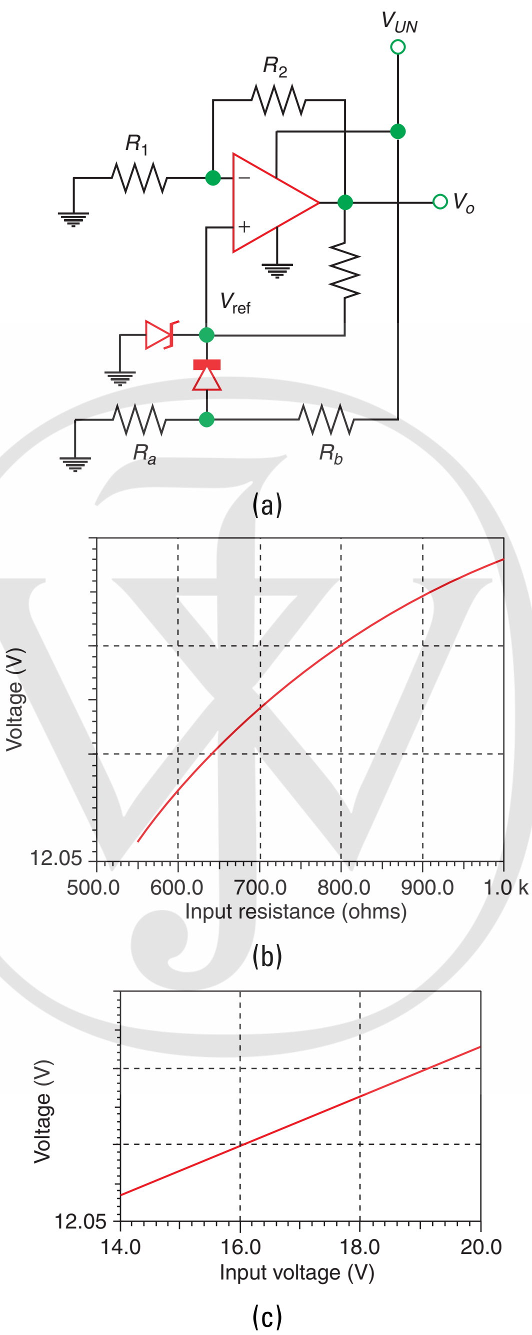


Figure 10-2 (a) A regulator system with startup; (b) load regulation output; (c) line regulation output

Table 10-1 Variation of load regulation with load current in an LDO

S. No.	Load Current	Output Voltage
1		
2		
3		
4		

10.4 Measurements to be Taken

1. Obtain the **Load Regulation:** Vary the load such that load current varies and obtain the output voltage; see the point till where output voltage remains constant. After that, the output will fall as the load current increases.
2. Obtain the **Ripple Rejection:** Apply the input ripple voltage and see the output ripple voltage; with the input ripple voltage, the output ripple voltage will rise.
3. Obtain the **Line Regulation:** Vary the input voltage and plot the output voltage as a function of the input voltage. Until the input reaches a certain value, the output voltage remains constant; after this point, the output voltage will rise as the input voltage is increased.
4. Calculate the **Output Impedance.**

10.5 What Should you Submit

1. Simulate the systems and compute the output characteristics, transfer characteristics and ripple rejection.
2. Take the plots of output characteristics, transfer characteristics and ripple rejection from the oscilloscope and compare them with simulation results.

Table 10-2 Variation of line regulation with input voltage in an LDO

S. No.	Input Voltage	Output Voltage
1		
2		
3		
4		

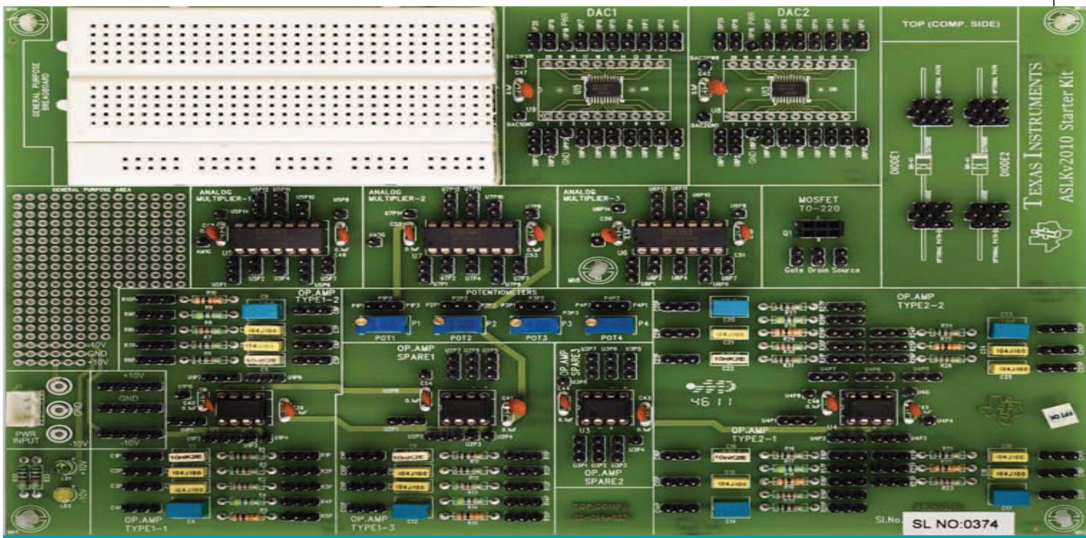
Table 10-3 Ripple rejection

S. No.	Ripple Input Voltage	Ripple Output Voltage
1		
2		
3		
4		

10.6 Exercises

1. Perform the same experiment with the specialized IC for LDO from Texas Instrument TLV700xx family and compare the characteristics of both the systems.

APPENDIX A



ICs used in ASLKv2010 Starter Kit

Texas Instruments Analog ICs used in ASLKv2010 Starter kit

A.1 TL082: JFET-Input Operational Amplifier

A.1.1 Features

1. Low power consumption.
2. Wide common-mode and differential voltage ranges.
3. Input bias and offset currents.
4. Output short-circuit protection.
5. Low total harmonic distortion: . . . 0.003% Typ.
6. High input impedance: . . . JFET-input stage.
7. Latch-up-free operation.

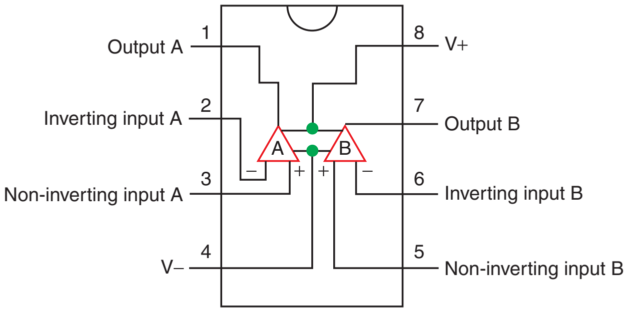


Figure A-1 TL082 – JFET-input operational amplifier

8. High slew rate: . . . 13 V/μs Typ.
9. Common-mode input voltage range includes V_{CC+}.

A.1.2 Applications

1. Instrumentation Amplifiers
2. Filters

A.1.3 Description

The TL08x JFET-input operational amplifier family is designed to offer a wider selection than any previously developed operational amplifier family. Each of these JFET-input operational amplifiers incorporates well-matched, high-voltage JFET and bipolar transistors in a monolithic integrated circuit. The devices feature high slew rates, low input bias and offset currents, and low offset voltage temperature coefficient. Offset adjustment and external compensation options are available within the TL08x family. The C-suffix devices are characterized for operation from 0°C to 70°C. The I-suffix devices are characterized for operation from −40°C to 85°C. The Q-suffix devices are characterized for operation from −40°C to 125°C.

A.1.4 Download Datasheet

`http://focus.ti.com/lit/ds/symlink/tl082.pdf`

A.2 MPY634: Wide-Bandwidth Analog Precision Multiplier

A.2.1 Features

1. Wide-bandwidth: 10 MHz Typ
2. 0.5% max four-quadrant accuracy
3. Internal wide-bandwidth `Op-Amp`

A.2.2 Applications

1. Precision analog signal processing.
2. Modulation and demodulation.
3. Voltage-controlled amplifiers.
4. Video signal processing.
5. Voltage-controlled filters and oscillators.

