

EEE 211  
ANALOG ELECTRONICS  
LECTURE NOTES

Hayrettin Köymen

BİLKENT  
UNIVERSITY

**This work is published by Bilkent University as the textbook for EEE 211 Analog Electronics course, taught in Bilkent University. All rights are reserved. No part of this work may be reproduced in any form except by written permission of the author. All rights of translation are reserved. Printed in Bilkent University, Ankara, Turkey.**

**First Edition, rev. 3.4  
April 2004**

## CONTENTS

Chapter 1: SIGNALS AND COMMUNICATIONS	1-1
1.1. Frequency	1-1
1.2. Oscillators	1-3
1.3. Modulation	1-5
1.4. Amplifiers	1-7
1.5. Mixers	1-8
1.6. Filters	1-9
1.7. Receivers	1-11
1.8. TRC-10	1-12
1.9. Bibliography	1-14
1.10. Laboratory exercises	1-14
1.11. Problems	1-15
Chapter 2: CIRCUIT THEORY PRIMER	2-1
2.1. Energy sources	2-1
2.2. Resistors	2-3
2.2.1. Resistive circuits	2-5
2.3. Analysis of electrical circuits	2-5
2.4. Capacitors	2-8
2.4.1. Power and energy in capacitors	2-10
2.4.2. RC circuits	2-11
2.5. Diodes	2-15
2.5.1. Diodes as rectifiers	2-17
2.5.1. Zener diodes as voltage sources	2-19
2.6. Inductors	2-20
2.7. Transformers	2-23
2.8. Circuit protection devices	2-24
2.8.1. Varistors	2-24
2.8.2. PTC Thermistors	2-25
2.8.3. Circuit protection	2-25
2.8. Bibliography	2-26
2.9. Laboratory exercises	2-26
2.10. Problems	2-34
Chapter 3: AUDIO CIRCUITS	3-1
3.1. Linear circuits	3-1
3.1.1. Power	3-4
3.2. Impedance and Transfer Function	3-5
3.3. Sources and Equivalent Circuits	3-10
3.4. Operational amplifiers	3-12
3.5. OPAMP circuits	3-17
3.5.1. Offset voltage in OPAMPs	3-20
3.5.2. OPAMP Linear Voltage Regulator	3-21
3.6. Bibliography	3-24

3.7. Laboratory exercises	3-24
3.8. Problems	3-40
Chapter 4: TUNED CIRCUITS	4-1
4.1. Parallel resonance	4-1
4.1.1. Energy in tuned circuits	4-2
4.1.2. Quality of resonance	4-4
4.2. Series tuned circuits	4-8
4.3. Equivalence of series and parallel RLC circuits	4-9
4.4. Real inductors	4-12
4.4.1. Air core inductors	4-14
4.4.2. Powdered iron core inductors	4-14
4.4.3. Core losses	4-15
4.4.4. Copper losses	4-16
4.4.5. RF choke inductors	4-17
4.5. RF Amplification using OPAMPs	4-19
4.5.1. OPAMP Open Loop Frequency Response	4-19
4.5.2. Voltage gain and power gain	4-21
4.5.3. Slew Rate	4-22
4.5.4. Input Bias Current	4-22
4.6. Maximum power transfer	4-22
4.7. Bibliography	4-24
4.8. Laboratory exercises	4-24
4.9. Problems	4-33
Chapter 5: FILTERS	5-1
5.1. Motivation	5-1
5.2. Polynomial filters	5-3
5.2.1. Butterworth filters	5-3
5.2.2. Harmonic filter	5-6
5.2.3. Butterworth HPF	5-8
5.2.4. Butterworth BPF	5-9
5.3. Impedance matching	5-10
5.3.1. Transformers	5-10
5.3.2. Real transformers	5-11
5.3.3. Matching by resonant circuits	5-14
5.3.4. Impedance inverters	5-15
5.4. Crystal filters	5-16
5.4.1. Quartz crystals	5-16
5.4.2. IF ladder filter	5-17
5.5. Bibliography	5-20
5.6. Laboratory exercises	5-20
5.7. Problems	5-30
Chapter 6: DIODES IN TELECOMMUNICATIONS	6-1
6.1. Diode detector	6-1
6.1.1. Real diodes in envelope detectors	6-3

6.2. TX/RX switch	6-5
6.2.1. Diode biasing	6-6
6.2.2. PIN diode switch	6-8
6.2.3. RF gain control	6-11
6.3. Audio gain control	6-12
6.4. Bibliography	6-13
6.5. Laboratory exercises	6-14
6.6. Problems	6-19

## Chapter 7: FREQUENCY CONVERSION 7-1

7.1. Amplitude modulators	7-1
7.1. Mixers in telecommunication circuits	7-2
7.1.1. Switch mixer	7-2
7.1.2. Double-balanced mixers	7-4
7.2. Analog multipliers	7-5
7.2.1. Conversion gain	7-7
7.3. Oscillators	7-7
7.3.1. Oscillator concept	7-8
7.3.2. Frequency control	7-10
7.4.3. Phase considerations in oscillators	7-11
7.4. Bibliography	7-12
7.5. Laboratory exercises	7-12
7.6. Problems	7-23

## Chapter 8: ON THE AIR 8-1

8.1. Antenna concept	8-1
8.1.1. Radiation from an antenna	8-1
8.1.2. Receiving antenna	8-6
8.2. Antenna impedance	8-6
8.3.1. Self impedance	8-7
8.3.2. Mutual impedance	8-7
8.3. Half wave dipole	8-8
8.4. Monopole antenna	8-8
8.5. Antenna feeders	8-10
8.5.1. Balanced-unbalanced transformation	8-11
8.5.2. TRC-10 short dipole feeder	8-12
8.6. Using amateur frequency bands	8-14
8.7. Bibliography	8-14
8.8. Laboratory exercises	8-14
8.9. Problems	8-16

## Appendix A: Circuit diagrams and PCB layout

## Appendix B: List of components

## Appendix C: Data sheets

## Appendix D: Complex numbers

## Chapter 1 : SIGNALS AND COMMUNICATIONS

Electronic communications is exchanging signals. While these signals are symbolic in many communication schemes, they are almost exact electrical replicas of original information in analog wireless communications. Sound and vision are all such signals. Signals are converted into a form, by a *transmitter*, so that they can be transmitted in the air as part of *electromagnetic spectrum*, and are received by a *receiver*, where they are converted back to the original form. Two communicating parties can be quite far away from each other, and therefore the term *telecommunications* is used to describe this form of communications. What follows in this chapter is a descriptive theory of analog signal processing in communications.

Transceivers are wireless transmitters (TX) and receivers (RX) combined in a single instrument. This book is structured around building and testing a transceiver, TRC-10, operating in the 10-meter amateur band (28-29.7 MHz). The name is generic: *TRC* stands for transceiver and *10* indicate that it works in 10-meter band.

TRC-10 is an *amplitude modulation superheterodyne transceiver*. We have to make some definitions in order to understand what these terms mean.

### 1.1. Frequency

The two variables in any electrical circuit is voltage,  $V$ , and current,  $I$ . In electronics, all signals are in form of a voltage or a current, physically. Both of these variables can be time varying or constant. Voltages and currents that do not change with respect to time are called *d.c.* voltages or currents, respectively. The acronym *d.c.* is derived from *direct current*.

Voltages and currents that vary with respect to time can, of course, have arbitrary forms. A branch of applied mathematics called *Laplace analysis*, or its special form *Fourier analysis*, investigates the properties of such time variation, and shows that all time varying signals can be represented in terms of linear combination (or weighted sums) of sinusoidal waveforms.

A sinusoidal voltage and current can be written as

$$v(t) = V_1 \cos(\omega t + \theta_v), \text{ and}$$

$$i(t) = I_1 \cos(\omega t + \theta_i).$$

$V_1$  and  $I_1$  are called the *amplitude* of voltage and current, and have units of Volts (V) and Amperes (A), respectively. “ $\omega$ ” is the radial frequency with units of radians per second (rps) and  $\omega = 2\pi f$ , where “ $f$ ” is the frequency of the sinusoid with units of Hertz (Hz). “ $\theta$ ” is the phase angle of the waveform. These waveforms are periodic, which means that it is a repetition of a fundamental form in every  $T$  seconds, where  $T = 1/f$  seconds (sec).

Quite often, sinusoidal waveforms are referred to by their *peak amplitudes* or *peak-to-peak amplitudes*. Peak amplitude of  $v(t)$  is  $V_1$  Volts peak (or  $V_p$ ) and peak-to-peak amplitude is  $2 V_1$  Volts peak-to-peak (or  $V_{pp}$ ).

Now we can see that a d.c. voltage is in fact a sinusoid with  $f = 0$  Hz. Sinusoidal voltages and currents with non-zero frequency are commonly referred to as *a.c.* voltages and currents. The acronym *a.c.* comes from *alternating current*.

If we know the voltage  $v(t)$  across any element and current  $i(t)$  through it, we can calculate the power delivered to it as

$$P(t) = v(t)i(t) = V_1 I_1 \cos(\omega t + \theta_v) \cos(\omega t + \theta_i)$$

or

$$P(t) = \frac{V_1 I_1}{2} \cos(\theta_v - \theta_i) + \frac{V_1 I_1}{2} \cos(2\omega t + \theta_v + \theta_i).$$

$P(t)$  is measured in watts (W), i.e.  $(1V) \times (1A) = 1$  W.

In case of a resistor, both current and voltage have the same phase and hence we can write the power delivered to a resistor as

$$P(t) = \frac{V_1 I_1}{2} + \frac{V_1 I_1}{2} \cos(2\omega t + 2\theta_v).$$

We shall see that the phase difference between voltage and current in an element or a branch of circuit is a critical matter and must be carefully controlled in many aspects of electronics.

$P(t)$  is called the *instantaneous power* and is a function of time. We are usually interested in the *average power*,  $P_a$ , which is the constant part of  $P(t)$ :

$$P_a = \frac{V_1 I_1}{2} \cos(\theta_v - \theta_i),$$

in general, and

$$P_a = \frac{V_1 I_1}{2}$$

in case of a resistor.

We note that if the element is such that the phase difference between the voltage across and current through it is  $90^\circ$ ,  $P_a$  is zero. Inductors and capacitors are such elements.

Radio waves travel at the speed of light,  $c$ . The speed of light in air is  $3.0 \text{ E}8$  m/sec (through out this book we shall use the scientific notation, i.e.  $3.0 \text{ E}8$  for  $3 \times 10^8$ ), to a very good approximation. This speed can be written as

$$c = f\lambda$$

where  $\lambda$  is the wavelength in meters. TRC-10 emits radio waves at approximately 30 MHz (actually between 28 and 29.7 MHz). The wavelength of these waves is approximately 10 meters. The amateur frequency band in which TRC-10 operates is therefore called *10-meter band*.

## 1.2. Oscillators

Electronic circuits that generate voltages of sinusoidal waveform are called “sinusoidal oscillators”. There are also oscillators generating periodic signals of other waveforms, among which square wave generators are most popular. Square wave oscillators are predominantly used in digital circuits to produce time references, synchronization, etc. Oscillator symbol is shown in Figure 1.1.

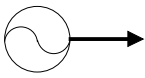


Figure 1.1 Oscillator symbol

We use oscillators in communication circuits for variety of reasons. There are two oscillators in TRC-10. The first one is an oscillator that generates a signal at 16 MHz fixed frequency. This oscillator is a square wave *crystal oscillator module*.

A square wave of 2 volts peak to peak amplitude is depicted in Figure 1.2.

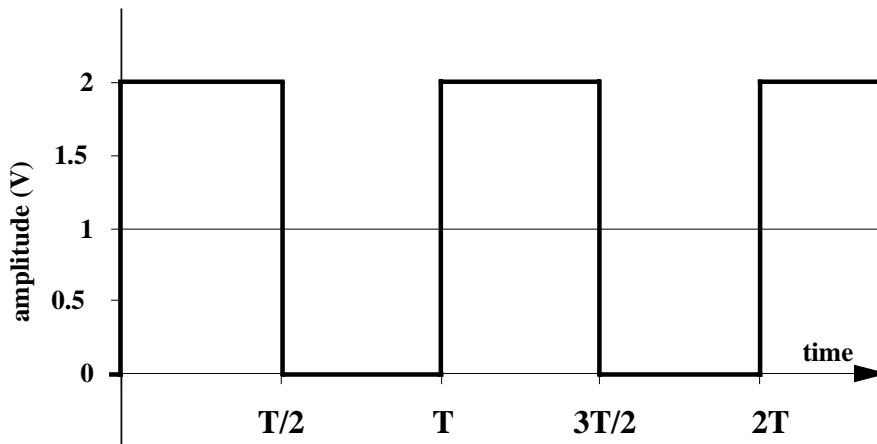


Figure 1.2 Square wave

In fact, such a square wave can be represented in terms of sinusoids as a linear combination:

$$s(t) = 1 + (4/\pi)\sin(\omega t) + (4/3\pi)\sin(3\omega t) + (4/5\pi)\sin(5\omega t) + (4/7\pi)\sin(7\omega t) + \dots$$

$$= a_0 + \sum_{n=1}^{\infty} b_n \sin(n\omega t)$$

where  $a_0$  is the average value of  $s(t)$ , 1 in this particular case, and  $b_n = (2/n\pi)[1 - (-1)^n]$

Note here that



- (i) there are an infinite number of sinusoids in a square wave;
- (ii) the frequencies of these sinusoids are only odd multiples of  $\omega$ , which is a property of square waves with equal duration of 2's and 0's- we call such square waves as 50% *duty cycle* square waves;
- (iii) the amplitude of sinusoids in the summation decreases as their frequency increases.

We refer to the sinusoids with frequencies  $2\omega, 3\omega, 4\omega, \dots, n\omega$  as *harmonics* of the *fundamental component*,  $\sin\omega t$ .

We can obtain an approximation to a square wave by taking  $a_0$ , fundamental, and only few harmonics into the summation. As we increase the number of harmonics in the summation, the constructed waveform becomes a better representative of square wave. This successive construction of a square wave is shown in Figure 1.3.

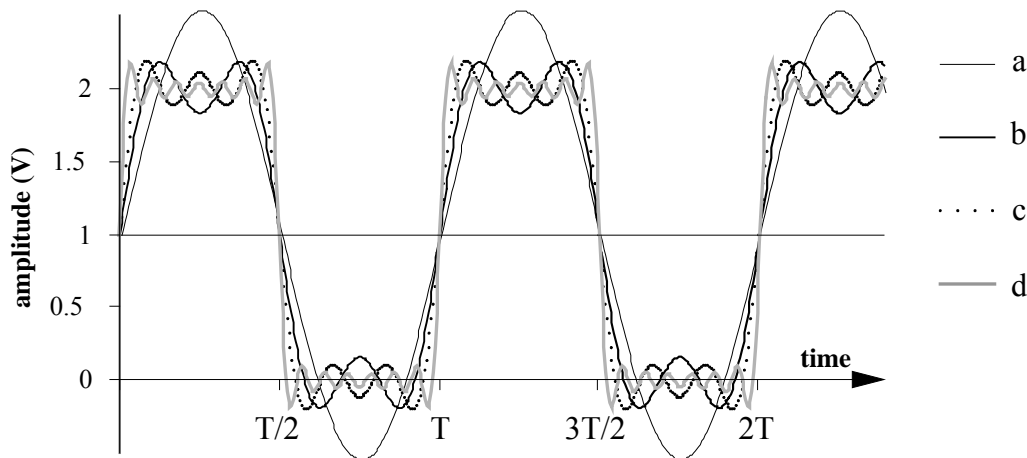


Figure 1.3 Constructing a square wave from harmonics, (a) only  $a_0$ + fundamental, (b) waveform in (a) + 3<sup>rd</sup> harmonic, (c) waveform in (b) + 5<sup>th</sup> harmonic, (d) all terms up to 13<sup>th</sup> harmonic.

Even with only 3 terms the square wave is reasonably well delineated, although its shape looks rather corrugated.

A common graphical representation of a signal with many sinusoidal components is to plot the line graph of the amplitude of each component versus frequency (either  $f$  or  $\omega$ ). This is called the *spectrum* of the square wave or its *frequency domain representation*. Spectrum of this square wave is given in Figure 1.4, which clearly illustrates the frequency components of the square wave.

Figure 1.4 clearly shows that the square wave, being a periodic signal, has energy only at discrete frequencies.

We need sinusoidal voltages in TRC-10, not square waves. Indeed, we must avoid the harmonics of our signals to be emitted from our transceiver, because such harmonics will interfere with other communication systems operating at that frequency. We use this fixed frequency square wave oscillator module, because such modules provide a

very accurate and stable frequency of oscillation and can be obtained at a low cost. We first filter out the harmonics of the waveform, when we use this module in our circuit.

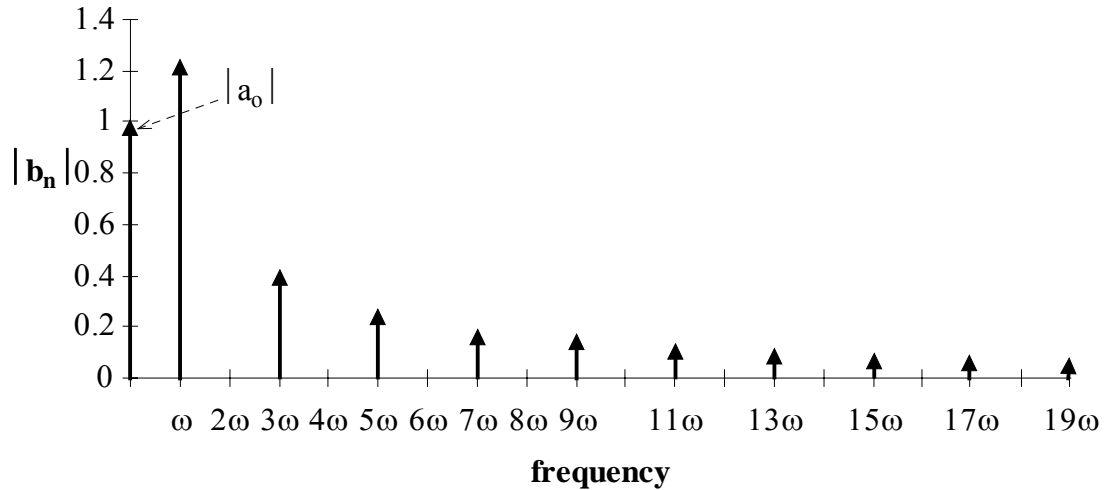


Figure 1.4 Spectrum of the square wave

The other oscillator is a *Variable Frequency Oscillator* (VFO). This oscillator produces a sinusoid of frequency that varies between 12 MHz and 13.7 MHz. This frequency is controlled by a d.c. voltage. We discuss the VFO in Chapter 7.

### 1.3. Modulation

A sinusoidal waveform does not carry any information on its own. In order to transmit any information, in our case voice, we need to make one parameter of a sinusoid dependent on this information.

In electronics, the information must of course be converted into an electrical signal, a voltage or a current, first. For example, a microphone converts sound into a voltage, which we first amplify and then use as information signal,  $v_m(t)$ . This signal is again a time varying signal, which can be represented as a linear combination of sinusoids. The frequency band of this signal, however, is not suitable for transmission in air, as it is. This frequency band is rather low for transmission and it is called *base-band*. The information signal occupying this band is referred to as *base-band signal*. Converting the information-carrying signal to a form suitable for (electromagnetic) transmission is called *modulation*.

There are three parameters that we can play with in a sinusoid: amplitude, frequency and phase. We must mount the information signal  $v_m(t)$  on a sinusoid of appropriate high frequency, so that it can be *carried* on air at that radio frequency. We call this operation, modulation. In TRC-10 we use *amplitude modulation (AM)*, which means that we make the amplitude of a sinusoid dependent on  $v_m(t)$ . Note that the frequencies of sinusoids that are present in a voice signal is within few kHz, while we wish to transmit this voice signal at 30 MHz. Let us assume that  $v_m(t)$  is a simple

signal,  $V_m \cos(\omega_m t)$ . In order to modulate the amplitude of a *carrier signal*,  $V_c \cos(\omega_c t)$ , we construct the signal,

$$v(t) = V_c \cos(\omega_c t) + v_m(t) \cos(\omega_c t) = V_c \left( 1 + \frac{V_m}{V_c} \cos(\omega_m t) \right) \cos(\omega_c t)$$

$v_m(t)$  is called the *modulating signal*. In AM, the maximum peak variation of  $|v_m(t)|$  must always be less than  $V_c$ , otherwise it cannot be *demodulated* by simple *envelope detector* circuits which can be used with this modulation scheme, and  $v_m(t)$  cannot be recovered.  $V_c [1 + (V_m/V_c) \cos(\omega_m t)]$  part in AM signal is called the *envelope*. An AM signal is depicted in Figure 1.5(b).

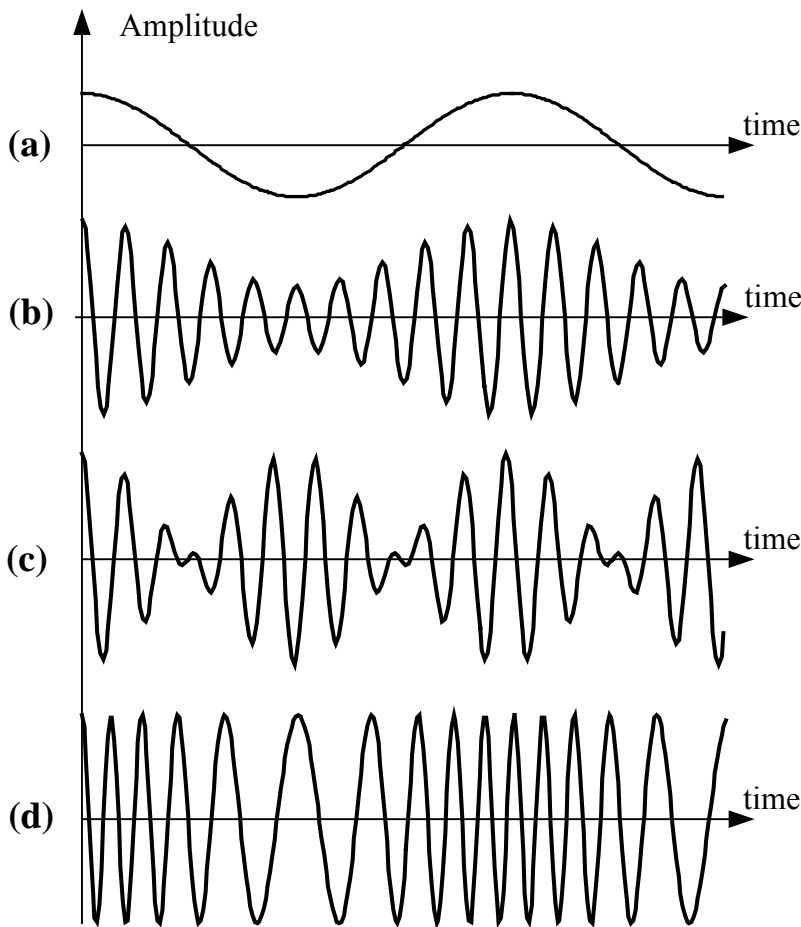


Figure 1.5 Modulated signals: (a) modulating signal, (b) AM signal, (c) DSBSC AM signal and (d) FM signal

The *depth of modulation* is determined by the maximum value of the normalized modulation signal  $|v_m(t)/V_c|$ . If

$$\left| \frac{v_m(t)}{V_c} \right|_{\max} = 1,$$

AM signal is said to have *100% modulation*, or *modulation index is one*. Similarly, if this maximum is, for example, 0.5, then the modulation index is 0.5 and the AM signal has 50% modulation.

A special case of AM is *double side-band suppressed carrier AM (DSBSC AM)*,

$$v(t) = v_m(t)\cos(\omega_c t) = V_m \cos(\omega_m t)\cos(\omega_c t).$$

In this scheme a different demodulation must be used. A DSBSC AM waveform is shown in Figure 1.5(c).

Other parameters that can be modulated in a sinusoid are frequency and phase. In analog communication systems different forms of amplitude modulation and frequency modulation (FM) are used. In FM, we construct the following signal:

$$v(t) = V_c \cos\{\omega_c t + k_f \int v_m(t) dt\} = V_c \cos\{\omega_c t + \beta V_m \sin(\omega_m t)\},$$

such that  $\beta = k_f/\omega_m$ , and  $\omega(t) = d\{\theta(t)\}/dt = \omega_c + k_f v_m(t)$ .

Here we change the frequency of the carrier signal around the carrier frequency,  $\omega_c$ , according to the variation of modulating (information) signal. An FM modulated signal is shown in Figure 1.5(d).

Long wave and middle wave radio broadcasting is done by AM, and radio broadcasting in 88-108 MHz band is done by FM. Analog terrestrial television broadcasting employs a version of AM (called *vestigial side-band AM*) for image and FM for sound.

#### 1.4. Amplifiers

The most frequently done operation on signals is amplification. The signal received at an antenna is often very weak, may be at power levels of few tens of *fW* (1 femtoWatt is 1 E-15 W). This power level corresponds to a few  $\mu V$  (*micro volt*, micro is 1 E-6) into a 50  $\Omega$ , which is a typical value of input resistance for a receiver. This signal level must be increased so that it can be demodulated, and further increased so that it can be heard. The device that performs this function is called *amplifier*. We use operational amplifiers (OPAMP) for amplification in TRC-10. Amplifiers relate the signal at their input and at their output by a gain. We are usually interested in two types of gain, the voltage gain and power gain. We shall denote voltage gain by A and power gain by G:

$$A = V_o / V_i, \text{ and}$$

$$G = P_o / P_i.$$

Gain is a quantity, which does not have any units. We use *decibels* (dB) to describe the amount of gain. We can express the same gain in decibels as,

$$A = 20 \log (V_o / V_i) \text{ dB, and}$$

$$G = 10 \log(P_o/P_i) \text{ dB.}$$

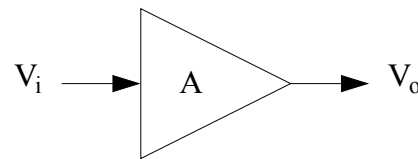
The coefficient of 10-base logarithm is 10 for power gain and 20 for voltage or current gain. With this definition, both a voltage gain and the corresponding power gain yield the same value in dB. For example, if a peak voltage of  $V_1$  appears across a resistor  $R$ , then the peak current through  $R$  is  $V_1/R$ , and average power delivered to  $R$  is  $V_1^2/2R$ . Now, if this voltage is amplified two folds and applied across the same resistor, then there is a voltage gain of  $A = 2$  and power gain of  $G = 4$ . In decibels, the value of both  $A$  and  $G$  is 6dB. Also note that 3-dB corresponds to a power gain of 2 and a voltage gain of  $\sqrt{2}$ .

Decibel is also used to define absolute levels. For example 0.5 miliwatt of power is expressed in decibels as “-3 dBm”. Here, “m” denotes that this value is relative to 1 miliwatt. Similarly, 20 Watts is expressed as “13 dBW”. Another way of writing absolute levels in decibels is to directly write what it is relative to. For example we can write 32 microvolts as “30 dB re  $\mu V$ ”.

Amplifier block diagram and some easy to remember approximate dB values are given in Figure 1.6.

dB	$A=V_o/V_i$	$G=P_o/P_i$
0	1	1
3	1.42	2
6	2	4
7	2.24	5
9	2.83	8
10	3.16	10
20	10	100

(a)



(b)

Figure 1.6 (a) gain conversion table, and (b) amplifier block diagram

### 1.5.Mixers

We want to transmit and receive voice signals, which are limited to a few kHz frequency band. We are allowed to do that at frequencies orders of magnitude higher, because all frequency bands are shared between different services and carefully regulated both nationally and internationally. We must, therefore, have the capability of shifting the frequency of information signal, up for transmission, and down for reception. Mixers do this by multiplying two signals.

Consider the amplitude modulated signal,

$$v(t) = V_c \cos(\omega_c t) + v_m(t) \cos(\omega_c t) = V_c [1 + (V_m/V_c) \cos(\omega_m t)] \cos(\omega_c t).$$

Let us define

$$m(t) = \left( \frac{V_m}{V_c} \right) \cos(\omega_m t)$$

so that

$$v(t) = V_c[1+m(t)]\cos(\omega_c t).$$

The multiplication of  $1+m(t)$  with carrier  $\cos(\omega_c t)$  can be done by a mixer. In Figure 1.7 a mixer performing this operation is shown.

The signal received by the antenna must be *mixed* down to a frequency where it is properly *filtered* and converted back to *audio* signal. This is called *heterodyne* reception.

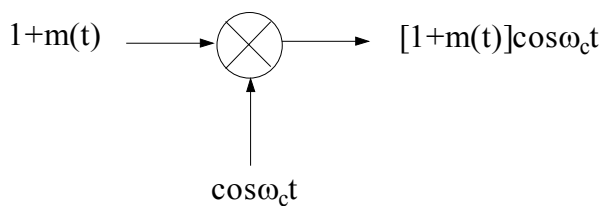


Figure 1.7 Mixer

## 1.6.Filters

Analog electronics is all about signals. Every signal occupies a frequency band in the spectrum. This band can be quite large, 8 MHz as in the case of a television signal, or very small such as for a single sine wave. The most important matter is that all signals are in the same frequency spectrum, they share the spectrum. Also apart from the signals of well-defined nature like the ones that we are interested in, there exists *natural* and *man-made noise*, which further complicates the task of electronic engineers. Whenever a particular signal is to be processed, we must make sure that we use a good and clean sample of that signal without any irrelevant and unwanted components in it. We employ *filters* for this purpose.

We use *low-pass* filters, *high-pass* filters and *band-pass* filters in TRC-10. As the names imply, low-pass filters (LPF) allow the signals below a certain frequency to pass through the filter, and *attenuate* (decrease their amplitude) the signals of higher frequency. This threshold frequency is called *cut-off* frequency. Filtering effect is not abrupt, but it is gradual. The signal components and noise beyond cut-off frequency are not completely eliminated, but attenuated more and more as their frequencies are further away from cut-off. This behavior of a LPF is shown in Figure 1.8(a). The filtering property of filters is described by their transfer function,  $H(\omega)$ .  $H(\omega)$  is a complex function (in algebraic sense) of radial frequency,  $\omega$ . At those frequencies where  $|H(\omega)|$  is equal to or close to 1, then the signals at the input of the filters pass through the filter unaffected. On the other hand, if  $|H(\omega)|$  is significantly lower than 1, the signals at those frequencies are attenuated. The cut-off frequency  $\omega_{-3dB} = 2\pi f_{-3dB}$  is defined as the frequency where  $|H(\omega)|$  is  $1/\sqrt{2}$ .

The transfer function of a filter is also described in decibels. We can plot  $|H(\omega)|$  as  $20\log|H(\omega)|$  versus  $\omega$ . In that case cut-off frequency is the frequency where  $20\log|H(\omega)|$  is  $-3\text{dB}$ .  $20\log|H(\omega)|$  of the same LPF is also depicted in Figure 1.8(b).

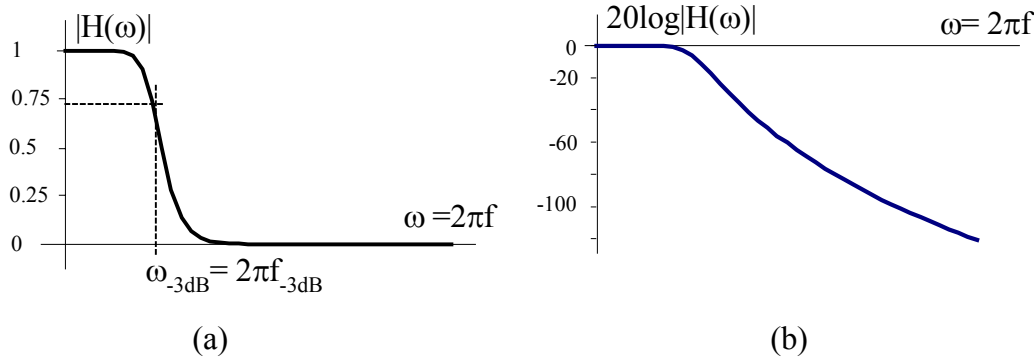


Figure 1.8 Low-pass filter (a)  $|H(\omega)|$ , and (b)  $20\log|H(\omega)|$

High-pass filters (HPF) eliminate the signals below the cut-off frequency. Band-pass filters (BPF) allow the signals that fall in a certain frequency band to pass through the filter and attenuate other signals. HPF and BPF are shown in Figure 1.9.

Again, the *bandwidth* (BW) of a BPF is defined as  $\Delta\omega = \omega_2 - \omega_1$  in rps, where  $\omega_2$  and  $\omega_1$  are the  $-3\text{dB}$  *upper* and *lower cut-off frequencies*. Often these are expressed in terms of Hz, where  $\Delta f = f_2 - f_1$ , and  $f_2$  and  $f_1$  are  $\omega_2/2\pi$  and  $\omega_1/2\pi$  respectively. The *center frequency* of a BPF,  $f_0$ , is the geometric mean of  $f_2$  and  $f_1$ . BPF usually have the lowest attenuation at  $f_0$ .

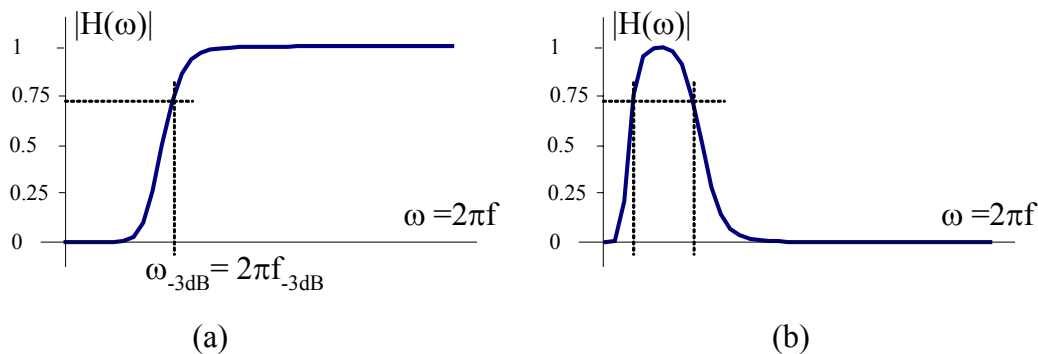


Figure 1.9 (a) High-pass filter, (b) Band-pass filter

Now consider that we wish to filter out the fundamental component of the square wave at 16 MHz, using the BPF in Figure 1.9(b). We must tune the filter so that center frequency is in the vicinity of the fundamental, and the bandwidth is sufficiently small to attenuate particularly the second harmonic sufficiently (which is nearest). This situation is depicted in Figure 1.10.

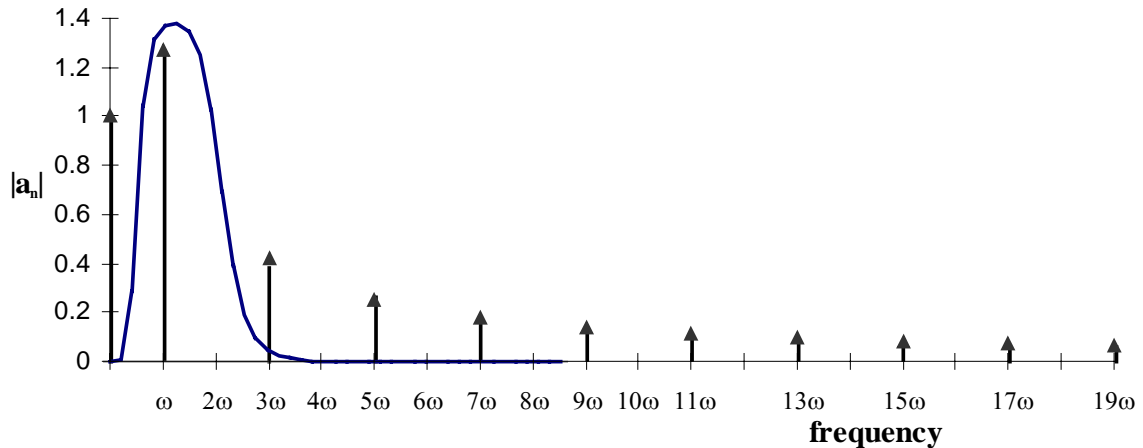


Figure 1.10 A BPF superimposed on the spectrum of the square wave

### 1.7. Receivers

We modulate a carrier to transmit the information signal over a radio frequency (*RF*). When received, we must demodulate this RF signal, and *detect* the information. We use *envelope* or *diode detectors* to do this. Envelope detectors separate the envelope from the carrier. This is the simplest way of detecting AM signals. We use envelope detector in the receiver of TRC-10.

Another way of detecting AM signals, is to reverse the modulation process. We can use mixers to demodulate AM signals, simply to multiply the received signal by an *exact* replica of the carrier signals. Indeed this is the only way to recover the information signal in DSBSC AM, and can of course be used in AM.

Frequency deviation carries the information in FM. The frequency of the received signal must be estimated in order to get the information signal, in this modulation scheme. There are different ways of doing this, among which, *FM discriminators* are the ones, which are most widely used.

When we modulate the amplitude of a carrier with a voice signal, the transmitted signal becomes:

$$v(t) = V_c \left( 1 + \frac{V_m}{V_c} \cos(\omega_m t) \right) \cos(\omega_c t)$$

where  $V_m \cos(\omega_m t)$  represents the voice signal (this would sound like a whistle) and  $\cos(\omega_c t)$  is the carrier. If we carry out the algebra, we can see that  $v(t)$  contains three sinusoids,

$$v(t) = V_c \cos(\omega_c t) + (V_m/2) \cos[(\omega_c - \omega_m)t] + (V_m/2) \cos[(\omega_c + \omega_m)t]$$

Now, assume that  $\omega_m$  is  $2\pi \times 3E3$ , i.e. a 3kHz signal (a *very* high pitch whistle sounds like this). On the other hand,  $\omega_c$  is anything between  $2\pi \times 28E6$  and  $2\pi \times 29.7E6$  in



TRC-10, let us take it as  $2\pi \times 29E6$ . Then the frequencies of the three components become 29MHz, 28.997MHz, and 29.003MHz, respectively. The information content in the modulated signal is confined to only a 6kHz bandwidth around 29MHz. This bandwidth is only 0.2% of the carrier frequency. All the signals and noise outside this bandwidth is irrelevant to this communication setup, and hence they must be eliminated as much as possible. We must use a filter with such a narrow bandwidth. Of course, if we had chosen another carrier frequency in 10-meter band, then the center frequency of the filter should be adjusted to that frequency.

