Chapter 3 : AUDIO CIRCUITS

The most natural way of communication for people is to speak to each other. The voice is transmitted and received in electronic communications, to enable people communicate over large distances. The first thing that must be done is to convert voice into an electrical signal, and process it before transmission. The last process in a transceiver, on the other hand, is to recover voice from the received RF signal. The audio circuits of TRC-10 are discussed in this chapter. The mathematical tools necessary to analyze circuits used in TRC-10 are also developed.

3.1. Linear circuits

A good understanding of exponential function with an imaginary argument, $\exp(j\phi)$, is fundamental in electronic engineering. There is a relation between sinusoids and exponential function, as follows:

$$\exp(j\phi)= \cos(\phi)+j\sin(\phi)$$

This is called Euler's formula. In other words, $\cos(\phi)$ is the real part of $\exp(j\phi)$, and $\sin(\phi)$ is the imaginary part. Sinusoids can be expressed as

$$\cos(\phi)= \mathrm{Re}[\exp(j\phi)]$$

or

$$\cos(\phi)= \{\exp(j\phi)+ \exp(-j\phi)\}/2$$

and

$$\sin(\phi)=\mathrm{Im}[\exp(j\phi)]$$

or

$$\sin(\phi)=\{\exp(j\phi) - \exp(-j\phi)\}/2j,$$

in turn. The magnitude of this exponential function is

$$|\exp(j\phi)|=1,$$

regardless of the value of the argument ϕ.

Let us consider a sinusoidal voltage $v(t)= V_p\cos(\omega t+\theta)$. In terms of exponential function,

$$v(t)= V_p\mathrm{Re}[\exp j(\omega t+\theta)] = \mathrm{Re}[V_p\exp j(\omega t+\theta)].$$

Linearity is also a fundamental concept in circuit analysis. Consider the block diagram in Figure 3.1. A circuit is called *linear* if it satisfies the following relation:

"If an input $x_i(t)$ (voltage or current) yields an output $y_i(t)$ (again voltage or current) in a *linear circuit*, then a linear combination of inputs, $ax_1(t)+ bx_2(t)$ yields the same combination of the individual outputs, $ay_1(t)+by_2(t)$". The number of inputs is not limited to two, but can be unlimited.

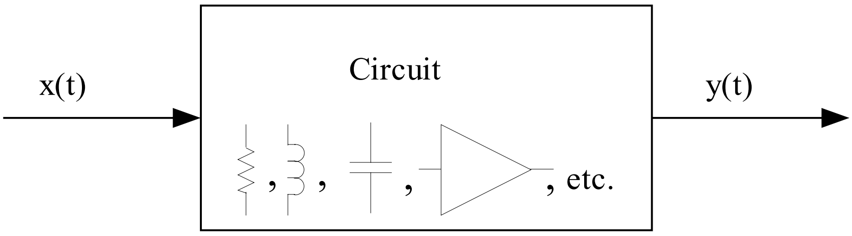


Figure 3.1 Linear circuit and linear circuit elements

This relation can be expressed as

If $x_1(t)\Rightarrow y_1(t)$ and $x_2(t)\Rightarrow y_2(t)$, then $ax_1(t)+ bx_2(t) \Rightarrow ay_1(t)+by_2(t)$

symbolically. Consider a circuit formed by a single resistor. If the input to the resistor is a current $i_1(t)$, and the output $v_1(t)$ is the voltage developed across it, then

$v_1(t) =R\, i_1(t)$.

If we apply a combination of two inputs $3i_1(t) +5i_2(t)$, then the total voltage developed across the resistor is

$v(t)=R[3i_1(t) +5i_2(t)]=3Ri_1(t)+ 5Ri_2(t)=3v_1(t) + 5v_2(t)$,

where $v_2(t)$ is the voltage corresponding to $i_2(t)$. Hence resistor is a linear circuit element.

Similarly for an inductor, since

$$v_1(t) = L\frac{di_1(t)}{dt},$$

then,

$$v(t) = L\frac{d[ai_1(t)+bi_2(t)]}{dt} = aL\frac{di_1(t)}{dt} + bL\frac{di_2(t)}{dt} = av_1(t) + bv_2(t),$$

where $v_2(t)$ is the voltage induced on the inductor by $i_2(t)$. Hence inductor is also a linear circuit element. Capacitor is a linear element too.

A large circuit, which contains only linear components, is also linear.

When a sinusoidal current, $I_p\cos(\omega t+\theta)$, passes through a resistor, voltage developed across it, is

$v(t) = RI_p\cos(\omega t+\theta) =V_p\cos(\omega t+\theta)$.

The only parameter modified by the resistor is the amplitude of the signal, i.e. $RI_p=V_p$. In case of an inductor,

$$
\begin{aligned}
v(t) &= Ld[I_p\cos(\omega t+\theta)]/dt \\
&= -\omega LI_p\sin(\omega t+\theta) \\
&= \omega LI_p\cos(\omega t+\theta+\pi/2).
\end{aligned}
$$

Hence waveform for v(t) is the same as that of the current, but the amplitude is scaled as ωLI_p (which has units of volts) and phase is shifted by $\pi/2$. The frequency is not changed. This is a very significant property of linear circuits.

We can write v(t) as,

$$
\begin{aligned}
v(t) &= \omega LI_p\mathrm{Re}\{\exp[j(\omega t+\theta+\pi/2)]\} \\
&= \mathrm{Re}\{\omega LI_p\exp[j(\omega t+\theta+\pi/2)]\} \\
&= \mathrm{Re}\{\omega LI_p\exp[j(\pi/2)]\exp(j\theta)\exp(j\omega t)\} \\
&= \mathrm{Re}\{j\omega LI_p\exp(j\theta)\exp(j\omega t)\}
\end{aligned}
$$

If we compare this expression with that of the input current $i(t) = I_p\cos(\omega t+\theta) = \mathrm{Re}[I_p\exp(j\theta)\exp(j\omega t)]$, we immediately notice that the only difference between the input and the output is $I_p\exp(j\theta)$ is scaled to $j\omega LI_p\exp(j\theta)$. There is no change in the time function $\exp(j\omega t)$, as far as the complex terms (mathematically speaking, otherwise it is simple) in the brackets are concerned.

The complex expressions $I= I_p\exp(j\theta)$ and $V= j\omega LI_p\exp(j\theta)$ are called *phasors*. Note that the voltage phasor across the inductor and its current phasor are related by only a complex number multiplier:

$$I = \frac{1}{j\omega}\frac{V}{L},$$

if the time waveform is sinusoidal at an angular frequency of ω. Both phasors *are scalar functions* of ω. All the information available about the circuit is contained in the phasors, for given angular frequency.

Special note on notation: It is common practice to denote phasors as upper case letter functions of ω, such as

$$V(\omega) = j\omega LI_p\exp(j\theta)$$

and

$$I(\omega) = I_p\exp(j\theta),$$

and corresponding time waveforms as lower case letter functions of time, as

$$v(t) = \mathrm{Re}[V(\omega)\exp(j\omega t)]$$

and

$i(t) = \mathrm{Re}[I(\omega)\exp(j\omega t)]$

respectively. The magnitude of the phasor

$|V(\omega)| = \omega L I_p$

is the amplitude of the sinusoidal time waveform, and the phase of the phasor,

$\angle V(\omega) = \theta + \pi/2$

is the phase of the time waveform. We can express v(t) as,

$v(t) = |V(\omega)| \cos[\omega t + \angle V(\omega)]$.

In cases when the entire discussion is at a standard frequency, like line frequency , the "(ω)" argument of the phasor is dropped, and only capital letter V and I are used. In such cases the fixed standard angular frequency argument, e.g. $V(2\pi\times 50)$, is implicit.

There are two trends in the definition of the magnitude of the phasors. In most of the *electrical engineering* books the magnitude of the phasors are defined as the *rms value of the quantity* (voltage or current). The other trend is to use the amplitude directly as we defined above.

The voltage and current phasor relations for a resistance, a capacitance and an inductor are as follows:

$V_R(\omega) = R\, I_R(\omega)$,

$I_C(\omega) = j\omega C\, V_C(\omega)$,

and

$V_L(\omega) = j\omega L\, I_L(\omega)$.

3.1.1. Power

We know that

$P(t) = v(t)i(t)$

defines the instantaneous power delivered to the circuit, where v(t) is the voltage across the circuit and i(t) is the current into the circuit. P(t) accounts both for the power dissipated in the circuit, the average power, and the power that goes to the inductors and capacitors, which is the *reactive power*.

In phasor analysis, power is related to voltage and current phasors as

$P = VI^*/2$.

Here, P is the *complex power*, which is again the sum of average and reactive power in the circuit. I^* is the complex conjugate of current phasor I. The dissipated power is the average power,

$P_a = \mathrm{Re}\{VI^*/2\}$

and the reactive power is

$P_r = \mathrm{Im}\{VI^*/2\}$.

3.2. Impedance and Transfer Function

We have observed that the definition of phasors allowed us to convert the differential relations in time into algebraic relations in angular frequency. We can now analyze circuits, which contain capacitors and inductors by solving linear algebraic equations that result from KVL and KCL. The variables and coefficients of these equations are, of course, complex variables and complex coefficients, respectively.

Consider the circuit given in Figure 3.2. The current in the circuit is

$$i(t) = C\frac{dv_C(t)}{dt}$$

and

$v_{in}(t) = V_i\cos(\omega t) = R\, i(t) + v_C(t)$,

in time domain.

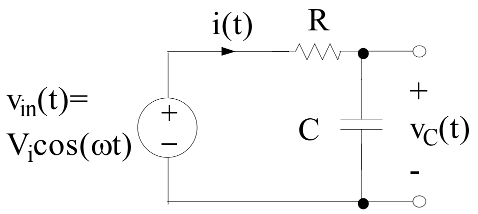


Figure 3.2 Series RC circuit with sinusoidal voltage excitation.

Differentiating both sides, we obtain

$$\frac{dv_{in}(t)}{dt} = -\,\omega V_i \sin(\omega t) = R\frac{di(t)}{dt} + \frac{i(t)}{C}.$$

The current in the circuit is the solution of the differential equation

$$\frac{di(t)}{dt} + \frac{i(t)}{RC} = -\,\omega\frac{V_i}{R}\ \sin(\omega t).$$

Let us analyze the circuit using phasors instead of solving the above equation. The input voltage phasor is $V_{in}(\omega)=V_i\exp(j0)= V_i \angle 0 = V_i$, since the phase of $v_{in}(t)$ is zero and its amplitude is V_i. Let the phasor of the circuit current, i(t), be I(ω), and voltage phasor of capacitance be $V_C(\omega)$. Then,

$$\begin{aligned} V_{in}(\omega) &= R\, I(\omega) + V_C(\omega) \\ &= R\, I(\omega) + I(\omega)/j\omega C \\ &=(R+1/ j\omega C)\, I(\omega) \end{aligned}$$

using KVL. Therefore

$$\begin{aligned} I(\omega) &= V_{in}(\omega)[j\omega C/(1+ j\omega CR)] \\ &= V_i\, [j\omega C/(1+ j\omega CR)]. \end{aligned}$$

Now, magnitude of I(ω) is $|I(\omega)|=\omega CV_i /[1+ (\omega CR)^2]^{1/2}$, and its phase is $\angle I(\omega) = \pi/2 - \arctan(\omega CR) = \theta$. The current, i(t), in the circuit is,

$$i(t) = \mathrm{Re}\{\omega CV_i /[1+ (\omega CR)^2]^{1/2} \exp(j\theta) \exp(j\omega t)\}$$

$$= \omega CV_i /[1+ (\omega CR)^2]^{1/2} \cos(\omega t+\theta).$$

If we know the numerical values of V_i, ω, C, and R, we can calculate $|I(\omega)|$ and θ, and evaluate i(t) numerically.

The phasors of voltage across the circuit, $V_{in}(\omega)$, and the current through it, I(ω), are related as

$$\begin{aligned} V_{in}(\omega) &= [(1+ j\omega CR)/j\omega C]\, I(\omega) \\ &= Z\, I(\omega). \end{aligned}$$

The *complex* expression $[(1+ j\omega CR)/j\omega C]$ behaves like a resistance, but it is not *real* like a resistance. We call this term *impedance*, and denote it by letter *Z*. Impedance has *real* and *imaginary* parts,

$$\begin{aligned} Z &= R - j(1/\omega C) \\ &= R + jX. \end{aligned}$$

Real part is called resistance and denoted by letter R for obvious reasons, and imaginary part is called *reactance* and is denoted by *X*. The units of impedance and reactance are ohms (Ω), like resistance.

Inverse of Z is called *admittance* and is denoted by *Y*,

$$Y = j\omega C/(1+ j\omega CR)$$

$$\begin{aligned} &= R(\omega C)^2/[1+ (\omega CR)^2]+ j\omega C/[1+ (\omega CR)^2] \\ &= G + jB. \end{aligned}$$

G is the real part of admittance and is called conductance, and *B* is the imaginary part and referred to as *susceptance*. Both admittance and susceptance are measured in siemens (S), like conductance.

The RC circuit in Figure 3.2 has a very important property: it is a basic filter block. The voltage across the capacitor is

$$\begin{aligned} V_C(\omega) &= I(\omega)/j\omega C \\ &= [1/(1+ j\omega CR)]\, V_{in}(\omega). \end{aligned}$$

The circuit transfers the input voltage to the output, $V_C(\omega)$, after dividing it by $1+j\omega CR$. The ratio of output phasor to the input phasor,

$$V_C(\omega)/V_{in}(\omega) = 1/(1+ j\omega CR) = H(\omega)$$

is called the *transfer function*. The magnitude and phase of the transfer function for this circuit is

$$|H(\omega)| = 1/[1+ (\omega CR)^2]^{1/2}$$

and

$$\angle H(\omega) = -\arctan(\omega CR).$$

$H(\omega)$ specifies the output with respect to input for any radial frequency ω. Variation of the transfer function of this circuit with respect to ω is shown in Figure 3.3(a).

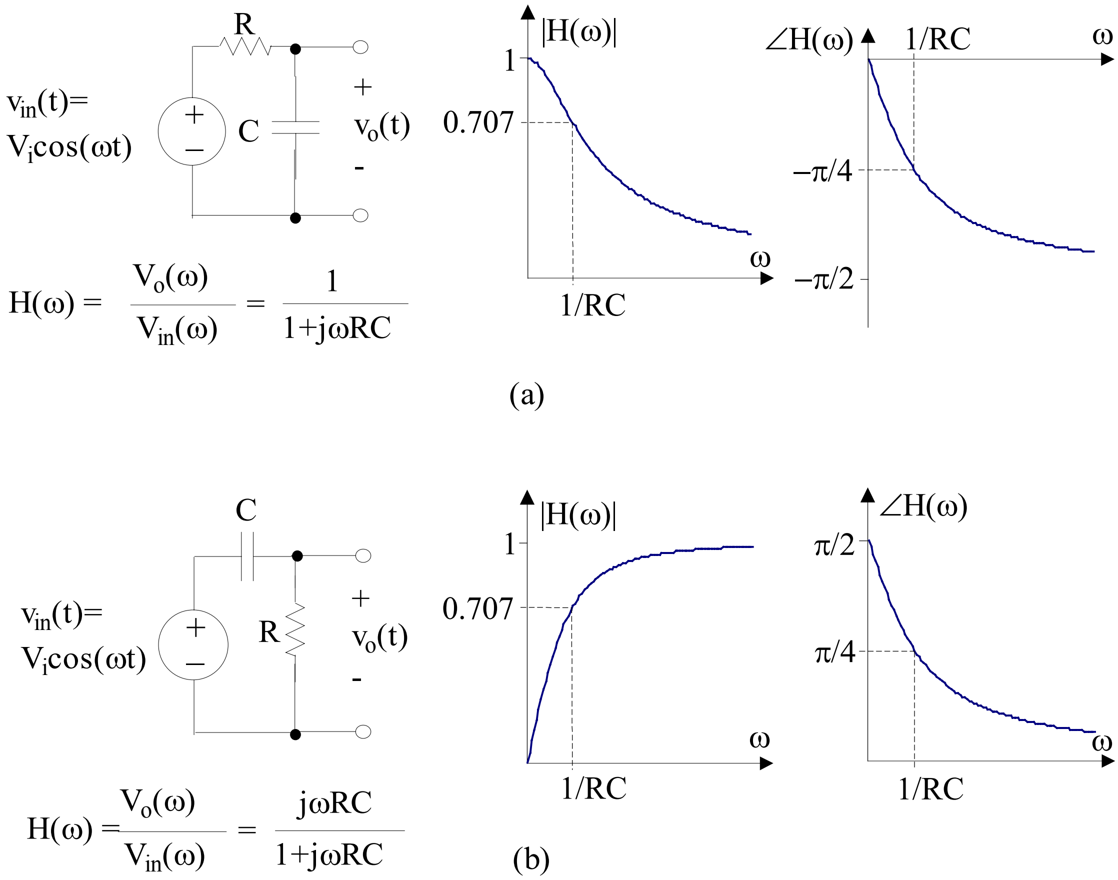
Note that for values of angular frequency near $\omega = 0$, $H(\omega) \approx 1$, and therefore the output is approximately the same as input for such frequencies. However $H(\omega)$ gets smaller and smaller as ω increases. The output approaches to zero as frequency increases. This circuit is called *RC low pass filter*, because of this property. $|H(\omega)|=1/\sqrt{2} = 0.707$ at $\omega = 1/RC$.

Now, since

$$\begin{aligned} 20\log|H(1/RC)| &= 20\log(0.707) \\ &= -3 \text{ dB}, \end{aligned}$$

$f_c=1/2\pi RC$ is called the "3-dB cut-off frequency".

