



VANIER
CÉGEP/COLLEGE
My choice. My future.

Electrical Engineering Technologies
Industrial Electronics
Instrumentation & Automation

Vanier College

Industrial Electronics Computerized Systems

Signal Processing 243-334-VA A Text Lab Manual

Rev: August 2012

Prepared by Patrick Bouwman:

SIGNAL PROCESS
55512122107



VANIER
CÉGEP/COLLEGE
My choice. My future.

Electrical Engineering Technologies
Industrial Electronics
Computerized Systems

Introduction

This student manual will provide the fundamental concepts and applications of signal processing and signal conditioning circuits. This course will extend the students' knowledge of circuit analysis to include specific signals as provided by the different types of transducers, used in instrumentation.

Students will be introduced to the fundamental techniques of analog and digital circuits and systems. Analog and digital signal processing and signal conditioning circuits (SCC's) will be studied and tested. Students will prepare, assemble measure, analyse, troubleshoot, and document typical analog and digital circuits used in processing signals. Software used to acquire and display data from low voltage circuits will be applied to simple digital signal processing algorithms.

Table of Content

Section 1 Notes

Section 2 Labs



Section 1 Notes

Table of Content

	Introduction (what is Signal Processing, Analog and Digital)
1	History of Op-Amps
2	Operational Amplifiers
3	The decibel (dB)
4	Signals and Noise
5	Bode Plots
6	Active Filters
7	Analog to Digital (A/D)
8	Intro to Digital Filters
9	Troubleshooting Op amps
10	Functional Diagrams for Instrument Control Systems



A brief history of the Operational Amplifier

The first commercially produced operational amplifier (the Model K2.W--) from Philbrick Laboratory in 1946 was a vacuum tube unit built to perform operational computation (today known as analog simulation). See Figure 1.

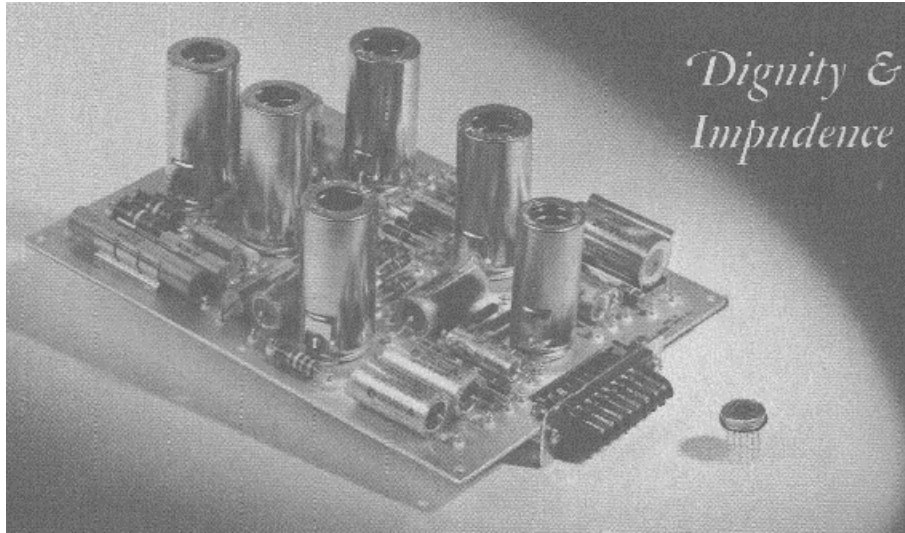


Figure 1. Picture of the first commercial operational amplifier next to a modern integrated circuit operational amplifier.

In 1948, the first semiconductor device was developed at Bell Laboratories. The device was called a transistor. Ten years later, in 1958, we saw the first integrated circuit (IC) constructed. In 1962 the first solid-state operational amplifier (op-amp) was introduced.

In 1963, Fairchild Semiconductor produced the first integrated circuit commercial op-amp, the $\mu A702$. The device had a low voltage gain (by today's standards) and used a bipolar power supply of -6 V and $+12\text{ V}$ dc. Although it burned out easily when the output was temporarily shorted, the $\mu A702$ was the best device of its time.

In 1965, Bob Widlar, a device engineer for Fairchild, designed the $\mu A709$ IC, which had a much higher gain and a lower input current than existing operational amplifiers and it used a balanced $\pm 15\text{ V}$ power supply voltage. Ever since then, operational amplifiers have been the analogue-design engineer's equivalent of a Lego building block.

In 1967, National Semiconductor introduced the LM101, which had a gain of up to 160,000. Equipped with short-circuit protection, the LM101 also had simplified frequency



compensation through the use of an externally wired capacitor. This arrangement eliminated internally generated oscillations in the direct-coupled amplifier network.

Fairchild Semiconductor introduced its $\mu A741$ in 1968. Similar to the LM101, it also had internal frequency compensation, simplifying the design of circuits that use operational amplifiers. The 741 is even today considered an industry standard and is produced by most semiconductor manufacturers in some form or other.

Chips containing multiple op-amps were introduced in 1974, when Raytheon Semiconductor developed the RC4558. The RCA CA3130 introduced FET input op-amps, which had smaller input current requirements.

Many types and configurations of op-amps have been introduced since that time.

Application of op-amps

The characteristics of the operational amplifier that lead to its widespread use and many applications are of particular interest to the technologist involved in industrial electronics. The op-amp is a powerful device that can be used as a linear amplifier, a signal-processing device, and/or a device for performing mathematical operations. It can also be used for the electronic isolation of in-circuit components.

Linear amplifiers usually operate as a multiplying type of function (technically known as a class-A amplifier) with the output signal being an exact reproduction of the input signal. The op-amp can be used to correct leading and trailing edges of pulses and to establish thresholds for control applications and modifying signals. Because of its ideal and unique characteristics, it can be used in relatively simple circuits to perform addition, subtraction, multiplication, and/or division of input voltages to give results that reflect mathematical operations. Differentiation and integration are performed with similar ease. The result is that the operational amplifier, with the addition of relatively few components, is an analog computer circuit capable of doing sophisticated mathematical computations. Such computation is required in process control and signal processing. Because of its high impedance, the op-amp can be used to isolate devices in active circuits without any significant loading effects. Op-amps are also used in logic and audio circuits.



Characteristics of typical op-amps

The LM747 is a typical integrated circuit op-amp. This “chip” contains two 741 op-amps in one package. Figure 1 shows the internal circuit of one 741 op-amp, and Figure 2 illustrates different package configurations for chip.

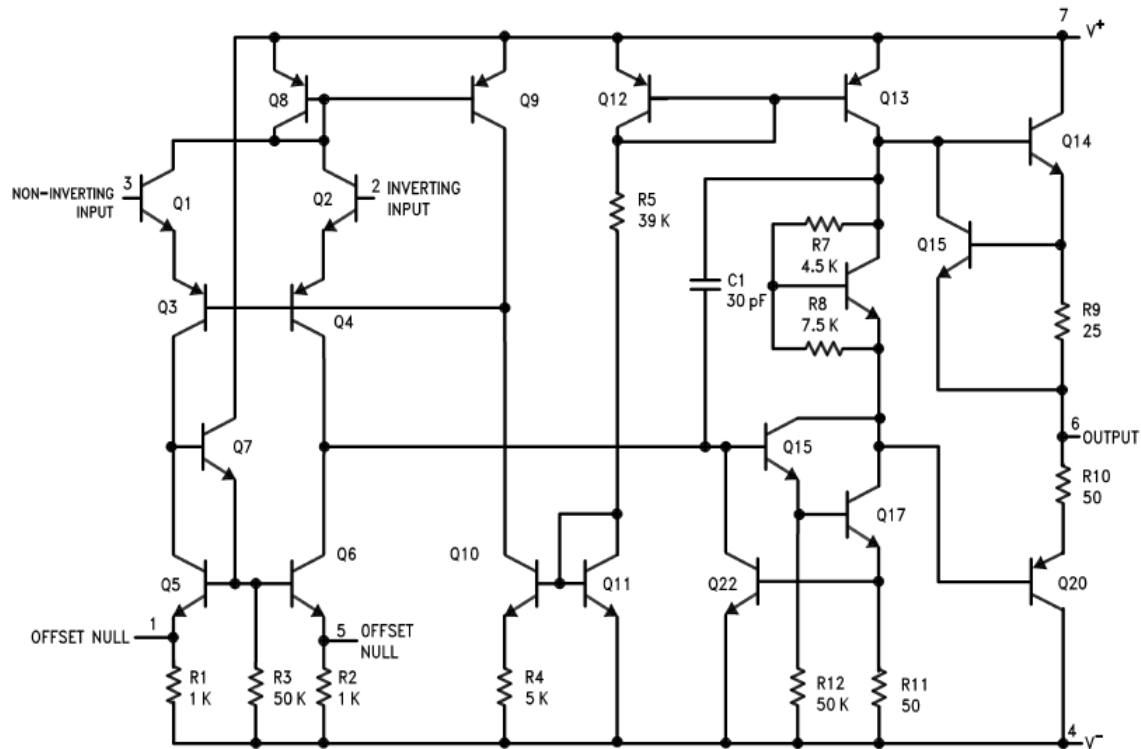


Figure 2. Schematic diagram of a 741 op-amp.

In Figure 2, transistors Q1 and Q2 are, respectively, non-inverting and inverting inputs to a differential amplifier configuration. Transistors Q16 and Q17 constitute a high-gain Darlington driver. Transistors Q14 and Q20 form a class-AB complementary symmetry output stage. A 10 kΩ potentiometer can be connected between pins 3 and 14 (for op-amp A) or pins 5 and 8 (for op-amp B) with the wiper tied to $-V_{cc}$, providing offset null adjustments. (Offset null adjustments are used to compensate for small irregularities in the manufacturing of the op-amp.) Tiny differences in the characteristics of transistors Q1 and Q2 sometimes mean that one transistor will conduct a little more than the other. When this slight imbalance is amplified many thousands of times, an output may occur even without any input. The offset null is used to ‘zero’, or cancel this so that with zero input there would be a zero output.

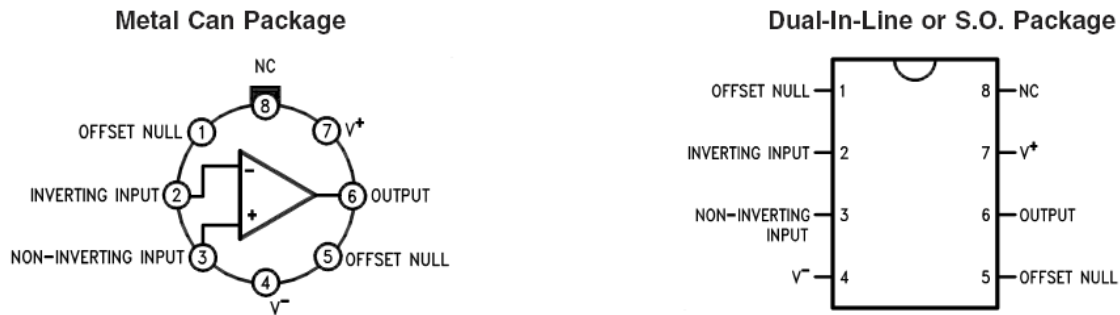


Figure 3. Connection diagram of a 747chip.

An ideal operational amplifier should have the following characteristics:

- Infinite gain
- Infinite input impedance
- Zero output impedance
- Infinite bandwidth
- Zero output voltage with no input voltage
- Zero input current (bias current)

The 741 is not an ideal amplifier, but its characteristics are impressive. The following table compares the 741 actual data sheet to the ideal:

- | | |
|-----------------------------|-------------------------------|
| – Infinite Gain | 200,000 |
| – Infinite input impedance | 1 M Ω to 50 M Ω |
| – Zero output impedance | less than 75 Ω |
| – Infinite bandwidth | between 1 MHz and 5 MHz |
| – Zero input offset voltage | 7.5 mV max |
| – Zero input offset current | 500 nA max |

The student should study a typical op-amp datasheet and locate the various parameters.



Operational Amplifiers

Introduction

Linear integrated circuits are circuits where many transistors, resistors, diodes, and capacitors are used to form a functional circuit (on a chip) that behaves in a linear fashion. A perfect example is the Operational Amplifier (op-amp). The op-amp can be treated as a single device with linear characteristics. The op-amp is the most versatile and widely used of all linear integrated circuits.

The universal symbol for the Operational Amplifier, shown in Figure 1, is a triangle that indicates the causal direction of the signal flow.

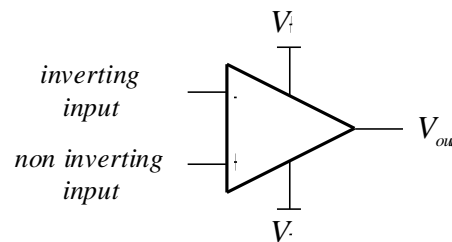


Figure 1. Operational Amplifier Symbol

The inputs are denoted with a (+) and a (–) sign indicating the non-inverting (positive) or the inverting (negative) input. Power supply connections are shown at the top (positive) and at the bottom (negative) in accordance with standard schematic diagram practices. When the power supply connections are not shown on the schematic diagram, they are tabulated on the schematic diagram.

Operational amplifiers, are dc coupled semiconductor amplifiers with very large voltage gains, and are almost always used with negative feedback. At this point we will not go into the details of all op-amp specifications and characteristics, but rather deal with them as an idealized building block. Op-amps that are used with no feedback, or with positive feedback, are referred to as comparitors, and will be discussed later.



Ideal Op-Amp Characteristics

When ideal op-amps are used as the building blocks in circuit design and analysis, they can be characterized as shown in Figure 2.

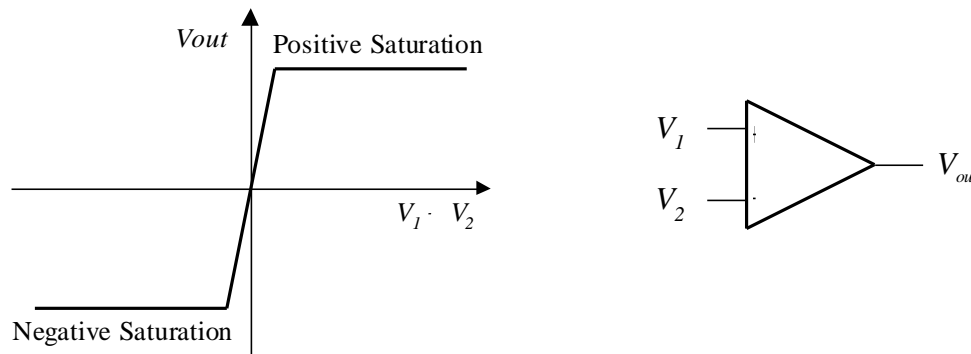


Figure 2. Op Amp Characteristics

The op-amp's open loop voltage gain, A_{ol} , is so high that a fraction of a millivolt *between* the input terminals V_1 and V_2 will drive the output into saturation.

The output of the op amp is given by $V_{out} = A_{ol}(V_1 - V_2)$, where A_{ol} is the open loop gain of the amplifier. Usually this is in the 10's or 100's of thousands.

For this reason the op-amp is usually used with feedback. This brings us to two fundamental concepts or rules that apply when working with op-amps that have negative feedback.

Rule 1 - The output attempts to drive its feedback circuit to make the voltage *differential* at the inputs zero. This means that the non-inverting and inverting inputs are always assumed to be equal or at the same potential.

This does not mean that the op-amp changes the voltage at its inputs, but rather the that output will swing either positive or negative to bring the *differential* voltage as close to zero as it can.

Rule 2 - The input to the op-amp will draw no current. This is the same thing as saying that the op-amp's input impedance is infinite.

By applying these two rules and basic DC and AC circuit analysis, we can analyze and/or design most op-amp circuits.



Basic Op-Amp Circuits

Voltage Follower

Figure 3. shows the simplest of op-amp circuits.

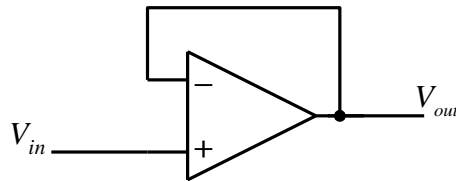


Figure 3. Voltage Follower

Despite its simplicity this circuit is very popular, and is the equivalent of an emitter follower, with the same impedance isolation or buffering advantages, namely high input impedance and low output impedance. Note; that the dual power supply connections are not shown. For *functional diagrams*, like this, they can be omitted, but for actual schematic diagrams they should always be shown or tabulated.

Voltage follower circuits are usually constructed using special op-amps, like the LM310 and the TL068 (which comes in a 3-pin transistor type package), designed specifically for this purpose.

In analyzing this circuit, we need only apply the two previously mentioned rules.

Rule 1 says the output will drive its feedback circuit (in this case a wire) to make the voltage difference at the input zero. This means that whatever the voltage signal applied to the circuit's input, V_{in} , in this case the non-inverting input, the output will apply exactly the same voltage signal to the inverting input, causing the difference between the inputs to stay at zero. The end result is that the output signal V_{out} is exactly the same as the input signal V_{in} .

Mathematically: $V_{out} = V_{in}$ or as a the transfer function

$$\frac{V_{out}}{V_{in}} = 1 \quad \text{meaning the gain of the circuit is 1.}$$



Inverting Amplifier

Figure 4 shows an op-amp configured as an inverting amplifier. This means that the output of the op-amp (V_{out}) is given by both amplifying and inverting (180° out of phase) the input signal (V_{in}).

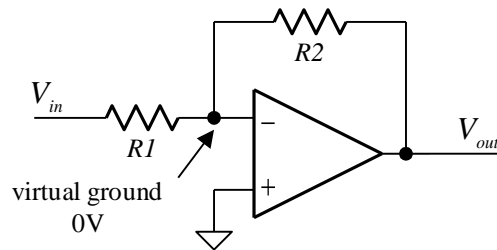


Figure 4. Inverting Amplifier

In analyzing this circuit we will apply both rules.

The non-inverting input of the op-amp is connected to ground then, by rule 1, the inverting input is also at ground level, or more accurately *virtual ground*. This then means that the voltage across $R1$ is equal to V_{in} ($V_{in} - 0$). Looking at the current, we can say that the current through $R1$ is equal to the current through $R2$ (rule 2 – zero input current).

$$I_1 = I_2$$

$$\frac{V_{in} - 0}{R1} = \frac{0 - V_{out}}{R2}$$

Mathematically:

$$\frac{V_{in}}{R1} = -\frac{V_{out}}{R2}$$

$$\frac{V_{out}}{V_{in}} = -\frac{R2}{R1}$$

If the voltage gain is defined as $\frac{V_{out}}{V_{in}}$ then the gain is equal to: $-\frac{R2}{R1}$.

This equation indicates that the output voltage is inverted (–) and has a gain factor (or attenuation) that is set by the resistor ratio $\frac{R2}{R1}$.



A second way of looking at the same circuit is to note that resistor $R1$ causes a current to flow of $I = \frac{V_{in}}{R2}$, and this current must flow through $R2$ (rule 2). The resistor $R2$ then produces a voltage of $V_{out} = -I \times R2$ referenced to ground.

This point of view will be useful when the feedback element is a device other than a resistor.



Non-Inverting Amplifier

Figure 5 is a non-inverting amplifier, and the analysis is again straightforward.

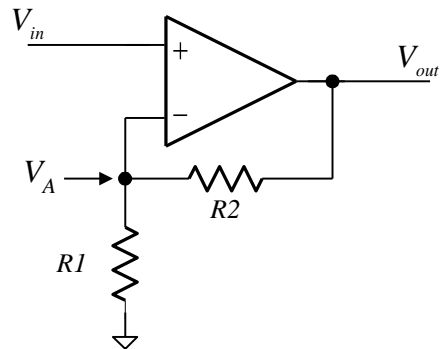


Figure 5. Non-Inverting Amplifier

Since V_A comes from a voltage divider made from $R1$ and $R2$, that is connected between V_{out} and ground.

$$\text{Then: } V_A = V_{out} \times \frac{R1}{R1 + R2}$$

But V_{out} drives the voltage divider such that

$$V_A = V_{in}$$

$$V_{out} \times \frac{R1}{R1 + R2} = V_{in}$$

$$\frac{V_{out}}{V_{in}} = \frac{R1 + R2}{R1}$$

$$\text{Therefore: } \frac{V_{out}}{V_{in}} = 1 + \frac{R2}{R1}$$

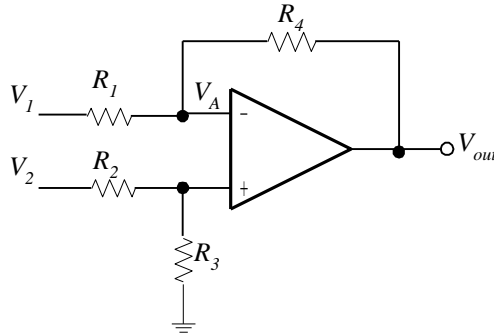
This equation says that the output voltage has a gain factor that is set by resistors $R1$ and $R2$ as follows $1 + \frac{R2}{R1}$. Note that in this circuit, the gain is always greater than unity.

Our analysis seems almost too simple, and in some ways this is true. Many important aspects of the circuits have been ignored. For example, in the inverting amplifier the input resistance is reduced to the value of $R1$, in other words the op-amp's high input impedance is lost in the final circuit. This is usually not a problem, but it is important to note that we are only looking at the functionality of the circuits, and not the complete picture.



Difference Amplifier

A difference amplifier provides an output that is proportional to the differential inputs V_1 and V_2 , or their mathematical difference $V_2 - V_1$.



Differential Amplifier Configuration

When we analyze the circuit we assume that the inverting and noninverting inputs are at the same potential, V_A , and that the amplifier inputs draw no current. The equations for the circuit become:

$$I_1 = \frac{V_1 - V_A}{R_1} = \frac{V_A - V_{out}}{R_4} \quad \text{and}$$
$$I_2 = \frac{V_2 - V_A}{R_2} = \frac{V_A - 0}{R_3}$$

Now solve both equations for V_A :

$$V_A = \frac{V_1 R_4 + V_{out} R_1}{R_1 + R_4} \quad \text{and} \quad V_A = \frac{V_2 R_3}{R_2 + R_3}$$

Equate the two equations and solve for V_{out} :

$$\frac{V_1 R_4 + V_{out} R_1}{R_1 + R_4} = \frac{V_2 R_3}{R_2 + R_3}$$
$$V_{out} = V_2 \frac{R_3}{R_1} \times \frac{R_1 + R_4}{R_2 + R_3} - V_1 \frac{R_4}{R_1}$$

This equation can be significantly simplified if we make $R_1 = R_2 = R_i$ and $R_3 = R_4 = R_f$. Then the equation becomes:



$$V_{out} = (V_2 - V_1) \frac{R_f}{R_i}$$

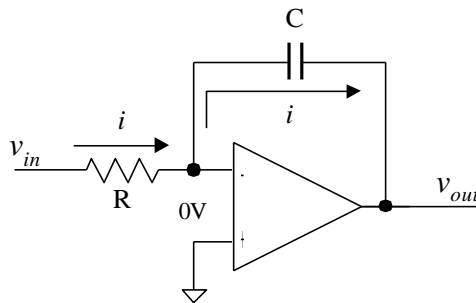
$$\frac{V_{out}}{V_2 - V_1} = \frac{R_f}{R_i}$$

Note that if we connected V_2 to ground we would have the same equation as for the inverting amplifier. Connecting a resistor to ground at the noninverting input does not affect the inverting amplifier's functionality since no current is flowing. The inverting amplifier can be considered a special configuration of the difference amplifier.



The Integrator

The actual circuit to do the integration is an operational amplifier with a capacitor as the feedback element (see figure below).



Resistor R produces a current proportional to its input voltage, and this current then charges the capacitor C . The output voltage of the circuit is the voltage across the capacitor produced by the current from R .

The voltage across the capacitor can then be expressed as:

$$v_C = \frac{1}{C} \int i \, dt ,$$

and the current as $i = \frac{v_{in}}{R}$

the output voltage $v_C = -v_{out}$

Substituting this in the above equation we get

$$-v_{out} = \frac{1}{C} \int \frac{v_{in}}{R} \, dt$$

or $v_{out} = -\frac{1}{RC} \int v_{in} \, dt$

This circuit has an output voltage proportional to the integral of the input voltage V_{in} , and has a gain of $-\frac{1}{RC}$.

Normally in simulation circuits we make $C=1\mu\text{F}$, and $R=1\text{M}\Omega$, so that the final integrating circuit has the following input/output relationship:

$$v_{out} = -\int v_{in} \, dt \quad \text{or an inverting integral.}$$

If we differentiate both sides of the equation, we obtain



$$\frac{d}{dt}v_{out} = -v_{in} \quad \text{or} \quad v_{in} = -\frac{d}{dt}v_{out}$$

which implies that the integrating block is the building block for the solution of a differential equation.

Capacitor

To understand how the integrator works, it is important to review how a capacitor charges. Recall that the charge Q on a capacitor is proportional to the charging current (I_c) and the time (t).

$$Q = I_c \times t$$

Also in terms of the voltage, the charge on a capacitor is

$$Q = C \times V_c$$

From these two relationships, the capacitor voltage can be expressed as

$$I_c \times t = C \times V_c$$

$$V_c = \frac{I_c \times t}{C}$$

$$V_c = \frac{I_c}{C} t$$

This expression has the form of an equation for a straight line that begins at zero with a

constant slope of $\frac{I_c}{C}$. Remember from algebra that the general formula for a straight line is

$y = mx + b$. In this case, $y = V_o$, $m = I_c/C$, $x = t$, and $b = 0$.

Recall that the capacitor voltage in a simple RC circuit is not linear but is exponential.

This is because the charging current continuously decreases as the capacitor charges and causes the rate of change of the voltage to continuously decrease. The key thing about using

an op-amp with an RC circuit to form an integrator is that the capacitor's charging current is made constant, thus producing a straight-line (linear) voltage rather than an exponential voltage. Now let's see why this is true.

In Figure 13-32, the inverting input of the op-amp is at virtual ground (0 V), so the voltage

across R_i equals V_{in} . Therefore, the input current is

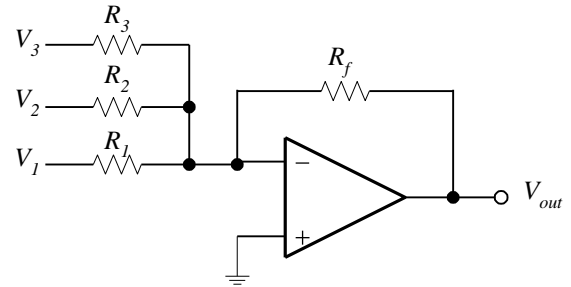


Summing Amplifier

A summing amplifier configuration is similar to the inverting amplifier, but the input resistance is now replaced with multiple input resistors connected to a summing node. If the noninverting input is connected to ground (making both inputs to the amplifier 0V) then the current equation is:

$$I = -\frac{V_{out}}{R_f} = \frac{V_1}{R_1} + \frac{V_2}{R_2} + \frac{V_3}{R_3}$$

giving
$$V_{out} = -\left(\frac{V_1}{R_1} + \frac{V_2}{R_2} + \frac{V_3}{R_3}\right) \times R_f$$



Summing Amplifier Configuration

and if we make $R_1 = R_2 = R_3 = R_i$ we get:
$$V_{out} = -\left(V_1 + V_2 + V_3\right) \times \frac{R_f}{R_i}$$

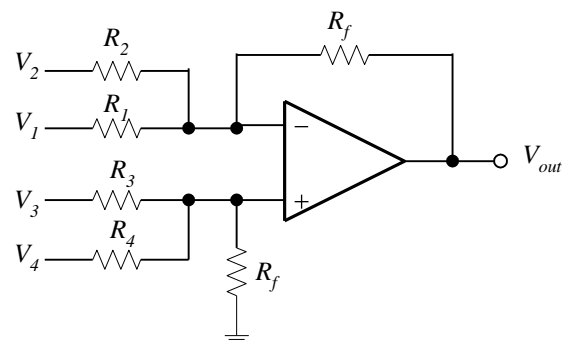
and if $R_i = R_f$ we get:
$$V_{out} = -\left(V_1 + V_2 + V_3\right)$$

The number of inputs can be increased or decreased as needed. The same principle will also apply to the noninverting amplifier, or in general to the differential amplifier.

Differential Summing Amplifier

The figure at right shows a differential summing amplifier. This configuration is capable of voltage addition, subtraction, and multiplication (gain). For simplicity the feed back resistors and the resistor connected to the noninverting input and ground are made equal.

The derivation of the relationship of the output voltage with respect to the input voltages is left as an exercise for the student.



Differential Summing Amplifier



VANIER
CÉGEP/COLLEGE
My choice. My future.

Electrical Engineering Technologies
Industrial Electronics
Instrumentation & Automation

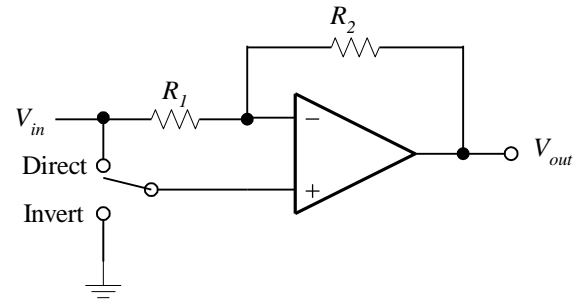
The differential summing amplifier is used in many applications where a linear transformation of the signal is required.



Inverting / Noninverting Amplifiers

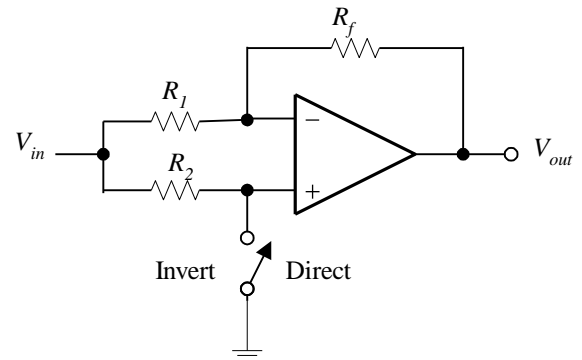
Signal inversion is often required as an option. The following circuits allow a signal to be switched between inverting and noninverting.

The figure at right shows a configuration that connects the signal to either the inverting or noninverting input of the operational amplifier. Resistors R_1 and R_f should be equal. In the noninverting mode resistors R_1 and R_2 do not interfere with the operation.



Inverting / Noninverting Amplifier

The figure at right shows an alternate configuration. Again R_1 and R_f should be equal for proper operation. The inverting configuration is exactly as the previous inverting amplifier, except the input is connected to an additional load of R_2 .



Inverting / Noninverting Amplifier



Current Source

Current sources are often required either as part of a specific design or as the final element of a transmitter. The transmitter is basically a current source, but it is difficult to control. Combining a transistor with a basic op-amp makes for a stable voltage controlled current source. The figure at right shows the basic configuration for a source. A current sink can be obtained by using a *pnp* transistor instead of a *nnp* transistor.

At the inputs to the op-amp we have voltage V_A , set by the voltage divider equation

$$V_A = \frac{R_2}{R_1 + R_2} V_{CC}$$

The output current is set by the resistor R_3 .

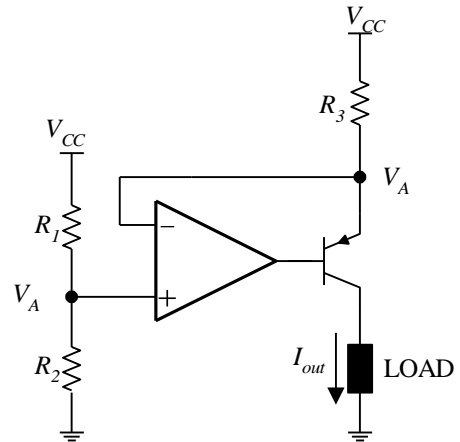
$$I_{out} = \frac{V_{CC} - V_A}{R_3}$$

Note that in this circuit, the operational amplifier is powered from a single V_{CC} power source

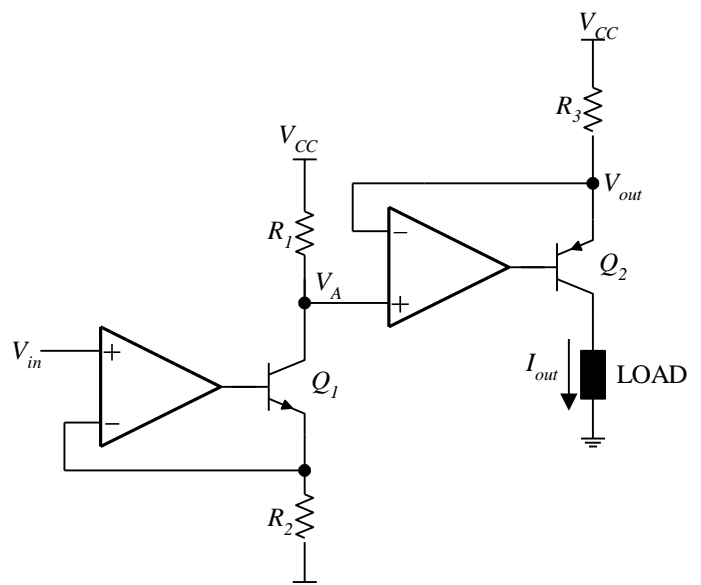
1 – 5 V to 4 – 20 mA Converter

With the above circuit the output current is proportional to the voltage drop below V_{CC} . To get a voltage to current converter you need a circuit with a variable input voltage. This problem is solved using the circuit at right.

The addition of another op-amp and transistor inverts V_A and scales it to the desired values.



Voltage Controlled Current Source



1 - 5 volt to 4 - 20 mA Converter



Let $V_A = 1 \text{ V}$ at a current of 20 mA, and with a power supply of $\pm 12 \text{ V}$ (24 V). Then

$$R_3 = \frac{V_{CC} - V_A}{I_{out}} = \frac{12 - 1}{20\text{m}} = 550 \, \Omega$$

and at 4 mA

$$V_A$$

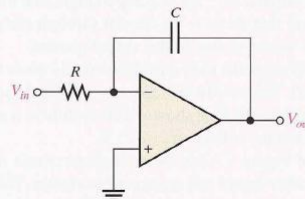


The Op-Amp Integrator

An ideal **integrator** is shown in Figure 13–31. Notice that the feedback element is a capacitor that forms an RC circuit with the input resistor. Although a large-value resistor is normally used in parallel with the capacitor to limit the gain, it does not affect the basic operation and is not shown for purposes of this analysis.

► **FIGURE 13–31**

An op-amp integrator.



How a Capacitor Charges To understand how the integrator works, it is important to review how a capacitor charges. Recall that the charge Q on a capacitor is proportional to the charging current (I_C) and the time (t).

$$Q = I_C t$$

Also, in terms of the voltage, the charge on a capacitor is

$$Q = CV_C$$

From these two relationships, the capacitor voltage can be expressed as

$$V_C = \left(\frac{I_C}{C} \right) t$$

This expression has the form of an equation for a straight line that begins at zero with a constant slope of I_C/C . Remember from algebra that the general formula for a straight line is $y = mx + b$. In this case, $y = V_C$, $m = I_C/C$, $x = t$, and $b = 0$.

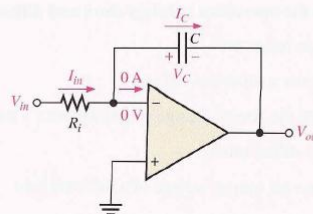
Recall that the capacitor voltage in a simple RC circuit is not linear but is exponential. This is because the charging current continuously decreases as the capacitor charges and causes the rate of change of the voltage to continuously decrease. The key thing about using an op-amp with an RC circuit to form an integrator is that the capacitor's charging current is made constant, thus producing a straight-line (linear) voltage rather than an exponential voltage. Now let's see why this is true.

In Figure 13–32, the inverting input of the op-amp is at virtual ground (0 V), so the voltage across R_i equals V_{in} . Therefore, the input current is

$$I_{in} = \frac{V_{in}}{R_i}$$

► **FIGURE 13–32**

Currents in an integrator.





If V_{in} is a constant voltage, then I_{in} is also a constant because the inverting input always remains at 0 V, keeping a constant voltage across R_i . Because of the very high input impedance of the op-amp, there is negligible current at the inverting input. This makes all of the input current go through the capacitor, as indicated in Figure 13–32, so

$$I_C = I_{in}$$

The Capacitor Voltage Since I_{in} is constant, so is I_C . The constant I_C charges the capacitor linearly and produces a linear voltage across C . The positive side of the capacitor is held at 0 V by the virtual ground of the op-amp. The voltage on the negative side of the capacitor, which is the op-amp output voltage, decreases linearly from zero as the capacitor charges, as shown in Figure 13–33. This voltage is called a *negative ramp* and is the consequence of a constant positive input.

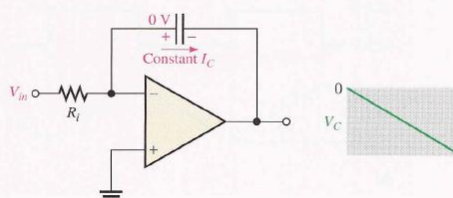


FIGURE 13–33

A linear ramp voltage is produced across C by the constant charging current.

The Output Voltage V_{out} is the same as the voltage on the negative side of the capacitor. When a constant positive input voltage in the form of a step or pulse (a pulse has a constant amplitude when high) is applied, the output ramp decreases negatively until the op-amp saturates at its maximum negative level. This is indicated in Figure 13–34.

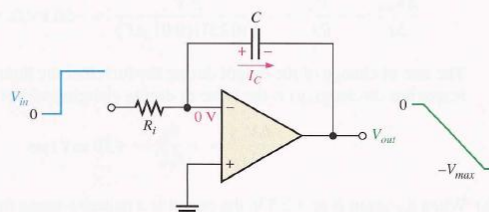


FIGURE 13–34

A constant input voltage produces a ramp on the output of the integrator.

Rate of Change of the Output The rate at which the capacitor charges, and therefore the slope of the output ramp, is set by the ratio I_C/C , as you have seen. Since $I_C = V_{in}/R_i$, the rate of change or slope of the integrator's output voltage is $\Delta V_{out}/\Delta t$.

$$\frac{\Delta V_{out}}{\Delta t} = -\frac{V_{in}}{R_i C}$$

Equation 13–7

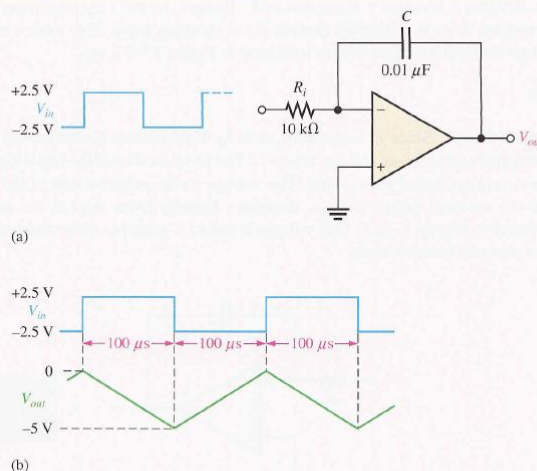
Integrators are especially useful in triangular-wave oscillators as you will see in Chapter 16.



EXAMPLE 13-10

- (a) Determine the rate of change of the output voltage in response to the input square wave, as shown for the integrator in Figure 13-35(a). The output voltage is initially zero. The pulse width is $100\ \mu\text{s}$.

► **FIGURE 13-35**



- (b) Describe the output and draw the waveform.

Solution (a) The rate of change of the output voltage during the time that the input is positive (capacitor charging) is

$$\frac{\Delta V_{out}}{\Delta t} = -\frac{V_{in}}{R_f C} = -\frac{5\text{ V}}{(10\text{ k}\Omega)(0.01\ \mu\text{F})} = -50\text{ kV/s} = -50\text{ mV}/\mu\text{s}$$

The rate of change of the output during the time that the input is negative (capacitor discharging) is the same as during charging except it is positive.

$$\frac{\Delta V_{out}}{\Delta t} = +\frac{V_{in}}{R_f C} = +50\text{ mV}/\mu\text{s}$$

- (b) When the input is at $+2.5\text{ V}$, the output is a negative-going ramp. When the input is at -2.5 V , the output is a positive-going ramp.

$$\Delta V_{out} = (50\text{ mV}/\mu\text{s})(100\ \mu\text{s}) = 5\text{ V}$$

During the time the input is at $+2.5\text{ V}$, the output will go from 0 to -5 V . During the time the input is at -2.5 V , the output will go from -5 V to 0 V . Therefore, the output is a triangular wave with peaks at 0 V and -5 V , as shown in Figure 13-35(b).

Related Problem Modify the integrator in Figure 13-35 to make the output change from 0 to -5 V in $50\ \mu\text{s}$ with the same input.



Open the Multisim file E13-10 in the Examples folder on your CD-ROM. Using the function generator, apply the indicated pulse waveform to the input and verify that the output voltage is correct.



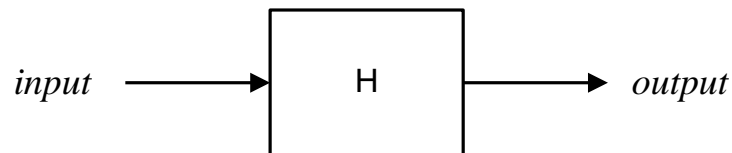
The decibel (dB)

Transfer Function

Recall that the **transfer function** (or gain) of a system is the ratio of the output of the system over the input of the system, and is usually represented as H .

Mathematically: Transfer Function = $\frac{\text{output}}{\text{input}} = H$

Graphically in block diagram form this is represented as:



The transfer function is dimensionless, provided that both input and output are specified or measured in the same units. In electronics we normally measure input and output in watts (power), volts, or amps (current).

If the output power (voltage or current) of a system is greater than the input power (voltage or current), the signal is said to be **amplified**. If the output power of a system is less than the input power, then the signal is said to be **attenuated**. The numeric ratio of the transfer function of a system can sometimes be very large or very small making it inconvenient to express the ratio as a simple numbers. An alternative is to express the ratio as a logarithmic function, compressing the range.

The decibel (dB)

The bel or decibel (dB) is named in honour of Alexander Graham Bell. (note that the 'unit' is spelled with one l).

This value describes the logarithmic ratio of the attenuation, or gain of signals.

The bel is defined as $bel = \log\left(\frac{P_2}{P_1}\right)$ note: log here is to the base 10

normally P_1 is the reference to which P_2 is compared.



In practice the more convenient unit of decibel (dB) is used (*deci* meaning $\frac{1}{10}$). In effect, 10 dB = 1 Bel or

$$A_P(\text{dB}) = 10 A_P(\text{Bel})$$

$$A_P(\text{dB}) = 10 \log \frac{P_{out}}{P_{in}}$$

Since dB is a ratio, it is really dimensionless. However when using this method to calculate attenuation, gain, or any other ratio we do speak of dB as though it were a 'unit'. In reality it describes a method of expressing or calculating a ratio.

Example:

What is the power gain, in decibels, of a system that has an input power of 1 mW and an output power of 500 mW ?

$$A_P(\text{dB}) = 10 \log \frac{P_{out}}{P_{in}} = 10 \log \frac{500}{1} = 10 \log 500 = 27 \text{ dB}$$

Note that the equal sign is not mathematically correct, unless we clearly specify dB. The dB designation, then clearly, defines how the ratio is calculated.

At this point, you may be wondering what the big advantage of the decibel system is. To answer this, recall a few log identities: In the log system, normal multiplication becomes addition, division becomes subtraction, and powers and roots become multiplication and division respectively. Because of this, two important facts arise, first, ratios of change become constant offsets in the decibel system and second, the entire range of values diminishes in size. The result is that a very wide range of gains can be represented within a fairly small scope of values, and the corresponding calculations can be done easier and faster.

With the aid of your hand calculator, it is easy to show the following:

Signals expressed in watts (power)	
Gain Factor G	dB Value $G_{dB} = 10 \log (G)$
1	0 dB
2	3.01 dB
4	6.02 dB
8	9.03 dB
10	10.0 dB



We can also look at fractional factors or losses (attenuation) instead of gains:

Signals expressed in watts (power)	
Attenuation Factor G	dB Value $G_{dB} = 10 \log (G)$
0.5	-3.01 dB
0.25	-6.02 dB
0.125	-9.03 dB
0.1	-10.0 dB

There are two values, which are useful to commit to memory. If you look carefully at the tables of gain/attenuation factors, you will notice that a doubling is represented by an increase of approximately 3 dB. A multiplier of 4 is, in essence, two doublings and therefore, it is equivalent to 3 dB + 3 dB, or 6 dB. Remember that since we are using log, multiplication turns into simple addition. In a similar manner, a halving is represented by approximately -3 dB. The negative sign indicates a reduction or attenuation. To simplify things, think of multipliers/dividers of 2 as ± 3 dB, with the sign indicating whether you are increasing (multiplying) by 2, or decreasing (dividing) by 2. Also notice a multiplier of 10 works out to a very convenient 10 dB. By remembering these two relationships you can often estimate a decibel conversion without the use of your calculator.