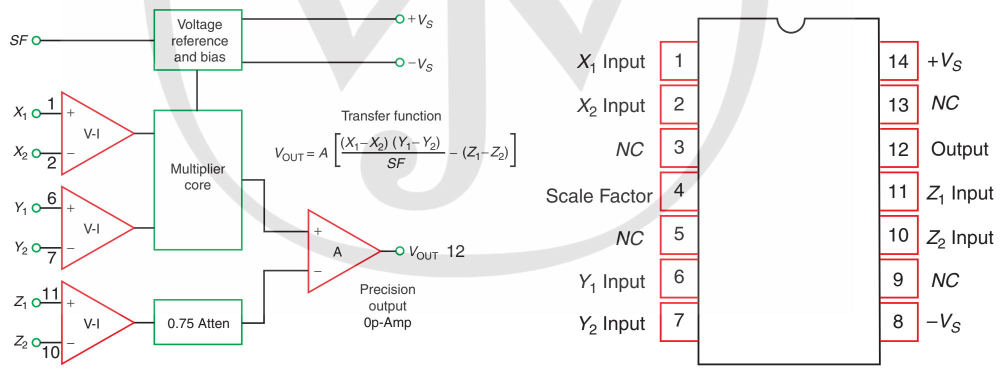


Figure A-2 MPY634 – Analog multiplier

Texas Lab Manual: "tlm" - 2012/8/14 - 14:55 - page 99 - #123

A.2.3 Description

The MPY634 is a wide bandwidth, high accuracy, four-quadrant analog multiplier. Its accurately laser-trimmed multiplier characteristics make it easy to use in a wide variety of applications with a minimum of external parts, often eliminating all external trimming. Its differential X, Y and Z inputs allow configuration as a multiplier, squarer, divider, square-rooter and other functions, while maintaining high accuracy. The wide bandwidth of this new design allows signal processing at IF, RF and video frequencies. The internal output amplifier of the MPY634 reduces design complexity compared to other high frequency multipliers and balanced modulator systems. It is capable of performing frequency mixing, balanced modulation and demodulation with excellent carrier rejection. An accurate internal voltage reference provides precise setting of the scale factor. The differential Z input allows user-selected scale factors from 0.1 to 10 using external feedback resistors.

A.2.4 Download Datasheet

`http://focus.ti.com/lit/ds/symlink/mpy634.pdf`

A.3 DAC 7821: 12 Bit, Parallel, Multiplying DAC

A.3.1 Features

1. 2.5 V to 5.5 V supply operation.
2. Fast parallel interface: 17 ns write cycle.
3. Update rate of 20.4 MSPS.
4. 10 MHz multiplying bandwidth.
5. 10 V input.
6. Low glitch energy: 5 nVs.

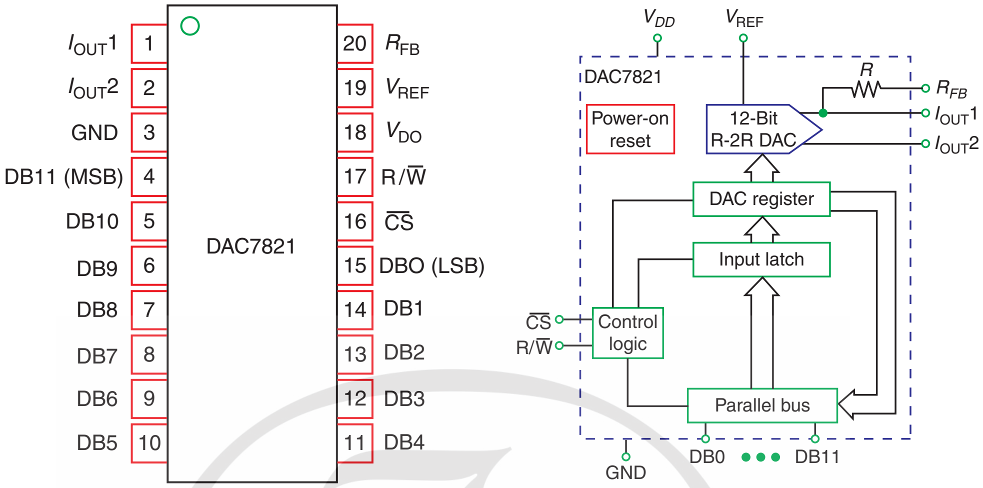


Figure A-3 DAC 7821 – Digital to analog converter

7. Extended temperature range: −40°C to +125°C.
8. 20-lead TSSOP packages.
9. 12-Bit monotonic.
10. 1LSB INL.
11. Four-quadrant multiplication.
12. Power-on reset with brownout detection.
13. Read back function.
14. Industry-standard pin configuration.

A.3.2 Applications

1. Portable battery-powered instruments.
2. Waveform generators.
3. Analog processing.
4. Programmable amplifiers and attenuators.

5. Digitally controlled calibration.
6. Programmable filters and oscillators.
7. Composite video.
8. Ultrasound.

A.3.3 Description

The DAC7821 is a CMOS 12-bit current output digital-to-analog converter (DAC). This device operates from a single 2.5V to 5.5V power supply, making it suitable for battery-powered and many other applications. This DAC operates with a fast parallel interface. Data read back allows the user to read the contents of the DAC register via the DB pins. On power-up, the internal register and latches are filled with zeroes and the DAC outputs are at zero scale. The DAC7821offers excellent 4-quadrant multiplication characteristics, with a large signal multiplying and width of 10 MHz. The applied external reference input voltage (V_{ref}) determines the full-scale output current. An integrated feedback resistor (R_{FB}) provides temperature tracking and full-scale voltage output when combined with an external current-to-voltage precision amplifier. The DAC7821 is available in a 20-lead TSSOP package.