The phase of $H(\omega)$ at $\omega = 1/RC$ is $\angle H(\omega) = -\arctan(\omega CR) = -\arctan(1) = -45°$.



$$H(\omega) = \frac{V_o(\omega)}{V_{in}(\omega)} = \frac{1}{1+j\omega RC}$$

$$H(\omega) = \frac{V_o(\omega)}{V_{in}(\omega)} = \frac{j\omega RC}{1+j\omega RC}$$

Figure 3.3 Series RC filters (a) LPF, (b) HPF

A part of receiver audio circuit is shown in Figure 3.4. This circuit is one of the LP filters which limits the bandwidth of the received and detected audio signal to approximately 3 KHz.

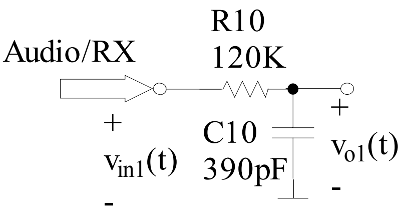


Figure 3.4 Audio/RX LPF

Let $V_{in1}(\omega)$ and $V_{o1}(\omega)$ be input and output signal (voltage) phasors. $RC = R_{10}C_{10} = 4.68$ E-5 sec. in this circuit and, therefore, the cut-off frequency f_c is 3.4 KHz. The transfer function $H_1(\omega)$ is

$$\begin{aligned} H_1(\omega) &= V_{o1}(\omega)/V_{in1}(\omega) \\ &= 1/(1+j\omega RC) \\ &= 1/[1+j\omega/(2\pi\times 3.4\text{ E3})] \\ &= 1/(1+jf/3.4\text{E3}) \end{aligned}$$

Note that we have written the transfer function in terms of frequency f, rather than the angular frequency ω, in the final expression. It is easier to perceive the function of the filter physically when expressed in f, while it is easier to carry out filter calculations when expressed in ω. The choice of one or the other is a personal matter.

The transfer function tells us that this filter attenuates all signal components in $V_{in1}(\omega)$ with frequencies larger than 3.4 KHz, by more than 3-dB. In fact, this LPF is affected by the presence of 560 K resistor R17. The cut-off frequency is slightly larger than 3.4 KHz. We discuss how to handle this effect later in this chapter.

If we interchange the positions of resistor and capacitor in Figure 3.3 (a), the circuit in Figure 3.3 (b) comes out. This is also a basic filter block, and it is called *RC HPF*. Consider the circuit depicted in Figure 3.5. This is the HPF right after the microphone in Audio/TX circuit.

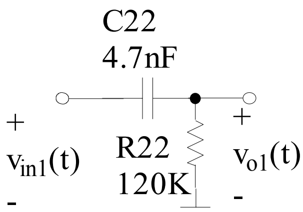


Figure 3.5 Audio/TX HPF

The total impedance that appears across $V_{in1}(\omega)$, Z, is

$$Z = R_{22}+1/j\omega C_{22}$$

in this circuit. Since the current in the circuit is $V_{in1}(\omega)/Z$,

$$V_{o1}(\omega) = R_{22}\{V_{in1}(\omega)/Z\}$$

$$= V_{in1}(\omega)\, R_{22}/(R_{22}+1/j\omega C_{22}).$$

The transfer function is

$$H(\omega) = V_{o1}(\omega)/V_{in1}(\omega)$$
$$= j\omega C_{22}R_{22}/(1+ j\omega C_{22}R_{22}).$$

The magnitude and phase of the transfer function of a RC HPF is depicted in Figure 3.3 (b). HPF attenuates all signal components with frequencies less than f_c by more than 3-dB.

$RC = R_{22}C_{22} = 5.64E-4$ sec. and $f_c = 282$ Hz in the circuit of Figure 3.5. Hence

$$H(\omega) = jf/282/(1+jf/282),$$

and this HPF attenuates all components with frequencies less than approximately 300 Hz. It is important to notice that H(0)=0. This means that this HPF never passes d.c.

voltages. The series component is a capacitor, and its reactance is $X_C(\omega) = 1/\omega C$. Thus $X_C(0) = \infty\ \Omega$. *A series capacitor stops d.c. voltage completely.*

3.3. Sources and Equivalent Circuits

An integral impedance quite often accompanies real sources, voltage or current supplies. Batteries, for example, always have an internal resistance, and they can be modeled as a series combination of a voltage source and a resistor. The equivalent circuit of an AA size 1.5 V battery is given in Figure 3.6 (a).

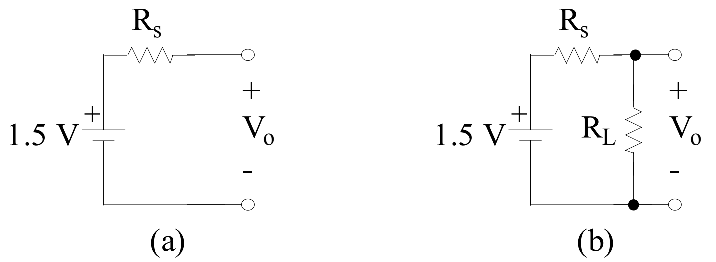


Figure 3.6 Battery equivalent circuit. (a) Unloaded, and (b) Loaded

R_s is the source resistance (the internal resistance of the battery). When there is not anything connected to the battery, the voltage that we measure across its terminals, using a multimeter, is its nominal voltage 1.5V. However when a load is connected across its terminals, like a light bulb or a radio, there will be a current flowing in the circuit. A load resistance connected to the battery is shown in Figure 3.6 (b). This current is $I = 1.5/(R_s+R_L)$ amps. The voltage that we shall measure across the battery terminals is now equal to the voltage across the load, which is

$V_o = 1.5\ R_L/(R_s+R_L)$ volts.

This value is always less than 1.5 V, the nominal voltage of the battery. Indeed $1.5\ R_s/(R_s+R_L)$ volts drops across the source resistance and the voltage that appears across battery terminals is $1.5 – I\ R_s$.

Note that here we have modeled a real voltage source by an *ideal* voltage source and a *series* source impedance. This is called *Thevenin equivalent circuit*. This model is applicable to all real voltage sources.

The real current sources can also be modeled similarly. An ideal current source, which can supply the designated current to the circuit connected to it whatever the voltage across it may be, and a *parallel (source) impedance* which takes off some current from it, model a real source accurately. This model is called the *Norton equivalent circuit* and is shown in Figure 3.7 (a). Note that the source impedance is Z_s, which can be complex, in this circuit.

A real current source can also be modeled as a voltage source and used in circuit analysis. In order to find the *voltage equivalent* of the source in Figure 3.7 (a), we have to calculate the *open circuit output voltage*, which is V_o in this circuit:

$V_o= Z_sI_N(\omega)$.

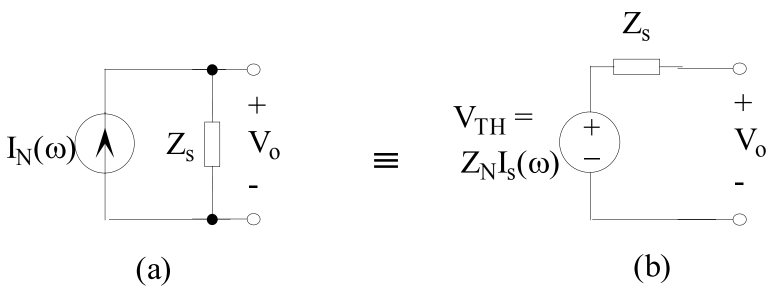


Figure 3.7 (a) Norton current source equivalent circuit, (b) its Thevenin equivalent voltage source

Open circuit output voltage is called the *Thevenin equivalent voltage* and is equal to the value of the ideal voltage source in the equivalent circuit. The series impedance of the new voltage source is the open circuit impedance of the circuit at hand. This means that first we must set the value of the ideal source in the circuit to zero, (in this circuit we have only $I_N(\omega)$, hence set $I_N(\omega) = 0$), and then find the impedance across the terminals. When a current source is set to zero, it exhibits an open circuit, since no current can flow through its terminals. Hence Z_s is the only impedance that appears across V_o. Thevenin equivalent circuit of the circuit in Figure 3.7 (a) comprises a voltage source $V_{TH} = Z_sI_N(\omega)$ and a series impedance Z_s. This model is given in Figure 3.7 (b).

Thevenin and Norton equivalent circuits are used extensively in circuit analysis. These circuits provide a very efficient way to simplify the circuit to be analyzed. Whenever a piece of linear circuit is connected to another circuit through two terminals, the equivalent circuit analysis is often the simplest way to understand how the latter is affected. Consider the circuit in Figure 3.8. As far as circuit 2 is concerned, every effect of first circuit is summarized by its equivalent circuit at the interconnection terminals.

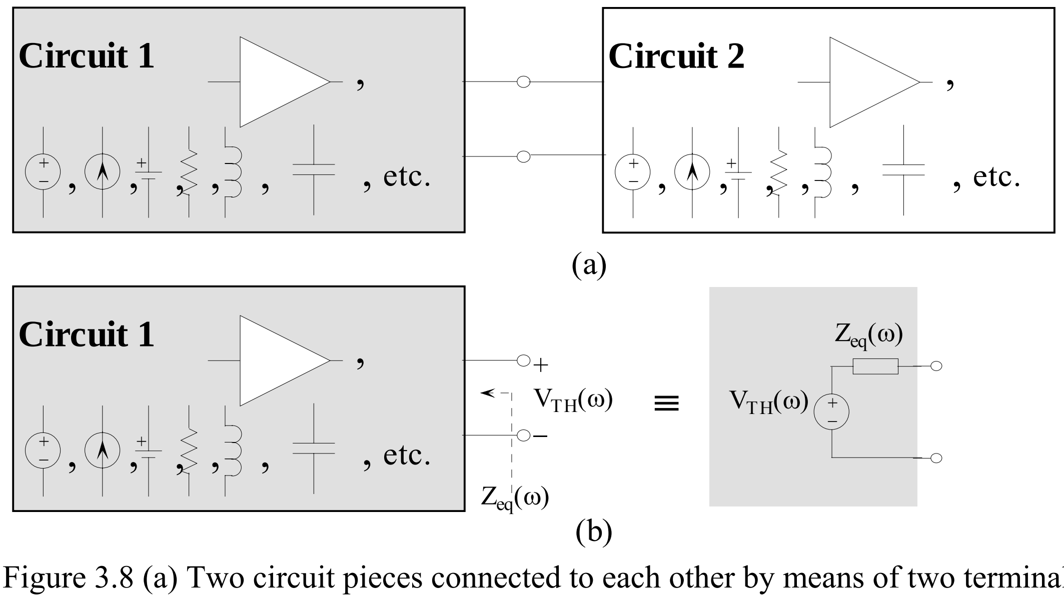


Figure 3.8 (a) Two circuit pieces connected to each other by means of two terminals, (b) Thevenin equivalent of circuit 1.

Thevenin equivalent voltage, $V_{TH}(\omega)$, is the *open circuit voltage* that appears across the interconnection terminals. This means that the two circuits must first be *disconnected*, and then the voltage that appears across the terminal is calculated as $V_{TH}(\omega)$. $Z_{eq}(\omega)$, on the other hand is calculated as follows:

1. First all sources in the circuit 1 are set to zero, which means that voltage sources are shorted and current sources are open circuited, i.e. removed.
2. Then the impedance that appears across the disconnected terminals is calculated as $Z_{eq}(\omega)$.

Once Thevenin equivalent circuit is obtained, circuit 1 can be replaced by its equivalent as in Figure 3.9 and circuit 2 can be analyzed.

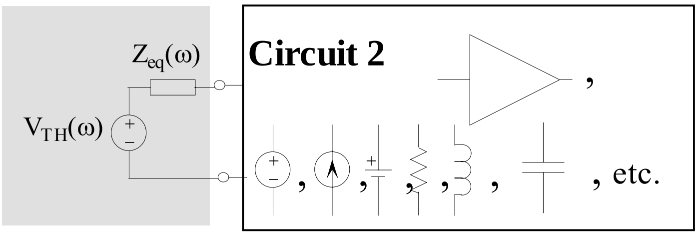


Figure 3.9 Thevenin equivalent of circuit1 connected to circuit 2.

Same approach is also possible using Norton equivalent of circuit 1. The equivalent impedance of Norton equivalent circuit is the same as that of Thevenin equivalent circuit. Norton equivalent current source, however, is calculated by first shorting these the disconnected terminals of circuit 1, as in Figure 3.10. The short circuit current calculated is the Norton equivalent current, $I_N(\omega)$. The equivalent circuit becomes a parallel combination of this current and the equivalent impedance, $Z_{eq}(\omega)$.

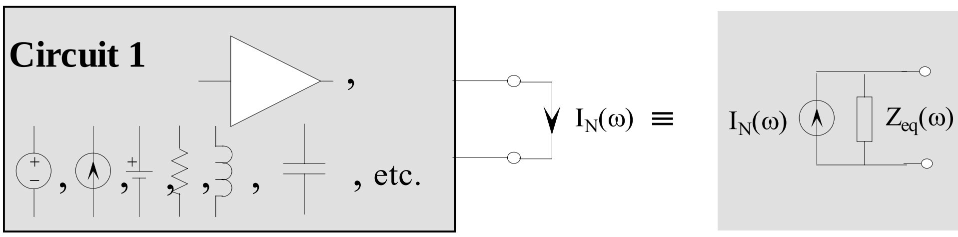


Figure 3.10 Norton equivalent circuit

Thevenin and Norton equivalent circuits are also equivalent. In other words, open circuit voltage of Norton circuit, $I_N(\omega)Z_{eq}(\omega)$, yields $V_{TH}(\omega)$, as discussed above. Hence, $Z_{eq}(\omega)$ can also be obtained by first calculating the open circuit voltage $V_{TH}(\omega)$ and short circuit current $I_N(\omega)$, and then calculating their ratio as

$$Z_{eq}(\omega) = V_{TH}(\omega)/ I_N(\omega).$$

Consider the circuit in Figure 3.11. Voltage across 2.7 k resistor can be found in many different ways.

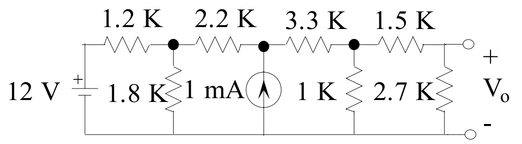


Figure 3.11

One of the most effective ways of solving such problems is depicted in Figure 3.12. First, the Thevenin equivalent of a part of the circuit, which contains the battery, is evaluated in the starting step. Note that the equivalent voltage is the open circuit voltage across 1.8 K resistor and the equivalent resistor is 1.2K // 1.8K, in this part. This equivalent circuit is connected to the remaining part and a Norton circuit is evaluated. To do this we disconnect the part of the circuit at the dotted line and short it. The short circuit current is equivalent current and the equivalent resistor is 0.72K+2.2K. In third step, we Notice that the equivalent current and 1 mA current source are in parallel. Combining them in a single source of 3.47 mA, we convert the Norton circuit to its Thevenin equivalent. After another Thevenin equivalent conversion, output voltage is obtained as 0.9V.

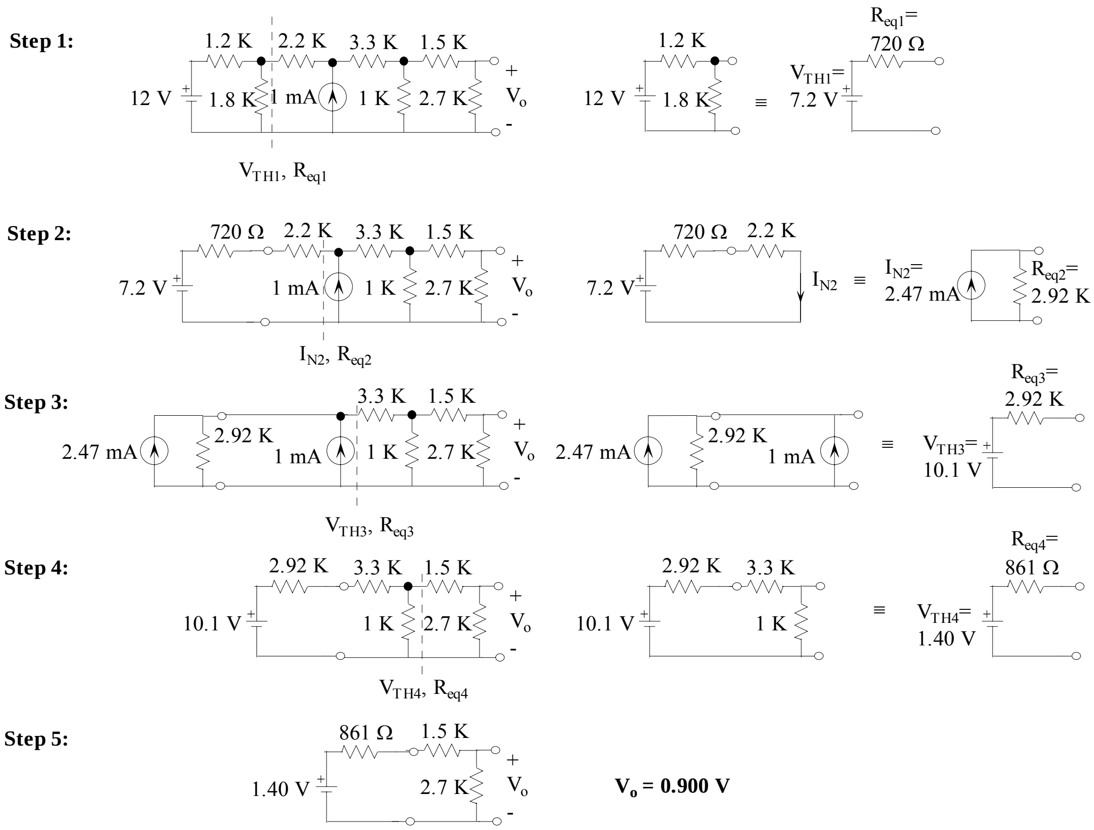


Figure 3.12 Analysis of circuit s using equivalent circuits.