Example:

*An amplifier produces an output power level of 200mW for an input of 10mW.
What is the power gain in decibels?*

The amplifier has a gain of $\frac{200}{10} = 20$

20 can be written as 2×10

The factor of 2 is 3 dB, and the factor of 10 is 10 dB

The answer can then be estimated to be 3 dB + 10 dB, or 13 dB

This can be verified by a direct calculation: $G_{dB} = 10 \times \log \left(\frac{200}{10} \right) = 13.01$ dB

Example:

An amplifier has a power gain of 800. What is its decibel power gain ?

$G(dB) = 10 \times \log 800 = 10 \times 2.903 = 29.03$ dB

we could also use our estimation technique:



$$G = 800 = 8 \times 10^2$$

8 is equivalent to 3 factors of 2, or $2 \times 2 \times 2$, which can be expressed as:

3 dB + 3 dB + 3 dB, which is, of course, 9 dB

100 is equivalent to 2 factors of 10, or $10 \text{ dB} + 10 \text{ dB} = 20 \text{ dB}$

The result is $9 \text{ dB} + 20 \text{ dB} = 29 \text{ dB}$.

Note that if the leading digit is not a power of 2, the estimation will not be exact. For example, if the gain is 850, you know that the decibel gain is just a bit over 29 dB. You also know that it must be less than 30 dB ($1000 = 10^3$ which is 3 factors of 10 or 30 dB). As you can see, by using the decibel form you tend to concentrate on the *magnitude* of the gain, and not so much on trailing digits.

Example:

An attenuator reduces signal power by a factor of 10,000.

Express this power loss in decibels?

$$G(\text{dB}) = 10 \log \left(\frac{1}{10,000} \right) = 10 \times (-4) = -40 \text{ dB}$$

By using the approximation we can say; $\frac{1}{10,000} = \frac{1}{10^4} = 10^{-4}$

The negative exponent tells us we have a loss (negative decibel value), of 4 factors of 10. Then we can use the following calculation:

$$G(\text{dB}) = -(10 + 10 + 10 + 10) = -40 \text{ dB}$$

Remember if an increase in signal is produced, the output signal is greater than the input signal, the result will be a positive decibel value. A decrease in signal, the output signal is less than the input signal, will result in a negative decibel value. A signal, which is unchanged, (output signal is equal to the input signal, indicates a gain of unity or 0 dB.

To convert from decibels to ordinary form, just invert the steps.

Mathematically: $G = \log^{-1} \left(\frac{G(\text{dB})}{10} \right)$

On most hand calculators, base 10 antilog is denoted as 10^x . In most computer languages and spreadsheets, you just raise 10 to the appropriate power.

Example

An amplifier has a power gain of 23 dB. If the input power is 1 mW, what is the output power?



In order to find the output power we first need to find the power gain ratio.

$$G = \log^{-1}\left(\frac{G(dB)}{10}\right) = \log^{-1}\left(\frac{23}{10}\right) = 199.5$$

Therefore: $P_{out} = G \times P_{in} = 199.5 \times 1 \text{ mW}$ or 199.5 mW.

We can also use the approximation technique in reverse. To do this, break up the decibel gain into chunks of 10 dB and 3 dB:

$$23\text{dB} = 3\text{dB} + 10\text{dB} + 10\text{dB}$$

Now replace each chunk with the appropriate factor, and *multiply* them together.

(Remember: when going from log to ordinary form, addition turns into multiplication.)

$$3 \text{ dB} = 2\times \quad \text{and} \quad 10 \text{ dB} = 10\times$$

$$\text{then } G = 2 \times 10 \times 10 = 200$$

While the approximation technique appears to be slower than the calculator, practice will show otherwise. Being able to quickly estimate decibel values will prove to be a handy skill in the electronics field. This is particularly true in larger, multistage systems.

Example:

A three-stage amplifier has gains of 10 dB, 16 dB, and 14 dB respectively.

What is the total decibel gain ?

Since decibel gains are a log form, just add the individual stage gains to arrive at the system gain:

$$G(dB) = 10 + 16 + 14 = 40 \text{ dB}$$

As you may have noticed, all of the examples up to this point have used power gain and not voltage gain. You may be tempted to use the same equations for voltage gain. In a word, **DON'T**.



The dB voltage ratio

The voltage gain of a system can also be expressed in dB. To arrive at the voltage gain equation, we start with the definition of dB and the power equation. The equation we need has power in terms of resistance and voltage.

$$P = \frac{V^2}{R}$$

and substituting this in the dB definition gives

$$G(dB) = 10 \times \log \left[\frac{V_2^2 / R_2}{V_1^2 / R_1} \right]$$

Where R_2 and R_1 are the equivalent resistances in which the powers P_2 and P_1 are dissipated, and V_2 and V_1 are the voltages across R_2 and R_1 respectively.

And if $R_1 = R_2$ then we get

$$G(dB) = 10 \times \log \frac{V_2^2}{V_1^2} = 10 \times \log \left(\frac{V_2}{V_1} \right)^2 = 20 \times \log \frac{V_2}{V_1}$$

or $G(dB) = 20 \log \frac{V_2}{V_1}$

It is a good idea at this point to construct the previous tables using voltage gain

Signal expressed in volts (voltage) (see above)	
Gain Factor G	dB Value $G_{dB} = 20 \log(G)$
1	0 dB
2	6.02 dB
4	12.04 dB
8	18.06 dB
10	20.0 dB



And again for attenuation:

Signal expressed in volts (voltage) (see above)	
Attenuation Factor G	dB Value $G_{dB} = 20 \log (G)$
0.5	-6.02 dB
0.25	-12.04 dB
0.125	-18.06 dB
0.1	-20.0 dB

Example:

An amplifier has a 2 V output signal for an input signal of 50 mV.

What is its gain in dB ?

First find the gain ratio:

$$G = \frac{2}{.05} = 40$$

Now convert to decibel form.

$$G(dB) = 20 \log 40 = 20 \times 1.602 = 32.01 \text{ dB}$$

The approximation technique yields:

$$40 = 2 \times 2 \times 10, \text{ or } 6 \text{ dB} + 6 \text{ dB} + 20 \text{ dB} = 32 \text{ dB.}$$

Again the same can be done in reverse.

Example:

An amplifier has a gain of 26 dB. If the input signal has a value of 10 mV.

What is the output ?

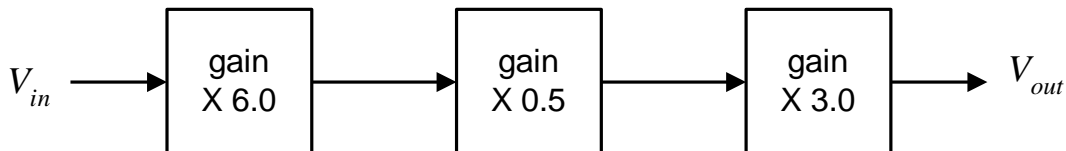
$$G_{\text{volts}} = \log^{-1} \left(\frac{G(dB)}{20} \right) = \log^{-1} \left(\frac{26}{20} \right) = 19.95$$

$$V_{\text{out}} = 19.95 \times 10 \text{ mV} = 199.5 \text{ mV}$$



Cascading multiple gain block.

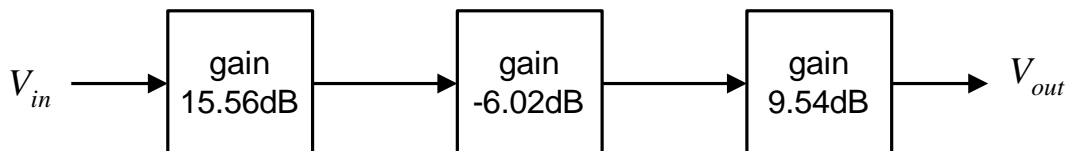
Consider the following system with the gains as indicated:



The total gain can be calculated as

$$\frac{V_{out}}{V_{in}} = 6 \times 0.5 \times 3 = 9 \quad \text{or} \quad 20 \times \log(9) = 19.08 \text{ dB}$$

The same calculation can be done by first converting each block to its dB equivalent as follows:



Now the total gain can be calculated as

$$\frac{V_{out}}{V_{in}} = 15.56 + (-6.02) + 9.54 = 19.08 \text{ dB, giving the same result.}$$

Recall from mathematics that $\log(a \times b) = \log a + \log b$

This will become very convenient later.



dBm

Often measurements are made with an established point of reference. Power gain is normally measured in reference to 1mW, and we speak of dBm to indicate this.

Then
$$dBm = 10 \times \log \left(\frac{P_2}{1mW} \right)$$

This unit is commonly used to specify output signals of amplifiers or transmitters.

VU

When the dBm is used to measure a sinusoidal signal into a 600 Ω resistive load, we refer to it as Volume Unit or VU. This term is commonly used in the audio field, and we see this “unit” on audio equipment.

Signal Representation in dBW and dBV

As you can see from the preceding section, it is possible to spend considerable time converting between decibel gains and ordinary voltages and powers. Since the decibel form does offer advantages for gain measurement, it would make sense to use a decibel form for power and voltage levels as well. This is a relatively straightforward process. There is no reason why we can't express a power or voltage in a logarithmic form. Since a decibel value simply indicates a ratio, all we need to do is decide on a reference (ie, a comparative base for the ratio). For power measurements, a likely choice would be 1 watt. In other words we can describe a power as being x decibels above or below 1 watt. Positive values will indicate power levels of greater than 1 watt, while negative values will indicate power levels of less than 1 watt. In general equation form:

$$P(dB) = 10 \log \left(\frac{P}{reference} \right)$$

The answer will have units of dBW, when the reference is 1 watt.

Example:

*A power amplifier has a maximum output power of 120 W.
What is this power expressed in dBW ?*

$$P(dB) = 10 \log \left(\frac{P}{reference} \right) = 10 \log \left(\frac{120W}{1W} \right) = 20.8 \text{ dBW}$$



There is nothing sacred about the 1-watt reference, short of its convenience. We could just as easily choose a different reference as was seen earlier.

Example:

*A small personal audio tape player delivers 200 mW to its headphones.
What is this output power in dBW ? and in dBm ?*

For an answer in units of dBW, use the 1-watt reference:

$$P(\text{dB}) = 10 \log \left(\frac{P}{\text{reference}} \right) = 10 \log \left(\frac{200 \text{ mW}}{1 \text{ W}} \right) = -7 \text{ dBW}$$

For units of dBm, use a 1-milliwatt reference:

$$P(\text{dB}) = 10 \log \left(\frac{P}{\text{reference}} \right) = 10 \log \left(\frac{200 \text{ mW}}{1 \text{ mW}} \right) = 23 \text{ dBm}$$

200 mW, -7 dBW, and 23 dBm are three ways of expressing the same thing. Note that the dBW and dBm values are 30 dB apart. This will always be true since the references are a factor of 1000 (30 dB) apart.

In order to transfer a value in dBW (or similar units) into watts, reverse the process:

$$P = \log^{-1} \left(\frac{P(\text{dB})}{10} \right) \times \text{reference}$$

Example:

*A studio microphone produces a 12 dBm signal while recording normal speech.
What is the output power in watts ?*

$$P = \log^{-1} \left(\frac{P(\text{dB})}{10} \right) \times \text{reference} = \log^{-1} \left(\frac{12 \text{ dBm}}{10} \right) \times 1 \text{ mW} = 15.8 \text{ mW or } 0.0158 \text{ W}$$

For voltages we can use a similar system. A logical reference is 1 volt with the resulting units being dBV. As before, these voltage measurements will require a multiplier of 20 instead of 10:

$$V(\text{dB}) = 20 \log \left(\frac{V}{\text{reference}} \right)$$



Example

A test oscillator produces a 2 volt output signal. What is the signal value in dBV?

$$V(dB) = 20 \log \left(\frac{V}{\text{reference}} \right) = 20 \log \left(\frac{2V}{1V} \right) = 6.02 \text{ dBV}$$

When both circuit gains and signal levels are specified in decibel form, analysis can be very quick. Given an input level, simply add the gain to it in order to find the output level. Given input and output levels, subtract them in order to find the gain.

Example

A floppy disk read/write amplifier exhibits a gain of 35 dB.

If the input signal is -43 dBV, what is the output signal ?

$$V_{out}(dB) = V_{in}(dB) + A(dB) = -43 \text{ dBV} + 35 \text{ dB} = -7 \text{ dBV}$$

Note that the final units are dBV and not dB, thus indicating a voltage and not merely a gain.

Example

A guitar power amplifier needs an input of 20 dBm to achieve an output of 25 dBW.

What is the gain of the amplifier in decibels ?

First, it is necessary to convert the power readings so that they share the same reference unit. Since dBm represents a reference 30 dB below the dBW reference, just subtract 30dB to compensate.

$$20 \text{ dBm} = -10 \text{ dBW}$$

$$G_{dB} = P_{out} - P_{in} = 25 \text{ dBW} - (-10 \text{ dBW}) = 35 \text{ dB}$$

Note that the units are dB and not dBW. This is very important. Saying that the gain is “so many” dBW is the same as saying the gain is “so many” watts. Remember that gains are “pure” numbers and do not carry units such as watts or volts.



In the Lab

To make life in the lab even easier, it is possible to take measurements directly in decibel form. If you do this, you need not convert when troubleshooting a design. For general purpose work, voltage measurements are the norm, and therefore a dBV scale is often used.

When using a digital meter on a dBV scale it is possible to “underflow” the meter if the signal is too weak. This will happen if you try to measure 0 volts for example. If you attempt to calculate the corresponding dBV value your calculator will probably show “error.” The effective value is negative infinite dBV. The meter will certainly have a hard time showing this value. Another item of interest revolves around the use of dBm measurements. It is common to use a voltmeter to make dBm measurements, in lieu of a wattmeter. While the connections are considerably simpler, a voltmeter cannot measure power. As long as the circuit impedance is known, power can be derived from a voltage measurement. A common impedance in audio and communication systems (such as recording studios) is 600 Ω , so a meter can be calibrated to give correct dBm readings by using the Power Law. If this meter is used on non-600 Ω circuit the reading will no longer reflect accurate dBm values (but will still properly reflect changes in dBm).



Sample Question

1. A decibel
 - a) is equal to ten bels
 - b) is one tenth of a bel
 - c) is never used in calculating power gain
 - d) is never used in calculating voltage gain

2. The basic gain equation is
 - a) $\text{gain} = \frac{\text{output}}{\text{input}}$
 - b) $\text{gain} = \frac{\text{input}}{\text{output}}$
 - c) $\text{gain} = \frac{\text{power output}}{\text{voltage output}}$
 - d) $\text{gain} = \frac{\text{power output}}{\text{voltage input}}$

3. An amplifier produces an output power level of 200mW for an input of 10mW.
What is the amplifiers gain expressed in dB.

4. An amplifier has a power gain of 800.
What is its decibel power gain ?

5. An attenuator reduces signal power by a factor of 10,000.
What is this loss expressed in decibels ?

6. An amplifier has a power gain of 23 dB.
If the input is 1 mW, what is the output ?



7. A three-stage amplifier has gains of 10 dB, 16 dB, and 14 dB per section.
What is the total decibel gain ?
8. An amplifier has an output signal of 2 V for an input of 50 mV.
What is its gain in dB ?
9. An amplifier has a gain of 26 dB. If the input signal is 10 mV.
What is the output ?
10. A power amplifier has a maximum output of 120 W.
What is this power in dBW ?
11. A small personal MP3 player delivers 200 mW to its headphones.
What is this output power in dBW and in dBm ?
12. A studio microphone produces a 12 dBm signal while recording normal speech.
What is the output power in watts ?
13. A test oscillator produces a 2 volt signal.
What is this value in dBV ?
14. A disk read/write amplifier exhibits a gain of 35 dB. The input signal is -43 dBV,
What is the output signal ?
15. A guitar power amplifier needs an input of 20 dBm to achieve an output of 25 dBW.
What is the gain of the amplifier in decibels ?
16. What is the decibel power gain of 200 ?
17. An attenuator reduces signal power by a factor of 25.
What is the loss expressed in decibels ?
18. An amplifier has a power gain of 23 dB, and the input is 12W.
What is the input power ?



19. A three-stage amplifier has gains of 10 dB, 16 dB, and 14 dB per section. What is the total decibel gain ?
20. An amplifier has an output signal of 0.7 V_{rms} for an input of 175 mV_{rms}. What is the gain expressed in decibels ?
21. An amplifier has a gain of 20 dB. If the input signal is 10 mV. What is the output voltage ?
22. An audio amplifier delivers 34 dBm of power into a 16 Ω speaker.
 - a) What is the output power in watts ?
 - b) What is the output voltage ?
23. Compute the power level of a signal that is 30 dBm.
24. The input power of an amplifier is 10 mW when the output power is 1W. What is the gain expressed in dB ?
25. A transducer with a gain of 10 dB is connected to a transmitter amplifier with a gain of 14 dB. The transmission line attenuates the signal by 3 dB; it is then connected to a receiving amplifier with a gain of 35 dB.
 - a) Draw a block diagram of the system.
 - b) Develop the transfer function of the system
26. The input power of an amplifier is 1 mW, and the amplifier's output power is 150 mW. What is the gain expressed in dB ?
27. A passive network has an attenuation of 1.5 dB. If the input power is 150 mW. What is the output power from the network ?
28. Compute the power, in watts, of a signal that is 15 dBm.



Signals and Noise

There are essentially two variables or points of view that can be used in describing and classifying signals. These are the amplitude versus time, and the amplitude versus frequency (or frequency distribution).

In the discussion of signals, the subject of noise is unavoidable. In fact in most cases it is easier to define a signal by what is not part of the signal than to actually define the signal of interest. This brings us directly to the topic of noise.

Noise

Noise can then be described as any signal that interferes with the signal of interest.

The three main sources of noise encountered when making measurements are: *interference signals*, *drift noise*, and *device noise*.

Interference signals consist of any unwanted analog signal that is coupled to the signal of interest. It is usually random in nature, and unpredictable. An example of this would be a motor that starts and stops as part of its normal operation and results in large current surges on the power lines. Each time the motor starts or stops, it produces a magnetic field that couples inductively into the electrical measurement. The result is an erroneous measurement. A more common example is the 60 Hz coupling of power transmission throughout a manufacturing facility.

Drift noise is primarily due to the finite instability of electronics and transducers used in a measurement chain. Electronic components tend to vary in stability over time. Temperature, primarily, as well as other environmental changes are often the cause of drift. As a result, any measurement or signal processing conducted over a period of time may be subject to drift and related instability.

Device noise is common to any electronic component, circuit, or transducer. Device noise is caused by individual charge carrying electrons in components and in conductors. This random collision produces variations in currents, and in turn voltages. Device noise is most often analyzed from a statistical point of view.

Device noise, interference signals, and drift noise all limit the ultimate sensitivity and performance of a measurement chain. Thus we need to understand how to detect and measure noise.

One of the most straightforward and frequently used expressions of the relation of a signal to its noise content is the **signal-to-noise ratio (SNR)**. The signal to noise ratio is defined as the ratio of a signal's power to its noise and is normally expressed in dB.

$$SNR = 10 \times \log \left(\frac{P_{signal}}{P_{noise}} \right)$$



Noise Detection and Classification

Both signal and noise must of course be measured in the same units. Signal or noise amplitudes are usually measured in units of volts. The frequency of a signal or noise is measured in Hertz (cycles per second). By plotting the amplitude versus frequency (or time) of the composite signal and noise waveforms, one can usually distinguish the noise from the signal.

Graphically the two points of view are illustrated in the following figures. Figure 1 illustrates a graph of a signal amplitude versus time plot, and is usually referred to as time domain representation. The graph shows a 2 Hz sine wave superimposed on a 16 Hz sine wave. Depending on the application, either sine wave can be considered the signal or the noise of the composite signal. Later we will learn to separate or filter to extract either sine wave. Figure 2 illustrates a similar signal. However this time the separation of the two sine waves is not that obvious. The graph shows a 25 Hz sine wave superimposed on a 50 Hz sine wave. The two graphs show the same type of representation that is obtained by most oscilloscopes and chart recorders. It shows the signal amplitude variation as a function of time, or as time passes. Figure 3 illustrates the same signal as Figure 2, but this time the signal is transformed into its frequency domain. This type of representation clearly shows the two different frequencies that compose the signal. This type of graph shows the signal amplitude as a function of its frequency.

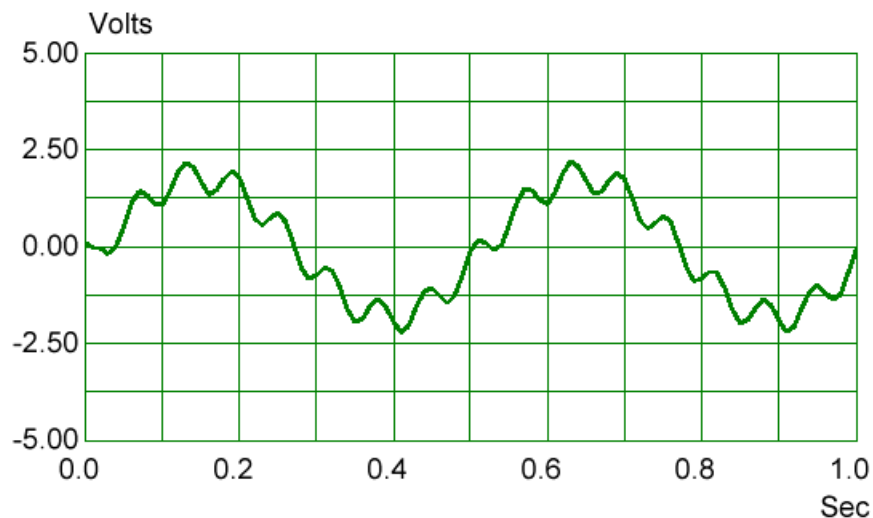


Figure 1. Time Domain Graph of a Noisy Sine Wave.

Many signals contain a variety of frequencies of interest, and the noise is usually distributed over a wider range of frequencies than the signal. The frequency domain representation can usually be obtained with the use of a spectrum analyzer. A spectrum analyzer is an instrument that transforms a time domain signal into an amplitude versus frequency plot usually in real time.



Spectrum analyzers that work in real time are very expensive. However, the transformation can be defined mathematically and performed on any personal computer, as was done to obtain the frequency domain graphs shown below.

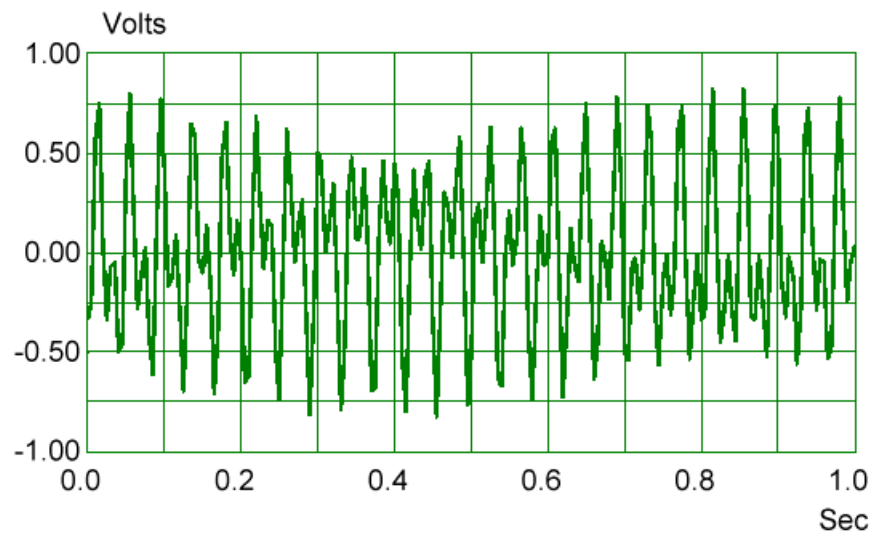


Figure 2. Time Domain Graph of 25 Hz and 50 Hz sine waves

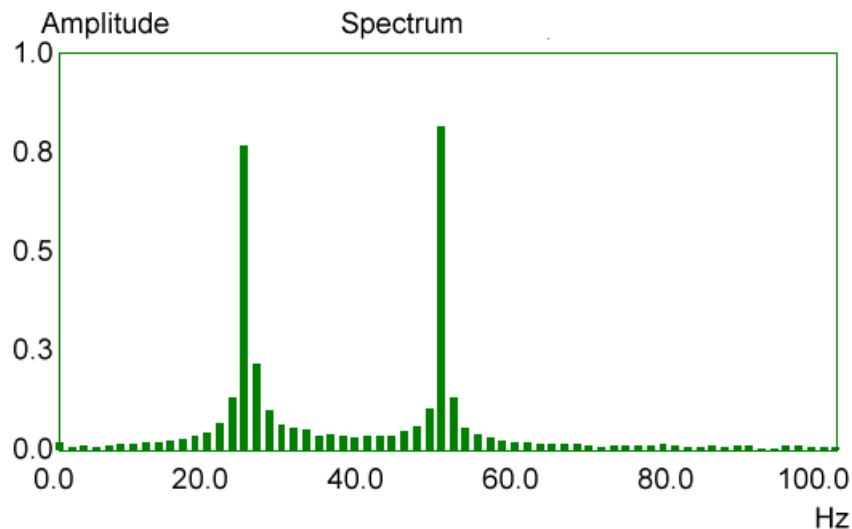


Figure 3. Frequency Domain Graph of the 25 Hz and 50 Hz sine waves.

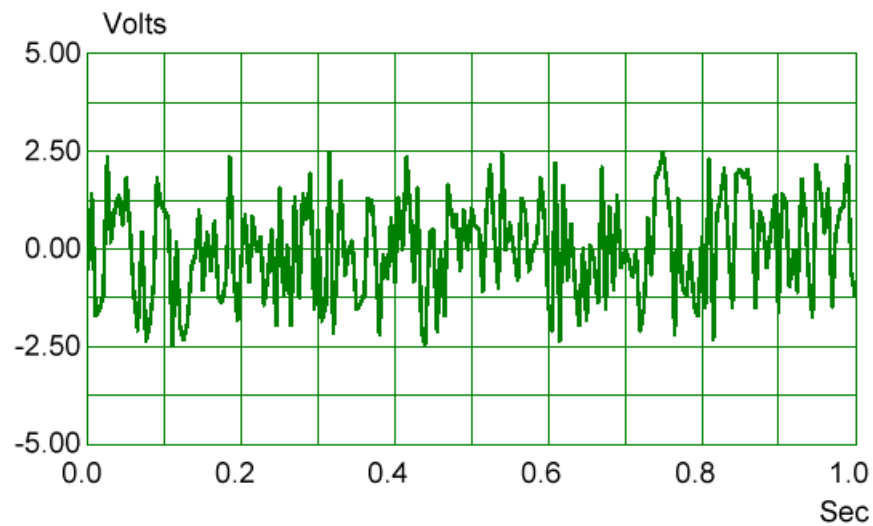


Figure 4. Time Domain Graph of random noise.

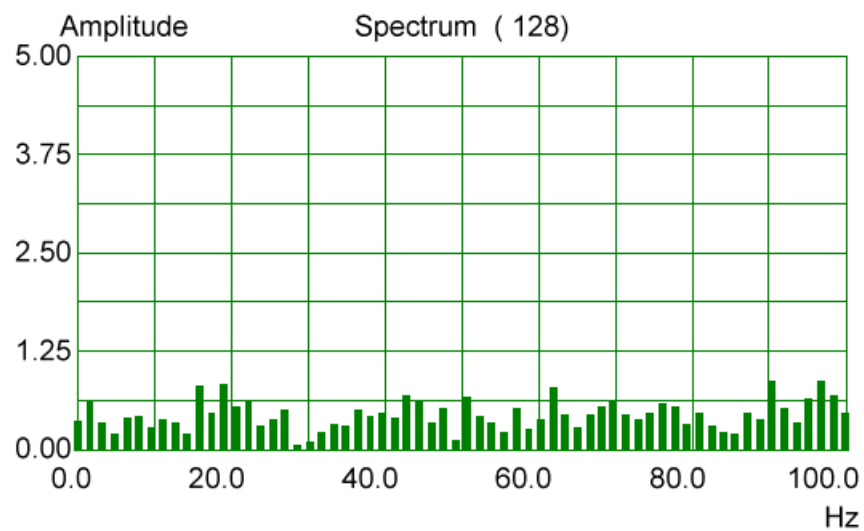


Figure 5. Frequency Domain Graph of random noise.

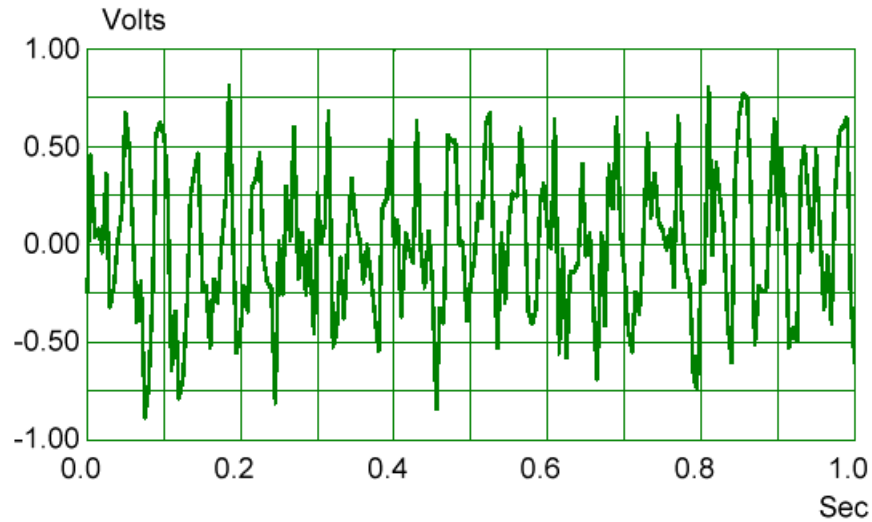


Figure 6. Time Domain Graph of a 25 Hz sine wave signal, with random noise added is shown.

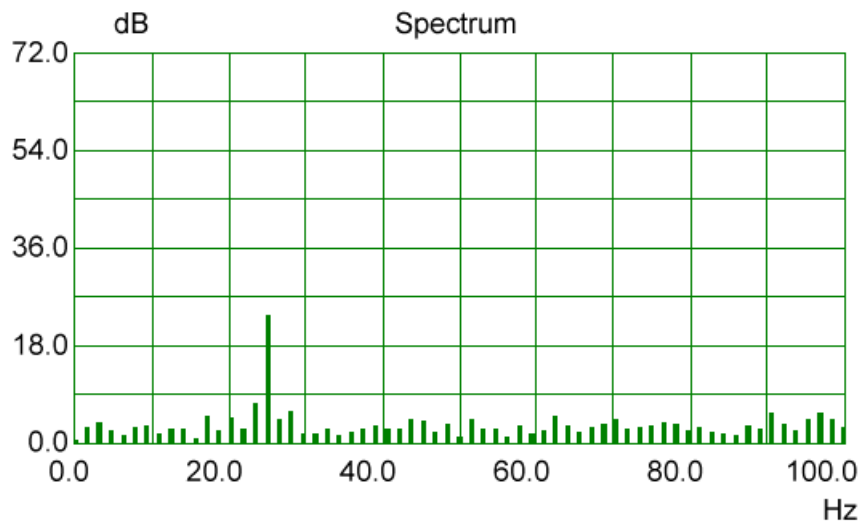


Figure 7. Frequency Domain Graph of a 25 Hz sine wave, with added random noise is shown.
The spike represents the 25 Hz sine wave signal.



Types of Signal Sources

When configuring signals or making signal connections, first determine whether the signal source is floating or ground referenced. A description of these two types of signals is given below.

Floating Signal Sources

A floating signal source does not connect in any way to earth ground but has an isolated ground-reference point. Some examples of floating signal sources are transformer outputs, thermocouples, battery-powered devices, optical isolator outputs, and isolation amplifiers. The ground reference of a floating signal should be tied to the analog input ground to establish a local or reference for the signal. Otherwise, the measured input signal varies or appears to float. An instrument or device that supplies an isolated output falls into the floating signal source category.

Ground-Referenced Signal Sources

A ground-referenced signal source connects in some way to earth ground and is, therefore, already connected to a common ground point with respect to the instrument

Nonisolated outputs of instruments and devices that plug into the power supply, fall into this category. The difference in ground potential between two instruments connected to the same power supply is typically between 1 and 100 mV but can be much higher if power distribution circuits are not properly connected. The following connection instructions for grounded signal sources eliminate this ground potential difference from the measured signal.

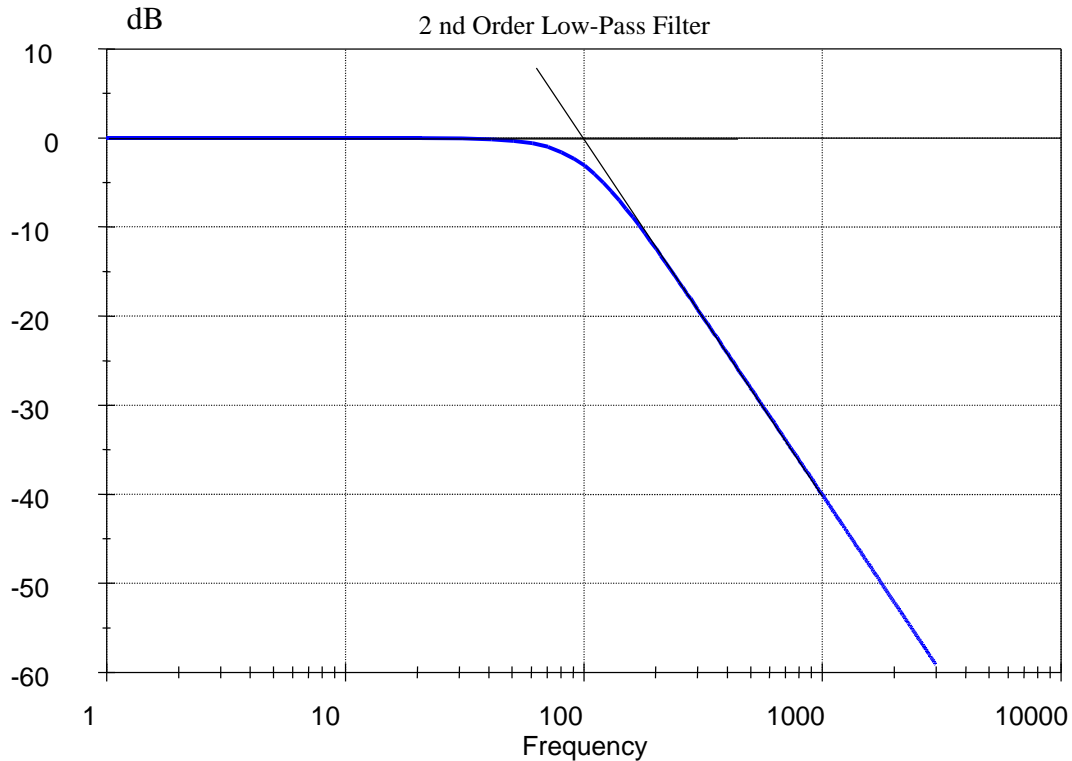


Signal Source Type		
Input	Floating Signal Source (Not Connected to Earth Ground)	Grounded Signal Source
	Examples Ungrounded Thermocouples Signal Conditioning with Isolated Output Battery devices	Examples Plug-in Instruments with Non-isolated Outputs
Differential		
Referenced Single-Ended Ground		<p>NOT RECOMMENDED</p> <p>Ground-Loop Losses, V_g, are added to measured signal</p>
NON-Referenced Single-Ended Ground		



Bode¹ Plots

This method of representing the frequency response of filters, systems or signals simplifies interpretation, and saves a tremendous amount of time. Through the use of straight line segments, the frequency amplitude and phase relationship of a system can be found efficiently and accurately.



¹ Named after Hendrick Wade Bode (1905-1981) VP of Bell Laboratories, and Harvard University professor.



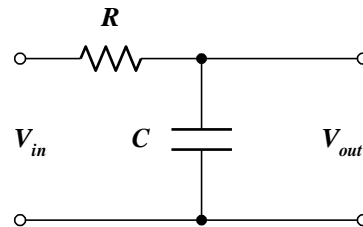
Frequency Response (introduction to filters)

Filters provide an electrical means of removing unwanted signals that are outside the frequency range of interest.

Eg. Consider the following RC low-pass filter circuit.

Let $s = j\omega = j2\pi f$

then
$$\frac{V_{in}}{R + \frac{1}{sC}} = \frac{V_{out}}{\frac{1}{sC}}$$



or the attenuation can be expressed as:

$$G(s) = \frac{V_{out}}{V_{in}} = \frac{1}{1 + sRC}.$$

Note that this equation is dependent on the frequency of the input signal, and has a complex component. Also this equation represents the **transfer function**, $G(s)$, of the filter.

We can also calculate the filter gain in dB,

$$\begin{aligned} G(\text{dB}) &= 20 \log |G(s)| \\ &= 20 \log \left| \frac{1}{1 + sCR} \right| \end{aligned} \quad \text{where the vertical lines represent magnitude.}$$

Which gives:

$$\begin{aligned} G(\text{dB}) &= 20 \log \frac{1}{\sqrt{1^2 + (\omega RC)^2}} \\ &= 20 \log \left(1 + (\omega CR)^2 \right)^{-1/2} \\ &= \left(-\frac{1}{2} \right) 20 \log \left(1 + (\omega CR)^2 \right) \end{aligned}$$

or
$$G(\text{dB}) = -10 \log \left(1 + (\omega CR)^2 \right)$$

If $\omega \ll \frac{1}{RC}$ then $\omega RC \approx 0$ and $G(\text{dB}) \approx -10 \log 1 = 0 \text{ dB}$



If $\omega \gg \frac{1}{RC}$ than $\omega RC \gg 1$ and $G(dB) \approx -10 \log(\omega RC)^2 = -20 \log(\omega RC)$

If $\omega = \frac{1}{RC}$ than $\omega RC = 1$ and $G(dB) \approx -10 \log(1+1^2) = 3 \text{ dB}$

Transfer Function

Recall that the **transfer function** (or gain) of a system is the ratio of the output of the system to the input of the system.

Mathematically: $\text{Transfer Function} = \frac{\text{output}}{\text{input}}$

In electronics and automation, it is often required to express the transfer function as a function of frequency. Usually, when we graph the transfer function we use dB on the vertical axis, and plot the horizontal axis (frequency) on a log scale. This type of plot is known as a Bode Plot.



Introduction

One of the basic building blocks in signal processing circuits is the filter. The purpose of filters is generally to achieve some frequency selectivity by processing input signals so that desired signal frequencies are passed through the filter (sometimes amplified as well) and undesired frequencies are attenuated.

The advent of active components has led to tremendous improvements in filter performance, as indicated in the “Pro’s and Con’s” list below:

<i>Pros of Active Filters</i>	<i>Cons of Active Filters</i>
<ol style="list-style-type: none">1. Elimination of inductors2. Low cost (largely due to item 1)3. Smaller size and weight (due to item 1)4. High isolation (high input impedance, low output impedance)5. Characteristics relatively independent of loading (due to item 4)6. User-defined gain	<ol style="list-style-type: none">1. Requirement for power2. Limited dynamic range (lower limit due to noise, upper limit due to clipping)3. Limited frequency range (lower limit due to large capacitors, upper limit due to active device performance)



What is a filter ?

A filter is a device that passes electric signals at certain frequencies or frequency ranges while attenuating (preventing the passage of) others.

Filter circuits are used in many applications. In the field of telecommunication, band-pass filters are used in the audio frequency range (0 kHz to 20 kHz) for modems and speech processing. High-frequency band-pass filters (several hundred MHz) are used for channel selection in telephone central offices. Data acquisition systems require anti-aliasing low-pass filters as well as low-pass noise filters in their signal reconstruction conditioning stages. System power supplies often use band-rejection filters to suppress the 60-Hz line frequency and high frequency transients.

In addition, there are filters that do not filter any frequencies of a input signal, but just add a linear phase shift to each frequency component, as a result contribute to a constant time delay. These are called all-pass filters.

At high frequencies (> 1 MHz), most of these filters usually consist of passive components such as inductors (L), resistors (R), and capacitors (C). They are then called passive RLC filters.

In the lower frequency range (0 Hz to 1 MHz), the inductor (L) value becomes very large and the inductor itself gets quite bulky, making economical production difficult.

In these cases, active filters become important. Active filters are circuits that use an operational amplifier (op amp) as the active device in combination with resistors (R) and capacitors (C) to provide an RLC-like filter performance at low frequencies (Figure 1).

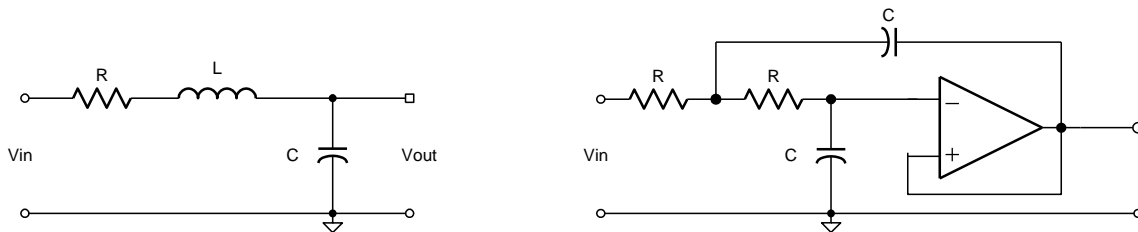


Figure 1 – Second-Order Passive Low-Pass Filter and Active Filter

Filters are classified by Order, such as first-order, second-order, etc. The order refers to the number of Reactive Components (RLC) the filter has.

In this section we will covers active filters. It introduces the three main filter optimizations (Butterworth, Tschebyscheff, and Bessel), followed by a sections describing the most common active filter applications: low-pass, high-pass, band-pass, band-rejection, and all-pass filters.



Frequency Response

Filters provide an electrical means of removing unwanted signals that are outside the frequency range of interest.

Eg. Consider the following RC low-pass filter circuit.

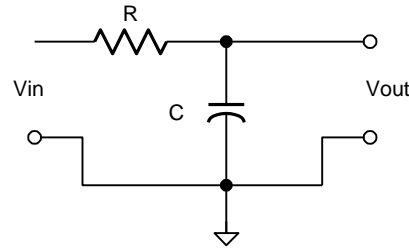
Let $s = j\omega$ and $\omega = 2\pi f$

Then $s = j2\pi f$

Then the current through R and C is

$$I = \frac{V_{in}}{R + X_c} = \frac{V_{out}}{X_c}$$

Then
$$\frac{V_{in}}{R + \frac{1}{sC}} = \frac{V_{out}}{\frac{1}{sC}}$$



or the attenuation can be expressed as:

$$G(s) = \frac{V_{out}}{V_{in}} = \frac{\frac{1}{sC}}{R + \frac{1}{sC}} = \frac{1}{1 + sRC}.$$

Note that this equation is dependent on the frequency f of the input signal, and has a complex component (s). Also this equation represents the **transfer function**, $G(s)$, of the filter.

We can also calculate the filter transfer function magnitude (gain) in dB,

$$G(dB) = 20 \log G(s) \quad G(s) \text{ is a voltage ration}$$

then $G(dB) = 20 \log |G(s)|$ where the vertical lines $| |$ represent magnitude.

And
$$G(dB) = 20 \log \left| \frac{1}{1 + sCR} \right|$$

Which gives:



$$\begin{aligned}
 G(dB) &= 20 \log \frac{1}{\sqrt{1^2 + (\omega RC)^2}} \\
 &= 20 \log \left(\frac{1}{\sqrt{1 + (\omega CR)^2}} \right)^{1/2} \\
 &= \left(-\frac{1}{2} \right) 20 \log \left(1 + (\omega CR)^2 \right)
 \end{aligned}$$

or
$$G(dB) = -10 \log \left(1 + (\omega CR)^2 \right)$$

If $\omega \ll \frac{1}{RC}$ than $\omega RC \approx 0$ and $G(dB) \approx -10 \log 1 = 0 \text{ dB}$

Given a flat response

If $\omega \gg \frac{1}{RC}$ than $\omega RC \gg 1$ and $G(dB) \approx -10 \log (\omega RC)^2 = -20 \log (\omega RC)$

This means that the slope is 20dB/dec

If $\omega = \frac{1}{RC}$ than $\omega RC = 1$ and $G(dB) \approx -10 \log (1^2) = -3 \text{ dB}$

This is referred to as the corner frequency

Transfer Function

Recall that the **transfer function** (or gain) of a system is the ratio of the output of the system to the input of the system.