It is very difficult to design a filter that both has a very narrow bandwidth and can be tuned to arbitrary center frequencies. However it is possible to have a very precise narrow bandwidth filter at a fixed frequency, easily. In this case we must shift the frequency of the received signal to the center frequency of the filter using a mixer, since we cannot let the filter to follow the signal frequency. The receivers that use this concept are called *superheterodyne receivers*. The frequency to which the carrier is shifted, is called the *intermediate frequency*, or simply *IF*, and the filter is called the *IF filter*.

TRC-10 is a superheterodyne receiver with 16MHz IF.

### 1.8. TRC-10

The block diagram of TRC-10 is given in Figure 1.11. The adventure of a voice signal through a transceiver can be observed on this figure.

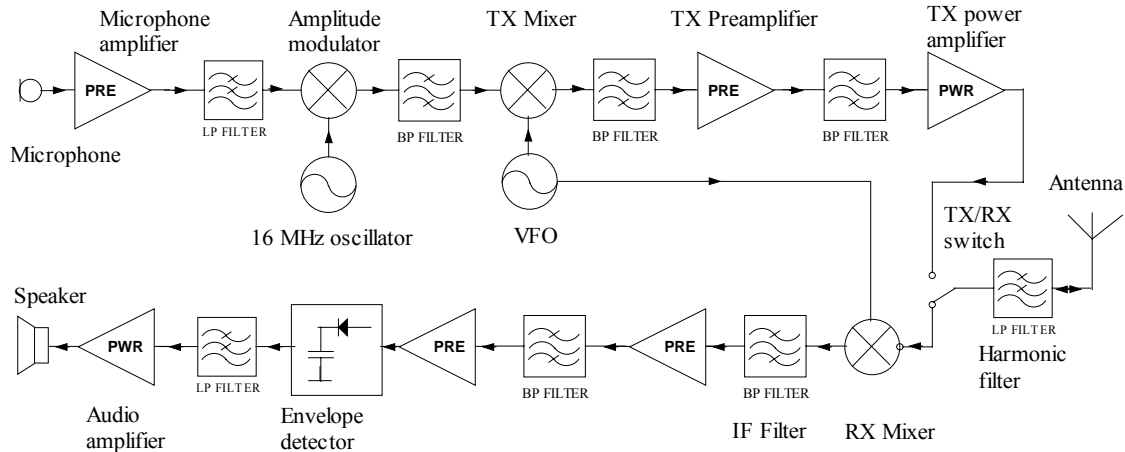


Figure 1.11 TRC-10 block diagram

Acoustic voice signal is first converted into an electrical signal at the microphone. The signal power is very low at this point, at pW level. Microphone amplifier increases the signal power level to about a mW. Voice or speech signal is not a periodic signal, and therefore its frequency spectrum is not a *line spectrum* like square wave spectrum. Speech signal has a continuous energy distribution over the frequency domain. Most of the energy contained in this signal is confined to a band between 100Hz and 3 kHz, approximately. Orchestral music, for example, is also an audible sound, but has a richer frequency spectrum. Orchestral music has a bandwidth of

approximately 10 kHz. The LPF after the microphone amplifier limits the bandwidth of the audio signal to 3 kHz, since we only speak to the transceiver microphone, and any possible noise like signals with higher frequency content are avoided.

The filtered audio signal is then used to modulate the amplitude of a 16 MHz carrier. The signal spectra at the input and at the output of the amplitude modulator are shown in Figure 1.12.

*Fourier analysis* tells us that any signal with a *real algebraic model* must have a spectrum defined for *negative* frequencies as well as for positive frequencies (with a lot of symmetry properties), in order to be mathematically correct. Audio signal is such a signal and can be represented by a real function of time,  $m(t)$ . Hence, we show the spectrum of  $m(t)$  by a two-sided spectrum in Figure 1.12, although it is customary to show a single side for positive frequencies only.

Once modulated, the voice spectrum is shifted such that it is now centered on IF frequency instead of zero. Also there is a sinusoidal carrier.

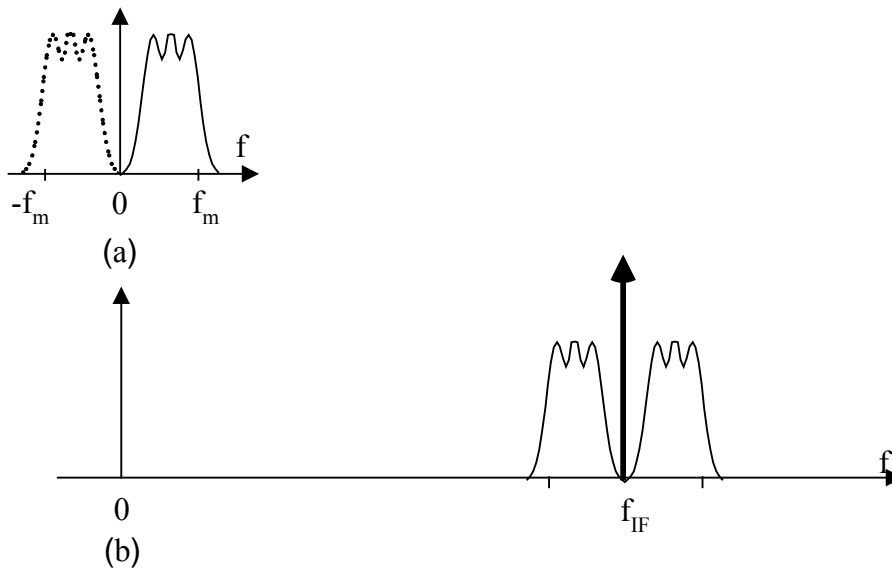


Figure 1.12 Modulator (a) input and (b) output signal spectra (not scaled)

TX mixer shifts that further to the transmission frequency,  $f_c = f_{IF} + f_{VFO}$ . This second mixing produces two spectral components. First one is the required one, and second one is at  $f_{IF} - f_{VFO}$ , which falls into the frequency band 2.3 MHz to 4 MHz. This component is an unwanted component and must be eliminated. We filter this component out by two filters, before and after the TX preamplifier. This process is depicted in Figure 1.13.

The RF signal is once more amplified by TX power amplifier to increase the power level to 40 mW. It is then filtered once more, to suppress the possible harmonics, and fed to the antenna.

The voice is now “on air”.

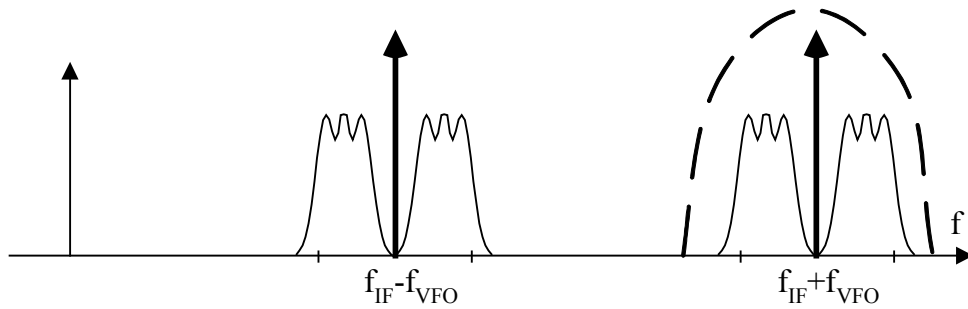


Figure 1.13 Signal at the TX Mixer output (not scaled)

When the transceiver is switched to reception mode, the diode switch isolates the antenna from TX and connects it to the RX input. Antenna receives all the signals and noise that reach to it, in air. Therefore what comes from antenna is first band pass filtered to 28-29.7 MHz, and then mixed with VFO signal down to 16 MHz. At this stage, the signal is passed through a very narrow band filter of just 6 kHz bandwidth. This ensures that we filter out all signals other than the one we are interested in.

The IF signal is then amplified by two amplifiers and then fed into the diode detector. Detector produces a replica of the envelope, which is the received voice signal. This signal is now at base band, and hence amplified in the audio amplifier before we hear it from the speaker.

## 1.9. Bibliography

- a. D.B. Rutledge's book "The Electronics of radio", published by Cambridge University Press, 1999.
- b. "The ARRL Handbook", published by ARRL, new edition every year, any recent edition.
- c. P. Nahin's book "The Science of Radio", 2<sup>nd</sup> Edition, published by Springer-Verlag.

## 1.10. Laboratory Exercises

### *Soldering exercise*

1. You will learn good soldering in this exercise. You will build a house on copper clad board using resistors and capacitors. Soldering is not gluing. Soldering is a chemical process to form an alloy of solder and the soldered metal pieces. Soldering iron must be hot and its tip must be shiny in order to make good solder joint. Put some water on the soldering sponge and keep it wet through out the soldering session. Turn the soldering iron on and wait until it is hot. Solder must immediately melt on the tip when it is hot enough. Put some solder on the tip and wipe the tip with wet sponge. The tip will shine. This process is called *tinning*. Now the iron is ready to make a solder joint. If the tip is not shiny, the heat transfer from the tip to the component is poor.

The joint to be soldered must be mechanically sturdy enough before solder is applied, so that when the solder is hot and in fluid form, the joint must not

move. Place the tip in contact with the joint, touching all parts to be soldered. Place the solder in contact with the parts (*not the tip*) opposite to the tip. Solder must melt within a second. Remove the tip and the solder.

### 1.11. Problems

1. Signal  $s(t) = 4 \cos(\omega_s t)$  is multiplied by a carrier  $c(t) = \cos(\omega_o t)$  in a mixer. Calculate the signal at the output of the mixer as sum of sine waves. Plot the magnitude of the individual sine wave components wrt frequency if  $\omega_s=1000$  rps and  $\omega_o=5000$  rps, as in Figure 1.4.
2. Let  $s(t) = \cos(\omega_s t) + 2 \cos(2\omega_s t) + 2 \cos(3\omega_s t) + \cos(4\omega_s t)$ , where  $\omega_s = 2\pi f_s$  and  $f_s = 300$  Hz.  $s(t)$  is mixed with  $\cos(\omega_o t)$  where  $f_o = 5000$  Hz. Calculate the signal at the output of the mixer as a sum of sinusoids (*no powers, no products*). Plot the magnitude of the individual sinusoidal wave components in this output signal and in  $s(t)$  wrt frequency.
3. Let  $s(t) = A(t)\cos(\omega_s t)$ .  $s(t)$  is mixed and filtered to obtain  $A(t)\cos(\omega_o t)$ . What is the signal that  $s(t)$  must be mixed with and what kind of filter is needed?
4. Show that
 
$$\tan(A + B) = \frac{\tan A + \tan B}{1 - \tan A \tan B}.$$
5. Construct a square wave with 3 and 5 components and calculate the mean square error, using anything, a spreadsheet, etc.
6. Do we need a filter at the output of amplitude modulator in TRC-10?
7. What must be the BW of the filter after detector?
8. A particular filter is experimentally measured and the following data is obtained:

$ H(\omega) $	1	0.986	0.95	0.832	0.707	0.6	0.447	0.351	0.287	0.148	0.01
f -Hz	0	500	1000	2000	3000	4000	6000	8000	10000	20000	30000

Draw  $|H(\omega)|$  with respect to  $\omega$ . Calculate  $|H(\omega)|$  in decibels and plot against  $\omega$ .



## Chapter 2 : CIRCUIT THEORY PRIMER

Electronics is not an abstract subject. Electronics is about designing and constructing instruments to fulfill a particular function, and be helpful to mankind. Electrical energy, voltage and current are all measurable, physical phenomena. Circuits are made up of resistors, capacitors, inductors, semiconductor devices, integrated circuits, energy sources, etc., in electronics. We design circuits employing these components to process electrical energy to perform a particular function. Circuits may contain a very large number of components. TRC-10 is an instrument with modest component count, yet it has about 200 components. If there should not be a set of rules, which tell how these components must be used, electronics would have never been possible. Algebra and differential equations are the tools that are used to both define the functions of elements and their inter-relations. The mathematics of circuit analysis and synthesis, models and set of rules developed for this purpose is altogether called *circuit theory*.

### 2.1. Energy sources

All circuits consume energy in order to work. Energy sources are either in form of *voltage sources* or *current sources* in electronic circuits. For example, batteries are voltage sources.

The voltage and current source symbols are shown in Figure 2.1. We need the concept of *ideal source* in order to model the real sources mathematically. An ideal voltage source is capable of providing the defined voltage across its terminals regardless of the amount of current drawn from it. This means that even if we short circuit the terminals of a voltage source and hence draw infinite amperes of current from it, the ideal voltage source, e.g. the one in Figure 2.1(a), will keep on supplying  $V_0$  to the short circuit. This inconsistent combination obviously means that the ideal supply is capable of providing infinite amount of energy. Similarly an ideal current source can provide the set current whatever the voltage across its terminals may be.

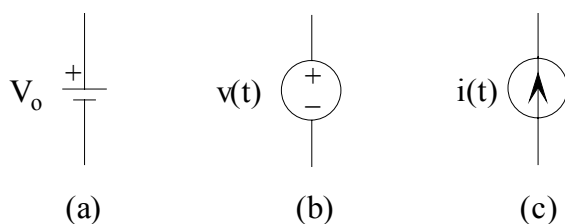


Figure 2.1 (a) d.c. voltage source, (b) general voltage source, (c) current source

Energy sources of infinite capacity are not available in nature. Should that be possible, then we should not worry about energy shortages, or on the contrary, we should be worrying a lot more on issues like global warming. The concept of ideal source, however, is very important and instrumental in the analysis of circuits.

*Real* sources deviate from ideal sources in only one aspect. The voltage supplied by a real source has a dependence on the amount of current drawn from it. For example, a battery has an internal resistance, and when connected to a circuit, its terminal voltage

decreases by an amount proportional to the current drawn from it. This is depicted in Figure 2.2(a).

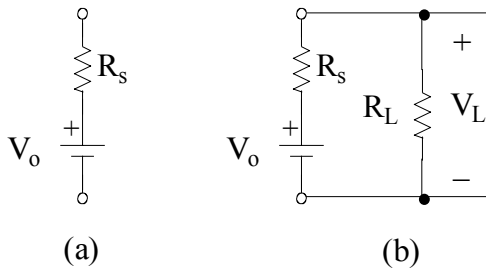


Figure 2.2 (a) Equivalent circuit of a battery, (b) a battery circuit

The internal resistance of the battery is denoted as  $R_s$ , in Figure 2.2. When there is no current drawn from the battery, the voltage across the terminals is  $V_o$ . When a *load* resistance  $R_L$ , such as a flash light bulb, is connected to this battery, the voltage across the battery terminals is no longer  $V_o$ . This circuit is given in Figure 2.2(b).

Ohm's Law says that the voltage across a resistor is proportional to the current that passes through it, and the proportionality constant is its *resistance*,  $R$ :

$$R = \frac{V}{I}.$$

$R$  is measured in *ohms*, denoted by  $\Omega$  (Greek letter omega), such that  $1V/1A=1\Omega$ . Inverse of  $R$  is called *conductance* and denoted by  $G$  and measured in terms of *siemens*,  $S$ :

$$G = \frac{I}{V} = \frac{1}{R}.$$

$1/(1\Omega)$  is  $1S$ . Kirchhoff's Voltage Law (KVL) states that, when the resistors are connected in series, the total voltage across all the resistors is equal to the sum of voltages across each resistor and the current through all the resistors is the same. Therefore in a series connection the total resistance is equal to the sum of the resistances.

In the circuit in Figure 2.2(b) we have  $R_s$  and  $R_L$  connected in *series*. Hence the total resistance that appears across  $V_o$  (the ideal source voltage) is

$$R_T = R_s + R_L$$

and the current through them is

$$I = \frac{V_o}{R_s + R_L}.$$

The voltage across the battery terminals (or  $R_L$ )  $V_L$  is  $IR_L$  or  $[V_o/(R_s + R_L)] R_L$ . This value is also equal to  $V_o - [V_o/(R_s + R_L)] R_s$ . Therefore the terminal voltage of a battery

is less than its nominal value when loaded by a resistance. Note that here we *modeled* a real source by a combination of an ideal voltage source,  $V_o$ , and a resistor,  $R_s$ .

The power drawn from the battery is

$$P_b = IV_o = \frac{V_o^2}{R_s + R_L}$$

where as the power delivered to the load,  $R_L$  is

$$P_L = IV_L = \frac{V_o^2 R_L}{(R_s + R_L)^2}.$$

The difference between  $P_b$  and  $P_L$ ,  $P_b[1 - R_L / (R_s + R_L)]$  is consumed by the internal resistor,  $R_s$ . Note that as  $R_s$  gets smaller, this power difference tends to zero.

Supplies in TRC-10 are not batteries. There are +15V, -15V and a +8V dc voltage supplies in TRC-10, all of which are obtained from mains by *a.c. to d.c conversion* and *voltage regulation*. Such voltage supplies behave differently compared to batteries. There is a specified output current limit for this kind of real voltage supplies. When the current drawn from the supply is below this limit, the terminal voltage of the supply is almost exactly equal to its nominal no-load level. When the limit is exceeded the terminal voltage drops abruptly. The regulated supplies behave almost like ideal supplies as long as the drawn current does not exceed the specified limit.

## 2.2. Resistors

Resistors dissipate energy. This means that they convert all the electrical energy applied on them into heat energy. As we increase the power delivered to a resistor it warms up.

When resistors are connected in parallel, the voltage,  $V$ , across every one of them will be the same, but each one will have a different current passing through it:

$$I_i = \frac{V}{R_i}.$$

Figure 2.3 depicts parallel-connected resistors.

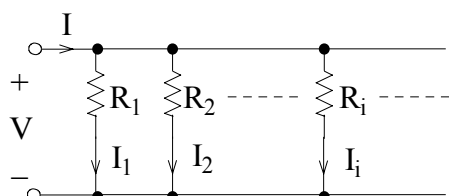


Figure 2.3 Parallel resistors



Kirchhoff's Current Law (KCL) says that the total current that flow into a node is equal to the total current that flow out of a node. The total current that flow into this parallel combination is I. Hence

$$I = \sum_i I_i$$

But  $I_i = V/R_i$  and total resistance of the parallel combination is

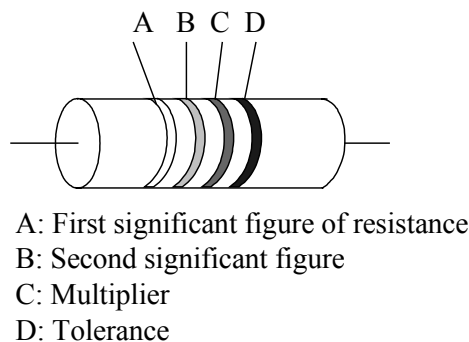
$$R_T = \frac{V}{I} = \left( \frac{1}{R_1} + \frac{1}{R_2} + \dots + \frac{1}{R_i} + \dots \right)^{-1}$$

or

$$G_T = \frac{I}{V} = G_1 + G_2 + \dots + G_i + \dots$$

We denote parallel combination of resistors as  $R_1 // R_2$  when  $R_1$  and  $R_2$  are in parallel and as  $R_1 // R_2 // R_3$  when  $R_1$ ,  $R_2$  and  $R_3$  are in parallel.

The resistors that we use in electronics are made of various materials. Most abundant are carbon resistors. There is a color code for resistance values. The resistance of a resistor is expressed in terms of a sequence of colored bands on the resistance. The color code is given in Figure 2.4.



Color	Significant figure	Multiplier	Tolerance
Black	0	E0	
Brown	1	E1	
Red	2	E2	
Orange	3	E3	
Yellow	4	E4	
Green	5	E5	
Blue	6	E6	
Violet	7	E7	
Gray	8	E8	
White	9	E9	
Gold			%5
Silver			%10
No color			%20

Figure 2.4 Resistor color codes

Most of the common resistors are available in standard values. The two significant figures of standard resistor values are:

10, 12, 15, 18, 22, 27, 33, 39, 47, 56, 68, and 82.

Hence, a 100  $\Omega$  resistor is marked as brown-black-brown and a 4.7 K $\Omega$  resistor is marked as yellow-violet-red.

## 2.2.1. Resistive circuits

Electrical circuits can have resistors connected in all possible configurations. Consider for example the circuit given in Figure 2.5(a). Two resistors are connected in series, which are then connected in parallel to a third resistor, in this circuit.

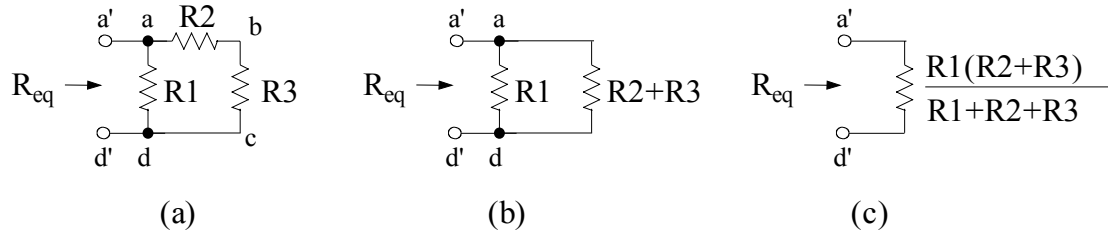


Figure 2.5 (a) Resistive circuit, (b) series branch reduced to a single resistance and (c) equivalent resistance.

In order to determine the overall equivalent resistance  $R_{eq}$ , which appears across the terminal  $a'-d'$ , we must first find the total resistance of  $a-b-c$  branch.  $R2$  and  $R3$  are connected *in series* in this branch. The total branch resistance is  $R2+R3$ . The circuit decreases to the one given in Figure 2.5(b). Only two resistances,  $R1$  and  $(R2+R3)$ , are connected in parallel in this circuit. Hence,  $R_{eq}$  can be determined as

$$R_{eq} = \left[ \frac{1}{R1} + \frac{1}{R2+R3} \right]^{-1} = \frac{R1(R2+R3)}{R1+R2+R3}.$$

Another resistive circuit is depicted in Figure 2.6(a). In this case we must reduce the two parallel resistors across the terminals  $b-c$  into a single resistance first, as shown in Figure 2.6(b).  $R1$  is now series with the parallel combination of  $R2$  and  $R3$ .  $R_{eq}$  can now be found readily as

$$R_{eq} = R1 + \frac{R2 R3}{R2 + R3},$$

as shown in Figure 2.6(c).

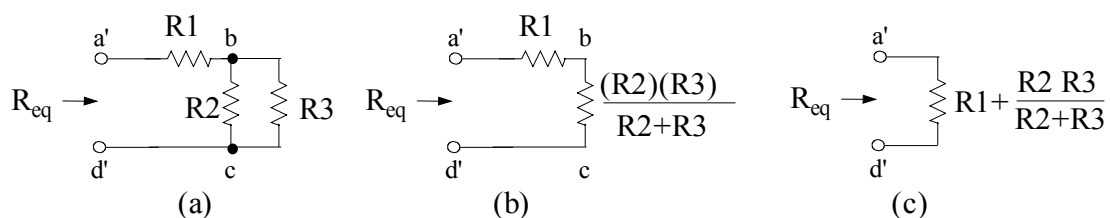


Figure 2.6 A resistive circuit

## 2.3. Analysis of electrical circuits

The knowledge of the value of current through each branch or the voltage across each element is often required. The circuits are analyzed to find these quantities. There are two methods of analysis. The first one is the *node-voltage* method, or *node analysis*.

A node is a point in the circuit where more than two elements are connected together. For example  $b$  in Figure 2.6(a) is a node but  $b$  in Figure 2.5(a) is not.

We follow a procedure outlined below to carry out the node analysis:

1. Select a common node so that all other node voltages are defined with respect to this node. Usually zero-potential node or *ground* node is taken as common node in electrical circuits.
2. Define the voltage difference between the common and all other nodes as unknown *node voltages*.
3. Write down the KCL at each node, expressing the branch currents in terms of node voltages and sources.
4. Solve the equations obtained in step 3 simultaneously.
5. Find all branch currents and voltages in terms of node voltages.

Consider the circuit in Figure 2.5(a), with a current source connected across terminals  $a'-d'$ . This circuit is given in Figure 2.7(a) with numerical values assigned to circuit parameters.

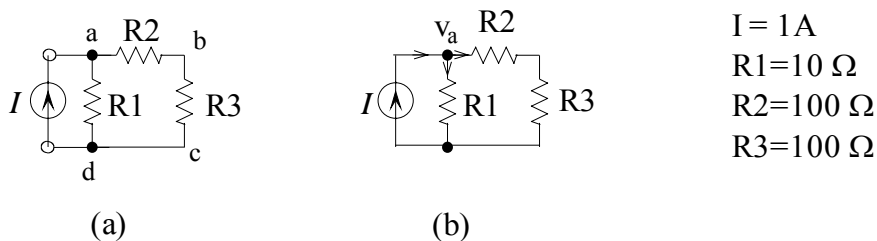


Figure 2.7 Node analysis

Let us analyze this circuit to find all element voltages and currents using node analysis procedure:

1. Choice of common node is arbitrary; we can choose either  $a$  or  $d$ . Choose  $d$ .
2. Define the node voltage  $v_a$  as the potential at node  $a$  minus the potential at node  $d$ , as shown in Figure 2.7(b).
3. KCL at node  $a$  (considering the assumed directions of flow):  
Source current (flowing into the node) = Current through  $R_1$  (flowing out) + Current through  $R_2$  and  $R_3$  (flowing out)

or

$$I = \frac{v_a}{R_1} + \frac{v_a}{R_2 + R_3}.$$

4. Solve for  $v_a$  in terms of the source current and resistors:

$$v_a = \frac{I}{\frac{1}{R_1} + \frac{1}{R_2 + R_3}} = I \frac{R_1(R_2 + R_3)}{R_1 + R_2 + R_3}$$

$$= 9.52 \text{ V.}$$

5. Branch currents in terms of  $v_a$  are already given in step 3. Current flowing through  $R_1$  is  $v_a/R_1$ , or 0.952 A, flowing from node  $a$  to node  $d$ . Current flowing through  $R_2$  and  $R_3$  is  $v_a/(R_2+R_3)$ , or 47.6 mA.

The voltage across  $R_3$  is  $(R_3) [v_a/(R_2+R_3)] = (100 \Omega)(47.6 \text{ mA}) = 4.76 \text{ V}$ . Similarly, voltage across  $R_2$  can be found as 4.76 V, either from  $(R_2) [v_a/(R_2+R_3)]$  or from  $v_a - (R_3) [v_a/(R_2+R_3)]$ .

Hence all currents and voltages in the circuit are determined.

Another example is the circuit in Figure 2.6(a), driven by a voltage source as shown in Figure 2.8(a).

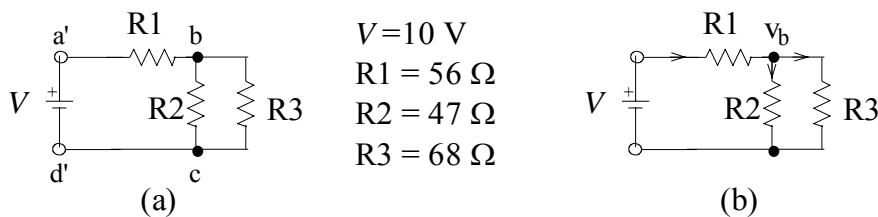


Figure 2.8 Node analysis

Node analysis procedure for this circuit is as follows:

1. Chose node  $c$  as common.
2. Define the node voltage  $v_b$ .
3. KCL at node  $b$ :  
Current through  $R_2$  (out of node) + Current through  $R_3$  (out of node) = Current through  $R_1$  (from the supply to the node)  
Or  

$$\frac{v_b}{R_2} + \frac{v_b}{R_3} = \frac{V - v_b}{R_1}$$
4. Solve for  $v_b$ :  

$$v_b = V (R_1 \parallel R_2 \parallel R_3) / R_1 = 3.3 \text{ V}.$$
5. Voltage across  $R_1$  is  $V - v_b = 6.7 \text{ V}$ . Current through  $R_2$  and  $R_3$  are  $v_b/R_2 = 70.6 \text{ mA}$  and  $v_b/R_3 = 48.8 \text{ mA}$ , respectively.

There is only one node voltage and therefore only one equation is obtained in step 3, in both of above examples. Consider the circuit in Figure 2.9(a).

Node analysis for this circuit is as follows:

1. Connection points  $d$ ,  $e$  and  $f$  are the same node in this circuit. Chose this point as common node.
2. Two other nodes are  $b$  and  $c$ . Define  $v_b$  and  $v_c$ .
3. KCL at node  $b$ :  
Choice of current directions is arbitrary (if the initial choice is opposite to that of actual flow for a branch, the solution comes out with negative sign). Let us

choose current directions such that all branch currents flow out of node  $b$  and all branch currents flow into node  $c$ , as shown in Figure 2.9(b).

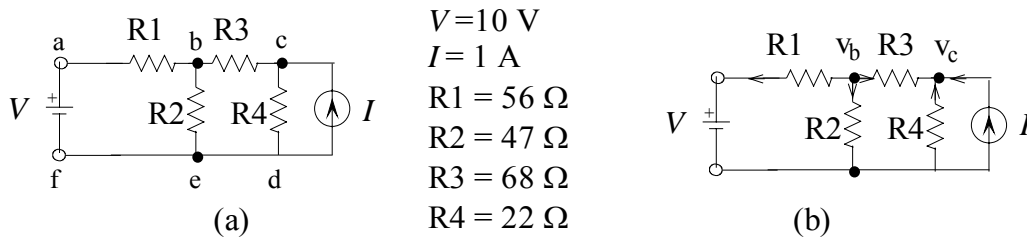


Figure 2.9 Two-node circuit

All currents coming out of node  $b = 0$ , or

$$\frac{v_b - V}{R1} + \frac{v_b}{R2} + \frac{v_b - v_c}{R3} = 0 \Rightarrow v_b \left( \frac{1}{R1} + \frac{1}{R2} + \frac{1}{R3} \right) - \frac{v_c}{R3} = \frac{V}{R1}.$$

KCL at node  $c$ :

All currents flowing into node  $c = 0$ , or

$$\frac{v_b - v_c}{R3} + \frac{-v_c}{R4} + I = 0 \Rightarrow \frac{v_b}{R3} - v_c \left( \frac{1}{R3} + \frac{1}{R4} \right) = -I.$$

4. KCL yields two equations for node voltages  $v_b$  and  $v_c$ , in terms of known resistances and source values. Solving them simultaneously yields  $v_b = 8.4 \text{ V}$  and  $v_c = 18.6 \text{ V}$ .
5.  $R1$  current (in the chosen direction) is  $(v_b - V)/R1 = -28.4 \text{ mA}$ ;  
 $R2$  current (in the chosen direction) is  $v_b/R2 = 178.7 \text{ mA}$ ;  
 $R3$  current (in the chosen direction) is  $(v_b - v_c)/R3 = -150 \text{ mA}$ ;  
 $R4$  current (in the chosen direction) is  $-v_c/R4 = -845.5 \text{ mA}$ .  
 The branch currents with negative values flow in the direction opposite to the one chosen initially.

The other method of circuit analysis is called the *mesh* analysis. In this method, the currents around the loops in the circuit are defined as mesh currents, first. Then KVL is written down for each mesh in terms of mesh currents. The two methods are mathematically equivalent to each other. Both of them yield the same result. Mesh analysis is more suitable for circuits, which contain many series connections. Node analysis, on the other hand, yields algebraically simpler equations in most electronic circuits.

## 2.4. Capacitors

Capacitors are charge storage devices. In this respect they resemble batteries. However capacitors store electrical charge again in electrical form, whereas batteries

store electrical energy in some form of chemical composition. Charge,  $Q$ , stored in a capacitor is proportional to the voltage,  $V$ , applied across it:

$$C = \frac{Q}{V},$$

where  $C$  is the *capacitance* of the capacitor, and is measured in farads (F), and charge  $Q$  is in *coulombs*. Note that capacitance differs from a resistance, where current through a resistance is proportional to the voltage across it.

The relation between the charge on a capacitor and the current through it is somewhat different and *time dependent*. If we let a current,  $i(t)$ , of arbitrary time waveform to pass through a capacitor, the amount of charge accumulated on the capacitor within a time interval, e.g.  $(0, t_1)$ , is given as

$$Q = \int_0^{t_1} i(t) dt.$$

If  $i(t)$  is a d.c. current,  $I_{dc}$ , then the charge accumulated on the capacitor is simply  $Q = I_{dc} t_1$ . For example a d.c. current of 1 mA will accumulate a charge of  $10^{-9}$  coulomb on a capacitor of 1  $\mu$ F in 1  $\mu$ -second. This charge will generate 1mV across the capacitor.

More generally this relation is expressed as

$$Q(t) = \int_0^t i(\xi) d\xi,$$

where the difference between the real (may be *present* or *observation*) time instant  $t$  and the integration time variable  $\xi$ , which operates on all passed time instances between 0 and  $t$ , is emphasized. Using  $C = Q/V$ , we can now relate the voltage across a capacitor and the current through it:

$$v(t) = \frac{1}{C} \int_0^t i(\xi) d\xi.$$

If we differentiate both sides of this equation with respect to  $t$ , we obtain

$$i(t) = C \frac{d}{dt} v(t).$$

Hence the current through a capacitor is proportional to the time derivative of voltage applied across it.

There are two major types of capacitors. The first type is *non-polar*, i.e. the voltage can both be positive and negative. Most of the capacitors of lower capacitance value are of this type. However as the capacitance values become large, it is less costly to use capacitors, which has polarity preferences, like *electrolytic* or *tantalum* capacitors. For these capacitors the voltage must always remain positive in the sense indicated. Symbols of both types are depicted in Figure 2.10(a).

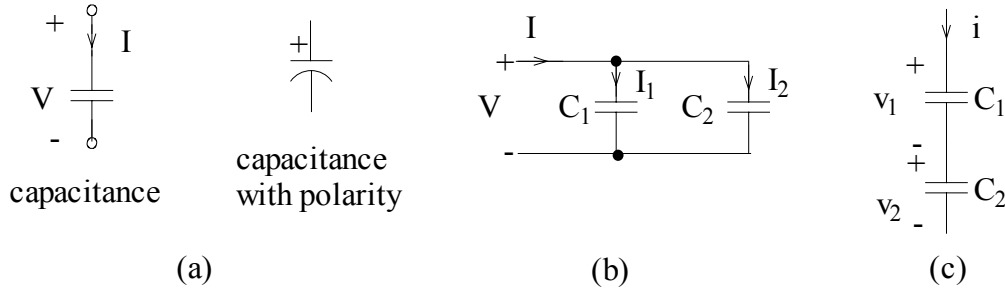


Figure 2.10 Capacitor circuits. (a) capacitance, (b) parallel connected capacitances, and (c) series connected capacitances

When capacitors are connected in parallel, as shown in Figure 2.10(b),  $I_1$  will charge  $C_1$  and  $I_2$  will charge  $C_2$ . If  $Q_1$  and  $Q_2$  are the charges accumulated on these two capacitors respectively, then the total charge is,

$$Q = Q_1 + Q_2 = C_1 V + C_2 V = (C_1 + C_2) V.$$

Hence when capacitors are connected in parallel, total capacitance is  $C = C_1 + C_2$ .

KVL tells us that in any loop, the sum of branch voltages is zero. When the capacitors are connected in series, as shown in Figure 2.10(c), sum of branch voltages is  $v_1 + v_2 = v$  and hence,

$$v = v_1 + v_2 = \frac{1}{C_1} \int_0^t i(\xi) d\xi + \frac{1}{C_2} \int_0^t i(\xi) d\xi = \left[ \frac{1}{C_1} + \frac{1}{C_2} \right] \int_0^t i(\xi) d\xi$$

Therefore, when the capacitors are connected in series, total capacitance is

$$C = \left[ \frac{1}{C_1} + \frac{1}{C_2} \right]^{-1}$$

#### 2.4.1. Power and energy in capacitors

The power delivered to the capacitor is

$$P(t) = v(t)i(t) = C v(t) \frac{dv(t)}{dt}.$$

$P(t)$  can be written as

$$P(t) = \frac{d}{dt} \left( C \frac{v^2}{2} \right)$$

which cannot contain a non-zero average term. Therefore, the average power delivered to a capacitor is always zero, whatever the voltage and current waveforms may be. Physically this means that capacitors cannot dissipate energy.