Now consider the HPF in Figure 3.5 again. Assume that this filter is driven by a current source $I_s(\omega)$ (see Laboratory exercise 1), which has a parallel impedance of R_s=1.2K (R20 in the Audio/TX circuit). HPF is redrawn in Figure 3.13 (a) together with the source.

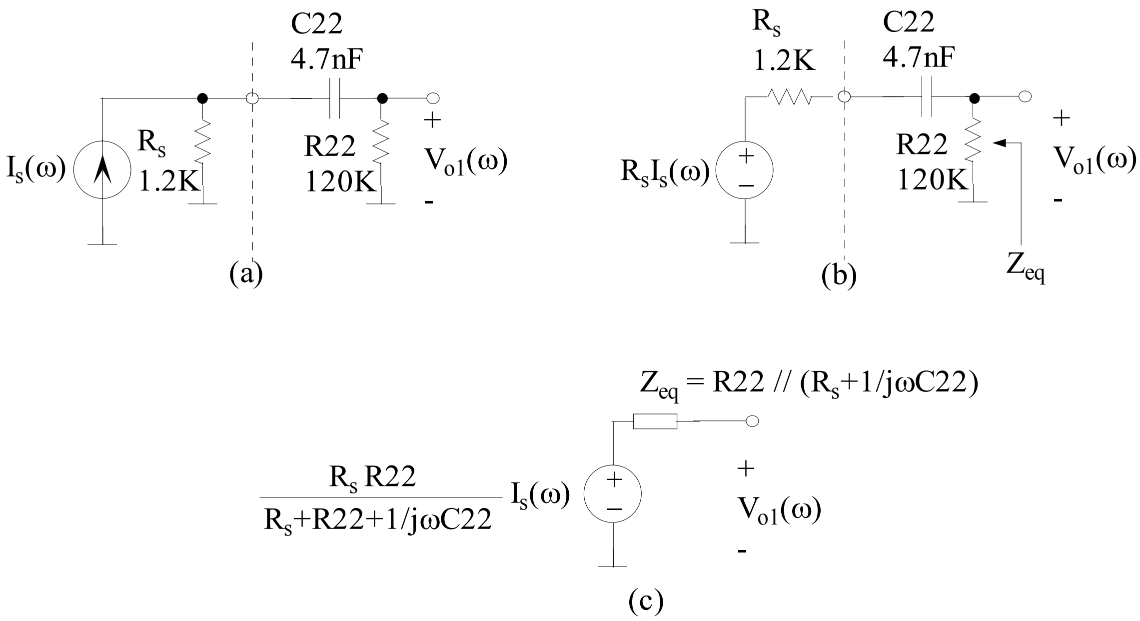


Figure 3.13 (a) Microphone input circuit, (b) and (c) Thevenin equivalent circuits

Conversion of the current source and the parallel resistance into its Thevenin equivalent yields the circuit in Figure 3.13 (b).

If the only thing that we are interested in is the voltage applied to the circuit following the terminals of V_{o1}, we can convert all of the circuit in Figure 3.13 (a), into an equivalent circuit. The open circuit voltage at the output terminals is,

$$V_{o1}(\omega) = R_sI_s(\omega)\ R22 / (R22+R_s+1/j\omega C22)$$

in Figure 3.13 (b). Since R_s<<R22

$$\begin{aligned} V_{o1}(\omega) &\approx R_sI_s(\omega)\ j\omega C22\ R22 / (1+ j\omega C22\ R22) \\ &= I_s(\omega)R_s\times jf / \{282\times(1+jf/282)\}. \end{aligned}$$

The equivalent resistance is found by first setting the source to zero, i.e. $R_sI_s(\omega) = 0$. Note that now the supply is a voltage source. Killing a voltage source means that the potential difference between the terminals it is connected is set to zero, i.e. *shorted*. The total impedance that appears across the output terminals, with supply shorted, is R22//(R_s+1/jωC22),

$$\begin{aligned} Z_{eq}(\omega) &= R22(\ 1+j\omega C22\ R_s) /[1+ j\omega C22(R_s+R22)] \\ &\approx R22(\ 1+j\omega C22\ R_s) /(1+ j\omega C22R22) \\ &= 120\ (\ 1+j\omega\times 5.64E{-}6) /(1+ j\omega\times 5.64E{-}4)\ K\Omega. \end{aligned}$$

For example, at f = 1 KHz, this impedance is

$$Z_{eq}(2\pi\times 1E3) \approx 10K\Omega - j31\ K\Omega\ .$$

Z_{eq} looks like a 10K resistor connected in series with a 5.1 nF capacitor, at 1 KHz.

3.4. Operational amplifiers

Operational amplifiers (*OPAMP*) are general-purpose integrated circuit amplifiers. Their characteristics are nearly ideal amplifier characteristics within their designated operating conditions.

The symbol of an OPAMP is given in Figure 3.14 (a).

It is important to introduce the *ideal OPAMP* concept, because quite often we are allowed to use the ideal model. The impedance between the two inputs and the voltage gain of an ideal OPAMP are both infinite (∞). Also the series output impedance of an ideal OPAMP is zero Ω. The equivalent circuit of an ideal OPAMP is shown in Figure 3.14 (b).

A real OPAMP, on the other hand, has input impedance R_{in}, output impedance R_{out}, and a finite gain A.

Notice that there is a voltage source with the value of "A ($V_{in1}-V_{in2}$)" at the output in the model. Such sources are called *controlled sources*, because its value is determined by some parameter in the circuit, $V_{in1}-V_{in2}$ in this case. Otherwise they are ideal sources.

We use two types of OPAMP, TL082 and LM7171, in TRC-10. TL082 is an audio OPAMP and has the following parameters at low frequencies:

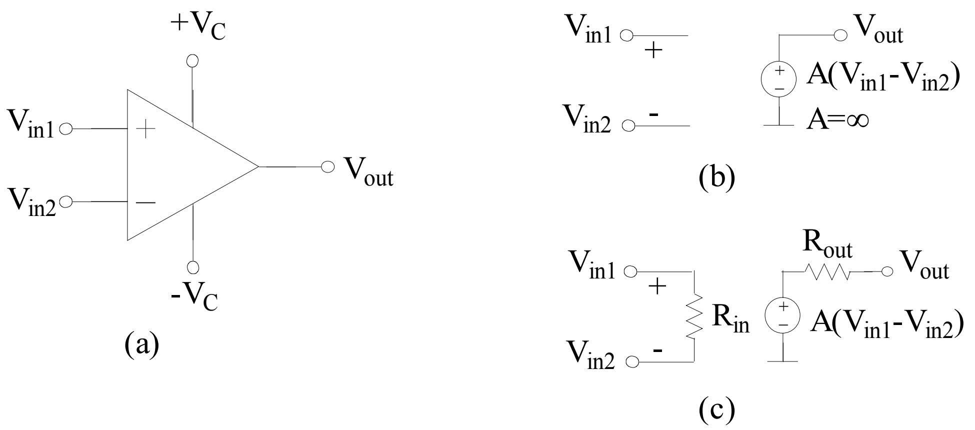


Figure 3.14 OPAMP (a) OPAMP symbol, (b) ideal OPAMP equivalent circuit, and (c) real OPAMP equivalent circuit

$R_{in} \approx 1$ E12 Ω = 1 tera Ω
$R_{out} \approx 10$ Ω
$A \approx 3.0$ E5

These values are close to that of an ideal OPAMP for many applications.

OPAMPs are always used with peripheral circuits for amplification of signals. An inverting amplifier circuit is given in Figure 3.15 (a).

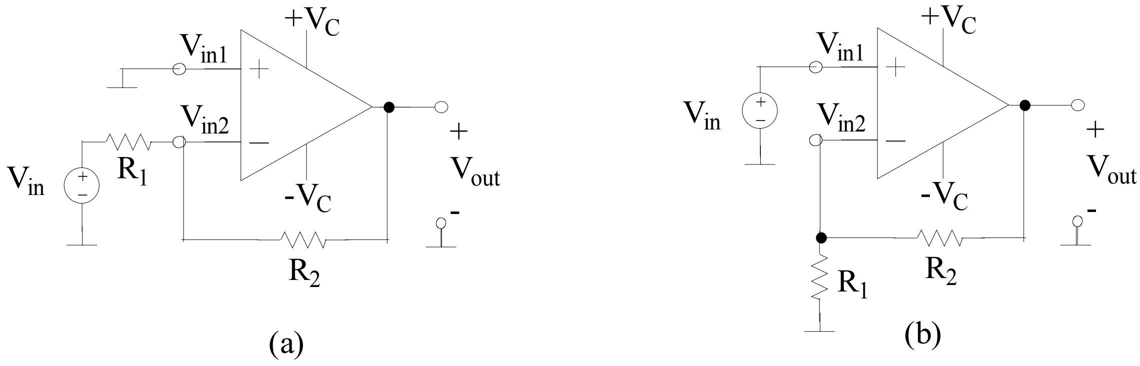


Figure 3.15 OPAMP (a) inverting amplifier, and (b) non-inverting amplifier

We first make some observations on the inverting amplifier circuit, in order to analyze it. The + input of the OPAMP is connected to ground, therefore $V_{in1} = 0$. Hence $V_{in1} - V_{in2} = -V_{in2}$. Let us assume that both resistances R_1 and R_2 are >> R_{out} and << R_{in}. Now we can employ the model of the ideal OPAMP and construct the equivalent circuit of inverting amplifier, in Figure 3.16.

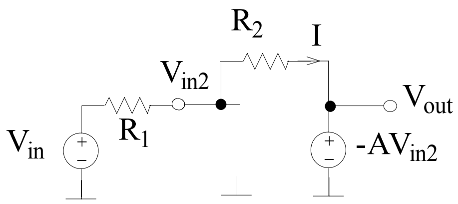


Figure 3.16 Equivalent circuit of inverting amplifier

There are two voltage sources in the circuit of Figure 3.16, and we want to find V_{out} in terms of V_{in}. Current I must be determined first:

$$I = [V_{in} - (-A\ V_{in2})]/(R_1 + R_2)$$
$$= (V_{in} + A\ V_{in2})/(R_1 + R_2).$$

But,

$$V_{in2} = V_{in} - I\ R_1 = V_{in} - R_1(V_{in} + A\ V_{in2})/(R_1 + R_2),$$

$$V_{in2}[1 + A\ R_1/(R_1 + R_2)] = V_{in}[1 - R_1/(R_1 + R_2)]$$

Solving for V_{in2},

$$V_{in2} = V_{in}\ R_2/(A\ R_1 + R_1 + R_2).$$

Now,

$$V_{out} = -A\ V_{in2} = -\ AV_{in}\ R_2/(A\ R_1 + R_1 + R_2),$$

and

$V_{out}/V_{in} = -\,(R_2/R_1)[A/(A+1+R_2/R_1)]$.

A is very large, $A\to\infty$. If we substitute this very high value of A, we obtain,

$V_{out}/V_{in} = -\,(R_2/R_1)$ for $A\to\infty$.

The overall gain of the inverting amplifier is $-R_2/R_1$. Note that it is negative (and hence the name "inverting") and is specified only by external circuit elements, R_2 and R_1, and not by any parameter of the OPAMP (not even A). Therefore if we want a gain of –100, we choose R_2 and R_1 such that $R_2/R_1=100$.

Couple of more observations:

1. $V_{in2} = V_{in}\, R_2/(A\, R_1+R_1+R_2) \approx 0$ for $A\to\infty$.
2. $I = (V_{in}+A\, V_{in2})/(R_1+R_2) \approx V_{in}/R_1$ for $A\to\infty$.

V_{in2}, indeed $V_{in1}-V_{in2}$, is forced to become almost zero volts by the very large value of A.

We can treat the node where two resistors meet, as ground. This is referred to as *virtual ground*. Knowing these facts, we can analyze the circuit in a simpler way:

$V_{in2} \approx 0$,
$I \approx V_{in}/R_1$,
$V_{out} = V_{in2} - I\, R_2 = -I\, R_2$
$= -V_{in}\, R_2/R_1$.

We used the concept of *feedback* in this amplifier. R_2 is connected between the output of the amplifier and its negative input. This resistor *feeds back* a sample of the output signal to its input and R_2 is usually called *feedback resistor*. It is only by feedback that we can control the gain of OPAMPs.

Two resistors, R_1 and R_2 determine the amount of feedback. Let us write V_{in2} in terms of V_{in} and V_{out} from Figure 3.16:

$$V_{in2} = V_{in} - IR_1 = V_{in} - R_1(V_{in} - V_{out})/(R_1+R_2)$$

$$= V_{in}\, R_2/(R_1+R_2) + V_{out}\, R_1/(R_1+R_2).$$

The amount of feedback, the feedback ratio, is $R_1/(R_1+R_2)$.

The above equation tells us that R_1 and R_2 behave like a *divider*; they divide V_{in} by $R_2/(R_1+R_2)$ and V_{out} by $R_1/(R_1+R_2)$. Impedances connected as in Figure 3.17 are called voltage dividers.

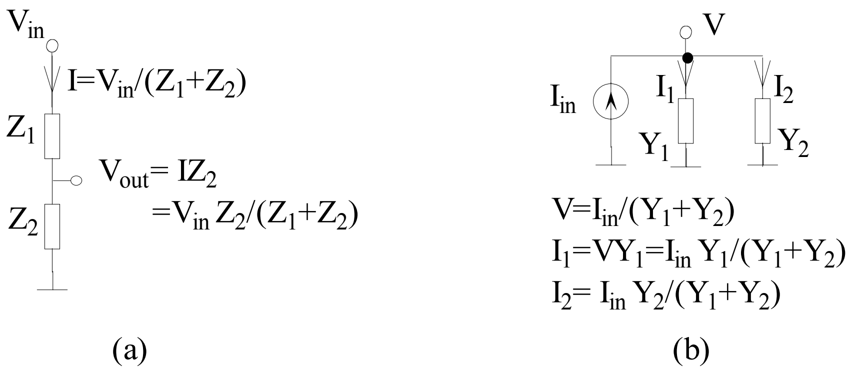


Figure 3.17 Dividers. (a) Voltage divider, and (b) Current divider

Another important observation in above equation is that the contribution of two sources, V_{in} and V_{out}, are simply added to obtain V_{in2}. This is a property of linear circuits, and it is called *superposition*. Whenever we want to calculate a branch current or a node voltage in a circuit which contains more than one source, we find the contribution of each source individually, with all other sources killed (set to zero), and add up these individual contributions. The above equation can also be obtained as follows:

1. Kill V_{out} (i.e. set V_{out}=0) and find the contribution of V_{in} on V_{in2} as $V_{in}R_2/(R_1+ R_2)$ (V_{in} divided by $R_1$and R_2);
2. Kill V_{in} (i.e. set V_{in}=0) and find the contribution of V_{out} on V_{in2} as $V_{out}R_1/(R_1+ R_2)$ (V_{out} divided by $R_1$and R_2);
3. Sum them up to find V_{in2}.

The non-inverting amplifier of Figure 3.15 (b) has similar properties. The input voltage is applied directly to the + input of the OPAMP while R_1 is terminated by ground in this circuit. Using voltage divider relation,V_{in2} becomes,

$$\begin{aligned} V_{in2} &= (0)\, R_2/(R_1+ R_2)+ V_{out}\, R_1/(R_1+ R_2) \\ &= V_{out}\, R_1/(R_1+ R_2). \end{aligned}$$

Since

$$V_{out} = A(V_{in1}- V_{in2}) = A[V_{in} - V_{out}\, R_1/(R_1+ R_2)],$$

solving for V_{out}, we get

$$V_{out} = V_{in}\, A(R_1+ R_2)/(A\, R_1+R_1+ R_2)$$

$$= V_{in}\, (1+ R_2/ R_1) \text{ for } A\rightarrow \infty.$$

The overall gain in non-inverting OPAMP is $(1+ R_2/ R_1)$ and it is positive.

Notice that we provide feedback always to the *negative* input of OPAMP in amplifier applications. This type of feedback is called *negative feedback*.

A particular OPAMP application is non-inverting amplifier with unity feedback, as shown in Figure 3.18.

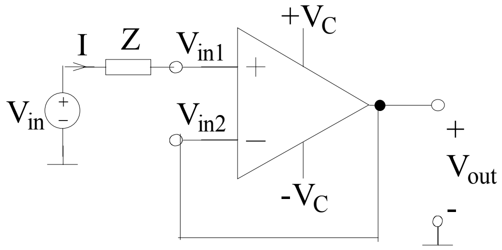


Figure 3.18 Unity gain amplifier

The first observation in this circuit is the input current is

$I = (V_{in} - V_{in1})/(Z+R_{in}) = 0$ for $R_{in} \rightarrow \infty$

where R_{in} is input impedance of the OPAMP. The input current is almost zero in non-inverting amplifiers because of the very high input impedance of OPAMP. The source impedance of the input voltage, Z in this case, does not have any effect on circuit parameters.