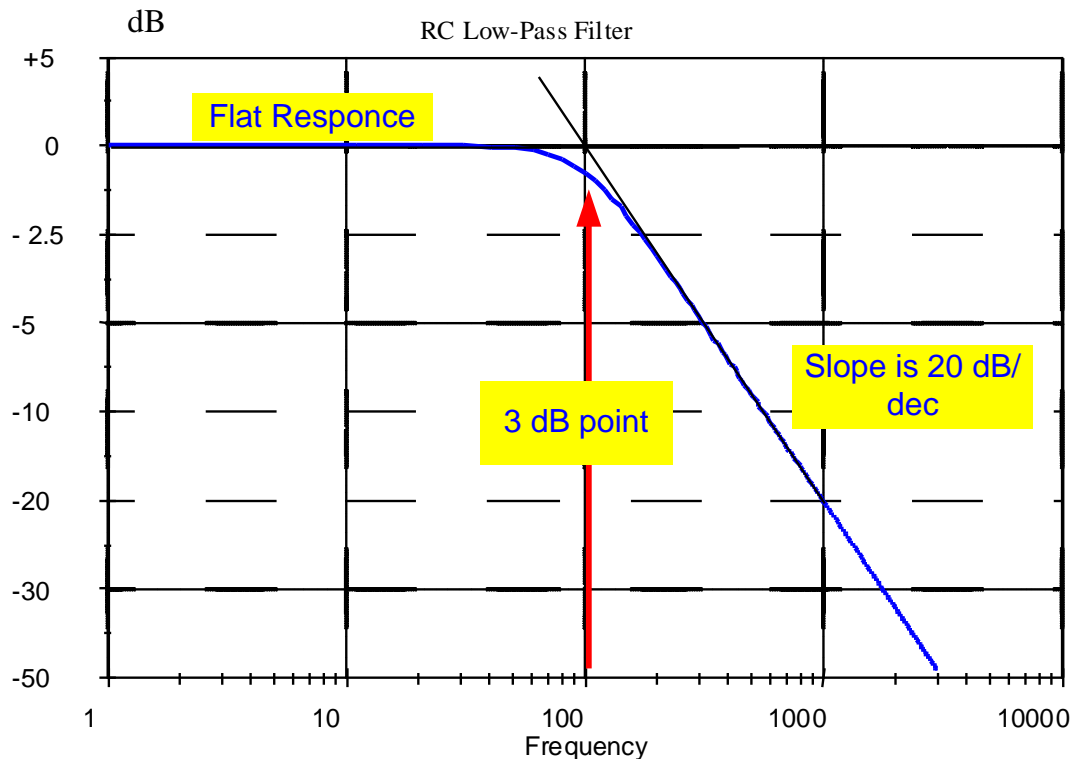
Mathematically:
$$\text{Transfer Function} = \frac{\text{output}}{\text{input}}$$

In electronics and automation, it is often required to express the transfer function as a function of frequency. Usually, when we graph the transfer function we use dB on the vertical axis, and plot the horizontal axis (frequency) on a log scale. This type of plot is known as a Bode Plot.



Bode¹ Plots

This method of representing the frequency response of filters, systems or signals simplifies interpretation, and saves a tremendous amount of time. Through the use of straight line segments, the frequency amplitude and phase relationship of a system can be found efficiently and accurately.



¹ Named after Hendrick Wade Bode (1905-1981) VP of Bell Laboratories, and Harvard University professor.



The most simple low-pass filter is the passive RC low-pass network shown in Figure 2.

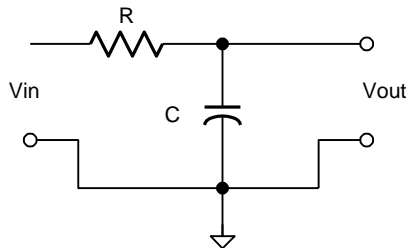


Figure 2 – First-order Passive Low-Pass Filter

The transfer function is

$$\frac{V_{out}}{V_{in}} = \frac{1}{sC} = \frac{1}{R + \frac{1}{sC}} = \frac{1}{1 + \frac{1}{sRC}}$$

where the complex frequency variable, $s = j\omega$, allows for any time variable signals.

For a normalized presentation of the transfer function, s is referred to the filter's corner frequency, or -3 dB point frequency, ω_c , and has these relationships:

$$s = \frac{s}{\omega_c} = \frac{j\omega}{\omega_c} = \frac{j2\pi f}{2\pi f_c} = j \frac{f}{f_c} = j\Omega$$

With the corner frequency of the low-pass in Figure 2 being $f_c = \frac{1}{2\pi RC}$,

s becomes $s = sRC$ and the transfer function $A(s)$ results in:

$$A(s) = \frac{1}{1 + s}$$

The magnitude of the gain response is:

$$|A| = \frac{1}{\sqrt{1 + \Omega^2}}$$

For frequencies $\Omega \gg 1$, the rolloff is 20 dB/decade. For a steeper rolloff, multiple filter stages can be connected in series as shown in Figure 3. To avoid loading effects, op amps, operating as voltage followers (impedance converters), separate the individual filter stages.

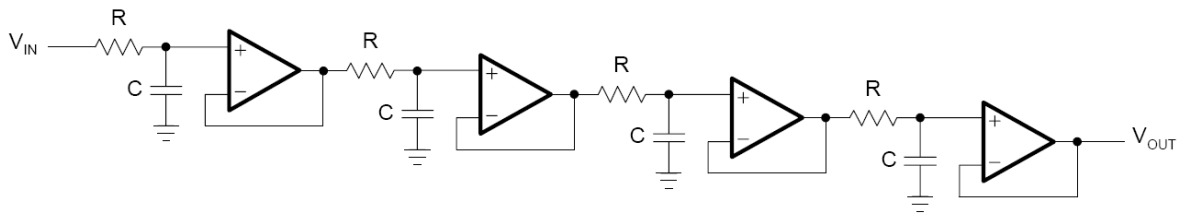


Figure 3 – Fourth-order Passive RC Low-Pass Filter.

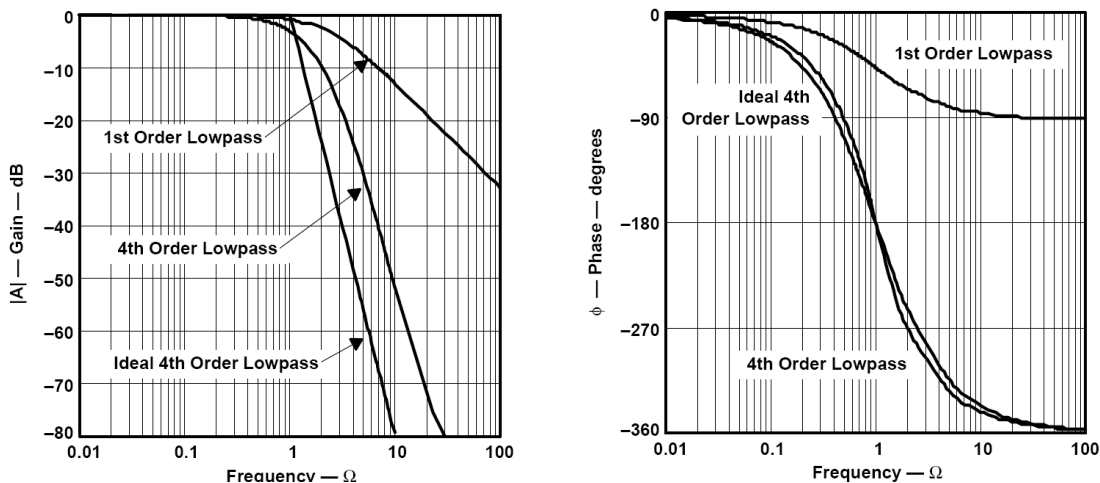


Figure 4 - Frequency and Phase Responses of a Fourth-Order Passive RC Low-Pass Filter

The following three types of predetermined filter responses are available:

- 1 - Butterworth – optimized pass-band for maximum flatness.
- 2 - Tschebyscheff – sharp transition from pass-band into the stop-band.
- 3 - Bessel – linear phase response up to f_c

The transfer function of a passive RC filter does not allow for this type of optimization, and these passive filters are mainly used for simple or high frequency applications. In the lower frequency range (< 1 MHz) active filters are used.



The following will introduce the stated most commonly used filter optimizations.

Butterworth Low-Pass Filters

The Butterworth low-pass filter provides maximum pass-band flatness. Therefore, a Butterworth low-pass filter is often used as an anti-aliasing filter in data acquisition system applications where precise signal levels are required across the entire pass-band.

Figure 5 plots the gain response of different orders of Butterworth low-pass filters versus the normalized frequency axis, Ω ($\Omega = f/f_c$); the higher the filter order, the longer the pass-band flatness.

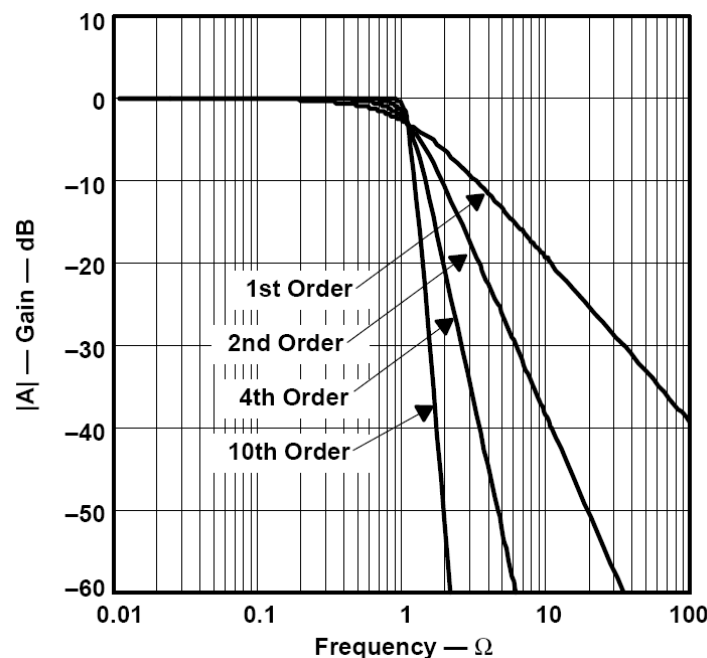


Figure 5 - Amplitude Responses of a Butterworth Low-Pass Filters

Tschebyscheff Low-Pass Filters

The Tschebyscheff low-pass filters provide an even higher gain roll-off above f_c . However, as Figure 6 shows, the pass-band gain contains ripples of constant magnitude. For a given filter order, the higher the pass-band ripples, the higher the filter's roll-off.

With increasing filter order, the influence of the ripple magnitude on the filter rolloff diminishes.

Each ripple accounts for one second-order filter stage. Filters with even order numbers generate ripples above the 0-dB line, while filters with odd order numbers create ripples



below 0 dB.

Tschebyscheff filters are often used in filter banks, where the frequency content of a signal is more important than constant amplitude.

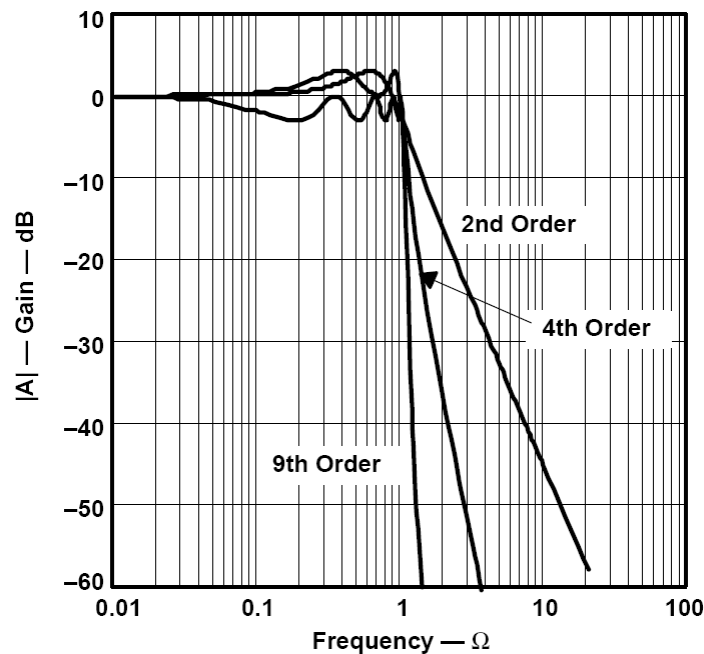


Figure 6 - Gain Responses of Tschebyscheff Low-Pass Filters

Bessel Low-Pass Filters

The Bessel low-pass filters have a linear phase response (Figure 7) over a wide frequency range, which results in a constant group delay (Figure 8) in that frequency range. Bessel low-pass filters, provide an optimum square-wave transmission behavior. However, the pass-band gain of a Bessel low-pass filter is not as flat as that of the Butterworth low-pass, and the transition from pass-band to stop-band is by far not as sharp as that of a Tschebyscheff low-pass filter (Figure 9).

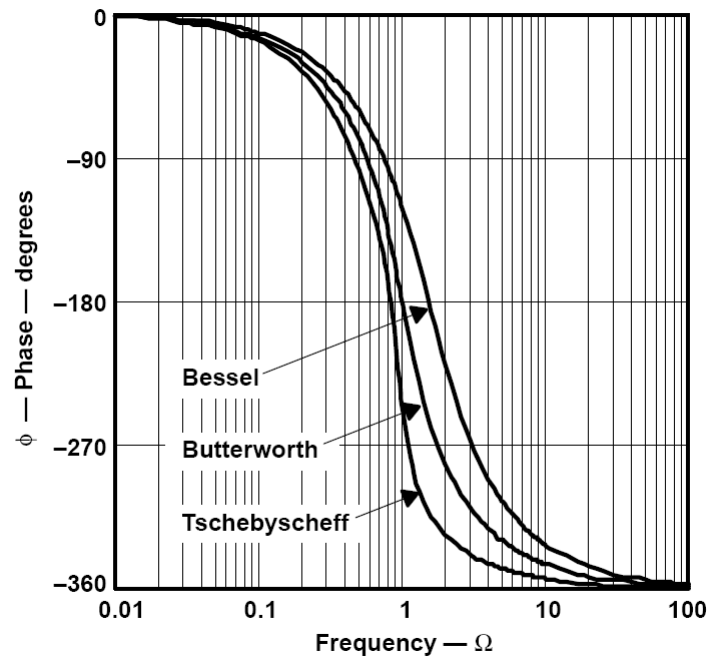


Figure 7 - Comparison of Phase Responses of a Fourth-Order Low-Pass Filters

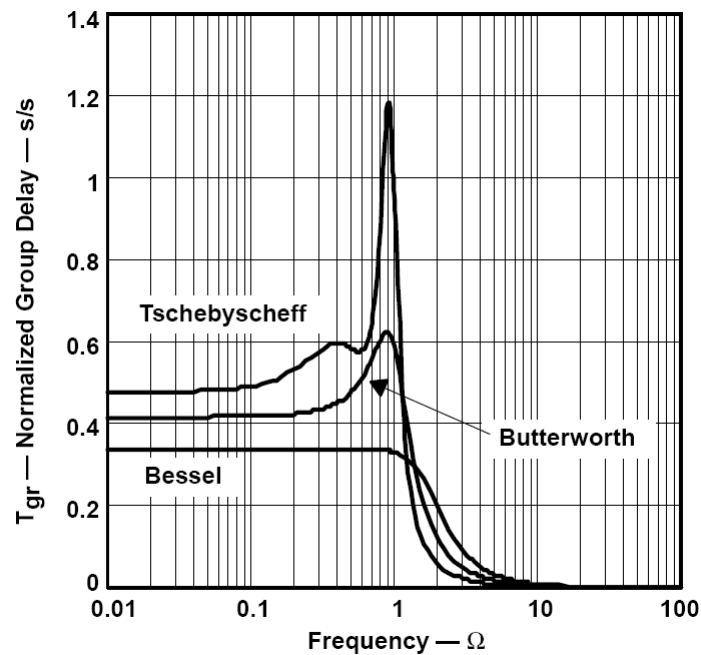


Figure 8 - Comparison of Normalized Group Delay (T_{gr}) of a Fourth-Order Low-Pass Filters

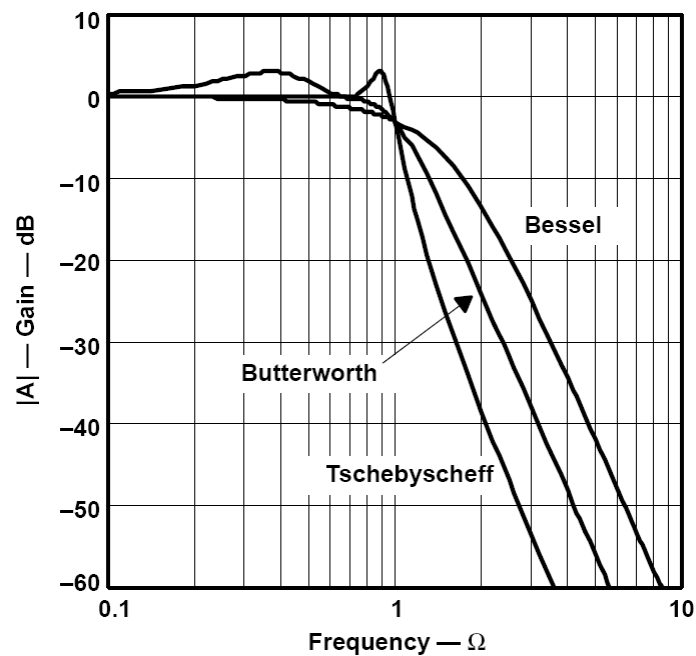


Figure 9 - Comparison of Gain Responses of Fourth-Order Low-Pass Filters

Filter implementation (design)

First-Order Low-Pass Filter

Figures 12 and 13 show a first-order low-pass filter in the inverting and in the non-inverting configuration.

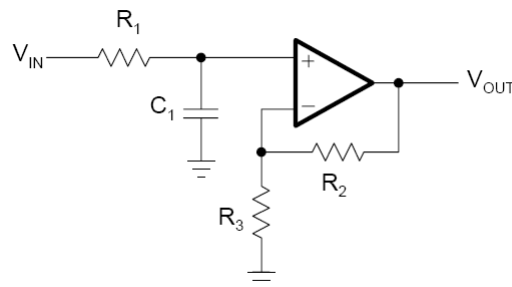


Figure 16-12. First-Order Non-inverting Low-Pass Filter

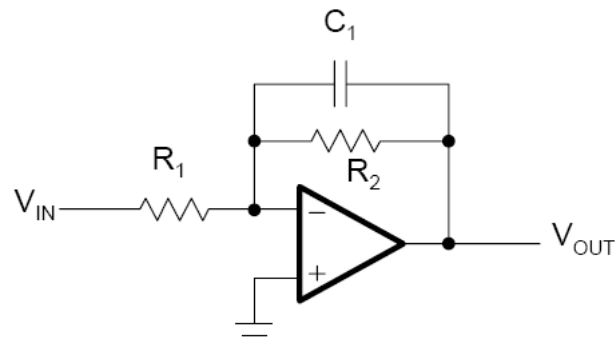


Figure 13 - First-Order Inverting Low-Pass Filter

The transfer functions of the circuits are:

$$A(s) = \frac{1 + \frac{R_2}{R_3}}{1 + \omega_c R_1 C_1 s} \quad \text{and} \quad A(s) = \frac{-\frac{R_2}{R_1}}{1 + \omega_c R_2 C_1 s}$$

The negative sign indicates that the inverting amplifier generates a 180° phase shift from the filter input to the output.

The filters cut-off frequency f_c (3db point) are:

$$f_c = \frac{1}{2\pi R_1 C_1} \quad \text{non-inverting filter} \quad \text{and}$$

$$f_c = \frac{1}{2\pi R_2 C_1} \quad \text{inverting filter}$$

The filters gain (A) are:

$$A = 1 + \frac{R_2}{R_1} \quad \text{non-inverting filter} \quad \text{and}$$

$$A = -\frac{R_2}{R_1} \quad \text{inverting filter}$$



Second-Order Low-Pass Filter

There are two topologies (circuit configurations) used for a second-order low-pass filter, the Sallen-Key and the Multiple Feedback (MFB) topology.

Sallen-Key Topology

The general Sallen-Key topology (Figure 15) allows for separate gain setting via $A_o = 1 + \frac{R_f}{R_i}$.

However, the unity-gain topology of Figure 16 is usually applied in filter designs with high gain accuracy, and unity gain.

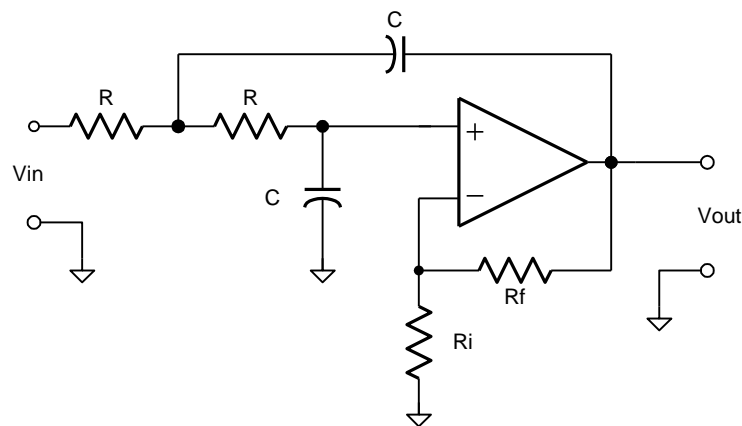


Figure 15 - General Sallen-Key Low-Pass Filter

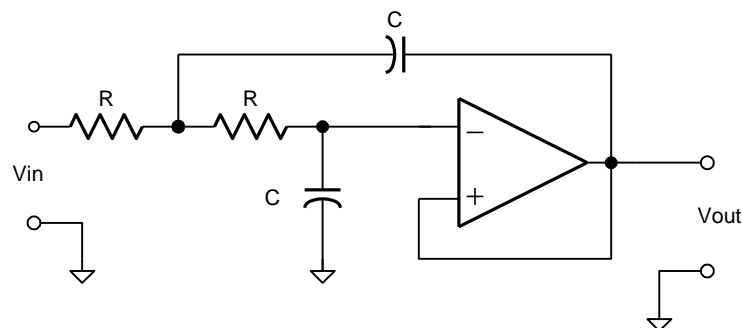


Figure 16 - Unity-Gain Sallen-Key Low-Pass Filter

Different filter cut-off frequencies can be obtained by changing the values of R and C, and different filter characteristics can be realised by changing the negative feedback gain of the filter (R_f and R_i).

Since the $\text{Gain(DC)} = 1 + \frac{R_f}{R_i}$



If we set the Gain(DC) = K

then we can consider K to be the gain (DC) of the filter.

Different values of K will give different filter responses or types as shown in the following table.

$K = 1$	$Rf = 0$ (short), and $Ri = \infty$ (open)
$K = 1.268$	Bessel
$K = 1.586$	Butterworth
$K = 2.234$	Chebyshev

The damping factor ζ , which normally determines the filter response characteristics, is related to K as:

$$2\zeta = 3 - K$$

This means that different filter types using this topology will have different DC gains. The 3dB point is then measured as *3dB below the filters DC gain*.

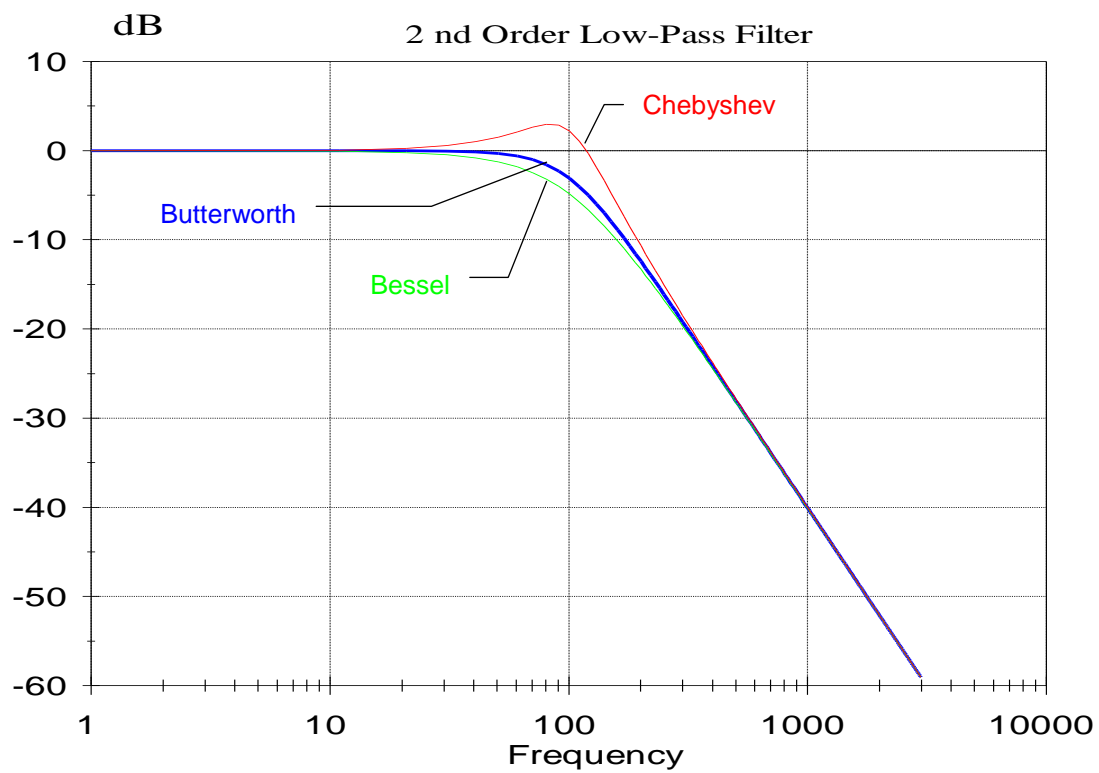
The cut-off frequency is given by the following relation:

$$f_o = \frac{1}{2\pi RC}$$

and the filters roll-off slope is 40 dB/dec.

The following Bode Plot shows a typical second order low-Pass filter with a cutoff frequency of 100Hz, and a dc gain of 1.

The following graph shows the three different responses, with a normalized gain of 1.





Introduction to interfacing Analog Signals to Digital Systems

Analog and Digital Signal Processing is a technology that is widely used in many applications, such as automotive, consumer, graphics and imaging, industrial, instrumentation, medical, military, telecommunication, and voice/speech applications. Today, most music that we listen to, phone conversations with our friends, and the programs we watch on TV are all brought to us with the magic of analog and digital signal processing.

Signal processing is derived from and incorporates basic electrical engineering technology, analog and digital circuit, mathematics, software programs, and microcontrollers to manipulate signals.

Converting Analog to Digital

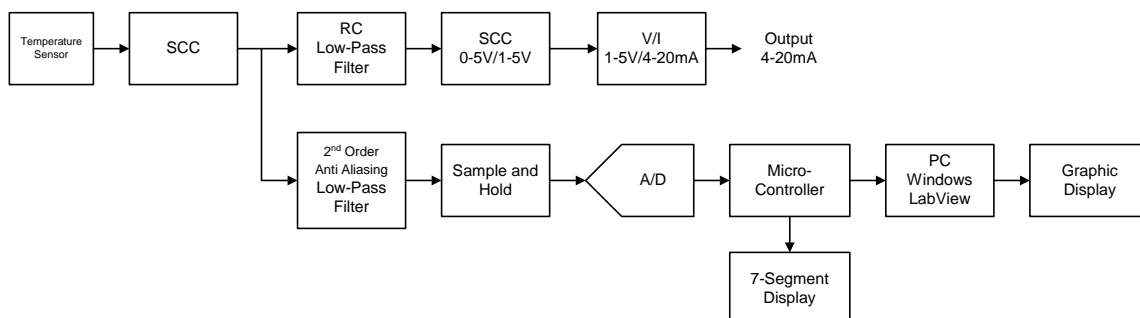
In Order to process signals using digital techniques, the incoming analog signal must be converted into a digital form.

After completing this section, you will be able to:

- Explain the basic process of converting an analog signal to digital
- Describe the purpose of the sample-and-hold function
- Define the Nyquist frequency
- Define the reason for *aliasing* and discuss how it is eliminated
- Describe the purpose of an ADC

Sampling

An anti-aliasing filter and a sample-and-hold circuit are two functions typically found in a digital signal processing system. The sample-and-hold function does two operations





Sampling is the process of taking a sufficient number of *discrete* values at points on a waveform that will define the shape of waveform. The more samples you take, the more accurately you can define a waveform. Sampling converts an analog signal into a series of *impulses*, each representing the amplitude of the signal at a given instant in time. Figure 1 illustrates the process of sampling.

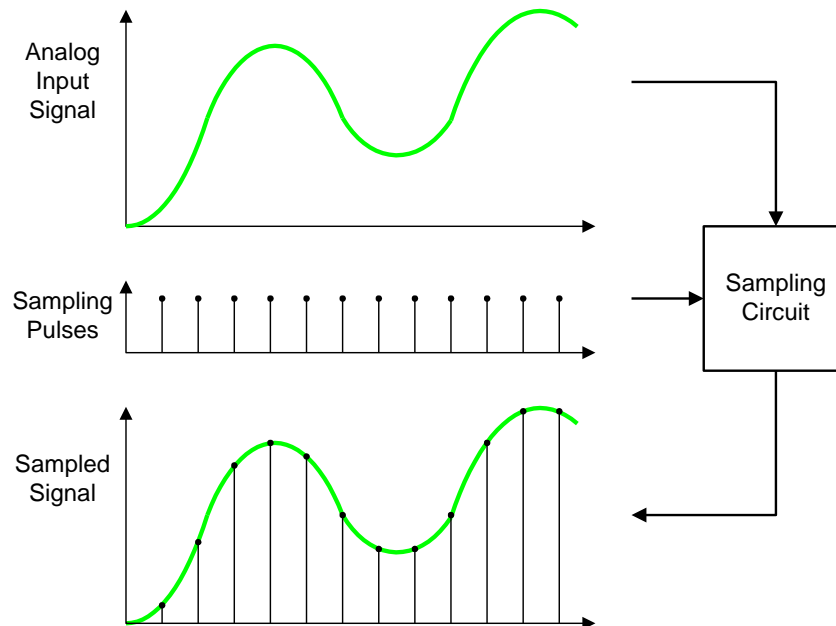


Figure 1 - Sampling process

When an analog signal is to be sampled, there are certain criteria that must be met in order to accurately represent the original signal. All analog signals (except a pure sine wave) contain a spectrum of component frequencies. For a pure sine wave, these frequencies appear in multiples called harmonics. The harmonics of an analog signal are sine waves of different frequencies and amplitudes. When the harmonics of a given periodic waveform are added, the result is the original signal. Before a signal can be sampled, it must be passed through a low-pass filter (anti-aliasing filter) to eliminate harmonic frequencies above a certain value as determined by the Nyquist frequency.

The Sampling Theorem

Notice that in Figure 1 there are two input waveforms to the Sampling circuit. One is the analog signal and the second is the sampling pulses. The sampling theorem states that, in order to represent an analog signal, the sampling frequency, f_{sample} , must be at least twice the highest frequency component of the analog signal. Another way to say this is that the highest analog frequency can be no greater than one-half the sampling frequency.



The frequency $f_a(\max)$ is known as the *Nyquist* frequency. This can be expressed mathematically as:

$$F_{\text{sample}} \geq 2f_a(\max)$$

In practice, the sampling frequency should be more than twice the highest analog frequency.

To intuitively understand the sampling theorem, a simple "bouncing-ball" analogy may be helpful, and illustrate the basic idea. If a ball is photographed (sampled) at one instant of time during a single bounce, as illustrated in Figure 2(a), you cannot tell anything about the path of the ball except that it is off the floor. You cannot tell whether it is going up or down or the distance of its bounce. If you take photos at two equally-spaced instants of time during one bounce, as shown in part (b), you can obtain only a minimum amount of information about its movement and nothing about the distance of the bounce. In this particular case, you know only that the ball has been in the air at the time between the two photos were taken and that the maximum height of the bounce is at least equal to the height shown in each photo. If you take four photos, as shown in part (c), then the path that the ball followed during a bounce begins to emerge. The more photos (samples) that you take, the more accurately you can determine the path of the ball as it bounces.

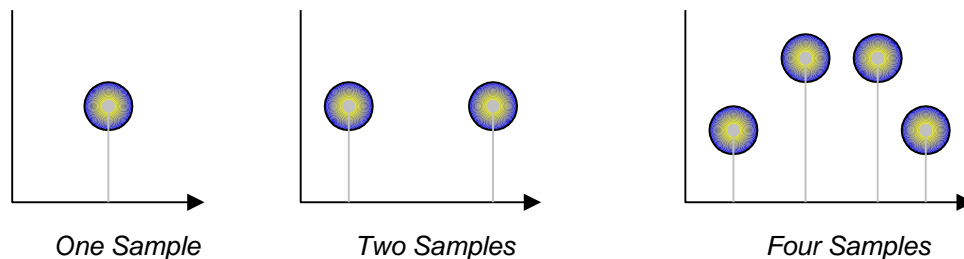


Figure 2 - Bouncing ball analogy of the sampling theory.

Filtering

Low-pass filtering is necessary to remove all frequency components (harmonics) of the analog signal that exceed the Nyquist frequency. If there are any frequency components in the analog signal that exceed the Nyquist frequency, an unwanted condition known as *aliasing* will occur. An alias is a signal produced when the sampling frequency is not at least twice the signal frequency. An alias signal has a frequency that is less than the highest frequency in the analog signal being sampled and therefore falls within the spectrum or frequency band of the input analog signal causing distortion. Such a signal is actually "posing" as part of the analog signal when it really is not, thus the term *alias*.

Another way to view aliasing is by considering that the sampling pulses produce a spectrum of harmonic frequencies above and below the sample frequency, as shown in Figure 3. If the analog signal contains frequencies above the Nyquist frequency, these



frequencies overlap into the spectrum of the sample waveform as shown and interference occurs. The lower frequency components of the sampling waveform become mixed in with the frequency spectra of the analog waveform, resulting in an aliasing error.

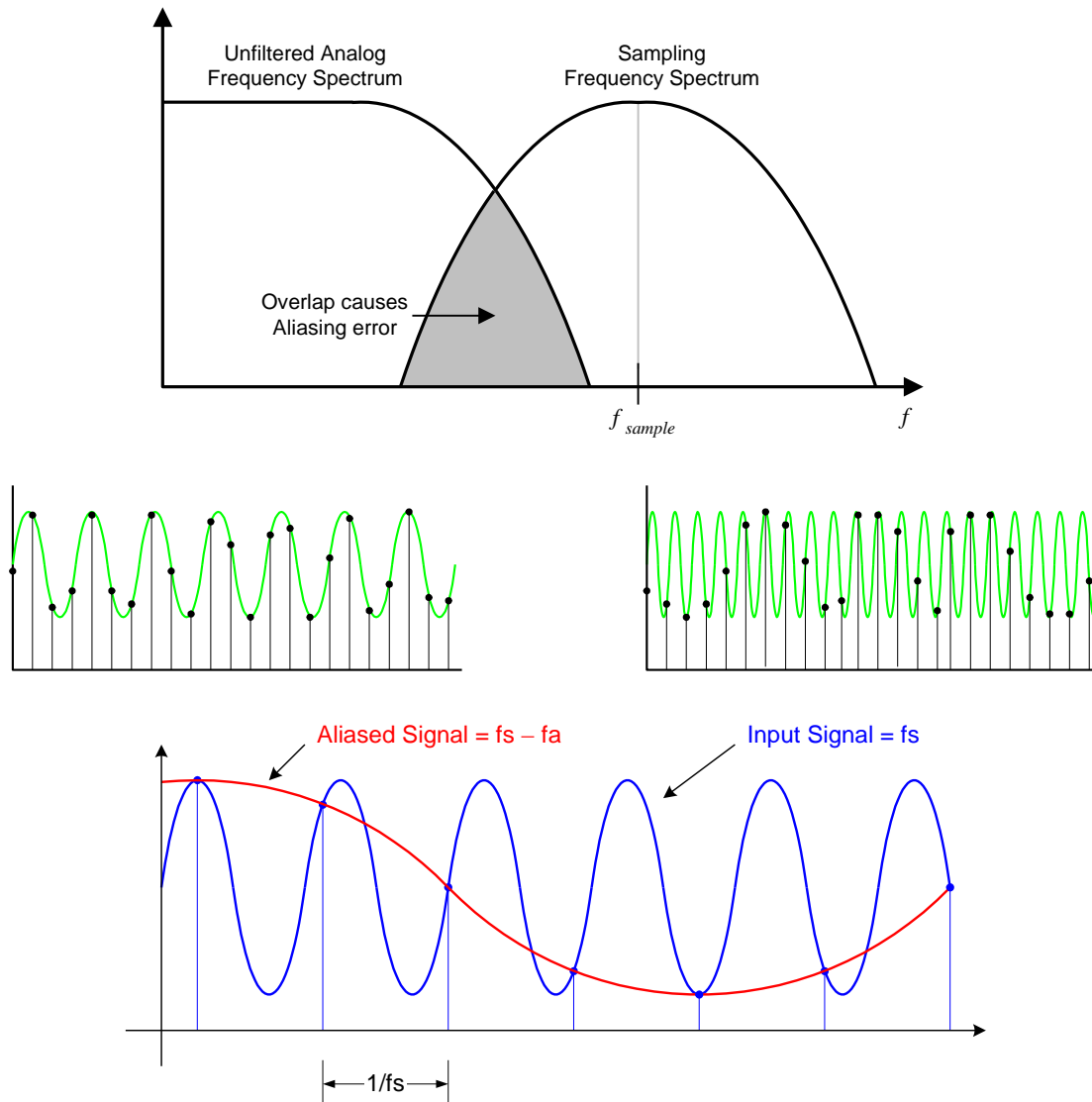


Figure 3 - A basic illustration of the condition: $F_{\text{sample}} \geq 2f_a(\text{max})$

A low-pass anti-aliasing filter is normally used to limit the frequency spectrum of the analog signal for a given sample frequency. To avoid an aliasing error, the filter must at least eliminate all analog frequencies above the minimum frequency in the sampling spectrum, as illustrated in Figure 4. Aliasing can also be avoided by sufficiently increasing the sampling frequency. However, the maximum sampling frequency is usually



limited by the performance of the analog-to-digital converter, the real time computations required, and the microcontroller in use.

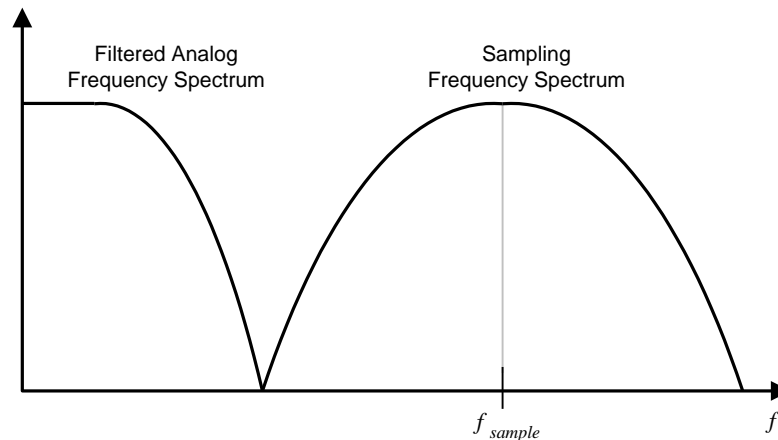


Figure 4 - After low-pass filtering, the frequency spectra of the analog and the sampling signals do not overlap, eliminating aliasing error.

Typical Application

An example of the application of sampling is in digital audio equipment. The sampling rates used are 32 kHz, 44.1 kHz, or 48 kHz (the number of samples per second). The 48 kHz rate is the most common, but the 44.1 kHz rate is used for audio CDs and pre-recorded tapes. According to the Nyquist rate, the sampling frequency must be at least twice the audio signal. Therefore, the CD sampling rate of 44.1 kHz captures frequencies up to about 22 kHz, which exceeds the 20 kHz specification that is common for most audio equipment

Many applications do not require such a wide frequency range to obtain reproduced sound that is acceptable. For example, human speech contains some frequencies near 10kHz and, therefore, requires a sampling rate of at least 20 kHz. However, if only frequencies up to 4 kHz (ideally requiring an 8 kHz minimum sampling rate) are reproduced, voice is very understandable. On the other hand, if a sound signal is not sampled at a high enough rate, the effect of aliasing will become noticeable with background noise and distortion.

Holding the Sampled Value

The holding operation is the second part of the sample-and-hold function. After filtering and sampling, the sampled level must be held constant until the next sample occurs. This is necessary for the ADC to have sufficient time to convert the analog signal to a digital representation ie, sufficient time to process the sampled value. This sample-and-hold



operation results in a "stairstep" waveform that approximates the analog input waveform, as shown in Figure 5.

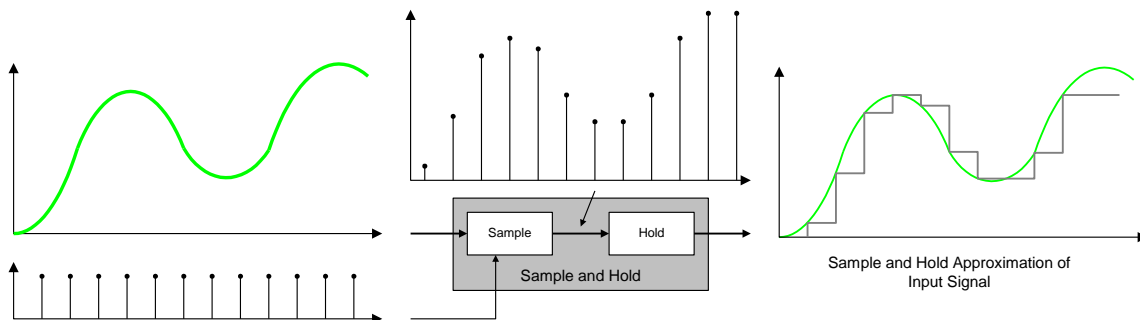
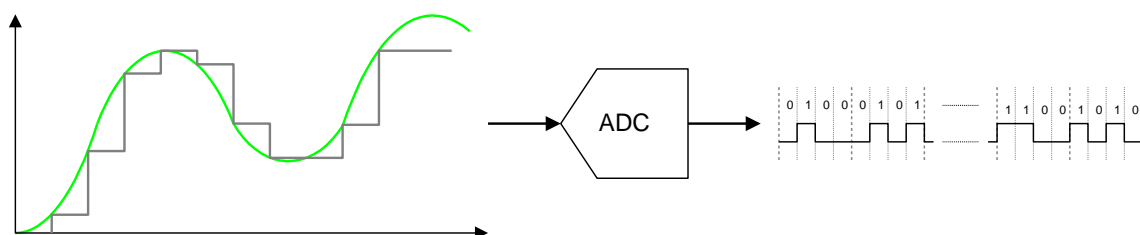


Figure 5 - Illustration of a sample-and-hold operation.

Analog-to-Digital Conversion

Analog-to-digital conversion is the process of converting the output of the sample-and-hold circuit to a series of binary numbers (codes) that represent the amplitude of the analog input at each of the sample times. The sample-and-hold process keeps the amplitude of the analog input signal constant between sample pulses; therefore, the analog-to-digital conversion can be done using a constant value rather than having the analog signal change during a conversion interval, which is the time between sample pulses. Figure 6 illustrates the basic function of an analog-to-digital (ADC) converter. The sample intervals are indicated by dashed lines.