In case of sinusoidal voltages, with

$$v(t) = V_1 \sin(\omega t + \theta), \text{ and}$$

$$i(t) = C \frac{dv(t)}{dt} = \omega C V_1 \cos(\omega t + \theta),$$

$P(t)$  becomes

$$P(t) = (\omega C V_1^2 / 2) \sin(2\omega t + 2\theta).$$

On the other hand, capacitors store energy. If we sum up (or integrate) the power delivered to the capacitor we must obtain the total energy delivered to it. Assuming that initially capacitor has no charge, i.e.  $v_C(0) = 0$ , and  $v(t)$  is applied across the capacitor at  $t=0$ , the energy delivered to the capacitor at time  $t_1$  is,

$$E = \int_0^{t_1} P(t) dt = \int_0^{t_1} d\left\{C \frac{v^2(t)}{2}\right\} = C \frac{v^2(t_1)}{2}$$

If the applied voltage across the capacitor is a d.c. voltage  $V_{dc}$ , the energy stored in the capacitor is

$$E = C \frac{V_{dc}^2}{2}.$$

#### 2.4.2. RC circuits

We combine resistors and capacitors in electronic circuits. When a resistor is connected to a charged capacitor in parallel, as in Figure 2.11, the circuit voltages become a function of time. Assume that initially capacitor is charged to  $V_0$  volts (which means that it has  $Q = CV_0$  coulombs stored charge). At  $t=0$  we connect the resistance,  $R$ . KCL says that

$$i = C \frac{d}{dt} v_C(t) = C \frac{d}{dt} v(t)$$

and

$$i = \frac{v_R}{R} = -\frac{v}{R}$$

since  $v_C = v$  and  $v_R = -v$  at all times. In other words

$$\frac{dv}{dt} + \frac{v}{RC} = 0.$$

This equation is called a *first order differential equation*. Its solution for  $t \geq 0$  is



$$v(t) = V_0 \exp(-t/RC) \quad \text{for } t \geq 0.$$

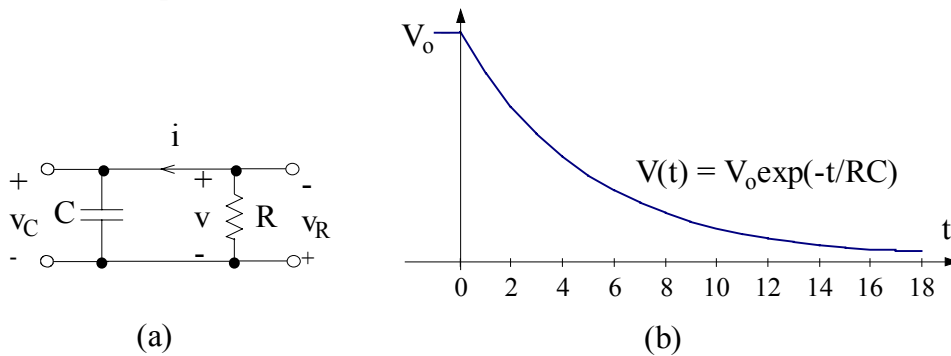


Figure 2.11 RC circuit

$v(t)$  is shown in Figure 2.11(b). Above expression tells that as soon as the resistor is connected the capacitor voltage starts decreasing, i.e. it discharges on R. The speed with which discharge occurs is determined by  $\tau = RC$ .  $\tau = RC$  is called *time constant* and has units of time ( $1\Omega \times 1F = 1$  second).

The current, on the other hand, is

$$i(t) = C \frac{dv}{dt} = -\frac{V_0}{R} \exp(-t/RC) \quad \text{for } t \geq 0$$

Also  $v_C(t) = -v_R(t) = v(t)$ .

We found the above differential equation by using KCL. We could use KVL, in which case sum of the voltages,  $v_C$  and  $v_R (=iR)$ , in the loop must yield zero:

$$v_C + v_R = v_C + iR = v + RC \frac{dv}{dt} = 0,$$

which is the same differential equation. Note that we used a *sign convention* here. *Once we have chosen the direction of current, the sign of the voltage on any element must be chosen such that the positive side is the one where the current enters the element.* Hence in this circuit the current is chosen in the direction from R to C at top, and thus the polarity of capacitor voltage  $v_C$  is similar to  $v$ , whereas the polarity of the resistor voltage  $v_R$  must be up side down. On the other hand, current actually flows from C to R on top of the figure, in order to discharge C. Therefore the value of the current is found negative, indicating that it flows in the direction opposite to the one chosen at the beginning.

Usually we do not know the actual current directions and voltage polarities when we start the analysis of a circuit. We assign directions and/or polarities arbitrarily and start the analysis. But we must carefully stick to the above convention when writing down KVL and KCL equations, in order to find both the values and the signs correctly, at the end of the analysis. The importance of this matter cannot be over emphasized in circuit analysis, indeed in engineering.

The *magnitude* of the current in the above circuit is at its maximum,  $V/R$ , initially, and decreases towards zero as time passes. This is expected, as the voltage across the capacitor similarly decreases.

If we connect a voltage source,  $v_s(t)$ , in parallel to the circuit in Figure 2.11, then the voltage on the capacitance is  $v_s(t)$  directly. This means that since voltage source can supply indefinite amount of current when demanded, capacitor can charge up to the value of the voltage source at any instant of time without any delay. Similarly the current through the resistor is simply  $v_s(t)/R$ . This is shown in Figure 2.12(a).

Here  $v_C(t) = -v_R(t) = v_s(t)$  at all times, and  $i = -v_s(t)/R$ .

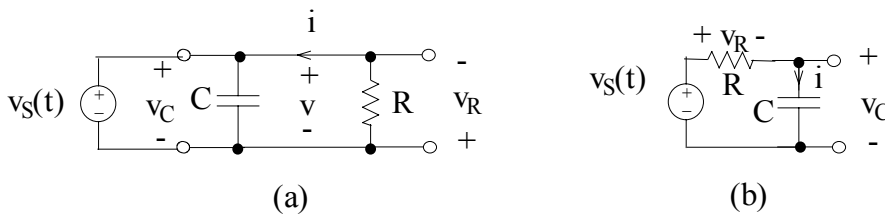


Figure 2.12 Parallel and series RC circuit

Figure 2.12(b) shows the case when  $R$  is connected in *series* to the capacitor instead of *parallel* in the same circuit. Assumed polarities (of voltage) and directions (of current) are clearly shown. The simplest way of analyzing this circuit is to apply KVL:

$$v_C(t) + v_R(t) - v_s(t) = 0.$$

The terminal relations of the two components are:

$$v_R(t) = R i(t)$$

and

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

with given polarities and directions. Substituting the terminal relations into KVL equation, we obtain,

$$v_C(t) + RC \frac{dv_C(t)}{dt} = v_s(t).$$

The solution of this equation for a general time function  $v_s(t)$  requires the knowledge of differential equations. However, we do not need this knowledge for the scope of this book. We shall confine our attention to the types of functions that represents the signals we shall use in TRC-10. Let us assume that  $v_s(t)$  is zero until  $t = 0$  and is a constant level  $V_S$  afterwards. Such time waveforms are called *step functions*. This is

similar to a case of replacing  $v_S(t)$  with a battery and a switch connected in series with it, and the switch is closed at  $t=0$ . This is modeled in Figure 2.13(a).

In order to find how  $v_C(t)$  behaves for all  $t>0$ , we must know its value just before the switch is closed. We call this value, the *initial value*,  $v_C(0)$ . Let us also assume that  $v_C(0) = 0$ . We can see that  $i(t) = 0$  and  $v_C(t) = v_R(t) = 0$  for  $t<0$  by inspecting Figure 2.12(b). When the supply voltage jumps up to  $V_S$  at  $t = 0$ , current  $i(t)$  starts flowing from the supply to the capacitor through  $R$ . Hence the circuit current magnitude cannot be any larger than  $V_S/R$ .  $R$  limits the amount of current that capacitor can drain from the supply, because it is connected in series. Hence the voltage across the capacitor, in this case cannot follow the changes in the supply voltage immediately.

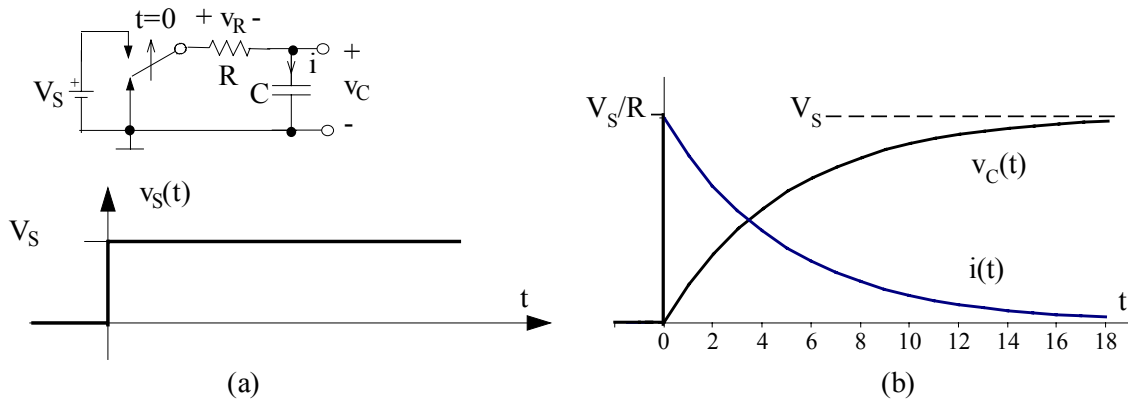


Figure 2.13 (a)  $v_S(t)$  as step voltage, (b)  $i(t)$  and  $v_C(t)$  vs. time

Initially, just after  $t = 0$ , the amount of current in the circuit is  $i(0) = [V_S - v_C(0)]/R = V_S/R$ . With this initial current capacitor starts charging up until the amount of charge accumulated on it reach to  $Q = CV_S$ . When this happens, the voltage across the capacitance is equal to that of the supply and it cannot charge any more. Hence there must not be any current flowing through it, which means that  $i(t)$  must become zero eventually.

As a matter of fact, if we write the KVL equation

$$v_C(t) + RC \frac{dv_C(t)}{dt} = V_S$$

for  $t>0$ , in terms of  $i(t)$  instead of  $v_C(t)$  (i.e. differentiating the entire equation first and then substituting  $i(t)/C$  for  $dv_C(t)/dt$ ), we have

$$i(t) + RC \frac{di(t)}{dt} = 0.$$

In order to obtain this, we first differentiate  $v_C(t)$  equation and then substitute  $i(t)/C$  for  $dv_C/dt$ . This equation is similar to the case of parallel RC, and its solution is

$$i(t) = \begin{cases} 0 & \text{for } t < 0 \\ (V_S/R) \exp(-t/RC) & \text{for } t \geq 0 \end{cases}$$

since the initial value of  $i(t)$  is  $V_S/R$ , as we determined by inspection above. Note that  $i(t)$  becomes zero as  $t$  becomes indefinitely large. Having found  $i(t)$ , we can write down the circuit voltages by direct substitution:

$$v_R(t) = R i(t) = \begin{cases} 0 & \text{for } t < 0 \\ V_S \exp(-t/RC) & \text{for } t \geq 0 \end{cases}$$

and

$$v_C(t) = \frac{1}{C} \int_0^t i(\xi) d\xi \\ = V_S [1 - \exp(-t/RC)] \quad \text{for } t \geq 0$$

since we assumed  $V_C(0)=0$  above.  $i(t)$  and  $v_C(t)$  are depicted in Figure 2.13(b).

## 2.5. Diodes

Diodes are *semiconductor* devices. They are nonlinear resistors, resistance of which depends on the voltage across them. We use different types of diodes in TRC-10 circuit:

1N4001	silicon power diode
1N4448	silicon signal diode
MPN3404 (or BA479)	PIN diode
KV1360NT	variable capacitance diode (varicap)
	zener diode

and a diode bridge rectifier. The I-V characteristics of a diode can be well approximated by an exponential relation:

$$I_D = I_S (\exp(V_D/\gamma) - 1)$$

where  $I_D$  and  $V_D$  are current and voltage of the diode and  $I_S$  and  $\gamma$  are the physical constants related to the material and construction of the diode. The symbol for a diode and typical I-V characteristics of a *silicon* diode are given in Figure 2.14.

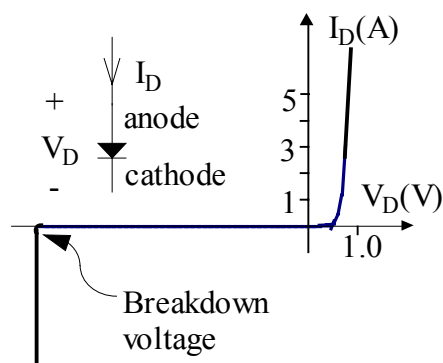


Figure 2.14 Diode

$V_D$  is defined as the voltage difference between *anode* and *cathode* terminals of the diode. This I-V characteristics shows that the current through a diode is effectively zero as long as the voltage across it is less than approximately 0.7 volt, i.e. it can be assumed open circuit. The current increases very quickly when the voltage exceeds 0.7 volt, hence it behaves like a short circuit for these larger voltages. Also note here that there is a negative voltage threshold for  $V_D$ , determined by the *breakdown voltage* in real diodes, below which the diode starts conducting again. This breakdown voltage is usually large enough such that the magnitudes of all prevailing voltages in the circuit are below it, and hence it can be ignored. In *zener diodes*, however, this breakdown voltage is employed to stabilize d.c. voltage levels.

It is useful to introduce the concept of *ideal diode* at this stage in order to model real diodes in circuit theory. Ideal diode is a device, which is an open circuit for all negative voltages and is a short circuit for positive voltages. The symbol and I-V characteristic of an ideal diode is shown in Figure 2.15.

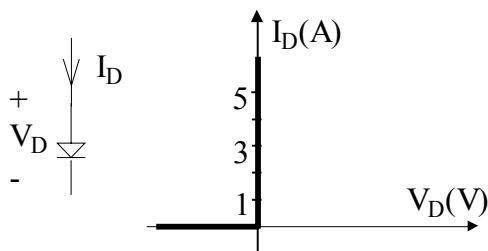


Figure 2.15 Ideal diode equivalent circuit and characteristics

Having defined the ideal diode, we can now develop approximate *equivalent circuits* for diodes. The simplest one is an ideal diode and a voltage source connected in series. In this model the voltage source represents the *threshold voltage*  $V_o$  ( $\approx 0.7$  V), we observed previously. This equivalent circuit and its I-V characteristics are shown in Figure 2.16(a).

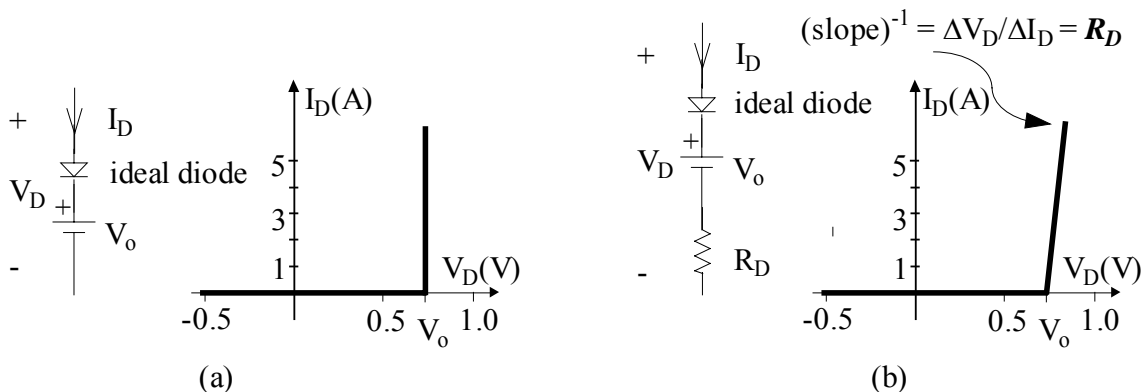


Figure 2.16 Piecewise linear diode equivalent circuits and characteristics

The equivalent circuit given in Figure 2.16(a) is a good enough approximation for many applications. However when the *on resistance* of the diode,  $R_D$  (i.e. the incremental resistance when  $V_D$  is larger than  $V_o$ ), becomes significant in a circuit, we can include  $R_D$  to the equivalent circuit as a series resistance as shown in Figure

2.16(b). A diode is called “*ON*” when it is conducting, and “*OFF*” otherwise. Note that  $R_D$  is zero in the simpler model.

### 2.5.1. Diodes as rectifiers

Diodes are used for many different purposes in electronic circuits. One major application is *rectification*. Electrical energy is distributed in form of alternating current. Although this form of energy is suitable for most electrical appliances, like machinery, heating and lighting, direct current supplies are necessary in electronic instrumentation. Almost all electronic instruments have a *power supply* sub-system, where a.c. energy supply is converted into d.c. voltage supplies in order to provide the necessary energy for the electronic circuits. We first rectify the a.c. voltage to this end. Consider the circuit depicted in Figure 2.17 below.

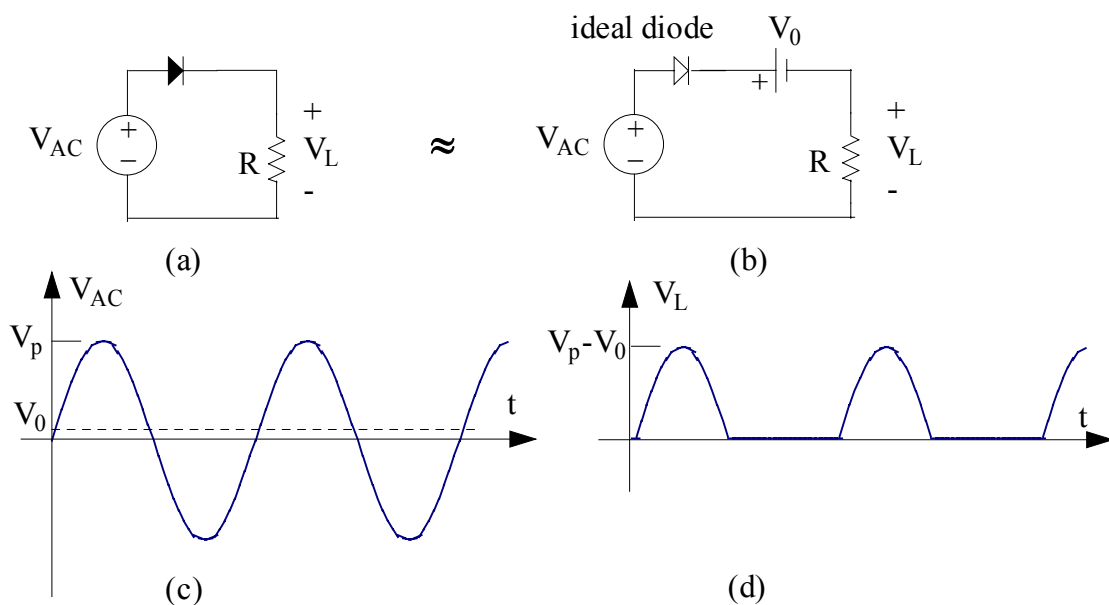


Figure 2.17 (a) Diode rectifier, (b) equivalent circuit, (c) input a.c. voltage and (d) voltage waveform on load resistance.

There is a current flowing through the circuit in Figure 2.17(b) (or (a)) during the positive half cycles of the a.c. voltage  $v_C$  at the input, while it becomes zero during negative half cycles. Here we assume that  $V_p$  is less than the breakdown voltage of the diode. The current starts flowing as soon as  $v_{AC}$  exceeds  $V_0$  and stops when  $v_{AC}$  falls below  $V_0$ . The voltage that appears across the load,  $v_L$ , is therefore sine wave tips as depicted in Figure 2.17(d). This voltage waveform is neither an a.c. voltage nor a d.c. voltage, but it is always positive.

When this circuit is modified by adding a capacitor in parallel to  $R$ , we obtain the circuit in Figure 2.18(a), and its equivalent Figure 2.18(b). The capacitor functions like a filter together with the resistance, to smooth out  $v_L$ .

When  $v_{AC}$  in Figure 2.18(c) first exceeds  $V_0$  (at  $t = t_1$ ), diode starts conducting and the current through the diode charges up the capacitor. The resistance along the path is

the diode *on* resistance. In the equivalent circuit we have chosen for this application (Figure 2.18(b)) this resistance is zero so that the time constant for charging up period is also zero. This means that the capacitor voltage, hence  $v_L$ , will follow  $v_{AC}$  instantly. Charge up continues until  $v_L$  reaches the peak,  $V_p - V_o$ , at  $t = t_2$ . After peak voltage is reached, the voltage at the anode of the diode,  $v_{AC}$ , falls below  $V_p$  and hence the voltage across the diode,  $V_D = v_{AC} - v_L$ , becomes negative. The current through the diode ceases flowing. We call this situation as the *diode is reverse biased*. Now we have a situation where the a.c. voltage source is isolated from the parallel RC circuit, and the capacitor is charged up to  $V_p - V_o$ . Capacitor starts discharging on R with a time constant of RC. If RC is small, capacitor discharges quickly, if it is large, discharge is slow. The case depicted in Figure 2.18 (c) is when RC is comparable to the period of the sine wave.

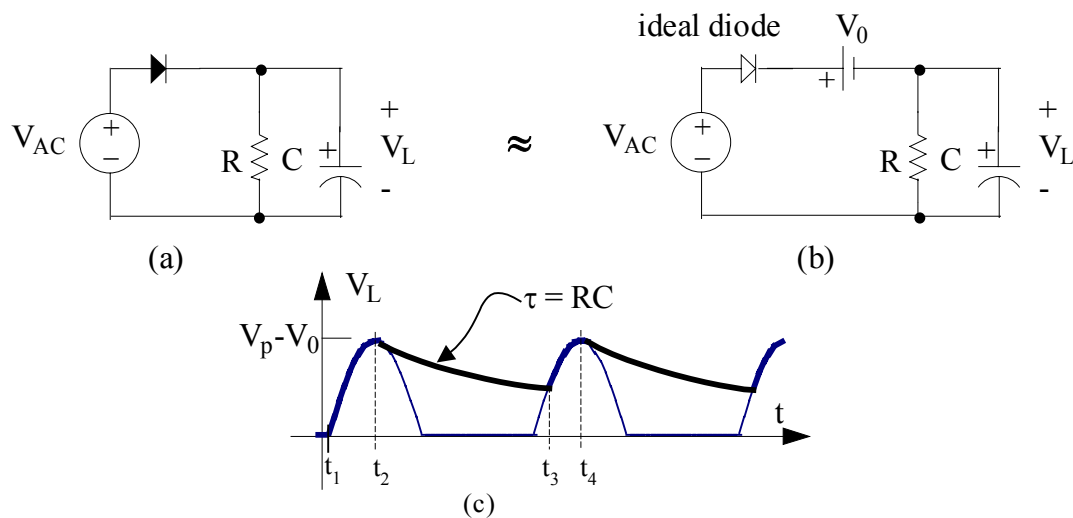


Figure 2.18 (a) Diode rectifier with RC filter, (b) equivalent circuit and (c) voltage waveform on load resistance.

As  $v_{AC}$  increases for the next half cycle of positive sine wave tip, it exceeds the voltage level to which the capacitor discharged until then, at  $t = t_3$ , and *diode is switched on* again. It starts conducting and the capacitor is charged up to  $V_p - V_o$  all over again ( $t = t_4$ ).

The waveform obtained in Figure 2.18(c) is highly irregular, but it is obviously a better approximation to a d.c. voltage compared to the one in Figure 2.17(d).

We preferred electrolytic capacitor in this circuit, which possesses polarity. In this circuit, the voltages that may appear across the capacitor is always positive because of rectification, and hence there is no risk in using such a capacitor type. On the other hand, large capacitance values can be obtained in small sizes in these type of capacitors. Large capacitance values allow us to have more charge storage for the same voltage level, large time constant even with smaller parallel resistances, and thus smoother output waveforms.

A better way of rectifying a.c. voltage is to use four diodes instead of one, as shown in Figure 2.19. In this case we utilize the negative half cycles as well as positive ones.

The four-diode configuration is called a *bridge*, and the circuit is called *bridge rectifier*.

When  $v_{AC}$  is in its positive phase, D2 and D4 conducts, and current flows through D2, capacitor and D4, until capacitor is charged up to the peak value,  $V_p - 2V_0$ . The peak voltage for  $v_L$  is less than the one in single diode case, because the charging voltage has to overcome the threshold voltage of two diodes instead of one. During the negative half cycles, D1 and D3 conducts and the capacitor is thus charged up in negative phase as well. Since the capacitor is charged twice in one cycle of  $v_{AC}$  now, it is not allowed to discharge much. The waveform in Figure 2.19(c) is significantly improved towards a d.c voltage, compared to single diode case.

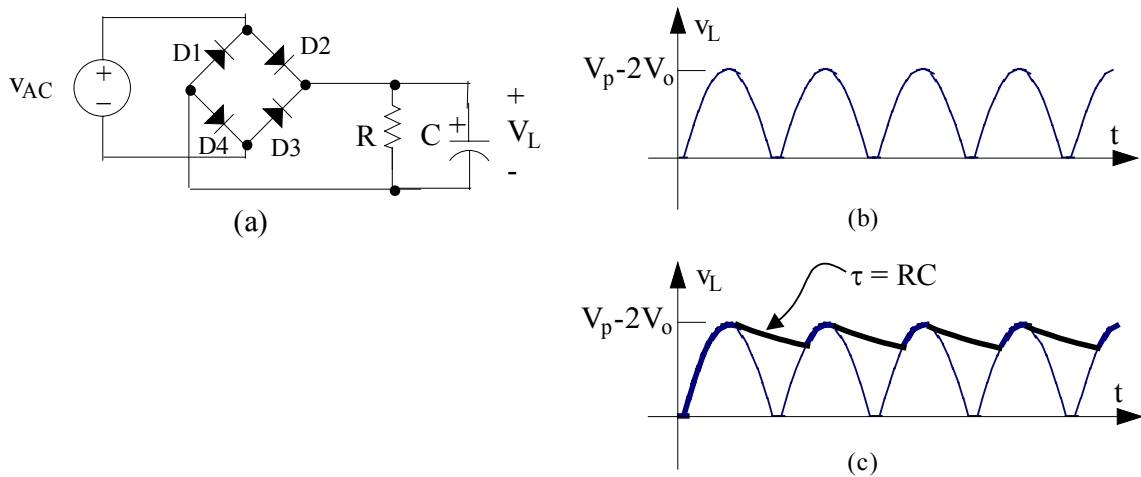


Figure 2.19 (a) Bridge rectifier, (b) rectified voltage without capacitor, and (c) filtered output voltage

### 2.5.2. Zener diodes as voltage sources

Zener diodes are used as d.c. voltage reference in electronic circuits. Zener diodes are used in the vicinity of breakdown voltage as shown in Figure 2.20, as opposed to rectification diodes. The symbol for zener diode is shown in the figure.

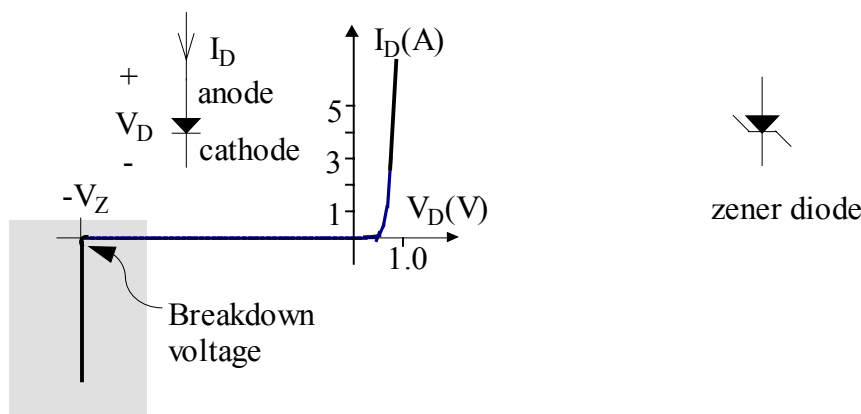


Figure 2.20 Zener diode and its characteristics



When a zener diode is used in a circuit given in Figure 2.21(a), a reverse diode current

$$-I_D = (V_S - V_Z)/R$$

flow through the diode as long as  $V_S > V_Z$ .  $V_Z$  appears across the diode. When  $V_S$  is less than  $V_Z$ , the diode is no longer in the breakdown region and it behaves like an open circuit.

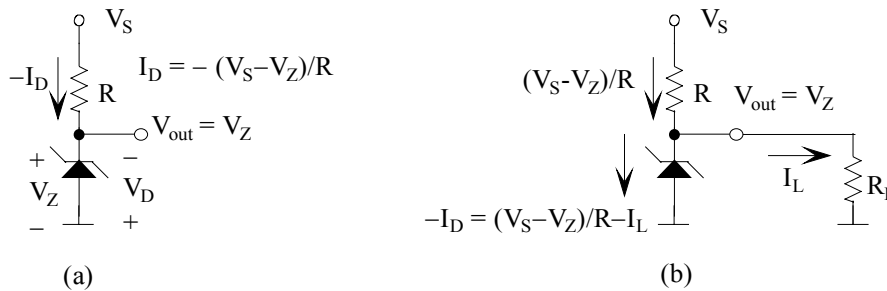


Figure 2.21 Zener diode in a voltage reference circuit

Assume that a load resistance  $R_L$  is connected across the diode, as shown in Figure 2.21(b). The current through  $R_L$  is

$$I_L = V_Z/R_L.$$

As long as the diode current  $-I_D$  is larger than zero, i.e.

$$I_L < (V_S - V_Z)/R,$$

diode remains in breakdown region and provides the fixed voltage  $V_Z$  across its terminals.

## 2.6. Inductors

When current flow through a piece of wire, a magnetic flux is generated around the wire. Reciprocally, if a conductor is placed in a time varying magnetic field, a current will be generated on it. From the electrical circuits point of view, this phenomena introduces the circuit element, *inductor*. Inductors are flux or magnetic energy storage elements. Inductance is measured in *Henries (H)*, and since most inductors used in electrical circuits have the physical form of a wound coil, a coil symbol is used in circuit diagrams to represent inductance (Figure 2.22(a)). The terminal relations of an inductance is given as

$$v(t) = L \frac{di(t)}{dt}$$

where  $v(t)$  and  $i(t)$  are current through and voltage across the inductance, and  $L$  is the value of inductance in henries. We note that voltage is proportional to the time derivative of current in an inductor. If  $i(t)$  is a d.c. current, its derivative is zero, and hence the voltage induced across inductor is zero. Putting this in another way, if we

apply a d.c. voltage across an inductor, the current that will flow through the inductor will be indefinitely large ( $= \infty$ ), and the applied voltage is shorted.

TRC-10 employs few different types of inductors. Some inductors are made by simply shaping a piece of wire in the form of a helix. These are called *air core inductors*. When larger inductance values are required in reasonable physical sizes, we wind the wire around a material which has higher *permeability* compared to air. This material is referred to as *core*, and such inductors are symbolized by a bar next to the inductance symbol, as shown in Figure 2.22(a).

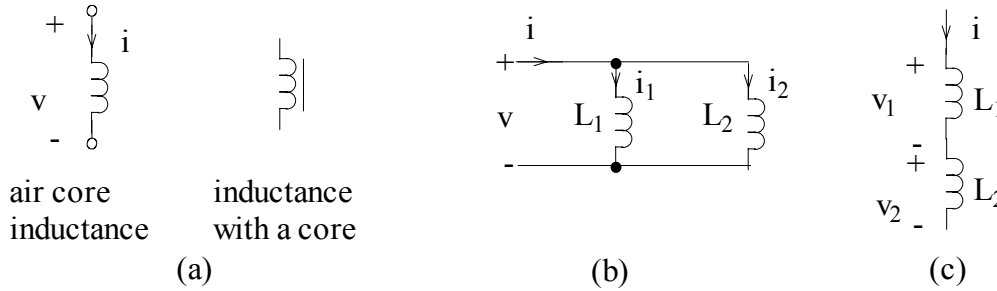


Figure 2.22 Inductor circuits. (a) inductance, (b) parallel inductances, and (c) series inductances.

The parallel and series combination of inductors are similar to resistance combinations, as can be understood from the terminal relation above. For parallel connected inductors as in Figure 2.22(a), the total inductance is

$$L = \left( \frac{1}{L_1} + \frac{1}{L_2} \right)^{-1} ,$$

whereas for series connected inductors (Figure 2.22(c)),

$$L = L_1 + L_2 .$$

The net magnetic energy stored in an inductor in the interval  $(0, t_1)$  is

$$E = \int_0^{t_1} P(t)dt = \int_0^{t_1} v(t)i(t)dt = \int_0^{t_1} \left( L \frac{di(t)}{dt} \right) i(t)dt = L \int_0^{t_1} d \left( \frac{i^2(t)}{2} \right) = L \frac{i^2(t_1)}{2} - L \frac{i^2(0)}{2}$$

If  $i(t)$  above is a d.c. current applied at  $t = 0$ , i.e.  $i(t) = I_{dc}$  for  $t \geq 0$  and  $i(t) = 0$  for  $t < 0$ , then the net energy stored in the inductor is

$$E = L \frac{I_{dc}^2}{2} .$$

When we connect a parallel resistance to an inductor with such stored energy,  $E = LI_i^2/2$  (we changed  $I_{dc}$  to  $I_i$  to avoid confusion), we have the circuit shown in Figure 2.23(a). Initially the inductor current  $i_L$  is equal to  $i(0) = I_i$ . Since  $R$  and  $L$  are connected in parallel, they have the same terminal voltage:

$$v = Ri_R = L \frac{di_L}{dt}$$

and

$$i(t) = i_L = -i_R.$$



Figure 2.23 LR circuit

Therefore

$$\frac{di(t)}{dt} + \frac{R}{L}i(t) = 0.$$

This equation is again a differential equation similar to capacitor discharge equation. Its solution is also similar:

$$i(t) = I_i \exp(-t/\tau) \quad \text{for } t \geq 0.$$

where the time constant is  $\tau = L/R$  in this case. Now let us assume that we connect a step voltage source  $v_S(t)$ , like the one in Figure 2.13(a), in series with  $R$ , instead of  $R$  alone. This circuit is given in Figure 2.23(b). Again the initial loop current is equal to the current stored in the inductor:

$$i(0) = I_i$$

and the KVL equation is

$$-v_S(t) + Ri(t) + L \frac{di(t)}{dt} = 0.$$

$v_S(t)$  is zero for  $t < 0$ , and  $v_S(t) = V_S$  for  $t \geq 0$ . If we differentiate the above equation with respect to  $t$ , we obtain

$$\frac{dv_S(t)}{dt} - R \frac{di(t)}{dt} - L \frac{d^2i(t)}{dt^2} = 0.$$

$dv_S(t)/dt$  term is zero for  $t > 0$ , since  $v_S(t)$  is constant. Substituting  $v_L/L$  for  $di(t)/dt$  in the above equation, we obtain

$$\frac{dv_L(t)}{dt} + \frac{R}{L} v_L(t) = 0.$$

We can write down the solution of this equation after determining the initial value of  $v_L$ . Just after  $t=0$ ,

$$v_L(0) = V_S - R i(0) = V_S - R I_i.$$

Hence

$$v_L(t) = (V_S - R I_i) \exp(-t/\tau) \quad \text{for } t > 0$$

where  $\tau = L/R$ , and

$$i(t) = \frac{1}{L} \int v_L(\xi) d\xi = (I_i - V_S/R) \exp(-t/\tau) + K_\infty$$

The integration constant  $K_\infty$  can be determined by using the initial value of  $i(t)$ :

$$i(0) = I_i = I_i - V_S/R + K_\infty \Rightarrow K_\infty = V_S/R.$$

$i(t)$  is drawn in Figure 2.24.  $V_S/R$  term is the value that the inductor current will reach at  $t = \infty$ . Indeed as time passes, the derivative term in KVL equation must diminish, since the variation with respect to time slows down. Then for large  $t$ ,  $i(t) \approx V_S/R$ . This value that the solution of the differential equation reaches at  $t = \infty$  is called *steady-state value*, in electrical engineering.

Once we determined the initial value and the steady-state value of the solution, we can write down the solution of a first order differential equation directly, as we have done above.

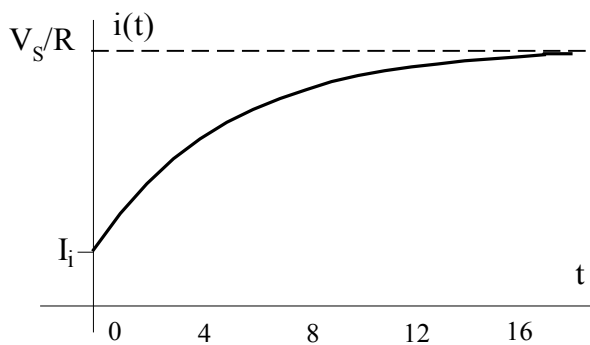


Figure 2.24 Inductor current in series RL circuit with step voltage input

## 2.7. Transformers

Transformers are two or more *coupled* inductors. They are coupled to each other by means of the same magnetic flux. In other words, they share the same flux. The circuit

symbol of a transformer with two *windings* (i.e. two inductors) is given in Figure 2.25.

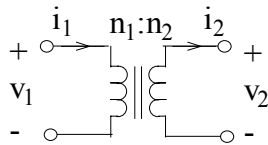


Figure 2.25 Transformer

The windings are referred to as *primary winding* and *secondary winding* respectively, in transformers. Transformers transform the voltage and current amplitude that appears across the primary winding to another pair of amplitudes at the secondary, and vice versa. The amount of transformation is determined by the *turns ratio*  $n_1:n_2$ . The relations in an *ideal transformer* are as follows:

$$\frac{v_2}{v_1} = \frac{n_2}{n_1}$$

where as

$$\frac{i_2}{i_1} = \frac{n_1}{n_2}$$

There are four transformers in TRC-10. The first one is the *mains transformer* used in power supply sub-system, and other three are RF transformers. We shall delay a detailed discussion of transformers until we consider the RF circuits. The mains transformer, as used in TRC-10, can well be modeled as an ideal transformer.

## 2.8. Circuit Protection Devices

There is always a possibility that voltages much larger than envisaged levels can appear in electronic circuits. For example, when a lightning strikes to a power line, it is likely that very high voltage spikes can appear on the voltage supply. Similarly very high currents can be drawn from supplies because of mishandling, such as short circuits. It is likely that many of you will experience short circuits in the Lab Exercises of TRC-10. We use *varistors* (*Variable Resistors*) as over voltage protection devices, and *positive temperature coefficient thermistors* (*PTC*) as over current protection devices.

### 2.8.1. Varistors

Varistors are nonlinear resistors made of ceramic-like materials like sintered zinc oxide or silicon carbide. The I-V characteristics of a varistor is depicted in Figure 2.26, together with its symbol.

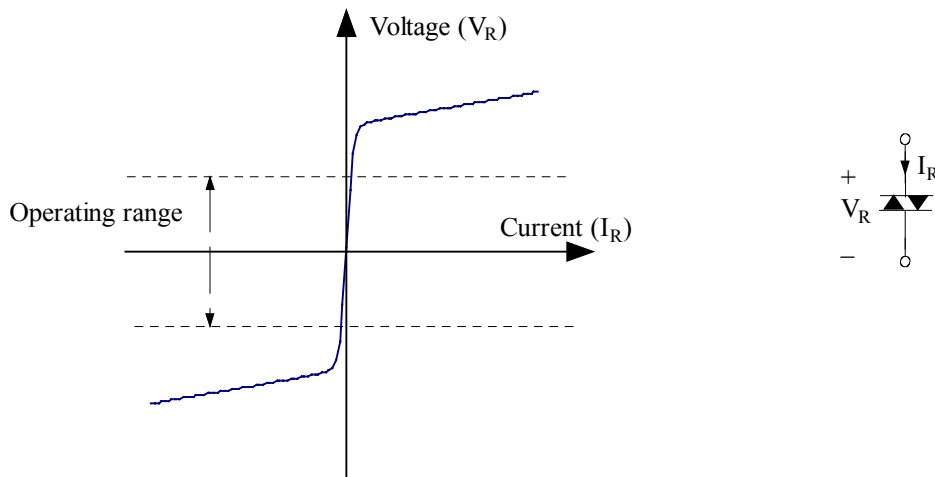


Figure 2.26 Varistor characteristics and symbol

When the voltage across the varistor is within the operating range, varistor exhibits a very large resistance. When the voltage increases, the resistance falls rapidly, thus taking most of the excess current due to over-voltage.