A.3.4 Download Datasheet

`http://focus.ti.com/lit/ds/symlink/dac7821.pdf`

A.4 TPS40200: Wide-Input, Non-Synchronous Buck DC/DC Controller

A.4.1 Features

1. Input voltage range 4.5 to 52 V.
2. Output voltage (700 mV to 90% V_{in}).

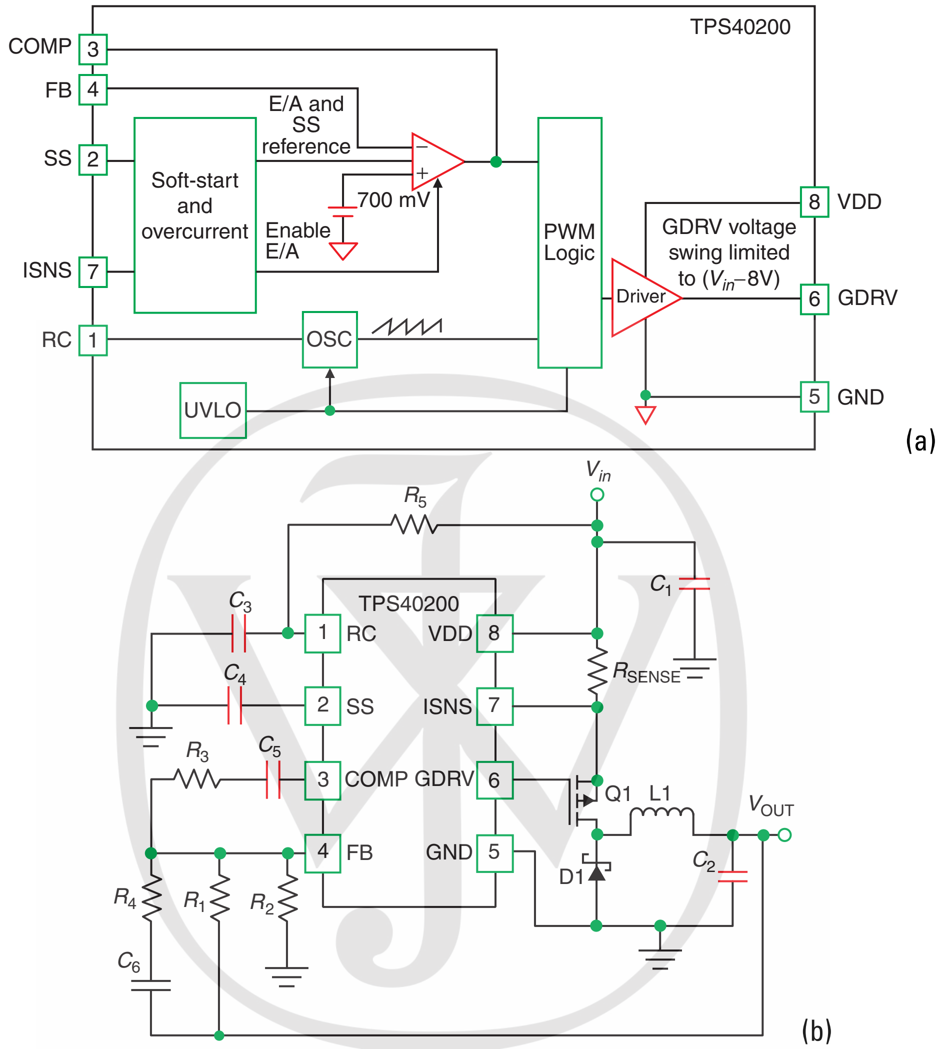


Figure A-4 TPS40200 – DC/DC controller

3. 200 mA internal P-Channel FET driver.
4. Voltage feed-forward compensation.
5. Under-voltage lockout.

6. Programmable fixed frequency (35–500 kHz) operation.
7. Programmable short circuit protection.
8. Hiccup overcurrent fault recovery.
9. Programmable closed loop soft start.
10. 700 mV 1% reference voltage.
11. External synchronization.
12. Small 8-pin SOIC (D) and QFN (DRB) packages.

A.4.2 Applications

1. Industrial control.
2. Distributed power systems.
3. DSL/Cable modems.
4. Scanners.
5. Telecom.

A.4.3 Description

The TPS40200 is a flexible non-synchronous controller with a built in 200 mA driver for P-channel FETs. The circuit operates with inputs up to 52 V with a power-saving feature that turns off driver current once the external FET has been fully turned on. This feature extends the flexibility of the device, allowing it to operate with an input voltage up to 52 V without dissipating excessive power. The circuit operates with voltage-mode feedback and has feed-forward input-voltage compensation that responds instantly to input voltage change. The integral 700 mV reference is trimmed to 2%, providing the means to accurately control low voltages. The TPS40200 is available in an 8-pin SOIC, and supports many of the features of more complex controllers.

A.4.4 TPS40200EVM-002

The TPS40200EVM-002 evaluation module (EVM) uses the TPS40200 non-synchronous buck controller to provide a resistor selected 3.3 V output voltage that delivers up to 2.5 A from a 24 V input bus. The EVM operates from a single supply and uses a single P-channel power FET and Schottky Diode to produce a low cost buck converter. The part operates at a 200 kHz clock frequency as determined by an external resistor and capacitor. TPS40200EVM-002 is designed to operate with an 18 to 36 V input and to produce a regulated 3.3 V output with a load current from 0.125 to 2.5 A. The TPS40200EVM-002 demonstrates using the TPS40200 in a typical buck converter application. The `ASLKv2010 Starter` kit sacrifices some layout density to provide ample test points for module evaluation. This EVM can be modified to support output voltages from 0.7 V to 5 V and above by changing a single feedback resistor. A table is included in the User Guide that lists specific 1% resistors for some common output voltages.

A.4.5 Download Datasheet

`http://focus.ti.com/lit/ds/symlink/tps40200.pdf`

A.5 TLV700xx: 200mA, Low IQ, Low Dropout Regulator for Portables

A.5.1 Features

1. Very low dropout:
2. 43 mV at I_{OUT} = 50 mA, V_{OUT} = 2.8 V.
3. 85 mV at I_{OUT} = 100 mA, V_{OUT} = 2.8 V.
4. 175 mV at I_{OUT} = 200 mA, V_{OUT} = 2.35 V.
5. 2% accuracy.

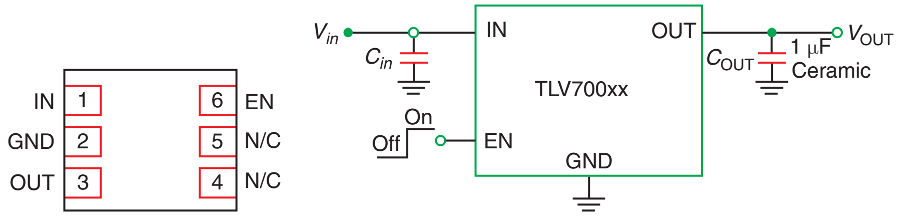


Figure A-5 TLV700XX – Low dropout regulators

6. Low IQ: 31 μA.
7. Available in fixed-output voltages from 0.7 V to 4.8 V.
8. High PSRR: 68 dB at 1 kHz.
9. Stable with effective capacitance of 0.1 μF.
10. Thermal shutdown and overcurrent protection.
11. Available in 1.5 mm × 1.5 mm SON-6, SOT23-5, and SC-70 packages.

A.5.2 Applications

1. Wireless handsets
2. Smart phones, PDAs
3. MP3 players
4. ZigBee™ Networks
5. Bluetooth™ Devices
6. Li-Ion operated handheld products
7. WLAN and other PC add-on cards

A.5.3 Description

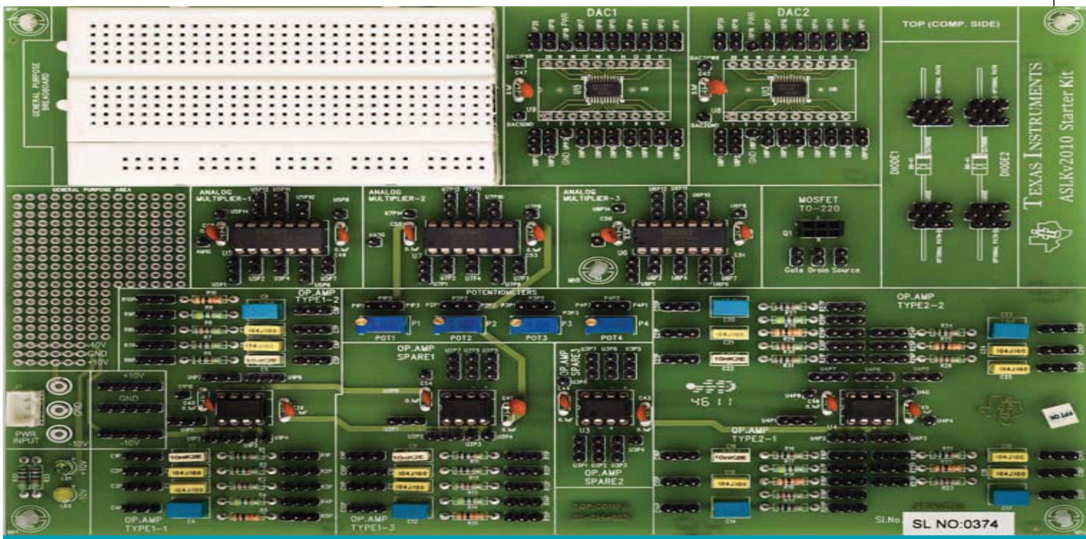
The `TLV700xx/TLV701xx` series of low-dropout (LDO) linear regulators from Texas Instruments are low quiescent current devices with excellent line and load transient performance. These LDOs are designed for power-sensitive applications. A precision bandgap and error amplifier provides overall 2% accuracy. Low output noise, very high power-supply rejection ratio (PSRR), and low dropout voltage make this series of devices ideal for most battery-operated handheld equipment. All device versions have thermal shutdown and current limit for safety. Furthermore, these devices are stable with an effective output capacitance of only 0.1 μF. This feature enables the use of cost-effective capacitors that have higher bias voltages and temperature derating.

A.5.4 TLV70018EVM-503 Evaluation Module

The evaluation module `TLV70018EVM-503` facilitates evaluation of the IC TLV70018 from Texas Instruments, which is a Low Dropout Regulator (200mA, low IQ LDO regulator in the DCK (2.0 x 2.1mm SC70-5) package.