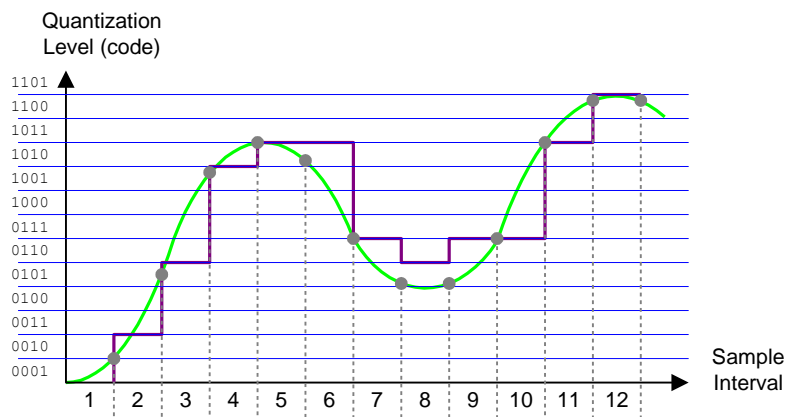
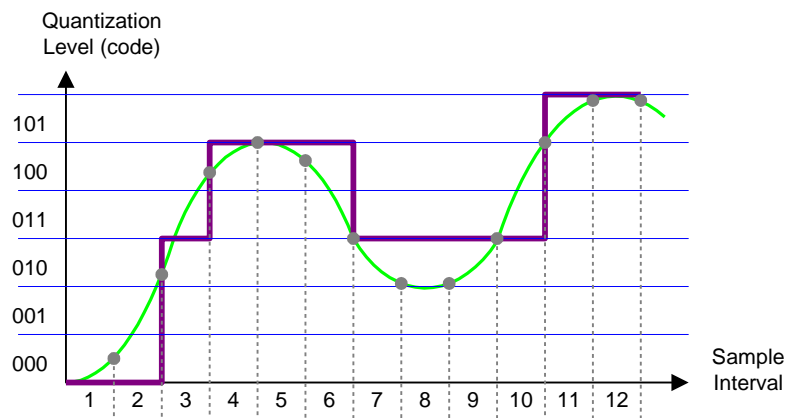
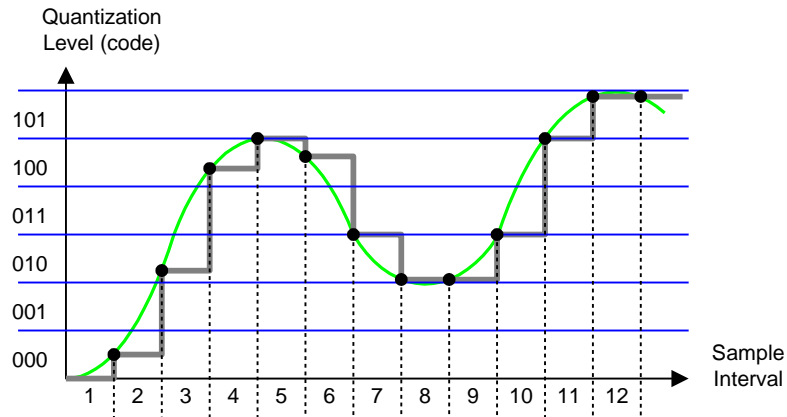
*Basic function of an analog-to-digital (ADC) converter
The ADC output waveform that represents the binary codes is also shown.*

Quantization

The process of converting an analog value to a code is called quantization. During the quantization process, the ADC converts each sampled value of the analog signal to a binary code. The more bits that are used to represent a sampled value, the more accurate is the representation.



To illustrate, we will quantize a reproduction of the analog waveform into four levels (0-3). Two bits are required for four levels. As shown in Figure 7, each quantization level is represented by a 2-bit code on the vertical axis, and each sample interval is numbered along the horizontal axis. The sampled data is held for the entire sample period. This data is quantized to the next lower level, as shown in Table 1 (for example, compare samples 3 and 4, which are assigned different levels).

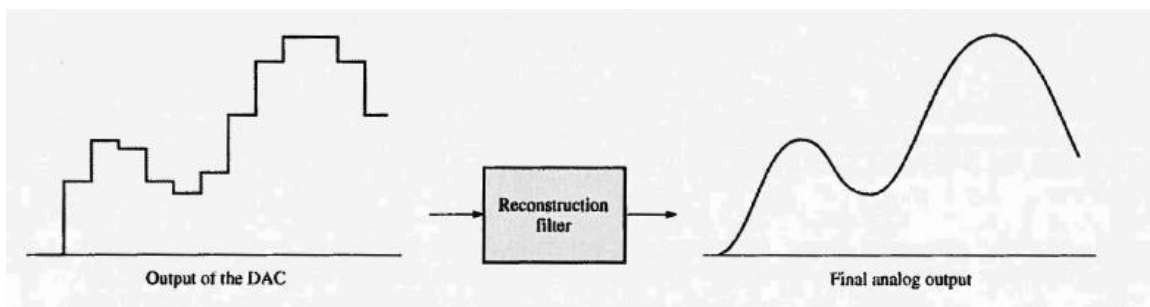




Digital to Analog

Reconstruction Filter

The output of the DAC is a "stairstep" approximation of the original analog signal after it has been processed by the DSP. The purpose of the low-pass reconstruction filter (sometimes called a postfilter) is to smooth out the DAC output by eliminating the higher frequency content that results from the fast transitions of the "stairsteps," as roughly illustrated in Figure 34.



Digital Signal Processing

Essentially, a digital signal processor (DSP) is a special type of microprocessor that processes data in real time. Its applications focus on the processing of digital data that represents analog signals. A DSP, like a microprocessor, has a central processing unit (CPU) and memory units in addition to many interfacing functions. Every time you use your cellular telephone, you are using a DSP, and this is only one example of its many applications.

The digital signal processor (DSP) is the heart of a digital signal processing system. It takes its input from an ADC and produces an output that goes to a DAC, as shown in Figure 37. As you have learned, the ADC changes an analog waveform into data in the form of a series of binary codes that are then applied to the DSP for processing. After being processed by the DSP, the data go to a DAC for conversion back to analog form.

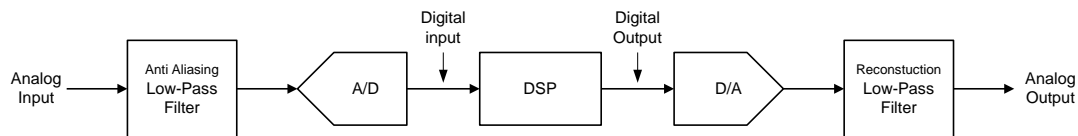


FIGURE The DSP has a digital input and produces a digital output.

DSP Programming

DSPs are typically programmed in C, or in some application specific program. Because programs written in assembly language can usually execute faster and because speed in some applications is critical these DSP applications.

DSP programs are usually much shorter than traditional micro-controller programs because of their very specialized applications where much redundancy is used. In general, the instruction sets for DSPs tend to be smaller than for microprocessors.

DSP Applications

The DSP, unlike the general-purpose microprocessor, must typically process data in *real time*; This means that the outputs it must occur in deterministic amount of time. Many applications in which DSPs are used cannot tolerate any noticeable delays, requiring the DSP to be extremely fast. In addition to cell phones, digital signal processors (DSPs) are used in multimedia computers, video recorders, CD players, hard disk drives, digital radio modems, and many other applications to improve the signal quality. Also, DSPs are becoming more common in television applications. For example, television converters use DSP to provide compatibility with various television standards.

An important application of DSPs is in signal compression and decompression. In CD systems, for example, the music on the CD is in a compressed form so that it does not use as much storage space. It must be decompressed in order to be reproduced. Also signal compression is used in cell phones to allow a greater number of calls to be handled simultaneously by the network provider in a local cell. Some other areas where it has had a major impact are as follows.

Telecommunications The field of telecommunications involves transferring all types of information from one location to another, including telephone conversations, television signals, and digital data. Among other functions, the DSP facilitates multiplexing many signals onto one transmission channel because information in digital form is relatively easy to multiplex and demultiplex.

At the transmitting end of a telecommunications system, DSPs are used to compress digitized voice signals for conservation of bandwidth. Compression is the process of



reducing the data rate. Generally, a voice signal is converted to digital form at 8000 samples per second (sps), based on a Nyquist frequency of 4 kHz. If 8 bits are used to encode each sample, the data rate is 64 kbps. In general, reducing (compressing) the data rate from 64 kbps to 32 kbps results in no loss of sound quality. When the data are compressed to 8 kbps, the sound quality is reduced noticeably. When compressed to the minimum of 2 kbps, the sound is greatly distorted but still usable for some applications where only word recognition and not quality is important. At the receiving end of a telecommunications system, the DSP decompresses the data to restore the signal to its original form.

Echoes, a problem in many long distance telephone connections, occur when a portion of a voice signal is returned with a delay. For shorter distances, this delay is barely noticeable; but as the distance between the transmitter and the receiver increases, so does the delay time of the echo. DSPs are used to effectively cancel the annoying echo, which results in a clear, undisturbed voice signal.

Sound cards used in computers use an ADC to convert sound from a microphone, audio CD player, or other source into a digital signal. The ADC sends the digital signal to a digital signal processor (DSP). Based on instructions from a ROM, one function of the DSP is to compress the digital signal so it uses less storage space. The DSP then sends the compressed data to the computer's processor which, in turn, sends the data to a hard drive or CD ROM for storage. To play a recorded sound, the stored data is retrieved by the processor and sent to the DSP where it is decompressed and sent to a DAC. The output of the DAC, which is a reproduction of the original sound signal, is applied to the speaker.

Music Processing The DSP is used in the music industry to provide filtering, signal addition and subtraction, and signal editing in music preparation and recording. Also, another application of the DSP is to add artificial echo and reverberation, which are usually minimized by the acoustics of a sound studio, in order to simulate ideal listening environments from concert halls to small rooms.

Speech Generation and Recognition DSPs are used in speech generation and recognition to enhance the quality of man/machine communication. The most common method used to produce computer-generated speech is digital recording. In digital recording, the human voice is digitized and stored, usually in a compressed form. During playback the stored voice data are uncompressed and converted back into the original analog form. Approximately an hour of speech can be stored using about 3 MB of memory.

Speech recognition is much more difficult to accomplish than speech generation. The DSP is used to isolate and analyze each word in the incoming voice signal. Certain parameters are identified in each word and compared with previous examples of the spoken word to create the closest match. Most systems are limited to a few hundred words at best. Also, significant pauses between words are usually required and the system must be "trained" for a given individual's voice. Speech recognition is an area of



tremendous research effort and will eventually be applied in many commercial applications.

Radar In *radio* detection and ranging (radar) applications, DSPs provide more accurate determination of distance using data compression techniques, decrease noise using filtering techniques, thereby increasing the range, and optimize the ability of the radar system to identify specific types of targets. DSPs are also used in similar ways in sonar systems.

Image Processing The DSP is used in image-processing applications such as the computed tomography (CT) and magnetic resonance imaging (MRI), which are widely used in the medical field for looking inside the human body. In CT, X-rays are passed through a section of the body from many directions. The resulting signals are converted to digital form and stored. This stored information is used to produce calculated images that appear to be slices through the human body that show great detail and permit better diagnosis.

Instead of X-rays, MRI uses magnetic fields in conjunction with radio waves to probe inside the human body. MRI produces images, just as cr, and provides excellent discrimination between different types of tissue as well as information such as blood flow through arteries. MRI depends entirely on digital signal processing methods. In applications such as video telephones, digital television, and other media that provide moving pictures, the DSP uses image compression to reduce the number of bits needed, making these systems commercially feasible.

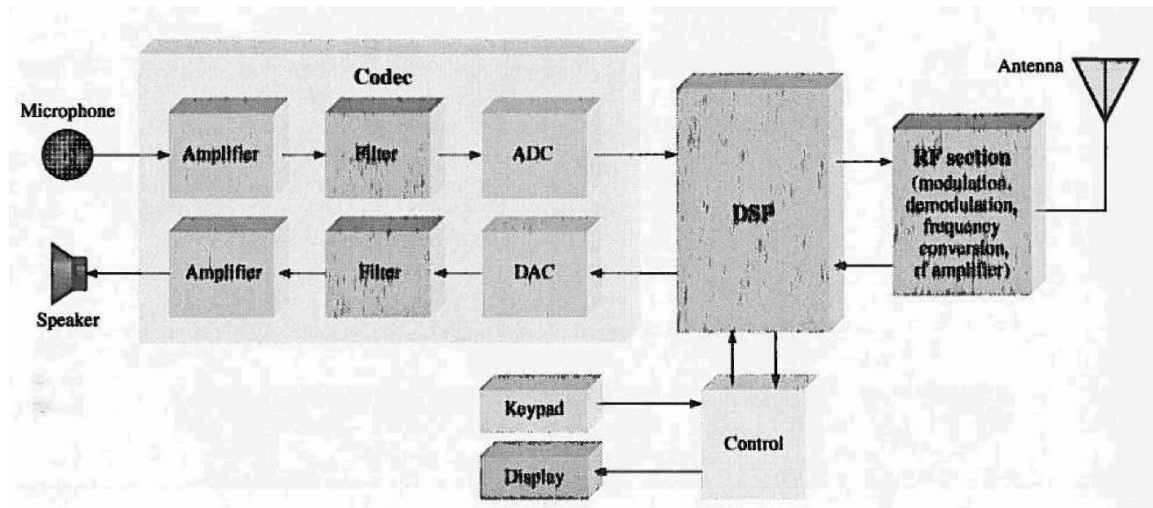
Filtering DSPs are commonly used to implement digital filters for the purposes of separating signals that have been combined with other signals or with interference and noise and for restoring signals that are distorted. Although analog filters are quite adequate for some applications, the digital filter is generally much superior in terms of the performance that can be achieved. One drawback to digital filters is that the execute time required produces a delay from the time the analog signal is applied until the time the output appears. Analog filters present no delay problems because as soon as the input occurs, the response appears on the output. Analog filters are also less expensive than digital filters. Regardless of this, the overall performance of the digital filter is far superior in many applications.

The DSP in a Cellular Telephone

The digital cellular telephone is an example of how a DSP can be used. Figure xx shows a simplified block diagram of a digital cell phone. The voice codec (codec is the abbreviation for coder/decoder) contains, among other functions, the ADC and DAC necessary to convert between the analog voice signal and a digital voice format. Sigma-delta conversion is typically used in most cell phone applications. For transmission, the voice signal from the microphone is converted to digital form by the ADC in the codec and then it goes to the DSP for processing. From the DSP, the digital signal goes to the rf (radio frequency) section where it is modulated and changed to the radio frequency for



transmission. An incoming rf signal containing voice data is picked up by the antenna, demodulated, and changed to a digital signal. It is then applied to the DSP for processing, after which the digital signal goes to the codec for conversion back to the original voice signal by the DAC. It is then amplified and applied to the speaker.



Simplified block diagram of a digital cellular phone.

Functions Performed by the DSP

In a cellular phone application, the DSP performs many functions to improve and facilitate the reception and transmission of a voice signal. Some of these DSP functions are as follows:

- *Speech compression.* The rate of the digital voice signal is reduced significantly for transmission in order to meet the bandwidth requirements.
- *Speech decompression.* The rate of the received digital voice signal is returned to its original rate in order to properly reproduce the analog voice signal.
- *Protocol handling.* The cell phone communicates with the nearest base in order to establish the location of the cell phone, allocates time and frequency slots, and arranges handover to another base station as the phone moves into another cell.
- *Error detection and correction.* During transmission, error detection and correction codes are generated and, during reception, detect and correct errors induced in the rf channel by noise or interference.
- *Encryption.* Converts the digital voice signal to a form for secure transmission and converts it back to original form during reception.



Successive Approximation Analog to Digital (A/D) Conversion

The Successive Approximation Register (SAR) Principle

The block diagram for a basic 4-bit SAR A/D is shown in Figure 1. Initially, the register is set to 0 (0000₂). The control logic then modifies the contents of the SAR, bit by bit. Each new binary number generated is fed to the DAC, where the analog voltage is produced (V_{out}) and sent to the comparator. When the conversion process is complete, the digital output reflects the input analog voltage at V_{in}.

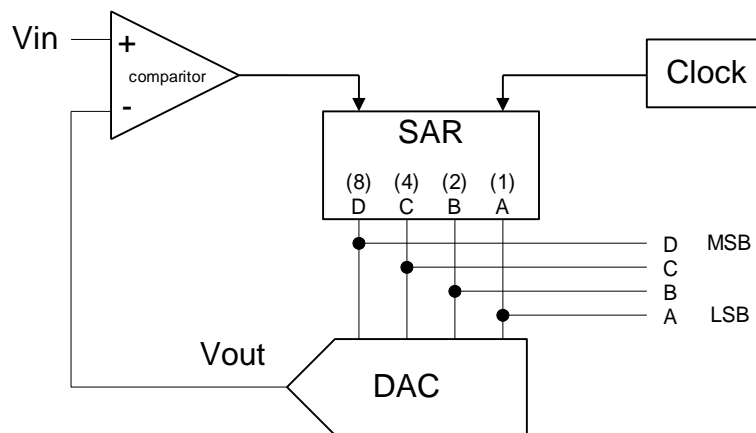


Figure 1. Block Diagram

To examine the concept a bit further. First, a clock pulse is sent to the control logic. It in turn, outputs a signal that sets the SAR's most significant bit (MSB) to 1. The register then contains the value 1000₂. The DAC then generates an analog output voltage, V_{out}, equivalent to the number 1000₂, and sends it to the inverting input of the comparator. If this voltage is greater than V_{in}, the comparator outputs a LOW, causing the MSB to reset to 0. If the voltage is less than V_{in}, the comparator outputs a HIGH, and the 1 is retained in the MSB location. This process now repeats with the next most significant bit, until all bits have been checked. After all the bits from the SAR have been placed, the conversion cycle is complete. The digital output then reflects the analog voltage input, within the resolution of the converter.

The following example will illustrate the circuit's operation, by analyzing how a 4-bit A/D converter, converts a 1.5V analog input to the equivalent binary number 0101₂.

Assume that the A/D converter has a range from 0 to 5. This means it has a resolution of

$$\frac{5}{2^4 - 1} = \frac{5}{15} = .333 \text{ V/bit. This means } V_{\text{out}} = 2.67 \text{ V for the D(8) bit (MSB),}$$

V_{out} = 1.33V for the C(4) bit, V_{out} = .667V for the B(2) bit, and V_{out} = .333V for the A(1) bit (LSB).



In Figure 2 we see the first step of the conversion cycle. With the first clock pulse, the MSB, D(4), is set to 1. The output of the DAC is now 2.67V. Since this voltage is greater than the V_{in} of 1.5V, the comparator's output goes LOW, causing the MSB in the SAR to reset to 0.

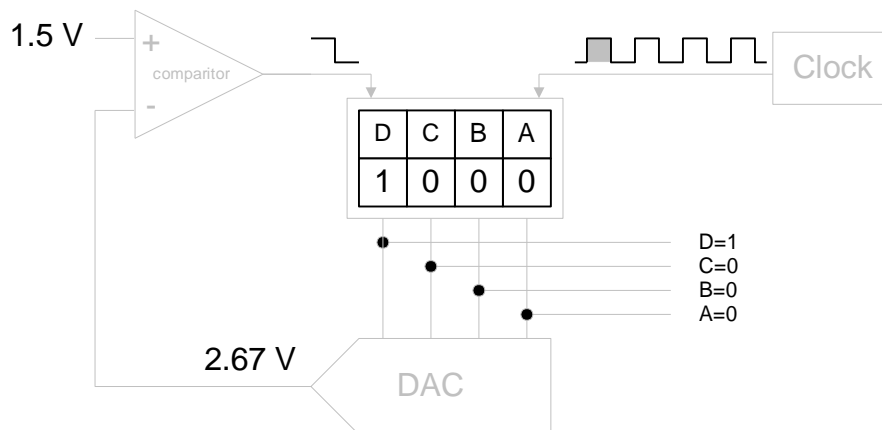


Figure 2.

Figure 3. illustrates the second step in the conversion cycle. With the second clock pulse, the C (4) bit is set to 1. The DAC output is now 1.33V. This voltage is less than the V_{in} of 1.5V. The comparator's output goes HIGH, and the 1 bit is retained in the SAR.

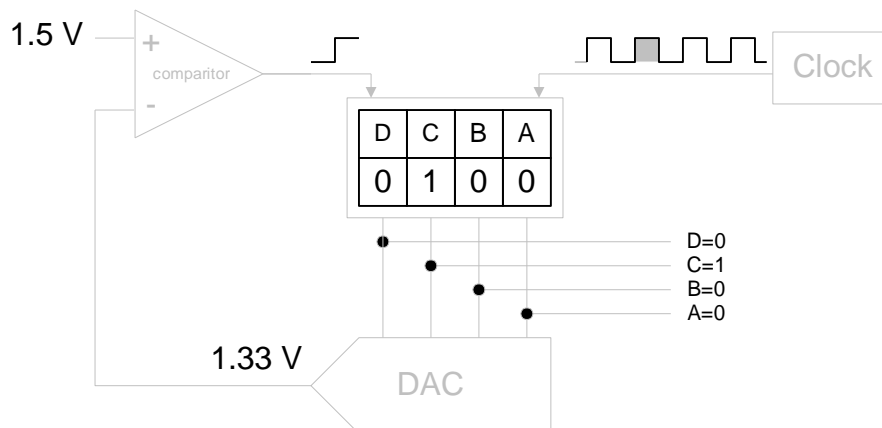


Figure 3.

Figure 4. shows the third step in the conversion cycle. With the third clock pulse, the B (2) bit is set to 1. Note, though, that the DAC now outputs 2.00V, not .667V as you might first expect. This is because the C and B bits are both set to 1, and giving a V_{out} of



$1.33 + .67 = 2.00\text{V}$. Since this voltage is greater than 1.5 V , the comparator's output goes LOW, and the B bit is reset to 0.

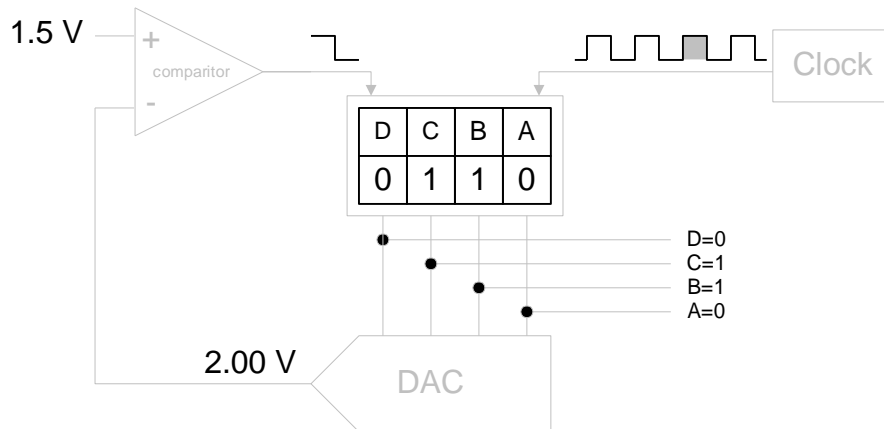


Figure 4.

Figure 5. illustrates the fourth, and last, step in the conversion cycle. With the fourth clock pulse, the A (1) bit is set to 1 and the DAC output is now 1.67 V (the A and C bits are set to 1).

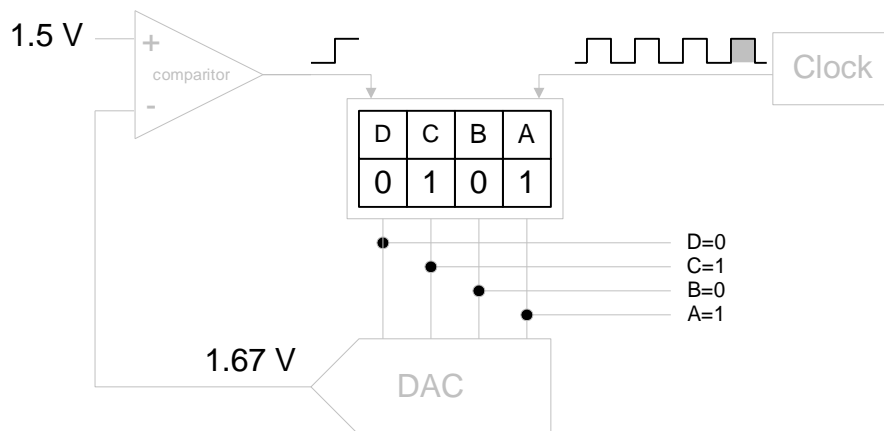


Figure 5.

At this point the conversion cycle is complete, since all 4 bits have been tried. The binary output of 0101_2 is the closest value to the 1.5 V input analog voltage. A new conversion cycle can now begin.

Note that during the conversion only four clock pulses were required. This illustrates an important advantage of the successive approximation A/D. It has a fixed conversion time that is independent of the input analog voltage, unlike other types of A/D converters. An N-bit SAR A/D will require N clock pulses to perform a conversion, regardless of the



level of the input analog voltage. The following timing diagram (Figure 6.) summarizes the sequence.

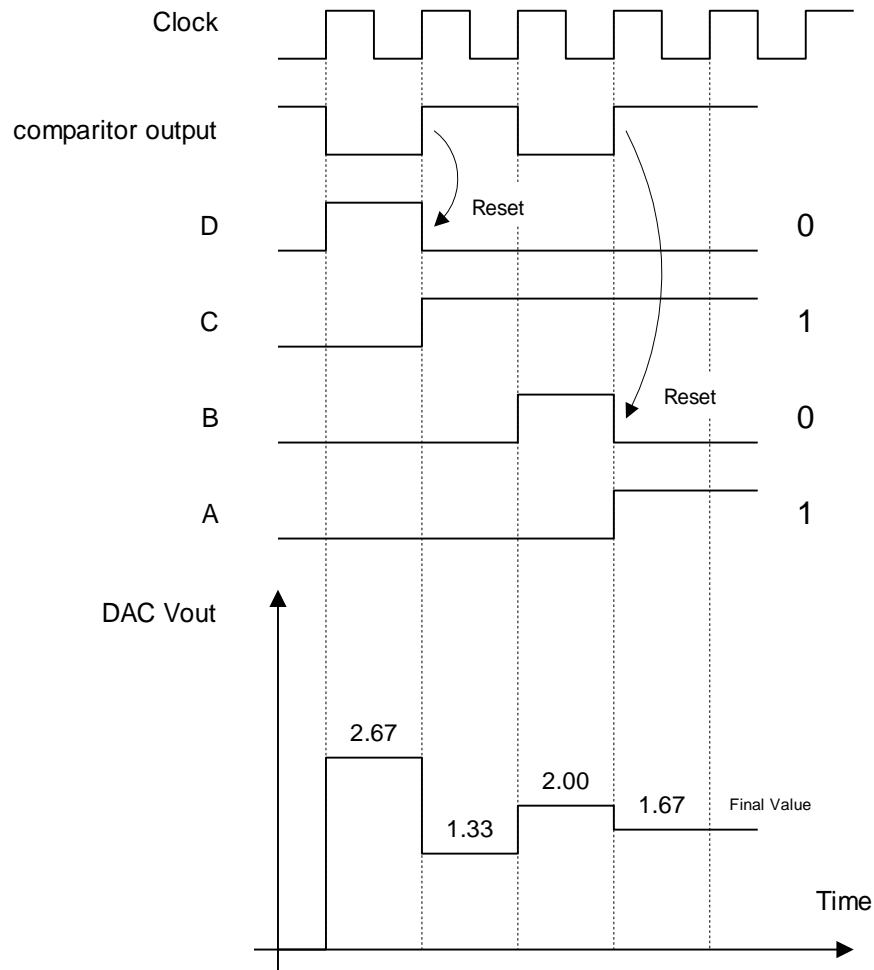


Figure 6. Timing Diagram



Summary

- Sampling converts an analog signal into a series of impulses, each representing the signal amplitude at a given instant in time.
- The sampling theorem states that the sampling frequency must be at least twice the highest sampled frequency (Nyquist frequency).
- Analog-to-digital conversion changes an analog signal into a series of digital codes.
- Four types of analog-to-digital converters (ADCs) are flash (simultaneous), dual-slope, successive-approximation, and sigma-delta.
- Digital-to-analog conversion changes a series of digital codes that represent an analog signal back into the analog signal.
- Two types of digital-to-analog converters (DACs) are binary-weighted input and *R/2R* ladder.
- Digital signal processing is the digital processing of analog signals, usually in real-time, for the purpose of modifying or enhancing the signal in some way.
- In general, a digital signal processing system consists of an anti-aliasing filter, a sample-and-hold circuit, an analog-to-digital converter, a DSP (digital signal processor), a digital-to-analog converter, and a reconstruction filter.
- A DSP is a specialized microprocessor optimized for speed in order to process data as it occurs (real-time).



Key Terms

Aliasing The effect created when a signal is sampled at less than twice the signal frequency. Aliasing creates unwanted frequencies that interfere with the signal frequency.

Analog-to-digital converter (ADC) A circuit used to convert an analog signal to digital form. Decode A stage of the DSP pipeline operation in which instructions are assigned to functional units and are decoded.

Digital-to-analog converter (DAC) A circuit used to convert the digital representation of an analog signal back to the analog signal.

DSP Digital signal processor; a special type of microprocessor that processes data in real time.

Quantization The process whereby a binary code is assigned to each sampled value during analog-to-digital conversion.

Sampling The process of taking a sufficient number of discrete values at points on a waveform that will define the shape of the waveform.



Sample Questions

Answer True/False

1. An analog signal is converted to a digital signal by an ADC.
2. A DAC is a digital approximation computer.
3. The Nyquist frequency is twice the sampling frequency.
4. A higher sampling rate is more accurate than a lower sampling rate for a given analog signal.
5. Resolution is the number of bits used by an analog-to-digital converter.
6. Successful approximation is an analog-to-digital conversion method.
7. Delta modulation is based on the difference of two successive samples.
8. Two types of DAC are the binary-weighted input and the $R/2R$ ladder.
9. The process of converting an analog value to a code is called *quantization*.
10. A flash ADC differs from a simultaneous ADC.

11. An ADC is an
 - (a) alphanumeric data code
 - (b) analog-to-digital converter
 - (c) analog device carrier
 - (d) analog-to-digital comparator
12. A DAC is a
 - (a) digital-to-analog computer
 - (b) digital analysis calculator
 - (c) data accumulation converter
 - (d) digital-to-analog converter
13. Sampling of an analog signal produces
 - (a) a series of impulses that are proportional to the amplitude of the signal
 - (b) a series of impulses that are proportional to the frequency of the signal
 - (c) digital codes that represent the analog signal amplitude
 - (d) digital codes that represent the time of each sample
14. According to the sampling theorem, the sampling frequency should be
 - (a) less than half the highest signal frequency
 - (b) greater than twice the highest signal frequency



- (c) less than half the lowest signal frequency
 - (d) greater than the lowest signal frequency
15. A hold action occurs
- (a) before each sample
 - (b) during each sample
 - (c) after the analog-to-digital conversion
 - (d) immediately after a sample
16. The quantization process
- (a) converts the sample-and-hold output to binary code
 - (b) converts a sample impulse to a level
 - (c) converts a sequence of binary codes to a reconstructed analog signal
 - (d) filters out unwanted frequencies before sampling takes place
17. Generally, an analog signal can be reconstructed more accurately with
- (a) more quantization levels
 - (b) fewer quantization levels
 - (c) a higher sampling frequency
 - (d) a lower sampling frequency
 - (e) either answer (a) or (c)
18. A flash ADC uses
- (a) an counter
 - (b) op-amps
 - (d) an integrator
 - (e) answer (a) and (c)
19. A dual-slope ADC uses
- (a) an counter
 - (b) op-amps
 - (d) a differentiator
 - (e) answer (a) and (c)
20. The output of a sigma-delta ADC is
- (a) parallel binary codes
 - (b) multiple-bit data
 - (c) single-bit data
 - (d) a difference voltage
21. In a binary-weighted DAC, the resistors on the inputs
- (a) determine the amplitude of the analog signal
 - (b) determine the weights of the digital inputs
 - (c) limit the power consumption
 - (d) prevent loading on the source

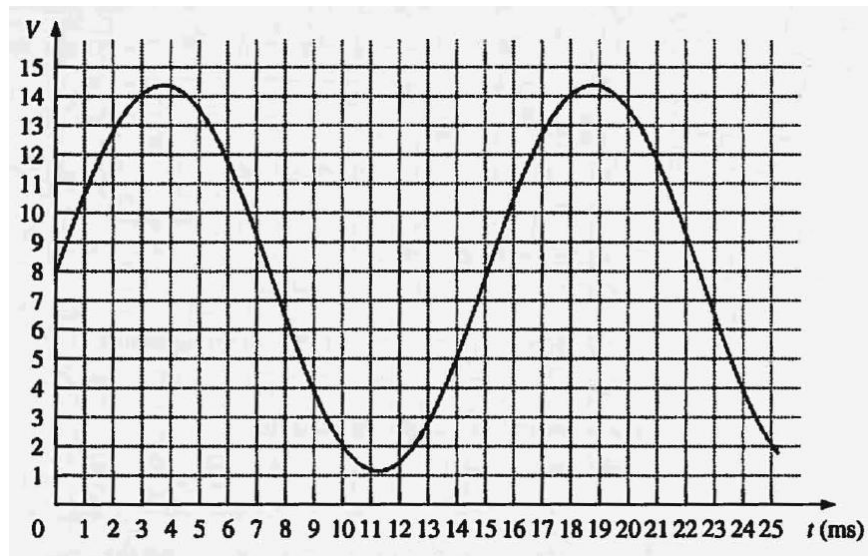


22. In an $R/2R$ DAC, there are
 - (a) four values of resistors
 - (b) one resistor value
 - (c) two resistor values
 - (d) a number of resistor values equal to the number of inputs
23. A digital signal processing system usually operates in
 - (a) real time
 - (b) imaginary time
 - (c) compressed time
 - (d) computer time
24. The term *Harvard architecture* means
 - (a) a CPU and a main memory
 - (b) a CPU and two data memories
 - (c) a CPU, a program memory, and a data memory
 - (d) a CPU and two register files
25. The minimum number of general-purpose registers in the TMS320C6000 series DSPs is
 - (a) 32
 - (b) 64
 - (c) 16
 - (d) 8
26. The two internal memories in the TMS320C6000 series each have a capacity of
 - (a) 1MB
 - (b) 512 kB
 - (c) 64 kB
 - (d) 32 kB
27. In the TMS320C6000 series pipeline operation, the number of instructions processed simultaneously is
 - (a) eight
 - (b) four
 - (c) two
 - (d) one
28. The stage of the pipeline operation in which instructions are retrieved from the memory is called
 - (a) execute
 - (b) decode
 - (c) accumulate
 - (d) fetch



Problems

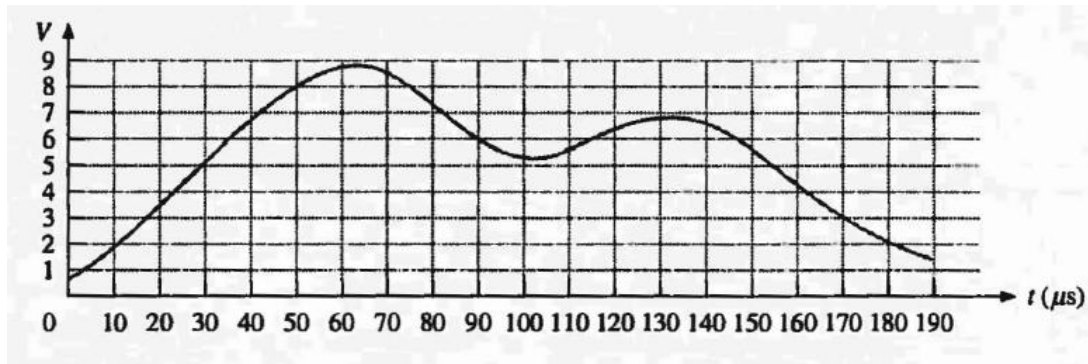
1. The waveform shown is applied to a sampling circuit and is sampled every 3 ms. Show the output of the sampling circuit. Assume a one-to-one voltage correspondence between the input and output.



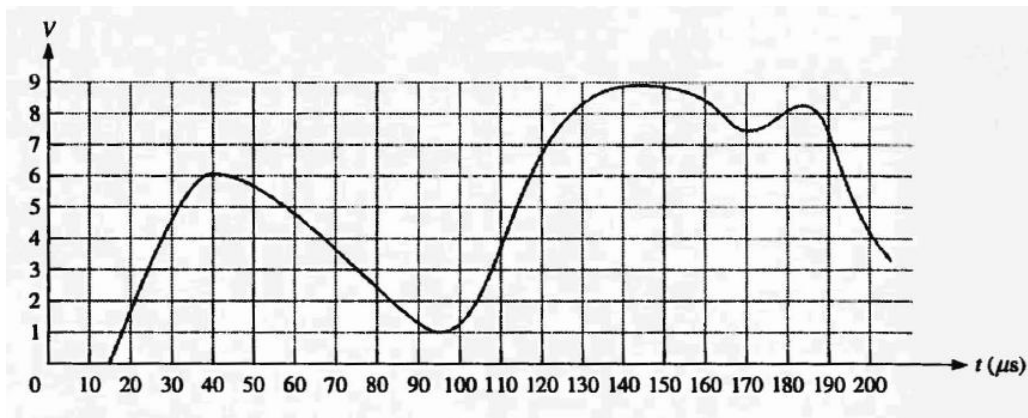
2. The output of the sampling circuit in Problem 1 is applied to a hold circuit. Show the output of the hold circuit.
3. If the output of the hold circuit in Problem 2 is quantized using two bits, what is the resulting sequence of binary codes?
4. Repeat Problem 3 using 4-bit quantization.
5. (a) Reconstruct the analog signal from the 2-bit quantization in Problem 3.
(b) Reconstruct the analog signal from the 4-bit quantization in Problem 4.
6. Graph the analog function represented by the following sequence of binary numbers:
1111, 1110, 1101, 1100, 1010, 1001, 1000, 0111, 0110, 0101, 0100, 0101, 0110, 0111, 1000, 1001, 1010, 1011, 1100, 1100, 1100, 1011, 1010, 1001.
7. The input voltage to a certain op-amp inverting amplifier is 10mV, and the output is 2 V. What is the closed-loop voltage gain?
8. To achieve a closed-loop voltage gain of 330 with an inverting amplifier, what value of feedback resistor do you use if $R_i = 1.0 \text{ K}\Omega$?
9. What is the gain of an inverting amplifier that uses a 47 $\text{K}\Omega$ feedback resistor if the input resistor is 2.2 $\text{K}\Omega$?



10. How many comparators are required to form an 8-bit flash converter?
11. Determine the binary output code of a 3-bit flash ADC for the analog input signal below.



12. Repeat Problem 11 for the analog waveform shown.





Design of Digital Filters

The design of a digital filter involves three major steps:

1. The filter's response characteristic has to be put into a mathematical form so that its behavior can be analyzed and evaluated. Analyzing the input output relationship of the analog filter.
2. Select an appropriate microcontroller to implement the numeric computation required.
3. Derive the digital implementation of the analog filter requires.

Discretization of Analog Systems

There are several ways to convert existing continuous or analog filters into discrete or digital systems. However, the conversion from the time domain or s-domain to the discrete domain causes some distortion. We will investigate the three most popular methods, and later discuss when to use each one.

A continuous function $F(t)$ can be represented digitally as $F(nT)$, where n is the n th sample, and T is the sampling period.

The first method is the simplest, and approximates a analog filter based on the fact that a *differential* can be approximated with the *difference* of two points or samples.

The second and third methods use what is known as the z transform, converting an s-domain transfer function into its discrete z-domain counterpart to then be converted to its required difference equation. The z-transform is considered the discrete (digital) version of the Laplace transform.

The second method is known as **zero-order hold**, and the third method is known as the bilinear **transformation**.

The zero-order hold method is the most accurate method of designing digital filters, and all the work is done in the digital domain. It is also the most difficult.

The bilinear transformation method takes an analog solution, and transforms it into a digital approximation.

Approximating a Analog Filter

The following procedure can be used to convert a continuous system expressed as a transfer function or differential equation into a digital system. The digital system will result in a *difference equation*, suitable to be processed by a computer.

1. Obtain the analog transfer function of the filter, and determine the required sampling time, T_s .
2. Convert the transfer function to a differential equation by taking the inverse Laplace transform (more on this later).



3. Convert the differential equation to a difference equation using the following relations:

$$y(t) = Y(n),$$

where (n) is the present value and

$$x(t) = X(n),$$

$(n-1)$ the previous value

$$s = \frac{d}{dt}$$

This is the inverse Laplace transform

$$dy(t) = Y(n) - Y(n-1), \text{ and}$$

Ts is the sampling time of the digitized system

$$dt = t(n) - t(n-1) = Ts$$

4. Solve the expression for the present value of the output in terms of the present input, past inputs, and past outputs as needed.

The following example will demonstrate the method.

Step 1

The following transfer function describes a simple RC low-pass filter with a cutoff frequency of 5.0 Hz, or a time constant of $RC = \frac{1}{2\pi \times 5.0} = 0.0318 \text{ sec.}$

$$\frac{Y(s)}{X(s)} = \frac{1}{1 + RCs}$$

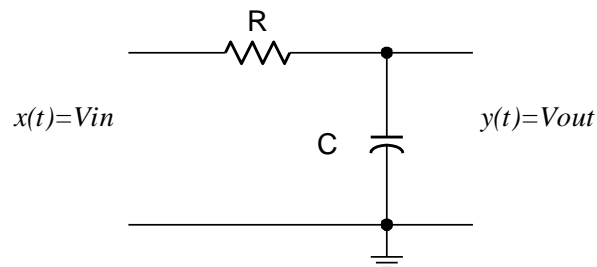


Figure 1 RC low-pass filter



Step 2

$$\frac{Y(s)}{X(s)} = \frac{1}{1 + RCs}$$

then $Y(s) \times (1 + RCs) = X(s)$

$$Y(s) + RCsY(s) = X(s)$$

Take the inverse Laplace transform, by replacing the s operator with its inverse Laplace transform: the differentiator $\frac{d}{dt}$, and expressing the input and output in the time domain.

$$y(t) + RC \frac{d}{dt} y(t) = x(t)$$

or $y(t) + RC \frac{dy(t)}{dt} = x(t)$

Step 3

With a sampling rate of $f_s = 100 \text{ Hz}$, or $T_s = 0.01$, digitize the equation by making the following replacements:

$$y(t) = Y(n),$$

$$x(t) = X(n),$$

$$dy(t) = Y(n) - Y(n-1), \text{ and}$$

$$dt = T_s$$

where (n) is the present value

and $(n-1)$ the previous value

then $Y(n) + RC \frac{Y(n) - Y(n-1)}{T_s} = X(n)$

Step 4

Solve for $Y(n)$ the present output

$$Y(n) + RC \frac{Y(n) - Y(n-1)}{T_s} = X(n)$$

$$Y(n) + \frac{RC}{T_s} Y(n) - \frac{RC}{T_s} Y(n-1) = X(n)$$



$$Y(n)\left(1 + \frac{RC}{T_s}\right) - \frac{RC}{T_s}Y(n-1) = X(n)$$

$$Y(n)\left(1 + \frac{RC}{T_s}\right) = X(n) + \frac{RC}{T_s}Y(n-1)$$

$$Y(n) = \frac{X(n) + \frac{RC}{T_s}Y(n-1)}{\left(1 + \frac{RC}{T_s}\right)} = \frac{X(n) + \frac{RC}{T_s}Y(n-1)}{\frac{T_s + RC}{T_s}}$$

$$Y(n) = \frac{T_s}{T_s + RC}X(n) + \frac{RC}{T_s + RC}Y(n-1)$$

substituting $RC = 0.0318$, and $T_s = 0.01$ we get our final difference equation

$$Y(n) = \frac{0.01}{0.01 + 0.0318}X(n) + \frac{0.0318}{0.01 + 0.0318}Y(n-1)$$

$$\underline{Y(n) = 0.24X(n) + 0.76Y(n-1)}$$

The difference equation

The following points calculated at 0.01sec intervals shows the response of the *difference equation* to a step input of 1. Also shown (continuous plot) is the theoretical response.

A look at the graph shows that the difference equation or computer approximation is very close to the original continuous solution.