Now since,

$V_{in1} = V_{in}$,

$V_{in2} = V_{out}$,

and

$V_{out} = A(V_{in1} - V_{in2}) = A(V_{in} - V_{out})$,

solving for V_{out}, we get

$$\begin{aligned} V_{out} &= V_{in}A/(A+1) \\ &= V_{in} \text{ for } A \rightarrow \infty \end{aligned}$$

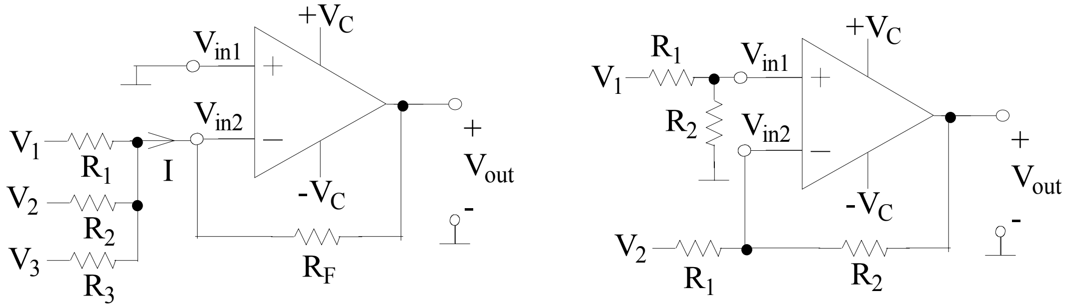
V_{out} is equal to V_{in} in this configuration. This amplifier is called by a few names such as, *unity gain amplifier*, *buffer amplifier* and *voltage follower*. Although it does not provide any voltage gain to the input signal, it is used to transfer the input voltage intact to the output while altering the impedance that appears at the terminals of V_{in} to the low output impedance of OPAMP. This can provide a large power gain, because the voltage at the source can now be applied to a relatively low impedance load.

The input power in the circuit of Figure 3.18 is essentially zero since V_{in} appears across an indefinitely large impedance. If a resistance of 50 Ω is connected across the output of the amplifier, however, the power delivered to this impedance is $(V_{in})^2/[2(50\ \Omega)]$. The power gain in this circuit is infinite, although there is no voltage gain.

3.5. OPAMP circuits

OPAMP is a basic building block in analog signal processing. We studied one of its applications, amplification, in the previous section. We can sum up signals and subtract signals using OPAMPs. A summing amplifier and a difference amplifier are given in Figure 3.19 (a) and (b), respectively.

The resistances determine the weighing coefficients when both summing and subtracting.



$V_{in2} \approx V_{in1} = 0$

$I = V_1/R_1 + V_2/R_2 + V_3/R_3$

$V_{out} = -(V_1 R_F/R_1 + V_2 R_F/R_2 + V_3 R_F/R_3)$

(a)

$V_{in1} = V_1 R_2/(R_1+R_2)$

$V_{in2} = V_2 R_2/(R_1+R_2) + V_{out} R_1/(R_1+R_2)$

$V_{out} = V_1 R_2/R_1 - V_2 R_2/R_1$

(b)

Figure 3.19 (a) Summing amplifier, and (b) Difference amplifier

The signals can also be differentiated and integrated in time domain, by using OPAMPs. Consider the circuit in Figure 3.20 (a). Since the V_{in2} node is virtual ground, i.e. V_{in2} =0, capacitor current is

$I_C(\omega)=j\omega C V_{in}(\omega)$, or

$i_C(t)= C dv_{in}(t)/dt$.

Therefore the differentiator output becomes

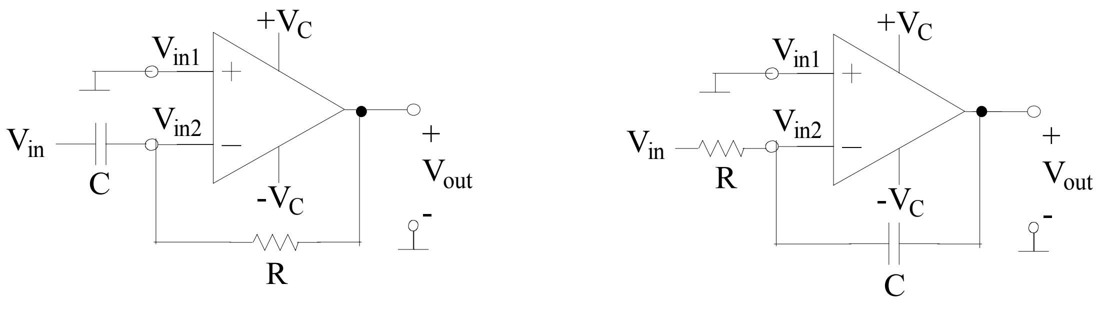
$$\begin{aligned} v_{out}(t) &= -\, R i_C(t) \\ &= -\, RC dv_{in}(t)/dt. \end{aligned}$$

Similarly, for the integrator, the resistor current is

$i_R(t) \quad = v_{in}(t)/R.$

$v_{out}(t)$ is the voltage generated on the capacitor by $-i_R(t)$, hence,

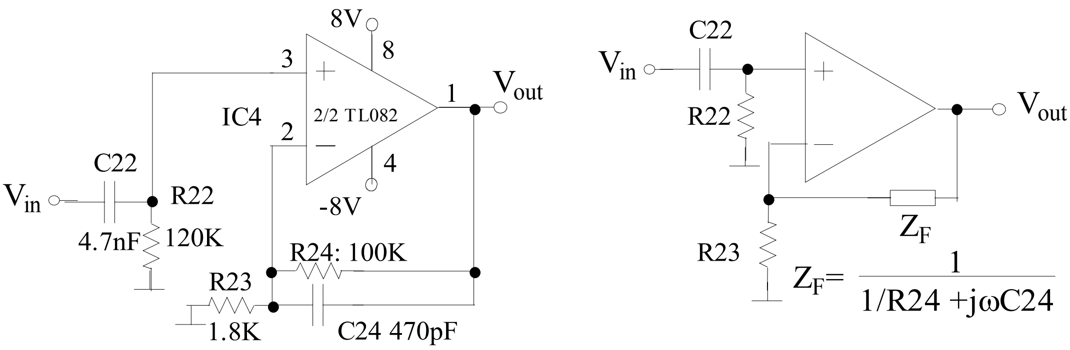
$$\begin{aligned} v_{out}(t) &= 1/C \int\{-i_R(t)\}dt \\ &= -1/(RC) \int v_{in}(t)dt. \end{aligned}$$



(a) (a)

Figure 3.20 (a) Differentiator, and (b) Integrator

With their very high input impedance, low output impedance and very high gain, OPAMPs are very instrumental in designing LP, HP and BP filters, using only resistors and capacitors. Consider the circuit in Figure 3.21, where a part of the microphone amplifier of TRC-10 is shown.



(a) (b)

Figure 3.21 Microphone amplifier (a) the actual circuit, and (b) the simplified circuit

TL082 is a *dual OPAMP*, i.e. there are two identical OPAMPs in the same 8-pin integrated circuit. We use both OPAMPs in Audio/TX circuit. “1/2 TL082” marked on the OPAMP in circuit diagram tells that we use the first OPAMP in integrated circuit IC4, which is TL082. The amplifier is configured in a non-inverting form where input signal is provided to the + input through a HPF. We examined this HPF above. The voltage that appears at the node where C22 and R22are connected, is,

$$V_{in1}(\omega) = V_{in}(\omega)\, j\omega C22\ R22/(1+ j\omega C22\ R22)$$
$$= V_{in}(\omega)\{jf/282/(1+jf/282)\}.$$

The feedback impedance, Z_F, is a parallel combination of a resistor and a capacitor. V_{out} can be obtained as

$$V_{out} = V_{in1}(1+ Z_F/R23) \text{ for } A \rightarrow \infty$$
$$= V_{in1}[1+ (R24/R23)/(1+j\omega R24C24)].$$

The time constant R24×C24 is 47 E−6 sec. Hence,

$$V_{out} = V_{in}(\omega) \times \frac{jf/282}{(1+jf/282)} \times \left[\frac{55.6}{(1+jf/3.4\ E3)} + 1 \right]$$

Note that the first part is a HPF and second part due to feedback impedance is a LPF with a cut off frequency at 3.4 KHz. The two filters are combined in this circuit, together with a mid-band gain of 56.6, or 35 dB.

3.5.1. Offset voltage in OPAMPs

Real OPAMPs are made of many transistors and resistors combined in an integrated circuit. OPAMPs consume electrical energy to provide this useful amplification. Both inputs of OPAMPs are kept at zero potential using proper design techniques. However, due to minor imperfections in the production process, certain imbalance occurs in the internal circuits of OPAMPs. One result of such imbalance is *input offset voltage*.

Input offset voltage is the differential d.c. input voltage, which inherently exists in every OPAMP. Input offset voltage can be modeled as in Figure 3.22.

V_{inos} represents the input offset voltage. Note that V_{out} is $A(V_{in1} - V_{in2})$ and is equal to $-AV_{inos}$ in this model, when $V_{in1} = V_{in2} = 0$. Maximum value of $|V_{inos}|$ is reported in the data sheets of OPAMPs. V_{inos} can be positive or negative. We can also model it as connected in series with positive input.

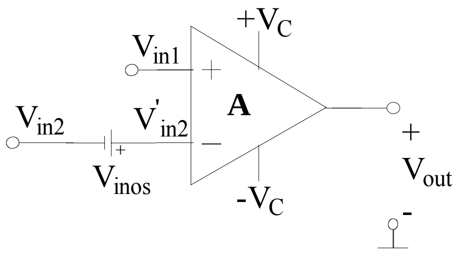


Figure 3.22 Input offset voltage

The effect of input offset voltage on output when there is feedback is considered in Figure 3.23. Notice that both inverting and non-inverting amplifier configurations given in Figure 3.15 reduces to this circuit when $V_{in} = 0$.

Here V_{oos} is the output offset voltage created by the OPAMP when there is no applied input voltage. We must express V_{oos} in terms of R_1, R_2 and V_{inos}. We start by writing V_{in2} as

$V_{in2} = [R_1/(R_1 + R_2)]\ V_{oos}$.

Hence

$V_{oos} = (1+R_2/R_1)\ V_{in2}$.

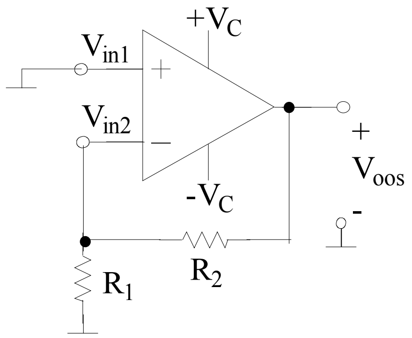


Figure 3.23 Offset voltage in feedback amplifier

Since $V_{in1} = 0$ and $V_{inos} = V_{in1} - V_{in2}$ with no externally applied input voltage, V_{oos} becomes

$V_{oos} = -\ (1+R_2/R_1)\ V_{inos}$.

V_{inos} is multiplied by the gain of the feedback amplifier to yield the output offset voltage.

It is possible to compensate for V_{oos} by an external circuit, which adds an appropriate amount of d.c. voltage to one of the inputs. Such compensation is necessary in circuits where very low frequency amplification is targeted, like in some measurement and instrumentation applications. We do not compensate for offset voltage in TRC-10.

$|V_{inos}|$ of TL082 is given in the datasheet as "typical 5mV; maximum 15 mV". The d.c. gain of the microphone amplifier is 1+R24/R23, or 56.6. Hence $|V_{oos}|$ becomes 283 mV typically, or 849 mV at maximum.

3.5.2. OPAMP Linear Voltage Regulator

We have observed in Chapter 2 that a voltage regulator converts a rectified a.c. voltage (and filtered by a capacitor) shown in Figure 3.24

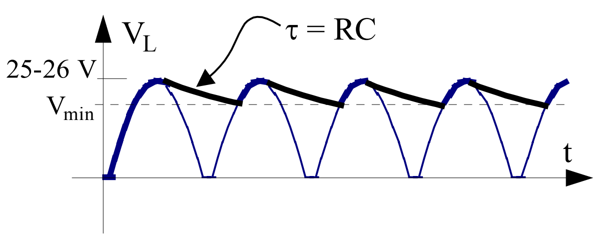


Figure 3.24 Full wave rectified and filtered a.c. supply voltage

to a well defined d.c. voltage, without any ripple, like the one depicted in Figure 3.25.

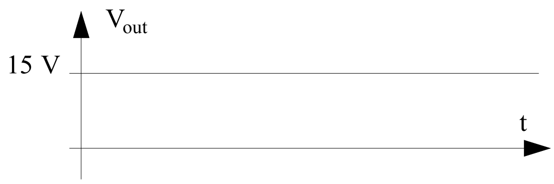


Figure 3.25 Regulated 15 V d.c. voltage

From the data sheet of voltage regulators (e.g. LM7808) we know that V_{min} in Figure 3.24 must be few volts more than the nominal output voltage of the regulator (8 V for LM7808).

Now let us design a voltage regulator using an OPAMP. We need a reliable voltage reference and an OPAMP. A zener diode can be used as voltage reference. To obtain a constant voltage reference using a zener diode, we must *reverse bias* the diode appropriately, as shown in Figure 3.26. In a reverse biased zener diode, a reverse diode current of $-(V_L - V_Z)/R$ flow as long as $V_L > V_Z$. When V_L is less than V_Z, the diode is no longer in the breakdown region, as we discussed in Chapter 2.

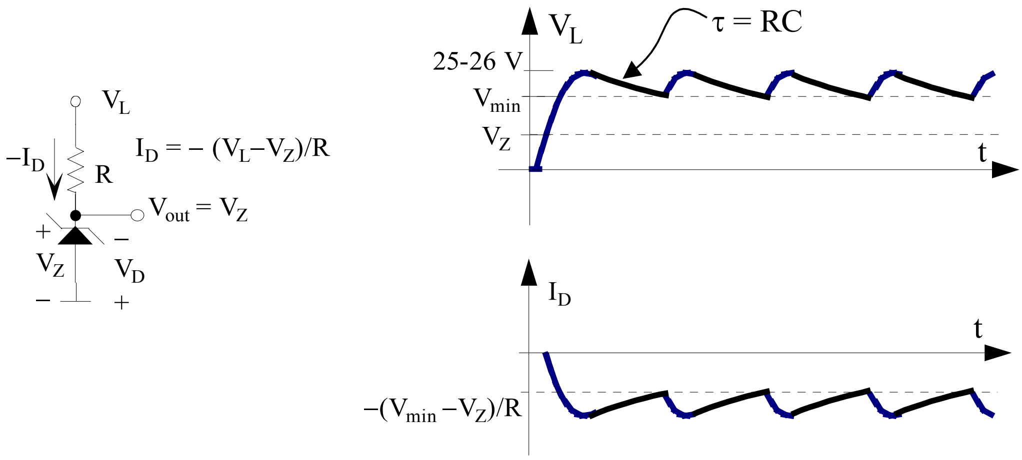


Figure 3.26 Biased zener diode and the current through it

For example for a zener diode with $V_Z = 5.1$ V, $V_{min} = 18$ V and with R = 1 K, -20.9 mA < I_D < -12.9 mA. The performance of this circuit is shown in the Figure 3.27. We obtain a constant voltage of 5.1 V at V_{out}. Note that, although the current through the zener diode varies between -12.9 mA and -20.9 mA, V_{out} is fixed at 5.1 V. This is due to the fact that all this current variation is mapped to the same diode voltage $-V_Z$, by the zener diode characteristics with 90° slope in breakdown region. However if we draw more than 12.9 mA to external circuits connected to V_{out}, the zener diode gets out of breakdown region and we no longer have 5.1 V at V_{out}.

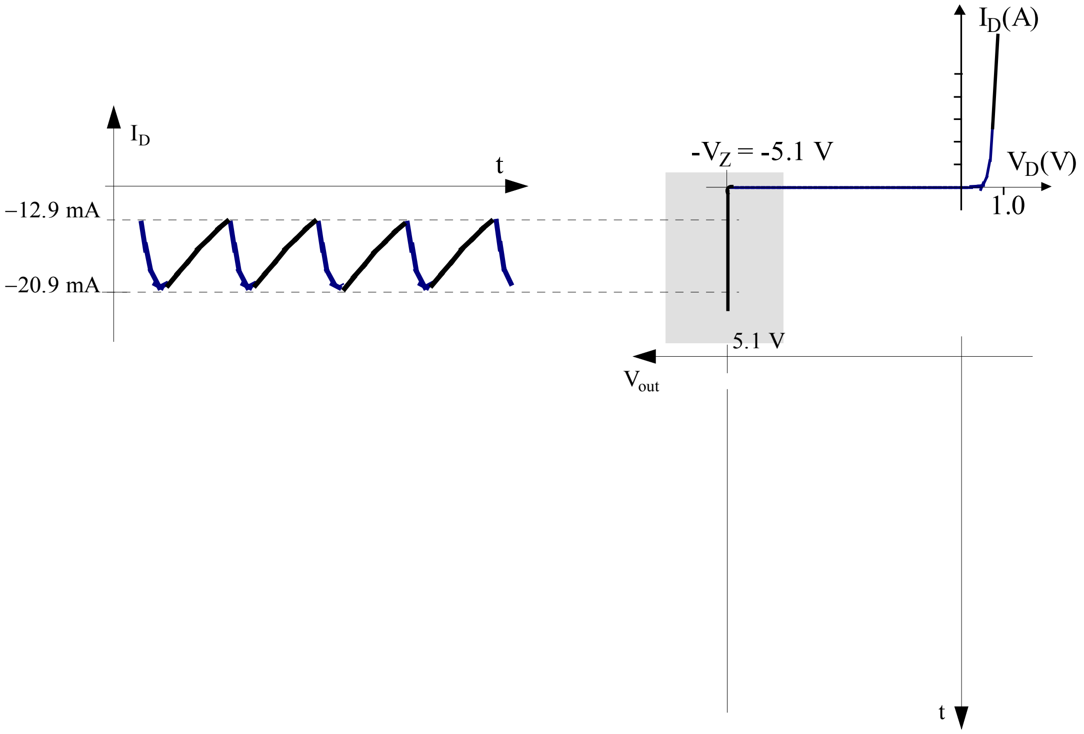


Figure 3.27 Zener voltage is fixed despite the variations on the current

For example, if we connect a load R_L across the diode, we obtain the circuit in Figure 3.28. In this circuit as long as the value of R_L is larger than $V_Z/|I_{Dmin}|$, where I_{Dmin} is $(V_{min}-V_Z)/R$ and 12.9 mA for the above example (minimum value of R_L is 400 Ω approximately), V_{out} remains at V_Z. If R_L is less than this minimum, then the load current exceeds I_{Dmin} and no more supply current can be spared to reverse bias the zener diode. The zener is out of breakdown region and basically no current can flow through it. In this case V_{out} becomes V_{min} $[R_L /(R_L+R)]$, which is 4.1 V for the above example when R_L= 300 Ω. The current I_L becomes 13.8 mA, and zener current is zero.