A.5.5 Download Datasheet

`http://focus.ti.com/lit/ds/symlink/tlv70012.pdf`



APPENDIX B

Introduction to Macromodels

Simulation models are very useful in the design phase of an electronic system. Before a system is actually built using real components, it is necessary to perform a "software breadboarding" exercise through simulation to verify the functionality of the system and to measure its performance. If the system consists of several building blocks $B_1, B_2, \ldots, B_n$, the simulator requires a mathematical representation of each of these building blocks in order to predict the system performance. Let us consider a very simple example of a passive component such as a resistor. Ohm's law can be used to model the resistor if we intend to use the resistor in a DC circuit. But if the resistor is used in a high frequency application, we may have to think about the parasitic inductances and capacitances associated with the resistor. Similarly, the voltage and current may not have a strict linear relation due to the dependence of the resistivity on temperature of operation, skin effect, and so on. This example illustrates that there is no single model for an electronic component. Depending on the application and the accuracy desired, we may have to use simpler or more complex mathematical models.

We will use another example to illustrate the above point. The MOS transistor, which is the building block of most integrated circuits today, is introduced at the beginning of a course on VLSI design. In a digital circuit, the transistor may be simply modeled as an ideal switch that can be turned on or off by controlling the gate voltage. This model is sufficient if we are only interested in understanding the functionality of the circuit. If we wish to analyze the speed of operation of the circuit or the power dissipation in the circuit, we will need to model the parasitics associated with the transistors. If the same transistor is used in an analog circuit, the model that we use in the analysis would depend on the accuracy which we want in the analysis. We may perform different kinds of analysis for an analog circuit – DC analysis, transient analysis, and steady-state analysis. Simulators such as SPICE require the user to specify the model for the transistor. There are many different models available today for the MOS transistor, depending on the desired accuracy. The level-1 model captures the dependence of the drain-to-source current on the gate-to-source and drain-to-source voltages, the mobility of the majority carrier, the width and length of the channel, and the gate oxide thickness. It also considers non-idealities such as channel length modulation in the saturation region, and the dependence of the threshold voltage on the source-to-bulk voltage. More complex models for the transistor are available, which have more than 50 parameters.

B.1 Micromodels

If you have built an operational amplifier using transistors, a straight-forward way to analyze the performance of the `Op-Amp` is to come up with the micromodel of the `Op-Amp` where each transistor is simply replaced with its corresponding simulation model. Micromodels will lead to accurate simulation, but will prove computationally intensive. As the number of nodes in the circuit increases, the memory requirement will be higher and the convergence of the simulation can take longer.

A macromodel is a way to address the problem of space-time complexity mentioned above. In today's electronic systems, we make use of analog circuits such as operational amplifiers, data converters, PLL, VCO, voltage regulators, and so on. The goal

Table B-1 Operational amplifiers available from Texas Instruments

	Characteristic	Number of Varieties
1	Standard Linear Amplifier	240
2	Fully Differential Amplifier	28
3	Voltage Feedback	68
4	Current Feedback	47
5	Rail to Rail	14
6	JFET/CMOS	23
7	DSL/Power Line	19
8	Precision Amplifier	641
9	Low Power	144
10	High Speed Amplifier ($\geq$ 50 MHz)	182
11	Low Input Bias Current/FET Input	38
12	Low Noise	69
13	Wide Bandwidth	175
14	Low Offset Voltage	121
15	High Voltage	4
16	High Output Current	54
17	LCD Gamma Buffer	22

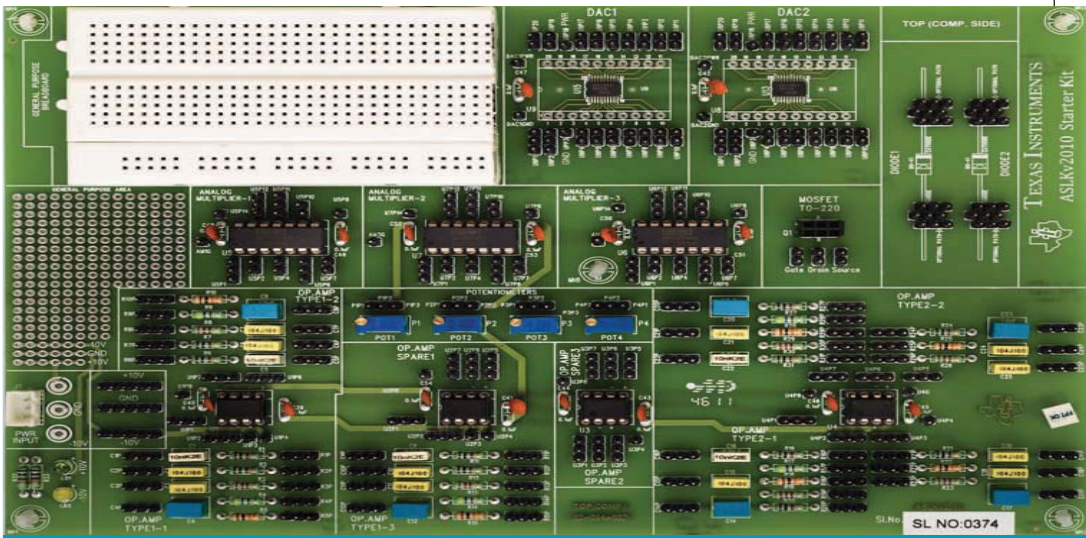
of the system designer is not only to get a functionally correct design, but also to optimize the cost and performance of the system. The system-level cost and performance depend on the way the building blocks $B_1, B_2, \ldots, B_n$ have been implemented. For example, if B_1 is an `Op-Amp`, we may have several choices of operational amplifiers. Texas Instruments offers a large number of operational amplifiers that a system designer can choose from. Refer to Table B-1. As you will see, there are close to 2000 types of operational amplifiers available! These are categorized into 17 different bins to make the selection simpler. However, you will notice that 240 varieties are available in the category of *Standard Linear* amplifiers! How does a system designer select from this large collection? To understand this, you must look at the characteristics of a standard linear amplifier – these include the number of operational amplifiers in a single package, the Gain Bandwidth Product of the amplifier, the CMRR, $V_{s(\min)}$, $V_{s(\max)}$, and so on. See `http://tinyurl.com/ti-std-linear`. The website allows you to specify these parameters and narrow your choices.

But how does one specify the parameters for the components? The overall system performance will depend on the way the parameters for the individual components have been selected. For example, the gain-bandwidth product of an operational amplifier B1 will influence a system-level parameter such as the noise immunity or stability. If one has n components in the system, and there are m choices for each component, there are $m \cdot n$ possible system configurations. Even if we are able to narrow the choices through some other considerations, we may still have to evaluate several system configurations. Performing simulations using micromodels will be a painstaking and non-productive way of selecting system configurations.

B.2 Macromodels

A macromodel is a mathematical convenience that helps reduce simulation complexity. The idea is to replace the actual circuit by something that is simpler, but is nearly equivalent in terms of input characteristics, output characteristics, and feed-forward characteristics. Simulation of a complete system becomes much more simple when we use macromodels for the blocks. Manufacturers of semiconductors provide macromodels for their products to help system designers in the process of system configuration selection. You can download the models for TI analog ICs from `http://tinyurl.com/ti-macromodels` – these files have a `.tsm` extension.

As you can guess, there is no single macromodel for an IC. A number of macromodels can be derived, based on the level of accuracy desired and the computational complexity that one can afford. A recommended design methodology is to start with a simple macromodel for the system components and simulate the system. A step-wise refinement procedure may be adopted and more accurate models can be used for selected components when the results are not satisfactory.