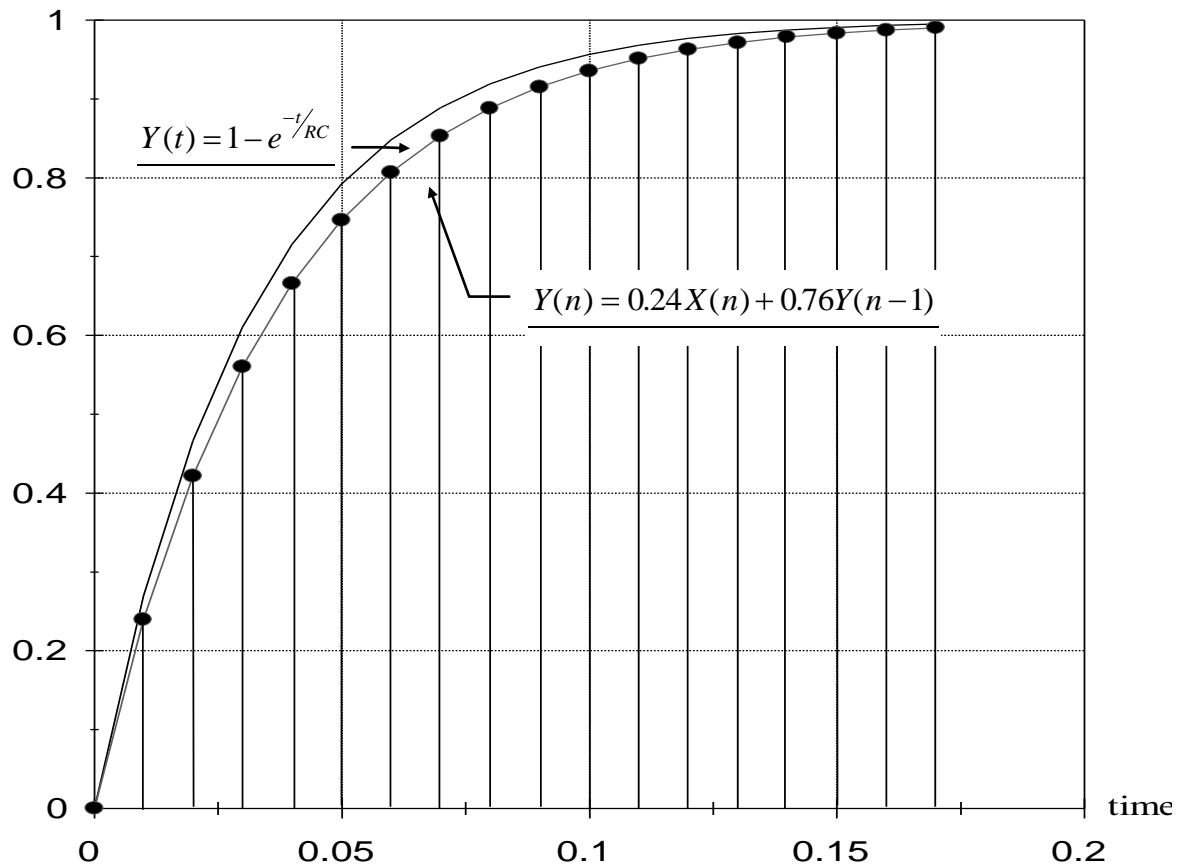


Figure 2 Time response

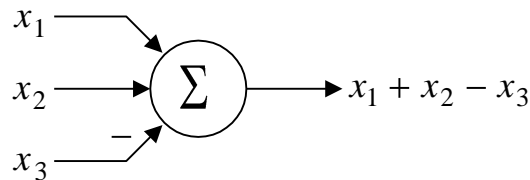
This example enables us to appreciate that a computer can approximate the solution to differential equations by using a *difference equation*, provided that an appropriately designed data acquisition system is available.

The student is to implement the above difference equation using a spreadsheet to reproduce the theoretical time response and its digitized approximation.

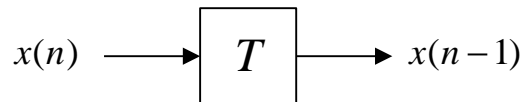


Flow Graphs

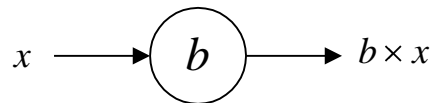
Flow Graphs are typically used to represent difference equations pictorially. They are composed of the following symbols (figure 4-1.):



Summation



Unit Time Delay



Multiplication

Figure 3 Signal Flow Symbols

The symbols represent summation (addition of all the input elements to produce an output), one unit discrete-time delay, and multiplication by a constant, respectively.

Using these symbols only, any difference equation can be represented graphically.

As an example we will represent the difference equation of our RC low-pass filter using the above symbols.

Our equation was: $Y(n) = 0.24X(n) + 0.76Y(n-1)$

This will produce the following Flow Graph:

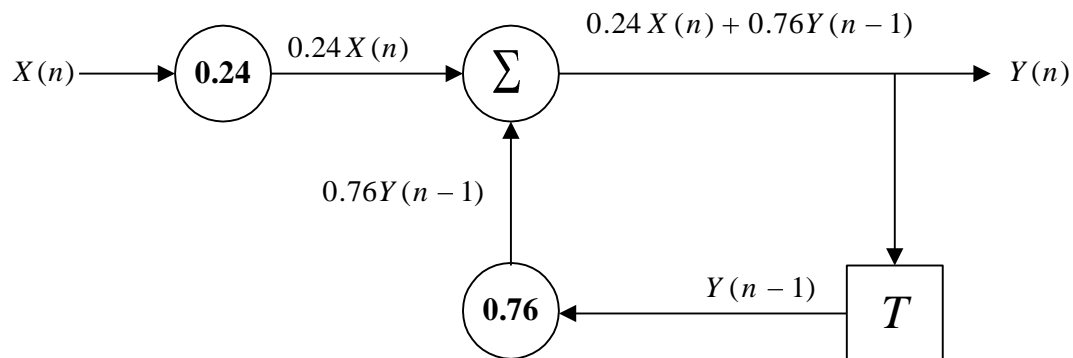


Figure 4-2 Flow Graph
of
 $Y(n) = 0.24X(n) + 0.76Y(n-1)$



Example

Derive the numeric difference equation using the difference approximation for the following differential equation, with a sampling time of 0.1 sec.

$$y(t) + 2 \frac{dy(t)}{dt} = x(t) \quad \text{where } y(t) \text{ is the output}$$

and $x(t)$ is the input

since the equation is provided in differential form, we can apply our substitution directly

$$\text{then } Y(n) + 2 \frac{Y(n) - Y(n-1)}{T_s} = X(n)$$

solving for $Y(n)$ gives

$$Y(n) = \frac{T_s}{T_s + 2} X(n) + \frac{2}{T_s + 2} Y(n-1) \quad \text{we can now substitute } T_s = 0.1$$

then $Y(n) = 0.0476X(n) + 0.9524Y(n-1)$ our final difference equation.

Example

Derive the difference equation of a simple RC high pass filter with a cutoff frequency of 20.0 Hz, See Figure 3-3.

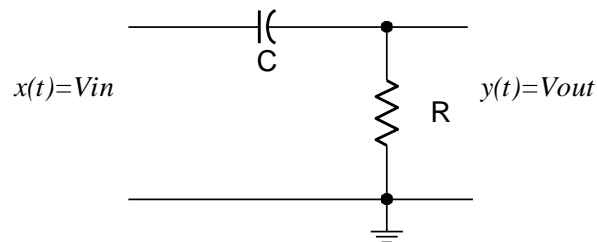


Figure 3-3 RC High Pass Filter

For a 20 Hz cutoff frequency, $f = \frac{1}{2\pi RC}$ or

$$RC = \frac{1}{2\pi \times 20} = 7.95 \text{ msec.}$$



Troubleshooting Op-Amp Circuits

Introduction

The ability to find and correct problems in a systematic manner is an exceptionally valuable skill. In Industrial Electronics, students are usually confronted with circuits that utilize many components, breadboards, and measuring equipment. This virtually guarantees that nothing will work the first time, and students are forced to develop effective troubleshooting skills.

Students should ensure that they are able to answer the following relevant questions before asking the lab instructor for help.

Carry out quick and easy checks first: start with a visual inspection, then use a multimeter, then the oscilloscope.

1. Is the schematic diagram complete, clear, properly prepared and available on the bench? A functional diagram is not sufficient to construct working circuits. The schematic diagram must contain all the required information. Do not keep the information necessary to constructing your circuit spread about in different places.
2. Are you sure of the pin-outs and polarity of each device? Check the specification sheet, and verify that the schematic is correct.
3. Do you understand how the circuit is supposed to work? If we do not know what to expect when making a measurement, we cannot tell if the results of the measurement are correct !

Power and Grounds

1. Is the power supply working, and properly connected ?
Check for stable outputs with a digital multimeter (DMM).
2. Are op-amps connected to both supply rails?
Check voltages with respect to ground at the appropriate pins of each device. Do not cause a short-circuits with the DMM probes.
3. Is the ground rail continuous ?
Check voltage between power supply rail and the points on the circuit that should be at ground.



Passive Components

1. Are the resistor and capacitor values correct ?
Check the color code, or use a DMM for resistor values. Remember to disconnect the components from the circuit before taking measurements. Don't simultaneously touch both probes with your hands when making the measurement or you will get misleading results.
2. Are potentiometers being used correctly ?
Check that the wiper voltage varies in a reasonable manner as the potentiometer is adjusted. Avoid using potentiometers as variable resistors.
3. Are the switches of the correct type ?
Have you checked your assumptions about which terminal is which, with a DMM ?

Op-Amps

If you suspect an op-amp IC is faulty, check it by substitution but first:

1. Is each required pin of the op-amp IC connected, and is it correct ?
Check visually that none of its pins have become wrapped under its body instead of being inserted into the board.
2. Is the output finite (saturated) ?
If the output is within a volt or so of either power rail, when it should not be in saturation, then it has either failed, or there is an excessive voltage at its inputs.
3. If an op-amp has negative feedback, is the input consistent with the output ?
Measure the voltage between the inverting and non-inverting inputs. It should be close to zero or within millivolts of zero.

Offset Voltages

A few millivolts of offset at the input of a system that has a high DC gain can be amplified to the point that it saturates the output stage. Many op-amps have offset adjust facilities which can reduce the offset by something like a factor of 10. Offsets are often temperature dependent.

Instability

Instability typically appears as high-frequency 'fuzz' on the output signal oscilloscope trace. Breadboard circuits are very prone to this because of inter-track capacitance and long components leads. Try the following:

1. Organize the physical layout of the circuit to keep the input and output stages separate.



2. De-couple the breadboard power supply rails with $0.1\mu\text{F}$ capacitors to ground.
3. Work out the loop-gain (see worksheet 1) and study the op-amp manufacturer's datasheet.
4. Don't use op-amps with an unnecessarily good frequency response, eg use a 741/747 or 324 type where possible.

Divide the circuit into stages

If you have a complex system involving several stages:

1. Divide the system into sub-circuits that can be tested individually.
If you want to do an open-loop test on a closed-loop system use a signal generator or voltage source to inject a simulation of the closed-loop signal at the point where you open the loop.
2. Check the signal at the input and output of each section, using an oscilloscope if appropriate.
It is possible for a faulty section to load its predecessor in the chain.

How to Kill a Working Circuit

1. Short-circuit a supply rail to something sensitive with the DMM probes when checking the power supply or by dropping something (eg. a screwdriver) onto the working circuit.
2. Apply power for an instant to only one rail of a circuit that requires two rails.
3. Make changes to the circuit **without switching off the power supply first**

For more information, check the tips on breadboarding op-amp circuits in Lab 2.



ISA 5.1 2009 Signal processing function block symbols

- 57 -

ANSI/ISA-5.1-2009

Table 5.6 — Signal processing function block symbols

Note: Numbers in parentheses refer to explanatory notes in Clause 5.3.8.


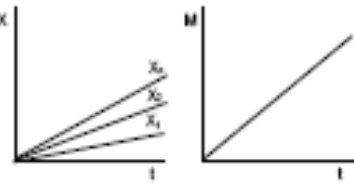


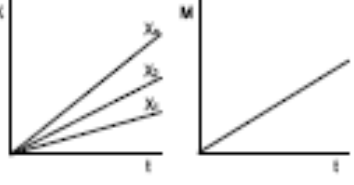


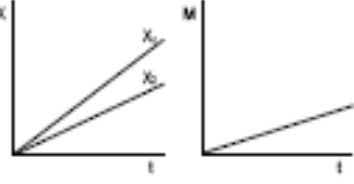


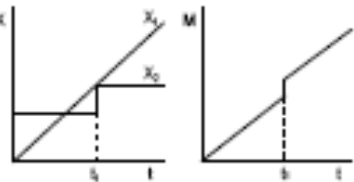

OPER. FUNCTIONS IN MICROPROCESSORS result as supervisory mode in control units.				
No	Function	Equation	Definition	
	Symbol (1) (2)	Graph		
1	Summation	$M = X_1 + X_2 \dots + X_n$	<ul style="list-style-type: none">• Output equals algebraic sum of inputs.	
				
				
2	Average	$M = (X_1 + X_2 \dots + X_n)/n$	<ul style="list-style-type: none">• Output equals algebraic sum of inputs divided by number of inputs.	
				
				
3	Difference	$M = X_1 - X_2$	<ul style="list-style-type: none">• Output equals algebraic difference of two inputs.	
				
				
4	Multiplication	$M = X_1 \times X_2$	<ul style="list-style-type: none">• Output equals product of two inputs.	
				
				



Table 5.6 — Signal processing function block symbols

Note: Numbers in parentheses refer to explanatory notes in Clause 5.3.8.



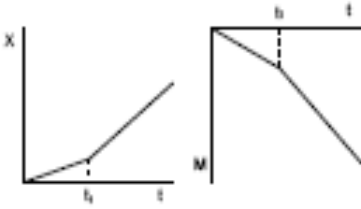




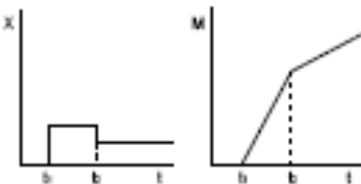
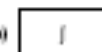



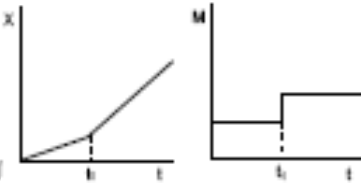



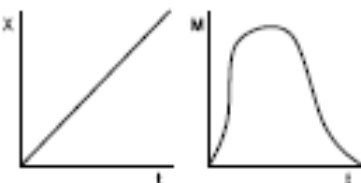

No	Function	Equation	Definition
	Symbol (1) (2)	Graph	
9	Reverse proportion	$M = -KX$ or $M = -PX$	<ul style="list-style-type: none"> Output inversely proportional to input. Replace '-K' or '-P' with '1:1' for volume boosters. Replace '-K' or '-P' with '1:2:1', '1:3:1', etc., for integer gains.
	(3) a)  b) 		
	(3) a)  b) 		
10	Integral	$M = (1/T_I) \int X dt$	<ul style="list-style-type: none"> Output varies with magnitude and time duration of input. Output proportional to time integral of input. T_I = integral time constant.
	(3) a)  b) 		
	(3) a)  b) 		
11	Derivative	$M = T_D (dx/dt)$	<ul style="list-style-type: none"> Output proportional to time rate of change of input. T_D = derivative time constant.
	(3) a)  b) 		
	(3) a)  b) 		
12	Unspecified function	$M = f(X)$	<ul style="list-style-type: none"> Output is a nonlinear or unspecified function of the input. Function defined in note or other text.
			
			



Table 5.6 — Signal processing function block symbols

Note: Numbers in parentheses refer to explanatory notes in Clause 5.3.8.


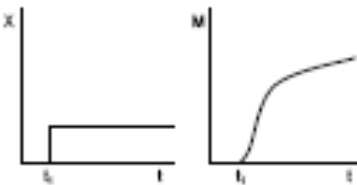


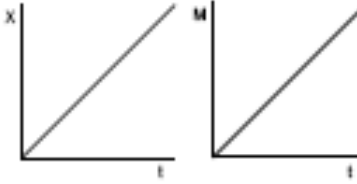


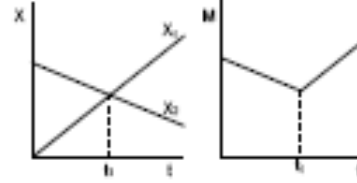


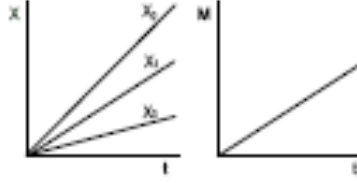

No	Function	Equation	Definition
	Symbol (1) (2)	Graph	
13	Time function	$M = X_f(t)$	<ul style="list-style-type: none"> Output equals a nonlinear or unspecified time function times the input. Output is a nonlinear or unspecified time function. Function defined in note or other text.
			
			
14	Conversion	$I = P, P = I, \text{ etc.}$	<ul style="list-style-type: none"> Output signal type different from that of input signal. Input signal is on the left and output signal is on the right. Substitute any of the following signal types for 'P' or 'I': A = Analog H = Hydraulic B = Binary I = Current D = Digital O = Electromagnetic E = Voltage P = Pneumatic F = Frequency R = Resistance
			
			
15	High signal select	$M = X_1 \text{ for } X_1 > X_2$ $M = X_2 \text{ for } X_1 \leq X_2$	<ul style="list-style-type: none"> Output equals greater of 2 or more inputs.
			
			
18	Middle signal select	$M = X_1 \text{ for } X_2 > X_3 \text{ or } X_2 > X_1 > X_3$ $M = X_2 \text{ for } X_1 > X_2 > X_3 \text{ or } X_2 > X_3 > X_1$ $M = X_3 \text{ for } X_1 > X_2 > X_3 \text{ or } X_2 > X_3 > X_1$	<ul style="list-style-type: none"> Output equals middle value of three or more inputs.
			
			



Table 5.6 — Signal processing function block symbols

Note: Numbers in parentheses refer to explanatory notes in Clause 5.3.6.


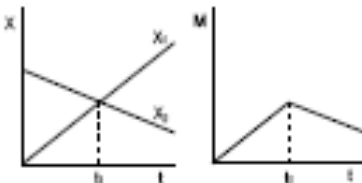


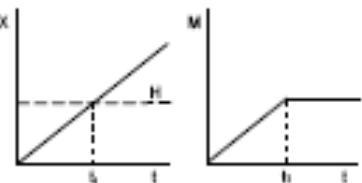


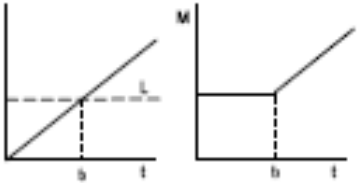


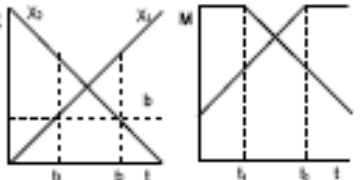

No	Function	Equation	Definition
	Symbol (1) (2)	Graph	
17	Low signal select	$M = X_1 \text{ for } X_1 \leq X_2$ $M = X_2 \text{ for } X_1 \geq X_2$	<ul style="list-style-type: none">Output equals lesser of 2 or more inputs.
			
			
18	High limit	$M = X \text{ for } X \leq H$ $M = H \text{ for } X \geq H$	<ul style="list-style-type: none">Output equals the lower of the input or high limit values.
			
			
19	Low limit	$M = X \text{ for } X \geq L$ $M = L \text{ for } X \leq L$	<ul style="list-style-type: none">Output equals the higher of the input or low limit values.
			
			
20	Positive bias	$M = X_1 + b$ $M = [-X_2] + b$	<ul style="list-style-type: none">Output equal to input plus an arbitrary value.
			
			



Table 5.6 — Signal processing function block symbols

Note: Numbers in parentheses refer to explanatory notes in Clause 5.3.8.



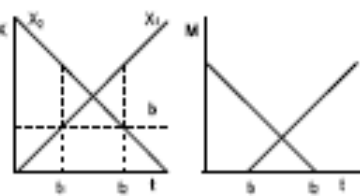

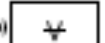
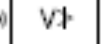
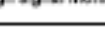
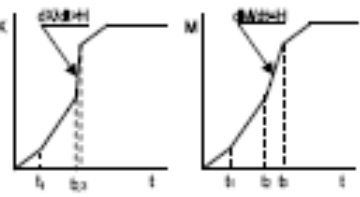


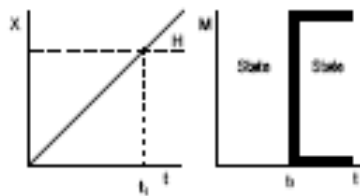


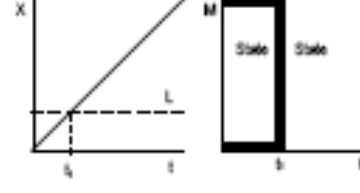
No	Function	Equation	Definition
	Symbol (1) (2)	Graph	
21	Negative Bias	$M = X_1 - b$ $M = [-X_2] - b$	<ul style="list-style-type: none"> Output equal to input minus an arbitrary value.
	 		
22	Velocity limiter	$dM/dt = dX/dt$ for $dX/dt \leq H$, $M = X$ $dM/dt = H$ for $dX/dt \geq H$, $M \neq X$	<ul style="list-style-type: none"> Output equals input as long as the input rate of change does not exceed the limit value that establishes the output rate of change until the output again equals the input.
	(3)  (3)   		
23	High signal monitor	(State 1) $M = 0$ @ $X < H$ (State 2) $M = 1$ @ $X \geq H$	<ul style="list-style-type: none"> Output state is dependent on value of input. Output changes state when input is equal to or higher than an arbitrary high limit.
	 		
24	Low signal monitor	(State 1) $M = 1$ @ $X \leq L$ (State 2) $M = 0$ @ $X > L$	<ul style="list-style-type: none"> Output state is dependent on value of input. Output changes state when input is equal to or lower than an arbitrary low limit.
	 		



Table 5.6 — Signal processing function block symbols

Note: Numbers in parentheses refer to explanatory notes in Clause 5.3.8.

No	Function	Equation	Definition
	Symbol (1) (2)	Graph	
25	High/low signal monitor	(State 1) $M = 1 @ X \leq L$ (State 2) $M = 0 @ L < X < H$ (State 3) $M = 1 @ X \geq H$	<ul style="list-style-type: none">• Output states are dependent on value of input.• Output changes state when input is equal to or lower than an arbitrary low limit or equal to or higher than an arbitrary high limit.
26	Analog signal generator	No equation	<ul style="list-style-type: none">• Output equals a variable analog signal that is generated:<ul style="list-style-type: none">a. Automatically and is not adjustable by operator.b. Manually and is adjustable by operator.
		No graph	
27	Binary signal generator	No equation	<ul style="list-style-type: none">• Output equals an on-off binary signal that is generated:<ul style="list-style-type: none">a. Automatically and is not adjustable by operator.b. Manually and is adjustable by operator.
		No graph	
28	Signal transfer	(State 1) $M = X_1$ (State 2) $M = X_2$	<ul style="list-style-type: none">• Output equals input that is selected by transfer.• Transfer actuated by external signal.

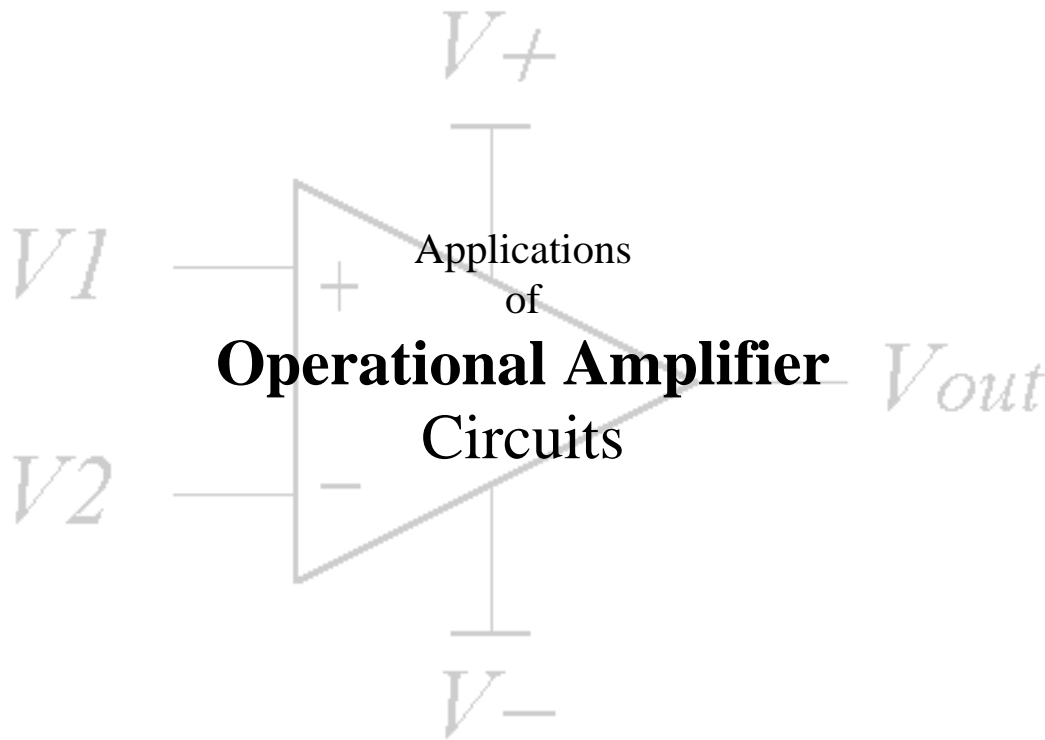


Section 2 Labs

Table of Content

Lab Activity

1	Operational Amplifier Circuits: follower, inverting, non-inverting
2	Frequency Response and Bode Plots
3	Signal Conditioning Circuits: $y = mx + b$
4	Voltage to Current (1 to 5V input 4 to 20mA output)
5	Second Order Active Low-Pass Filter
6	Operational Amplifier Circuits: Comparators
7	Operational Amplifier Circuits: Integration
8	Waveform Generation
9	Common Mode Rejection
10	
11	
12	Operational Amplifier Circuits with Diodes
13	Sample and Hold
14	Analog to Digital (A/D) Conversion
15	Digital to Analog (D/A) Conversion



Objectives

- Identify operational amplifier part numbers
- Breadboard simple operational circuits
 - Voltage Follower
 - Inverting amplifier
 - Non-Inverting amplifier
- Test breadboarded circuits

These applications will reinforce what we have learned about operational amplifiers, by building and verifying actual circuit operation.



The Operational Amplifier Part Number

Most integrated circuit (IC) op-amps are identified by their part numbers, similar to digital IC's.

The part number can be divided into four elements.

1. Type (eg. 741)
2. Manufacturer (eg. Analog devices)
3. Specification (eg. temperature range)
4. Mechanical Packaging (eg. DIP)



DIP



metal can
TO5

For example



The letter prefix usually consists of two or three letters that identify the manufacturer

Manufacturers	AD	Analog Devices
	CA	RCA
	LM	National Semiconductors
	MC	Motorola
	NE/SE	Signetics
	TL	Texas Instruments
	μA	Fairchild

Specifications see data sheet

Temperature range	C	Commercial	0 to 70°C
	I	Industrial	-25 to 85°C
	M	Military	-55 to 125°C

Mechanical Packaging	N, P	Plastic dual-inline
	J	Ceramic dual-inline
	D	Plastic dual-inline for surface mount
	H	TO-99



Bread-boarding linear (opamps) IC circuits

The Power Supply

Power supplies for general-purpose linear, or op amp circuits are normally bipolar as shown in figure 1. , and typical commercially power supply outputs are $\pm 15V$ or $\pm 12V$. The common point between the positive supply and the negative supply is called the power supply common. It is shown with a ground symbol for two reasons. First, all voltage measurements are made with respect to this point. Second, power supply common is usually wired to the third wire of the line cord that extends to ground, and other equipment Figure 2. shows how to connect the lab power supply to give us the bipolar voltages required.

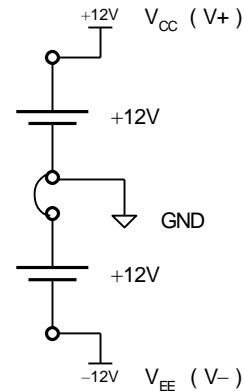


Figure 1. Dual Power Supply

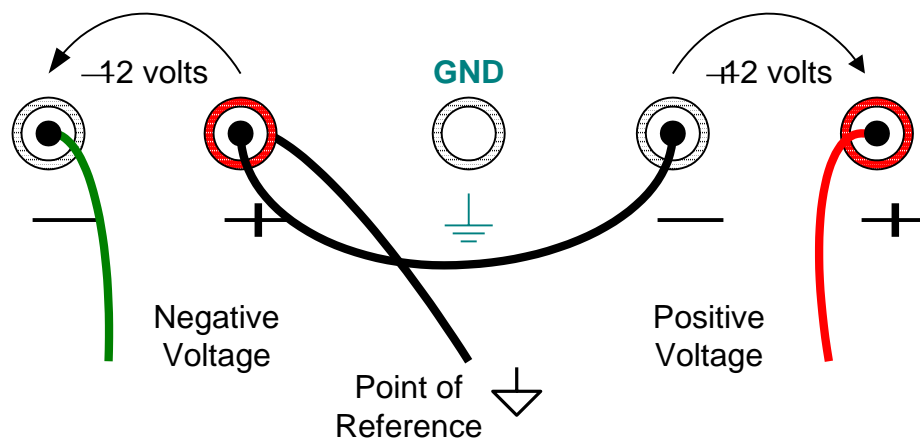


Figure 2. Actual Power Supply connection



Some IC op-amps are specified with a single supply (eg. LM324), these units can still function with dual power supply. Again the op-amp only sees the potential difference of its positive supply referenced to its negative supply. The signal with these units are also referenced to the supply center, or can be referenced to the op-amps negative terminal.



Breadboarding suggestions

Breadboarding is as much an art as a skill, but with a little effort and some common sense rules much frustration can be avoided, and successful circuits can be produced. The following ten rules summarize good practice.

1. Prepare a detailed schematic diagram and wiring diagram of the circuit to be breadboarded. This include checking for the correct pin numbers.
2. Document all connections and changes as you proceed step by step.
3. Do all wiring with power off. It is very important to remember that when changes are made to a circuit, the power must be disconnected first, before any components are changed or moved. Only after all changes have been made should power be turned on or reconnected.
4. Keep wiring and component leads short and flat.
5. Use multiple wires and tie points to make complex wire routings.
6. Wire the power supply and ground first, using the bus bars of the breadboard. (do not use the bus for signals)
7. Connect signal voltages only **after** the power is connected to the circuit. (see 9)
8. Take all measurement with respect to ground.
9. Disconnect the input signal before the dc power is removed.
Driving the input pins above or below the power supply may damage the IC (see 7)
10. Connect a 0.1 μ F capacitor between the power supply pins of the IC and ground if unwanted oscillation occurs at the output.

When possible section the circuit into functional blocks, build and check each block, then proceed to interconnect the blocks. (do not forget step 2)

For more information, read the notes on Troubleshooting Op-Amps.



Part 1 Voltage Follower

In this part you will assemble and test a voltage follower or unity gain noninverting amplifier. This amplifier has no voltage gain or loss, and the output signal is in phase with the input signal. Also, the input impedance of this amplifier is extremely high and its output impedance is extremely low.

The transfer function is given as:

$$\frac{V_{out}}{V_{in}} = 1$$

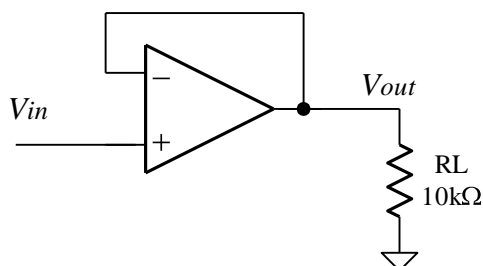


Figure 3. Voltage Follower

Procedure

1. Identify the part number of the operational amplifier provided, and using the specification sheet available on the network (*partslib*) identify the proper pin numbers to each terminal.
2. Draw a detailed schematic diagram (including test requirements) before assembling the circuit. Drawing an assembly diagram will also help in your construction
3. Assemble the circuit per your diagrams. Connect the 10 kΩ load resistor at the output
4. Apply ± 12 V to the circuits (first measure the power supply voltages before connecting to your circuit).
5. Using your Digital multimeter measure the DC voltages at the non-inverting input, inverting input, and output and record. All measurements are referenced to ground.
6. Connect the function generator to the input of the voltage follower, and set the function generator (using the oscilloscope) to supply the circuit with an input signal of 1 kHz, 1 Vp-p sine wave. Make sure the signal is zero referenced - no DC offset.
7. Using the oscilloscope, measure the peak-to-peak output voltage, and record. Use both channels if possible, and compare the phase relationship between input and output and record.
8. Decrease you power supply to ± 6 V.
Increase the amplitude of the input signal and note that the output voltage follows the input, until the output is at maximum voltage swing (just before clipping when the amplifier goes into saturation) and record.
9. Remove the signal source and power, before disassembling/assembling the next circuit.

Document all your results in your log book.



Part 2 *Inverting amplifier*

In this part you will assemble and test an inverting voltage amplifier, with a gain of 10. The input impedance of this amplifier unlike the follower is equal to the input resistance of the circuit (R_1). Its output impedance is still extremely low.

The transfer function is given as:

$$\frac{V_{out}}{V_{in}} = -\frac{R_2}{R_1}$$

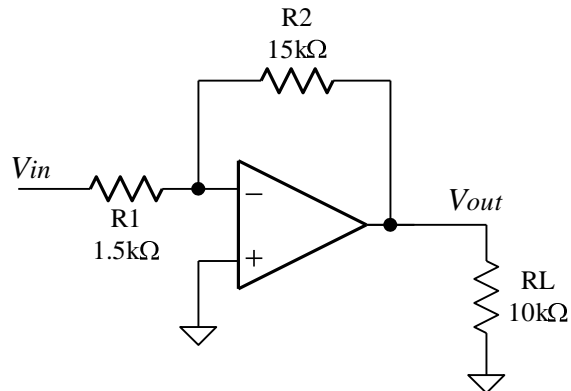


Figure 4. Inverting Amplifier

Procedure

Using the same procedure as in part 1, test the circuit with an input signal of 1 kHz, 1 Vp-p.

Also perform the following tests:

Adjust the function generator's offset voltage, and note the output voltage follows the input voltage with a gain of 10, but inverted (the negative sign). The output signal is 180° out of phase with respect to the input signal, or the output is inverted.

Measure the non-inverting input of the opamp. Does your measurement support the concept of *virtual ground*? Explain your answer.

Modify the circuit so that the voltage gain is reduced to 1.

Document all your results in your log book.



Part 3 Non-Inverting amplifier

In this part you will design assemble and test a non-inverting voltage amplifier, with a gain of 11. The input impedance of this amplifier, like the follower, is extremely high and its output impedance is extremely low.

The transfer function is given as:

$$\frac{V_{out}}{V_{in}} = 1 + \frac{R2}{R1}$$

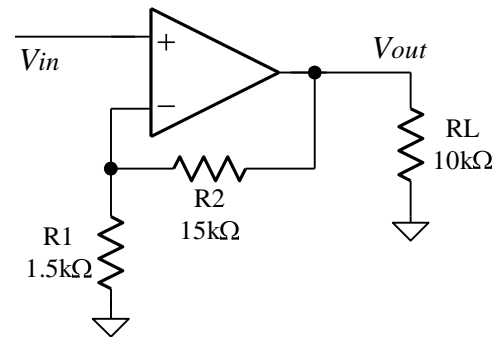


Figure 5. Non-Inverting Amplifier

Procedure

Using the same procedure as in part 1, test the circuit with an input signal of 1 kHz, 1 Vp-p.

Also perform the following tests:

Adjust the function generator's offset, but not its voltage, and note the output voltage follows the input voltage with a gain of 11. The output signal is in phase with respect to the input signal, or the output is not inverted.

Modify the circuit so that the voltage gain is reduced to 2.

Document all your results in your log book.



Lab Activity

Frequency Response and Bode Plots

Objectives

- Use a function generator and oscilloscope to determine the cutoff frequency and the frequency response of a RC low-pass and high-pass filter.
- Explain filter response in terms of dB, and Bode Plots.
- Observe the effect of ac coupling a signal.
- Observe the loading effect of concatenating multiple stages

This lab activity will provide practice in determining frequency response of circuit and systems, and in the construction of Bode Plots.



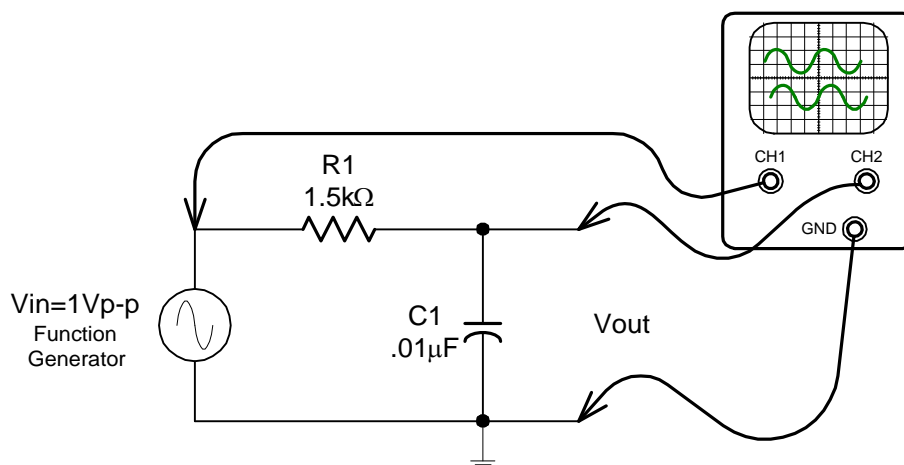
Part 1 RC Low Pass filter

In this part of the experiment you will construct and experimentally analyze a simple RC low-pass filter. You will make voltage measurements at different frequencies, at the filter's input and output and convert these measurements to decibels, and then construct a low-pass Bode Plot.

In DC coupling, the source is directly connected to the system. The DC coupling allows very slow changing signals to be acquired, but may not produce the desired display due to offset drift superimposed on top of the desired signal. See step 4

Procedure

1. Assemble the following RC lowpass circuit:



2. Make sure that your scope is set to DC, and the trigger is set to internal CH1
3. Calculate the theoretical cutoff frequency of the filter and record.
4. Set the function generator to produce a sine wave with an amplitude of 1Vp-p at 1kHz.
5. Slowly vary the function generator's offset adjustment, and note that the output follows the input exactly. This is when the signal is dc coupled. The offset adjustment superimposed on the ac signal can be considered a dc signal at low frequencies.
6. Before continuing, adjust the dc offset on the function generator back to zero.



7. Slowly adjust the frequency of the function generator until the filter's output is 3 dB ($0.707 V_{p-p}$) below its input voltage, and record the frequency.
8. Slowly vary the frequency and measure the filter's input and output voltage at different frequencies, and plot the filter's response on a dB vs log frequency scale ie. Bode Plot.

Note:

When measuring input and output voltages ratios on the oscilloscope, make sure that for each set of measurements both V_{in} and V_{out} are measured at the frequency of interest.

Select your frequencies as required to construct a appropriate Bode plot, noting that the horizontal axis is logarithmic, and that the bode plot consist primarily of two straight lines.



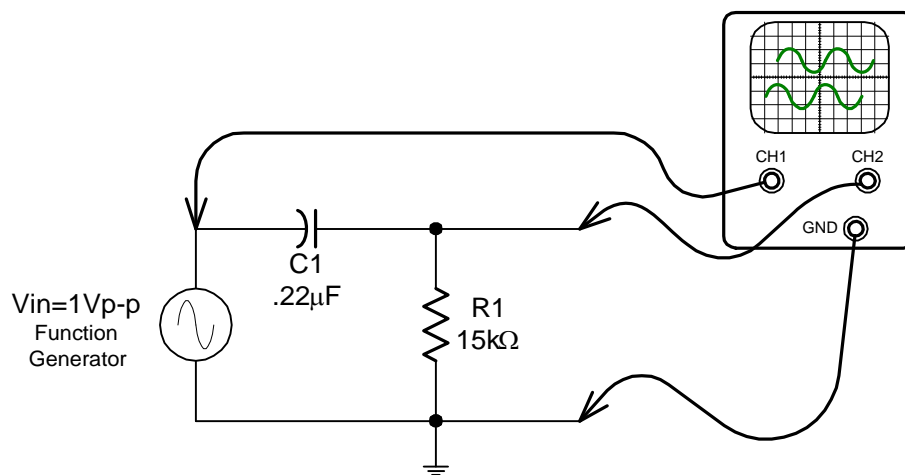
Part 2 RC High Pass filter (ac coupling)

In this part of the experiment you will construct and experimentally analyze a simple RC highpass filter. You will make voltage measurements for different frequencies at the filters input and output and convert these measurements to decibels to construct a highpass Bode Plot. You will also observe the effect of ac coupling a signal.

In AC coupling, the source is passed through a capacitor to block the DC component of the signal, passing the AC portion only. Note that in AC coupling an AC signal must be present. The DC signal usually presents itself as a slow changing drift superimposed on the desired AC signal. See step 4

Procedure

1. Assemble the following RC high pass filter circuit.



2. Make sure that your scope is set to DC, and the trigger is set to internal CH1
3. Calculate the theoretical cutoff frequency and record.
4. Set the function generator to produce a sine wave with an amplitude of 1Vp-p at 1kHz.
5. Slowly vary the function generator's offset adjustment, and note that the output *resists* to follow the input. Instead the output returns to zero offset. This is when a signal is *ac coupled*. The slow varying offset adjustment superimposed on the ac signal is removed from the higher frequencies.
6. Before continuing, adjust the dc offset to zero, and record your findings of step 4.



7. Slowly adjust the frequency of the function generator until the filter's output is 3 dB below its input, and record
8. Slowly vary the frequency and measure the filter's input and output voltage at different frequencies, and plot the filter's response on a dB vs log frequency scale ie. Bode Plot.



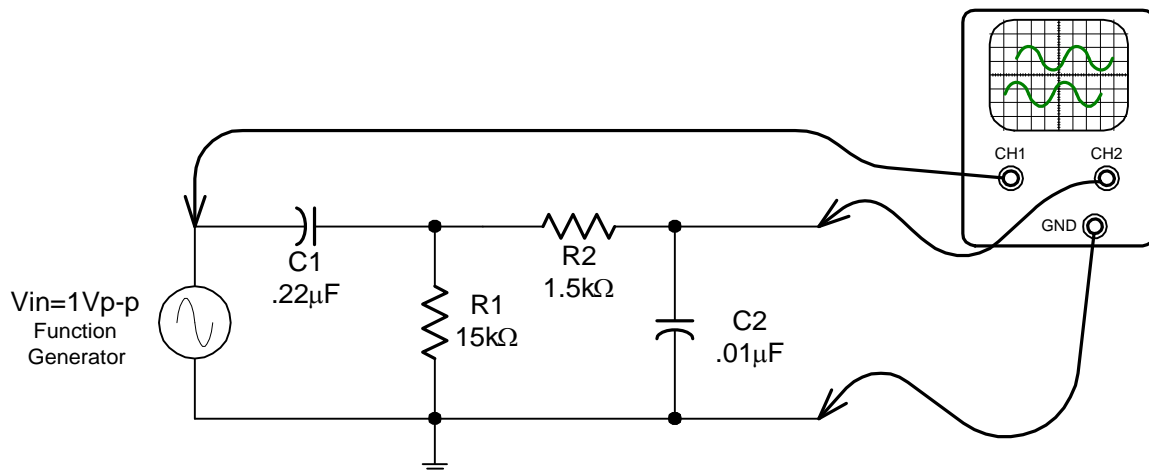
Part 3 RC Band Pass filter

All systems behave as a simple lowpass filter, with the cutoff frequency above the frequencies of interest. Most systems that deal with ac signals, also exhibit some type of ac coupling.

In this part of the experiment you will construct and experimentally analyze a simple RC bandpass filter. You will make voltage measurements for different frequencies at the filters input and output and convert these measurements to decibels to construct a bandpass Bode Plot.

Procedure

1. Assemble the following RC band-pass filter circuit:



2. Make sure that your scope is set to DC, and the trigger is set to internal CH1
3. Calculate the theoretical cutoff frequencies and record.
4. Set the function generator to produce a sine wave with an amplitude of 1Vp-p at 1kHz.
5. Adjust the function generator's dc offset to zero
6. Slowly adjust the frequency of the function generator until the filters output is 3 dB below its input, and record. Note that this should occur at two specific frequencies. Explain why.



7. Slowly vary the frequency and measure the filters input and output voltage at different frequencies, and plot the filters response on a dB vs log frequency scale ie. Bode Plot.
8. Compare your results (Bode plot) with the results obtained in part 1 and 2. Explain your observations.