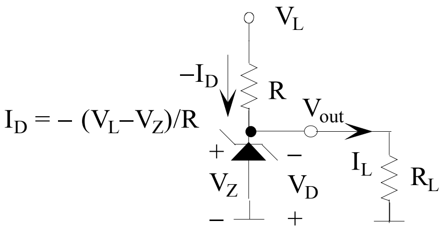


Figure 3.28 Reverse biased zener diode with load current

We can use a voltage follower at this node to improve the amount of current that can be drawn, as shown in the Figure 3.29.

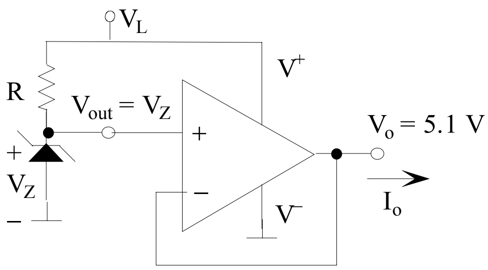


Figure 3.29 Voltage regulator

Note that here the current that can be drawn to external circuits, I_o, is limited only by the output current capacity of the OPAMP. OPAMP output is given by

$$V_o = A(V_Z - V_o) \Rightarrow \quad V_o = [A/(A+1)]\, V_Z \quad \Rightarrow V_o = V_Z \qquad \text{for } A \rightarrow \infty.$$

Also note that supply voltage V_L is not regulated. There is a ripple of about 26–18=8 V on V_L. If we consider an OPAMP like TL082, the *supply voltage rejection ratio* is given in the datasheet as *PSRR = 70 dB* minimum. This means that the ripple of 8 volts in the supply voltage is suppressed by –70 dB and it causes a ripple of at most

$$(8\text{ V})\,(10^{-70/20}) = 2.5\text{ mV}$$

at the output, V_o. This ripple is negligible. Hence we can use V_o as a regulated voltage supply.

3.6. Bibliography

References for circuit theory books given in Chapter 2 are also relevant here.

There are many excellent books on OPAMPs. A comprehensive treatment of OPAMP topics is given in *Op-Amps and Linear Integrated Circuits* by R.A.Gayakwad (Prentice-Hall, 2000).

Audio Engineering Handbook by K. B. Blair (McGraw-Hill, 1988) has a good chapter on microphones and another one on loudspeakers.

The internet paper *Powering Microphones* by T. Engdahl (http://www.hut.fi/Misc/Electronics/circuits/microphone_powering.html) describes how to use electret microphones in electronic circuits in various applications, in his web site (http://www.hut.fi/Misc/Electronics/circuits/#avdoc).

3.7. Laboratory Exercises

Microphone Amplifier

1. The microphone amplifier of TRC-10 is given in Appendix A. We analyzed part of this circuit in this chapter already. C22 and R22 form a HPF, R24 and C24 is a LPF, as well as providing 35 dB mid-band gain together with R23 and OPAMP.

The input signal was assumed to be supplied by a current source in parallel to 1.2K, R20. Actually, the signal is supplied by a microphone, which is a *transducer* that converts acoustic signal to electrical signal. The input circuit then preprocesses the electrical signal.

Many types of microphones are available. An electret condenser microphone is used in TRC-10, because it is suitable for speech applications and available at a very low cost. Electret microphones are exclusively used in all low cost applications, which occupies approximately 90% of the market commercially. Microphones used with sound cards of computers and cellular phones, tie-clip microphones, amateur video camera microphones are all electret type. Electret microphones are in the class of microphones called *condenser microphones*. The basic condenser microphone is a parallel plate capacitor, and is called *externally polarized* condenser microphone. The schematic of this type of microphone is shown in Figure 3.30.

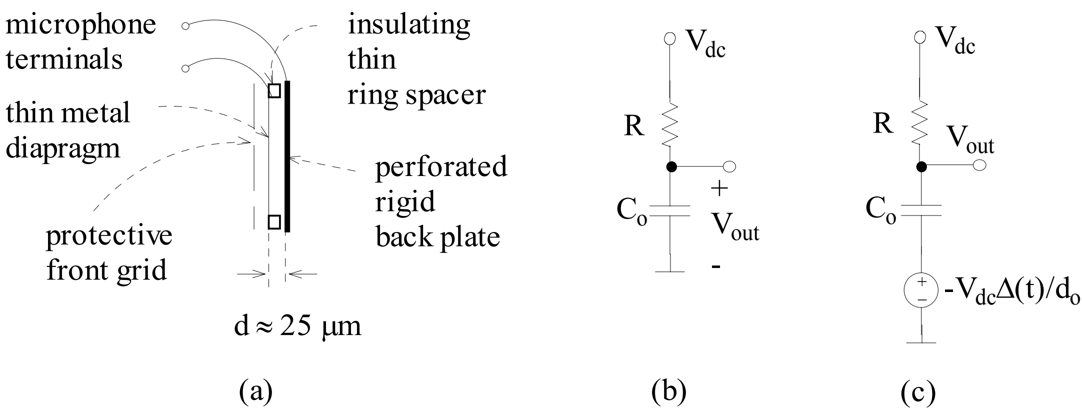


Figure 3.30 Condenser microphone, (a) Mechanical structure, (b) Microphone circuit, and (c) Equivalent circuit

Basic principle of operation of a condenser microphone is simple. When there is no sound, two parallel plates, the diaphragm and the back plate, form a capacitance, C_o. C_o is given as

$$C_o = \varepsilon_o A/d_o,$$

where ε_o is 8.854E-12 F/m, the permittivity of free space, A is the area of the plates and d_o is the spacing of the plates. These microphones are used in circuits like the one shown in Figure 3.30 (b). C_o is charged up to V_{dc} through R. The electric field between the plates is

$$E = V_{dc}/d_o$$

and $V_{out} = V_{dc}$.

When there is sound incident on the diaphragm, sound pressure forces the diaphragm to vibrate back and forth. The spacing is changed by this motion by very small amount and becomes,

$d = d_o + \Delta(t)$,

where $\Delta(t)$ is proportional to the sound signal amplitude, and $\Delta(t) << d_o$. Then the electric field becomes

$$\begin{aligned} E &= V_{out}/d \\ &\approx V_{dc}/[d_o + \Delta(t)] \\ &\approx (V_{dc}/d_o)(1 - \Delta(t)/d_o) \end{aligned}$$

and therefore,

$$\begin{aligned} V_{out} &= Ed \\ &\approx Ed_o \\ &= V_{dc} - V_{dc}\Delta(t)/d_o. \end{aligned}$$

The sound is thus converted to the electrical signal $V_{dc}\Delta(t)/d_o$. The variation $\Delta(t)$ is the membrane displacement and $|\Delta(t)|_{max}$ is at best a small fraction of a micrometer, for ordinary sound levels. Notice that the sensitivity of a microphone depends on the size of d_o: smaller d_o is, more sensitive the microphone is. Furthermore d_o must be kept constant over the lifetime of the microphone, otherwise the sensitivity will change by aging. A compromise between these two requirements is found at about 25 μm in most commercial microphones. The expression for V_{out} above, tells that there is a voltage source $-V_{dc}\Delta(t)/d_o$ additive to the capacitor (i.e. in series with the capacitor). Figure 3.30 (c) shows the overall equivalent circuit.

The microphones must be connected to amplifiers using a length of cable. For a microphone with a surface of 1cm^2, C_o is approximately 35 pF. However the capacitance of a pair of cables used in audio work is at least 40 pF/m. This means that if we try to connect the microphone directly by a cable to the power circuits and the amplifier, we shall reduce the sensitivity severely. Therefore a very high input impedance buffer amplifier is always used within the housing of the microphone to isolate this capacitance from the external circuits.

Condenser microphones usually serve the upper end of the market. Very high precision condenser microphones are made for professional use, and they are expensive. Furthermore, they are vulnerable against corrosion and must be protected in outdoor conditions. A modern variation to condenser microphones is electret microphones. A polymer membrane replaces the vibrating membrane, which is pre-polarized, in these microphones. Polyvinyl fluoride (PVF) polymers (similar to teflon) have a property of keeping a polarizing electric field for very long periods (like 10 years), once they are appropriately polarized by a strong electric field. These polymer film membranes are manufactured very thin, like 25 μm thick, and metal plated (aluminum, nickel or even gold) on both sides, by a technique called *sputtering*. These electret microphones can be manufactured in very large quantities, in automatic manufacturing processes. TRC-10 uses such a microphone. The structure and equivalent circuit of an electret microphone is given in Figure 3.31.

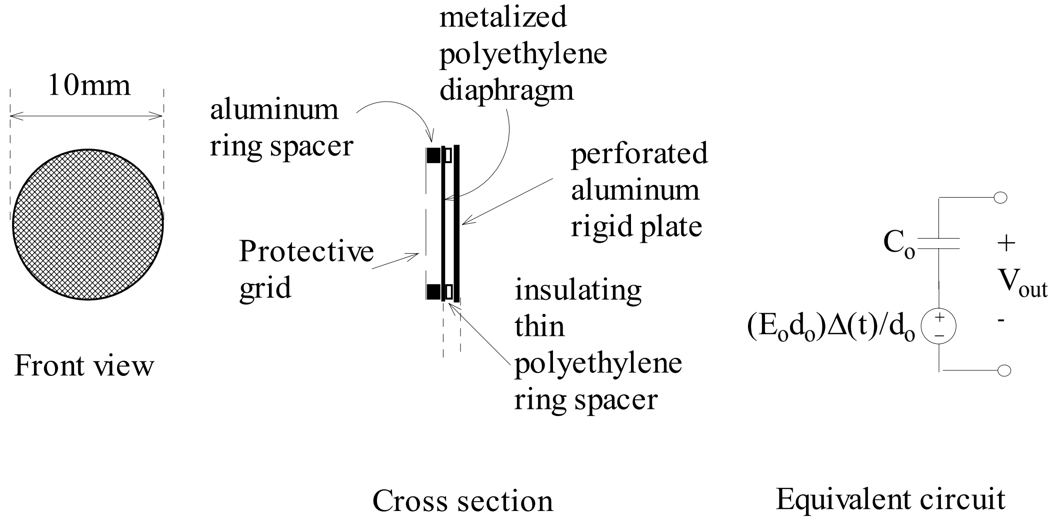


Figure 3.31 Electret condenser microphone

Note here that since the polymer membrane is pre-polarized, we do not have to provide an external d.c. supply, to make the microphone work. The electric field trapped in the membrane, E_o, which is equivalent to V_{dc}/d_o in externally polarized condenser type, enables the microphone to produce the equivalent electrical signal, in absence of any polarizing voltage. This type of microphone also suffers from capacitive output impedance drawback, and a buffer must be used.

Electret microphones are commercially available as capsules, which contain the microphone, the buffer and internal wiring. The electrical model of the microphone used in TRC-10 and its equivalent circuit in is given in Figure 3.32.

The microphone housing contains the microphone and a *field effect transistor* (FET), which is used as buffer. The internal wiring is such that one microphone terminal is connected to the *gate* of FET, and the other (which is also connected to the aluminum case of the housing) is connected to the *source* of the FET. FET must be provided with a voltage supply as shown in the figure in order to act as a buffer amplifier. 8 V d.c. voltage supply and resistors R20 and R21 provide the environment, where FET acts like a *voltage controlled current source*. V_o is the voltage produced by the microphone proportional to the sound pressure, and it is the controlling voltage for the current source. FET converts this into a *current source* output $g_m V_o$, where g_m is a parameter of FET, called *transconductance*. The terminals of the microphone are these two terminals of the FET. We shall therefore model the microphone output as a current source with a current output proportional to the voice signal, when appropriately connected to an external circuit.

Solder the microphone cable leads to two solder contacts on the microphone capsule.

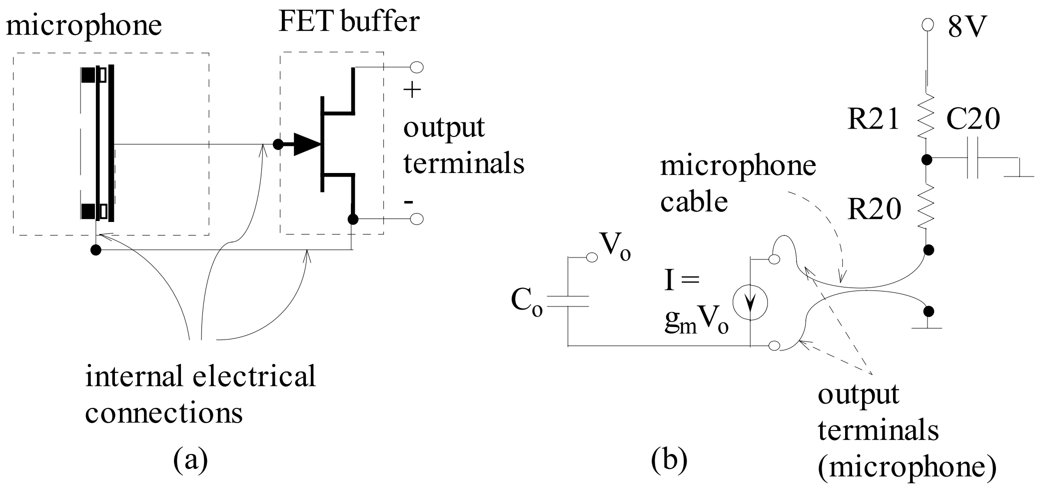


Figure 3.32 Electret condenser microphone of TRC-10 (a) components in the microphone housing, and (b) microphone equivalent circuit and the way it is used in TRC-10.

Cut 3 cm long piece of 6mm-diameter heat shrink sleeve, and work the cable into it. Push sleeve up to the microphone. Shrink the sleeve. Cut a 4 cm long piece of 1.2 cm diameter sleeve and put the microphone into it. The sleeve must cover the side of the microphone completely. Shrink the sleeve. Trim the sleeve, using a pocketknife, on the microphone side.

Solder the other end of the leads to the plug contacts. Make sure that the lead connected to ground on the plug corresponds to the lead soldered to the case contact on microphone.

Solder two 10-cm long cables on the microphone jack contacts. Crimp the contacts to the other end of the two cables for PCB connection. Fit the PCB connector jack J2. Mount the microphone jack on the panel.

Place the TL082 (IC4) on the component side of the PCB, into its holes. Solder all eight pins. Mount and solder capacitors C11, C12, C20, C22, and C24. Mount and solder all resistances R20 to R24. You must first shape the resistor leads into a ⊓ shape of correct size to fit into respective holes properly. Use your long-nose pliers for that.

The microphone preamplifier circuit is now partly finished. Check your connections with the multimeter one by one. Make sure that they match the circuit and the layout. Switch the power ON. Check the supply voltages +8 V and –8 V, with your multimeter. Make sure that you read the same voltages at the supply pins of IC4. Check the d.c. voltage at the output of the OPAMP, pin 1. It must read about few hundred milivolts at most. (Why not zero? See Section 3.5.1.) If you cannot read these voltages, switch the power off and check all your connections and correct them.

C25 forms a HPF with R26. Find the cut-off frequency of this filter. Mount and solder C25 and R26.

We now have the part of the circuit depicted in Figure 3.33 installed. Initially, we shall use the signal generator instead of microphone to supply the signal to the amplifier. In order to make the signal generator look like a current source; we use a 180 K resistor in series with it. Set the output level of the signal generator to 6.8 V pp or 2.4 V rms. **Make sure that *J12 connector is not connected***, because if it is connected, the microphone jack shorts the input of the amplifier at all times.

2. What is the Norton equivalent current source of the circuit in dotted box at the input of the amplifier, in Figure 3.33?

   The amplifier amplifies any interference signal or noise coupled to the microphone terminals. When we provide the supply for microphone buffer, we must make sure that it is a very clean d.c. voltage. Resistor R21 and capacitor C20 filter the +8 V supply. C20 is a tantalum capacitor.

   What kind of filter is the circuit formed by R21 and C20, if the input is +8 V supply terminal, and the output is the test point? What is the cut-off frequency of this filter?

   C11 and C12 are called by-pass capacitors, which are always used at the supply terminals of integrated circuits. Although they do not seem to be functional, they provide energy reserve to meet the demand by the IC when there are short duration high currents generated by the IC. Using by-pass capacitors meets this current demand at the closest point to the IC, instead of drawing that current all the way up from the supply circuit. They are particularly important at the HF part of the circuit.