Varistors are connected in parallel to the circuits to be protected.

### 2.8.2. PTC Thermistors

A PTC thermistor is a thermally sensitive ceramic resistor. Its resistance increases abruptly with increasing temperature beyond a specified limit (reference temperature). Using PTC as resettable fuse relies on the following consideration:

The current through the PTC under normal operating conditions is sufficiently low. At this current level the power dissipated by the PTC *on* resistance is again low enough, such that the PTC temperature does not exceed the reference temperature. When a short circuit occurs, the current through the PTC increases. The power dissipation increases the temperature over the reference temperature and PTC *trips* to high impedance state.

PTC's are usually specified by two current parameters. *Rated current* ( $I_N$ ) is the current level, below which the PTC reliably remains in low resistance mode. *Switching current* ( $I_S$ ) is the level beyond which the PTC reliably trips to high resistance mode. Another parameter of significance is  $R_N$ , the resistance of PTC at low resistance mode.

PTC thermistors are connected in series to the circuit to be protected.

### 2.8.3. Circuit protection

An over-voltage protection circuit typically has the form shown in Figure 2.27.

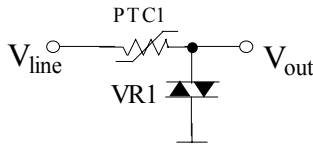


Figure 2.27 Over-voltage protection circuit

This circuit operates as follows:

VR1 and PTC1 are chosen such that, when there is not any over-voltage or surge current, the voltage across VR1 is in the normal range and the current through PTC1 is less than  $I_N$ . PTC exhibits a low resistance and VR exhibits a very high resistance.

When an over-voltage occurs on the line, the voltage across VR1 increases beyond its operating range. The current through VR1 increases rapidly due to the nonlinear nature of the varistor resistance. This current passes through PTC1 and warms the PTC up. PTC1 switches to high impedance mode isolating the line from the output, when this current exceeds  $I_S$ .

## 2.9. Bibliography

Chapter 24 in *ARRL Handbook* has comprehensive information and data on components.

There are excellent circuit theory books. *Electrical Engineering Uncovered* by D. White and R. Doering provides a very good introduction. N. Balabanian's *Electric Circuits* is one of excellent circuit theory books.

## 2.10. Laboratory Exercises

### Power supply sub-system

1. The construction of TRC-10 starts with the power supply. Examine the circuit diagram of the power supply given in the appendix. Familiarize yourself with the components in this circuit.
2. Read all exercises in this chapter carefully.
3. Place the mounting tray on your desk.
4. Mount the mains jack J1.
5. Mount the fuse holder.
6. Mount the mains switch S1.
7. Mains switch is a *double pole single throw* (DPST) switch. This means that it has a pair of single switches, connects two pair of lines when closed (double pole), and only disconnects when open (single throw). We shall connect the mains live and neutral lines to the transformer primary winding through the mains switch. The mains switch is a toggle switch marked as "I/O". When set to "I" we want the mains connected, when set to "O", disconnected. A neon bulb is fitted internally between the two contacts on one side of the switches, as shown in the figure below. We want that side connected to our circuit, so that the neon will be energized only when TRC-10 is switched on. The other side is toggle side and must be connected to mains. Neon bulbs are gas discharge bulbs, and emit light

only when the voltage across them is larger than approximately 90 volts. They draw very little current when emitting light. They are commonly used power indicators. Check between which contacts the neon is fitted.

To make the *live* connection, cut two pieces of 10 cm long *brown* colored wire, strip both ends for about 5mm each, and tin them. Solder one end of the wires to one of the two circuit taps on the switch. Solder one end of the other piece of wire to the corresponding toggle tap on the switch. Cut two 2 cm long pieces of 6 mm diameter heat-shrink sleeve and work each wire through one of them. Push the sleeves as far as you can such that the sleeves cover the taps entirely.

Heat-shrink sleeves shrink to a diameter, which is 30% to 50% of its original diameter when exposed to heat of about 90°C for a few seconds. It is an isolating material so that there will not be any exposed hot conducting surfaces. Take the hot air gun, adjust the temperature to 90°C and shrink the sleeve. This can be done by a lighter or a hair dryer instead of hot air gun. Hot air gun is a professional tool and must be handled with care. It can blow out very hot air reaching to 400 °C and can cause severe burns. Ask the lab technician to check your work.

Cut another piece of heat shrink sleeve and work the wire connected to the side tap through it. Solder the wire end to the hot tap on the mains jack. Push the sleeve so that it completely covers the tap. Using the hot air gun shrink the sleeve.

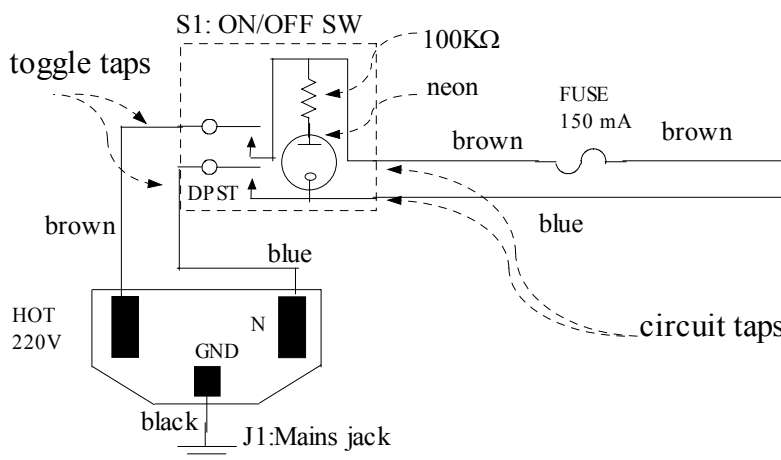


Figure 2.28

To make the *neutral* line connections, cut two 10 cm pieces of *blue* wire, and tin the ends. Make the connections to the other pair of taps on the switch, similar to the live one. Make sure that sleeves cover all conducting surfaces. **Safety first!** If you work carefully and tidily, there will never be any hazardous events. There must not be any hazardous events. Indeed a careful and clever engineer will **never** get a shock or cause any hazard to others.

Cut two more pieces of heat shrink sleeve and a piece of 10 cm long brown wire. Tin the wire ends. Solder this wire to one tap of the fuse holder. Work the brown



wire coming from the switch into a piece of sleeve and then solder it to the other tap of the fuse holder. Push the sleeve to cover the tap completely. Fit the remaining sleeve onto the other tap. Using hot air gun, shrink the sleeves.

Notice that we used a color code for mains connection: *brown* for live line and *blue* for neutral. We shall use *black* for earth connections.

8. Mount the mains transformer T1 on the TRC-10 tray, by means of four screws. Do not forget to use anti-slip washers. Otherwise the nuts and bolts may get loose in time.
9. Mains transformer is a 10 W 220V to 2X18 V transformer. That is, it converts 220V line to 18V a.c. There are two secondary windings, and hence we have two 18V outputs with a common terminal.
10. Cut two 2 cm pieces of heat shrink sleeve and work the two wires, brown coming from the fuse and blue coming from the switch, into each one of the sleeves. Solder the wires to the two primary windings of the transformer. Push the sleeves so that they cover the transformer terminals completely. Shrink the sleeves using hot air gun.

Now we have completed the mains connections. *Root mean square* (rms) definitions of voltage and current prevail particularly for a.c. circuits. Line voltages have sinusoidal waveform. The frequency of the line,  $f_L$ , differs from country to country, but it is either 50Hz or 60Hz. For some specific environments there are other line frequency standards (in aircraft for example, a.c. power line is 400Hz). If we express the line voltage as

$$v(t) = V \sin(\omega t + \theta)$$

where  $\omega = 2\pi f_L$  and  $V$  is the amplitude of the line voltage, then *rms* value of  $V$  is defined as

$$V_{\text{rms}} = \sqrt{\frac{1}{T} \int_0^T v^2(t) dt} = \frac{V}{\sqrt{2}}$$

where  $T$  is the period of the sine wave. Similarly rms value of a sine wave current is

$$I_{\text{rms}} = \frac{I}{\sqrt{2}}$$

with  $I$  the current amplitude. Note that with this definition, on a line of  $V_{\text{rms}}$  potential and carrying  $I_{\text{rms}}$  current, the total power is  $P = V_{\text{rms}} I_{\text{rms}}$ .

When a line voltage is specified, e.g. 220 V, it means that the line potential is 220 volts rms. Hence, the voltage on this line is of sinusoidal form with an amplitude of approximately (nominally) 310 V. Similarly, for a transformer specified as 220V to 18 V, it means that the transformer transforms line voltage of 310 V amplitude to a sinusoidal voltage of 25.5 volts amplitude, at the line frequency.

As a matter of fact, if we measure an a.c. voltage with a multimeter, the reading will be the rms value of the voltage.

Power line voltages also differ from country to country, but there are only few standards. Line voltages are 110 V<sub>rms</sub>, 120 V<sub>rms</sub>, 220 V<sub>rms</sub>, or 240 V<sub>rms</sub>. The only component sensitive to the line voltage specification in TRC-10 is the line transformer. The power line in this environment is 50Hz/220 V<sub>rms</sub> line and hence transformer is chosen accordingly.

Electric energy is generated in electric power plants. The generated power must be transported long distances before it can be used, since power plants can be quite far away to areas where large energy demand is. Voltage level is either 6.3 KV rms or 13.8 V rms at the terminals of the generator in the plant. In order to carry the power over long distances with minimum energy loss, the voltage of the line is stepped up to a very high level, usually to 154 or 380 KV<sub>rms</sub>. The transport is always done by means of *high voltage (HV) overhead lines (OHL)*. This voltage level is stepped down to a lower level of 34.5 KV *medium voltage (MV)*, in the vicinity of the area (may be a town, etc.) where the energy is to be consumed. Energy is distributed at this potential level (may be up to few tens of km). It is further stepped down to household voltage level (e.g. 220 V- the voltage referred to as “220 V rms” actually means a voltage level between 220 to 230 V rms) in the close vicinity of the consumer. All this step-up and step-down is done by using power transformers.

We are accustomed to see the electric energy coming out of household system as a supply of single phase voltage on a pair of lines, *live* and *neutral*. When energy is generated at the generator, it always comes out in three phases. If the phase voltage that we observe between the live and neutral is

$$v_1(t) = V_p \sin(\omega t),$$

then, it is always accompanied by two other related components,

$$v_2(t) = V_p \sin(\omega t + 120^\circ) \text{ and}$$

$$v_3(t) = V_p \sin(\omega t - 120^\circ).$$

This is necessitated by the economics of the technology employed in electromechanical power conversion. These three phases of line supply is distributed to the consumers such that all three phases are evenly loaded, as much as possible.

As far as phase voltage is concerned, 220 V rms refers to the voltage difference between any one of the phase voltages and neutral. On the other hand, the potential difference between any two phases, which is called *line voltage*, e.g. between  $v_1(t)$  and  $v_2(t)$ , is

$$\begin{aligned} \Delta v(t) &= v_1(t) - v_2(t) \\ &= \sqrt{3} V_p \sin(\omega t - 30^\circ) \end{aligned}$$

The potential difference between the phases is therefore 1.73 times larger than any one of phase voltage with respect to neutral. The line voltage level is 380-400 Vrms for a phase voltage of 220 V. The last step-down from MV to low voltage (LV) is depicted in Figure 2.29 below.

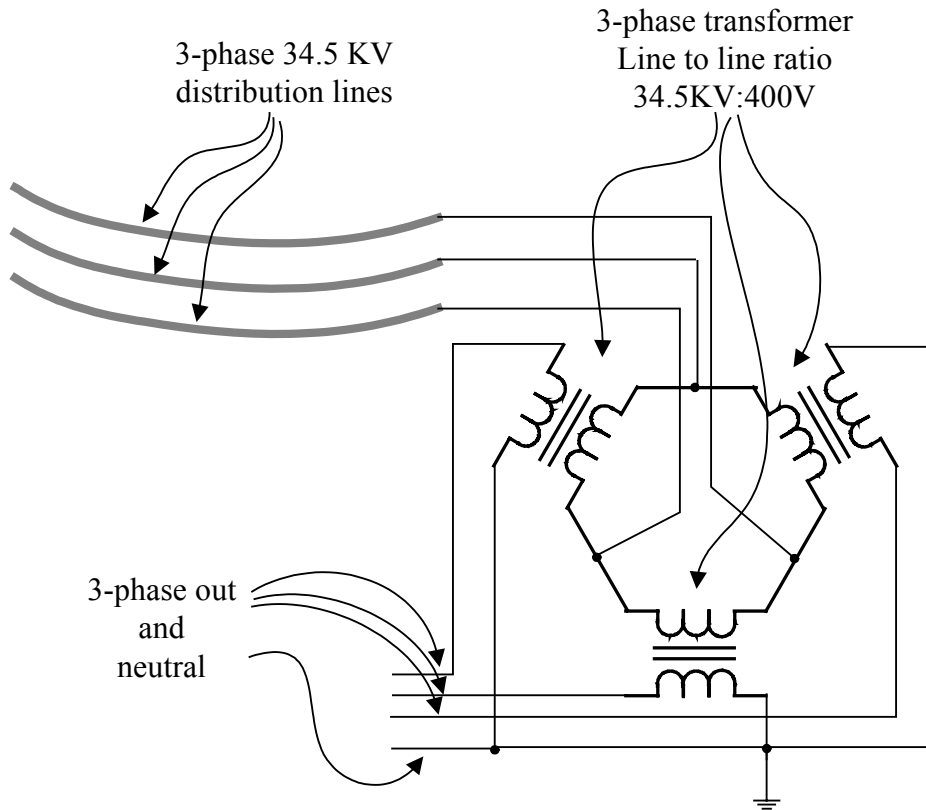


Figure 2.29 MV to LV transformer

Note that there is no neutral for 3-phase MV distribution lines (both HV and MV energy are carried as three phases only without neutral reference during the transportation). Once it is stepped down, one terminal of each of the secondary windings are grounded at the transformer site, and that node is distributed as neutral. Grounding is done by connecting that terminal to a large conducting plate or long conducting rods buried in earth. A separate line connected to earth is also distributed, since most household and professional equipment require a separate earth connection, not only for operational reasons but also for safety. Neutral is the return path of the current we draw from line. We do not expect any significant current on the earth connection, other than leakage.

When the energy is carried on three phases only, the nominal rms line voltages refer to the potential between the phases. 34.5 KV rms, for example, is the line voltage in MV lines.

A typical MV to LV transformer configuration is given in Figure 2.29. The three phase line voltage of 34.5 KV MV is connected to the primary windings of a three phase transformer, which is connected in a “Δ” configuration. The secondary terminals are LV terminals, and three windings are now configured in a “Y” form.

In other words, one terminal of each of the secondary windings is connected to earth, while there is no earth connection on the primary. The voltage transformation ratio in these transformers are always stated as the *ratio of line voltages* (i.e. the potential difference between the phases) of primary and secondary windings, although the physical turns ratio of primary and secondary windings correspond to 34.5 KV to 230 V.

Three 220V live lines, neutral and ground are distributed in the buildings through few distribution panels. Precautions against excessive current are taken at each panel. This is for reducing the fire risk in the building and is not useful to avoid electric shock. One can get electric shock either by touching both live and neutral at the same time or by touching live while having contact to ground. The first one is highly unlikely unless one is very careless.

Building floors have a connection to ground reference, although there may be some resistance in between. Therefore if one touches the line while standing on the floor, e.g. with shoes with natural soles (not an isolating sole like rubber), he will get a shock. It is likely that there is an extra precaution at the last panel, where a *residual current device* (RCD) is fitted. This device monitors the leakage current to the ground, and when it exceeds 30 mA, it breaks the circuit. This decreases the severity of the shock.

Ask the lab technician to show and explain the distribution panel in your laboratory.

11. Visit the medium voltage transformer site, which provides energy to your laboratory. Find out the diameter of the cable that delivers the MV energy.
12. Visit a local power plant. Find out what kind of primary energy (i.e. gas, coal, petroleum, wind, hydraulic, etc.) it uses.
13. The voltage reference for the secondary windings is the center tap. We must connect this tap to earth tap in J1. Cut two pieces of 10 cm long *black* colored wire, strip both ends for about 5mm each, and tin them. Join and solder one end of each wire to the center tap of the transformer. Cut two 2 cm long pieces of sleeve. Work two wires into one of the sleeves. Push the sleeve so that the transformer tap is completely covered. Shrink the sleeve. Work one of the wires into remaining sleeve. Solder that wire to earth tap on J1. Push the sleeve and shrink it. Crimp a lead to the other end of the cable and fit it into the center pin of *printed circuit board* (PCB) connector plug, J11. Cut a pair of 10 cm long red wires, strip and tin the ends. Solder one end of each to secondary winding terminals. Fit other ends to side pins of J11, after crimping the leads.
14. Check all connections with a multimeter. Make sure that they correspond to the circuit diagram. Connect the power cable and switch the power ON.

Using a *mains tester* check if there is mains leakage on the tray.

Set your multimeter for AC voltage measurement. Connect the leads across the center tap and one of the secondary winding taps. Measure and record the voltage.

This is the rms value of the secondary winding. Calculate the peak value. Measure the voltage across the other winding.

Switch the power OFF.

15. Rest of the power supply circuit is on the PCB. Mount and solder two 30 V varistors. Varistors have high impedance at low voltage levels and low impedance at high voltage levels. When the voltage across it exceeds protection level (30 V in this case), varistor effectively limits the voltage by drawing excessive current.
16. Study the data sheet of PTC thermistors in the Appendix. Determine the rated and switching current of PTC1. What is the  $I_N$ ,  $I_S$  and the on resistance  $R_N$  of this thermistor? Record these figures. Mount and solder PTC1 and PTC2. Check the connections. Trim the leads of the PTC's and varistors at the other side of the PCB using a side cutter.
17. The pin configuration of a diode bridge is marked on the package either by the schematic of bridge circuit or by a "+" sign at the pin as marked in the circuit diagram. Mount the bridge correctly and solder it. Trim the leads at the other side of the PCB using a side cutter.

Connect capacitors C1 and C2. These capacitors are *electrolytic* and have polarity. They contain a *liquid* electrolyte in their case. Either negative pin or positive pin is marked on the capacitor case. Take care to mount them correctly. Solder the capacitors. Check all the connections using a multimeter. Switch the power ON. Measure and record the voltage across C1 and C2 ground pin being common in both cases.

Switch the power OFF.

18. We use three *voltage regulators* in TRC-10, one regulator for each voltage supply except  $-8V$  supply. Voltage regulators are integrated circuits comprising many transistors, diodes etc. All regulators we use in this circuit have three pins: *input*, *output* and *ground*. Voltage regulators convert a rectified and filtered voltage level (like the one in Figure 2.18(c) or Figure 2.19(c)) at their input terminal and convert it into a clean d.c. voltage level *without any ripple*. The two positive supplies 15V and 8V are obtained at the output of two regulators LM7815 and LM7808, respectively.  $-15V$  is regulated by LM7915. Voltage regulator requires that the minimum voltage level that appears across its input be about 2V higher than the nominal output voltage, in order to perform regulation. For example, LM7815 requires that the minimum value of the unregulated voltage at its input is 17.5V, in order to provide a regulated nominal 15 V output. Output voltage nominal value has a tolerance. For LM7815, output voltage can be between 14.25 and 15.75 V, regulated. This does not mean that it is allowed to fluctuate between these values, but the level at which the output is fixed can be between these voltages.

The data sheets of LM78XX and LM79XX series regulators are given in Appendix D. Examine the data sheets. Can you find out the information given

above for 7815 in the data sheet? Find out the maximum current that can be drawn from 7815, while regulation is still maintained (peak output current).

Install LM7808 on the PCB and solder it. Make sure that you placed the IC pins correctly on the PCB. Check the connections. Switch the power ON. Measure and record the output voltage, with one decimal unit accuracy. Switch the power OFF.

Install LM7815 on the PCB and solder it. Check the connections. Switch the power ON. Measure and record the output voltage, with one decimal unit accuracy. Switch the power OFF.

Install LM7915 on the PCB and solder it. Check the connections. Switch the power ON. Measure and record the output voltage, with one decimal unit accuracy. Switch the power OFF.

Mount all remaining capacitors, C7, C8 and C9. All of them have polarities. Mount them accordingly. C7, C8 and C9 are *tantalum* capacitors. The electrolyte in tantalum capacitors is in solid form. We include tantalum capacitors to improve the filtering effect at higher frequencies, where the performance of electrolytic capacitors deteriorates.

Check the connections. Switch the power ON. Measure and record all output voltages, with one decimal unit accuracy. Compare these measurements with the previous ones. Switch the power OFF.

19. Mount and solder the protection diodes D12, D14 and D16. These diodes provide a discharge path to capacitors C7, C8 and C9 respectively, when the unregulated voltage input becomes zero. This is a precaution to protect the regulators.

Mount and solder the protection diodes D13, D15 and D17.

Check the connections. Switch the power ON. Measure and record all output voltages. Compare these measurements to see if they are the same with the ones in the previous exercise. Switch the power OFF.

20. Find out and record  $I_N$ ,  $I_S$  and the on resistance  $R_N$  of PTC3. Mount and solder three PTC thermistors in series with +15 V, -15 V and + 8 V regulator output terminals.

Check the connections. Switch the power ON. Measure and record the supply voltage levels after the PTC's.

Connect the multimeter in series with PTC3 as a current meter (not voltmeter). Connect the free end of the current meter to ground. What is the current meter reading? If there were not any PTC on the way, you should read a short circuit current and often the regulator would be burnt out!

Remove the short circuit and connect the multimeter across D13 as a voltmeter. Record the supply voltage.

21. We also need a  $-8\text{ V}$  supply in TRC10. The output current requirement on this supply is low. We use a simple circuit containing a zener diode to obtain this voltage (see problem 9).

Mount and solder resistor R01 and the zener diode D18.

Check the connections. Switch the power ON. Measure and record the output voltage. The output voltage is  $V_Z$  of the diode. Switch the power OFF.

22. The energy provided by the supplies can now be connected to the circuits. We need 20 jumper connections from the power supplies to the circuits of TRC-10. A white straight line between two connection points on the PCB shows each one of these. Locate these jumpers on the PCB. Cut appropriate lengths of wire for each and make the connections by soldering these wires.

### 2.11. Problems

1. Find  $R_{eq}$  in the circuits given below (two significant figures, in  $\Omega$ , K or M as appropriate):

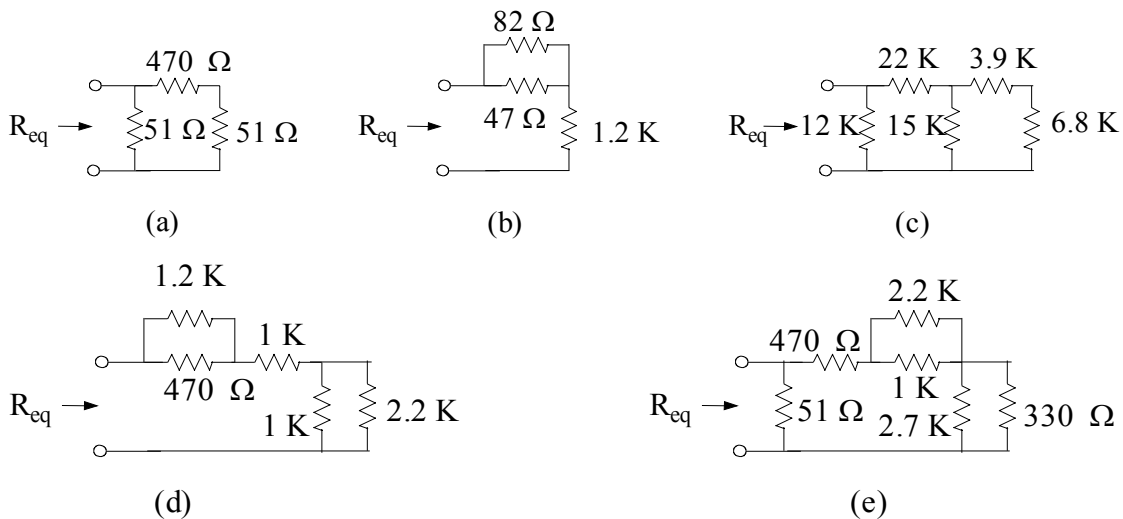
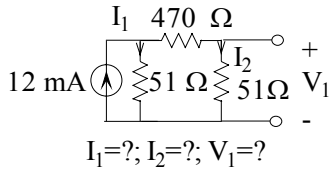


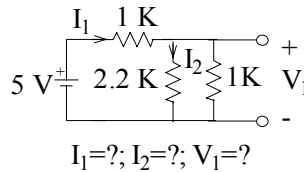
Figure 2.30 Problem 1

2. Write down the *rms* values of following (two significant figures, in scientific notation):
- (a)  $10 \cos(1000t)\text{ V}$       (b)  $1.4 \sin(314t+30^\circ)\text{ A}$       (c)  $28 \cos(\omega t+\theta)\text{ V}$
3. What are the frequencies of the waveforms in question 3 (three significant figures, in scientific notation)?
4. A rechargeable Li-ion (*lithium ion*) battery of a mobile phone has nominal voltage of  $3.7\text{ V}$  and a capacity of  $650\text{ mA-hr}$ . How long a charged battery can supply energy to a  $330\ \Omega$  resistor, at its rated voltage? Assume that the internal resistance is very small compared to  $330\ \Omega$ . What is the total energy (in *joules*) delivered to the resistor?

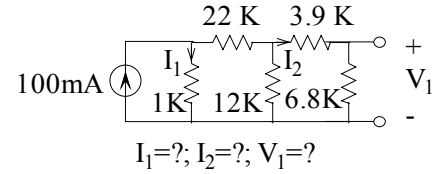
5. Find the marked variables using node analysis in the following circuits (three significant figures, in scientific notation):



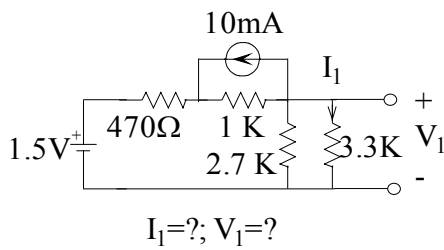
(a)



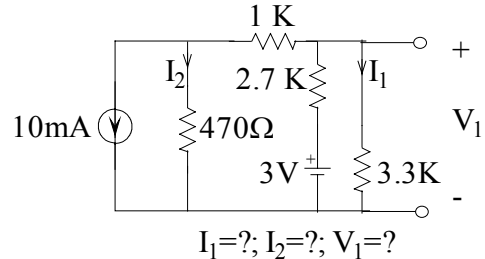
(b)



(c)

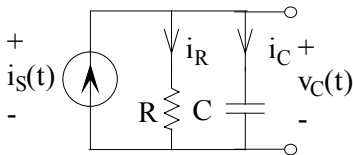


(d)

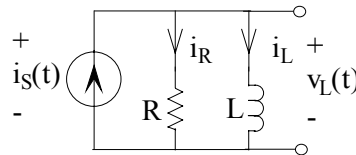


(e)

6. Consider the circuits (a) and (b) given below. Both circuits are driven by a step current source  $i_s(t)$ , shown in Figure 2.31. Find and sketch  $i_C(t)$ ,  $v_C(t)$ ,  $i_L(t)$ , and  $v_L(t)$ . Assume that  $v_C(0) = 0$  and  $i_L(0) = 0$ .



(a)



(b)

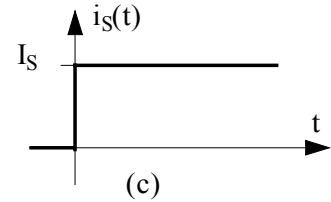


Figure 2.31 Circuit for problem 6.

7. Steel reinforced aluminum wires are used in long distance HVOHL. “954 ACSR” wire (954 tells the type of conductor and ACSR stands for “Aluminum Conductor-Steel Reinforced”) has a cross section of  $485 \text{ mm}^2$  and a resistance (per unit length) of  $0.059 \text{ } \Omega/\text{km}$ . HVOHL are carried by transmission line poles separated by approximately 400 meters, on the average. Considering that the wire is made of aluminum predominantly, calculate the mass of three-phase line between two poles. Calculate the power loss if 32 MW of power is carried over 200 km at a 380 KV<sub>rms</sub> line. What must be the cross section of the wire to have the same loss over the same distance, if the line voltage is 34.5 KV<sub>rms</sub>? Calculate the mass for this case.
8. How much energy is stored in C8? What is the amount of charge stored in it?



9. For the  $-8\text{ V}$  supply circuit formed by R01 and D18, connected to  $-15\text{ V}$  supply, calculate the maximum current that can be drawn by external circuits, while maintaining the diode voltage at  $-8\text{ V}$  approximately. What is the current drawn from  $-15\text{ V}$  supply when no external circuit is connected? Assume that  $V_Z$  of D18 is  $8\text{ V}$  and maximum power it can dissipate is  $0.25\text{ W}$  (D18 burns out if this power limit is exceeded).
10. Find the ratio of R1 to R2 such that the output is  $-8\text{ V}$  in the circuit given in Figure 2.32. Find a pair of standard resistor values (given in section 2.2) for R1 and R2 such that above ratio is satisfied as much as possible. What is the percent error? Assuming that  $-8\text{ V}$  is reasonably approximated, can this circuit be used instead of the one in problem 9? Why?

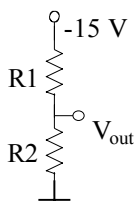
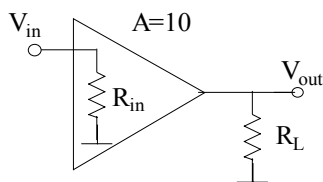


Figure 2.32

11. Consider the amplifier given in the figure, which has input impedance of  $R_{in}$ . The voltage gain of the amplifier is 10. Express the voltage gain in dB. What is the power gain when  $R_{in} = R_L$ ? What is the power gain when  $R_{in} = 10 R_L$ ? In dB?



## 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

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

or

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

and

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

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  $\phi$ .

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

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

*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.

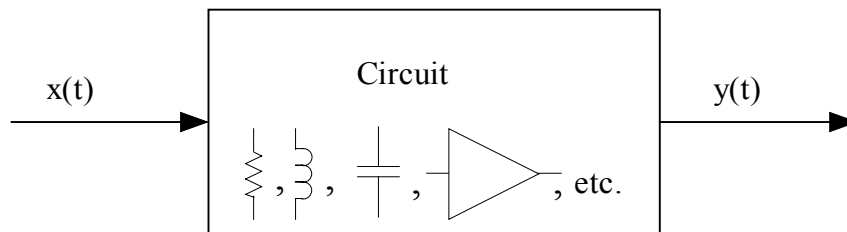


Figure 3.1 Linear circuit and linear circuit elements

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  $\omega LI_p$  (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,

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

If we compare this expression with that of the input current  $i(t) = I_p\cos(\omega t+\theta) = \operatorname{Re}[I_p\exp(j\theta) \exp(j\omega t)]$ , 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:

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

if the time waveform is sinusoidal at an angular frequency of  $\omega$ . Both phasors *are scalar functions* of  $\omega$ . 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  $\omega$ , 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

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

and

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

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 “ $(\omega)$ ” 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,

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

and the reactive power is

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

### 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.

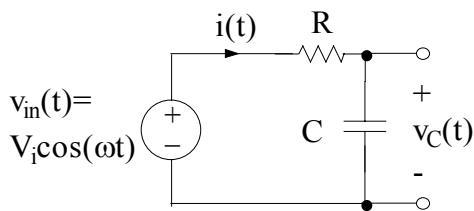


Figure 3.2 Series RC circuit with sinusoidal voltage excitation.

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

$$\begin{aligned} i(t) &= \text{Re} \{ \omega C V_i / [1 + (\omega CR)^2]^{1/2} \exp(j\theta) \exp(j\omega t) \} \\ &= \omega C V_i / [1 + (\omega CR)^2]^{1/2} \cos(\omega t + \theta). \end{aligned}$$

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

The phasors of voltage across the circuit,  $V_{in}(\omega)$ , and the current through it,  $I(\omega)$ , 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 ( $\Omega$ ), like resistance.

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

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

$G$  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  $\omega$ . Variation of the transfer function of this circuit with respect to  $\omega$  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  $\omega$  increases. The output approaches to zero as frequency increases. This circuit is called *RC low pass filter*, because of this property.

$$|H(\omega)| = 1/\sqrt{2} = 0.707 \text{ at } \omega = 1/RC.$$

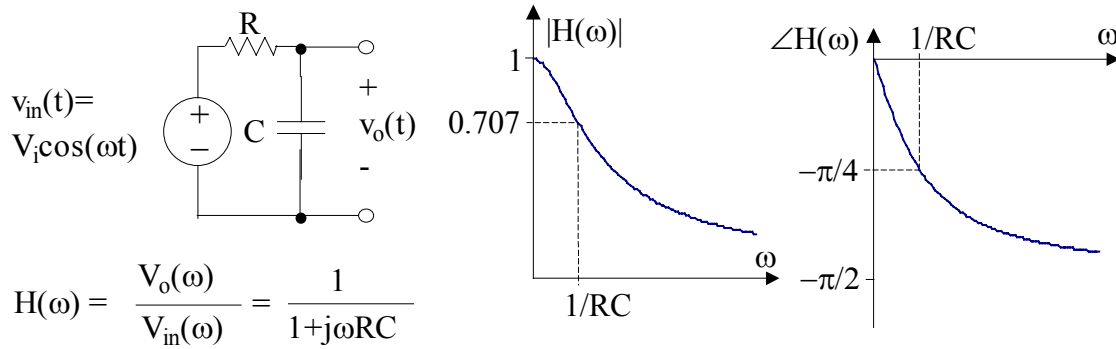
Now, since

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

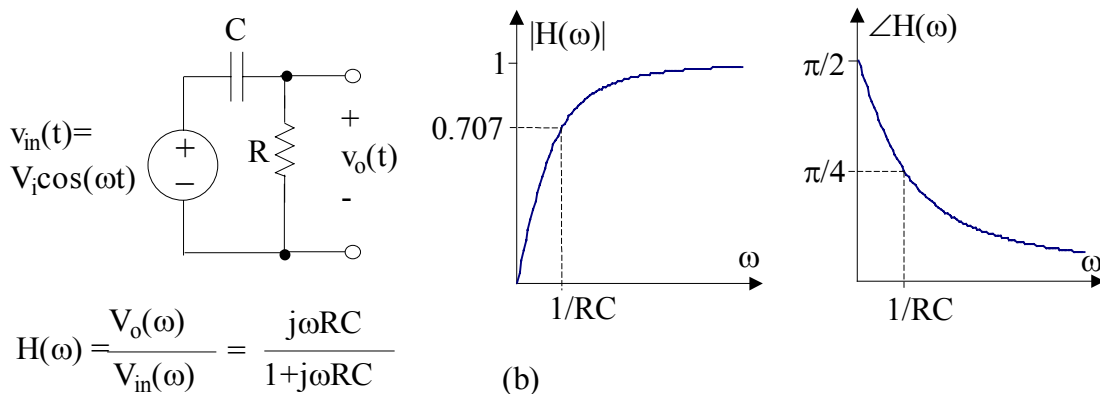
$f_c = 1/2\pi RC$  is called the “3-dB cut-off frequency”.

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





(a)



(b)

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.

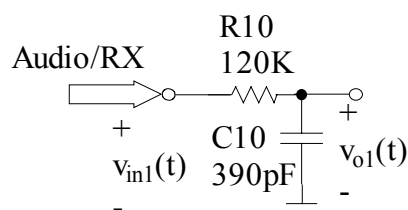


Figure 3.4 Audio/RX LPF

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

$$\begin{aligned}
 H_1(\omega) &= V_{o1}(\omega) / V_{in1}(\omega) \\
 &= 1 / (1 + j\omega RC) \\
 &= 1 / [1 + j\omega / (2\pi \times 3.4 \text{ E}3)] \\
 &= 1 / (1 + jf / 3.4 \text{ E}3)
 \end{aligned}$$

Note that we have written the transfer function in terms of frequency  $f$ , rather than the angular frequency  $\omega$ , 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  $\omega$ . 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.

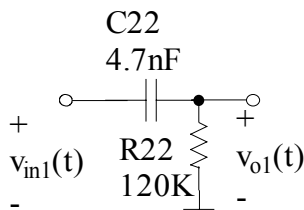


Figure 3.5 Audio/TX HPF

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  $V_{in1}(\omega)/Z$ ,

$$\begin{aligned} V_{o1}(\omega) &= R_{22} \{V_{in1}(\omega)/Z\} \\ &= V_{in1}(\omega) R_{22}/(R_{22} + 1/j\omega C_{22}). \end{aligned}$$

The transfer function is

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

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).