APPENDIX C

Activity: To Convert your PC/Laptop into an Oscilloscope

C.1 Introduction

In any analog lab, an oscilloscope is required to display waveforms at different points in the circuit under construction in order to verify circuit operation and, if necessary, redesign the circuit. High-end oscilloscopes are needed for measurements and characterization in labs. Today, solutions are available to students for converting a PC into an oscilloscope [31]. These solutions require some additional hardware to route the analog signals to the PC for observation; they also require software that provides the graphical user interface to convert a PC display into an oscilloscope. Since most students have access to a PC or laptop today, we have designed the **Analog System Lab** such that a PC-based oscilloscope solution can be used along with `ASLKv2010 Starter` kit. We believe this will reduce the dependence of the student on a full-fledged lab. In this chapter, we will review a solution for a PC-based oscilloscope. The components on the `ASLKv2010 Starter` kit can be used to build the interface circuit needed to convert the PC into an oscilloscope.

One of the solutions for a "PC oscilloscope" is Zelscope [36] which works on personal computers running MS/Windows XP. The hardware requirements for the PC are modest (300+ MHz clock, 64+ MB memory). It uses the sound card in the PC for converting the analog signals into digital form. The Zelscope software, which requires about 1 MB space, is capable of using the digitized signal to display waveforms as well as the frequency spectrum of the analog signal.

At the "line in" jack of the sound card, the typical voltage should be about 1 V AC; hence it is essential to protect the sound card from over voltages. A buffer amplifier circuit is required to protect the sound card from over voltages. Two copies of such a circuit are needed to implement a dual-channel oscilloscope. The buffer amplifier circuit is shown in Figure C-1 and has been borrowed from [35].

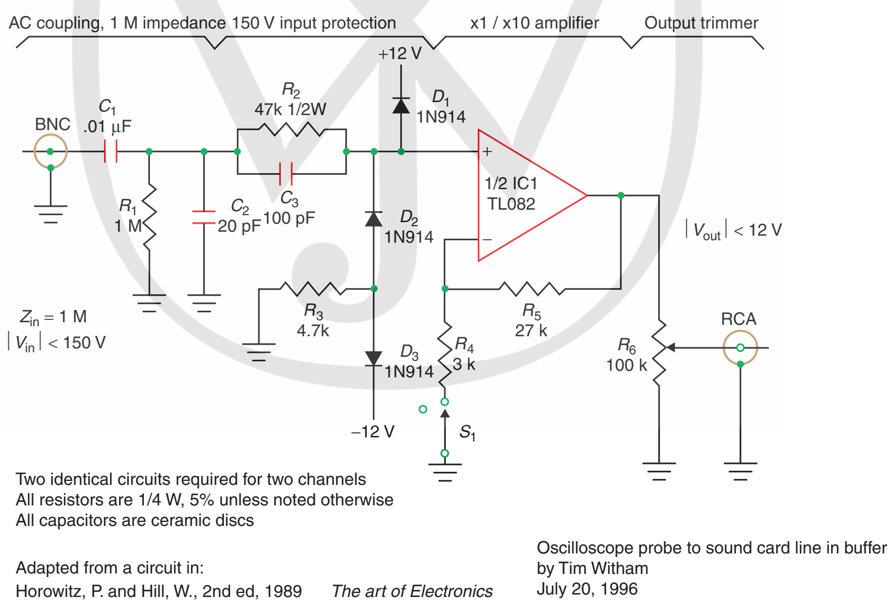
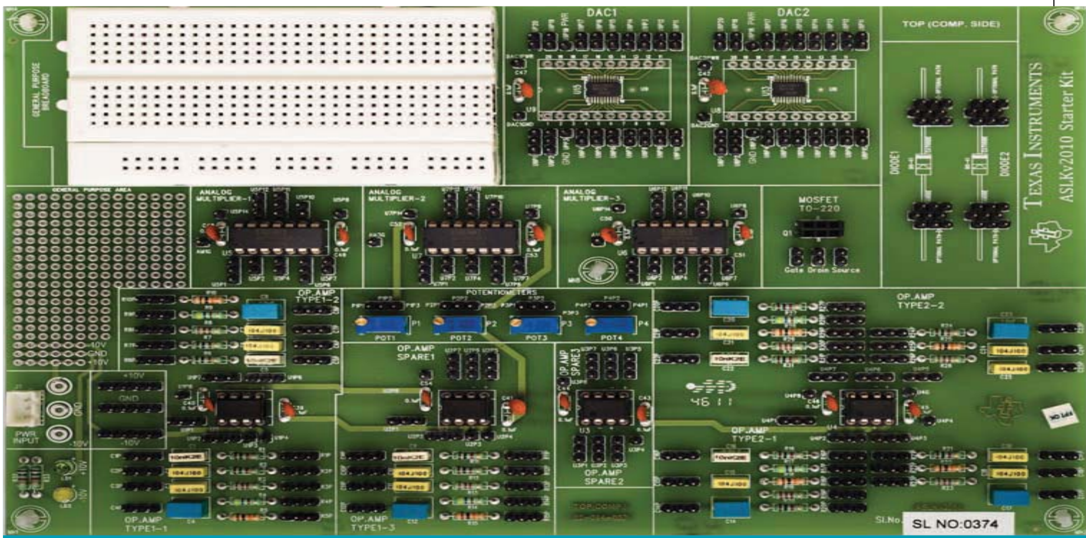


Figure C-1 Buffer circuit needed to interface an analog signal to oscilloscope

C.2 Limitations

1. Not possible to display DC voltages (as the input capacitor of sound card blocks DC)
2. Low frequency range (10 Hz–20 kHz)
3. Measurement is not very accurate

APPENDIX D



System Lab Kit ASLKv2010 Starter Kit Connection Diagrams

Figure D-1 shows the overall floorplan of the ASLKv2010 Starter kit. We have shown the power connections in ASLKv2010 Starter in Figure D-2. Note that the ±10 V power and ground connections have to be connected to the power inlets at the side of the kit; the power and ground is internally connected to the OPAMPs and analog multipliers. If you wish to carry out an experiment using the DAC integrated circuits on the board, you must use an extern 5 V supply and ground connection, as shown in Figure D-2. There are three potmeters included in the kit, each of which is connected across 10 V and ground. The output of the potmeter can be used to derive a voltage in the range 0 to 10 V; this can be useful in generating a reference voltage or even in generating a 5 V power supply for the DAC.

The student must become familiar with the general floorplan of the kit. Begin by understanding the power and ground connections to the kit. Note that ±10 V and ground connections must be fed to the kit from the inlets at the left-hand side. This automatically powers the operational amplifiers and the analog multipliers. Each of the

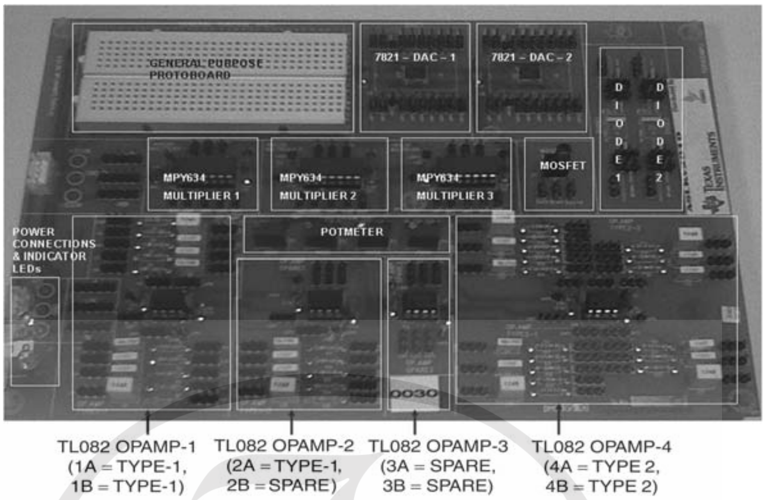


Figure D-1 Floorplan of the `ASLKv2010 Starter` kit

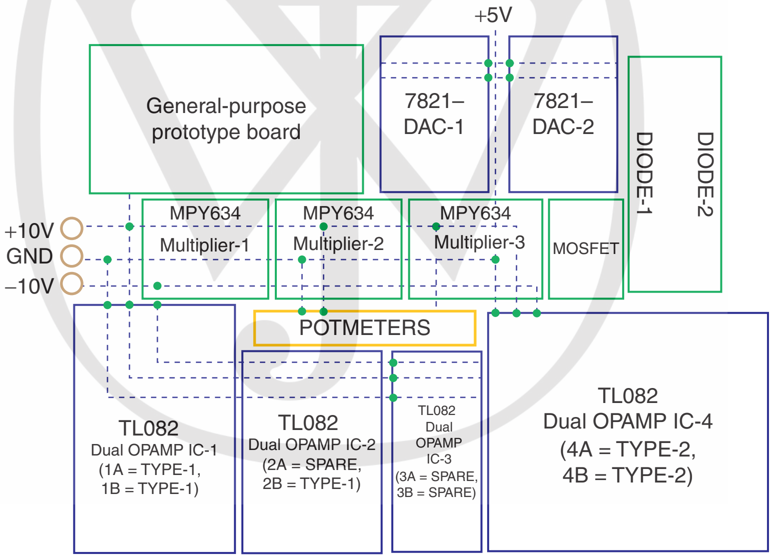


Figure D-2 Power connections in `ASLKv2010 Starter` kit. Only connect ±10 V and ground connections