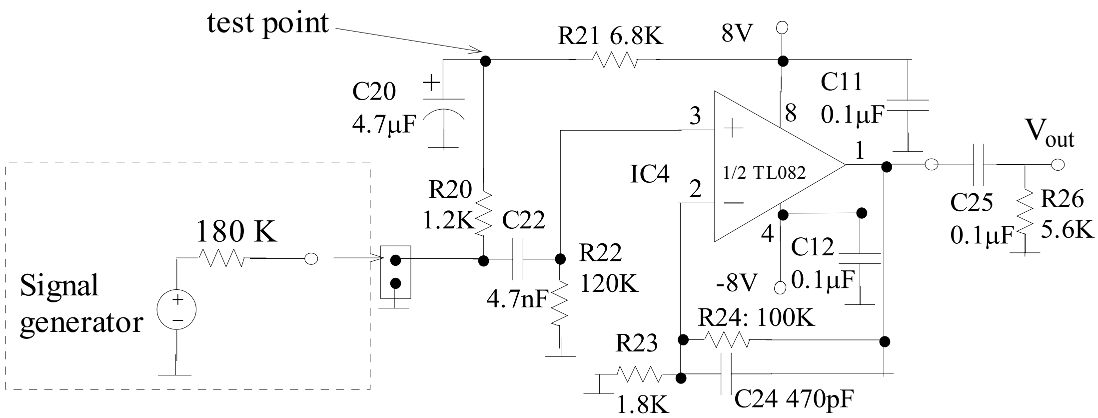


Figure 3.33 Microphone amplifier

The data sheet of TL082 is given in appendix D. Examine this data sheet. What is the lead temperature for soldering? What is the supply current demand of this amplifier from ±8 supply, when the input signal is zero volts? What is the *open loop voltage gain* at 10 KHz as a ratio (not dB)? What is the output impedance at 1 KHz? What is the input impedance?

Calculate the gain of the amplifier in mid-band, i.e. calculate the gain ignoring the effects of C20, C22, C24 and C25. *Ignoring the effect of a capacitance* means, assuming that it is a short circuit if it is connected in series, and it is open circuit if it is connected in parallel. To *ignore* these capacitances we replace C20, C22 and C25 by a short circuit and C24 by an open circuit. Hence redraw the circuit with C20, C22 and C25 replaced by a short circuit and C24 removed. Calculate the gain of this all-resistive circuit. This is the mid-band gain.

Now include C22, C24 and C25 into the circuit. Find the transfer function of the amplifier (it is actually already found in Section 3.5) for the frequency range of 30 Hz to 30 KHz. Calculate the transfer function in dB, and plot the transfer function on a graph paper with logarithmic scales (transfer function on y-axis and frequency on x-axis). 15 to 20 frequency points should be sufficient to show the variation, if these frequencies are chosen appropriately.

3. Compensate both of your oscilloscope probes. First consider the equivalent circuit of scope front end. The oscilloscope channel amplifier and a typical probe can be modeled as shown in Figure 3.34. The input impedance of the channel amplifier is always written on the oscilloscope, just next to the channel amplifier probe connector. Read the values of R_{in} and C_{in}.

   The *attenuation ratio* of the probe is always written on the probe, as “1:1” or “10:1”, etc. Read the attenuation ratio of the probe. Assuming that capacitors do not exist, find the value of R_p such that V_i:V_o is exactly equal to attenuation ratio $R_{in}/(R_{in}+R_p)$. A d.c. impedance measurement can reveal the value of R_p, but the value of R_p is usually too high to be measured by an ordinary multimeter even for a 10:1 probe.

   C_c is the capacitance of probe cable and is parallel to C_{in}. C_o is the compensation capacitance, usually mounted on the probe connector. It is an adjustable capacitor and it is also parallel to C_{in}. Its effect can be better understood if we consider the transfer function from V_i to V_o. Letting $C_T = C_c + C_o + C_{in}$, we can find this transfer function as

   $$V_o(\omega)/V_i(\omega) = [1+(R_p/R_{in})]^{-1}(1+j\omega\, C_pR_p)/(1+j\omega C_T\, R_{in}||R_p).$$

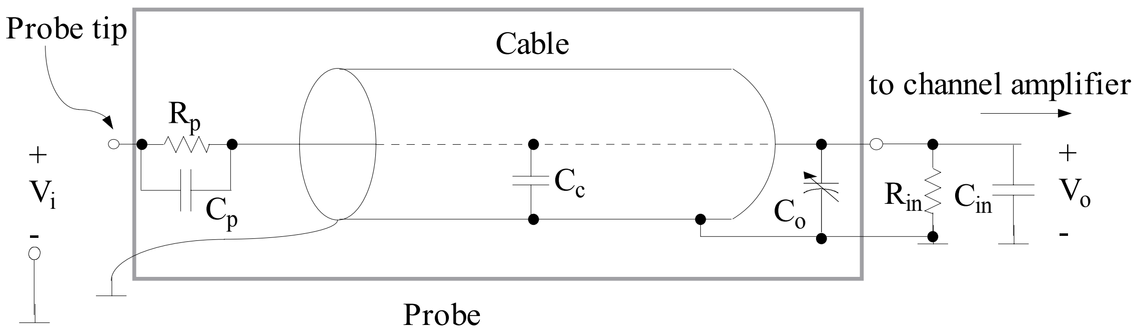


Figure 3.34 Probe model and probe compensation

When C_o is adjusted such that $(C_c+ C_o + C_{in})R_{in}||R_p = C_pR_p$, or equivalently $(C_c+ C_o + C_{in})R_{in} = C_pR_p$, the transfer response becomes $[1+(R_p/R_{in})]^{-1}$ for all ω. Hence $V_o(\omega)/V_i(\omega)$ is now independent of frequency. Such circuits are called *all-pass filters*. C_p is a low valued capacitor, typically about 10 pF. Probe cable capacitance is typically about 40 pF.

Determine the approximate value of C_o using the calculated value of R_p, values of R_{in} and C_{in} as written on the scope, and the approximate values of C_c and C_p given above. Setting C_o to this value is called *probe compensation.*

Oscilloscopes have a special terminal for compensation, usually marked “CAL” for calibration. This is an output where a square wave signal is provided. Connect the probe tip to this terminal (earth connection is not necessary) to compensate the probe. Display the signal on the scope. The signal is probably a distorted square wave. This is because the probe is uncompensated. Now take an alignment tool and turn the screw of variable capacitor C_o. If you turn it in correct direction, the signal on the screen will approach a well-defined square wave. Adjust until you get a well-defined square wave with right-angled corners. This is called *low frequency compensation*. The probe is compensated. Always use **10**× setting.

Connect one probe to the input, across the PCB connector, and the other across V_{out}. Set the frequency of the signal generator to approximately 3 KHz and apply a sine wave. Measure and record the amplitude of both the input signal and V_{out}. Make sure that your measurement for both signals is of the same form, i.e. they are both peak-to-peak or both peak voltage. Make this measurement for all the frequencies you calculated the transfer response. Adjust the scope time axis appropriately for every measurement frequency. Record your results. Calculate the ratio of amplitudes of V_{out} and input voltage in dB and plot them on the same transfer function graph. Do the calculations and measurements agree? Point out the disagreements, if there are any. Disconnect the signal generator.

4. Connect J12 and the microphone plug to the jack on the panel. Connect one probe to V_{out}. Adjust the time base of the oscilloscope to 5 ms/div. Pick the microphone and speak to it, while watching the waveform on the scope. There will be three types of signals on the scope:
   - your voice,
   - wind noise due to air blown out of your mouth while speaking,
   - noise due to the movement of microphone in your hand.

   We are interested in the first signal, your voice. The third signal appears as bursts of short duration. We can eliminate them by holding the microphone lightly at the cable end. To decrease the second type of signal, adjust your voice tone, and the distance between your mouth and microphone.

   Is your voice a regular signal, like a sine wave or a square wave?

   Now try to whistle at a fixed tone without generating wind noise. Keep the microphone at least about 15 cm away from your mouth. Whistle gently and slowly, but continuously, without producing any significant air flow. Does the signal look like a *periodic signal*? Can you measure the period of this waveform? What is its frequency? Record your observations and measurements. Switch the power OFF. Disconnect J12 and keep it disconnected until Chapter 8.

5. Now we have amplified and filtered the signal. Intelligibility is of prime importance in speech communication systems. Information content in speech is contained mainly in the frequency variation of the voice, where as emotional content is in amplitude variation. The voice signals are *clipped* in speech communication systems, in contrast to HiFi (*High Fidelity*, i.e. high quality music reproduction) systems. Clipping limits the amplitude of the signal and therefore severely distorts the voice signal, but all zero crossings of the signal, hence its frequency variation, is well preserved. A circuit comprising a series resistor and two diodes shown in Figure 3.35 clips the waveform.

   Assume that both diodes can be modeled with the piecewise linear model of Figure 2.11 (a) with $V_o = 0.6$ V. If v_{in} is a sine wave of amplitude 2.5 V peak, find and sketch the output waveform, $v_{out}(t)$, assuming that potentiometer R31 is set to maximum. If the input signal amplitude is indefinitely increased, can you tell what kind of output signal is obtained?

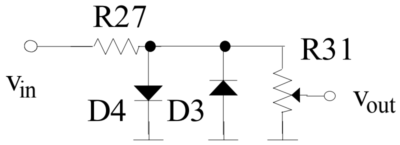
   

   Figure 3.35 Diode clipper

   D3 and D4 are both 1N4001. These diodes are in fact capable of handling large currents, and are used abundantly in circuits because of their low cost. The data sheet of this diode is in Appendix D. Examine this sheet. R27 is a 15K resistor. Assuming that the diode voltage will not change significantly for lower forward currents, the peak diode current will be approximately 0.5 mA even for a v_{in} peak amplitude of 8 V (the maximum that OPAMP can deliver). The *Forward voltage vs Forward current* graph in the data sheet reveals that the diode voltage is slightly less than 0.6 V when the forward current is 0.5 mA. Hence an approximate value of 0.6 V is an appropriate choice for V_o. What is the exact value of V_D when the diode current is 0.5 mA?

6. Mount and solder R27, R31, D3 and D4. Connect the signal generator and 180 K resistor, as in Figure 3.33. Adjust the frequency to 1 KHz and amplitude to 6.8 V peak. Switch the power ON. Connect a probe to the output of the OPAMP and another across the diodes. You must see a proper sine wave at OPAMP output with a peak voltage of about 2.5 V. Adjust the input amplitude if necessary. Measure the waveform across the diodes, i.e. the period, important voltage levels, etc., and sketch the waveform. Is the shape of this signal similar to what you have obtained in Exercise 5? What are the differences? What are the reasons for the differences, if there are any? Switch the power OFF.

7. The voice signal is now clipped to about 0.6 V. The maximum amplitude can now be 0.6 V. The audio signal carries the information to be transmitted and it is the modulating signal in TRC-10. The audio signal must be superimposed on (added to) a d.c. voltage before modulation in order to obtain AM signal, as discussed in Section 1.3. For example, if the d.c. voltage is twice the maximum amplitude of audio signal, then modulation index becomes 50%. We construct this signal in the circuit shown in Figure 3.36.

A d.c. voltage V_1 is first generated by R29 and R28 (*a trimmer potentiometer*- or *trimpot*) in this circuit. Trimpot R28 behaves like a variable divider. Calculate the range of d.c. levels that V_1 can assume.

Notice the similarity of this amplifier circuit to the preamplifier circuit. We have two inputs in this circuit. The effect of C26 is to filter out any a.c. voltage that may couple to V_1. Calculate the mid-band gain for input V_a in this amplifier (again *ignoring* C26 and C27). Write down the relation between V_{out}, V_1 and V_a. What is the minimum and maximum modulation index that we can have with this circuit? Calculate the 3-dB cut-off frequency of this amplifier.

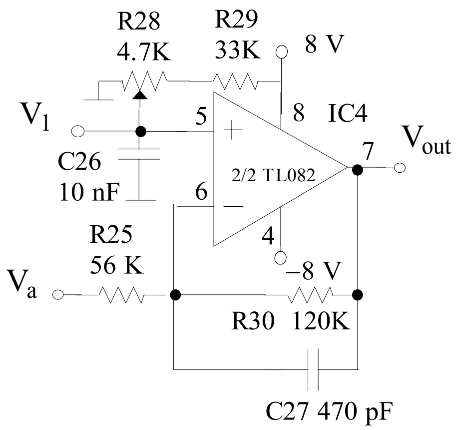


Figure 3.36 Audio signal output circuit

8. Mount the remaining components in microphone amplifier circuit. Check all connections carefully. Switch the power ON. Connect and adjust the signal generator exactly as in Exercise 1. Connect the oscilloscope probe to the output, V_{out}. Make sure that your scope amplifier is ***d.c. coupled***. Adjust R28 with your alignment tool and observe that you can shift the d.c. level of V_{out}. Now connect the multimeter across C26 and adjust R28 to its minimum and the reading is approximately 0 V d.c. Observe on the scope that V_{out} has zero or very low level d.c. component. Leave the setting of R28 at this level for the rest of the exercises of this chapter. We shall readjust this pot together with R51 to set the modulation index.

   Switch the power OFF. **IMPORTANT!** Switching off the power after an exercise is finished, is not reminded in the rest of this book. You must make sure that you follow this *good engineering* procedure in every exercise in this course, and always in your professional life. After all, ***safety means no accident***.

Loudspeaker amplifier

9. The loudspeaker amplifier is given in Appendix A. We analyzed the LPF formed by R10 and C10 in this chapter already. This piece of circuit is part of the first stage in loudspeaker amplifier. This first amplifier stage is shown in Figure 3.37.

The LPF of R10 and C10 is also loaded by R17. Carry out the analysis of this inverting amplifier circuit to find the exact cut-off frequency of the LPF and the overall mid-band gain. *Ignore the effect* of C10 while finding the mid-band gain.

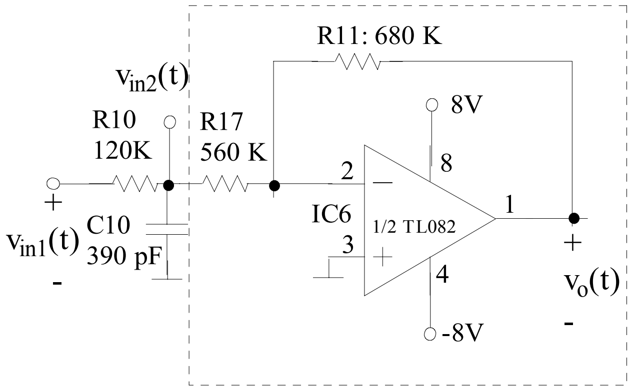


Figure 3.37 Loudspeaker amplifier-first stage

10. The output of this amplifier is connected to TDA7052A audio amplifier at two pins, 2 and 4. TDA7052A is not an OPAMP. It is an integrated circuit audio amplifier. The data sheet of TDA7052A is given in Appendix D. Examine the data sheet. Which type of package is your integrated circuit, i.e. what is *package pin position* and what is *package code* of your IC? Who is the producer?

    Can we use ±8 V supplies for this amplifier? What is the supply range of this IC?

    Pin 2 is the input pin for the audio signal to be amplified. What is the input impedance of the amplifier at this pin? Under which conditions can this impedance be approximated as an ideal OPAMP input impedance?

    Read the soldering information on page 11 in data sheet.

11. First amplifier is connected to TDA7052A by means of two filters. OPAMP output is connected to pin 2 through a HPF formed by R13, C15 and the input impedance (resistance) of TDA7052A (if you have not been able to find it in the data sheet for exercise above, it is *typically* 20 K). Calculate the cut-off frequency. What is the transfer function from OPAMP output to pin 2 of TDA7052A?

    The OPAMP output is connected to pin 4 through a LPF formed by R12, R14, C16, C17, R15 and R16. This pin is the *volume control* pin of the audio amplifier. The volume of the output (in fact the gain of the amplifier) is controlled by the d.c. voltage applied to this pin. The gain versus d.c. voltage level variation is given in the data sheet in Figure 3 on page 7. Examine this figure. Figure reveals that the gain of the amplifier can be changed between –70 dB and +35 dB, when the control voltage is changed between 0.4 V and 1.2 V. Find this gain range in terms of ratio.

    TRC-10 uses this pin for two purposes: manual gain control and *automatic gain control* (AGC). Manual gain control is done by R15 and R16. R15 is a *panel* *potentiometer*, and is used to adjust the output volume to a comfortable level, by supplying a d.c. voltage from +15 V supply. The AGC circuit uses the d.c. level coming from the diode detector in RX circuit, which indicates the level of received carrier signal. We discuss these in Chapter 6. The LPF must therefore separate the audio signal and this d.c. level.

What is the cut-off frequency of LPF? How much does the presence of R15 and R16 affect this cut-off frequency?

Call the d.c. component of the voltage at OPAMP output V_a and the voltage at pin 4 as V_4. Ignoring the effects of C16 and C17 (you must know why we can ignore C16 and C17 by now) and assuming R15 set to 0 Ω, find the value of V_4 in terms of V_a and +15 V. What is the value of V_4 if R15 is set to its full value?

12. Short the middle pin and one of the side pins of R15. Cut two 10 cm pieces of wire and solder them to the respective pins of R15. Crimp the contacts for PCB jack to their other ends. Mount R15 on the panel, using a pair of pliers. Use anti-slip washer. Take care not to round the nut. Fit a knob on the pot swindle. Fit the jack.

13. Mount and solder all the components in loudspeaker amplifier circuit excluding C10, R12 and R13. Also solder the PCB jacks for the loudspeaker and R15 connection. C13 and C14 are by-pass capacitors. Connect and solder a short jumper between +15 V supply and IC5 pin holes (or pads) of J20. This connection provides power supply to IC5.

    Check all connections carefully. Short circuit C10 end of R17 to ground, using a crocodile jumper. Switch the power ON. Check the d.c. levels by a multimeter at supply connections, both on the PCB and at supply pins of IC's. Check the d.c. levels at OPAMP input and output. Input voltage must be 0V d.c. and OPAMP output must be at most 20 milivolts d.c. Check the d.c. level at pin 4 of TDA7052A. Make sure that it is within the limits, and can be changed by adjusting R15.

    Switch the power off. Remove the crocodile jumper.