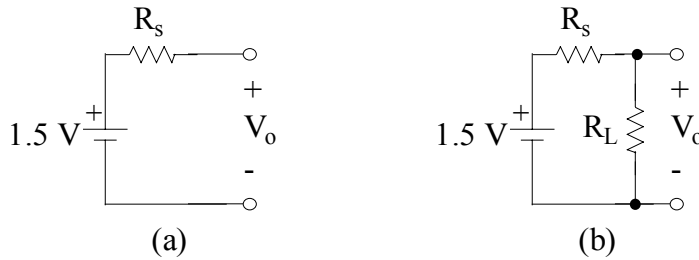


Figure 3.6 Battery equivalent circuit. (a) Unloaded, and (b) Loaded

$R_s$  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

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

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. There is an ideal current source, which can supply the designated current to the circuit connected across it whatever the voltage across it may be, and a *parallel impedance* connected to it, which takes off some current from it. 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:

$$V_o = Z_s I_s(\omega).$$

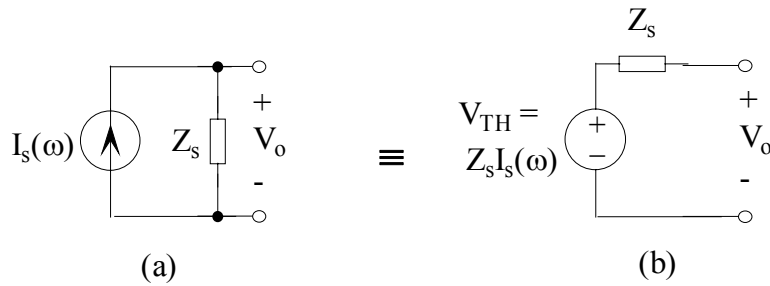


Figure 3.7 (a) Norton current source equivalent circuit, (b) its Thevenin equivalent voltage source

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  $I_s(\omega)$ , hence set  $I_s(\omega)=0$ ), 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  $V_{TH} = Z_s I_s(\omega)$  and a series impedance  $Z_s$ . This model is given in Figure 3.7 (b).

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  $R_s=1.2K$  ( $R_{20}$  in the Audio/TX circuit). HPF is redrawn in Figure 3.8 (a) together with the source.

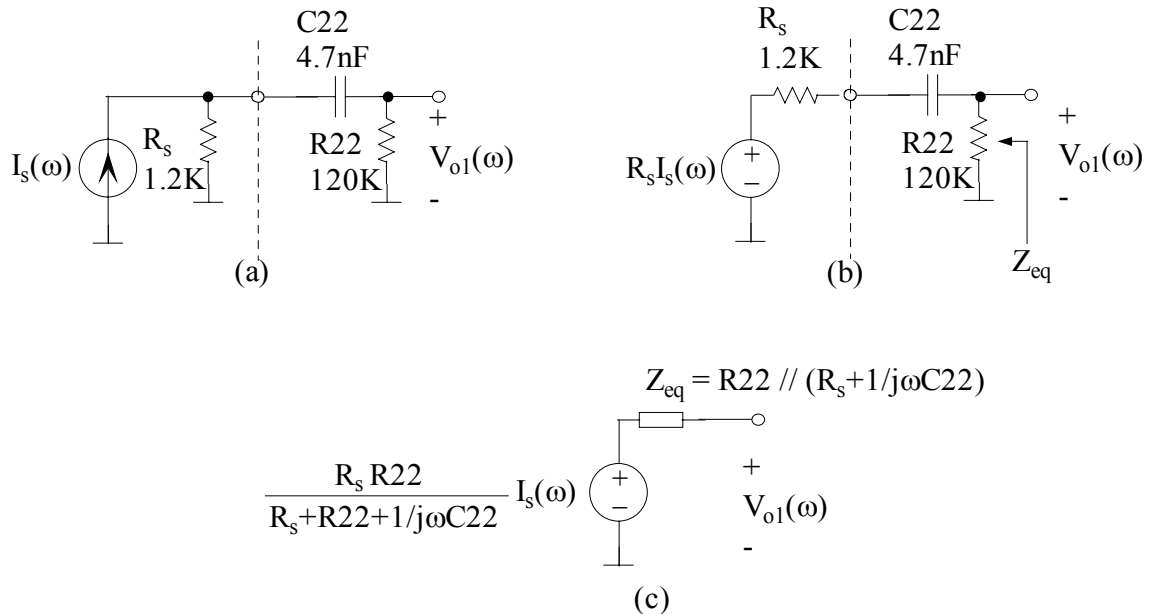


Figure 3.8 (a) Microphone input circuit, (b) and (c) Thevenin equivalent circuits

Conversion of the current source and the parallel resistance into its Thevenin equivalent yields the circuit in Figure 3.8 (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 3.8 (a), into an equivalent circuit. The open circuit voltage at the output terminals is,

$$V_{o1}(\omega) = R_s I_s(\omega) R_{22} / (R_{22} + R_s + 1/j\omega C_{22})$$

in Figure 3.8 (b). Since  $R_s \ll R_{22}$

$$\begin{aligned} V_{o1}(\omega) &\approx R_s I_s(\omega) j\omega C_{22} R_{22} / (1 + j\omega C_{22} R_{22}) \\ &= I_s(\omega) R_s \times jf / \{282 \times (1 + jf/282)\}. \end{aligned}$$

The equivalent resistance is found by first setting the source to zero, i.e.  $R_s I_s(\omega) = 0$ . 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  $R_{22} // (R_s + 1/j\omega C_{22})$ ,

$$\begin{aligned} Z_{eq}(\omega) &= R_{22} (1 + j\omega C_{22} R_s) / [1 + j\omega C_{22} (R_s + R_{22})] \\ &\approx R_{22} (1 + j\omega C_{22} R_s) / (1 + j\omega C_{22} R_{22}) \\ &= 120 (1 + j\omega \times 5.64E-6) / (1 + j\omega \times 5.64E-4) \text{ K}\Omega. \end{aligned}$$

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

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

$Z_{eq}$  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 3.9 (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 ( $\infty$ ). Also the series output impedance of an ideal OPAMP is zero  $\Omega$ . The equivalent circuit of an ideal OPAMP is shown in Figure 3.9 (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:

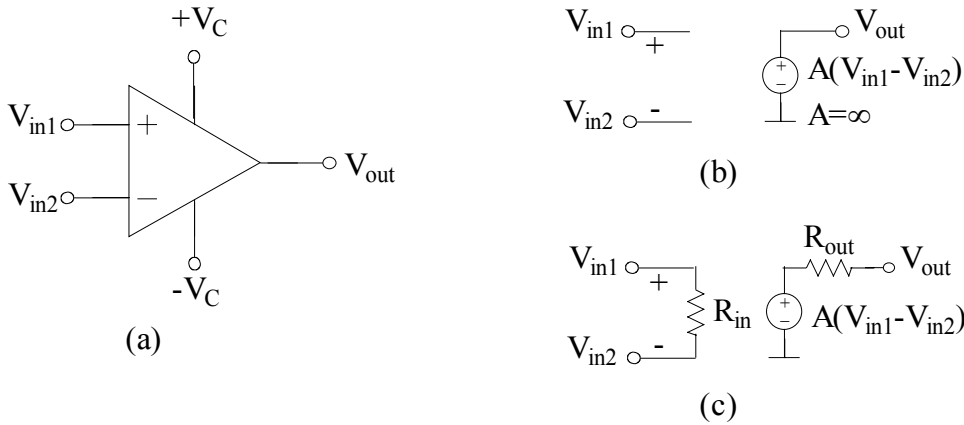


Figure 3.9 OPAMP (a) OPAMP symbol, (b) ideal OPAMP equivalent circuit, and (c) real OPAMP equivalent circuit

$R_{in} \approx 1 \text{ E}12 \ \Omega = 1 \text{ tera } \Omega$   
 $R_{out} \approx 10 \ \Omega$   
 $A \approx 3.0 \text{ E}5$

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 3.10 (a).

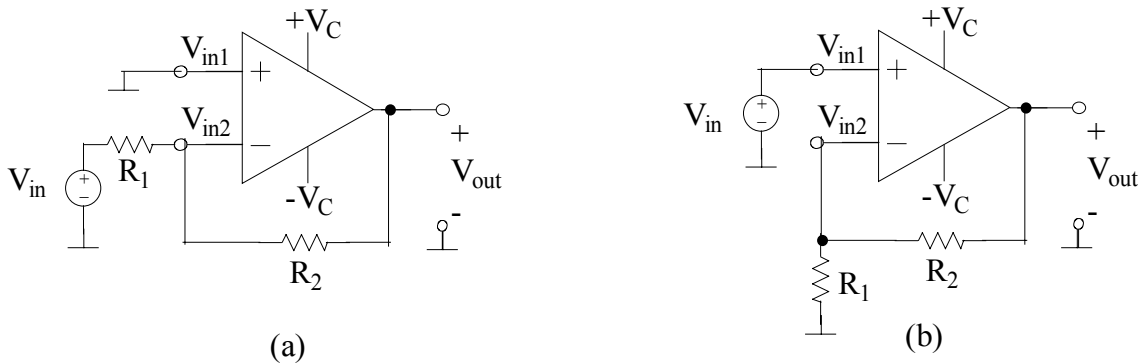


Figure 3.10 OPAMP (a) inverting amplifier, and (b) non-inverting amplifier

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  $\gg R_{out}$  and  $\ll R_{in}$ . Now we can employ the model of the ideal OPAMP and construct the equivalent circuit of inverting amplifier, in Figure 3.11.

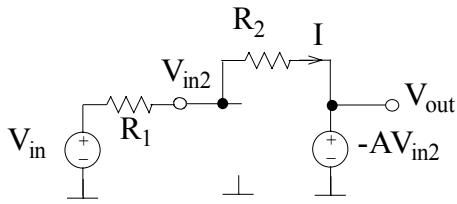


Figure 3.11 Equivalent circuit of inverting amplifier

There are two voltage sources in the circuit of Figure 3.11, 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,

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

and

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

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

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

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

$V_{in2}$ , 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:

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

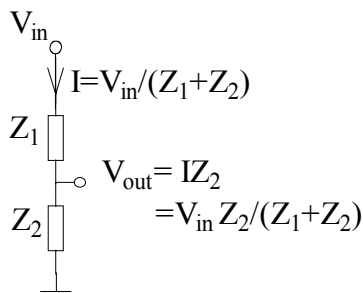
We used the concept of *feedback* in this amplifier.  $R_2$  is connected between the output of the amplifier and its negative input. This resistor *feeds back* a sample of the output signal to its input and  $R_2$  is usually called *feedback resistor*. It is only by feedback that we can control the 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 3.11:

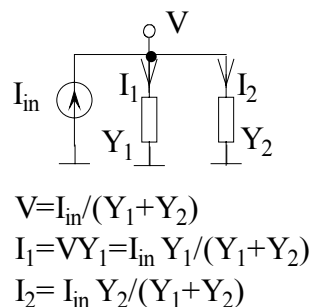
$$\begin{aligned} V_{in2} &= V_{in} - IR_1 = V_{in} - R_1(V_{in} - V_{out})/(R_1 + R_2) \\ &= V_{in} R_2/(R_1 + R_2) + V_{out} R_1/(R_1 + R_2). \end{aligned}$$

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 3.12 are called voltage dividers.



(a)



(b)

Figure 3.12 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 3.10 (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

$$\begin{aligned} V_{out} &= V_{in} A(R_1+ R_2)/(A R_1+R_1+ R_2) \\ &= V_{in} (1+ R_2/ R_1) \text{ for } A \rightarrow \infty. \end{aligned}$$

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 3.13.

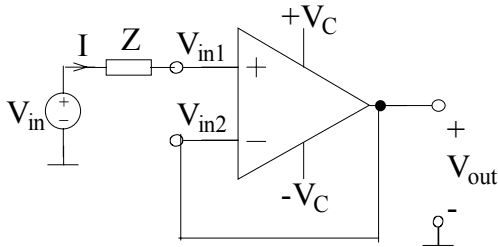


Figure 3.13 Unity gain amplifier

The first observation in this circuit is the input current is

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

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

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

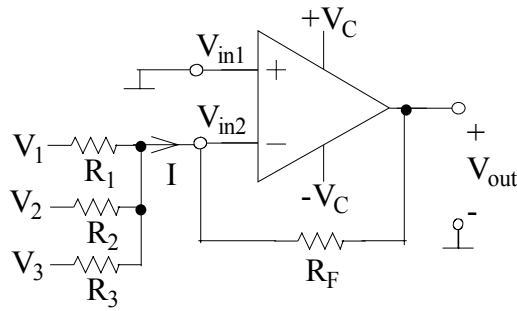
$V_{out}$  is equal to  $V_{in}$  in this configuration. This amplifier is called by a few names such as, *unity gain amplifier*, *buffer amplifier* and *voltage follower*. Although it does not provide any voltage gain to the input signal, it is used to transfer the input voltage intact to the output 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 3.13 is essentially zero since  $V_{in}$  appears across an indefinitely large impedance. If a resistance of  $50 \Omega$  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 3.14 (a) and (b), respectively.

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

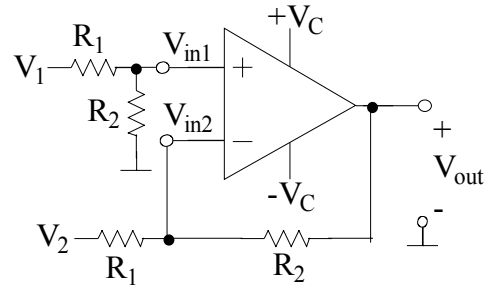


$$V_{in2} \approx V_{in1} = 0$$

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

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

(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 3.14 (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 3.15 (a). Since the  $V_{in2}$  node is virtual ground, i.e.  $V_{in2} = 0$ , capacitor current is

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

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

Therefore the differentiator output becomes

$$v_{out}(t) = -R i_C(t)$$

$$= -RC dv_{in}(t)/dt.$$

Similarly, for the integrator, the resistor current is

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

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

$$v_{out}(t) = 1/C \int \{-i_R(t)\} dt$$

$$= -1/(RC) \int v_{in}(t) dt.$$

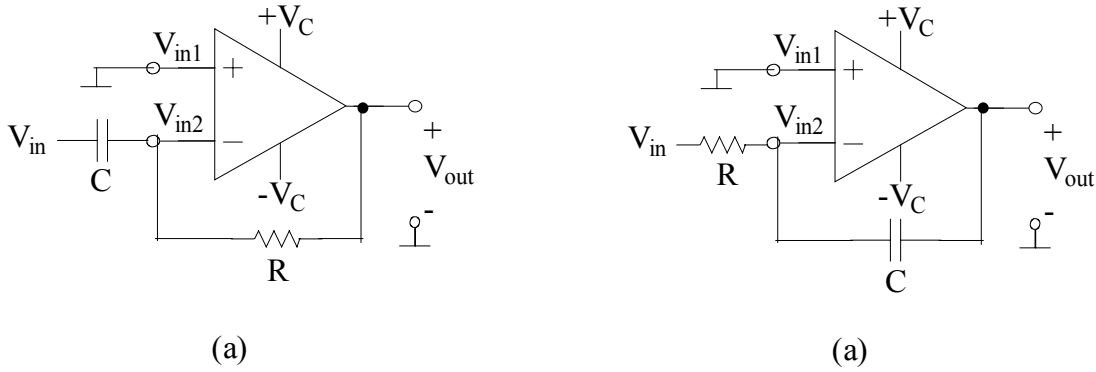


Figure 3.15 (a) Differentiator, and (b) Integrator

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 3.16, where a part of the microphone amplifier of TRC-10 is shown.

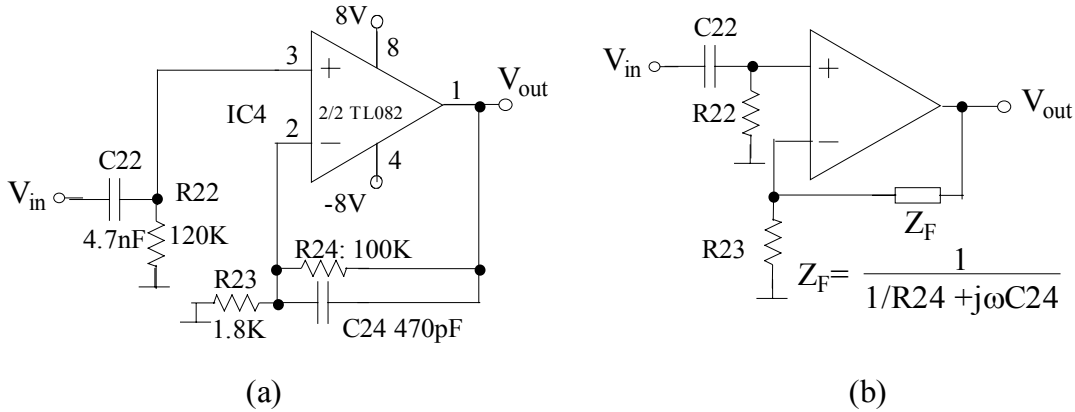


Figure 3.16 Microphone amplifier (a) the actual circuit, and (b) the simplified circuit

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 R22 are connected, is,

$$V_{in1}(\omega) = V_{in}(\omega) \frac{j\omega C22 R22}{(1 + j\omega C22 R22)}$$

$$= V_{in}(\omega) \{j\omega / 282 / (1 + j\omega / 282)\}.$$

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

$$V_{out} = V_{in1} (1 + Z_F / R23) \text{ for } A \rightarrow \infty$$

$$= V_{in1} [1 + (R24 / R23) / (1 + j\omega R24 C24)].$$

The time constant  $R24 \times C24$  is  $47 \text{ E}-6$  sec. Hence,

$$V_{out} = V_{in}(\omega) \times \frac{jf/282}{(1 + jf/282)} \times \left[ \frac{55.6}{(1 + jf/3.4 \text{ E}3)} + 1 \right]$$

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 3.17.

$V_{inos}$  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.

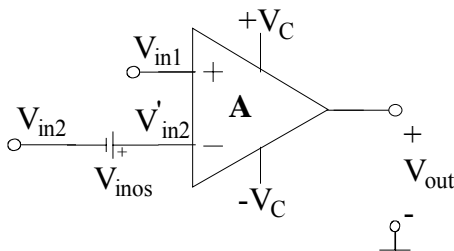


Figure 3.17 Input offset voltage

The effect of input offset voltage on output when there is feedback is considered in Figure 3.18. Notice that both inverting and non-inverting amplifier configurations given in Figure 3.10 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

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

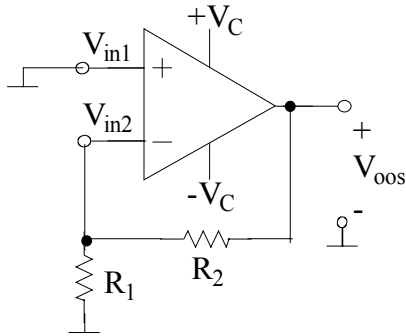


Figure 3.18 Offset voltage in feedback amplifier

Since  $V_{in1} = 0$  and  $V_{inos} = V_{in1} - V_{in2}$  with no externally applied input voltage,  $V_{00s}$  becomes

$$V_{00s} = - (1 + R_2/R_1) V_{inos}.$$

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

It is possible to compensate for  $V_{00s}$  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  $1 + R_{24}/R_{23}$ , or 56.6. Hence  $|V_{00s}|$  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 3.19

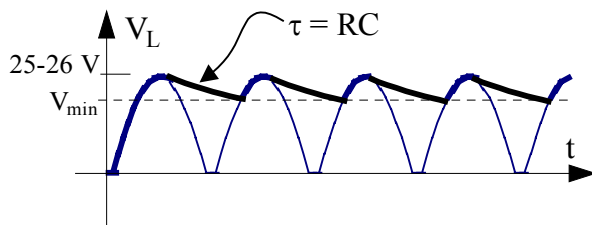


Figure 3.19 Full wave rectified and filtered a.c. supply voltage

to a well defined d.c. voltage, without any ripple, like the one depicted in Figure 3.20.

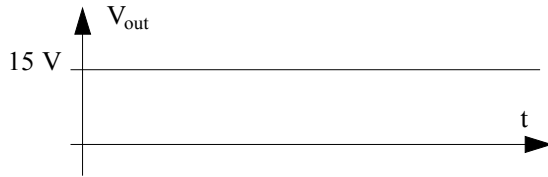


Figure 3.20 Regulated 15 V d.c. voltage

From the data sheet of voltage regulators (e.g. LM7808) we know that  $V_{min}$  in Figure 3.19 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 3.21. 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.

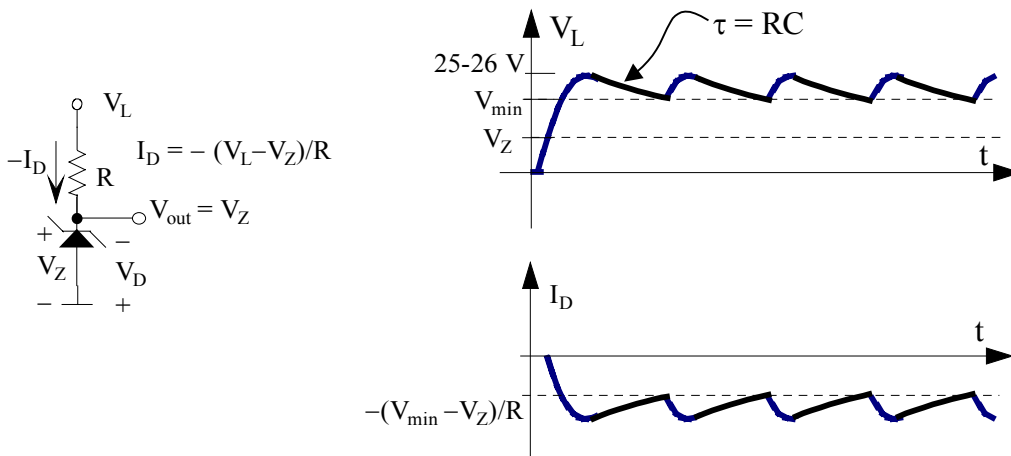


Figure 3.21 Biased zener diode and the current through it

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 3.22. 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  $90^\circ$  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  $V_{out}$ .

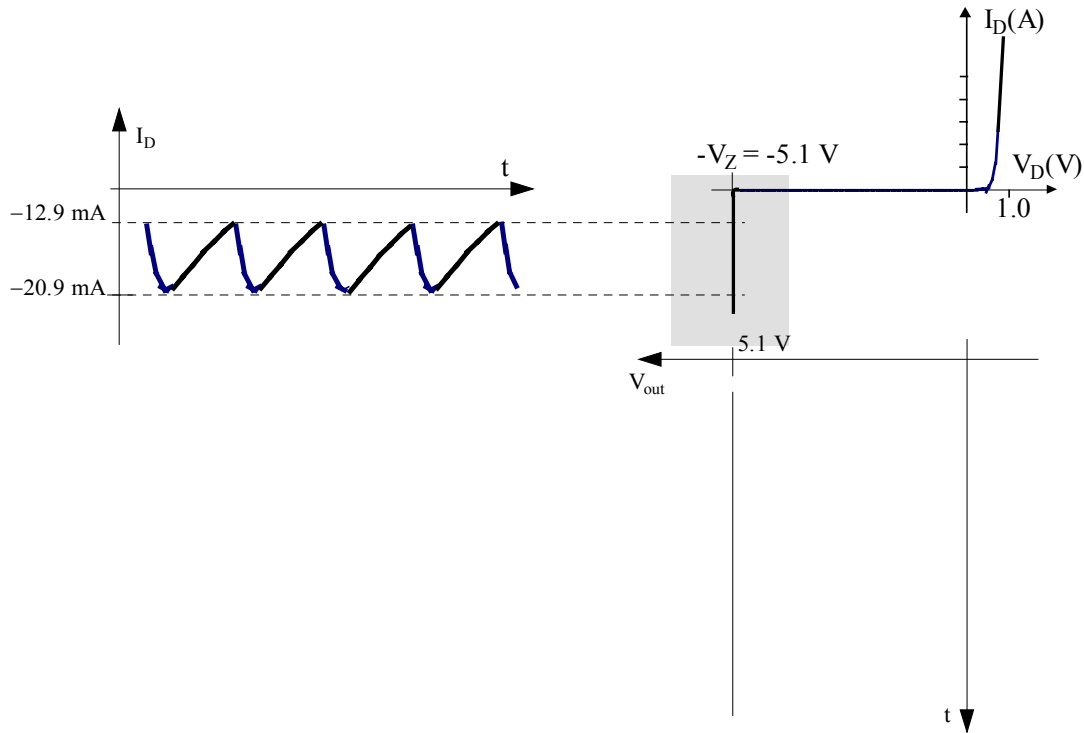


Figure 3.22 Zener voltage is fixed despite the variations on the current

For example, if we connect a load  $R_L$  across the diode, we obtain the circuit in Figure 3.23. 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  $\Omega$  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  $R_L=300 \Omega$ . The current  $I_L$  becomes 13.8 mA, and zener current is zero.

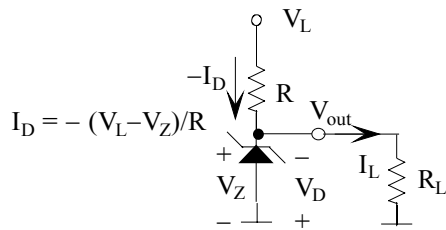


Figure 3.23 Reverse biased zener diode with load current

We can use a voltage follower at this node to improve the amount of current that can be drawn, as shown in the Figure 3.24.



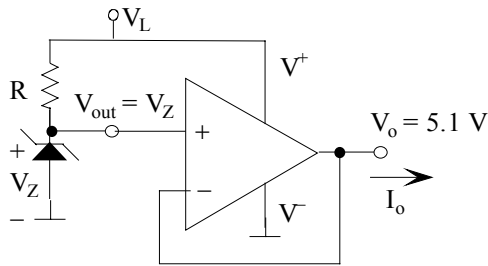


Figure 3.24 Voltage regulator

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

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

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](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 3.25.

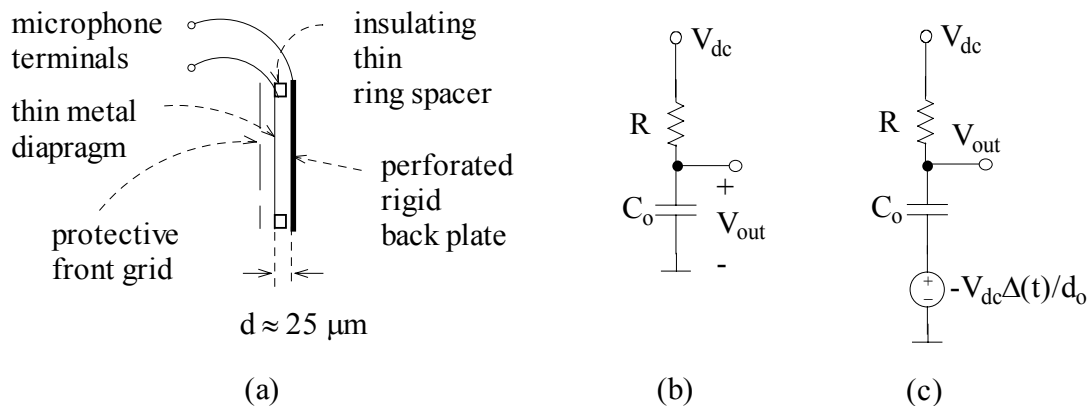


Figure 3.25 Condenser microphone, (a) Mechanical structure, (b) Microphone circuit, and (c) Equivalent circuit

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,  $C_0$ .  $C_0$  is given as

$$C_0 = \epsilon_0 A / d_0,$$

where  $\epsilon_0$  is  $8.854 \times 10^{-12}$  F/m, the permittivity of free space,  $A$  is the area of the plates and  $d_0$  is the spacing of the plates. These microphones are used in circuits like the one shown in Figure 3.25 (b).  $C_0$  is charged up to  $V_{dc}$  through  $R$ . The electric field between the plates is

$$E = V_{dc} / d_0$$

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

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,

$$d = d_0 + \Delta(t),$$

where  $\Delta(t)$  is proportional to the sound signal amplitude, and  $\Delta(t) \ll d_0$ . Then the electric field becomes

$$\begin{aligned} E &= V_{\text{out}}/d \\ &\approx V_{\text{dc}}/[d_0 + \Delta(t)] \\ &\approx (V_{\text{dc}}/d_0)(1 - \Delta(t)/d_0) \end{aligned}$$

and therefore,

$$\begin{aligned} V_{\text{out}} &= Ed \\ &\approx Ed_0 \\ &= V_{\text{dc}} - V_{\text{dc}}\Delta(t)/d_0. \end{aligned}$$

The sound is thus converted to the electrical signal  $V_{\text{dc}}\Delta(t)/d_0$ . The variation  $\Delta(t)$  is the membrane displacement and  $|\Delta(t)|_{\text{max}}$  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  $d_0$ : smaller  $d_0$  is, more sensitive the microphone is. Furthermore  $d_0$  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  $\mu\text{m}$  in most commercial microphones. The expression for  $V_{\text{out}}$  above, tells that there is a voltage source  $-V_{\text{dc}}\Delta(t)/d_0$  additive to the capacitor (i.e. in series with the capacitor). Figure 3.25 (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  $1\text{cm}^2$ ,  $C_0$  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  $\mu\text{m}$  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 3.26.

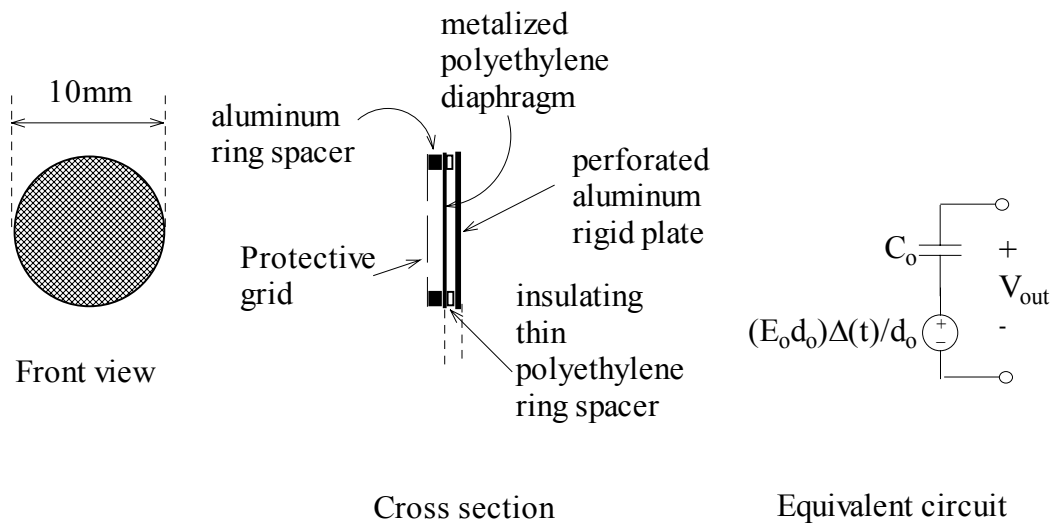


Figure 3.26 Electret condenser microphone

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,  $E_0$ , which is equivalent to  $V_{dc}/d_0$  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 3.27.

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.

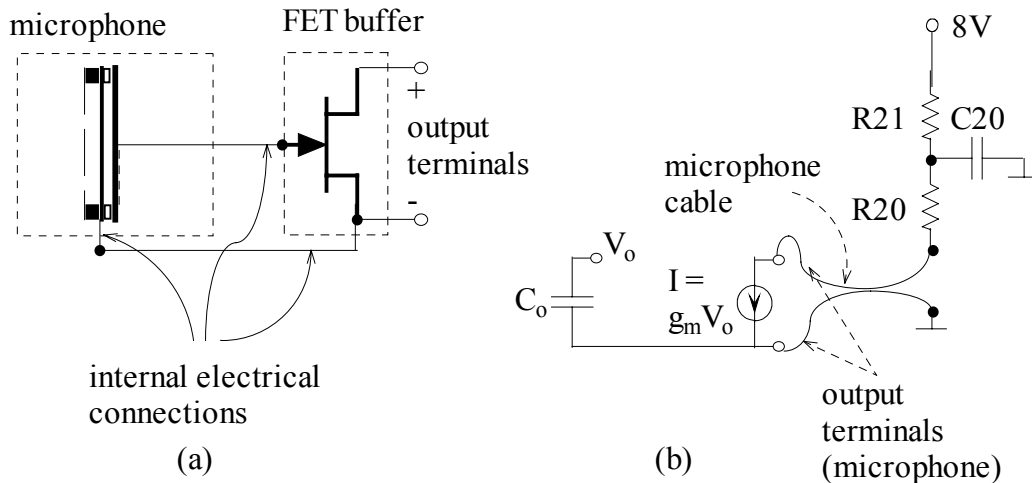


Figure 3.27 Electret condenser microphone of TRC-10 (a) components in the microphone housing, and (b) microphone equivalent circuit and the way it is used in TRC-10.

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  $\square$  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 millivolts 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 3.28 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 3.28?

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.

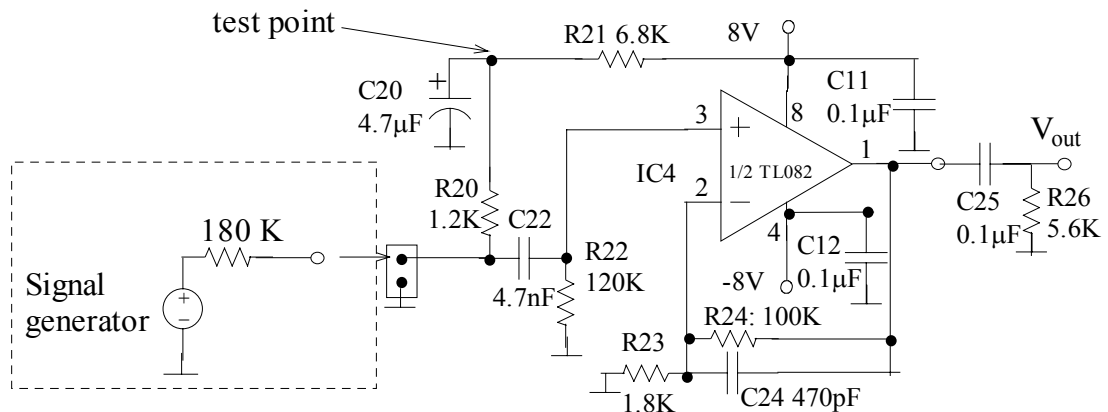


Figure 3.28 Microphone amplifier

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  $\pm 8$  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 3.29. 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

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

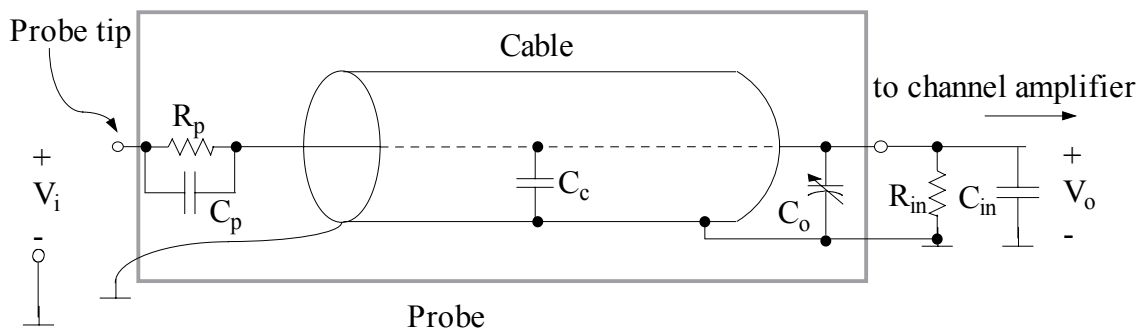


Figure 3.29 Probe model and probe compensation

When  $C_o$  is adjusted such that  $(C_c + C_o + C_{in})R_{in}||R_p = C_p R_p$ , or equivalently  $(C_c + C_o + C_{in})R_{in} = C_p R_p$ , the transfer response becomes  $[1+(R_p/R_{in})]^{-1}$  for all  $\omega$ . 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  $C_p$ . 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 3.30 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?

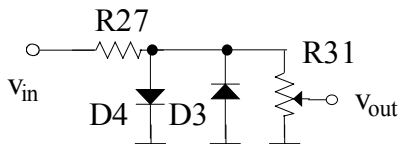


Figure 3.30 Diode clipper

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 3.28. 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 3.31.

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.

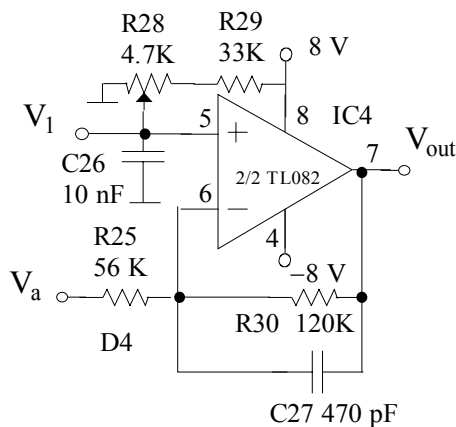


Figure 3.31 Audio signal output circuit

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 3.32.

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.*

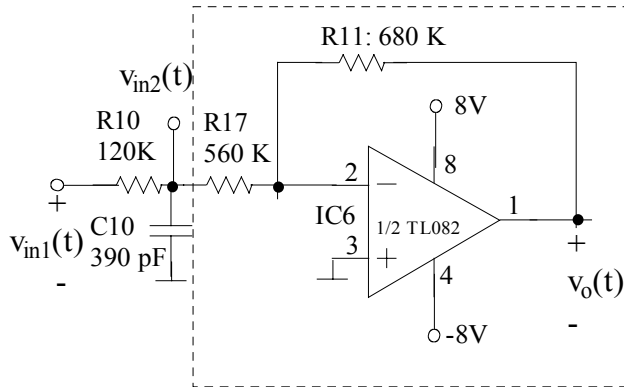


Figure 3.32 Loudspeaker amplifier-first stage

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  $\pm 8$  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  $0 \Omega$ , 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 millivolts 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  $\mu$ F 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 3.33.

Loudspeakers are most commonly made to have a circular symmetry. Figure 3.33 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.