*Frequency Response and Bode Plots **Lab Results***

Student's Name: _____ Date: _____

Part 1 RC Low Pass filter

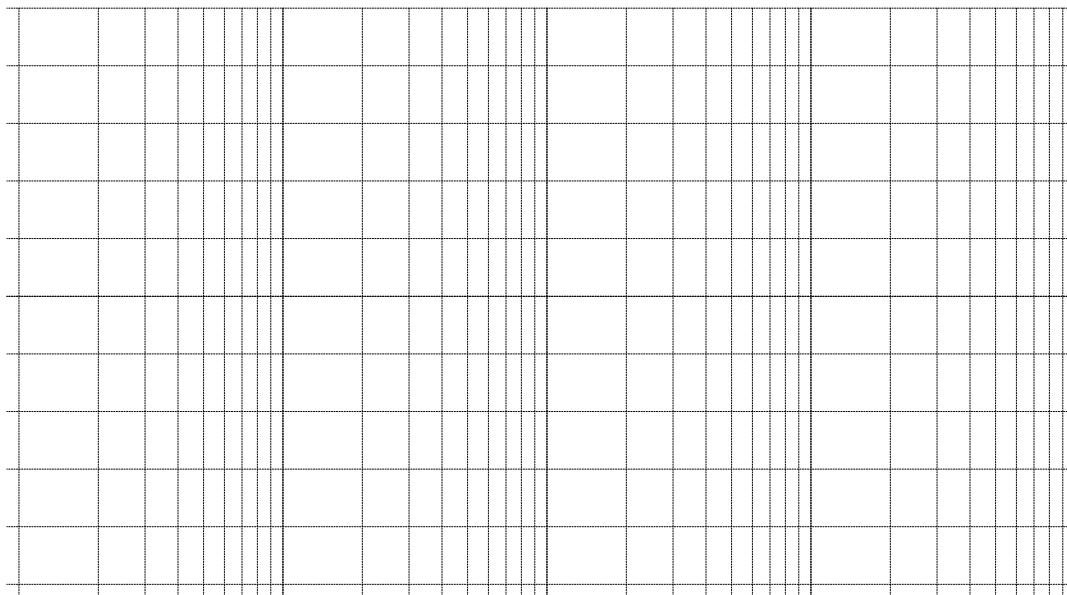
Calculated theoretical cutoff frequency: $f = \frac{1}{2\pi RC} =$ _____

3 dB point input voltage _____

 output voltage _____

 @ frequency _____

% deviation from theoretical _____





*Frequency Response and Bode Plots **Lab Results***

Student's Name: _____ Date: _____

Part 2 RC High Pass filter (ac coupling)

Calculated theoretical cutoff frequency: $f = \frac{1}{2\pi RC} =$ _____

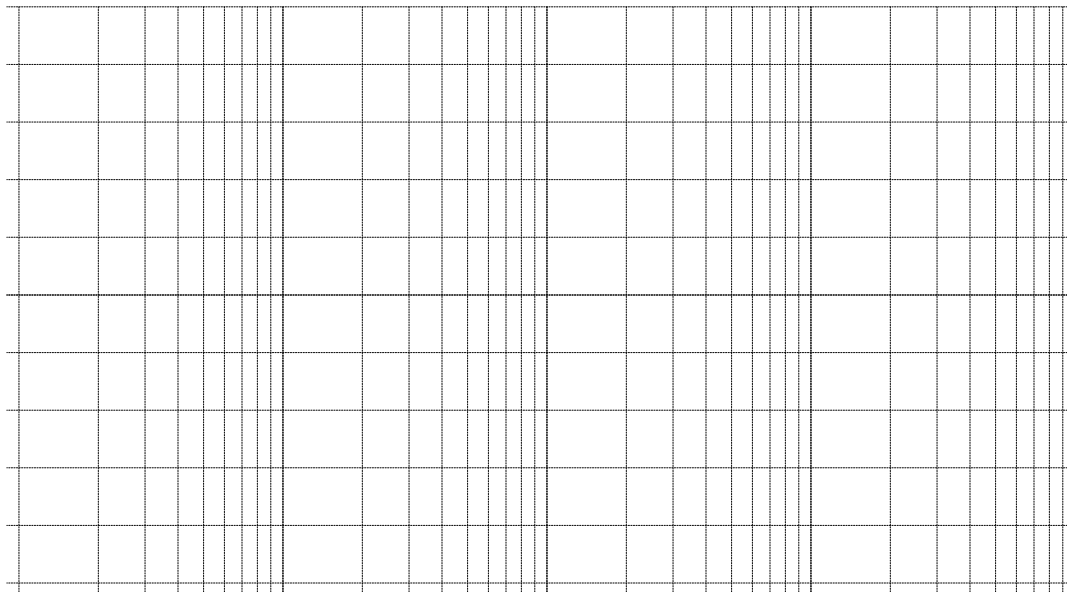
Note offset adjustment superimposed on the ac signal. _____

3 dB point input voltage _____

output voltage _____

@ frequency _____

% deviation from theoretical _____





*Frequency Response and Bode Plots **Lab Results***

Student's Name: _____ Date: _____

Part 3 RC Band Pass filter

Calculated theoretical cutoff frequencies: Low Pass _____

High Pass _____

3 dB point

Low Pass

input and output voltage _____

@ frequencies _____

% deviation from theoretical _____

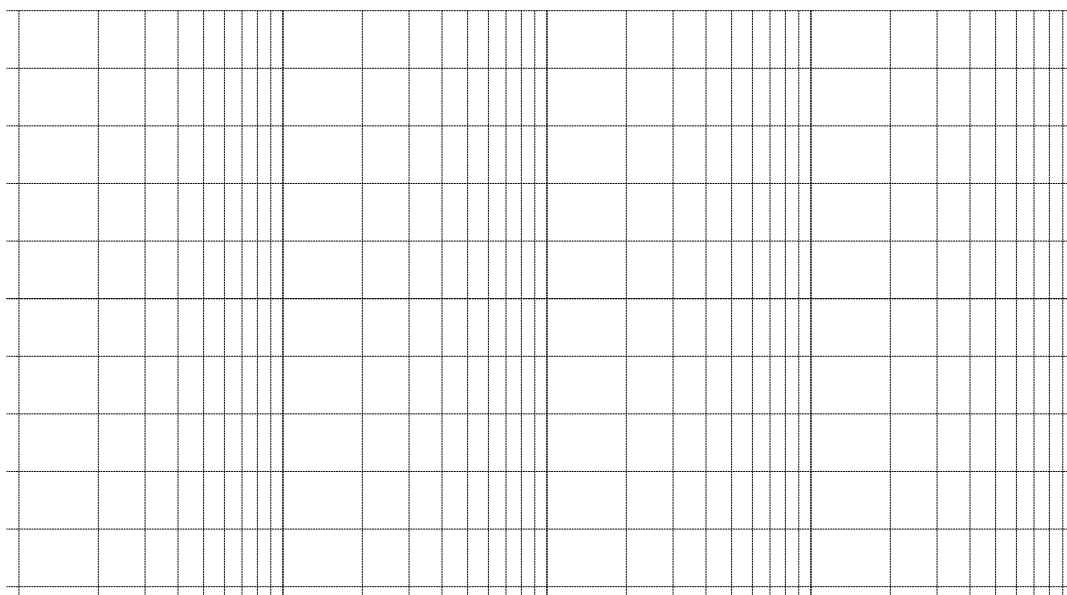
3 dB point

High Pass

input and output voltage _____

@ frequencies _____

% deviation from theoretical _____





Applications
of
Operational Amplifiers
Signal Conditioning Circuits
SCC

Objectives

- To study the linear equation $y = mx + b$, and linear circuits.
- Breadboard a Signal Conditioning circuit.
- Test the breadboard circuit.

See **Applications of Operational Amplifier Circuits Part A** for details on identifying operational amplifiers and breadboarding.

These applications will reinforce what we have learned about operational amplifiers, by building and verifying actual circuit operation.



Introduction

Non Inverting Signal Conditioning Circuit

The following linear circuit, is basically a non inverting amplifier with an input of $V1$. Instead of $R1$ being connected to ground it is connected to a second voltage $V2$.

The output of this circuit is:

$$V_{out} = \left(1 + \frac{R4}{R3}\right)V1 - \frac{R4}{R3}V2$$

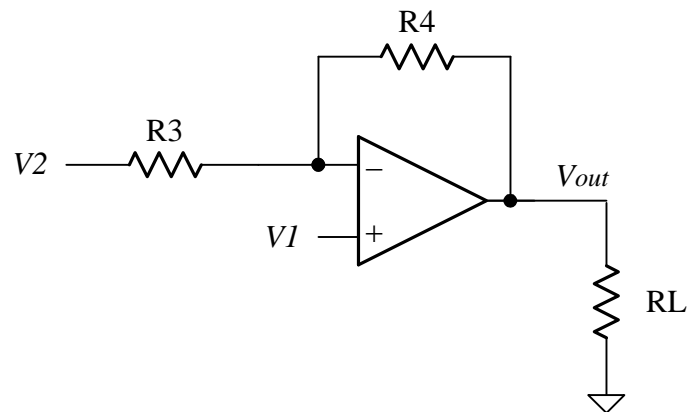
Which is again of the form:

$$y = mx + b$$

Note that the slope m is analogous to the voltage gain of $V1$, or

$Gain = \frac{R4 + R3}{R3} = m$, and the y-axis intercept b correspond to a dc offset of

$$Offset = -\frac{R4}{R3}V2 = b \text{ when } V2 \text{ is constant.}$$





For a noninverting gain of 3, we make

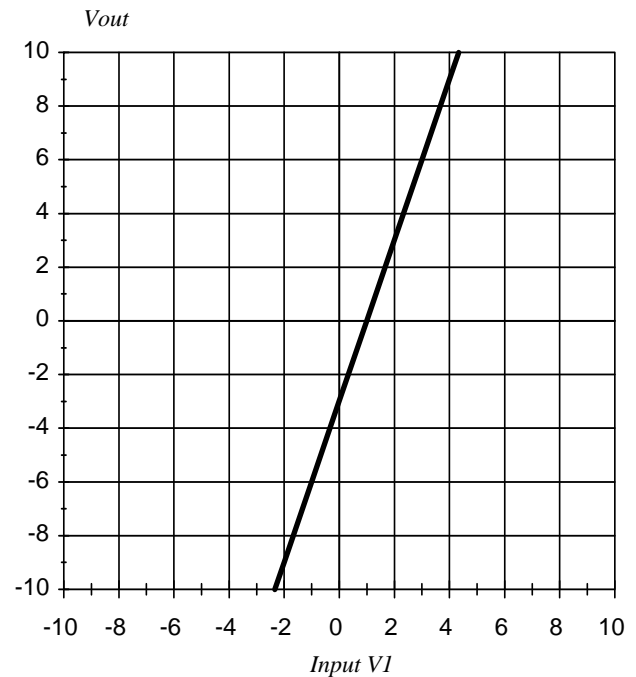
$$m = 3 = \frac{R4 + R3}{R3} = \frac{20k + 10k}{10k} \text{ and}$$

a dc offset of 3 Vdc, we make

$$b = -3 = \frac{R4}{R3} V2 = \frac{20k}{10k} (1.5V)$$

The graph on the right shows the final result.

$$V_{out} = 3 \times V1 - 3$$





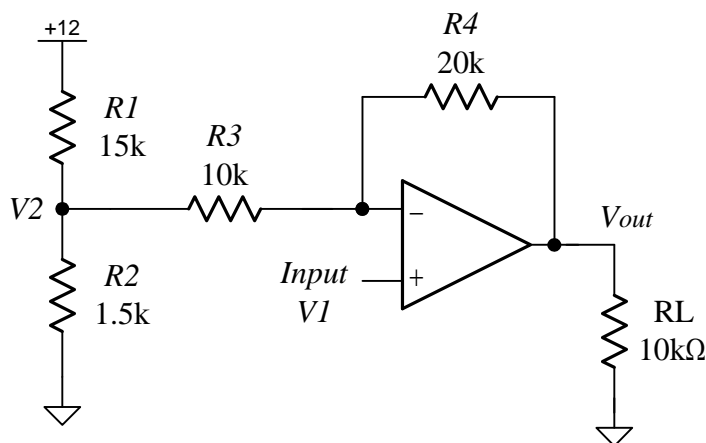
Non Inverting Signal Conditioning

In this part you will assemble and test a non inverting amplifier, with a dc offset.

The transfer function is given as:

$$V_{out} = \left(1 + \frac{R4}{R3}\right)V1 - \frac{R4}{R3}V2$$

V2 is the voltage developed by the resistor divider $R1$ and $R2$



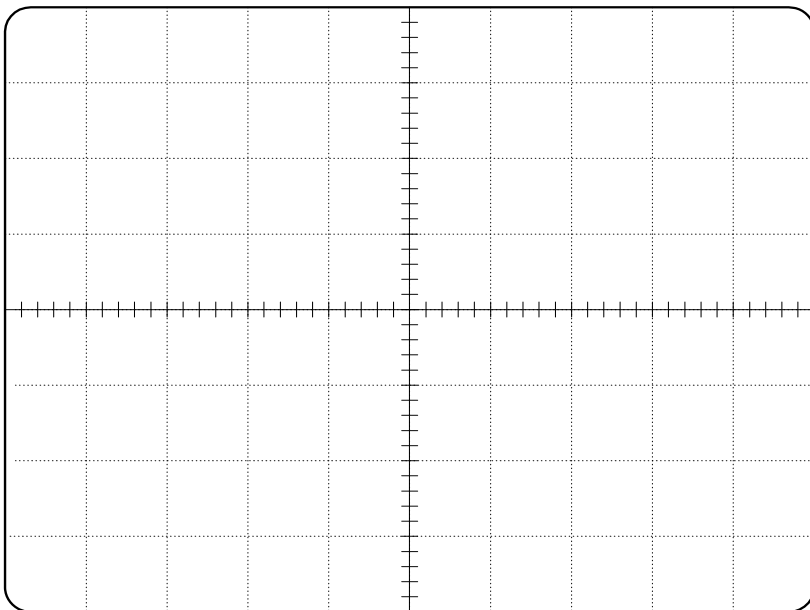
Procedure

1. Draw a detailed schematic diagram before assembling the circuit.
Drawing an assembly diagram will also help in your construction
2. Assemble the circuit per your diagrams.
3. Apply ± 12 V to the circuits (first adjust the power supply voltages before connecting to your circuit).
4. Using your Digital Millimeter measure the DC voltage V2 and verify that it is correct and record. All measurements are referenced to ground.
5. Connect the function generator to the *Input V1* of the circuit, and set the function generator (using the oscilloscope) to supply the circuit with an input signal of 1 kHz, 5 to VP-p saw tooth wave (\wedge). Make sure the signal is zero referenced, no DC offset.
6. Using the oscilloscope, measure the peak-to-peak output voltage, and record. Use both channels if possible, and compare the relationship between input and output and record.
7. Plotting the x-y characteristic of the circuit.
set the oscilloscope in the x-y mode, and adjust to clearly see the x-y relationship of the input and output.
8. Remove the signal source and power, before disassembling/assembling the next circuit.
9. Using your measurement, verify that the circuit represents the linear equation $y = mx + b$ by determining m and b.
10. Draw a graph showing the input voltage Vs the output voltage, and explain.



Lab Results

Student's Name: _____ Date: _____



Note the horizontal and vertical gain settings of your scope, and identify the zero reference for both the x and y coordinates.

Document all your results in your log book



Lab Activity

Second Order Active Low-Pass Filter

Objectives

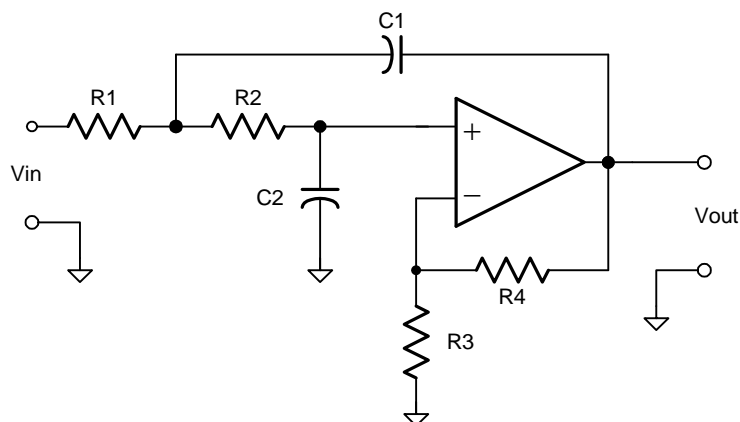
In this lab activity **each student** will assemble and test a second order active low-pass filter.

- Use a function generator and oscilloscope to determine the cutoff frequency and the frequency response of the second order low-pass filters.
- Calculate the components required to build one of the following Filter types:
 - Bessel
 - Butterworth
 - Chebyshev
- From the data obtained, construct a detailed and accurate Bode Plot, and compare your results to the theoretical values.



Introduction

The following circuit is one of the most popular topologies for a second order low-pass filter.



Different filter cutoff frequencies can be obtained by changing the values of $R1$, $R2$, $C1$, and $C2$.

If $R1 = R2 = R$, and $C1 = C2 = C$, the filter's cut-off frequency (3dB point) can be calculated by the simple equation:

$$f_o = \frac{1}{2\pi RC}$$

Different filter characteristics can be realized by changing the negative feedback gain of the filter ($R3$ and $R4$).

If we set

$$R4 = (K - 1)R_K$$

And $R3 = R_K$ then we can consider K to be the gain (DC) of the filter, since $Gain(DC) = 1 + \frac{R4}{R3} = 1 + \frac{(K - 1)R_K}{R_K} = K$.

Different values of K will give different filter responses or types as shown in the following table.

$K = 1$	$Rf = 0$ (short), and $Ri = \infty$ (open)
$K = 1.268$	Bessel
$K = 1.586$	Butterworth



$$K = 2.234 \quad \text{Chebyshev}$$

The damping factor ξ , which normally determines the filter response characteristics, is related to K as:

$$2\xi = 3 - K$$

This means that different filter types using this topology will have different DC gains. The 3dB point is then measured as *3dB below the filters DC gain*.

The filters roll-off slope is 40 dB/dec.

The following Bode Plot shows typical second order low-Pass filter with a cutoff frequency of 100Hz, and a dc gain of 1

The three different filter types that will be studied are,

Butterworth

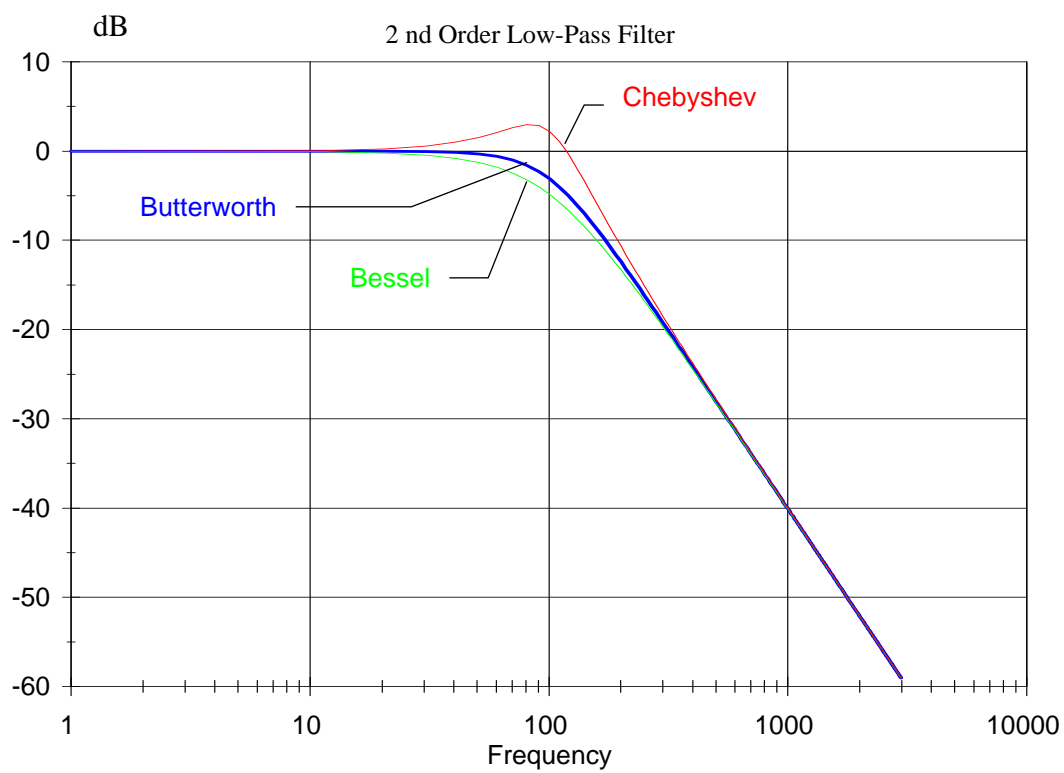
Provides a very flat response in the pass band. The phase response is not linear, and will cause overshoot when a pulse is applied to the filter.

Chebyshev (also spelled Tschebysheff or Tchebysheff)

Characterized by a rapid roll-off, but at the expense of a pass band ripple. The phase response is even more non linear, and will cause overshoot and oscillation when a pulse is applied to the filter.

Bessel

The Bessel response exhibits a linear phase characteristic, making it ideal for pulse waveforms since it produces almost no overshoot. This is at the expense of a poor roll-off in the stop band. The following graph shows the three different responses, with a normalized gain of 1.

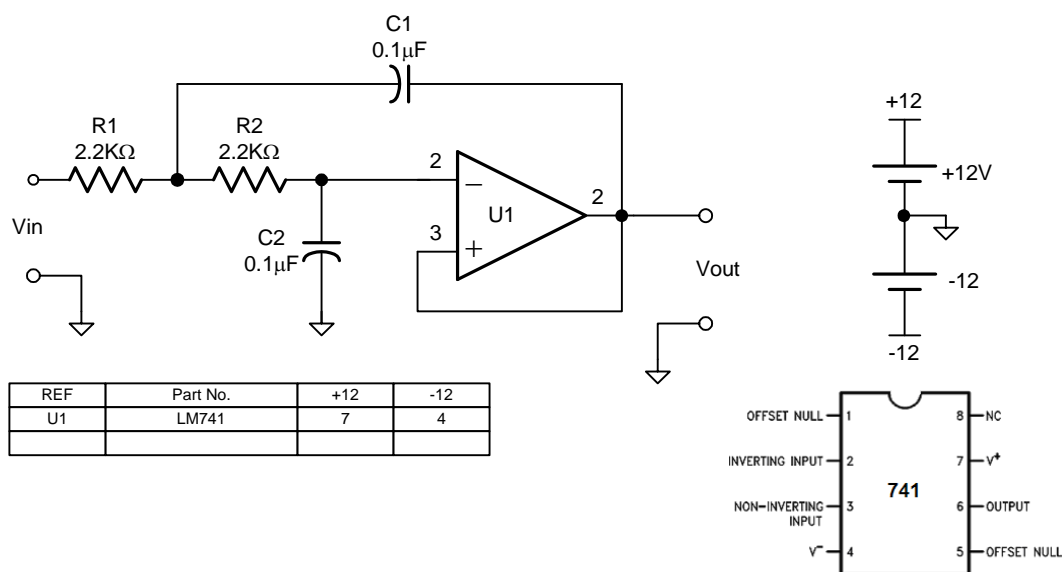




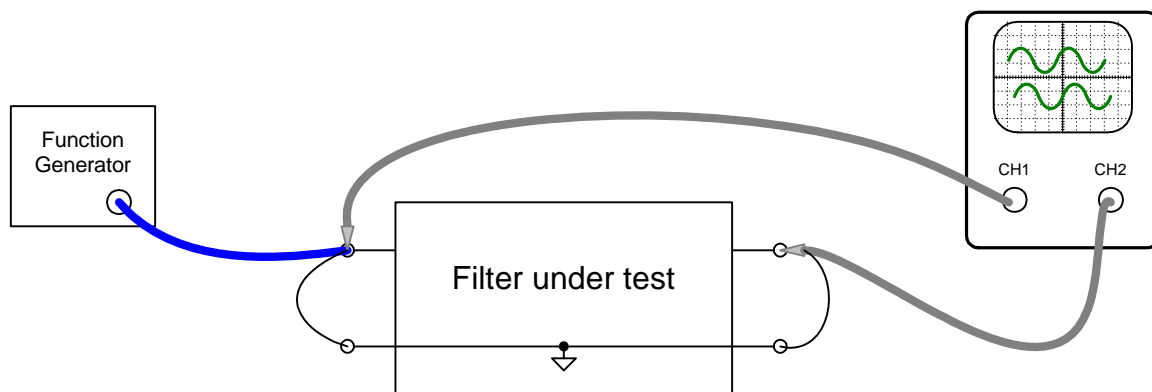
Preliminary

In this part of the experiment you will construct and experimentally analyze the following active lowpass filter. You will make voltage measurements for different frequencies at the filter's input and output and convert these measurements to decibels to construct a detailed Bode Plot.

Draw a complete schematic diagram of the filter with a gain of 1
($R_f = 0$, and $R_i = \infty$ not connected) in your logbook.



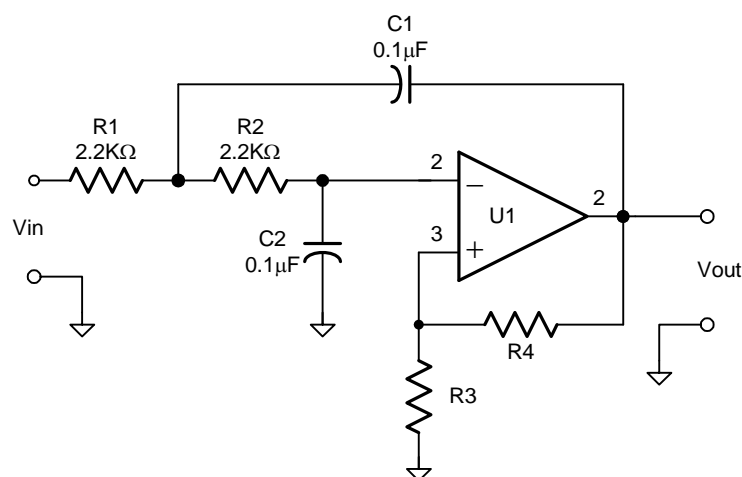
Test the circuit by connecting the function generator and the scope as required.





Part 1 *Butterworth Low-Pass filter*

1. Calculate appropriate values for R4 and R3, so that the filter is a Butterworth type $K = 1.586$. Choose standard values of resistors by using the ratio $R4/R3$. Calculate the theoretical cutoff frequency. Record all calculations.
2. Modify your circuit to include the gain resistors R3 and R4.



Frequency Response

1. Set the function generator to produce a sine wave with an amplitude of 2Vp-p at approximately 1kHz at the input. Before continuing, make sure to adjust the dc offset of the function generator to zero.
2. Slowly adjust the frequency of the function generator until the filter's output is at its -3 dB point (note that the filter has a gain), and record.
3. Slowly vary the frequency and measure the filter's input and output voltage at different frequencies, and plot the filter's response on a dB vs log frequency scale ie. Bode Plot.

Time Response

4. Readjust the function generator to produce a pulse (0 to 1V) at approximately 70Hz at the input.
5. Observe, comment, and record the output waveform

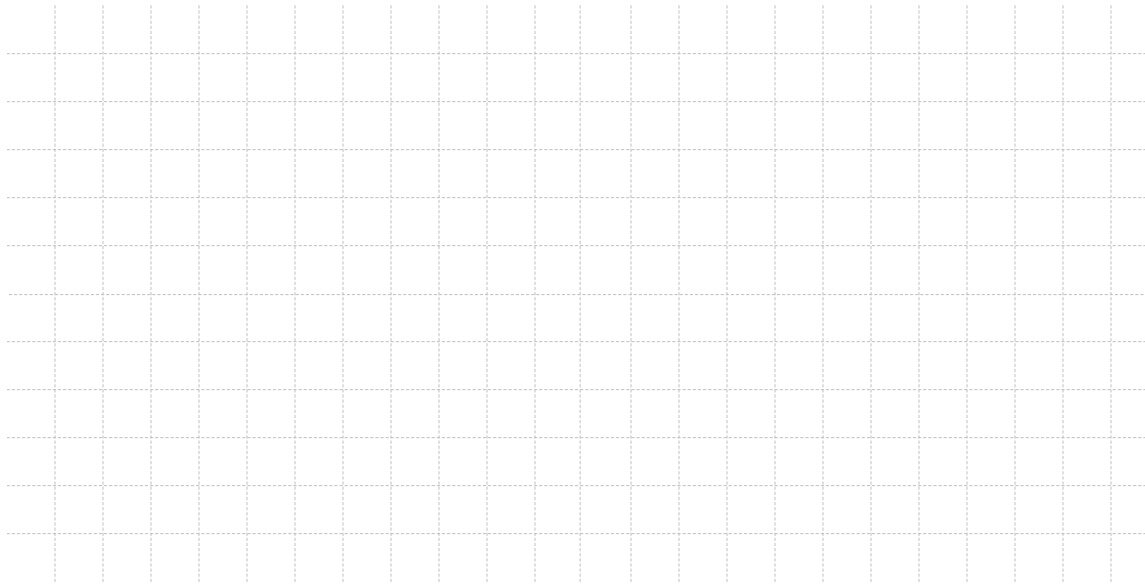


Frequency Response and Bode Plots

Lab Results Student's Name: _____ Date: _____

Part 1 Butterworth Low-Pass filter

Final Schematic Diagram



Calculated theoretical cutoff frequency: _____

Calculated R_f and R_i _____

Calculated DC gain _____

3 dB point input voltage _____

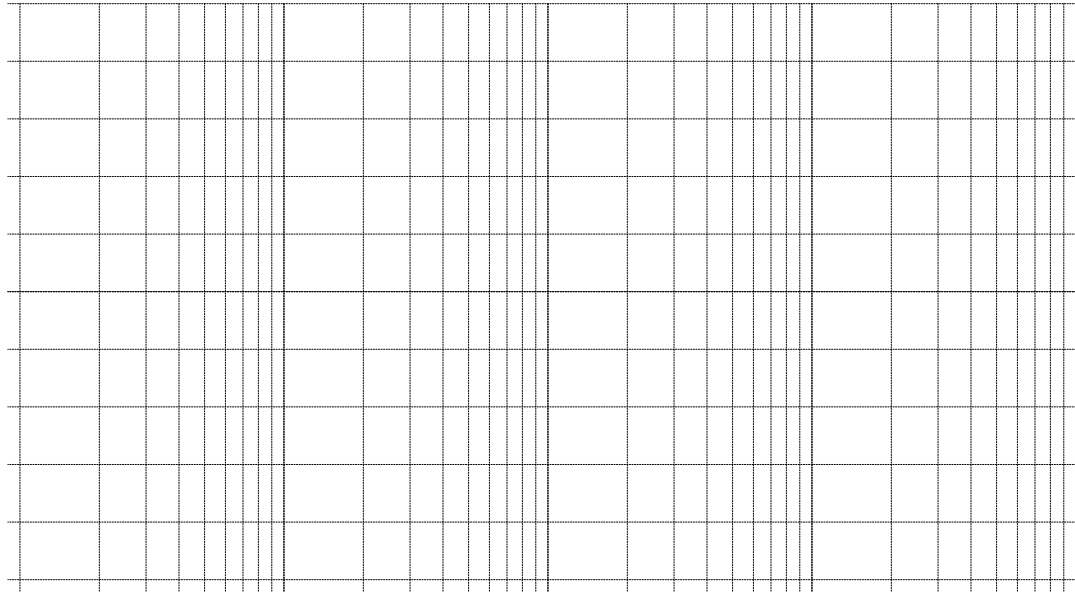
 output voltage _____

 @ frequency _____

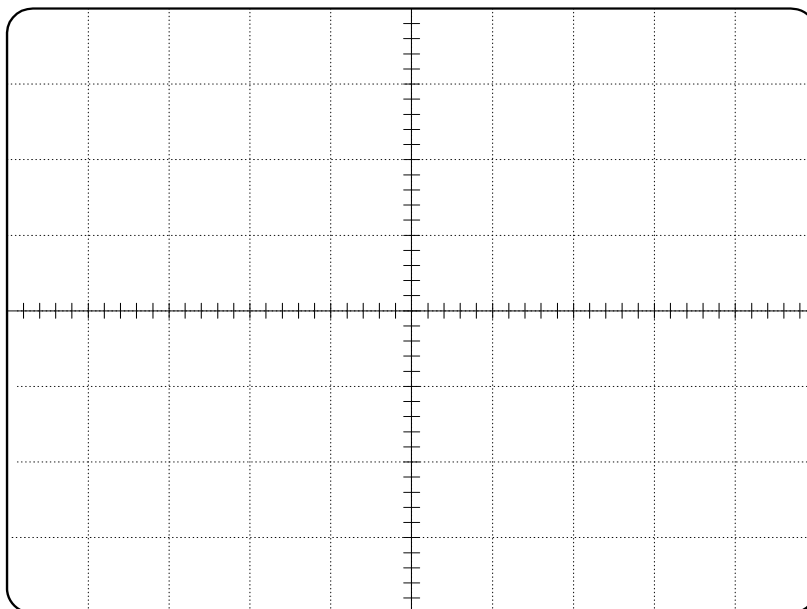
% deviation from theoretical _____



Bode Plot



Time Response



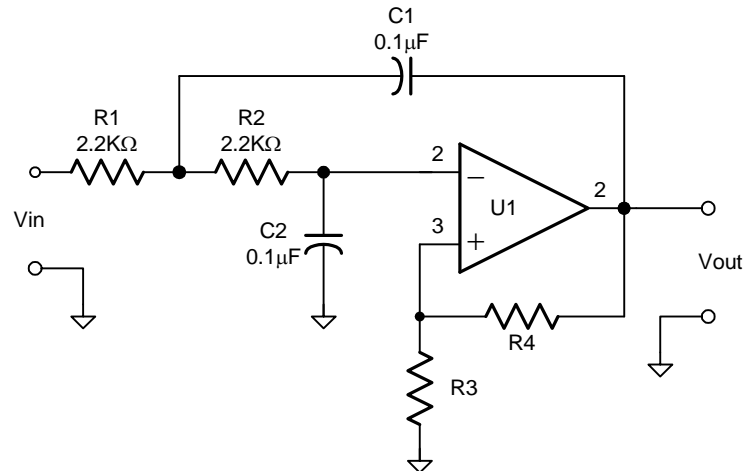
Note the horizontal and vertical gain settings of your scope, and identify the zero reference for each signal.



Part 2 Bessel

In this part of the experiment you will construct and experimentally analyze the Chebyshev active lowpass filter

Repeat part 1 using $K = 1.268$ Bessel



Part 2 Chebyshev

In this part of the experiment you will construct and experimentally analyze the Chebyshev active lowpass filter

Repeat part 1 using $K = 2.234$ Chebyshev

Prepare a detailed lab report comparing the filter types.



Applications of Operational Amplifier Circuits Comparators

Objectives

- To study the use of operational amplifiers in the use of comparator circuits.
- Breadboard and test
 - non-inverting comparator
 - inverting comparator
 - the schmitt trigger
- characterizing the schmitt trigger using the x-y oscilloscope function.

See **Applications of Operational Amplifier Circuits Part A** for details on identifying operational amplifiers and breadboarding.

These applications will reinforce what we have learned about operational amplifiers, by building and verifying actual circuit operation.



Introduction

Consider the following circuit.

Note that the circuit has no feedback connection, and is operating in what is referred to as open loop.

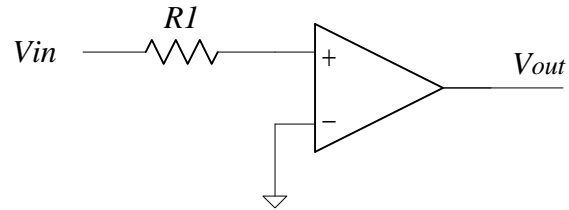
Recall that the op-amp, amplifies (with a very large gain) the differential voltage applied to its input terminals, or

$$V_{out} = A_{ol}(V^+ - V^-)$$

In the circuit shown, since V^- is connected to ground (zero volts), the equation reduces to:

$$V_{out} = A_{ol}V_{in}$$

This means that for $V_{in} > 0$, $V_{out} = +V_{sat}$
and $V_{in} < 0$, $V_{out} = -V_{sat}$



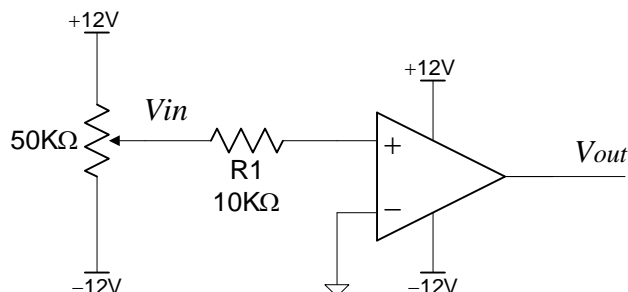


Non-inverting zero crossing detector

Assemble the following circuit using a 747 type op-amp connected to ± 12 volts.

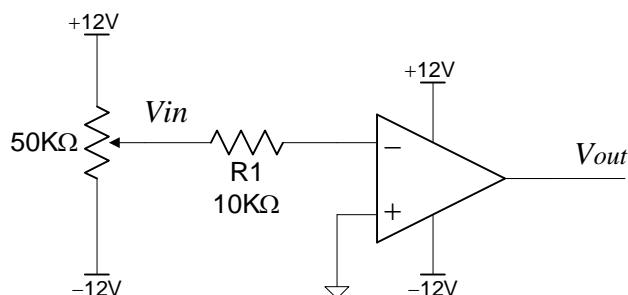
Connect V_{in} and V_{out} to the oscilloscope channel 1 and 2 respectively. Make sure that you have established a zero reference for each.

Slowly adjust V_{in} from $-2V$ to $+2V$, and observe the output voltage.



Inverting zero crossing detector

Modify the circuit as shown (interchanging the inputs), and repeat the above experiment.



Compare the input / output characteristics of both circuits.



The schmitt trigger circuit

Assemble the following circuit using a 747 type op-amp connected to ± 12 volts.

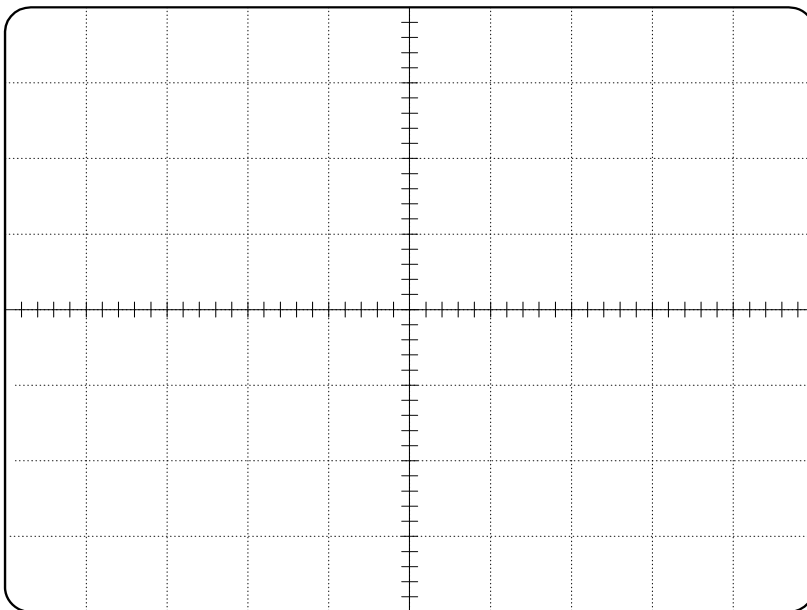
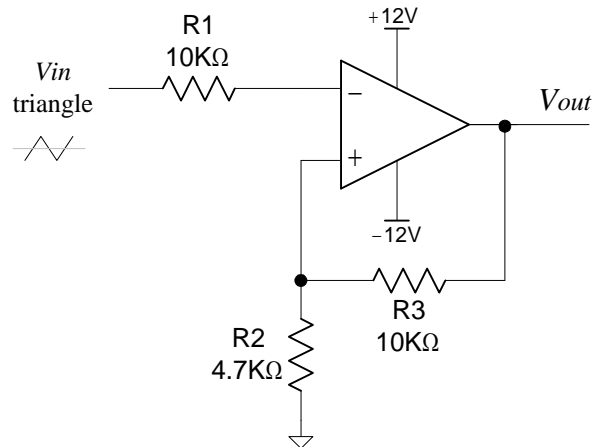
Connect V_{in} and V_{out} to the oscilloscope channel 1 and 2 respectively. Make sure that you have established a zero reference for each.

Adjust the function generator to make V_{in} a triangular waveform centered around zero with an amplitude of ± 1 volts, and a frequency of 100 Hz.

Note that V_{out} does not change. Explain why.

Increase V_{in} to ± 10 volts and note what happens. Explain why

Measure and record the input and output waveforms of the circuit, and draw the result accurately in the space provided.



Note the horizontal and vertical gain settings of your scope, and identify the zero reference for each signal.

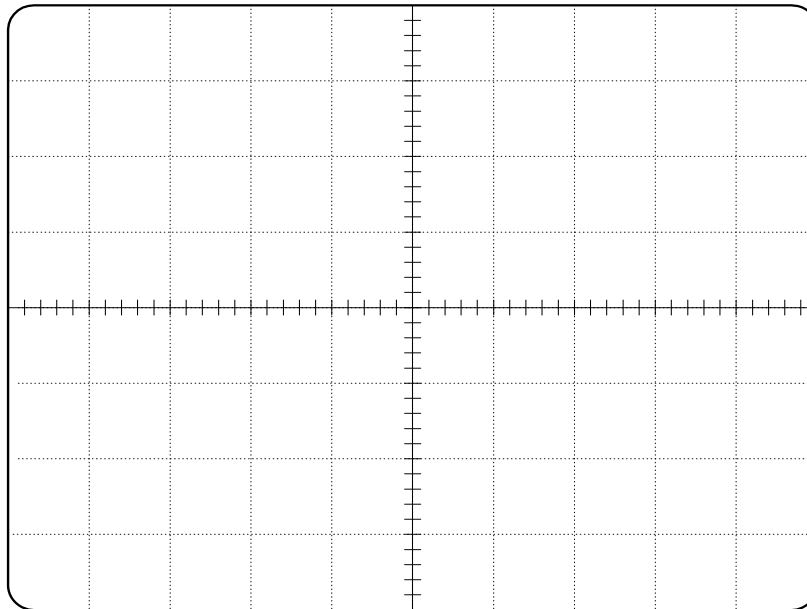


Comparator

Lab Results Student's Name: _____ Date: _____

Plotting the x-y characteristic of the schmitt trigger.

set the oscilloscope in the x-y mode, and adjust to clearly see the hysteresis of the schmitt trigger



Note the horizontal and vertical gain settings of your scope, and identify the zero reference for both the x and y coordinates



Applications of Operational Amplifier circuits **Integration**

Objectives

- The purpose of this activity is to demonstrate the response of a integrator to periodic waveforms.
- Breadboard and test a
Integrator
- Describe the basic integrator circuits
- Use the oscilloscope and function generator to test circuit operation

These applications will reinforce what we have learned about operational amplifiers, by building and verifying actual circuit operation.



Introduction

An active integrator circuit as shown. The circuit is an inverting configuration with a capacitor in the feedback. Resistor R produces a current proportional to its input voltage, and this current then charges the capacitor. The output voltage of the circuit is the voltage across the capacitor produced by the current from R.