    We must modify the loudspeaker amplifier slightly, in order to perform Exercise 15, where we use sub-audio frequency signals. Some advanced signal generators produce their output by a technique called Direct Digital Synthesis (DDS). These may have some low amplitude but very high frequency components (compared to the desired frequency) additively imposed on their output. The first modification is to use a 22 nF capacitor (or two 10 nF) in the place of C10, to convert R10-C10 section into a LPF with 60 Hz 3-dB cut-off frequency. Solder a 22 nF capacitor onto the solder pads of C10 temporarily.

    The second modification is necessary to let the sub-audio frequency signals reach the input of TDA7052A (pin2). Normally R13, C15 and the input resistance of TDA7052A (which is about 20 K) forms a HPF with a cut-off frequency of about 65 Hz. We pull that cut-off frequency below 1 Hz, by connecting a 56 K resistor temporarily in place of R13 and 2.2 μF C73 capacitor in parallel with C15. Do not trim the leads. Just solder temporarily on the soldering pads. You will disconnect them very soon. You need C73 later.

Check all the connections.

14. Loudspeakers are also electro-acoustic transducers that convert electrical energy into acoustic energy. Acoustic energy requires a presence of matter in the medium to propagate (it does not propagate in vacuum), unlike electromagnetic energy. Air is the medium of propagation for audio acoustics, and the matter that supports the propagation is air. A loudspeaker acts like a piston and forces air in its vicinity, to move to and fro at the frequency of the signal and at an amplitude proportional to the signal amplitude. The structure of an ordinary loudspeaker is given in Figure 3.38.

    Loudspeakers are most commonly made to have a circular symmetry. Figure 3.38 gives the cross section of the speaker. The cone section above is a cone shaped light diaphragm and it simply acts as the piston head to push the air. It is very lightly supported at the peripheral metal frame by corrugated suspension, both at top and at bottom. This support allows the diaphragm to move easily, but up and down only.

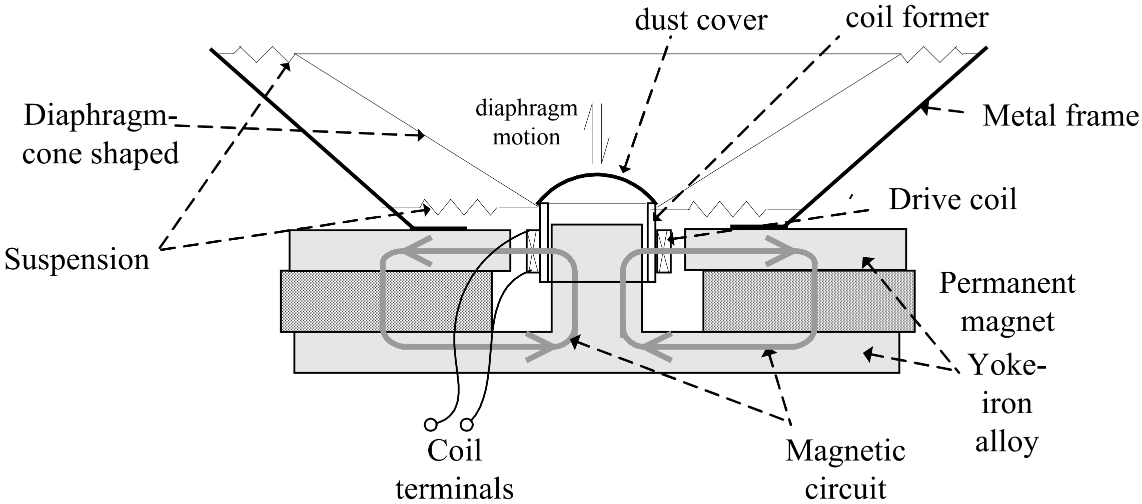


Figure 3.38 Loudspeaker

We need a “motor” to drive the piston head. The motor is at the lower part. Diaphragm is rigidly attached to the drive coil. The motor part consists of a magnetic circuit, which moves the drive coil up and down when there is a current flowing in the coil. Motor can be analyzed in two parts. The first one is the magnetic circuit. The magnetic circuit is shown in Figure 3.39.

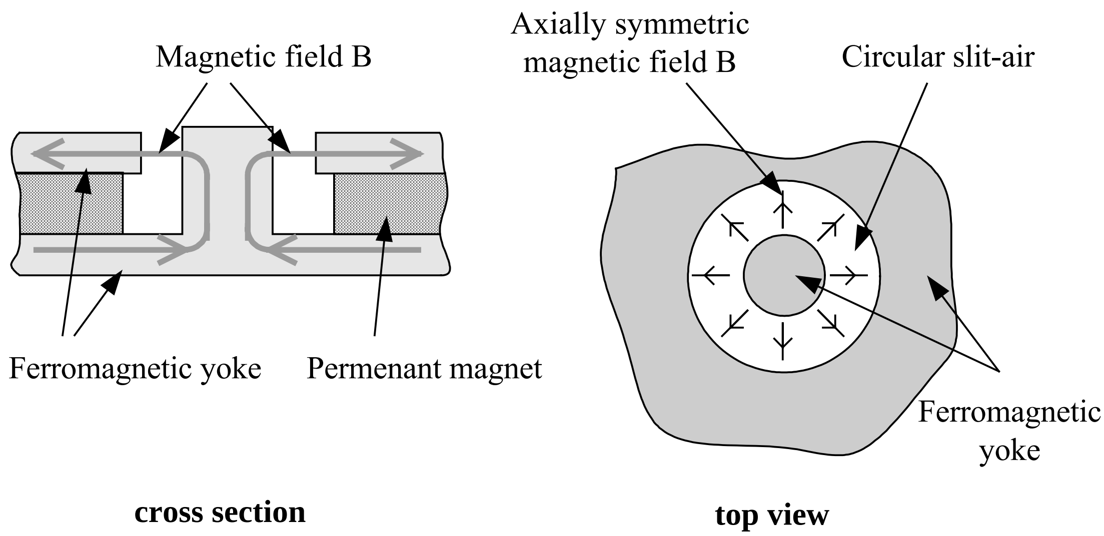


Figure 3.39 Magnetic circuit

The source of the magnetic field is the permanent magnet, whose N-S poles are aligned vertically in the cross section view. A magnetic flux emanates from the magnet in that direction as well. The function of the yoke is to concentrate the magnetic flux into the narrow circular air slit. Yoke is made of a ferromagnetic material like iron, which conducts the magnetic flux as copper conducts electric current. Thus almost all the flux (small amount of flux escapes into surrounding air medium) is concentrated in the slit, generating a circularly symmetric strong magnetic field, B (top view).

Secondly a circular drive coil is placed in this field. This is shown in Figure 3.40. When a current carrying conductor is placed in a magnetic field, the conductor experiences a force in a direction perpendicular to both the directions of the current and the magnetic field. Now since current and magnetic field both lies on the same plane, the direction of the generated force is perpendicular to that plane. For given directions of field and current, the magnetic force is in the direction shown in the figure. The magnitude of this force is given as

$F = N\,I\,B$ (Nt.- Newtons),

where I is the current in the coil (A), B is the magnetic field in Tesla (T) and N is the number of turns in the coil.

If the current in the coil is sinusoidal, then the force is obviously sinusoidal. Whatever the signal (current) is, the force generated is proportional to it. Therefore we must apply a current, proportional to the voice signal, to the drive coil of the loudspeaker. The generated magnetic force is then proportional to the voice and since the coil is rigidly fixed to the cone membrane (piston), the air in front of speaker is moved accordingly.

This is how loudspeaker works.

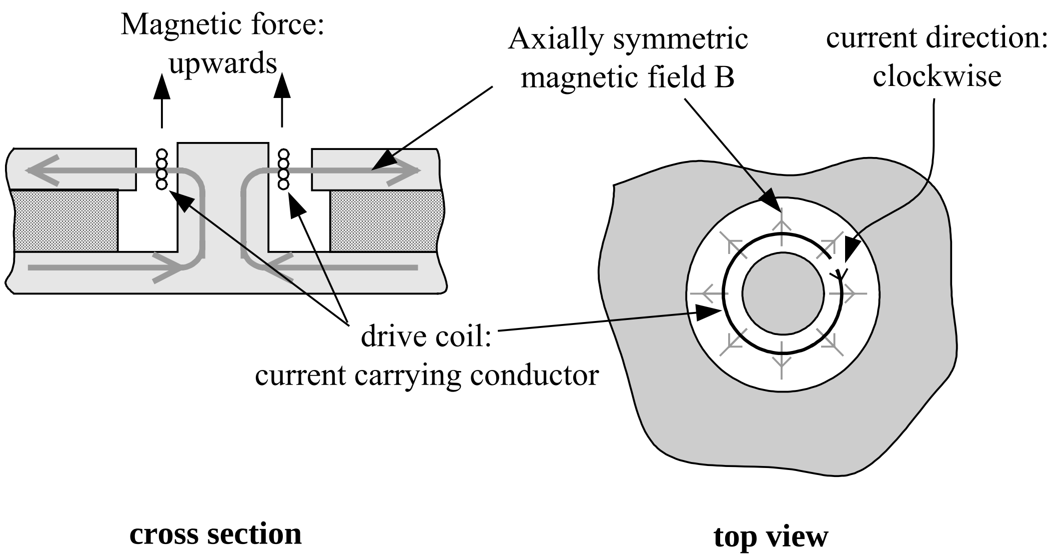


Figure 3.40 Current carrying coil in magnetic circuit

The efficiency, η, of this kind of loudspeakers are extremely low. Efficiency is defined as the ratio of power delivered to air, P_a, to the total power applied to the loudspeaker, P_{in},

$\eta = P_a / P_{in}$.

P_{in} is the sum of P_a and the electrical and mechanical power lost in the loudspeaker. This efficiency is few percent (2% is typical) for this kind of loudspeaker. Almost all of the input power is dissipated in the speaker.

Loudspeakers are specified by their input resistance. 8 Ω and 16 Ω are standard input resistance values for this type of loudspeakers.

Cut two 10 cm long pieces of wire. Solder one end of each to the lead points on the loudspeaker. Fit the PCB jack to the other ends of the wires.

15. You must use another type of loudspeaker than the one in TRC-10 kit, in order to perform this experiment step. The loudspeaker in TRC-10 is of cheapest type and its acoustic response at sub-audio frequencies is very poor. The suspension of the cone in this loudspeaker is stiff. You need a mid range loudspeaker or, still better, a “woofer” for this step. Borrow an appropriate loudspeaker from lab technician. Place the loudspeaker on a paperback book on the bench, cone facing upwards. Connect the PCB jack to the output of audio amplifier.

    Connect the signal generator to the input of loudspeaker amplifier, at Audio/RX pigtail. Set the signal generator to sine wave output, with a frequency of 1 Hz, and amplitude of 0.5V peak. Adjust R15 to mid position. Switch the power ON.

    While watching the surface of the cone diaphragm (rather the dust cover) carefully, increase the volume until you can see the motion of the diaphragm. Increase the volume until you can see the diaphragm motion clearly. You cannot hear anything but you can see that the cone surface moves up and down in a sinusoidal motion. You cannot hear, because this frequency is far below your auditory systems pass-band. This is simply vibration. The sinusoidal input voltage is converted into mechanical energy in form of sinusoidal motion. *Acoustic energy is a form of mechanical energy*.

Change the signal generator output to a square wave of 50% duty cycle at the same frequency. Observe the mechanical response of the cone surface.

Change the wave shape back to sinusoidal. While keeping the amplitude constant, increase the frequency gradually. Find and record the frequency where you can no longer follow the up and down motion of the dust cover (i.e. where you can only see a *blurred* dust cover). This frequency is approximately the perception threshold for motion of your visual system.

Keep on increasing the frequency. Find the frequency where your auditory system starts sensing sound, i.e. the lowest frequency you can hear. This frequency is approximately the lower cut-off frequency of your auditory system. Record the frequency.

Increase the frequency to 50 Hz. Listen to the sound carefully. You will hear this sound in your professional life quite often. It is an indication that you have messed up your electronic circuit and there is a breakthrough of line voltage into your low frequency circuits!

Switch off the power and disconnect the signal generator. Disconnect and return the loudspeaker you borrowed back to the lab technician. De-solder and remove 22 nF and 56K. De-solder and save C73. Solder C10, R12 and R13.

Connect the PCB jack of the TRC-10 loudspeaker to the output of audio amplifier. Mount the loudspeaker on the panel into its brackets. Check all new connections.

16. Connect the signal generator as you did in Exercise 15. Set the signal generator to about 300 Hz sine wave of approximately 100 mV peak amplitude. Connect a probe to signal generator output and see the sine wave on the oscilloscope. Adjust R15 to mid-range. Switch the power ON.

    Adjust R15 until you can hear a comfortable tone. While watching the signal on the screen, listen to the sound as you increase the frequency up to 20 KHz. Record the frequency above which you cannot hear anything. This frequency is the cut off frequency of your auditory system (subjectively, of course).

    Decrease the frequency to 300 Hz. Change the signal type to square wave of the same amplitude. Try to feel the difference between the sound produced by a square wave and a sine wave of same frequency and amplitude. Although both signals have the same period, square wave has additional harmonic components. As the frequency is increased, the filters in your audio circuits attenuate the harmonics of the square wave. Also harmonics frequencies eventually fall beyond your hearing cut off frequency.

Connect another probe to the output of OPAMP or to that end of R13. Make sure both waveforms are clear and stable on the scope. Increase the frequency while watching the waveforms, until the difference in the sound you hear from the sine wave and the square wave becomes insignificant. Record that frequency. Can you comment on the reason? (Hint: Consider the LPFs along the way and your auditory system transfer response)

Set the signal generator to *sweep* mode, between 300Hz and 3KHz. In sweep mode, the frequency of the signal generator output is continuously varied linearly between lower and upper limits and in a specified period. Set the period to approximately 1 second. Listen to the sound produced while watching the waveform.

Switch off the power.

17. Connect the microphone amplifier output (Audio/TX pigtail) to the Audio/RX input using a crocodile jumper cable. Connect the microphone. Ask your friend to speak to the microphone. Check that audio-audio chain works.

    Switch off the power. Disconnect the signal generator, the scope probes and the connection between Audio/TX and Audio/RX. De-solder the jumper on two pin holes (or pads) of J20.

    We are now ready to proceed with RF circuits.

3.8. Problems

1. Expand exp(jθ) in Taylor series, group the real and imaginary parts and show that real series corresponds to the expansion of cos(θ) and imaginary series correspond to the expansion of sin(θ).

2. Show that capacitance is a linear circuit element.

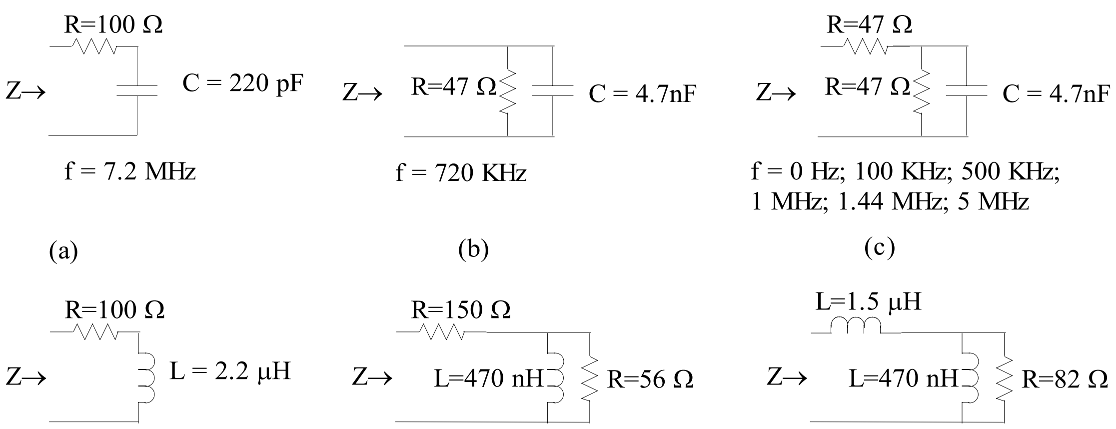
3. Show that, if a circuit satisfies the linearity definition for two arbitrary inputs, it also satisfies the linearity condition for an indefinite number of inputs.

4. A voltage amplifier input/output characteristics is $V_o(t) = AV_i(t)$, where $V_i(t)$ is the input and $V_o(t)$ is the output voltage, and A is a constant (gain). Show that an amplifier is a linear circuit component.

5. Design a LPF using a resistor and an inductor. Find the transfer function for this filter and plot its magnitude with respect to angular frequency.

6. Design a HPF using a resistor and an inductor. Find the transfer function for this filter and plot its magnitude with respect to angular frequency.

7. Find the impedance of the circuits given below at the specified frequency. Write the impedance in polar form, i.e. magnitude and phase (3 significant figures):



(a) f = 7.2 MHz

(b) f = 720 KHz

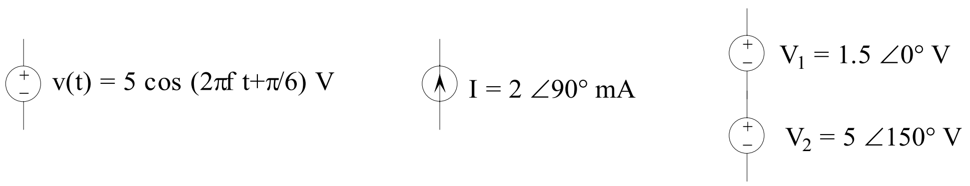
(c) f = 0 Hz; 100 KHz; 500 KHz; 1 MHz; 1.44 MHz; 5 MHz

(d) f = 7.2 MHz

(e) f = 1 MHz; 19 MHz

(f) f = 1 MHz; 36.5 MHz; 73 MHz

8. The voltage (current) sources given below are connected to the circuits in problem 7. The frequencies of the sources are as given in problem 7. Find the current through (voltage across) the circuits.