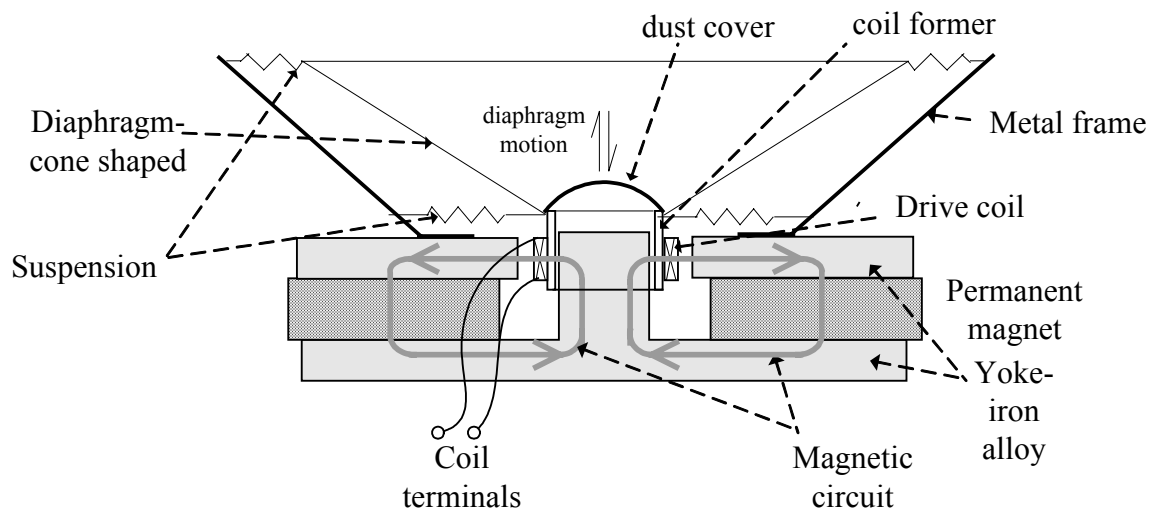


Figure 3.33 Loudspeaker

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 3.34.

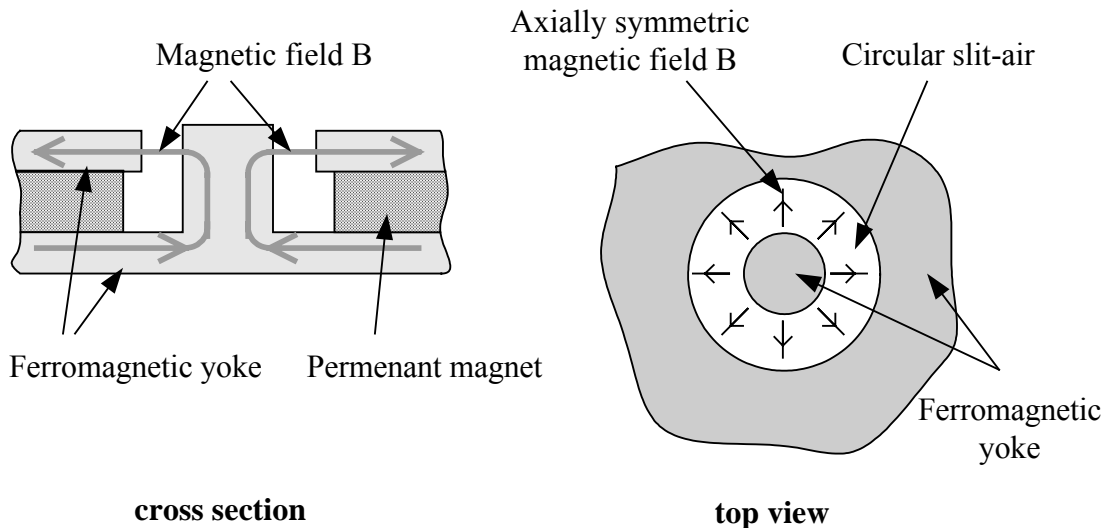


Figure 3.34 Magnetic circuit

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 3.35. 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 lie 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

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

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.

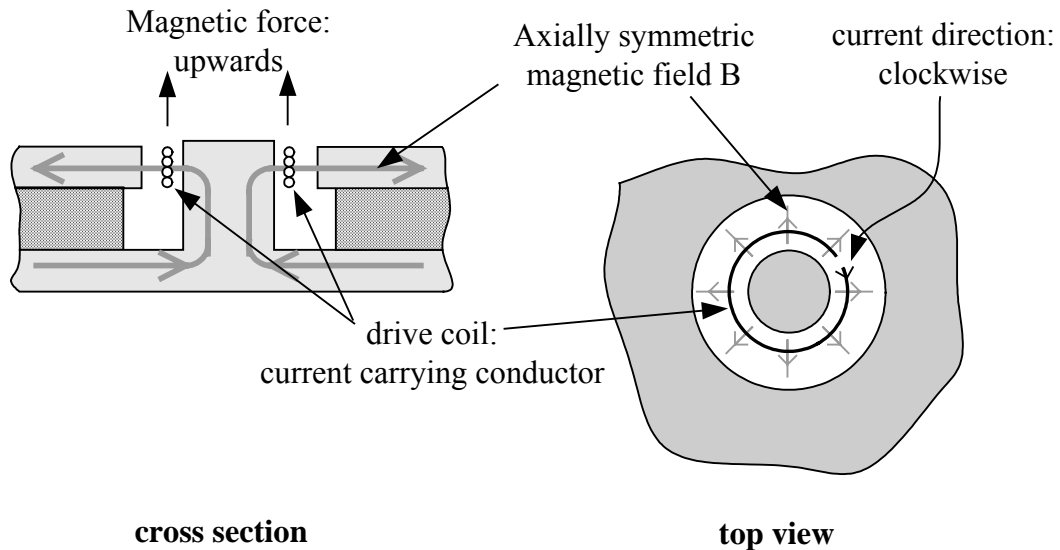


Figure 3.35 Current carrying coil in magnetic circuit

The efficiency,  $\eta$ , 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.  $8 \Omega$  and  $16 \Omega$  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.

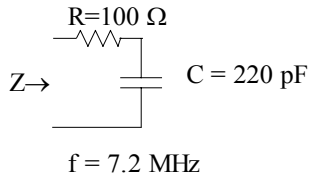
17. Connect the microphone amplifier output (Audio/TX pigtail) to the Audio/RX input using a crocodile jumper cable. Connect the microphone. Ask your friend to speak to the microphone. Check that audio-audio chain works.

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.

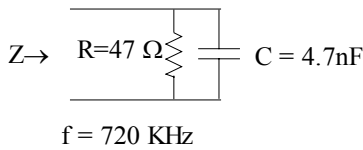
We are now ready to proceed with RF circuits.

### 3.8. Problems

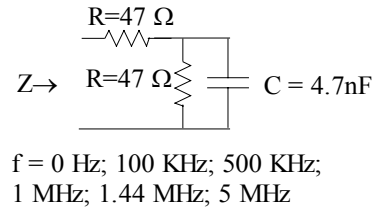
1. Expand  $\exp(j\theta)$  in Taylor series, group the real and imaginary parts and show that real series corresponds to the expansion of  $\cos(\theta)$  and imaginary series correspond to the expansion of  $\sin(\theta)$ .
2. Show that capacitance is a linear circuit element.
3. Show that, if a circuit 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):



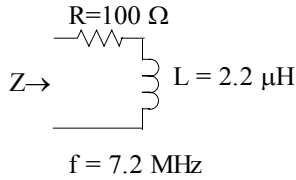
(a)



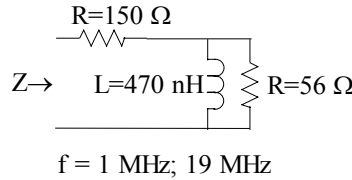
(b)



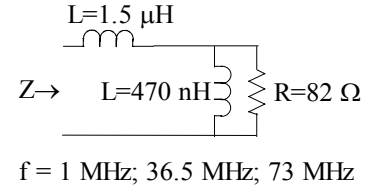
(c)



(d)

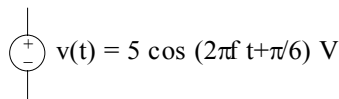


(e)

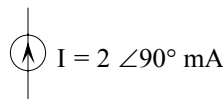


(f)

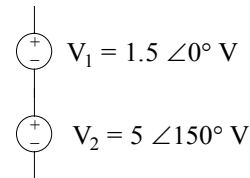
8. The voltage (current) sources given below are connected to the circuits in problem 7. The frequencies of the sources are as given in problem 7. Find the current through (voltage across) the circuits.



(a)



(b)



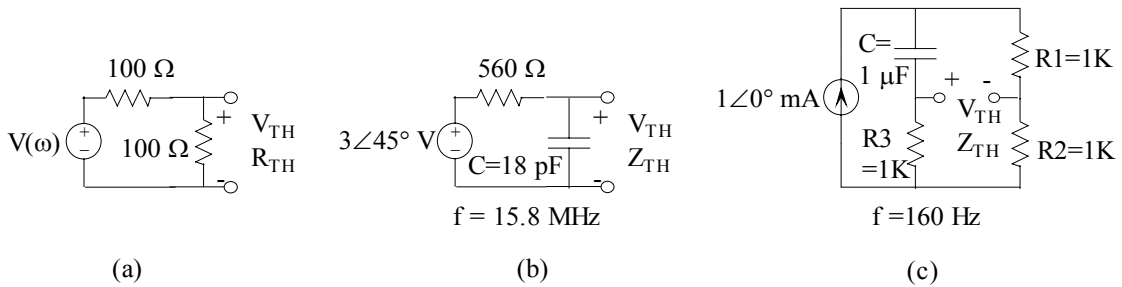
(c)

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  $\Omega$ ,  $K$  for resistance;  $\mu$ ,  $p$  for capacitance, etc.), for each circuit. Determine the frequency and angular frequency in each case.

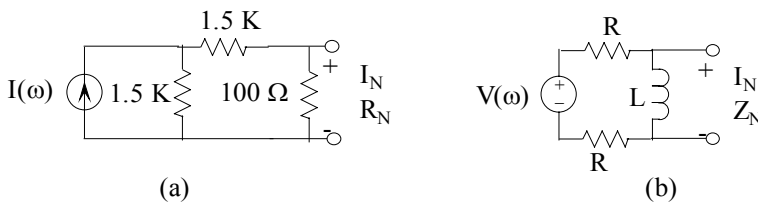
- |  |   |
|--|---|
| (a) $v(t) = 28.3 \cos(628t + 150^\circ) V$ ;           | $i(t) = 11.3 \cos(628t + 140^\circ) mA$           |
| (b) $v(t) = 5 \cos(2\pi \times 300t - 25^\circ) V$ ;   | $i(t) = 8 \cos(2\pi \times 300t + 5^\circ) mA$    |
| (c) $v(t) = 10 \cos(2\pi \times 796t - 150^\circ) V$ ; |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

11. A series circuit has a resistor  $R = 120 \Omega$  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  $\pi/4$ .

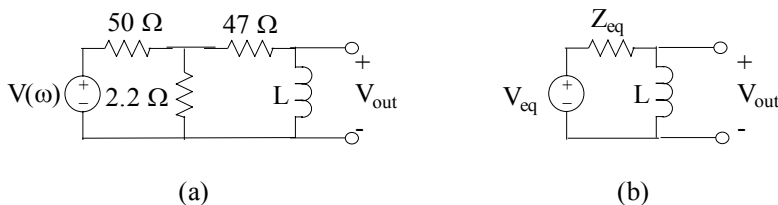
13. A series RC circuit has  $C = 470 \text{ pF}$ . Find  $R$  if the phase difference is  $30^\circ$  at  $1 \text{ KHz}$ .
14. The voltage and current of a two element series circuit at  $500 \text{ KHz}$  are  $V=3\angle-45^\circ \text{ V}$  and  $I = 1\angle-120^\circ \text{ mA}$ . When the frequency is changed to another value  $f$ , the phase difference between the voltage and current becomes  $30^\circ$ . 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.



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.



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



20. This problem illustrates how a unity feedback amplifier is used to avoid loading effects. Consider the divider circuit in the Figure 3.36(a). What is  $V_{out}$ ? Assume we want to apply  $V_{out}$  across a  $1\text{K}$  resistor as shown in part (b). What is  $V_{out}$  now? Now assume we place a buffer amplifier between the divider and  $1\text{K}$  resistor as in part (c). Find  $V_{out}$ .

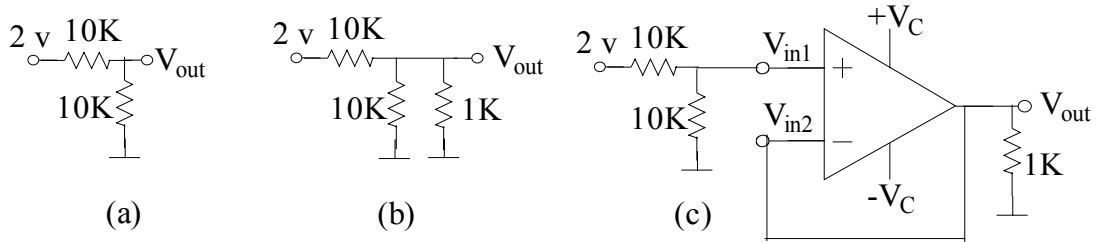
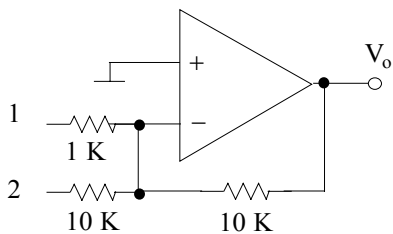


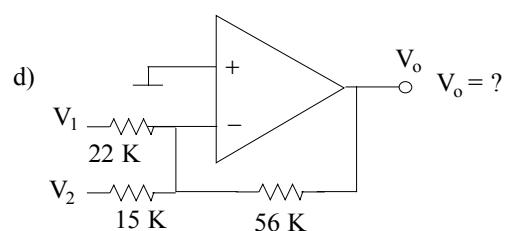
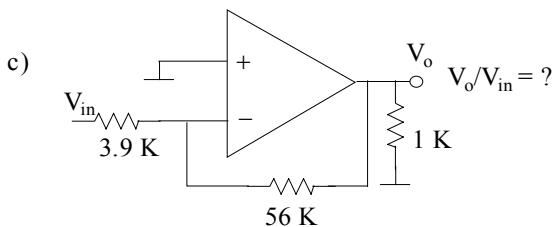
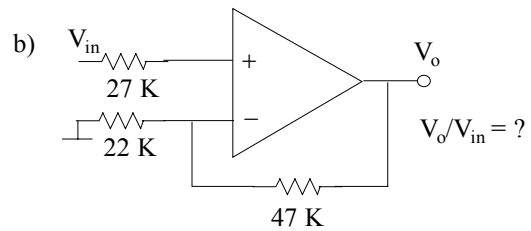
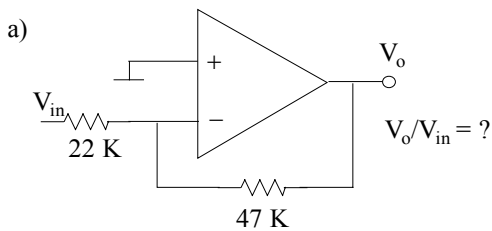
Figure 3.36

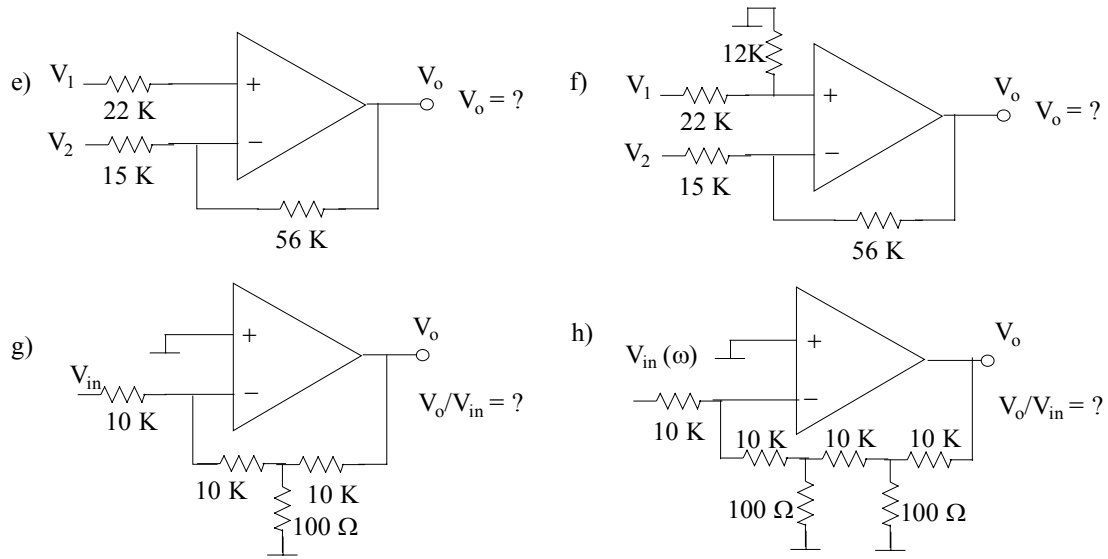
21. Assume there are two signals,  $V_1$  and  $V_2$ . Design a summing amplifier to produce  $V_{out} = 2 V_1 + 0.5 V_2$ , using at most (a) two OPAMPs, and (b) one OPAMP.
22. Make a table indicating what terminals to connect to the input signal source or the output in order to get all possible (different)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



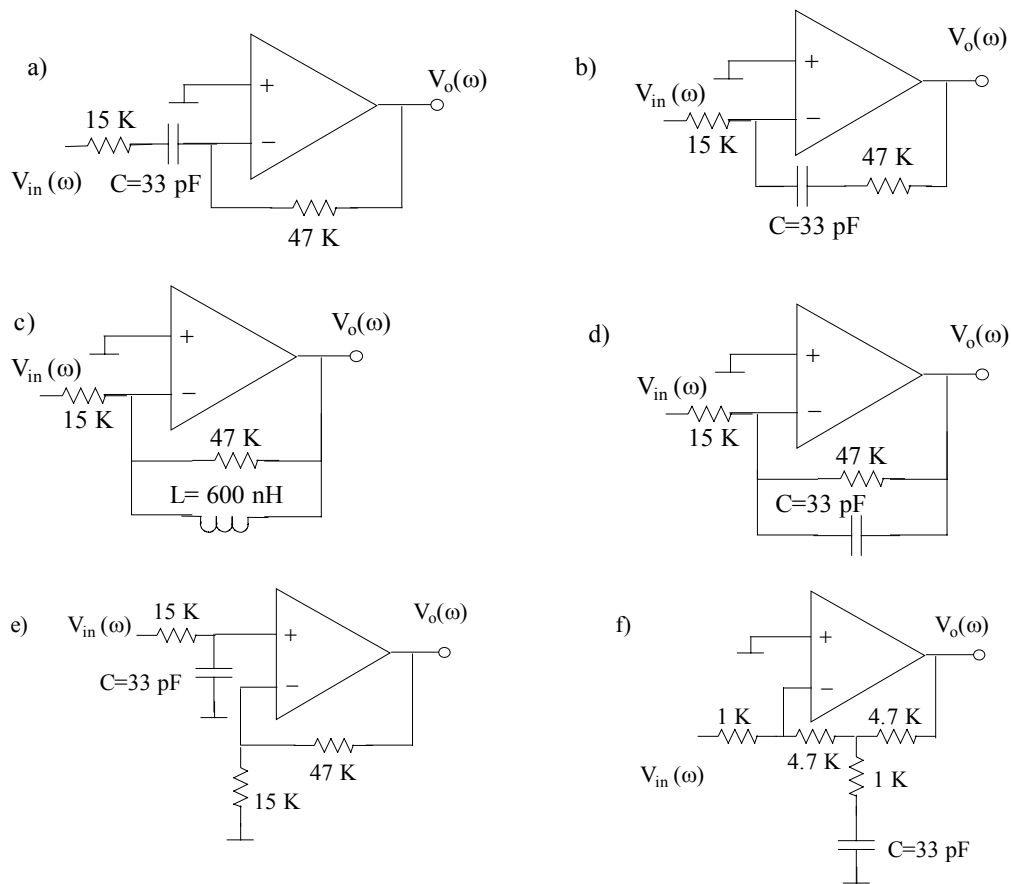
1	2	Gain	Rin
i/p	NC	-10	1K
i/p	o/p	-5	1K
NC	i/p		
i/p	i/p		
o/p	i/p		

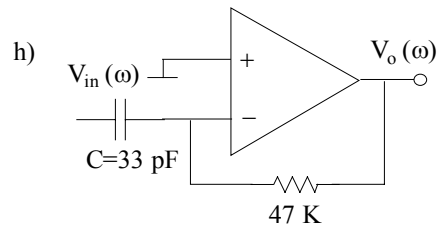
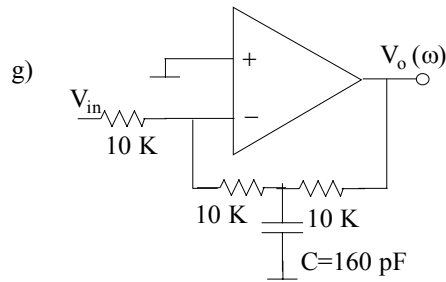
23. Assume that OPAMPs are ideal in the following problems.





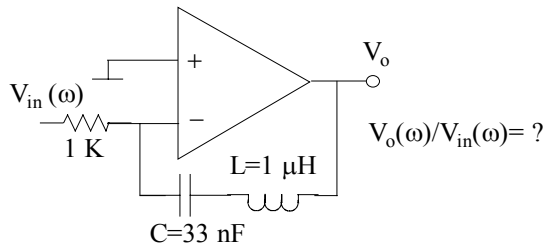
24. Find  $H(0) = V_o(0)/V_{in}(0)$  and  $H(\infty) = V_o(\infty)/V_{in}(\infty)$  in the following circuits.



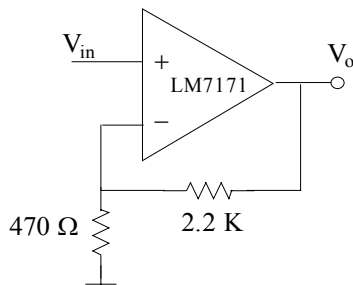


25. Find the transfer functions  $V_o(\omega)/V_{in}(\omega)$  for the circuits given in problem 24.

26. 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  $\infty$ ?



27. 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?





## Chapter 4 : TUNED CIRCUITS

*Frequency selectivity* is a fundamental concept in electronic communications. Communicating in a particular frequency band requires the ability of confining the signals into that band. Any filter is a frequency selective circuit. *Tuned circuits* are the most commonly used frequency selective circuits.

### 4.1. Parallel resonance

Consider the circuit in Figure 4.1(a). A capacitor and an inductor are connected in parallel and are driven by a current source.  $I$  is the current phasor  $I_p \angle 0^\circ$  of a sinusoidal source signal  $i(t) = I_p \cos \omega t$  at an arbitrary frequency  $\omega$ .

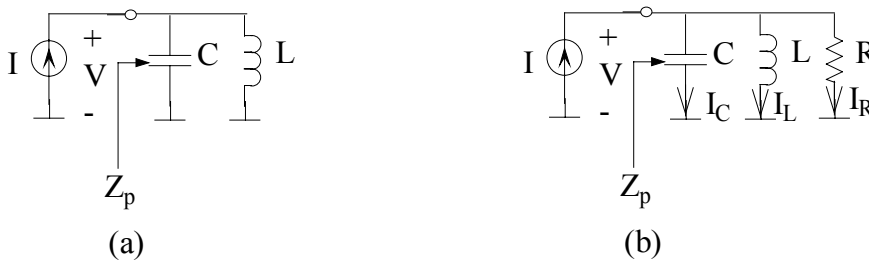


Figure 4.1 Parallel tuned circuit

The combined parallel impedance  $Z_p(\omega)$  at that frequency is

$$\begin{aligned} Z_p(\omega) &= (j\omega C + 1/j\omega L)^{-1} \\ &= j\omega L / (1 - \omega^2 LC) \end{aligned}$$

Notice that this expression is always imaginary and is equal to  $\infty$  when

$$\omega^2 LC = 1 \Rightarrow \omega = 1/(LC)^{1/2}.$$

This means that if the angular frequency is adjusted to this special frequency, then any amplitude of the current source, however small it may be, generates a voltage of infinitely large amplitude across this simple circuit. This phenomenon is called *resonance*, and the frequency  $f_0 = 1/2\pi(LC)^{1/2}$  or  $\omega_0 = 1/(LC)^{1/2}$  is called the *resonance frequency*. A resonating circuit formed by a capacitor and an inductor in parallel, is called a *parallel tuned circuit*.

This circuit does not contain any resistance. Resistor means power loss and this circuit is lossless. This is not a situation we face in real world. Now consider a slightly modified circuit given in Figure 4.2(b). Impedance now becomes,

$$\begin{aligned} Z_p(\omega) &= (1/R + j\omega C + 1/j\omega L)^{-1} \\ &= j\omega LR / (R + j\omega L - \omega^2 LCR) \\ &= j\omega L / [(1 - \omega^2 LC) + j\omega L/R]. \end{aligned}$$

This impedance is complex and it indicates a resonance. At frequency  $\omega_0 = 1/(LC)^{1/2}$ ,  $Z(\omega)$  becomes,



$$Z_p(\omega_0) = j\omega_0 L / [(0) + j\omega_0 L / R] \\ = R.$$

The imaginary part of the impedance vanishes at resonance.

We already know that if there would not be any resistance at all, the remaining parallel LC-circuit has infinite impedance at  $\omega_0$ . Infinitely large impedance in parallel with R yields R. Hence the voltage across the circuit at  $\omega = \omega_0$  is

$$V = IR$$

where V is the output voltage phasor. This means that the current through the resistor is exactly equal to the input current at resonance frequency, i.e. at  $\omega = \omega_0$

$$I_R = I.$$

Consider the other branch currents now. Since the voltage across the circuit is  $V = IR$ , the current in the capacitor branch is

$$I_C = IR / (1 / j\omega_0 C) = I (j\omega_0 CR).$$

Substituting the value of  $\omega_0$  into above expression, capacitor current phasor at resonance frequency becomes,

$$I_C = I (jR) \times (C/L)^{1/2}.$$

*Although there is no net current flowing into parallel LC at  $\omega_0$ , there is a finite current flowing through the capacitor.* This current is purely imaginary. Its magnitude is determined by the component values as  $R(C/L)^{1/2}$  times the input current and can be very large. This is a very interesting property of resonating circuits.

Similarly, the current phasor of inductor branch at resonance frequency is

$$I_L = IR / (j\omega_0 L) = I (-jR / \omega_0 L) = I (-jR) \times (C/L)^{1/2}.$$

Again there is a finite and imaginary current flowing through the inductor. But notice that its magnitude is exactly equal to that of  $I_C$  and its phase is exactly  $180^\circ$  out of phase with  $I_C$ , i.e. it is in exactly the opposite direction. Hence, when we add up the total current at the node joining L and C,  $I_C$  and  $I_L$  sums up to zero and we are left with  $I = I_R$  only, at resonance.

Resonance is a peculiar phenomenon!

#### 4.1.1. Energy in tuned circuits

It is shown in Section 2.3.1 that the energy stored in a capacitor at any time  $t_1$ , is given as

$$E = C v^2(t_1)/2$$

where  $C$  is the capacitance and  $v(t)$  is the voltage across the capacitor. Similarly the energy stored in an inductor is given as

$$E = L i_L^2(t_1)/2$$

in Section 2.5, where  $L$  is the inductance and  $i_L(t)$  is the current through the inductor. The voltage  $v(t)$  across the capacitor in the circuit of Figure 4.1(b) is

$$v(t) = I_p R \cos(\omega_0 t),$$

at resonance. The stored energy in this capacitor is

$$E_C = (C/2) [I_p R \cos(\omega_0 t)]^2 = (C/2) [(I_p R)^2/2] [1 + \cos(2\omega_0 t)].$$

The average value of the stored energy can be found from above expression as

$$E_{avC} = C (I_p R)^2/4$$

and the peak stored energy in the capacitor as

$$E_{peakC} = C (I_p R)^2/2.$$

Similarly, the energy stored in the inductor in the same circuit can be found from the inductor current. The inductor current phasor is  $I_p(-jR/\omega_0 L)$ , or  $I_p(R/\omega_0 L) \angle -90^\circ$ , which yields an inductor current  $i_L(t)$  of

$$i_L(t) = I_p(R/\omega_0 L) \cos(\omega_0 t - 90^\circ) = I_p(R/\omega_0 L) \sin(\omega_0 t).$$

The stored energy in the inductor becomes

$$E_L = L i_L^2(t_1)/2 = (L/2) [I_p(R/\omega_0 L)]^2 (1/2) [1 - \cos(2\omega_0 t)].$$

The average and peak stored energy in the inductor are

$$E_{avL} = L [I_p(R/\omega_0 L)]^2/4$$

and

$$E_{peakL} = L [I_p(R/\omega_0 L)]^2/2,$$

respectively. In any one of the reactive components,  $C$  and  $L$ , in this parallel tuned circuit, the stored average energy is half of stored peak energy.

Since  $(\omega_0^2 L)^{-1}$  is equal to  $C$  at resonance, we can write  $E_{avL}$  and  $E_{peakL}$  as

$$E_{avL} = [I_p^2 (R^2 C)] / 4 = E_{avC},$$

and

$$E_{\text{peakL}} = [I_p^2(R^2C)] / 2 = E_{\text{peakC}},$$

respectively. Therefore, both the average and peak stored energy in these components are equal in parallel tuned circuit. A closer examination of time dependent expressions for energy in either component reveals that when the stored energy in one of them reaches to a maximum, the stored energy in the other becomes zero.

It is useful to find the total stored energy  $E_{\text{av}}$  in the circuit:

$$\begin{aligned} E_{\text{av}} &= E_{\text{avC}} + E_{\text{avL}} \\ &= [I_p^2(R^2C)] / 2, \end{aligned}$$

which is equal to the peak stored energy in either component. As a matter of fact, when the stored energy in both components  $E_C$  and  $E_L$  are summed up at any time instant, the total stored energy  $E_s$  in the circuit also becomes

$$E_s = C (I_p R)^2 / 2.$$

#### 4.1.2. Quality of resonance

*Quality* of a resonating circuit is defined as *the ratio of the stored energy in the circuit to the dissipated energy during (1/ω<sub>o</sub>) seconds*. The dissipated power is, of course, the power delivered to the resistor,

$$P_d = I_R^2 R / 2 = I_p^2 R / 2,$$

and the dissipated energy during (1/ω<sub>o</sub>) seconds is,

$$E_d = I_p^2 R / (2\omega_o).$$

The quality factor is,

$$\begin{aligned} \mathbf{Q} &= E_s / E_d \\ &= \omega_o \mathbf{CR} \\ &= \mathbf{R} / \omega_o \mathbf{L} \end{aligned}$$

for a *parallel RLC circuit*.

Consider the impedance expression for the RLC-circuit of Figure 4.1(b),  $Z_p(\omega) = j\omega L / [(1 - \omega^2 LC) + j\omega L/R]$ . After a little complex algebra, this can be written as,

$$Z_p(\omega) = R / [1 + jQ(\omega/\omega_o - \omega_o/\omega)],$$

in terms of  $R$ ,  $Q$  and  $\omega_o$ . We can see that when  $\omega = \omega_o$ ,  $|Z_p(\omega)|$  reaches a maximum at  $Z_p(\omega) = R$  and as frequency deviates from  $\omega_o$ ,  $|Z_p(\omega)|$  decreases. The real and imaginary parts [ $Z_p(\omega) = R_p(\omega) + jX_p(\omega)$ ] of the impedance of a parallel circuit,

normalized to  $R$ , versus frequency is given in Figure 4.2. This impedance is calculated for a circuit, which has a  $Q$  of 10.

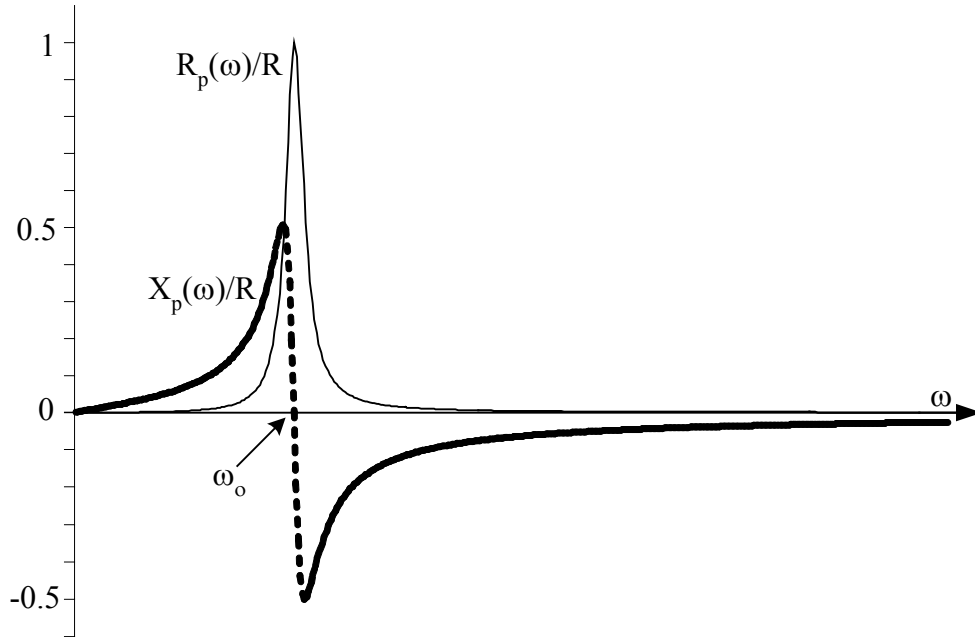


Figure 4.2 Normalized real and imaginary parts of  $Z_p$

While  $R_p(\omega)$  reaches  $R$  at resonance,  $X_p(\omega)$  becomes zero. There are two frequencies  $\omega_1$  and  $\omega_2$ , one of which makes the above expression  $Z_p(\omega_1) = R/(1-j)$  and the other,  $Z_p(\omega_2) = R/(1+j)$ .  $R_p(\omega)$  becomes  $R/2$  in both cases, while  $X_p(\omega_1) = R/2$  and  $X_p(\omega_2) = -R/2$ . These frequencies are  $\omega_{2,1} = \pm\omega_0/(2Q) + (1+4Q^2)^{1/2}\omega_0/(2Q)$ . Note that,  $\omega_{2,1} \approx \omega_0 \pm \omega_0/(2Q)$  within 5% for  $Q \geq 1.5$ , and within 1% for  $Q > 3.5$ . The difference between these frequencies,

$$\omega_2 - \omega_1 = \omega_0/Q$$

is called the *3-dB bandwidth (BW)* of the tuned circuit. The BW of the tuned circuit in the figure is 1/10 of resonance frequency, since  $Q$  is chosen as 10.

The resonance frequency is the geometrical mean of  $\omega_1$  and  $\omega_2$ , i.e.

$$\omega_0 = \sqrt{\omega_1 \omega_2} .$$

For large values of  $Q$ , approaches to the *center frequency*, the arithmetic mean of  $\omega_1$  and  $\omega_2$ .

The variation of the magnitude of  $Z_p(\omega)$  with respect to frequency is given in Figure 4.3.

In this figure, the magnitude of impedance for a circuit with  $Q = 10$ , as in Figure 4.2, is plotted together with the impedance of a circuit with  $Q=1$ , for comparison. Both circuits have the same  $C$  and  $L$  values, hence the same  $\omega_0$ , but the parallel resistance

of the high-Q circuit is 10 times larger than the other one. Note that  $|Z_p(\omega)|$  is  $R/\sqrt{2} \approx 0.7R$  at  $\omega_1$  and  $\omega_2$ , rather than  $R/2$ .

The ratio of the voltage magnitudes generated across the circuit in Figure 4.1(b) at resonance and at  $\omega_1$  (or  $\omega_2$ ) is,

$$|V(\omega_0)|/|V(\omega_1)| = |Z_p(\omega_0)|/|Z_p(\omega_1)| = \sqrt{2}$$

and

$$20\log|V(\omega_0)/V(\omega_1)| \approx 3 \text{ dB.}$$

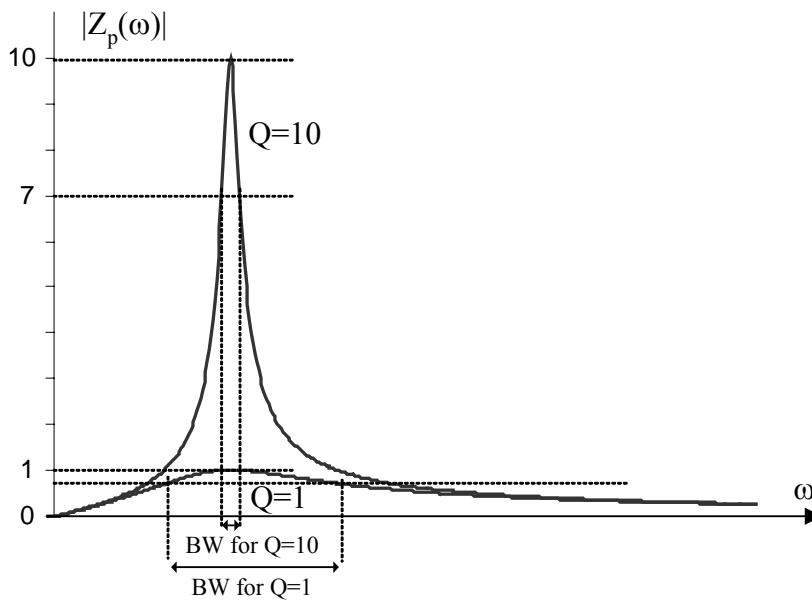


Figure 4.3  $|Z(\omega)_p|$  of two parallel tuned circuits

This is why the bandwidth between  $\omega_1$  and  $\omega_2$  is called 3-dB BW.

TRC-10 employs a parallel tuned circuit in its transmitter. C65 and L1 are the components that form the resonating part. This part of the circuit is given in Figure 4.4(a).

We discuss mixers in detail in Chapter 7. For the moment, we state that the equivalent circuit seen looking into the output pin 5 (or 4) of mixer chip, IC7 (SA602A or SA612A), can be well approximated by a parallel combination of a current source and a resistance of 1.5K. This circuit is depicted in Figure 4.4(b) as Eq.cct.1.

IC7 is TX mixer in Figure 1.11, and the signal at the output is as given in Figure 1.13. We want to eliminate the signal components around  $f_{IF}-f_{VFO}$  and preserve the ones near  $f_{IF} + f_{VFO}$ .  $f_{IF}$  is 16 MHz and  $f_{VFO}$  is between 12 to 13.7 MHz, making  $f_{IF} + f_{VFO}$  anywhere between 28 and 29.7MHz. In order that this tuned circuit preserves the wanted signal, it must have a minimum pass-band that covers 28-29.7 MHz band.