The voltage across a capacitor can be expressed as:

$$V_C = \frac{1}{C} \int i \, dt, \text{ and}$$

the current $i = \frac{V_{in}}{R}$ and the output voltage $V_C = -V_{out}$

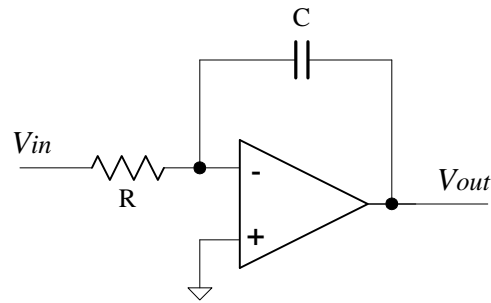
Substituting this in the above equation we get

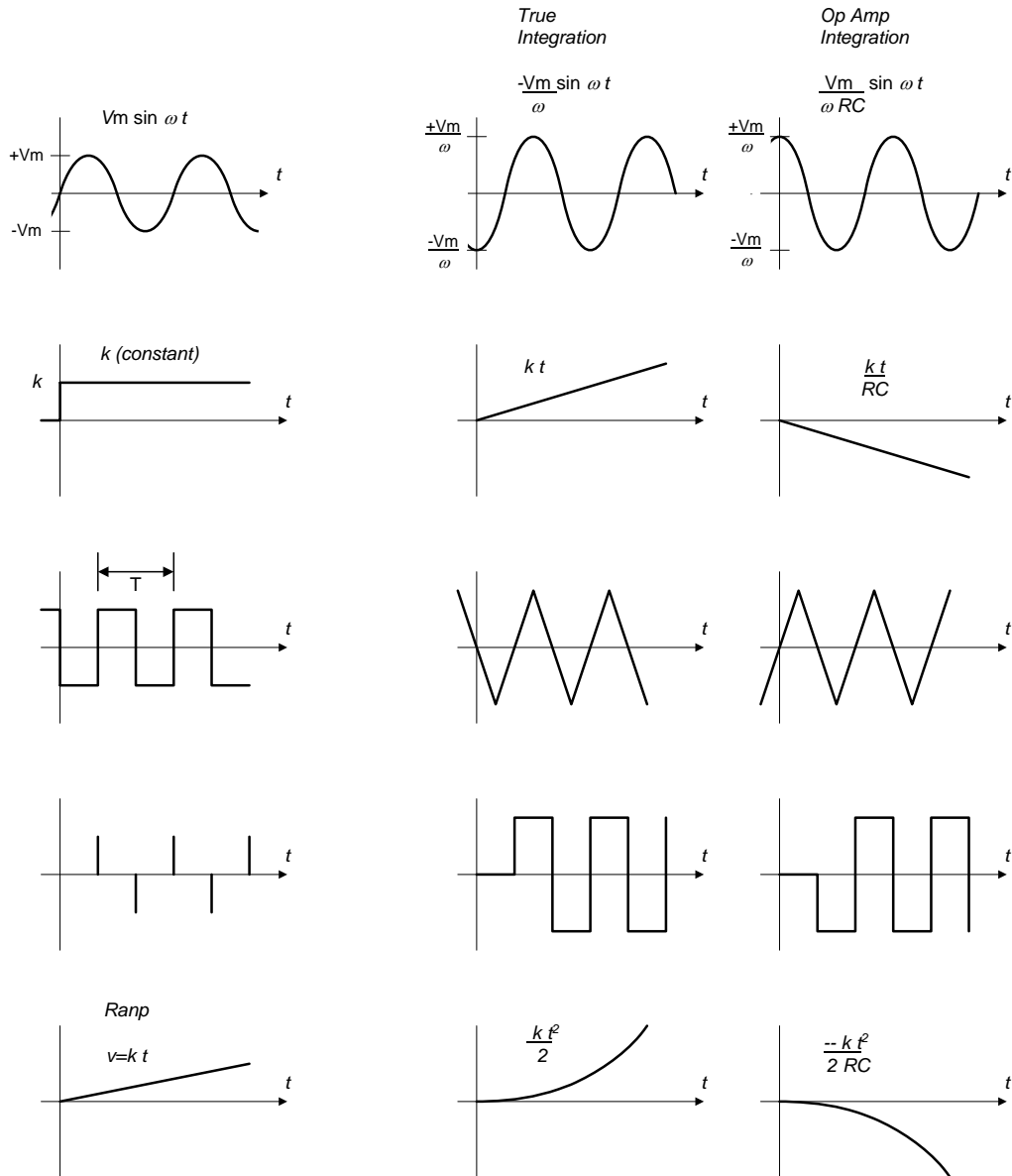
$$-V_{out} = \frac{1}{C} \int \frac{V_{in}}{R} \, dt \quad \text{or}$$

$$V_{out} = -\frac{1}{RC} \int V_{in} \, dt$$

This circuit has a output voltage proportional to the integral of the input voltage V_{in} , and with a gain of $-\frac{1}{RC}$.

The following demonstrates graphically different input signals, their integral, and the active integral based on the opamp circuit.

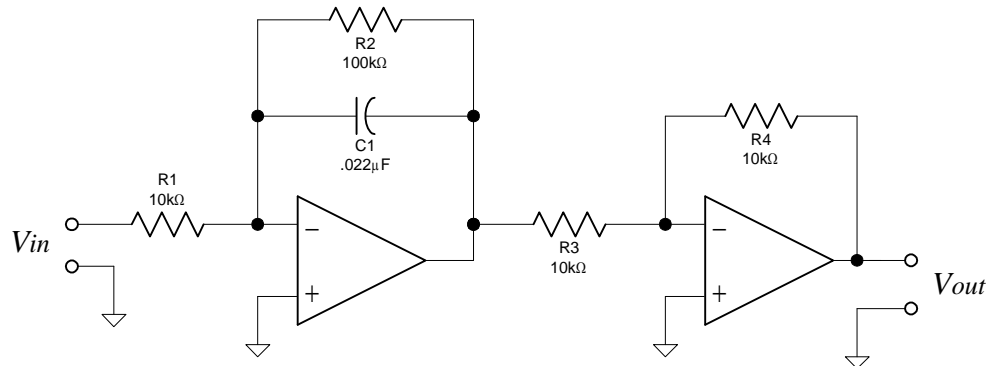






Procedure

1. Assemble the following circuit. The second op-amp is an inverting amplifier with a unity gain, making the output of the active integrator noninverting.



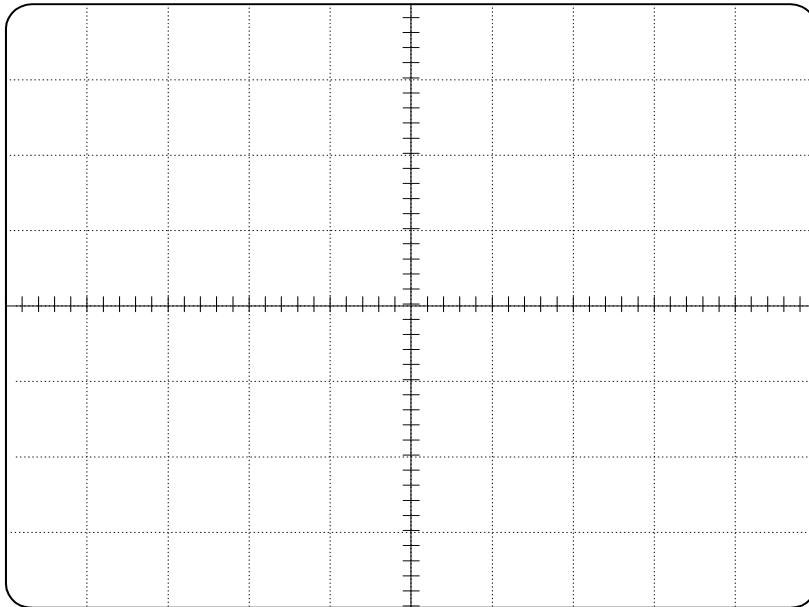
2. Use Channel 1 of your oscilloscope to monitor the input of the circuit and Channel 2 to monitor the output. The input mode select switch for each channel should be set in the DC position after using the GND position to establish a 0V reference line for the upper and lower traces. Use the center horizontal line on the screen for the 0 V reference line for the Channel 1 upper trace and the bottom horizontal line for the Channel 2 reference line. Set the volts/div control for Channel 1 to 0.5 V/div, and the volts/div control for Channel 2 to 2 V/div. The time/div control should be set at 0.5 ms/div. Trigger from Channel 1 in the Auto mode.
3. Use your function generator to apply a 0 to 1 V_{peak}, 1 kHz pulsating DC square wave to the input. Make sure the square wave starts at exactly 0V and rises to +1 V for each pulse. You need to adjust the DC offset control and the amplitude control several times to produce this square wave. Also, the square wave should have a 50% duty cycle.
4. Accurately draw the input and output signals of the screen in your log book.
5. Increase the frequency to 4kHz, and describe its effect on the output.
6. Lower the frequency of the pulsating DC until the DC signal displayed on Channel 2 becomes rippled (noticeable triangle waveform). This should be a very low frequency.
7. Using the integrator circuit, verify and experiment with different signals as shown in the introduction. Document each waveform, and comment on amplitude, and shape.
8. Answer all the following questions.



Integrator

Lab Results Student's Name: _____ Date: _____

1. Draw the input and output signals of step 4 in the space provided. Identify both the horizontal and vertical axis.



2. For the circuit assembled, what does the final output voltage represent ?



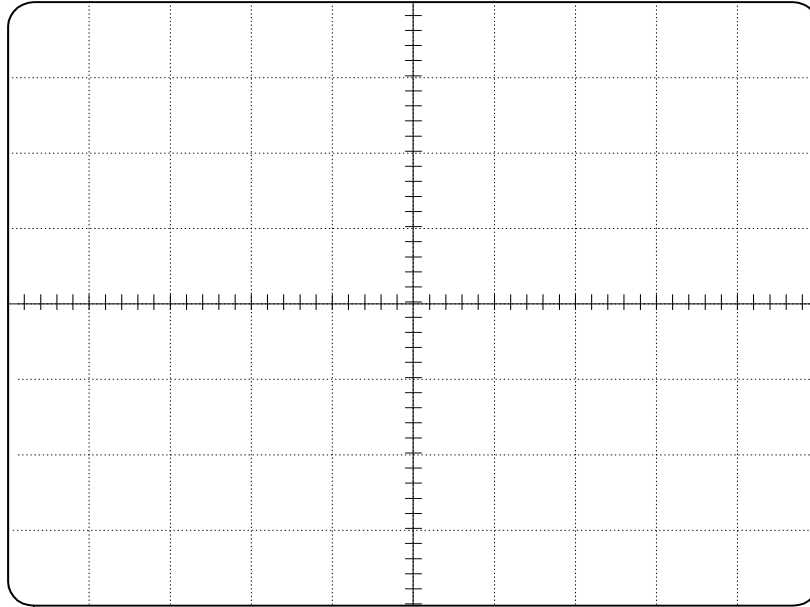


VANIER
CÉGEP/COLLEGE
My choice. My future.

Electrical Engineering Technologies

Industrial Electronics

Instrumentation & Automation





Waveform Generation

Objectives

- To be able to connect a integrated circuits from information provided by the manufacturer's data sheet.
- To investigate the effect of ac and dc coupling waveforms.
- To mix appropriate sinusoidal waveforms to produce complex waveforms.

Reference Reading

ICL8038 or XR-8038 datasheet



Introduction

The ICL8038 integrated circuit, has the capability of generating sinusoidal, triangular, and square waveforms. It forms the heart of some commercial signal generators and is a popular device for experimenters. The block diagram shows its simplified function.

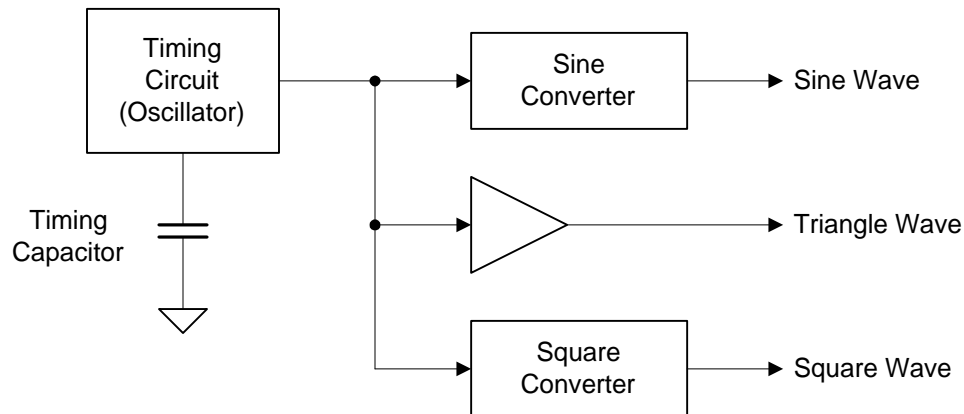


Figure 1. Block Diagram

No attempt will be made in this experiment to understand the internal operation of this integrated circuit. However students are encouraged to study its data sheet (reflib on the network), to fully understand its function. It is easy to use, however, by following and understanding the manufacturer's suggested configurations, and application information. The manufacturer's data specifies various ranges for particular components. The ranges picked for this experiment fall within the specified ranges, and will produce waveforms with a 50% duty cycles. The student will observe that some of the waveforms are less than perfect. The reason for this is that the circuit configuration is a simple one. More external design is needed to obtain optimum performance from the integrated circuit as explained in the data sheet. All details are provided in the data sheet, and we will apply some of these techniques later when required.

The three outputs of this chip each have a DC offset voltage along with the AC waveform component. This will allow the study of the effect of using a capacitor to couple a signal. It will be seen that the capacitor will remove the DC level, in the same way that a scope's AC coupling does.

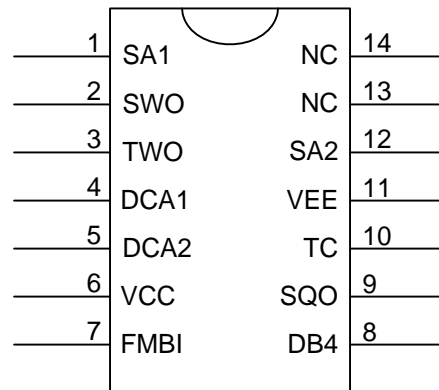


Figure 2. Pin Out

Before we apply the ICL8038 we will identify each pin, to fully understand its functional description in relation to its functional diagram. (also see Figure 2.)

<i>Pin Number(s)</i>	<i>Symbol</i>	<i>Description</i>
1	SA1	Sine Waveform Adjust Input 1. This pin is used together with pin 12 to adjust the harmonic distortion of the sine output.
2	SWO	Sine Wave Output
3	TWO	Triangle Wave Output
4	DCA1	Duty Cycle Adjustment Input 1. Duty cycle can be adjusted using this pin in combination with pin 5.
5	DCA2	Duty Cycle Adjustment Input 2
6	VCC	Positive Power Supply
7	FMBI	Frequency Modulation Input.
8	FMSI	Frequency Sweep Input.
9	SQO	Square Wave Output.
10	TC	Timing Capacitor Input.
11	VEE	Negative Power Supply.
12	SA2	Sine Waveform Adjust Input 2.
13	NC	No Connect.
14	NC	No Connect.



Procedure

1. Connect the test circuit as recommended in the manufacturer's data sheet. Do not turn on power until all of the circuit is connected.

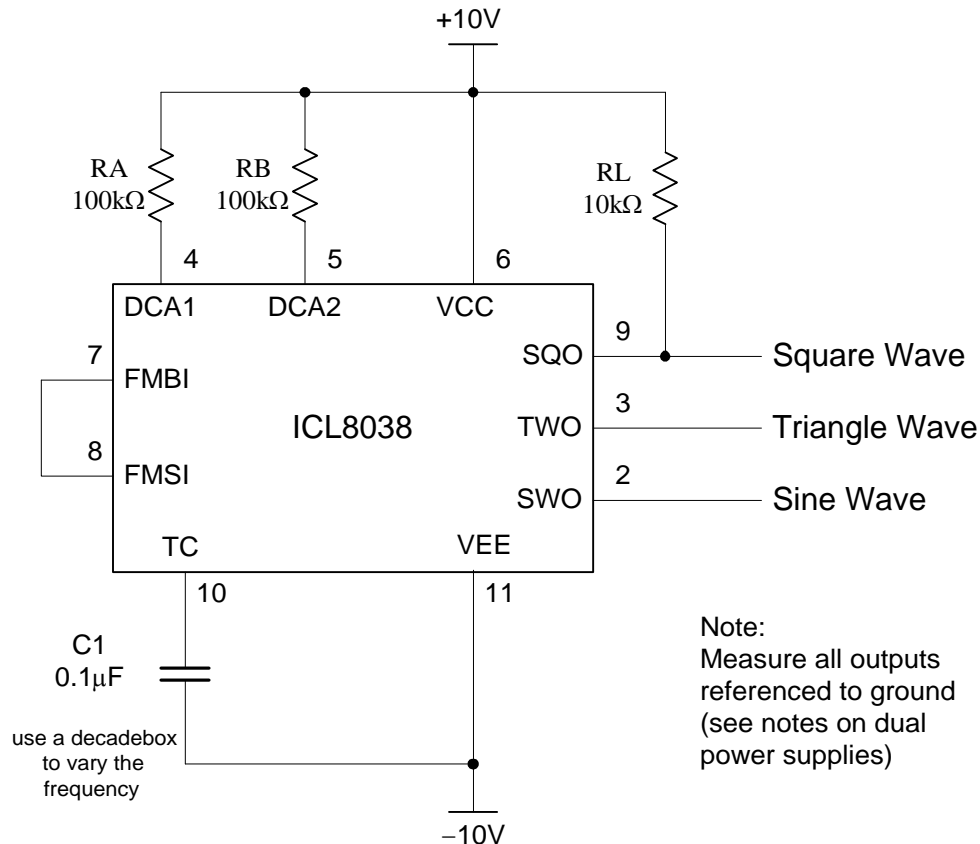


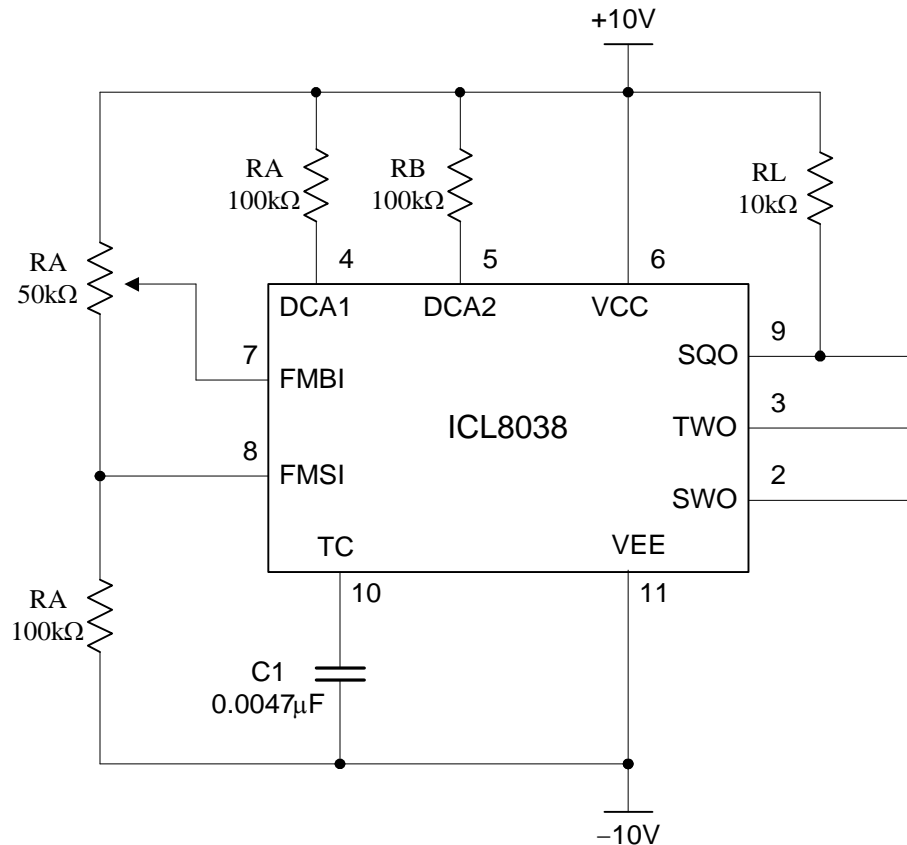
Figure 3. Test Circuit

1. Use the oscilloscope to measure the waveform at each specified output of the integrated circuit, and record. (frequency, shape, and amplitude; draw an accurate picture,)
2. Use a DC voltmeter to measure the DC and rms voltage levels of each waveform and record.

Change C1 repeat steps 1. Determine how the frequency changes as the value of the capacitor changes, Always turn off the power when making changes to the circuit.



3. Add a $0.1\mu\text{F}$ capacitor in series with the output of pin number 2. Observe the waveform on the scope with the input set for dc coupling. Measure on both sides of the capacitor in order to see the effect it has on the output waveform.
4. Modify the circuit as explained in the data sheet so that the frequency can be varied (modulated) using a potentiometer as explained in the data sheet. Use a $50\text{k}\Omega$ potentiometer and a $100\text{k}\Omega$ resistor. Explain why.





Lab Activity

Common Mode Rejection

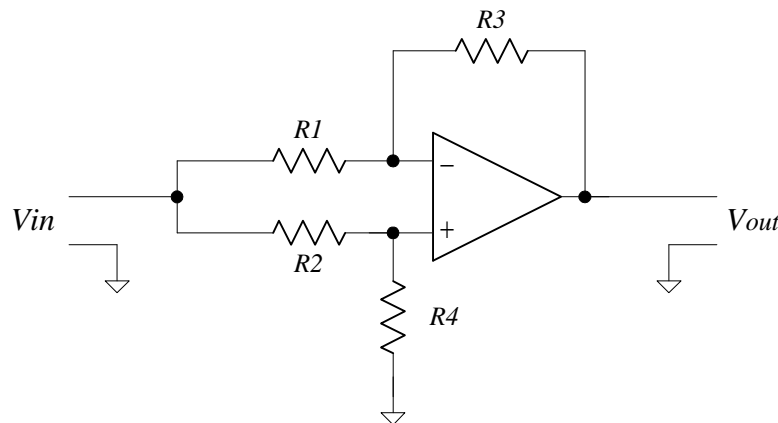
Objectives

- This lab activity is designed to measure the common node rejection ratio (CMRR) of a differential operational amplifier circuit.
- Express the CMRR in terms of dB.



Introduction

If the same signal is applied to both inputs of a differential amplifier, the input is called common-mode input, and then the output voltage of the differential amplifier should be zero. Since operational amplifiers and circuit components are not ideal, a small but finite output voltage will be produced when both inputs are the same. The ratio of the common-mode input voltage to the produced output voltage is termed the common-mode rejection ratio, or **CMRR**, and is normally expressed in decibels. The higher the **CMRR**, the better the rejection, resulting in a smaller output voltage. The following circuit will be used. Note that both inputs of the differential amplifier are connected together.



the differential gain is $A_d = \frac{R_3}{R_1} = \frac{R_4}{R_2}$ if $R_1=R_2$, and $R_3=R_4$

the common mode gain is $A_{cm} = \frac{V_{out}}{V_{in}}$

then $CMRR = \frac{A_d}{A_{cm}}$

expressed in dB $CMRR = 20 \times \log \left(\frac{A_d}{A_{cm}} \right)$ (in deciBel)

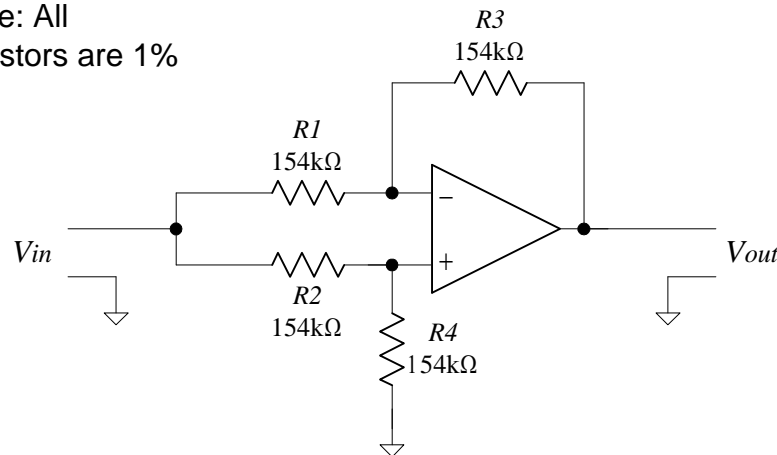
note gain is expressed in voltage



PROCEDURE

Assemble the following circuit using a 741 type op-amp, making sure all connections are short and close together.

Note: All resistors are 1%



Apply $\pm 12\text{V}$ power to the circuit.

Connect the input to the function generator and adjust to 1.0 V_{p-p} sine wave (no offset) at a frequency of 60 Hz using channel 1 of the oscilloscope. Also, concurrently, measure its RMS voltage using the DMM.

Measure both the RMS and peak-to-peak output voltage using channel 2, and record all results.

From your measurements, calculate the CMRR of the circuit.

Calculate the CMRR of the 741 operational amplifier based on the actual (**measured**) values of the resistors.

Compare your results with those published by the manufacturer of the 741 operational amplifier.

Repeat your measurements and calculations using a 324 type op-amp.



Applications of **Operational Amplifier Circuits with Diodes**

Objectives

- To study the following circuits
 - half wave rectifier
 - peak detector
 - full wave rectifier (absolute value)
 - signal limiter
- Breadboard and test the circuit.

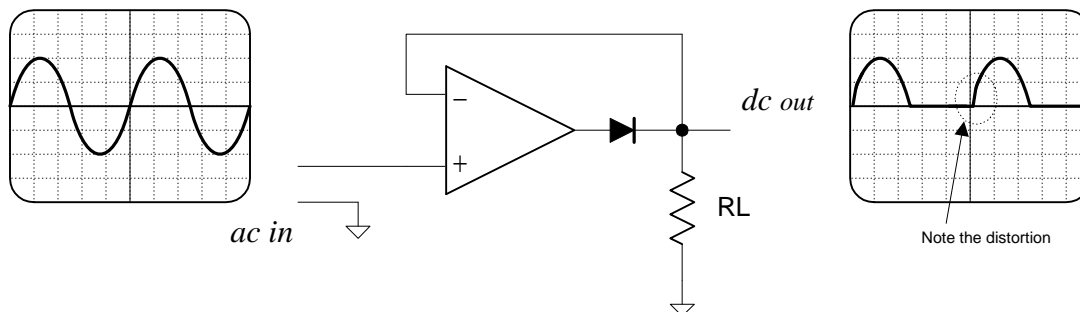
These applications will reinforce what we have learned about operational amplifiers, by building and verifying actual circuit operation.



Introduction

Half wave rectifier

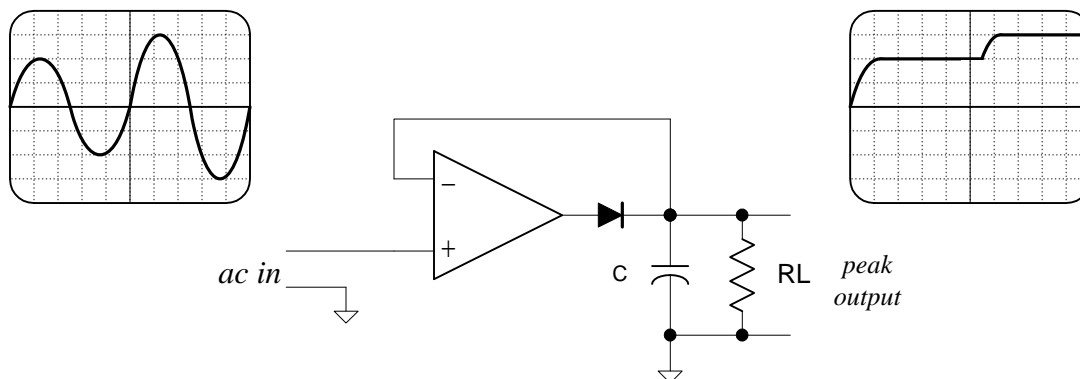
The following circuit is a positive precision half-wave rectifier. This circuit will be able to rectify signals that are smaller than a diode voltage drop (diode barrier potential). The operational amplifier enables the diode to behave like an ideal diode.



Notice that the output is taken from the input of the inverting amplifier. During the positive cycle, the amplifier is connected as a voltage follower with the diode forward biased. During the negative cycle, the amplifier will go into negative saturation, and the diode will be reverse biased, and the inverting input will remain at zero potential. Resistor R_L makes sure that the diode is forward biased. When the op-amp needs to return from negative saturation, it will require some time to recover. This causes the distortion as shown. We will see later how to avoid this

Peak detector

The same circuit, with the addition of a capacitor makes a peak detector.



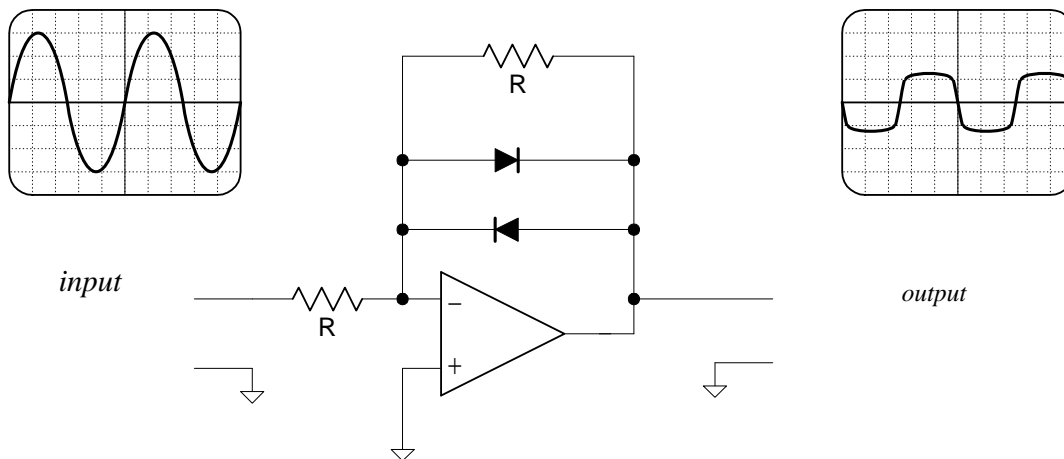
The capacitor during the positive cycle will charge until the output voltage equals the input voltage as long as the input voltage is greater than the voltage across the capacitor. At this point, the op-amp (as a comparator) switches, and the op-amp's output goes to



negative saturation. The diode is now reverse biased, and the capacitor will hold its charge. If later a *larger* input voltage occurs, the capacitor will charge to its *new* peak voltage.

Signal Limiter

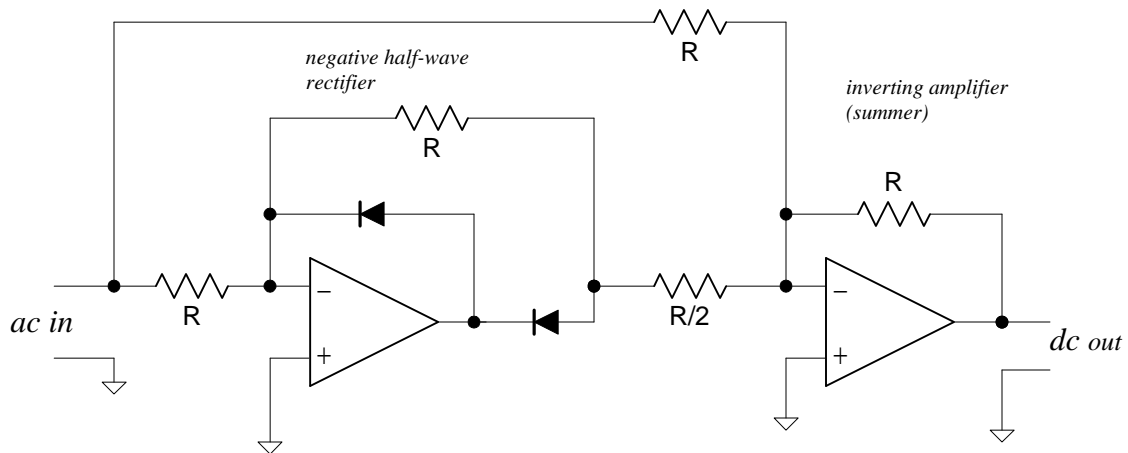
Diodes in the feedback are often used to limit a signal's amplitude. As long as the input signal is below approximately 1V_{pp} ($\pm 0.5\text{V}$) the signal will be inverted. If the signal exceeds the diode barrier potential, the signal will be amplitude limited.



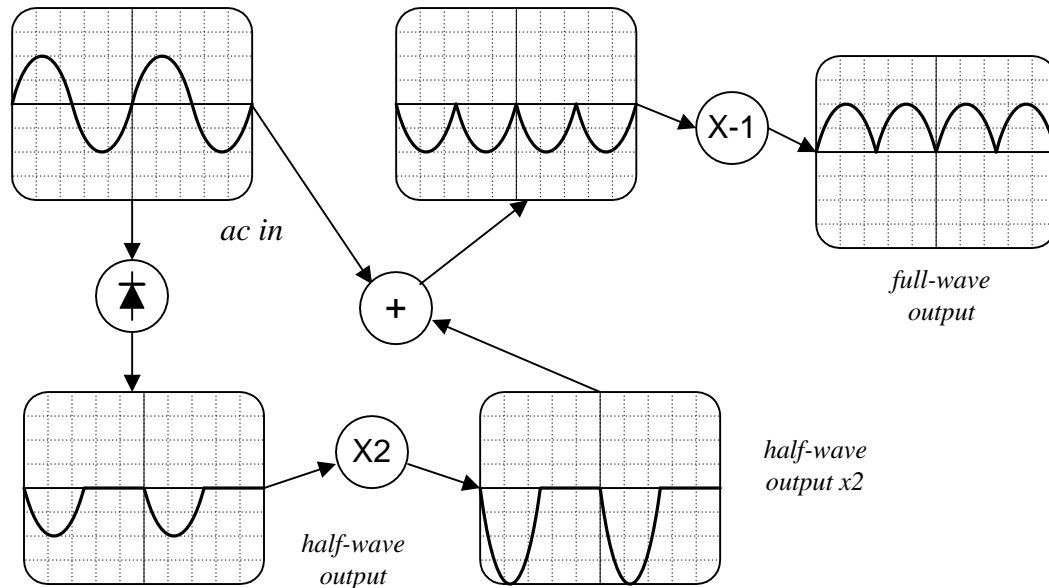


Full wave rectifier

The following circuit is a full-wave precision rectifier..



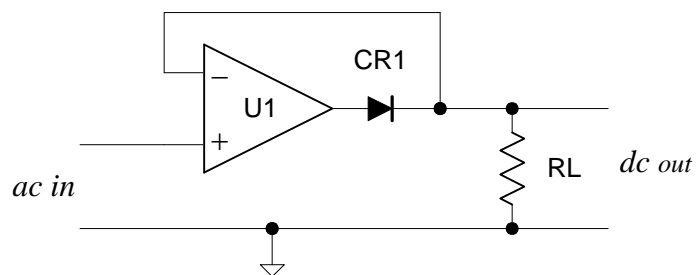
Looking at the circuit we see that the first part consists of a negative half-wave rectifier, and the second part is a summer. The summer adds the original signal plus twice the inverted half-wave rectified signal producing the full wave-rectification as shown.



Part 1 Half wave rectifier

Assemble the following circuit.

U1 747/324
CR1 1N914
RL 1k to 10k
Power +/- 12V

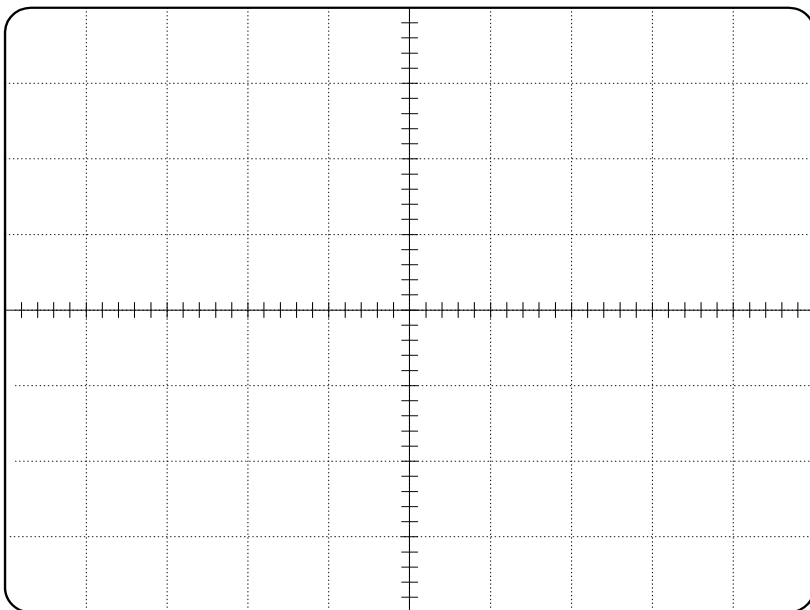


Using an appropriate input, verify that the circuit is functional. Document your results in the space provided.



VANIER
CÉGEP/COLLEGE
My choice. My future.

Electrical Engineering Technologies
Industrial Electronics
Instrumentation & Automation



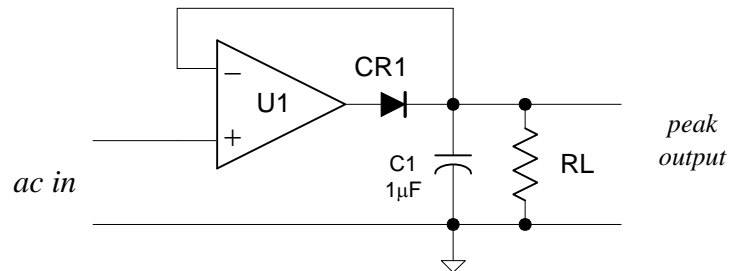
Record the horizontal and vertical gain settings of your scope, and identify the zero reference for each signal



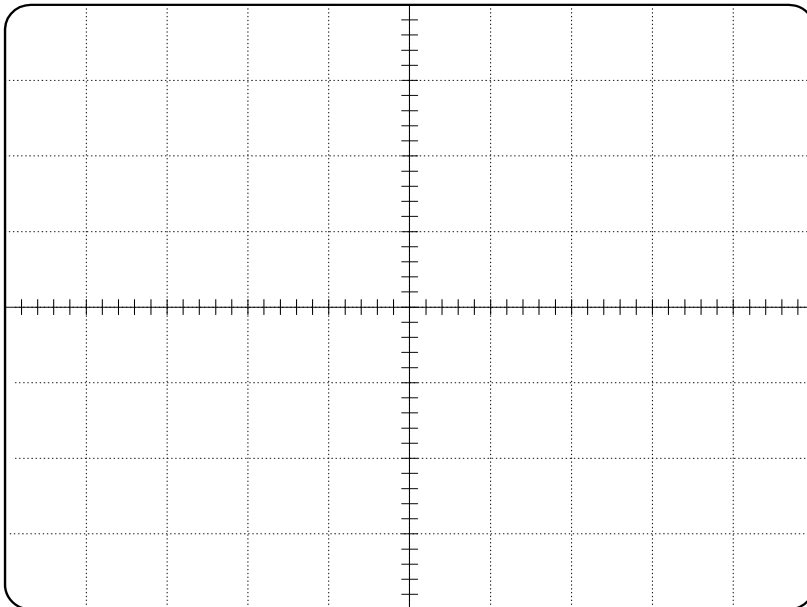
Part 2 Peak detector

Assemble the following circuit

U1 747/324
CR1 1N914
RL 1k to 10k
Power +/- 12V



Using an appropriate input, verify that the circuit is functional. Document your results in the space provided.

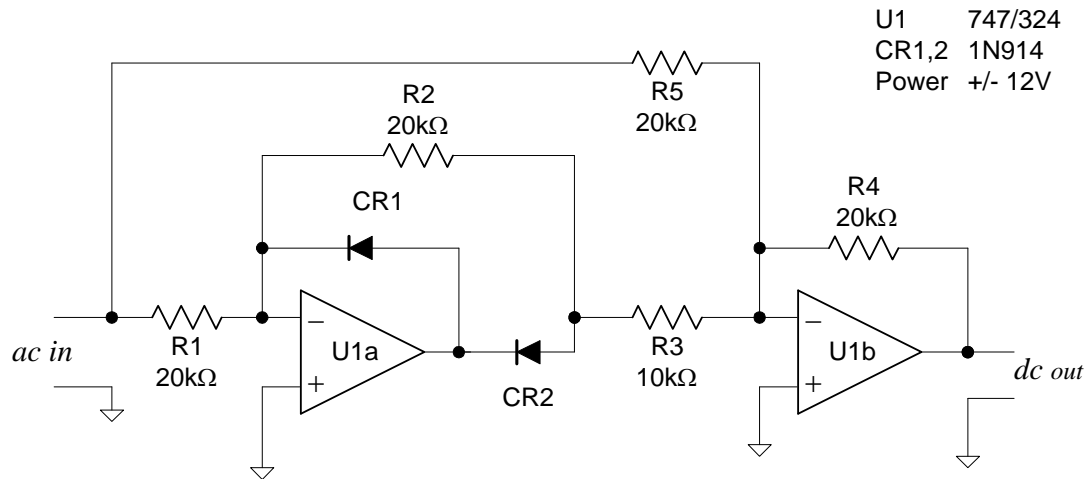


Record the horizontal and vertical gain settings of your scope, and identify the zero reference for each signal

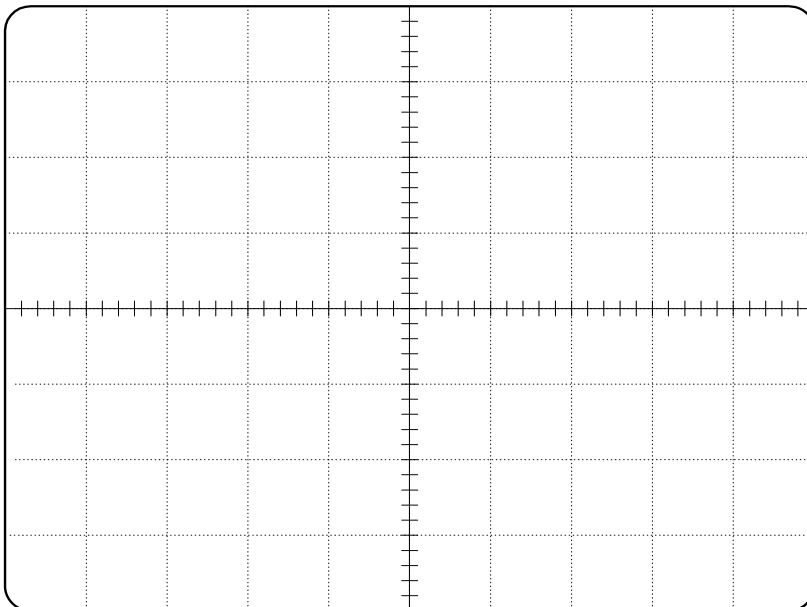


Part 3 Full wave rectifier

Assemble the following circuit



Using an appropriate input, verify that the circuit is functional. Document your results in the space provided.



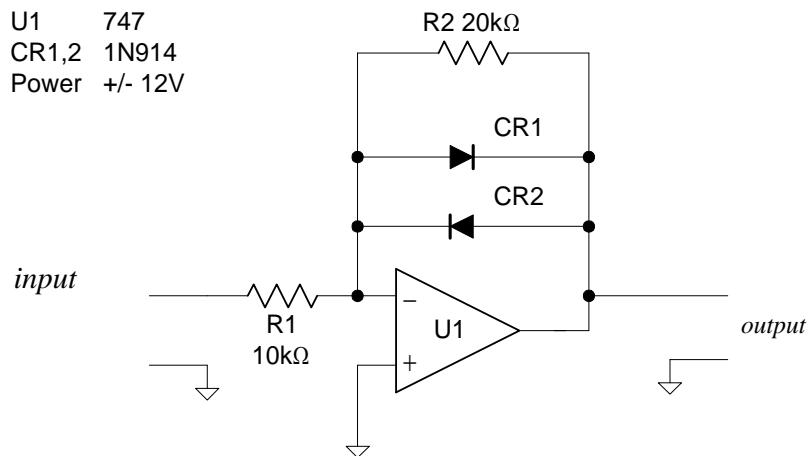
Record the horizontal and vertical gain settings of your scope, and identify the zero reference for each signal



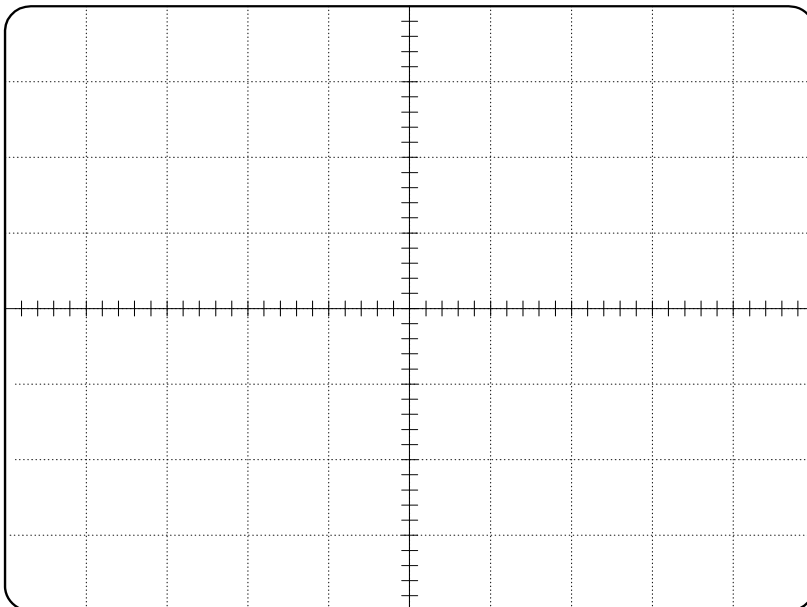
Part 4 Signal Limiter

Assemble the following circuit

U1 747
CR1,2 1N914
Power +/- 12V



Using an appropriate input, verify that the circuit is functional. Document your results in the space provided.