(a) (b) (c)

9. Calculate the current through the capacitor in problem 7(c) and inductor in problem 7 (e) and (f) using *node analysis*, when the sources in problem 8 are connected across the circuits.

10. The voltage across and the current through two element series circuits are given below. Find the component types and their values with two significant figure accuracy and in regular value notation (like *Ω, K* for resistance; *µ, p* for capacitance, etc.), for each circuit. Determine the frequency and angular frequency in each case.

(a) $v(t) = 28.3 \cos(628t + 150°)$ V; $i(t) = 11.3 \cos(628t + 140°)$ mA
(b) $v(t) = 5 \cos(2\pi\times 300t - 25°)$ V; $i(t) = 8 \cos(2\pi\times 300t + 5°)$ mA
(c) $v(t) = 10\cos(2\pi\times 796t - 150°)$ V; $i(t) = 1.333 \cos(2\pi\times 796t - 3\pi/8)$ mA
(d) $v(t) = 8 \cos(10^6 t + 45°)$ V; $i(t) = 8 \cos(10^6 t + 90°)$ mA
(e) $v(t) = 5 \cos(2\pi\times 10^6 t - 160°)$ V; $i(t) = 10 \cos(2\pi\times 10^6 t - 75°)$ mA

11. A series circuit has a resistor R = 120 Ω and an inductor L = 780 nH. A voltage of 10 V peak value with a frequency of 25 MHz (zero phase) is applied across this circuit. Find the current flowing through it and write down the expression for the time waveform. Find the current if the frequency is increased to 50 MHz.

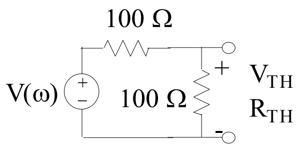
12. A series circuit has R =1K and C =120 pF. What is the frequency (not angular frequency) at which the phase difference between the current and voltage is π/4.

13. A series RC circuit has C = 470 pF. Find R if the phase difference is 30° at 1 KHz.

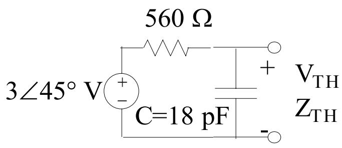
14. The voltage and current of a two element series circuit at 500 KHz are $V=3\angle-45°$ V and $I = 1\angle-120°$ mA. When the frequency is changed to another value f, the phase difference between the voltage and current becomes 30°. Find f.

15. Assume that the voltage and current pairs given in problem 10 are for two element parallel circuits. Determine the component types and their values.

16. Find and draw the Thevenin equivalent of the circuits given below. Express the equivalent voltages and impedances in polar form.

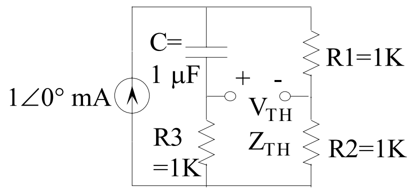


(a)



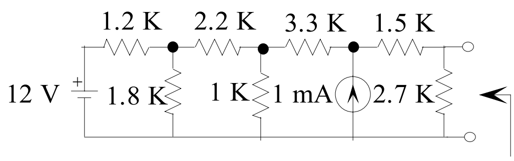
f = 15.8 MHz

(b)

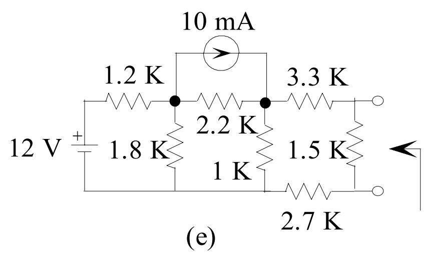


f =160 Hz

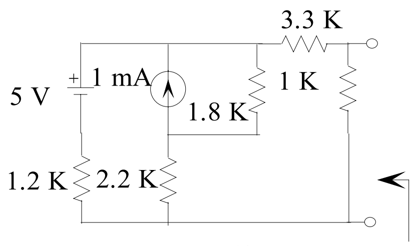
(c)



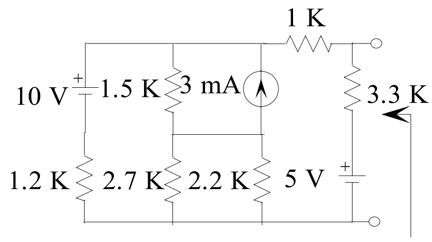
(d)



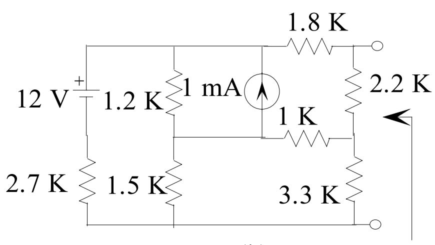
(e)



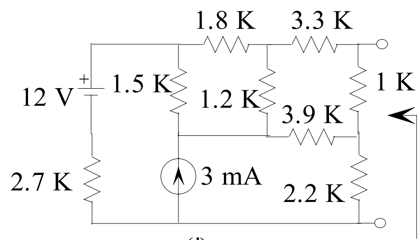
(f)



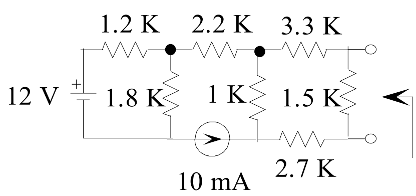
(g)



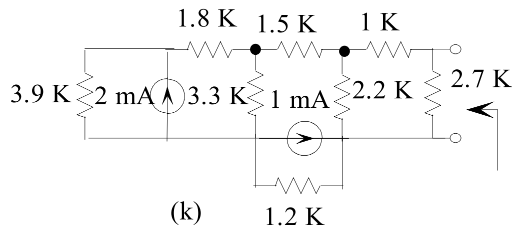
(h)



(i)



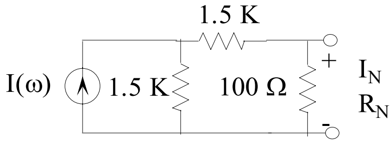
(j)



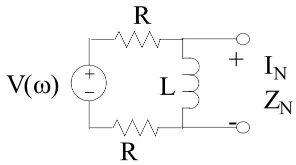
(k)

17. Convert the equivalent circuits found in problem 16 into Norton equivalent circuits and draw them.

18. Find and draw the Norton equivalent of the circuits given below. Express the equivalent currents and impedances in polar form.

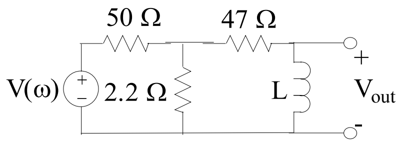


(a)

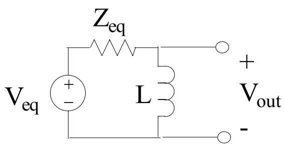


(b)

19. Find the V_{eq} and Z_{eq} such that the circuit given below in (a) can be represented as in (b).

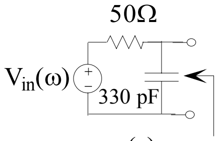


(a)

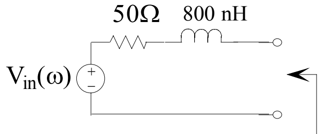


(b)

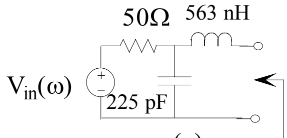
20. Find and draw the Thevenin and Norton equivalent circuits of the following at d.c., 20 MHz and 40 MHz.



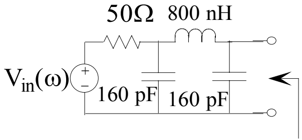
(a)



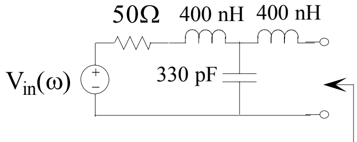
(b)



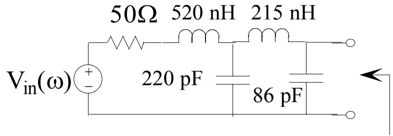
(c)



(d)



(e)



(f)

21. This problem illustrates how a unity feedback amplifier is used to avoid loading effects. Consider the divider circuit in the Figure 3.41(a). What is V_{out}? Assume we want to apply V_{out} across a 1K resistor as shown in part (b). What is V_{out} now?

Now assume we place a buffer amplifier between the divider and 1K resistor as in part (c). Find V_{out}.

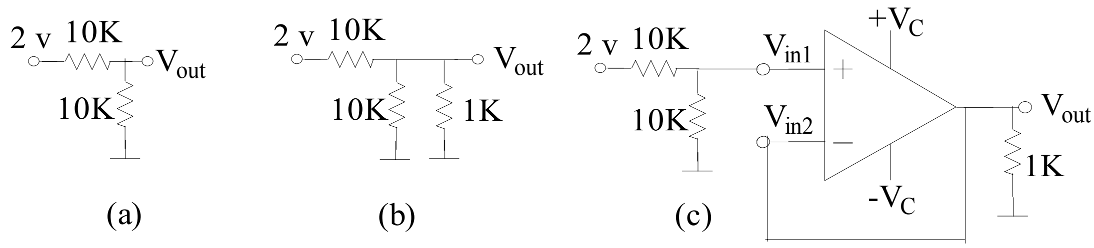
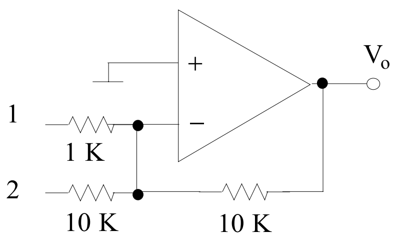


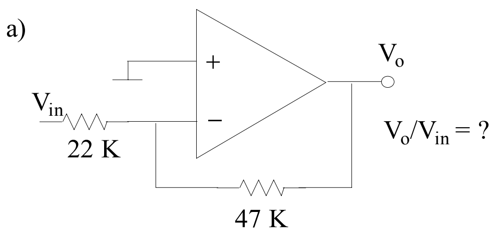
Figure 3.41

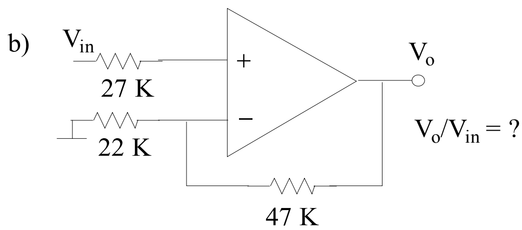
22. Assume there are two signals, V_1 and V_2. Design a summing amplifier to produce $V_{out}= 2\ V_1+ 0.5\ V_2$, using at most (a) two OPAMPs, and (b) one OPAMP.
23. Make a table indicating what terminals to connect to the input signal source or the output in order to get all possible (different) amplification factors, for the following circuit. Also calculate the resulting input impedance and what the possible gains are, and include them in the table.

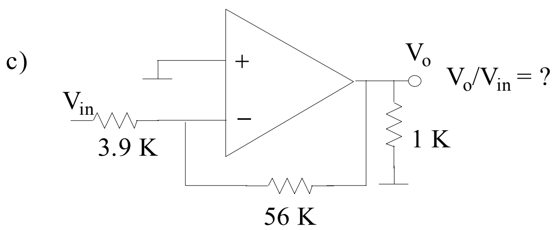


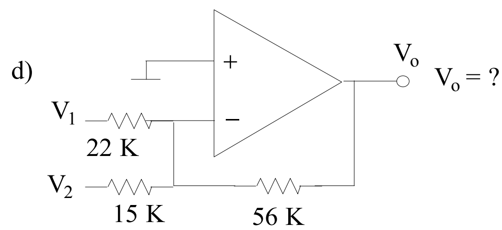
1	2	Gain	Rin
i/p	NC	-10	1K
i/p	o/p	-5	1K
NC	i/p		
i/p	i/p		
o/p	i/p		

24. Assume that OPAMPs are ideal in the following problems.

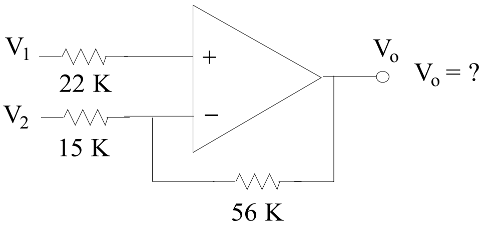




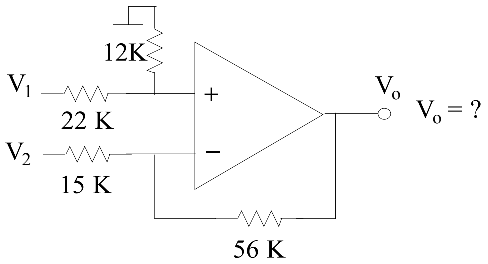




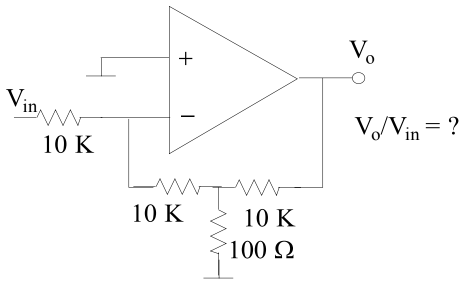
e)



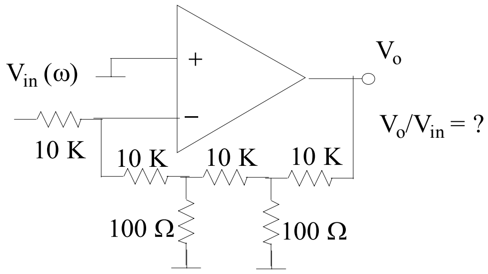
f)



g)

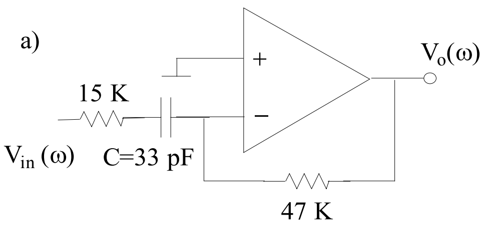


h)

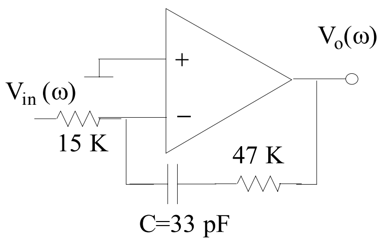


25. Find $H(0) = V_o(0)/V_{in}(0)$ and $H(\infty) = V_o(\infty)/V_{in}(\infty)$ in the following circuits.

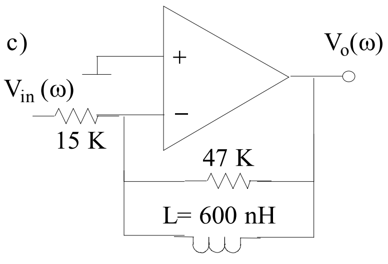
a)



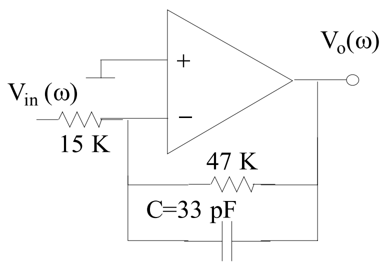
b)



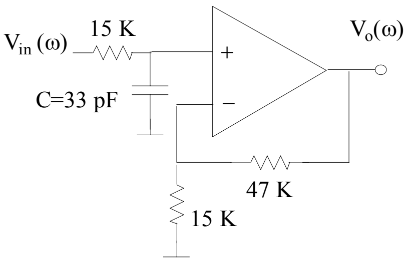
c)



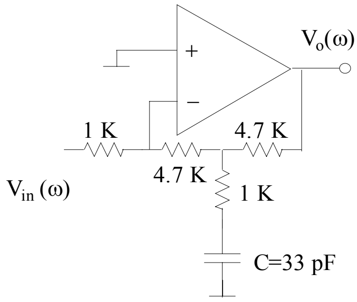
d)



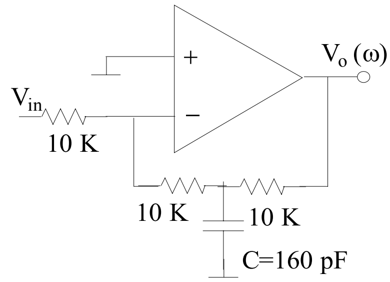
e)



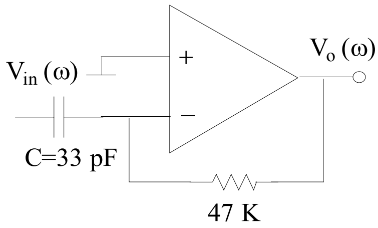
f)



g)

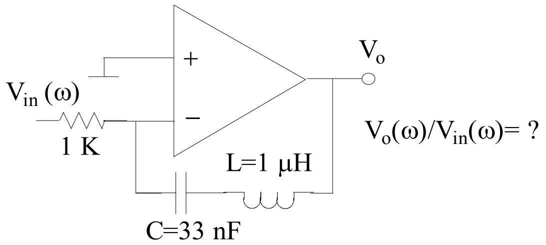


h)



26. Find the transfer functions $V_o(\omega)/V_{in}(\omega)$ for the circuits given in problem 24.

27. Find the transfer function of the following circuit. Is there a frequency at which the gain is zero? Which frequency? Is there a frequency at which the gain is ∞?



28. In the following circuit a RF OPAMP LM7171 is used in a non-inverting amplifier configuration. What is the magnitude of the typical d.c. voltage at the output due to the *input offset voltage* of LM7171? What is the maximum?