Considering that the 3-dB BW is a reasonable pass-band definition, tuned circuit must have a resonance frequency near 28.85 MHz and a Q less than 17.

Another circuit element is C64, the *d.c. decoupling capacitor*. IC7 is an electronic circuit, which needs to sustain certain d.c. voltages at various points in it in order to function properly. These d.c. levels are not related to the output signal, but must be carefully guarded in order that the chip works properly. When we connect an external circuit, like our tuned circuit, we must take care that we do not disturb the d.c. potential at pin 5. If for example we connect the tuned circuit directly to pin 5, i.e. without C64, inductance L1 will short out the d.c. potential at pin 5, because impedance of L1 at zero frequency is zero. To avoid that, we decouple the d.c. potentials and currents using a series capacitor. The impedance presented by this capacitor to d.c. components is  $\infty$ , hence d.c. voltages on the mixer side is well preserved. C64 is a 10nF capacitor and it has an impedance of  $0.55 \Omega$  at 28.9 MHz. We can approximate it as a short circuit, considering that all other prevailing resistances and reactance in the circuit are much larger.

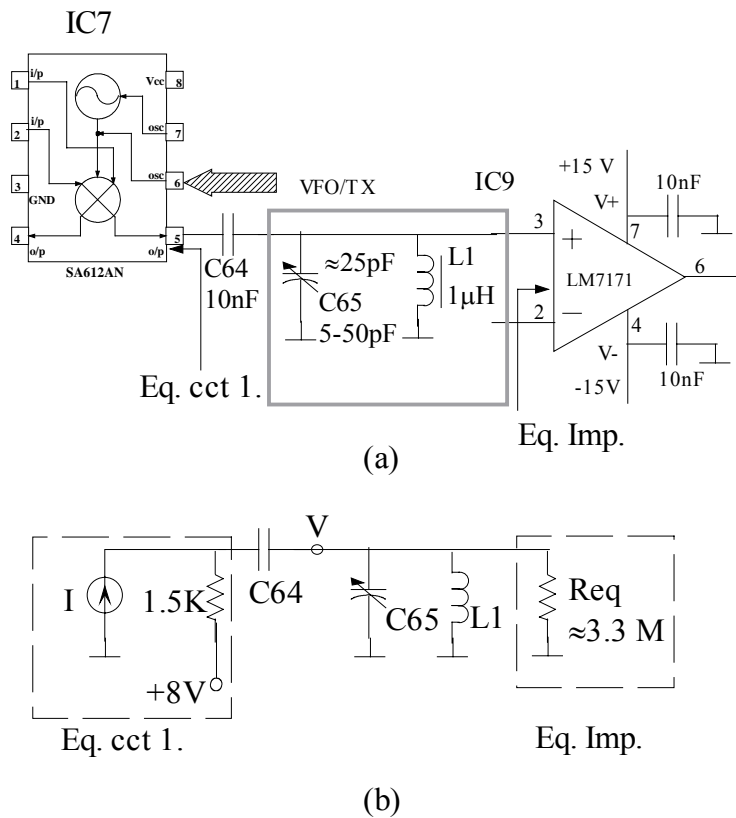


Figure 4.4 TX parallel tuned circuit (a) TX circuit part, (b) Equivalent circuit for analysis

Tuned circuit is connected to the input of an RF OPAMP, LM7171, used in a non-inverting amplifier configuration. We shall see later in this chapter that this amplifier has *differential mode* input impedance,  $R_{eq}$ , of about 3.3 M. This is very large compared to 1.5 K, and since the two resistors appear in parallel, we can ignore the effect of  $R_{eq}$ . Hence the circuit in Figure 4.4(b) reduces to the parallel tuned circuit of Figure 4.1 (b), with C, L, R replaced by C65, L1 and 1.5 K, respectively.

Recall that we require a resonance frequency of 28.85 MHz and a Q less than 17. Q of this circuit is,

$$Q = 1.5 \text{ K}/\omega_0 L = 8.3 < 17.$$

L1 is set to 1 $\mu$ H. To resonate at 28.9 MHz approximately, the total capacitance across it must be about 30 pF. We must expect few picofarad parallel capacitance at mixer output and at OPAMP input, which are called stray capacitances and are due to simple physical distances. In order to set the resonance frequency we must adjust the total parallel capacitance (including stray capacitances) to 30 pF. We do this by adjusting the variable capacitor C65 to the correct value. Adjusting the capacitance to resonate the circuit at the specified frequency is called, *tuning the circuit*, and the capacitor is called the *tuning capacitor*. Quite often resonant circuits are called tuned circuits.

#### 4.2. Series tuned circuits

When a capacitor and an inductor is connected in series, we have a series resonance. This is depicted in Figure 4.5(a). The impedance of this circuit is

$$Z_s(\omega) = 1/j\omega C + j\omega L = (1 - \omega^2 CL)/j\omega C.$$

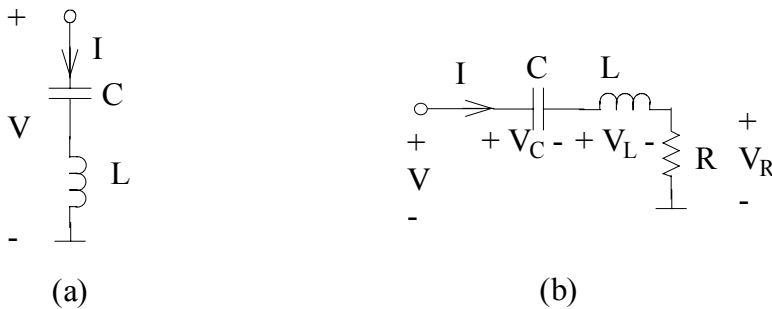


Figure 4.5 Series tuned circuit

$Z_s(\omega)$  is zero when  $1 - \omega^2 CL = 0$  or  $\omega = 1/(LC)^{1/2}$ . This is series resonance and the resonance frequency is  $\omega_0 = 1/(LC)^{1/2}$ , again. When we add a series loss element R into the circuit, we obtain the circuit in Figure 4.5(b). The impedance and admittance of series RLC circuit is,

$$\begin{aligned} Z_s(\omega) &= 1/j\omega C + j\omega L + R \\ &= [(1 - \omega^2 CL) + j\omega RC]/j\omega C \end{aligned}$$

and

$$Y_s(\omega) = 1/Z_s(\omega) = j\omega C/[(1 - \omega^2 CL) + j\omega RC],$$

respectively. The form of the  $Y_s(\omega)$  expression is the same as that of  $Z_p(\omega)$  for parallel RLC circuit, except numerator is changed to  $j\omega C$  and  $j\omega L/R$  is replaced by  $j\omega RC$ , in denominator. Indeed the quality factor of a series circuit can similarly be calculated as

$$\begin{aligned}
 Q &= E_s / E_d \\
 &= \omega_0 L / R \\
 &= 1 / (\omega_0 RC),
 \end{aligned}$$

where  $E_d = I^2 R / (2\omega_0)$ , and  $E_s = I^2 L / 2$ , at resonance frequency. Note that the expressions for  $Q$  are inverse of that of parallel resonance. An important matter to notice is that an increase in the value of  $R$ , increases the  $Q$  of a parallel tuned circuit while it decreases the  $Q$  of a series tuned circuit. This is an expected result, since a large resistance means lower dissipated power relative to stored power in parallel circuits, while it means relatively larger dissipated power in series circuits.

Admittance and impedance of a series circuit can be written, after a little complex algebra, as

$$Y_s(\omega) = 1 / \{R[1 + jQ(\omega/\omega_0 - \omega_0/\omega)]\}$$

and

$$Z_s(\omega) = R[1 + jQ(\omega/\omega_0 - \omega_0/\omega)],$$

respectively. Again, imaginary part of the impedance becomes zero at resonance.

Note that the expression for  $Y_s(\omega)$  is exactly the same as one for  $Z_p(\omega)$ , with  $R$  replaced by  $1/R$ . The variation of  $Y_s(\omega)$  with respect to  $\omega$  is similar to that of  $Z_p(\omega)$ , given in Figure 4.2 and Figure 4.3. The variation of  $Z_s(\omega) = R_s(\omega) + j X_s(\omega) = R + jRQ(\omega/\omega_0 - \omega_0/\omega)$  with respect to  $\omega$  is given in Figure 4.6.

The relation between  $\omega_1$ ,  $\omega_2$  and  $\omega_0$  is similar to the parallel case, and is shown in Figure 4.6.

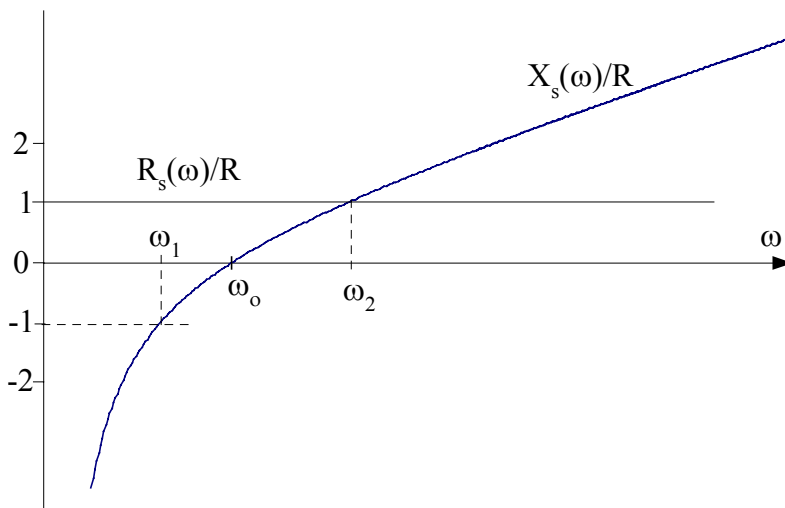


Figure 4.6 Real and imaginary parts of  $Z_s(\omega)$  normalized to series resistance

### 4.3. Equivalence of series and parallel RLC circuits



Quite often we use circuits, which do not look like the parallel or series tuned circuit morphologies we discussed above. One very common deviation is given in Figure 4.7. The inductors usually possess certain loss, which can be modeled by a series resistor. This kind of circuit is called the *tank circuit*. As far as resonance is concerned, this circuit can be viewed as a series resonance circuit containing series connected C, L<sub>S</sub> and R<sub>S</sub>. However we are interested in what appears across the capacitor terminals. The admittance of the tank circuit is

$$Y_T(\omega) = j\omega C + (j\omega L_S + R_S)^{-1}$$

$$= R / (\omega^2 L_S^2 + R_S^2) + j\omega(R^2 C - L + \omega^2 L^2 C) / (\omega L_S^2 + R_S^2).$$

The imaginary part must be zero at resonance:

$$R_S^2 C - L_S + \omega^2 L_S^2 C = 0.$$

This condition yields

$$\omega_0 = [1/L_S C - (R_S/L_S)^2]^{1/2}.$$

Note that R/L is  $\omega_0/Q$  for the series RLC circuit, i.e.

$$\omega_0 = [(1/L_S C)(1 - 1/Q^2)]^{1/2}.$$

If the quality of resonance is sufficiently high, resonance frequency is similar to that of the series R<sub>S</sub>L<sub>S</sub>C circuit. For example, for a quality factor of 3.5 resonance frequency is within 4% of  $1/(L_S C)^{1/2}$ .

This circuit can be modeled by an equivalent parallel RLC circuit over a certain frequency band. The series inductive branch impedance must be equal to the parallel L<sub>P</sub>R<sub>P</sub> section impedance at resonance frequency, for equivalence:

$$j\omega_0 L_S + R_S = (1/j\omega_0 L_P + 1/R_P)^{-1} = j\omega_0 L_P R_P / [R_P + j\omega_0 L_P].$$



Figure 4.7 Tuned circuit with lossy inductor

The equivalence relation for real and imaginary parts are

$$R_S = \text{Re} \{ j\omega_0 L_P R_P / [R_P + j\omega_0 L_P] \} = (\omega_0 L_P)^2 R_P / [R_P^2 + (\omega_0 L_P)^2] = R_P / [R_P^2 / (\omega_0 L_P)^2 + 1]$$

$$j\omega_0 L_S = \text{Im} \{ j\omega_0 L_P R_P / [R_P + j\omega_0 L_P] \} = j\omega_0 L_P R_P^2 / [R_P^2 + (\omega_0 L_P)^2]$$

$$= j\omega_0 L_P / [1 + (\omega_0 L_P)^2 / R_P^2]$$

respectively.

Then

$$R_p = R_s(Q^2 + 1)$$

and

$$L_p = L_s(Q^2 + 1)/Q^2,$$

where  $Q = R_p/(\omega_0 L_p)$  is the quality factor of the parallel circuit. This equivalence also maintains that the  $Q$ 's of two circuits are also equal:

$$R_p/\omega_0 L_p = \omega_0 L_s/R_s = Q$$

The equivalence holds at frequencies near the resonance frequency.

We use few series RLC circuits in TRC-10. One of them is the one formed by R58, L2, and C69, shown in Figure 4.8(a). This piece of circuit is the BPF between the TX preamplifier and TX output amplifier.

We must construct an equivalent circuit modeling the IC9 output and the IC10 input, again, in order to analyze the circuit. IC9 and IC10 are both LM7171 OPAMPs. IC10 is configured as a non-inverting amplifier, and its input impedance at pin 3 is  $R_{eq} = 3.3M$ . The output impedance of LM7171 is approximately  $15\Omega$ . With these values we obtain the equivalent circuit in Figure 4.8(b). The output of preamplifier (IC9) is modeled as a voltage source and a series resistance  $R_o = 15\Omega$ , where the input of IC10 is modeled by  $R_{eq}$ .

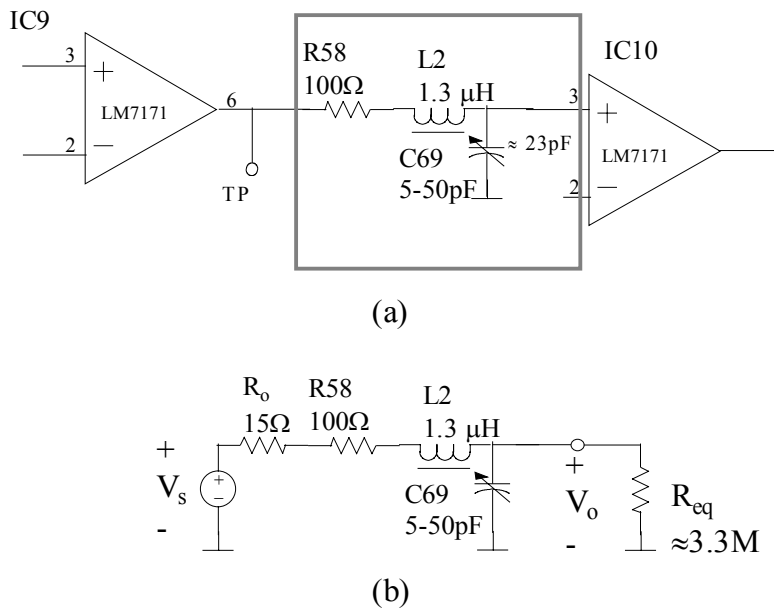


Figure 4.8 TX series RLC BPF, (a) Circuit diagram, and (b) Equivalent circuit

Assuming that the quality factor of this circuit is sufficiently high, the resonance frequency is approximately  $(1/L2C69)^{1/2}$ . L2 and C69 must resonate at 28.9MHz, hence

$$C69 = 1/\omega_0^2 L2 \approx 23 \text{ pF.}$$

The total series resistance  $R_S$  and  $Q$  of the series circuit are

$$R_S = R_o + R58 = 115\Omega,$$

and

$$Q = \omega_0 L2 / R_S \approx 2, \text{ respectively.}$$

We need to know the magnitude of the signal applied to the amplifier input. This is the voltage across C69. The parallel equivalent resistance of  $R_S$  in the tank circuit ( $V_s$  *killed*) is  $R_S(Q^2+1)$ , or 575  $\Omega$  approximately. This resistance is in parallel with  $R_{eq}$ , and together they yield  $R_S(Q^2+1) // R_{eq} \approx 575 \Omega$ ! The input impedance of IC10,  $R_{eq}$ , behaves like an open circuit compared to  $R_S(Q^2+1)$ , and we can *ignore* it.

Then the current in the circuit is

$$I = V_s / (R_o + R58)$$

at resonance and,

$$\begin{aligned} V_o &= I / (j\omega_0 C69) \\ &= V_s / [j\omega_0 C69 (R_o + R58)] \\ &= -j2 V_s. \end{aligned}$$

The voltage magnitude across the capacitor is twice the magnitude of the input voltage. The input voltage is effectively amplified, at the expense of increased equivalent resistance.

This is a very important property of series tuned circuits. Indeed, capacitor or inductor branch voltage magnitudes are always  $Q$  times the input voltage in series RLC circuits, similar to the branch currents in parallel RLC circuits.

#### 4.4. Real inductors

Inductors store electrical energy in a *magnetic field*. Magnetic field consists of lines of magnetic force or *flux*. Any conductor carrying a current produces a magnetic field. If a conductor is wound into a *solenoid*, as shown in Figure 4.9, the flux is intensified along the axis of the solenoid.

Flux is denoted by  $\phi(t)$  and its phasor by  $\Phi$ . Flux generated by a single loop is given by

$$\Phi_l = A_L I$$

and by a solenoid of  $N$  turns is given by

$$\Phi = N\Phi_l = NA_L I$$

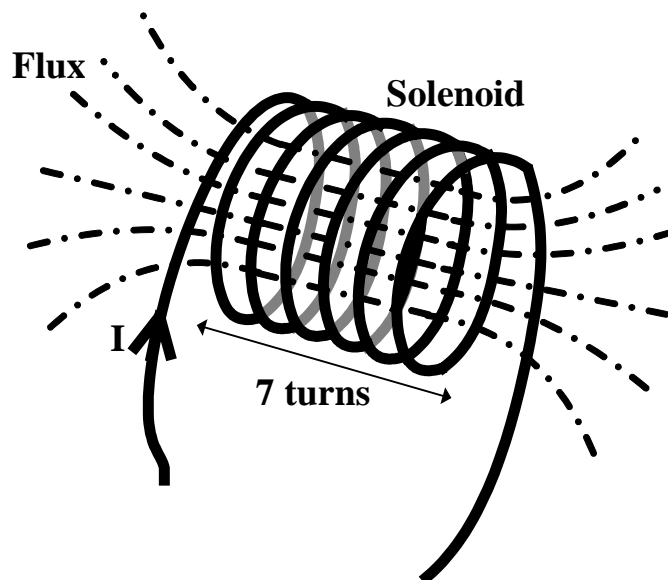


Figure 4.9 Inductor with 7-turn solenoidal wound conductor.

where  $I$  is the current carried by the conductor and  $A_L$  is the *inductance constant*. The voltage generated across *one loop*,  $v_1(t)$  and its phasor  $V_1$ , by this flux are

$$v_1(t) = d\phi/dt$$

and

$$V_1 = j\omega\Phi,$$

respectively.

There are  $N$  loops in the solenoid and they are all in series. We must add all single loop voltages up to obtain the voltage,  $V$ , across the solenoid:

$$\begin{aligned} V &= NV_1 \\ &= j\omega N\Phi \\ &= j\omega N^2 A_L I \end{aligned}$$

The inductance of a solenoid is, therefore,

$$L = N^2 A_L.$$

The inductance constant  $A_L$  depends on the size (e.g. diameter) of the loops and the type of the *core*, material on which the conductor is wound. Typical cores used for making inductors and transformers are:

- a. air,
- b. stacks of laminated steel sheets,
- c. various *ferrite* compounds (cores shaped as rods, beads, toroids, many other forms),
- d. powdered iron based ceramics (similar to ferrites but for higher frequencies).

The laminated steel core is used for mains transformers and audio applications. The core of the mains transformer we used for power supply is laminated steel type. Other three types are used for higher frequencies.

#### 4.4.1. Air core inductors

L6 and L7 in TRC-10 are air-core inductors. The easy way of designing an air-core inductor is to use the following formula, which gives the inductance value in terms of physical dimensions:

$$L (\mu\text{H}) = d^2 N^2 / (46d + 102\ell)$$

where  $L$  is the inductance value in  $\mu\text{H}$ ,  $d$  is coil diameter in cm,  $N$  is number of turns and  $\ell$  is the length of the coil in cm. This formula is quite accurate for coils having an aspect ratio  $\ell/d$  greater than 0.4.

Let us try to find the number of turns and the dimensions for a  $0.33 \mu\text{H}$  inductor. There are many choices for dimensions. Let us choose a coil geometry such that the length of the coil is equal to its diameter, i.e.,  $\ell = d$ . Then, for  $0.33 \mu\text{H}$  inductor,

$$N^2 d = 48.84 \text{ cm-turns}^2.$$

If we wish to have an inductor of length (and diameter in this case) 5mm, then number of turns becomes,  
 $N \approx 10$  turns.

Once it is wound, the value of an air-core inductor can be tuned within about 20%, by extending its length. As the length is increased, inductance falls. Inductance of a coil increases, on the other hand, as the diameter increases or as the number of turns increases.

#### 4.4.2. Powdered iron core inductors

Another inductance in TRC-10 is L1. We use an iron powder core of toroidal shape for this inductor. Such an inductor is shown in Figure 4.10. The core is Micrometals T37-7. Micrometals is the company producing the core, “T” stands for toroid, “37” tells that the outer diameter of the toroid is 0.37 inch and “7” tells that the core is made of mix-7, *Carbonyl-TH*. The data sheet of this core is given in Appendix D.

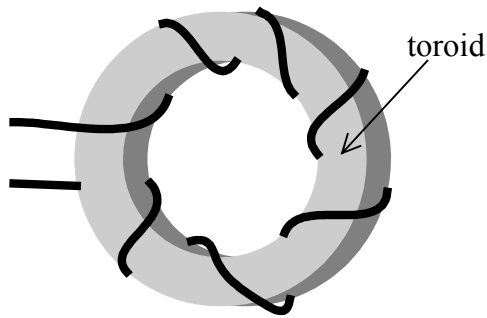


Figure 4.10 Inductor made of wire wound on a toroid

$A_L$  of this core is  $3.2 \text{ nH/turns}^2$ . We can calculate the number of turns necessary for  $L=1 \text{ }\mu\text{H}$ , directly as,

$$\begin{aligned} N^2 &= L/A_L \\ &= 312.5 \quad \Rightarrow \quad N \approx 18 \text{ turns.} \end{aligned}$$

The cores are used to obtain high inductance values with smaller turn diameters and smaller number of turns. This ability of the core material is determined by its *permeability*,  $\mu$ . The permeability of air (actually free space) is  $\mu_0 = 4\pi \times 10^{-7} \text{ H/m}$ . The permeability,  $\mu$ , of other materials are usually given in terms of a relative permeability,  $\mu_r$ , relative to that of air,  $\mu = \mu_r \mu_0$ . For paramagnetic materials suitable for use as cores,  $\mu_r$  is always larger than one, indeed much larger. Relative permeability of steel is about 5000. Relative permeability of ferrite ranges between 50 and few thousand and of iron powder materials, ranges between about 10 and 50.

As  $\mu_r$  increases we can get larger flux, and hence larger inductance, in a single turn. The overall size of an inductor decreases as the permeability of the material increases.

Cores confine the magnetic flux into the core volume, as much as possible. Their function is similar to the yoke in loudspeaker. The effectiveness of a core material in this respect is, again, proportional to the materials permeability.

#### 4.4.3. Core losses

Magnetic flux experiences loss in materials. The choice of a particular core material always depends on the amount of loss at the frequency range the inductor is used. Laminated steel sheet cores are useful only at audio frequencies. At higher frequencies, their loss becomes excessive. Ferrite materials are usable up to lower HF range, and iron powder cores can be used in applications up to VHF range. The loss at higher frequencies increase in materials of high permeability. The mechanism of this kind of power loss are discussed in texts on electromagnetism extensively. We shall confine our discussion to its effect in choice of core materials and in the design and modelling of real inductors. There is a trade-off between high  $\mu_r$  and low loss.

If there were not any loss in an inductor, its model would be an inductance only. The loss in the core introduces a resistance into the model. In inductor models, this resistance is usually included into the model in series with the inductance. Since the

loss of a core varies with the frequency, this resistance in the model has different values at different frequencies. *As a matter of fact, this loss resistance is zero for d.c.* Manufacturers usually provide loss data for materials in a variety of ways. One common way is *Q-graphs* with respect to frequency.

Q vs. frequency graph of T37-7 is given in Appendix D. This graph tells that an inductor made by winding 16 turns on T37-7 has an inductance of  $0.9 \mu\text{H}$  with a Q of 190, at approximately 12 MHz. This value of Q is maximum and it decreases at other frequencies. This means that if we form a series resonance circuit only with this inductor and a capacitor of value which sets the resonance frequency to 12 MHz (i.e. without any other resistance in the circuit), the series loss due to core power losses will limit the Q at 190. From this Q value, we can figure out that equivalent loss resistance is:

$$R_c = \omega L / Q = 0.36 \Omega \text{ at } 12 \text{ MHz.}$$

We use the same core with 18 turns for L1 at 29 MHz. The Q at this frequency is approximately 80. Hence equivalent resistance becomes  $2.3 \Omega$  at 29 MHz. But we use L1 in a parallel resonance circuit with C65. Using the parallel-series equivalence, we can find the equivalent parallel core loss resistance as

$$\begin{aligned} R_{cp} &= R_c(Q^2 + 1) \\ &= (2.3 \Omega) \times (80^2 + 1) \\ &= 14.7 \text{ K}\Omega. \end{aligned}$$

The modeling and conversion discussed above is shown in Figure 4.11.

Now, this loss resistance,  $R_{cp} = 14.7 \text{ K}\Omega$ , appears in parallel with  $1.5 \text{ K}$  IC7 output impedance and  $R_{eq} = 3.3 \text{ M}$  in Figure 4.4(b). Hence the total parallel resistance of the circuit is combination of these three resistances, which is  $1.36 \text{ K}$ . The effect of core losses of the inductor is a slight decrease in total parallel resistance. This also decreases the overall Q from 8.3 to 7.5, which is acceptable.

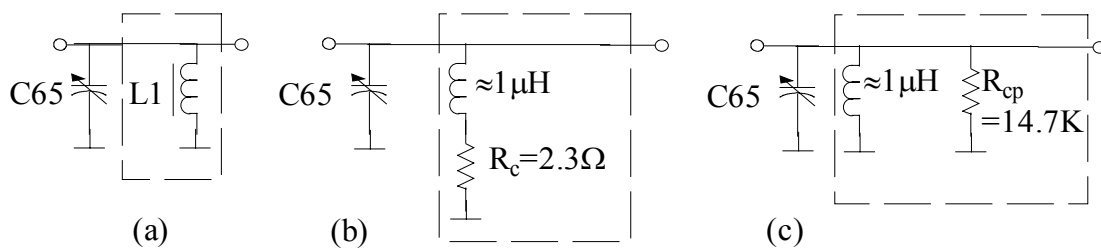


Figure 4.11 Core loss in L1(a) Parallel LC65 circuit, (b) Equivalent circuit of L1 at 29MHz, (c) Parallel equivalent RLC circuit

#### 4.4.4. Copper losses

The other type of loss in inductors is *copper loss* or *winding resistance*. This loss is due to the finite conductivity of the wire used in the winding. This loss resistance is the resistance of the wire, i.e.  $\rho l / A$ , at low frequencies, where  $\rho$  is the resistivity of the wire material,  $l$  is the length and  $A$  is the cross section of the wire. This loss is

further aggravated at RF because of a phenomenon called *skin effect*. As the frequency increases, the current is no longer homogeneously distributed across the cross section of the conductor, but it is confined to a thin cylindrical layer next to the conductor surface. Hence the cross section of the wire is effectively decreased, resulting in a larger winding resistance. The approximate cut-off frequency, above which skin effect becomes significant is given as,

$$f_{sc} \approx 0.08/d^2$$

where  $f_{sc}$  is frequency in MHz and  $d$  is the diameter of the wire in mm. The resistance of the wire,  $R$ , at a frequency  $f$  above  $f_{sc}$ , is roughly

$$R \approx R_{dc}(f/f_{sc})^{1/2}$$

where  $R_{dc}$  is the d.c. resistance of the wire. This corresponds to ten times increase in the resistance for an increase of two decades in frequency, above  $f_{sc}$ .

For example, a copper wire of 1 mm diameter has a d.c. resistance of 0.0215  $\Omega$ /m. If we use a length of 10 cm to make the winding, the total d.c. resistance becomes  $0.1\text{m} \times 0.0215 \text{ } \Omega/\text{m} = 2.15\text{E-}3 \text{ } \Omega$ , or 2.15 m $\Omega$ . The  $f_{sc}$  for this wire is 8E-2 MHz, or 80 KHz. Then, at 29 MHz,  $R$  becomes 41 m $\Omega$ .

The winding resistance is also in series with the inductance, in inductor model. Hence it is possible to include one loss resistance to the model, to represent the sum of core and winding losses. It is common practice to choose sufficiently thick wires in inductors, to make the winding resistance negligible compared to inductance reactance and other losses.

#### 4.4.5. RF choke inductors

While core loss degrades the performance of an inductor used in resonant circuits, it provides useful properties for wide band applications. A particularly important application area is the *Electromagnetic Compatibility (EMC)/Electromagnetic Interference (EMI)* engineering. The interference of RF signals within the same instrument or between instruments must be avoided in electronic circuits. Voltages or currents can couple to other parts of the circuits where they are not wanted, by electromagnetic means. Interference of irrelevant signal in a circuit can cause an instrument to malfunction. Studies related to keep this effect under control is called EMC/EMI.

A typical RF interference to the power supply line is shown in Figure 4.12(a).



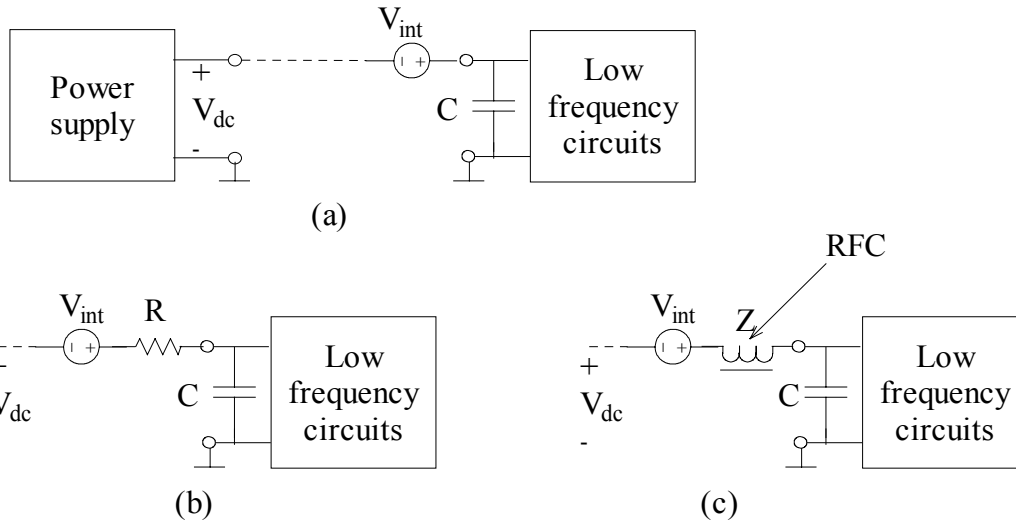


Figure 4.12 (a) Model for RF interference, (b) RC filter to decrease interference, (c) resistance replaced by RFC

A circuit is assumed in this model where a LF circuit is fed by a d.c. power supply, similar to the audio circuits in TRC-10. There is a by-pass capacitor,  $C$ , at the supply terminal of LF circuit. HF signal couples to the supply line between the power supply and the circuit electromagnetically, producing an interference signal  $V_{int}$ . The LF circuit expects to have  $V_{dc}$  only at its terminals, but it experiences  $V_{dc} + V_{int}$ .

One way of decreasing  $V_{int}$  is to include a series resistance to form an RC LPF, as in Figure 4.12 (b). The d.c. current drawn by the LF circuit will cause a d.c. voltage drop on  $R$  in this case. The d.c. voltage that will appear at the circuit terminals is less than  $V_{dc}$ , by an amount determined by  $R$  and the current drawn by the circuit. This is not always agreeable, particularly in circuits where the d.c. current demand is high or varies considerably, like the audio power amplifier in TRC-10.

A better solution is to use an inductor instead of a resistor, as shown in Figure 4.12(c). All we need in this inductor is that it must exhibit a high impedance at the frequency of  $V_{int}$ , and a low impedance (preferably zero impedance) at d.c. Therefore it need not be a high Q inductor. Such an inductor is called *radio frequency choke* (RFC).

A common way of making RFC is to wind few turns on a *ferrite core* with high permeability. The inductor behaves like a simple inductance at low frequencies, and its impedance is zero at d.c., since core losses of the ferrite are not effective, i.e.

$$Z(\omega) \Big|_{\omega=0} \approx 0.$$

The d.c. supply voltage,  $V_{dc}$ , appears at the terminals of LF circuit.

The impedance of the RFC at HF, on the other hand, is obviously

$$Z(\omega) \Big|_{\omega=HF} = R_c + j\omega L,$$

where  $R_c$  is the core loss resistance. Hence having a large core loss is preferable for this application. The effect of  $V_{int}$  at circuit terminals is what is left after the voltage division between  $Z$  and  $C$ :

$$V_{int}/(1 - \omega^2 LC + j\omega R_c C).$$

We use few RFCs in TRC-10. We use a hollow cylindrical core with a height of 5 mm, and with inner and outer diameters of 2 mm and 4 mm, respectively. Such cores are called *beads*. Neosid manufactures this particular bead, and the ferrite material is called F19. The specifications of this bead tells that a single turn on this core yields an impedance of  $|Z| = 27 \Omega$  at 25 MHz. Indeed 18 turns yield an impedance of 8 K at 29 MHz and this impedance is predominantly resistive (i.e.  $R_c$  dominates over reactive part). Hence if we use this RFC with a by-pass capacitor of 10 nF,  $V_{int}$  is attenuated by 1/15000 (or by 83 dB) approximately, while  $V_{dc}$  is maintained intact. The voltage at the terminals of LF circuit becomes approximately

$$V_{dc} + V_{int}/15000.$$

#### 4.5. RF Amplification using OPAMPs

We employed OPAMPs for amplification of audio signals in Chapter 3. There we implicitly assumed that the *open loop gain*,  $A$ , of the OPAMP is constant for all frequencies. If we examine the “Open Loop Frequency Response” graph in the data sheet of TL082, where gain is plotted versus frequency, we observe that the gain is constant upto about only 20 Hz. Above this frequency gain falls at 20 dB/decade as the frequency increases.

##### 4.5.1. OPAMP Open Loop Frequency Response

As a matter of fact, OPAMP gain is not constant, but it is a complex function of frequency, called *open loop frequency response*, which can be expressed as

$$A_{ol}(\omega) = A/(1 + j\omega/\omega_o)$$

where  $A_{ol}$  is the open loop gain,  $A$  is the open loop gain at d.c. (i.e.  $3.0E5$ , the value we used in Chapter 3), and  $\omega_o$  is the 3-dB frequency or *break frequency* where gain starts falling, 20 Hz for TL082.

The assumption of constant gain for all frequencies did not cause any errors in the analysis of audio circuits, because audio circuits are processing signals at few KHz. At those audio frequencies, the data sheet shows that  $|A_{ol}(\omega)|$  is about 70 dB, or 3000. We used the TL082 in a feedback configuration to deliver gains of about 50, and the assumption of  $A_{ol}(\omega) \rightarrow \infty$  was acceptable.

Should we try to use TL082 for amplifying signals at HF, we must first check the suitability of its frequency response. The same graph tells us that TL082 can provide a gain of 0 dB, or 1, at about 3 MHz. This means that it is not acting like an amplifier at HF.

We use high frequency OPAMPs, like LM7171, for amplification at HF. The data sheet of LM7171 is given in Appendix A. We can see that  $A = 85$  dB,  $|A_{ol}(\omega)| = 1$  at about 200 MHz and  $|A_{ol}(\omega)| \approx 20$  dB, or 10, at 30 MHz. The open-loop phase shift between input and output is often reported in OPAMP data sheets indirectly as *phase margin*. Phase margin is an important concept related to stability considerations, discussion of which is left to more advanced texts. The open-loop phase shift information at any frequency can be obtained from the phase margin data by subtracting  $180^\circ$ . The phase margin at 30 MHz is approximately  $85^\circ$  and hence the open-loop phase shift between the input and output is  $-95^\circ$ . This OPAMP is suitable for amplification at HF.

A single break frequency can cause a phase shift up to  $90^\circ$ . Other capacitive effects in the IC become significant at higher frequencies in OPAMPs, as in the case of LM7171. This can be modeled by a second break frequency as

$$A_{ol}(\omega) = A / [(1 + j\omega/\omega_{o1}) \times (1 + j\omega/\omega_{o2})].$$

Using the above data, we can express the open loop gain at 30 MHz as

$$A_{ol}(\omega) \Big|_{\omega=2\pi \times 30 \text{ E}6} = A / [(1 + j\omega/\omega_{o1}) \times (1 + j\omega/\omega_{o2})] \Big|_{\omega=2\pi \times 30 \text{ E}6} \approx 10 \angle -95^\circ.$$

We cannot see the first 3-dB frequency in the data sheet and therefore we cannot calculate the first break frequency. It is obvious that 30 MHz is much higher than the first break frequency, since the phase margin plot has reached a plateau long before frequency reaches 30 MHz. Mathematically this corresponds to  $(1 + j\omega/\omega_{o1}) \approx j\omega/\omega_{o1}$ . Therefore we can consider that the  $-90^\circ$  of the phase shift is due to first break frequency and  $-5^\circ$  is because of the second break frequency, i.e.

$$A_{ol}(\omega) \Big|_{\omega=2\pi \times 30 \text{ E}6} \approx A / [(j\omega/\omega_{o1}) \times (1 + j\omega/\omega_{o2})] \Big|_{\omega=2\pi \times 30 \text{ E}6} \approx 10 \angle -90^\circ \angle -5^\circ.$$

for LM7171. Hence the second break frequency,  $\omega_{o2}$ , can be found from

$$\omega/\omega_{o2} \Big|_{\omega=2\pi \times 30 \text{ E}6} = \tan(5^\circ)$$

as  $2.16 \text{ E} 9$  rad/sec, or 343 MHz, approximately.