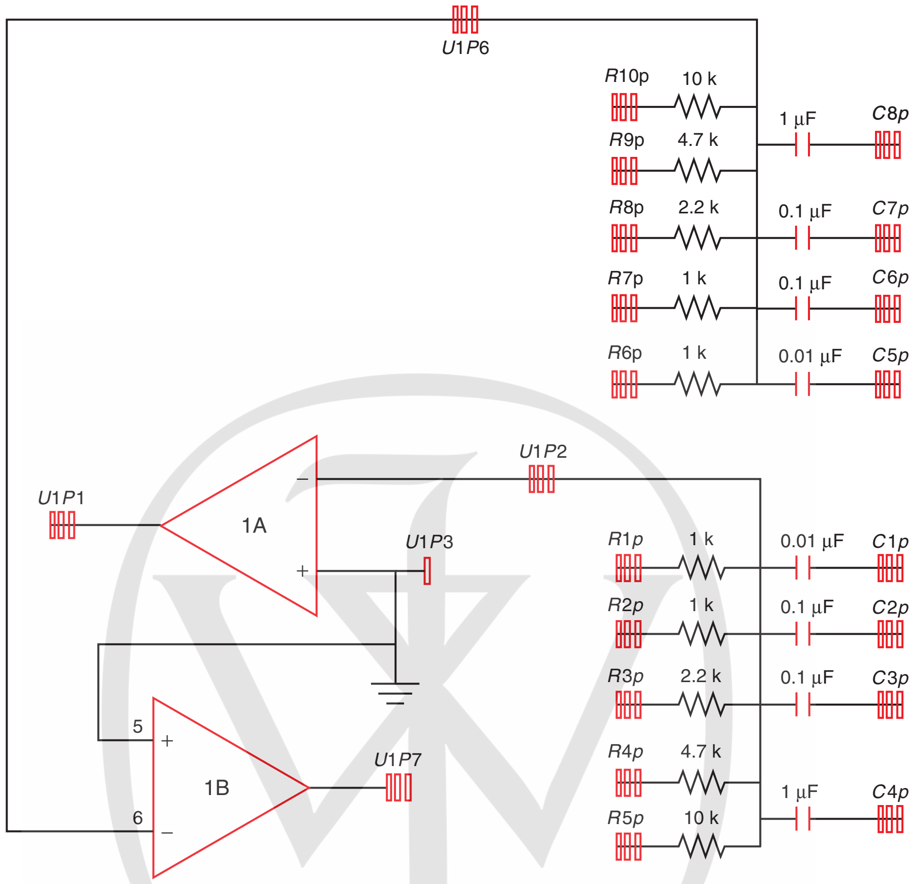


Figure D-3 Op-Amp IC-1 (Dual Op-Amp with two amplifiers, 1A and 1B) connected in Type-1 configuration (Inverting)

potmeters receives +10 V supply and ground connection. The output of a potmeter can be used as a DC reference voltage or even as power supply for the DAC.

Figure D-3 shows the connections for Op-Amp IC-1, which has two Op-Amps connected in Type-1 (inverting) configuration. The inverting terminal of Op-Amp 1-A is connected to resistors through Berg pin connections $R1p, R2p, \ldots, R5p$ and

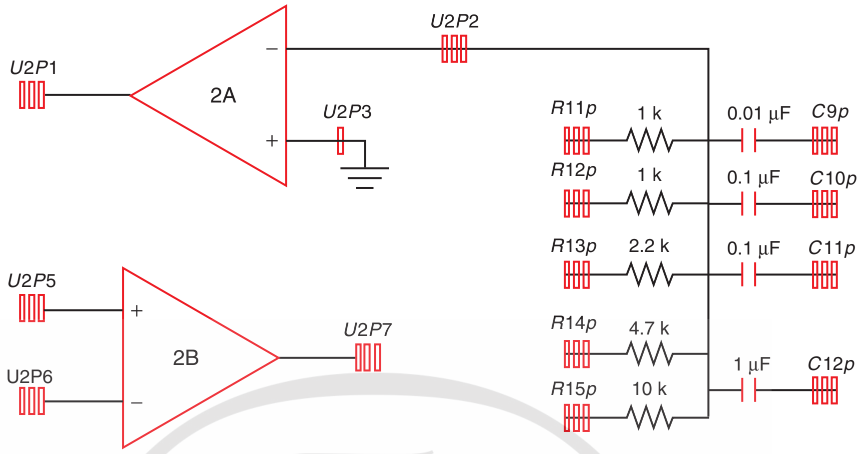


Figure D-4 Op-Amp IC-2 (Dual Op-Amp with two amplifiers, 2A and 2B. 2A can be connected in Type-1 configuration (Inverting; 2B is a spare)

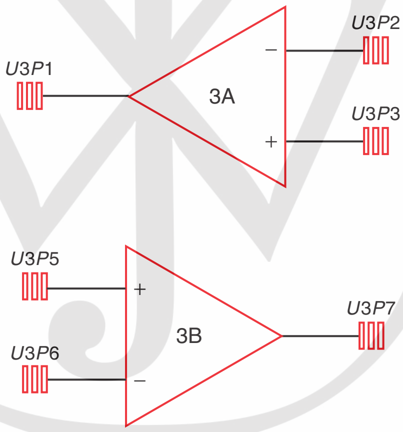


Figure D-5 Op-Amp IC-3 (Dual Op-Amp with two spare amplifiers, 3A and 3B)

to capacitors through the Berg pin connections *C*1*p*, *C*2*p*, *C*3*p*, *C*4*p*. Note that each Berg connection has three pins and the user can use any one of them for making an electrical connection. The Op-Amp 1-B is connected to resistors through Berg pin connections *R*6*p*, *R*7*p*, . . . , *R*10*p* and to capacitors through Berg pin connections

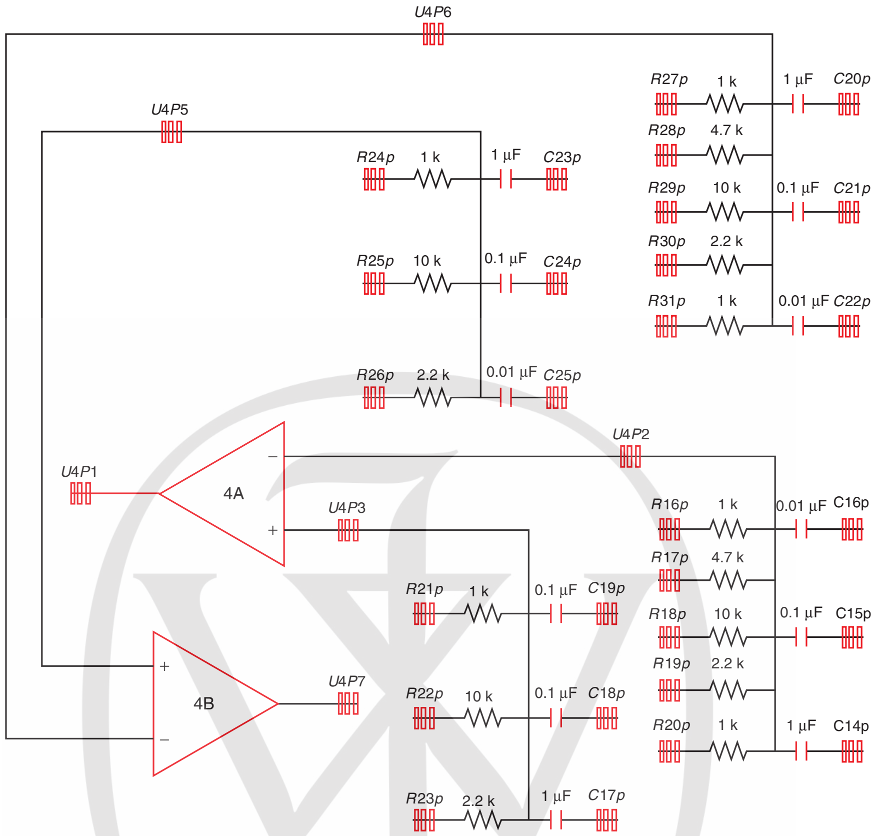


Figure D-6 Op-Amp IC-4 (Dual Op-Amp with amplifiers 4A and 4B. Op-Amp 4A and 4B can be used in inverting or non-inverting configuration)