Record the horizontal and vertical gain settings of your scope, and identify the zero reference for each signal



Sample and Hold

Objectives

The purpose of this activity is to assemble, troubleshoot, and demonstrate the operation of a Sample and Hold Circuit used in Data Acquisition and Control industry.

Specifically:

- To analyze the operation of a sample and hold circuit.
- To evaluate the relationship between the incoming analog frequency and the sampling frequency.
- To reconstruct an analog waveform from a sampled waveform.
- To observe the effect of aliasing distortion and its causes.

This experiments will reinforce what we have learned about Data Acquisition and the need for a Sample and Hold Circuit.



Materials Required

- Resistor: 10K Ω
- Capacitors: 47nF (2), 100 nF, 10 μ F (2)
- N-Channel Enhancement MOSFET VN10KM
- LM324 Op amp
- Oscilloscope



Introduction

The purpose of the circuit of Figure 1 is to allow an incoming analog signal to be sampled so that the discrete voltage sample levels may be converted to digital signals. To do this, the circuit must follow or track the analog signal and periodically hold the sample level long enough for the level to be converted to a digital value by a digital-to-analog (D/A) converter. In the circuit of Figure 1 the MOSFET acts as a simple switch, allowing the signal from the first op amp to appear across the 47nF hold capacitor whenever the MOSFET is on. The sampling frequency is usually a square wave applied to the gate, turning the MOSFET on whenever the gate voltage is positive. When the MOSFET is off, the 47nF capacitor holds its voltage level since it cannot discharge through the high impedance of the MOSFET or the high input impedance of the second op amp. The voltage level may now be converted to a digital word reflecting the voltage value. When the MOSFET switches on again, the 47nF capacitor quickly tracks back to the voltage of the analog signal. The analog signal may be most accurately reconstructed if many samples are taken within one analog cycle, consequently the sampling frequency is best when much higher than the analog frequency. At worst, the sampling frequency must be at least twice the analog frequency or aliasing distortion will occur, creating unwanted lower frequencies. Complete A/D conversion systems will usually include anti-aliasing filters to remove analog frequencies higher than half of the sampling frequency in order to prevent aliasing.



Procedure

Observing Sample and Hold Operation

- Step 1.** Connect the circuit of Figure 1 using 10 μ F noise-filtering capacitors across the 12 V power supplies for the op amps. Observe capacitor polarities to prevent an explosion hazard.

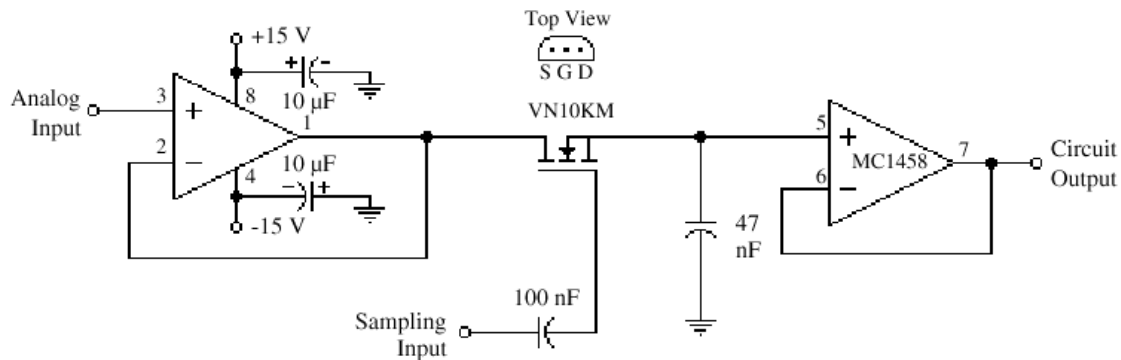


Figure 1. Sample and Hold Circuit

- Step 2.** Connect a 100 Hz sine wave generator as the analog input. Set its amplitude to 1.4 V_{p-p}. Set the frequency as close as you can to the 100Hz value.
- Step 3.** Connect a 1.00KHz square wave generator as the sampling input. Set its amplitude to 2.5 V_{p-p}. Set the frequency as close as you can to the 1.00KHz value.

From time to time in this experiment it may be necessary to momentarily short-circuit the 100nF capacitor so that the circuit can begin operation. If the sampling voltage is initially set too high and then decreased to the required setting, the circuit will sometimes fail to operate. If the sampling voltage is initially set too low and then increased to the required setting, the circuit will always operate correctly. A brief short across the 100nF capacitor will restore operation.

- Step 4.** With the oscilloscope triggered on the analog signal, observe the output waveform. You will probably see the small *hold samples* moving continuously through the 100 Hz waveform. This is normal because no matter how carefully you set the two frequencies, they will never be exactly a 10:1 ratio since they are separate unrelated generators, and each will drift to some extent.
- Step 5.** Adjust one of the generators very carefully and try to stop the motion of the sample levels, or at least slow the movement as much as possible.



- Step 6.** Count the number of sample points within one cycle of the 100Hz analog signal. Record.
- Step 7.** Adjust the analog frequency to 200 Hz and repeat steps 5 and 6. Repeat this for the frequencies setting of ??

Observing Signal Reconstruction

- Step 8.** Add a RC low pass filter to the output of the circuit as shown in Figure 2.

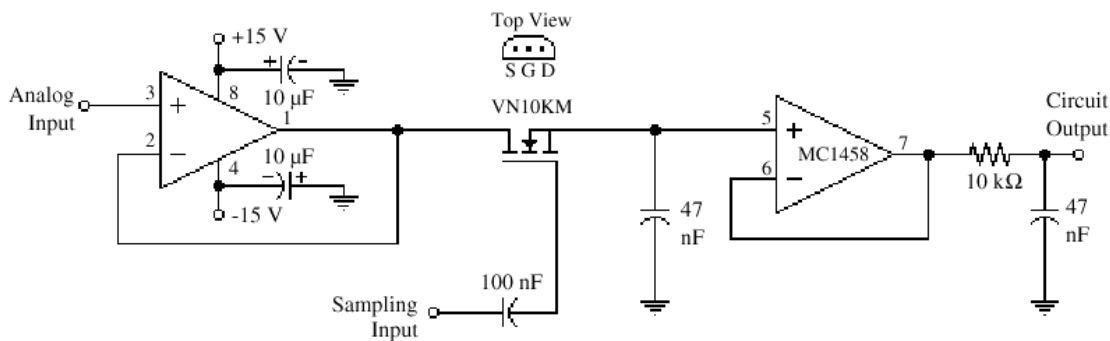


Figure 2. Reconstructed Output

- Step 9.** Set the analog signal back to 100Hz and observe the waveform across the 47nF capacitor of the RC low-pass filter. Record.

This would be similar to how a reconstructed waveform would appear if it had been digitized and then converted back to analog with a digital-to-analog (D/A) converter.

- Step 10.** Set the analog signal to 200Hz and observe the waveform across the 47nF capacitor of the RC filter. Record.
- Step 11.** Set the analog signal to 500Hz and observe the waveform across the 47nF capacitor of the RC low-pass filter. Record.

Observing Aliasing

- Step 12.** Remove the RC low pass filter from the circuit and set the analog frequency to about 980Hz. Set the scope sweep rate to 10 ms/div and observe the appearance of the alias frequency. record your observations in your log book.
- Step 13.** Slowly adjust the analog signal from 950Hz to 1050Hz and observe the various alias frequencies.



Observing High Sampling Rates

Step 14. Reset the analog signal to 100Hz and increase the sampling frequency to 10KHz. Observe that the output is now being sampled many times per cycle, more than you can easily count. This would represent a more typical relationship between the analog frequency and the sampling frequency for some but certainly not all applications.

Questions

1. When the VN10KM is on, the specification for its $R_{DS(on)}$ is 7.5Ω . What is the expected charging time constant for the hold capacitor? What is the time required to reach full charge ($5t$)?
2. To prevent aliasing distortion when the sampling frequency is 1.0KHz, what is the maximum allowable input analog frequency?
3. If the analog input frequency to the sample and hold circuit is 500Hz and its period is therefore 2.00ms, determine what percentage of this period time would be required for the 47nF capacitor to reach its full charge.
4. The time to reach full charge should be less than 0.5 % of the period of the input analog frequency. Is this circuit within an acceptable range?
5. What would happen if the hold capacitor were increased to $4.70\mu F$?



Application

Analog to Digital A/D Conversion

Objectives

This lab activity will study the use of a typical analog to digital converter, and its input output characteristics by breadboarding and testing the circuit.

- To build the A/D converter circuits
- Study its input output characteristics



Successive Approximation Analog to Digital (A/D) Conversion

The Successive Approximation Register (SAR) Principle

The block diagram for a basic 4-bit SAR A/D is shown in Figure 1. Initially, the register is set to 0 (0000₂). The control logic then modifies the contents of the SAR, bit by bit. Each new binary number generated is fed to the DAC, where the analog voltage is produced (V_{out}) and sent to the comparator. When the conversion process is complete, the digital output reflects the input analog voltage at V_{in}.

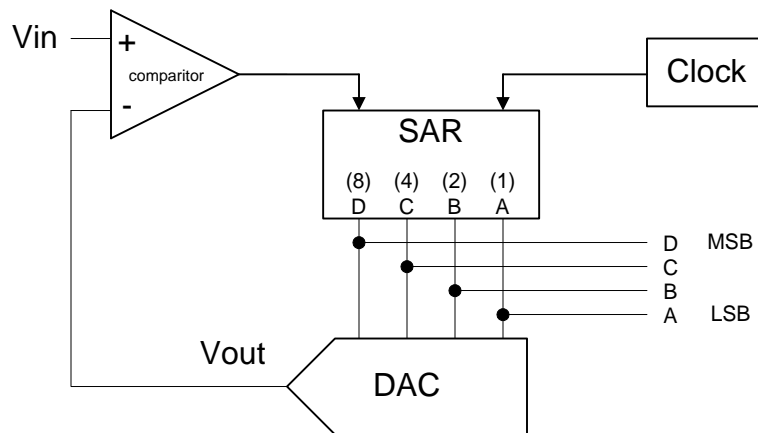


Figure 1. Block Diagram

To examine the concept a bit further. First, a clock pulse is sent to the control logic. It in turn, outputs a signal that sets the SAR's most significant bit (MSB) to 1. The register then contains the value 1000₂. The DAC then generates an analog output voltage, V_{out}, equivalent to the number 1000₂, and sends it to the inverting input of the comparator. If this voltage is greater than V_{in}, the comparator outputs a LOW, causing the MSB to reset to 0. If the voltage is less than V_{in}, the comparator outputs a HIGH, and the 1 is retained in the MSB location. This process now repeats with the next most significant bit, until all bits have been checked. After all the bits from the SAR have been placed, the conversion cycle is complete. The digital output then reflects the analog voltage input, within the resolution of the converter.

The following example will illustrate the circuit's operation, by analyzing how a 4-bit A/D converter, converts a 1.5V analog input to the equivalent binary number 0101₂.

Assume that the A/D converter has a range from 0 to 5. This means it has a resolution of

$$\frac{5}{2^4 - 1} = \frac{5}{15} = .333 \text{ V/bit. This means } V_{out} = 2.67\text{V for the D(8) bit (MSB),}$$



$V_{out} = 1.33V$ for the C(4) bit, $V_{out} = .667V$ for the B(2) bit, and $V_{out} = .333V$ for the A(1) bit (LSB).

In Figure 2 we see the first step of the conversion cycle. With the first clock pulse, the MSB, D(4), is set to 1. The output of the DAC is now 2.67V. Since this voltage is greater than the V_{in} of 1.5V, the comparator's output goes LOW, causing the MSB in the SAR to reset to 0.

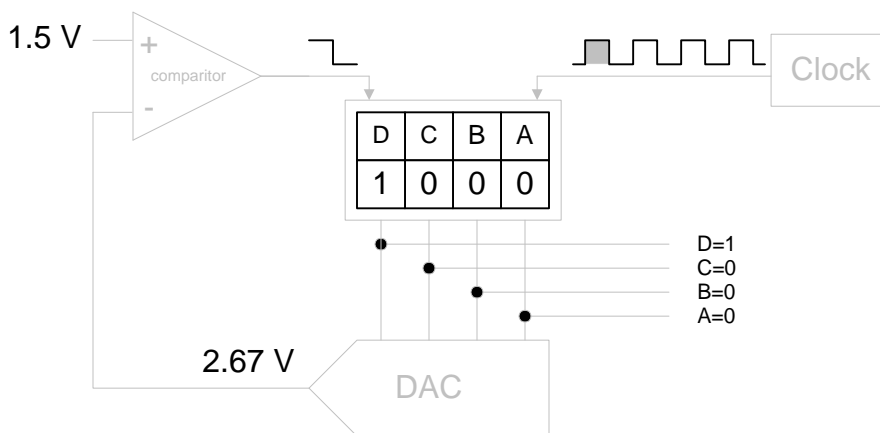


Figure 2.

Figure 3. illustrates the second step in the conversion cycle. With the second clock pulse, the C (4) bit is set to 1. The DAC output is now 1.33V. This voltage is less than the V_{in} of 1.5V. The comparator's output goes HIGH, and the 1 bit is retained in the SAR.

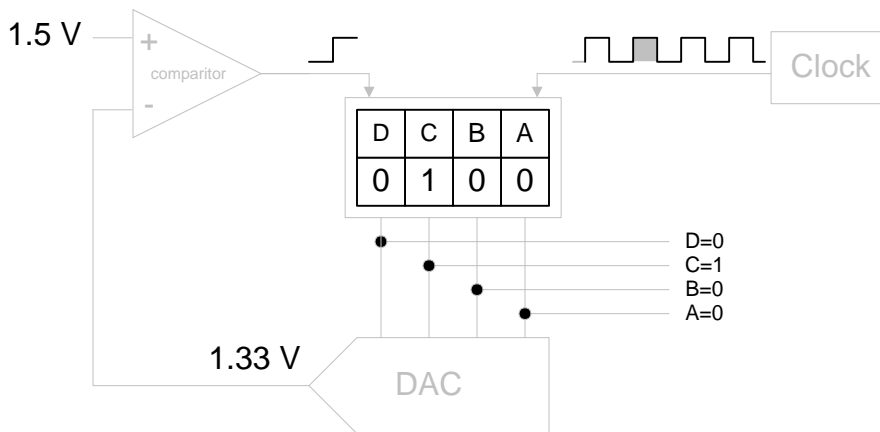


Figure 3.



Figure 4. shows the third step in the conversion cycle. With the third clock pulse, the B (2) bit is set to 1. Note, though, that the DAC now outputs 2.00V, not .667V as you might first expect. This is because the C and B bits are both set to 1, and giving a V_{out} of $1.33 + .67 = 2.00V$. Since this voltage is greater than 1.5 V, the comparator's output goes LOW, and the B bit is reset to 0.

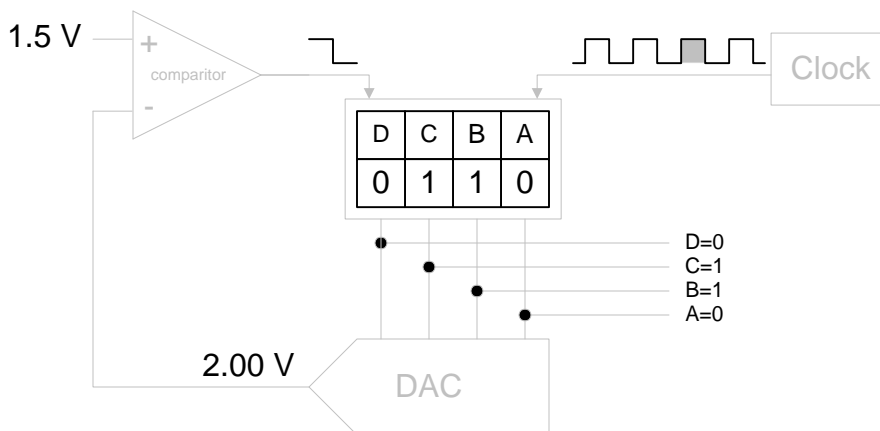


Figure 4.

Figure 5. illustrates the fourth, and last, step in the conversion cycle. With the fourth clock pulse, the A (1) bit is set to 1 and the DAC output is now 1.67 V (the A and C bits are set to 1).

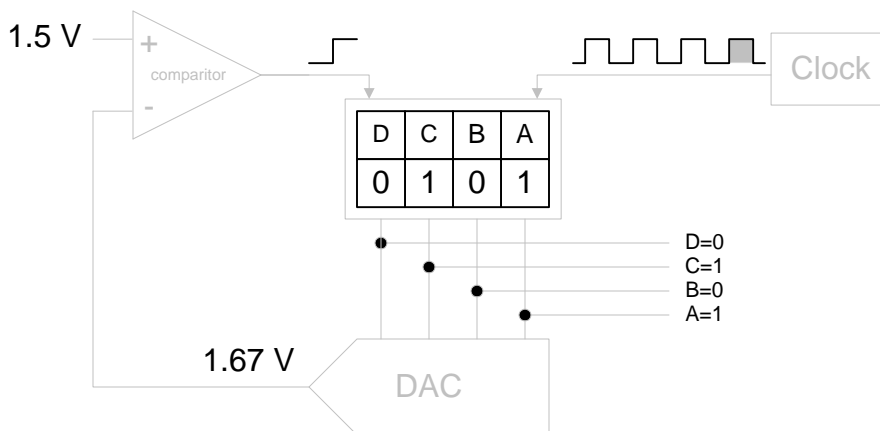


Figure 5.

At this point the conversion cycle is complete, since all 4 bits have been tried. The binary output of 0101_2 is the closest value to the 1.5 V input analog voltage. A new conversion cycle can now begin.



Note that during the conversion only four clock pulses were required. This illustrates an important advantage of the successive approximation A/D. It has a fixed conversion time that is independent of the input analog voltage, unlike other types of A/D converters. An N-bit SAR A/D will require N clock pulses to perform a conversion, regardless of the level of the input analog voltage. The following timing diagram (Figure 6.) summarizes the sequence.

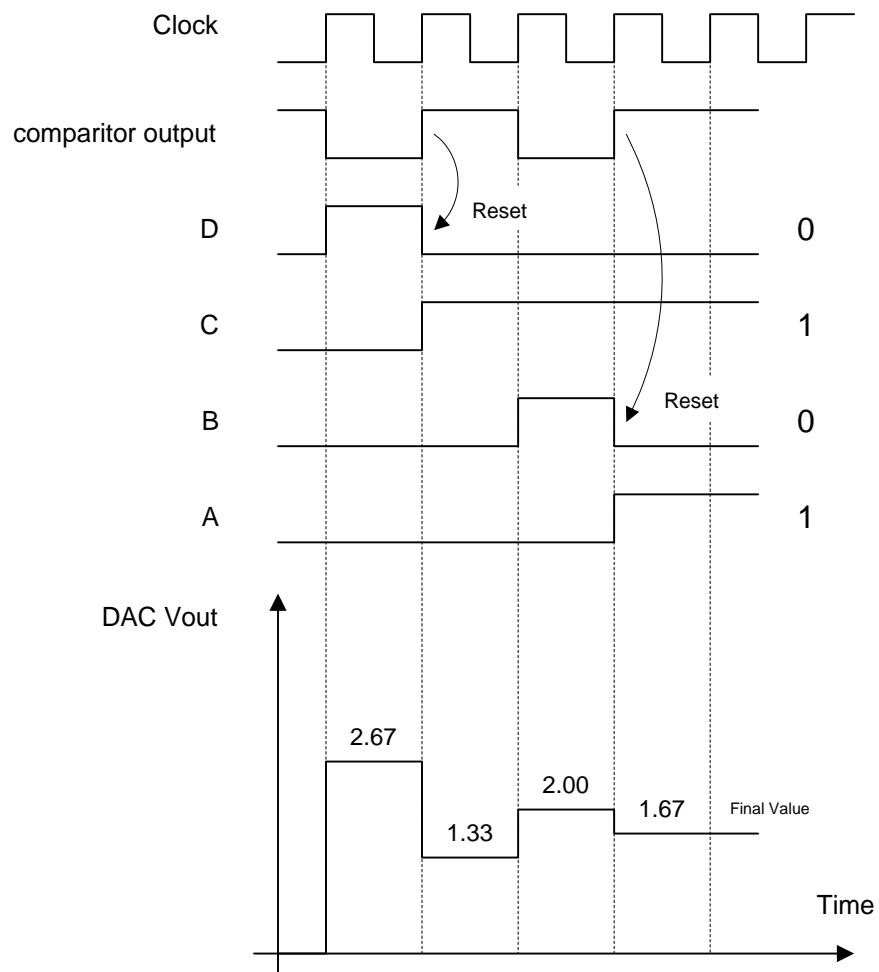


Figure 6. Timing Diagram



The ADC0804 A/D converter (the real thing)

Complete 8-bit successive-approximation A/D converters are available as a single IC (Integrated Circuit), usually in 18 or 20 pins DIP (Dual In-line Package). They contain an on-chip clock generator, the successive approximation register (SAR), an 8-bit DAC, all control logic, and three-state outputs to facilitate computer bus interfacing. Figure 7. shows the pin configuration for a typical 8-bit A/D converter chip, the ADC0804. The ADC0804 is a member of a family of A/D converters. For a detailed description and application information refer to its data sheet, **ADC0801.pdf** in the parts library. \overline{CS}



Figure 7. ADC0804 pin out

Before we apply the A/D we will identify each pin, to fully understand its functional description in relation to its functional diagram. Note that many of the pins are identified with computer terminology, since most ADC are used to interface with microprocessors.

Pin Number(s)	Symbol	Description
1	\overline{CS}	Active LOW chip select. For the ADC0804 to operate, it must be asserted LOW.
2	\overline{RD}	Active LOW read pin. When LOW, the tri-state outputs are active and the digital output DB7 - DB0 can be read.
3	\overline{WR}	An active LOW write pin. When it goes LOW, the A/D conversion process begins.
4	CLK IN	Clock input pin. A clock signal is available from pin 19, or



		any TTL level square wave signal source will do.
5	$\overline{\text{INT}}$	Active LOW interrupt, or halt pin. When LOW it signals that the conversion process is complete. If connected to $\overline{\text{WR}}$ the conversion process will immediately begin again.
6 and 7	Vin(+) and Vin(-)	Differential input. Usually, the Vin(+) input is used and the Vin(-) is grounded.
8	A GND	Analog ground connection for the analog input voltage.
9	Vref/2	Controls the voltage to which the analog input voltage is referenced. If left unconnected, the reference is Vcc.
10	D GND	Digital ground for the device and digital outputs.
11 - 18	DB7 - DB0	These are the tri-state buffered digital outputs.
19	CLK R	This is the output from the internal clock. Its frequency is determined by an external resistor and capacitor.
20	Vcc	This is the supply voltage, typically +5 V, though it can go as high as 6.5 V.

Application

In Figure 8. a test/demonstration circuit for the ADC0804 is detailed. This circuit will convert an analog input voltage of from 0 to 5 V (potentiometer R2 is acting as a voltage divider) and produce an equivalent digital output with full 8-bit resolution, in a conversion time of approximately 13 ms.

The external resistor capacitor R1, C1 will setup a clock frequency of

$$\frac{1}{1.1 \times RC} = \frac{1}{1.1 \times 10k \times 150p} \approx 600 \text{ Hz}$$

The connection from $\overline{\text{INTR}}$ to $\overline{\text{WR}}$ will cause the A/D to start a new conversion each time the $\overline{\text{INTR}}$ (end of conversion line goes low. S1 is used to make sure that an initial first conversion takes place.

The $\overline{\text{CS}}$ is grounded to enable the A/D converter chip.

The $\overline{\text{RP}}$ is grounded to enable the D0 - D7 outputs.

The analog input voltage is positive 0 - 5V referenced to ground, so it is connected to Vin(+) , and Vin(-) is connected to ground.



The outputs D0 - D7 are connected to light emitting diodes through a current limiting resistor of 330 Ω .

Since the analog input voltage is directly proportional to the digital output, the following ratio applies.

$$\frac{V_{in}}{V_{ref}} = \frac{D_{out}}{256} \quad \text{or} \quad D_{out} = \frac{V_{in}}{V_{ref}} \times 256 \quad \text{digital output in base 10}$$

Our resolution is $\frac{5}{2^8 - 1} = \frac{5}{255} = 19.6 \text{ mV/bit}$.

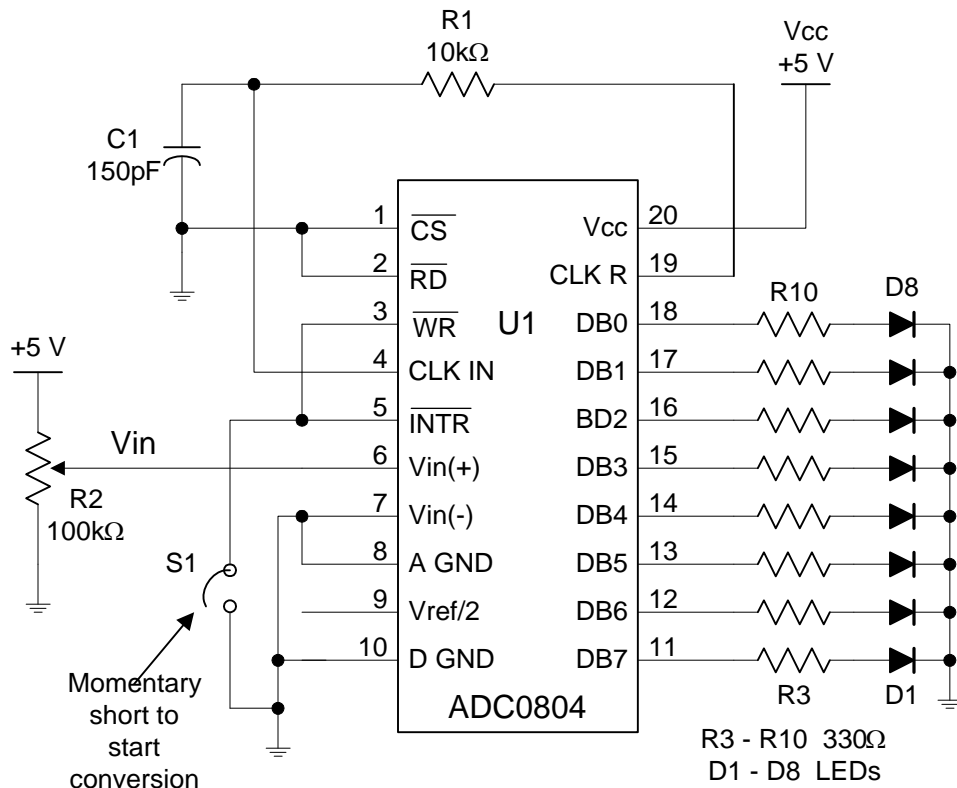
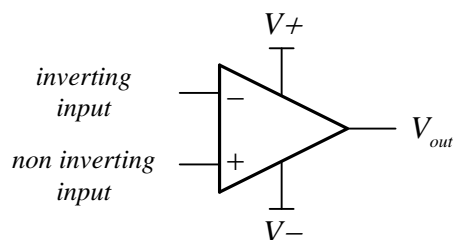
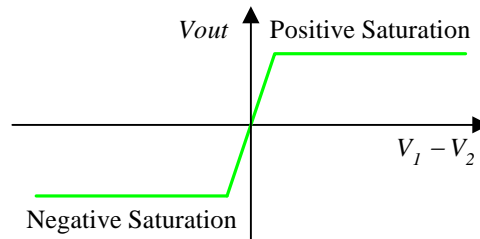
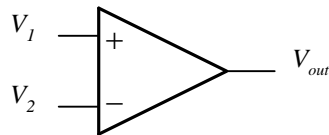


Figure 8. Connection for continuous conversion



The Operational Amplifier



Very high open-loop voltage gain
Very high input impedance ($\infty \Omega$)
Very low output impedance (0Ω)

Negative Feedback Circuits

Rule 1 - The output attempts to drive its feedback circuit to make the voltage *differential* at the inputs zero.

This means that the non-inverting and inverting inputs are always assumed to be equal or at the same potential.

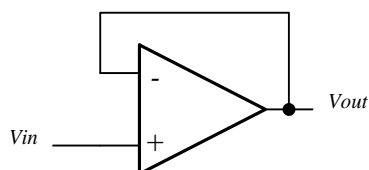
This does not mean that the op-amp changes the voltage at its inputs, but rather that the output will swing either positive or negative to bring the *differential* voltage as close to zero as it can.

Rule 2 - The input to the op-amp will draw no current. This is the same thing as saying that the op-amp's input impedance is infinite.

Very high input impedance ($\infty \Omega$)

For best performance, use resistors in the $1K\Omega$ to $100K\Omega$ range

Voltage Follower

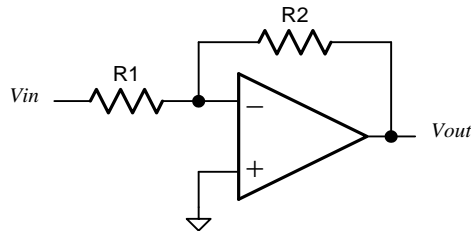


$$\frac{V_{out}}{V_{in}} = 1$$

Very high input impedance ($\infty \Omega$)
Very low output impedance (0Ω)

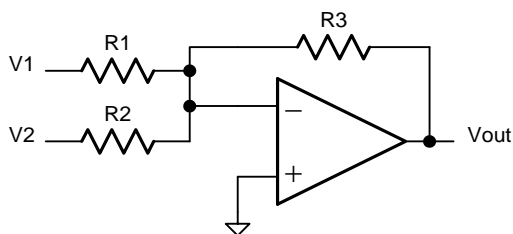


Inverting Amplifier



$$\frac{V_{out}}{V_{in}} = -\frac{R2}{R1}$$

Inverting Summing Amplifier

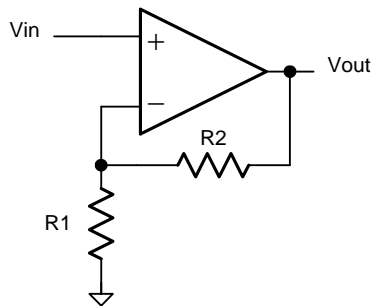


$$V_{out} = -\left(\frac{V_1}{R_1} + \frac{V_2}{R_2}\right) \times R_3$$

If $R_1 = R_2 = R_i$

$$V_{out} = -(V_1 + V_2) \times \frac{R_f}{R_i}$$

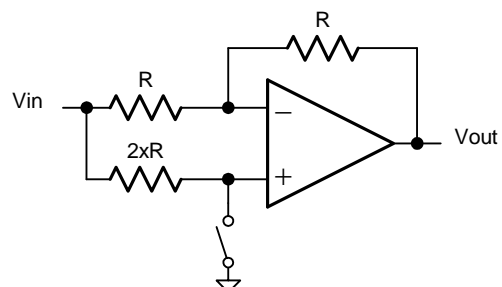
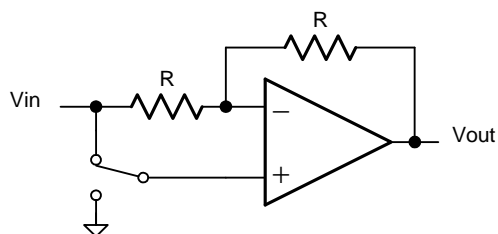
Non-inverting Amplifier



$$\frac{V_{out}}{V_{in}} = 1 + \frac{R2}{R1}$$

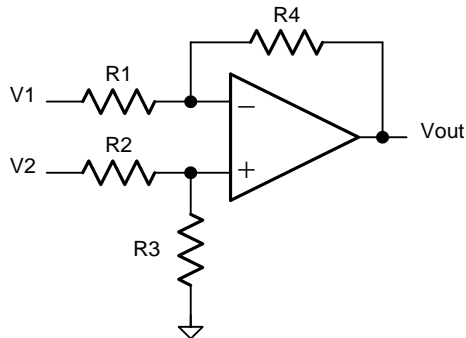
Very high input impedance ($\infty \Omega$)

Inverting / Noninverting Amplifiers





Differential Amplifier



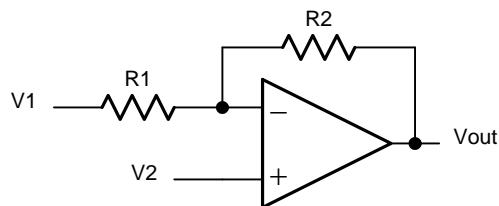
$$V_{out} = V_2 \frac{R_4}{R_1} \times \frac{R_2 + R_3}{R_2 + R_3} - V_1 \frac{R_4}{R_1}$$

If $R_1 = R_2 = R_i$ and $R_3 = R_4 = R_f$

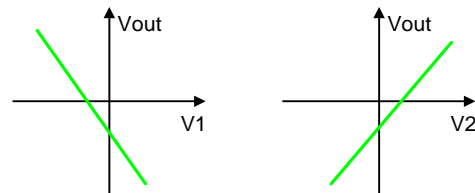
$$V_{out} = (V_2 - V_1) \times \frac{R_f}{R_i}$$

SCC

$$y = mx + b$$

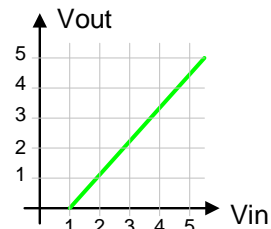
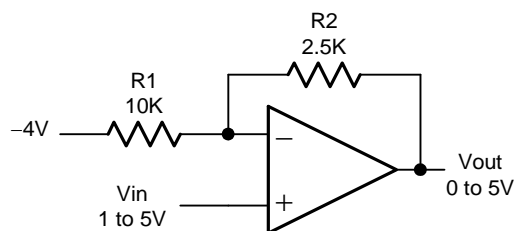


$$V_{out} = \left(1 + \frac{R_2}{R_1}\right)V_2 - \frac{R_2}{R_1}V_1$$



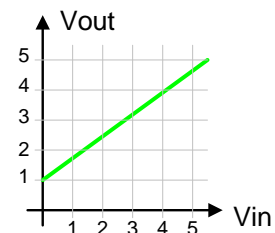
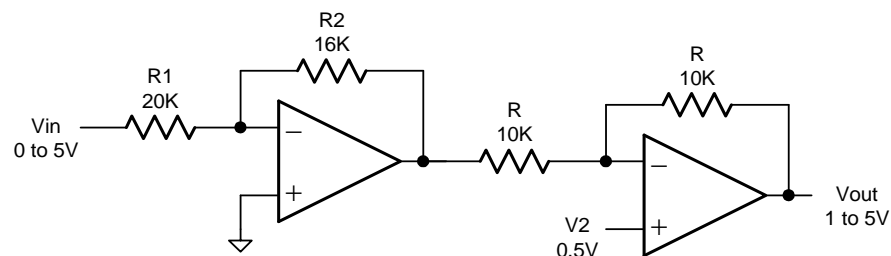
SCC

$$V_{out} = 1.25V_{in} + 1$$



SCC

$$V_{out} = 0.8V_{in} + 1$$

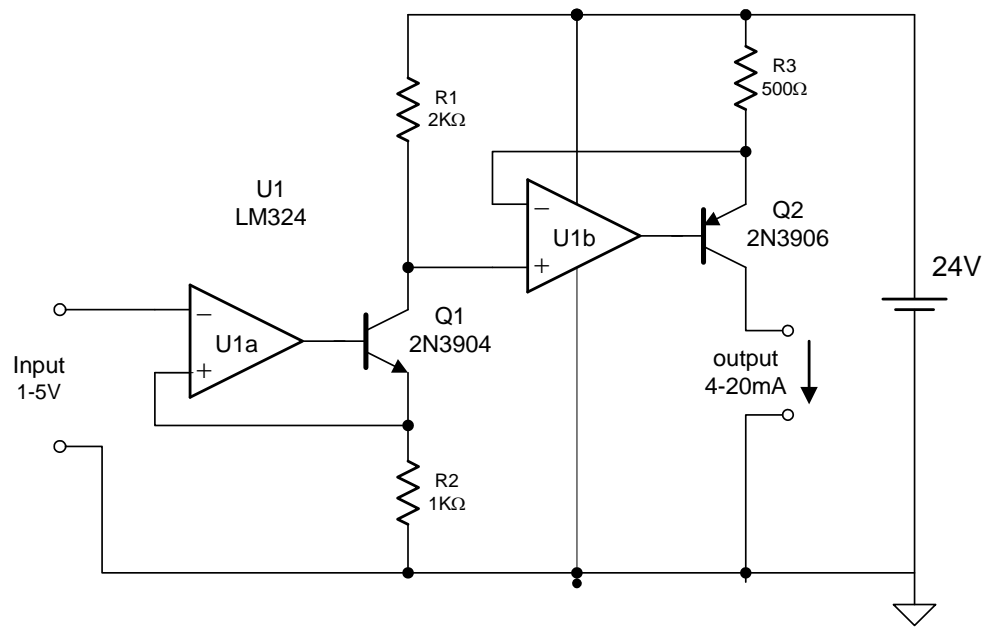




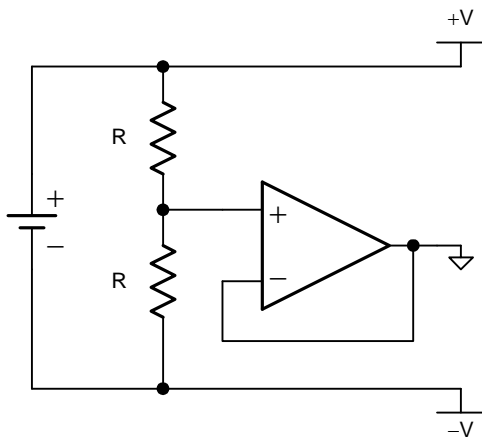
SCC

1-5V to 4-20mA

Note that the circuit operates from a single power supply.



Split Power

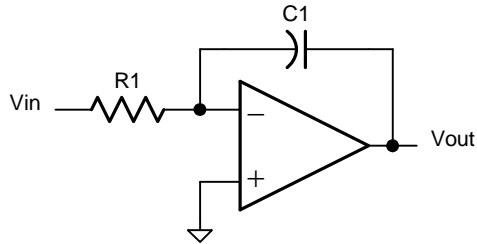


This circuit will split a single supply to produce a dual supply that can be used with all dual supply operational amplifiers circuits.



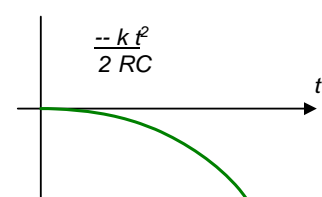
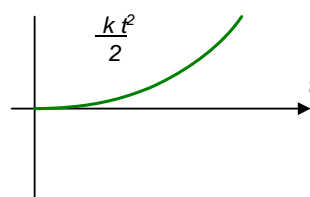
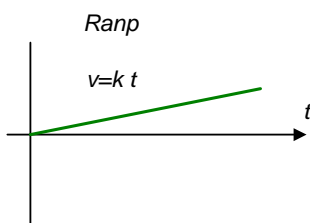
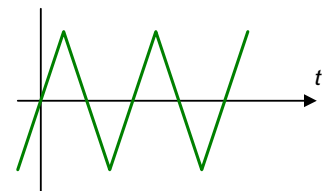
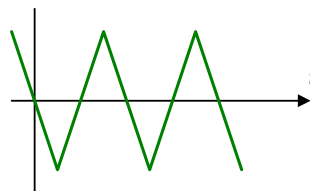
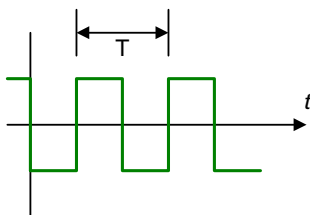
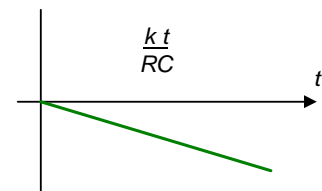
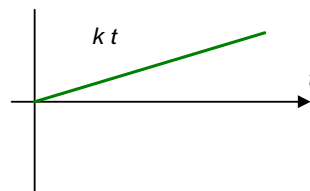
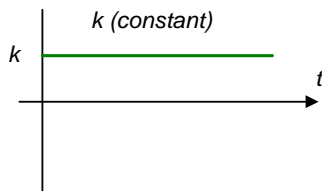
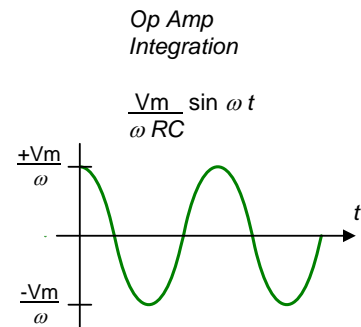
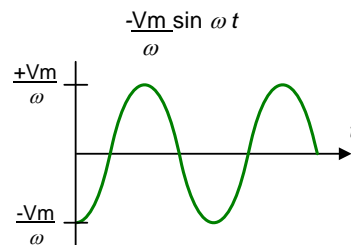
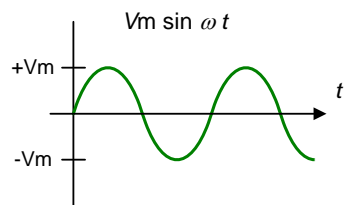
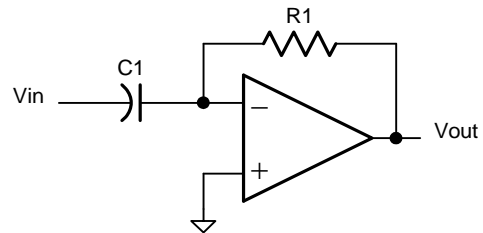
Integrator

$$v_{out} = -\frac{1}{C} \int v_{in} dt$$



Differentiator

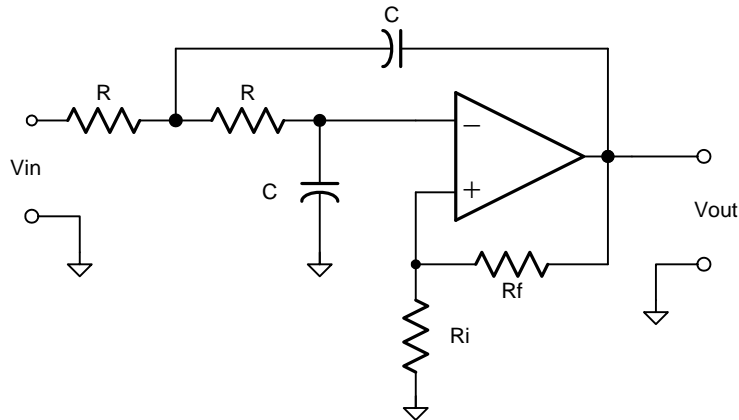
$$v_{out} = -\int v_{in} dt$$





2nd order Active Low-Pass Filter - *Sallen-Key*

3dB point $f_o = \frac{1}{2\pi RC}$ Drop off is 40dB/dec



If $R_f = (K-1)R_K$
and $R_i = R_K$
 $K = \text{gain (DC)}$

$$\text{Gain(DC)} = 1 + \frac{R_f}{R_i}$$

$K = 1$ $R_f = 0$ (short), and
 $R_i = \infty$ (open)

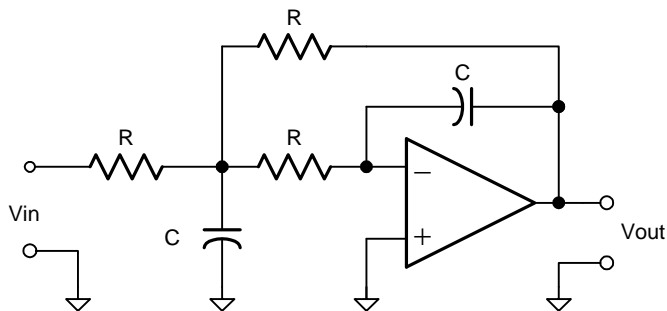
damping factor ζ

$$2\zeta = 3 - K$$

$K = 1.268$ Bessel
 $K = 1.586$ Butterworth
 $K = 2.234$ Chebyshev

2nd Order Low-Pass Filter - *MFB*

3dB point $f_o = \frac{1}{2\pi RC}$ Drop off is 40dB/dec



Integration