*C*5*p*, *C*6*p*, *C*7*p*, *C*8*p*. The Berg pin connection *U*1*P*1 can be used to connect the output of Op-Amp 1-A. Similarly, connnection *U*1*P*7 can be used to connect the output of Op-Amp 1-B.

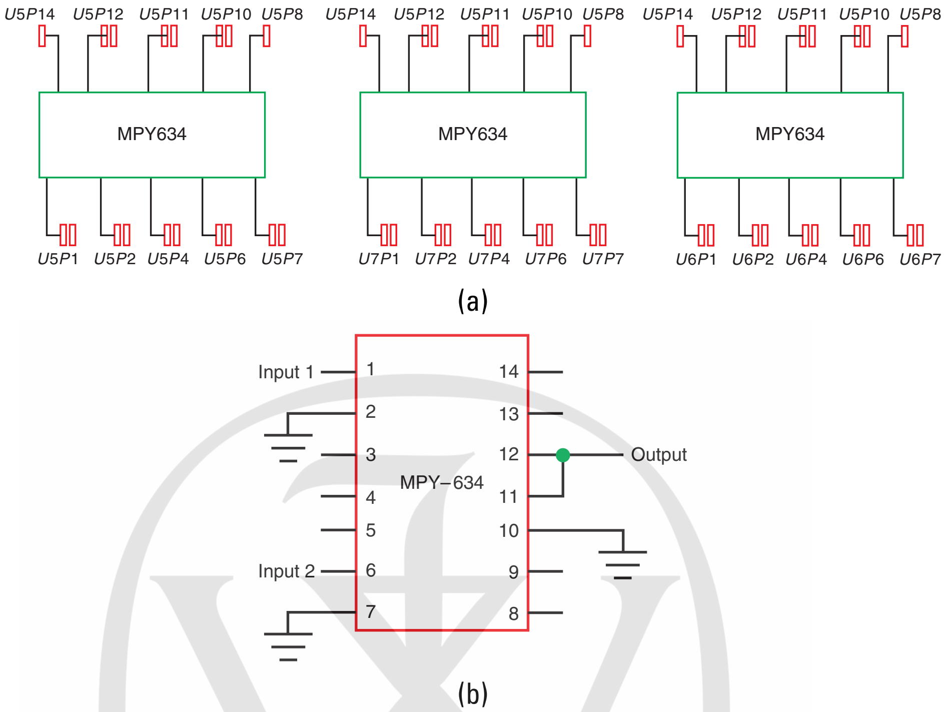


Figure D-7 (a) PCB connections for analog multipliers 1, 2 and 3 on `ASLKv2010 Starter`; (b) External connections needed to use the multiplier MPY634

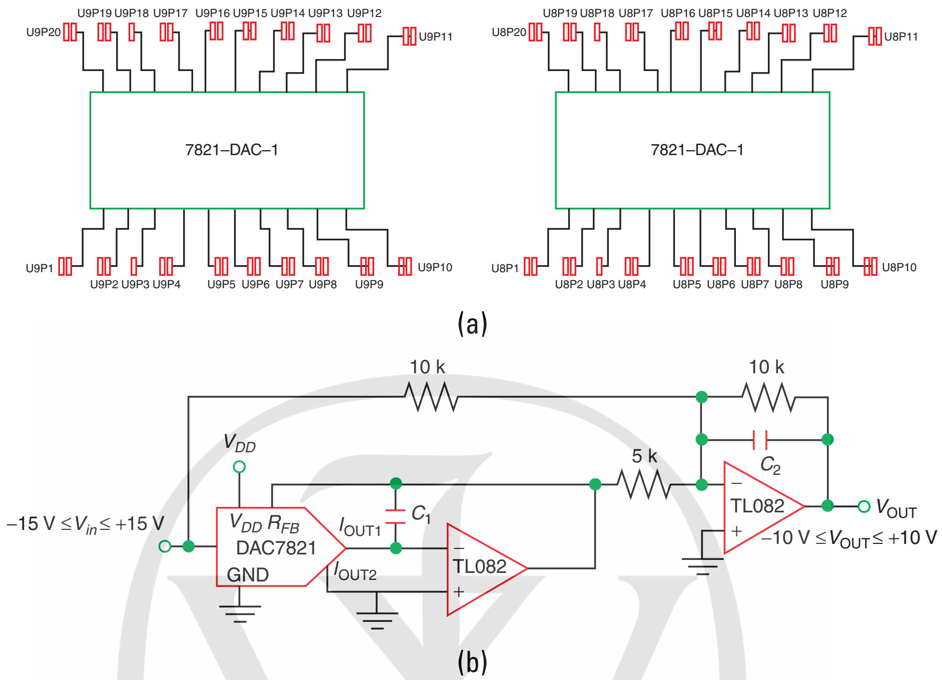
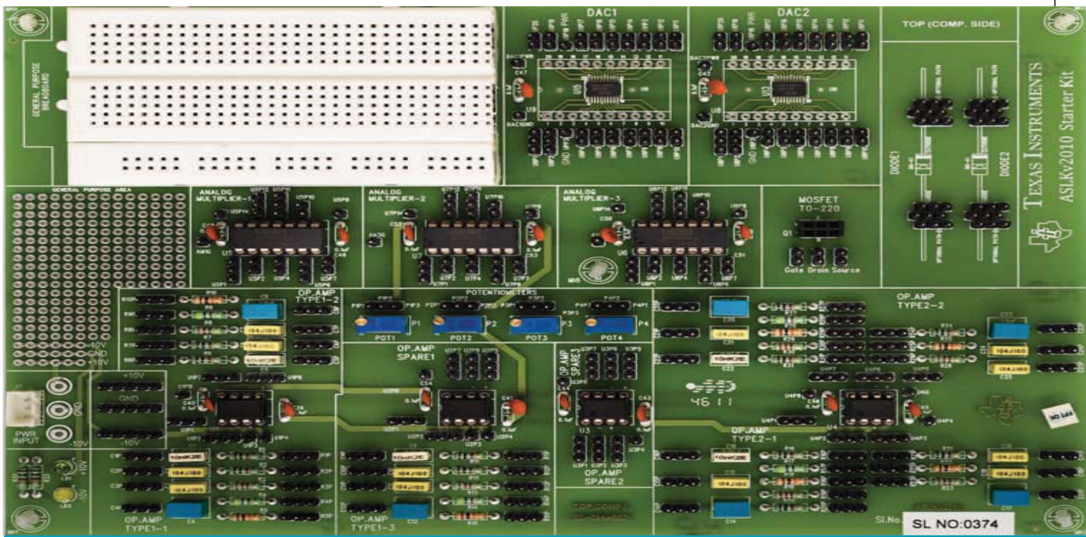


Figure D-8 (a) PCB connections for the DAC on `ASLKv2010 Starter`; (b) external connections needed to use the DAC to make it four-quadrant



Bibliography

[1] ADCPro (TM) – Analog to Digital Conversion Evaluation Software. Free. Available from http://focus.ti.com/docs/toolsw/folders/print/adcpro.html

[2] F. Archibald. Automatic Level Controller for Speech Signals Using PID Controllers. Application Notes from Texas Instruments. Available from http://focus.ti.com/lit/wp/spraaj4/spraaj4.pdf

[3] High-Performance Analog. Available from www.ti.com/analog

[4] Wide Bandwidth Precision Analog Multiplier MPY634, Burr Brown Products from Texas Instruments, Available from http://focus.ti.com/lit/ds/sbfs017a/sbfs017a.pdf

[5] B. Carter and T. Brown. Handbook Of Operational Amplifier Applications. Texas Instruments Application Report. 2001. Available from http://focus.ti.com/lit/an/sboa092a/sboa092a.pdf

[6] B. Carter. *Op Amp and Comparators – Don't Confuse Them!* Texas Instruments Application Report, 2001. Available from http://tinyurl.com/carter-opamp-comp

[7] B. Carter. Filter Design in Thirty Seconds. Application Report from Texas Instruments. Downloadable from http://focus.ti.com/lit/an/sloa093/sloa093.pdf

[8] B. Carter and R. Mancini. OPAMPS For Everyone. Elsevier Science Publishers, 2009.

[9] DesignSoft. TINA – the complete analog lab. www.tina.com

[10] FilterPro (TM) – Active Filter Design Application. Free software. Available from http://tinyurl.com/filterpro-download

[11] Thomas Kuehl and Faisal Ali. Active Filter Synthesis Made Easy With FilterPro V3.0. Tutorial presented in TI Technology Days 2010 (May), USA. Available from http://www.ti.com/ww/en/techdays/2010/index.shtml

[12] J. Molina. DESIGN A 60Hz Notch Filter with the UAF42. Application note from Burr-Brown (Texas Instruments), 2000. Available from http://focus.ti.com/lit/an/sbfa012/sbfa012.pdf

[13] J. Molina. Digitally Programmable, Time-Continuous Active Filter, 2000. Application note from Burr-Brown (Texas Instruments), http://focus.ti.com/lit/an/sbfa005/sbfa005.pdf

[14] George S. Moschytz. From Printed Circuit Boards to Systems-on-a-chip. IEEE Circuits and Systems magazine, Vol 10, Number 2, 2010.

[15] National Instruments. www.ni.com/multisim

[16] Phase-locked loop. Wikipedia entry. http://en.wikipedia.org/wiki/Phase-locked_loop

[17] R. Palmer. Design Considerations for Class-D Audio Amplifiers. Application Note from Texas Instruments. Available from http://focus.ti.com/lit/an/sloa031/sloa031.pdf

[18] K.R.K. Rao. Electronics for Analog Signal Processing – Part II. Op-Amp in Negative Feedback. Recorded lecture available through NPTEL. http://tinyurl.com/krkrao-nptel-lec7 and http://tinyurl.com/krkrao-nptel-lec8

[19] K.R.K. Rao. Electronics for Analog Signal Processing – Part II. Frequency Compensation in Negative Feedback. Recorded lecture available through NPTEL. http://tinyurl.com/krkrao-nptel-lec16 and http://tinyurl.com/krkrao-nptel-lec17

[20] K.R.K. Rao. Electronics for Analog Signal Processing – Part II. Instrumentation Amplifier. Recorded lecture available through NPTEL. http://tinyurl.com/krkrao-nptel-lec11

[21] K.R.K. Rao. Electronics for Analog Signal Processing – Part II. Active Filters. Recorded lecture available through NPTEL. http://tinyurl.com/krkrao-nptel-lec12

[22] K.R.K. Rao. Electronics for Analog Signal Processing – Part II. Positive Feedback (Regenerative). Recorded lecture available through NPTEL. http://tinyurl.com/krkrao-nptel-lec9

[23] K.R.K. Rao. Analog ICs. Self-Tuned Filter. Recorded lecture available through NPTEL. http://tinyurl.com/krkrao-nptel-ic-lec23

[24] K.R.K. Rao. Analog ICs. Phase Locked Loop. Recorded lecture available through NPTEL. http://tinyurl.com/krkrao-nptel-ic-lec24, http://tinyurl.com/krkrao-nptel-ic-lec25, http://tinyurl.com/krkrao-nptel-ic-lec26, and http://tinyurl.com/krkrao-nptel-ic-lec27

[25] K.R.K. Rao. Electronics for Analog Signal Processing – Part II. Voltage Regulators. Recorded lecture available through NPTEL. http://tinyurl.com/krkrao-nptel-26, http://tinyurl.com/krkrao-nptel-27, http://tinyurl.com/krkrao-nptel-28

[26] K.R.K. Rao. Electronics for Analog Signal Processing – Part II. Converters. Recorded lecture available through NPTEL. http://tinyurl.com/krkrao-nptel-28,

[27] K.R.K. Rao. Electronics for Analog Signal Processing – Part II. AGC/AVC. http://tinyurl.com/krkrao-nptel-33, http://tinyurl.com/krkrao-nptel-34, http://tinyurl.com/krkrao-nptel-35, http://tinyurl.com/krkrao-nptel-36

[28] K.R.K. Rao. Analog ICs. Voltage Controlled Oscillator. Recorded lectures available from http://tinyurl.com/krkrao-vco-1, http://tinyurl.com/krkrao-vco-2

[29] Thomas Kugesstadt. Active Filter Design Techniques. Texas Instruments. Available from http://focus.ti.com/lit/ml/sloa088/sloa088.pdf

[30] Oscilloscope Solutions from Texas Instruments – Available from http://focus.ti.com/docs/solution/folders/print/437.html

[31] PC Based Test and Instrumentation. Available from http://www.pctestinstruments.com

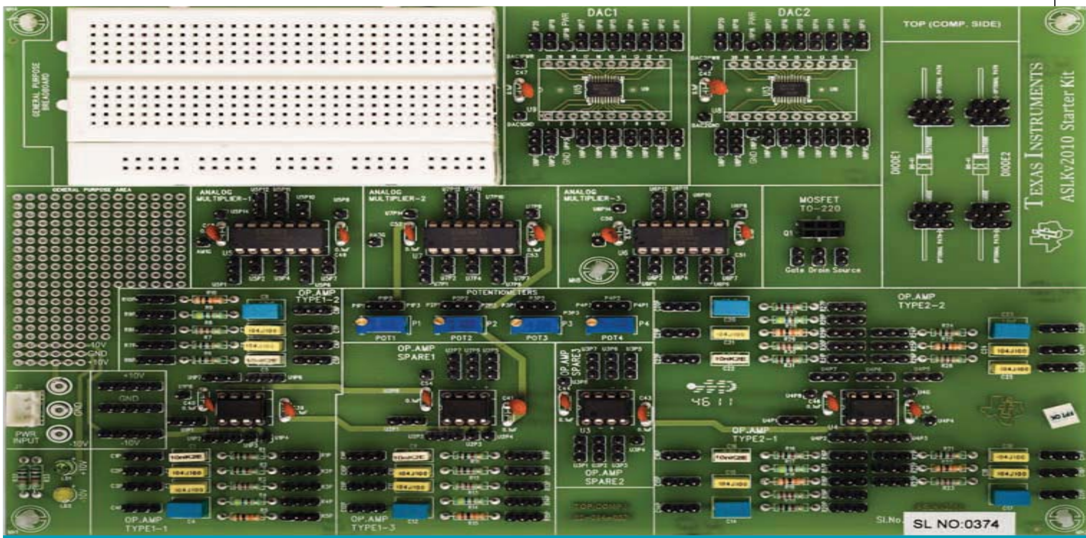
[32] PSpice. http://en.wikipedia.org/wiki/PSpice

[33] SwitcherPro (TM) – Switching Power Supply Design Tool. http://focus.ti.com/docs/toolsw/folders/print/switcherpro.html

[34] Texas Instruments Analog eLAB – SPICE Model Resources. Macromodels for TI analog ICs are downloadable from http://tinyurl.com/ti-macromodels

[35] How to use PC as Oscilloscope. Available from www.trickswindows.com

[36] Zelscope: Oscilloscope and Spectrum Analyzer. Available from www.zelscope.com


DAC1
DAC2
TOP (COMP. SIDE)
TEXAS INSTRUMENTS
ASLKv2010 Starter Kit
SL NO:0374

Index