Consider the TX preamplifier circuit of TRC-10, shown in Figure 4.13. This amplifier stage is a non-inverting amplifier and its input is the voltage developed across the parallel tuned circuit of C65 and L1.

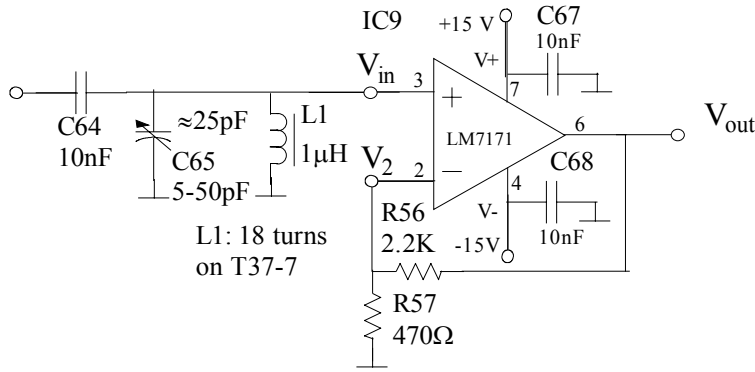


Figure 4.13 TX preamplifier

C67 and C68 are supply by-pass capacitors. Feedback of the amplifier is provided by R56 and R57. Now, if we carry out the analysis for non-inverting amplifier configuration, we obtain

$$V_2 = V_{out} \times R57 / (R56 + R57),$$

$$\begin{aligned} V_{out} &= A_{ol}(\omega)(V_{in} - V_2) \\ &= A_{ol}(\omega)[V_{in} - V_{out} \times R57 / (R56 + R57)]. \end{aligned}$$

Here we ignored the input impedance,  $R_{in}$ , of LM7171.  $R_{in}$  of LM7171 is (from the data sheet) is  $3.3 \text{ M}\Omega$ , which is very large compared to R56 and R57. The output impedance of the OPAMP is also ignored. From the data sheet, we can see that  $R_o$  is  $15 \Omega$  and compared to R56,  $R_o$  can be assumed zero without any loss in accuracy.

Solving for  $V_{out}$ , we get

$$V_{out} = V_{in} \times A_{ol}(\omega) / [1 + A_{ol}(\omega) \times R57 / (R56 + R57)].$$

The gain of the amplifier at 30 MHz can be obtained numerically by substituting the values of  $A_{ol}(\omega)$ , R56 and R57, as

$$\begin{aligned} A_{ol}(\omega) / [1 + A_{ol}(\omega) \times R57 / (R56 + R57)] \Big|_{\omega = 2\pi \times 30 \text{ E}6} &\approx (10 \angle -95^\circ) / [1 + (10 \angle -95^\circ) \times 0.176] \\ &= (10 \angle -95^\circ) / (0.847 - j1.753) = (10 \angle -95^\circ) / (1.95 \angle -64^\circ) \\ &\approx 5.1 \angle -31^\circ. \end{aligned}$$

Thus the overall gain of the non-inverting amplifier is 5.1 with a phase shift of  $-31^\circ$  between the input and the output.

Voltage division between R56 and R57 yields 0.176. Hence  $A_{ol}(\omega) \times R57 / (R56 + R57)$  in the denominator is  $1.95 \angle -64^\circ$ . Should we made the usual assumption (which is incorrect in this case) that

$$|1.95 \angle -64^\circ| \gg 1$$

in the gain expression, we could ignore the effect of “1” in  $[1+A_{oi}(\omega)\times R_{57}/(R_{56}+R_{57})]$  and obtain the gain approximately as  $1+R_{56}/R_{57}\approx 5.7$ , without any phase shift. Note that even with an open loop gain as low as 10, the gain we obtain is reasonably close to the gain obtained with ideal OPAMP assumption.

#### 4.5.2. Voltage gain and power gain

We calculated that the voltage gain of TX preamplifier stage as 5.1, or 14.2 dB. The power gain of this stage can also be calculated. Referring to Figure 4.4(b), the input voltage and power delivered to this stage are

$$V_{in} = I(1.5K)$$

and

$$P_{in} = \frac{1}{2} I^2(1.5K),$$

respectively, where 1.5 K is the output resistance of the mixer chip IC7. Since the gain of the amplifier is 5.1,  $V_{out}$  is  $(5.1)\times I(1.5K)$ . Referring to Figure 4.8(b), the load connected to the output of the amplifier at resonance, 30 MHz, is 100  $\Omega$ . The power delivered to the load is the current through the load resistor times the voltage across it,

$$\begin{aligned} P_{out} &= \frac{1}{2} [(V_{out})/(R_o+100\Omega)][(V_{out})\times(100\ \Omega)/(R_o+100\Omega)] \\ &= \frac{1}{2} V_{out}^2 [100/(15+100)^2]. \end{aligned}$$

Substituting the value of  $V_{out}$ , we obtain

$$P_{out} = \frac{1}{2} [(5.1)\times I(1.5K)]^2 [7.56E-3].$$

Hence the power gain is

$$P_{out}/P_{in} = (5.1)^2 \times (1.5K) \times (7.56E-3) = 295,$$

or 24.7 dB. The power gain is considerably larger compared to the voltage gain. This is because of the fact that, when the amplifier processes the signal, it is not only amplified, but the impedance that the signal appears across, decreases from 1.5 K to 100  $\Omega$ . There is an impedance transformation. As a result of this impedance transformation, we obtain a power gain, which is 10.5 dB larger than the voltage gain.

#### 4.5.3. Slew Rate

As far as the signal amplitudes are small, open loop transfer function is the limiting factor for output voltage amplitude. When the signal amplitudes become larger, another limitation due to *slew rate* of the OPAMP emerges. Slew rate is the maximum slope of the output voltage that can be amplified by the OPAMP without any distortion. Assume that the signal is a sinusoidal voltage  $v(t)=V_p\cos\omega t$ . Then the maximum slope of this signal is

$$\left| dv(t)/dt \right|_{max} = V_p\omega.$$

The product  $V_p\omega$  must not exceed the slew rate limit of the OPAMP in order to have amplification without any distortion. We can see in the data sheet that LM7171 has a very high slew rate of  $4100\text{V}/\mu\text{s}$  at unity gain. This means that, for example at 30 MHz,

$$V_p < 4100 (\text{V}/\mu\text{s}) / (2\pi \times 30\text{E}6 \text{ rad/sec}) = 21.75 \text{ V},$$

which is significantly larger than our requirement (notice that the supply voltage is  $\pm 15 \text{ V}$ , and hence output voltage amplitude,  $V_p$ , must be less than  $15 \text{ V}$  anyway).

#### 4.5.4. Input Bias Current

In order that an OPAMP can operate, it draws certain amount of d.c. current into its input pins from the external circuits. This current is called *input bias current*. The magnitude of this current is usually small, but must be carefully taken care of in every application. Circuits connected to both positive and negative input pins (i.e. pins 3 and 2 respectively in case of LM7171) must allow a d.c. current path to a supply or to ground, so that this bias current can flow and OPAMP can work. For example there must not be a series capacitor, blocking all the d.c. current. In the TX preamplifier in Figure 4.13, d.c. path for pin 3 is provided by the inductance L1 and for pin 2, by R56 and R57.

Input bias current of LM7171 is given as  $12 \mu\text{A}$  maximum ( $I_B$  in the data sheet). TL082 is an “FET input OPAMP” and has a very low input bias current of  $50 \text{ pA}$ .

#### 4.6. Maximum power transfer

Consider the amplifier circuit in Figure 4.14(a). An amplifier equivalent circuit is given in this figure, where the output is not loaded, i.e. the output is open. Here  $V_{in}$  is input voltage phasor. There is no output current and therefore no output power dissipation. If we connect a load resistor to the output terminal, as in Figure 4.14(b), an output current,  $I_o = AV_{in} / (R_1 + R_L)$ , will flow. The power delivered to the load is

$$P_L = \frac{1}{2} V_{out} I_o \\ = \frac{1}{2} (AV_{in})^2 R_L / (R_1 + R_L)^2.$$

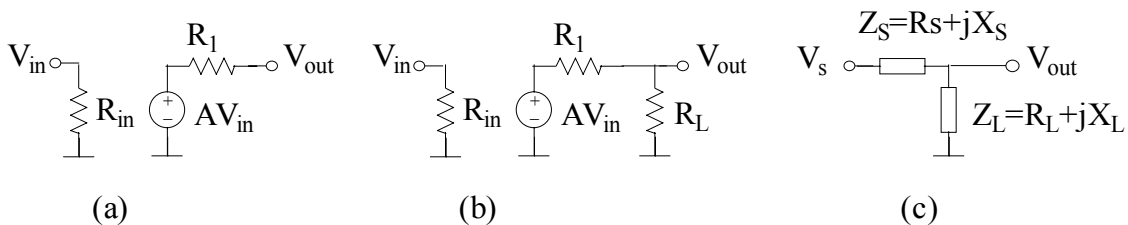


Figure 4.14 Maximum power transfer

Usually the maximum power that an amplifier can deliver to a load impedance is limited by few factors. For example, the output voltage amplitude is limited by the power supply level and slew rate, in OPAMPs. Maximum output amplitude is limited to about  $9 \text{ V}$  in LM7171, before slew rate, when operated from  $\pm 15 \text{ V}$  supply (“Output Swing” specification in the data sheet).

On the other hand, maximum output current is also limited in OPAMPs. For example, maximum output current amplitude is limited to about 90 mA in LM7171. Hence, the maximum power we can get from an amplifier using LM7171 is 0.4 W at best. If we further examine the data sheet, for example from undistorted output point of view, other limitations decrease this maximum power level to a significantly lower one at high frequencies.

Thus, the available output power is limited. Occasionally we find ourselves in a situation where we wish to deliver as much power as possible to the load. If we carefully examine the expression for  $P_L$  above, we can see that  $\frac{1}{2} (AV_{in})^2 / (R_1 + R_L)$  is the limited available power and we can only maximize  $P_L$  by good choice of resistances. To find the value of  $R_L$  which yields maximum  $P_L$ , we differentiate the expression for  $P_L$  with respect to  $R_L$  and equate it to zero:

$$dP_L/dR_L = 0 \quad \Rightarrow \quad R_L = R_1.$$

Hence *load resistance must be equal to source resistance for maximum power transfer* to the load.

Both source and load impedances can be complex. This case is depicted in Figure 4.14(c). The complex power delivered to the load in this case is

$$P_L = \frac{1}{2} [V_s Z_L / (Z_S + Z_L)] I_s^*,$$

Where  $I_s$  is the current flowing through  $Z_L$ . Substituting

$$I_s^* = [V_s / (Z_S + Z_L)]^*$$

we obtain

$$P_L = \frac{1}{2} (V_s V_s^*) Z_L / |Z_S + Z_L|^2.$$

The complex power  $P_L$  has two components. The real part is the average power and it is dissipated on  $R_L$ . The imaginary part is the reactive power and it is not dissipated. This component is the power that goes to the capacitors and/or inductors. However the reactive power must be available to the circuit in order that the circuit can function. There is no point to maximize the reactive part since this component is only temporarily borrowed by  $Z_L$ . The component to be maximized is the real part. The real power delivered to the load is the average power

$$P_{La} = \text{Re}[P_L] = \frac{1}{2} (V_s)^2 R_L / [(R_L + R_S)^2 + (X_L + X_S)^2].$$

First of all it is clear that  $X_L$  and  $X_S$  must sum up to zero to maximize  $P_L$ , i.e.  $X_L = -X_S$ . With this condition imposed, the rest of the expression reduces to the all-resistive case, which yields  $R_L = R_S$  for maximum  $P_L$ . Hence the general form of maximum power transfer condition is

$$Z_L = Z_S^*.$$

In other words, *load impedance must be equal to the complex conjugate of the source impedance* for maximum power transfer to load. The maximum power that can be delivered to load by a source, which is also called *available power*, is

$$P_L = P_A = V_s^2 / 8R_S$$

where  $V_{in}$  is the source voltage amplitude and  $R_S$  is the real part of source impedance.

#### 4.7. Bibliography

*The ARRL Handbook* has a very good chapter on real world components (Chapter 10).

Following web sites provide a wealth of information on various core materials and their applications:

[www.ferroxcube.com](http://www.ferroxcube.com)

[www.micrometals.com](http://www.micrometals.com)

[www.neosidcanada.com](http://www.neosidcanada.com)

<http://bytemark.com/amidon>

*Op-amps and Linear Integrated Circuits* by R.A.Gayakwad has a good chapter on OPAMP frequency response.

#### 4.8. Laboratory Exercises

1. Read all Laboratory Exercises of this chapter carefully.

##### ***Radio amplifiers***

2. The first RF circuit to be mounted is the TX preamplifier and the tank circuit at its input. This sub-circuit is given in Figure 4.13. We analyzed all parts of this circuit in the chapter.

Cut a 30 cm long piece of 0.5 mm thick enameled wire. Carefully wind 18 turns on a T37-7 toroid, to make 1  $\mu$ H inductor L1. Count the turns carefully. It is very easy to end up by one turn more. Number of turns is the number of times the wire passes through the toroid. Spread the turns evenly around the toroid, otherwise the inductance will be significantly larger than the calculated value. Trim the wires to leave about 1 cm length at each lead for connection. Strip the enamel at each end for about 3 mm length, using your pocket knife or sand paper. Burning the enamel coating with a lighter before using sand paper or pocketknife helps. Stripping the enamel fully is very important. If you do not strip it properly, you will have a bad solder connection and your circuit will work and stop intermittently. It may be very annoying and time consuming to correct this later. Coat the stripped part with a thin layer of solder. If the enamel is not stripped fully, solder will not cover the wire smoothly but beads of solder will form on the wire. L1 is now ready.

Install and solder L1 on the PCB. Tie it on the PCB using a plastic strip. C65 is a trimmer capacitor and has three pins. One pin is connected to the trimming screw. That pin must be connected to the ground. Install C65 and solder it. The tank circuit is now ready. Install and solder the supply by-pass capacitors C67 and C68.



Install the feedback resistors R56 and R57. Leave the resistors few mm above the board so that you can connect the oscilloscope probes. Solder the resistors. Install one lead of the by-pass capacitor C64 to the hole on the C65 side only and solder it. Leave the lead on the IC7 side on the surface of PCB, for measurements.

Examine the data sheet of LM7171 for pin configuration and soldering information. Install IC9 carefully. Make sure that the mark on the IC fits to the one on PCB. Check if all pins are at their proper positions. Solder all the pins carefully. Check all connections using a multimeter for connectivity and for short circuits.

TX preamplifier is now ready for testing and tuning.

3. Ordinary pair of cables are not suitable to make connections of length more than few tens of centimeters at HF. We already discussed that there is always an inductance associated with a length of wire and a capacitance between two wires. These parasitic elements become significant as the frequency of interest increases, particularly if the wires are long. We use special cables in which this inductance and capacitance are carefully controlled. Coaxial cables are such cables. Cables of all oscilloscope probes, for example, are coaxial cables. RG58/U is a common laboratory grade coaxial cable, which is composed of a central conductor wire and an outer conductor. Outer conductor is a cylindrical braided wire, concentric with the central conductor. In between, there is a solid but flexible insulator, made of an insulating plastic material, which maintains the concentricity. The outermost layer is an insulating plastic sheath.

The ratio of incremental inductance ( $L$ ) and capacitance ( $C$ ) along the cable is maintained constant in coaxial cables. In RG58/U,  $(L/C)^{1/2}$  is  $50 \Omega$ .  $(L/C)^{1/2}$  is called the *characteristic impedance* of the cable. Hence, RG58/U is a  $50 \Omega$  coaxial cable. Usually BNC type connectors are used with RG58/U. These are bayonet type twist-and-fit connectors.

Signal generators are not ideal voltage sources with zero output impedance, but have finite output impedance. Most widely used standard is  $50 \Omega$ . The output levels of a signal generator are specified assuming that there is a  $50 \Omega$  termination (load) impedance connected to its output. If a cable connects a  $50 \Omega$  load to the generator, it is required that the characteristic impedance of the cable must also be  $50 \Omega$  for proper operation at high frequencies.

Figure 4.15(a) shows how we must connect the signal generator, oscilloscope and the circuit to make the measurements. Figure 4.15(b) shows how the coaxial cable from the signal generator must be connected to the circuit and the oscilloscope probe connection.

To connect the signal generator to the circuit, we shall use a very reliable laboratory measurement technique. We use a coaxial cable of about 1 m length, with a BNC connector on one end. Remove the black insulating sheath for about 1.5-2 cm from the bare end using your pocketknife, without damaging the braided outer conductor. Undo the exposed braid and twist it into a thick stranded wire. Remove about 5 mm of the inner polyethylene insulator without damaging the

inner conductor. Twist the inner conductor into a stranded wire. Heat each of the two stranded conductors and wet them with solder.

The part of the circuit in gray bracket in Figure 4.15 (b) is not implemented on the PCB, since it is not part of TRC-10 circuit. We need  $51\ \Omega$  in order to terminate the signal generator output, connected via a  $50\ \Omega$  cable (actually we need  $50\ \Omega$ , but only  $47\ \Omega$ ,  $51\ \Omega$  and  $56\ \Omega$  are available as standard resistor values closest to  $50\ \Omega$ ). We use  $1.5\ \text{K}$  resistor to simulate the output impedance of IC7, in order to make the measurements.

We mount these two components using *ground plane construction* technique. This is shown in Figure 4.15(c). Find a clear spot on the PCB close to the IC7 end of C64. Shorten the leads of two resistors,  $1.5\ \text{K}$  and  $51\ \Omega$ . Solder one end of  $1.5\ \text{K}$  to the free lead of C64 as in Figure 4.15(c). Using long-nose pliers shape one lead of  $51\ \Omega$  resistor into an “L”. Solder that lead directly on the ground plane of PCB such that the resistor stands like a stud. Solder the other lead to  $1.5\ \text{K}$  resistor, as shown in Figure 4.15(c).

Now, place the soldered outer conductor of the cable on the ground plane and heat it up with the soldering iron. Put ample amount of solder on the braided wire, to form a good contact with the ground plane. Solder the central conductor of the cable to the node where  $51\ \Omega$  and  $1.5\ \text{K}$  resistors are connected.

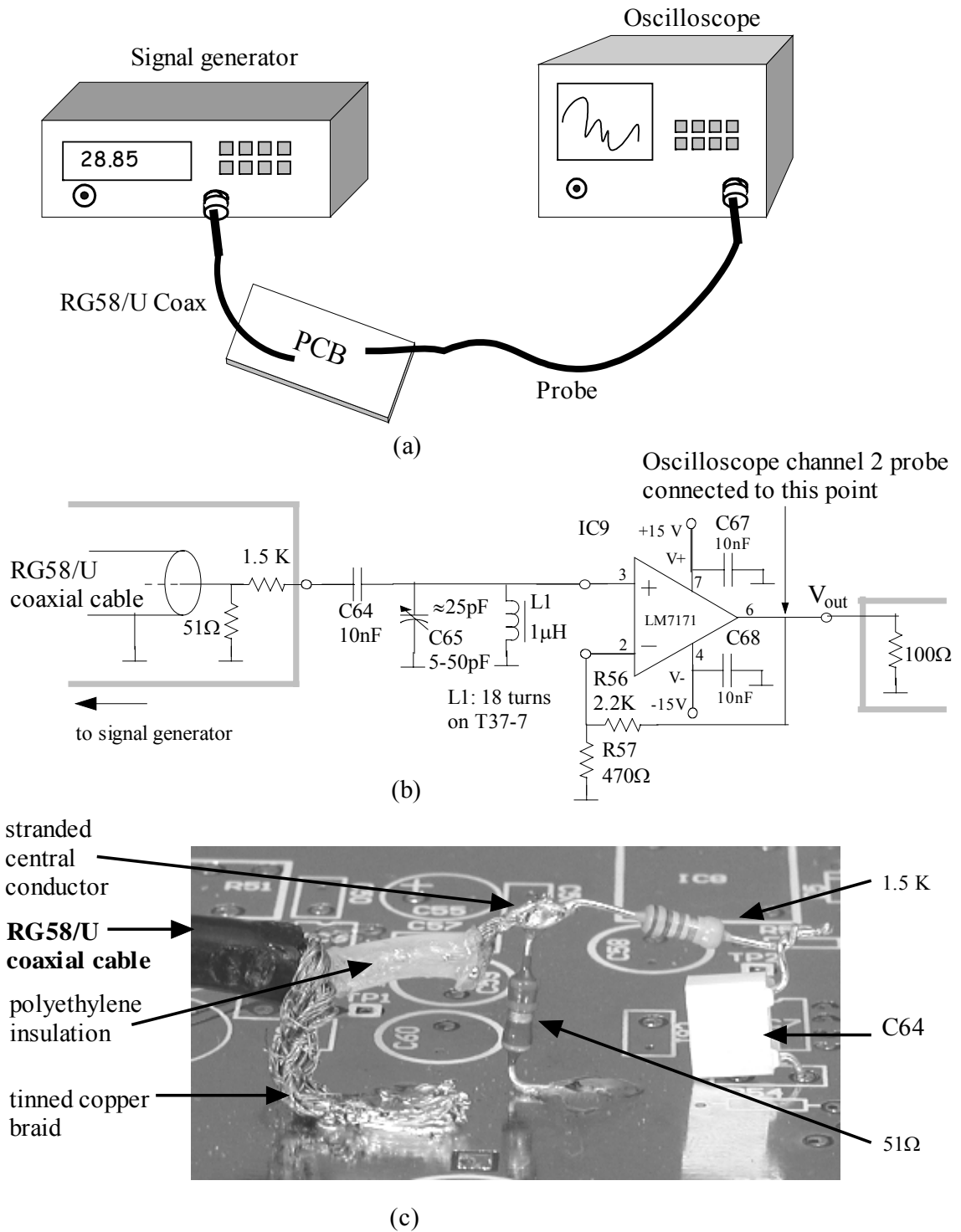


Figure 4.15 (a) Measurement set up for TX preamplifier, (b) circuit and connections on PCB, (c) ground-plane construction of test circuit components

$V_{out}$  node must be terminated by a  $100\ \Omega$  resistor also, as in the actual circuit. Install and solder one lead of R58 to the hole connected to  $V_{out}$ . Solder the other lead directly on the ground plane, for the time being.

*Make sure all soldered joints are very good joints.*

Ground plane construction technique provides an excellent trial and measurement environment. Parasitic effects are minimum in this circuit construction technique.

4. Before connecting the signal generator and the oscilloscope, switch the power ON, and check the circuit for d.c. voltages. Using a multimeter, measure the supply voltages at the OPAMP terminals, and make sure that they read  $\pm 15$  V.  $V_{\text{out}}$  must be within few mV of zero, and the voltages at pins 2 and 3 must read zero. Switch the power OFF.

Switch the signal generator ON and set the signal to CW (*continuous wave*) sinusoid at the frequency of 28.85 MHz. Set the amplitude to 100 mV (rms), or  $-7$  dBm (recall that  $-7$  dBm is 0.2 mW, the power delivered to  $50 \Omega$  resistor when 100 mV is applied across it). Switch the oscilloscope ON. Set the first channel to 0.1 V/div vertical and 50 nsec/div horizontal resolution. Trigger the oscilloscope and stabilize the waveform on the scope.

Switch the power of the circuit ON. Connect the oscilloscope (channel 2) probe to the output of the OPAMP, by hooking it to the corresponding pin of R58. Set the vertical axis to 0.2 V/div. Using alignment tool, adjust C65 such that the amplitude of  $V_{\text{out}}$  is made maximum. Measure the amplitude and record it. Congratulations! You had just tuned your tank circuit.

Calculate the absolute value of the voltage gain, and in dB, at 28.85 MHz. The output is what you observe on channel 2 and input is on channel 1. Calculate the power delivered to  $100 \Omega$  resistor.

Note that we base our calculations to the measurement made at the output of the amplifier only, rather than making another measurement directly across the tuned circuit. Making a direct measurement on the tuned circuit is not straightforward. Recall the Lab Exercise 3 of Chapter 3. Even when the probe is carefully compensated,  $C_p$ , the capacitance at the tip of the probe, appears directly across the circuit the probe is connected to (see problem 23). Although  $C_p$ , is a small capacitance in the order of 10 pF, it may load the circuits which has low parallel capacitance, at high frequencies. In this tuned circuit the parallel capacitance is only few tens of pF and a probe connection will definitely off-tune it. Special low-capacitance “FET probes” are used at higher frequencies.

5. The gain you have obtained experimentally may be less than 5.1, as calculated in Section 4.5.1. This is because of two reasons:
  1. Open Loop Gain of an OPAMP is usually reported for the case when the OPAMP is unloaded, or loaded by a large resistor. When the load impedance is small, the current demand from the amplifier increases and there can be a decrease in the gain. In TX preamplifier circuit, the load resistor is  $100 \Omega$ . In the data sheet, this variation of the gain with respect to load resistance is given for *large signal* case. Large signal case means that the amplitude of the signal at the output is close to the limits of the OPAMP. Signal levels at the output of TX preamplifier in the previous exercise can be considered as small signal. Hence there is not any data that we can use directly. However, we can deduce

from the large signal data that the nominal open loop gain can be 4 dB less than the nominal gain reported for 1 K load resistance. This corresponds to having a gain of 16 dB instead of 20 dB at 30 MHz.

2. There may be a difference in the open loop gain of OPAMPs coming from different production batches.

Calculate the voltage and power gain of the TX preamplifier stage if the open loop gain of the OPAMP is 16 dB with a phase margin of  $80^\circ$ , following the calculation method given in Section 4.5.1. Is the difference of 10.5 dB between the power gain and voltage gain is maintained, although the voltage gain decreased?

6. We shall now measure the transfer response of this stage using the set up in Exercise 3. Measure the output voltage amplitude at about 20 different frequencies between 1 MHz and 30 MHz (40 MHz if the signal generator allows), while keeping the input signal at  $-7$  dBm. Choose the frequencies cleverly in order to characterize both the pass band of the tank circuit around 29 MHz and its suppression at around 3 MHz. Calculate the voltage gain in dB and plot it versus frequency on a graph paper with a linear scaled vertical axis and logarithmic scaled horizontal axis.
7. Although the voltage gain we obtained in preamplifier stage may not be impressive, we obtained a good power gain and we filtered the transmit signal to some extent. We shall now increase this power level more by first increasing the voltage level using a series RLC circuit and then another OPAMP, in TX output amplifier.

L2 is a 1.3  $\mu$ H inductor. We shall make this inductor by winding 20 turns on T37-7 toroid. Use a 30 cm length of 0.2 mm diameter enamel coated wire and make the inductor carefully, following the procedure described for L1.

The circuit diagram of TX output amplifier is given in Figure 4.16. Make sure that the power of the circuit is switched OFF. Disconnect the probe from the circuit and coaxial signal cable from the signal generator, so that none of the equipment has any connection to the circuit. De-solder the lead of R58 which was connected to ground plane. Using de-soldering pump, remove the remaining solder on the ground plane. Install that lead of R58 to its proper hole and solder. Install and solder L2 and C69. Take care to place C69 such that its trimmer lead is connected to ground. Install and solder the OPAMP IC10, by-pass capacitors C70, C71, C54 and C55, and the feedback resistors R59 and R60. C54 and C55 have polarity and be careful to place them correctly before soldering. Also, be careful while installing LM7171 to place it correctly on the PCB.

This amplifier drives the output circuit, which is connected to the antenna. Antennas are designed usually to have a standard input impedance like 50  $\Omega$ , 75  $\Omega$ , etc. The output stage of TRC-10 is designed to drive an antenna with input impedance of 50  $\Omega$ . We simulate this impedance when testing the output amplifier, by using an extra 56  $\Omega$  resistor and ground plane construction, as shown in the gray bracket in at the output.

Install and solder one lead of R61 (56  $\Omega$  resistor) to the hole connected to the OPAMP output pin, pin 6. Using another 56  $\Omega$  resistor, make a stud exactly similar to the one we made with a 51  $\Omega$  resistor in Exercise 3, close to the other lead of R61. Complete the ground plane construction by making a soldered node with the two free leads of R61 and 56  $\Omega$  resistor.

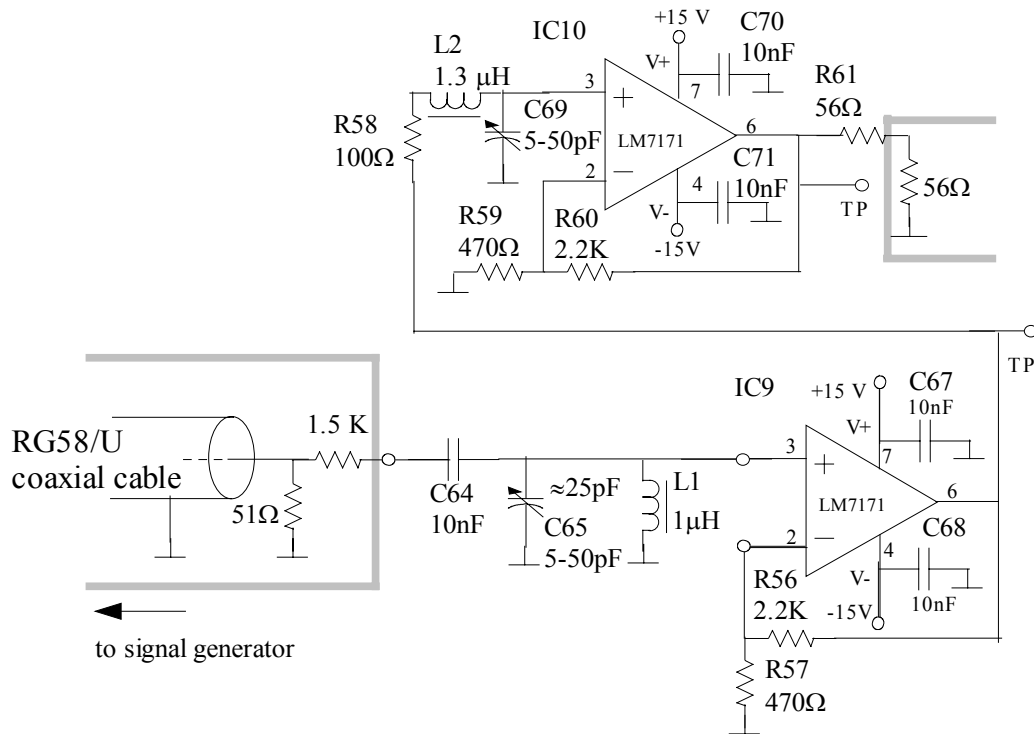


Figure 4.16 Measurement set up for TX preamplifier and TX output amplifier

Switch the power ON. Check the d.c. voltages in the circuit, as in Exercise 2. Connect the signal cable to the signal generator and the scope probe back to its position in Exercise 2. Check if the tank circuit still properly tuned. Decrease the signal generator output to  $-10$  dBm and set its frequency to 28.85 MHz. Hook up the probe across the 56  $\Omega$  output resistor. Set the vertical resolution to 1V/div.

If the signal amplitude is large there will be distortion on the output sinusoid. Decrease the signal generator output if necessary, to have an undistorted sinusoidal waveform at the output. Record the input and output voltage levels. Calculate the power delivered to 56  $\Omega$  load resistor.

Trim C69 such that the output waveform amplitude is maximized. The series RLC circuit is tuned. Measure the output voltage amplitude and record it. Calculate the overall gain of two stages.

8. Repeat Exercise 6 to find and plot the combined transfer response of two stages. Switch the power and the equipment OFF. Disconnect the probe.

9. We shall measure the amount of distortion on the output waveform using a *spectrum analyzer*. Spectrum analyzer is an instrument, which measures and plots the frequency components of a signal with respect to frequency. The frequency components are measured in terms of power (dBm) delivered to  $50\ \Omega$  input impedance of the analyzer.

TX output amplifier is expected to deliver about 40 mW to the antenna, at 29 MHz. Although the amplifier is a linear amplifier, there is distortion on the RF signal when the OPAMP is driven at its limits. Distortion means that there are signal components clustered around harmonics of 29 MHz. Signals produced by transmitters at frequencies other than the transmission frequency are called *spurious emissions* (like harmonics of the carrier), which must be suppressed.

The TX amplifier is a linear amplifier and the carrier waveform at the output is predominantly sinusoidal. The data sheets of LM7171 reveal that we must be able to get an output signal with about 3% distortion at full power.

Spectrum analyzers usually have  $50\ \Omega$  input impedance and can handle a maximum of 0 dBm input power. We must attenuate the TX output by more than 10 dB before connecting to the analyzer. We use the set up in Figure 4.16 with  $56\ \Omega$  termination resistance replaced by a three resistor attenuator and connection to spectrum analyzer, as shown in Figure 4.17. The circuit enclosed in gray bracket must be mounted using ground plane construction technique. The 10 nF capacitor is a d.c. decoupling capacitor. It blocks the d.c. path between the IC10 output and the spectrum analyzer. The circuit in the shaded box is the attenuator. It attenuates the IC10 output delivered to  $56\ \Omega$  load in Figure 4.16,  $V_{o1}$ , before it is presented to the spectrum analyzer as  $V_{o2}$ , while it maintains approximately  $50\ \Omega$  termination at both ends.

Calculate the output impedance of the attenuator  $Z_o$ , assuming that the output impedance of IC10 is zero and 10 nF capacitor is virtually short circuit above 28 MHz. Calculate the attenuation from  $V_{o1}$  to  $V_{o2}$  in dB, considering the impedance across the coaxial cable is the input impedance of the spectrum analyzer,  $50\ \Omega$ . What is the maximum signal power that IC10 can deliver to the analyzer now?

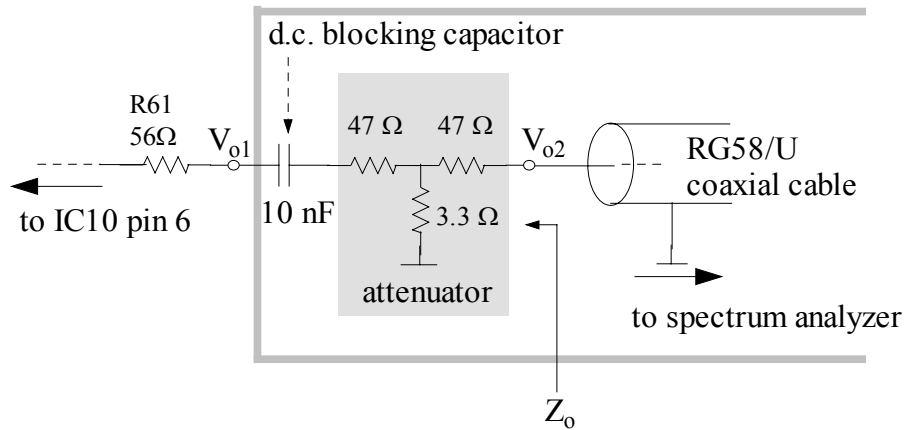


Figure 4.17 Measurement set up in Figure 4.16 modified for harmonic measurement

- De-solder and remove  $56\ \Omega$  termination resistor at the output in Figure 4.16. Remove excess solder on the PCB, using de-soldering pump. Construct part of the circuit in gray bracket in Figure 4.14, comprising  $10\ \text{nF}$  capacitor, two  $47\ \Omega$  resistors and  $3.3\ \Omega$  resistor.

The coaxial cable to the spectrum analyzer is actually a short cable terminated by the *BNC antenna jack*. We mount this jack on the back panel and the cable from the antenna is connected to it. We use this jack to make harmonic measurements in this exercise.

A neat and geometrically correct connection of BNC jack to RG58/U is very important. Ask the lab technician to show you a correctly connected jack. Cut a 25 cm long RG58/U cable. Strip the plastic cover at both ends for about 1 cm. Strip the polyethylene insulation from the ends for about 3 mm. Twist the central conductor and put a solder cover on it. First fit the central conductor to its place on the jack and make a very neat solder connection. Then fit and solder the braid.

Mount the antenna BNC jack to its place at the back panel tightly, with the nut provided.

Prepare the free end of the coaxial cable for ground plane construction and connect it to the attenuator output, as we had done in Exercise 3. Check your connections.

- Connect the cable with BNC plugs at both ends between the antenna jack and the spectrum analyzer. Ask the assistant or the lab technician to help you with the spectrum measurements. Switch the power on.
- Set the signal generator frequency to 28.85 MHz and its voltage to the level you recorded in Exercise 7. Measure the fundamental component and second and third harmonic power of the output signal. Correct these values using the attenuation you had calculated, and record them. Compare the fundamental component power to the one calculated in Exercise 7.



13. Disconnect the spectrum analyzer and the signal generator. Disconnect the coaxial cable to the spectrum analyzer. De-solder the other end of antenna jack cable from the PCB. You need the rest of the set up for harmonic measurements after the harmonic filter is installed. This measurement is the first experimental exercise in Chapter 5.

#### 4.9. Problems

- In a series RLC circuit,  $R = 100 \Omega$ ,  $L = 10 \mu\text{H}$  and  $C = 3 \text{ pF}$ . Plot the magnitude and the phase of the impedance as a function of  $\omega$  for  $0.6\omega_0 < \omega < 1.5\omega_0$ .
- A voltage  $10\angle 0^\circ$  is applied to the series circuit of Problem 1. Find the voltage across each element for  $f = 28, 29$  and  $29.7 \text{ MHz}$ .
- A series circuit with  $R = 50 \Omega$ ,  $C = 39 \text{ pF}$  and variable inductor  $L$  has an applied voltage  $V = 10\angle 0^\circ$  with a frequency of  $16 \text{ MHz}$ .  $L$  is adjusted until the voltage across the resistor is maximum. Find the voltage across each element.
- A series circuit has  $R = 50 \Omega$ ,  $L = 1 \mu\text{H}$  and variable capacitor  $C$ . Find  $C$  for series resonance at  $f = 16 \text{ MHz}$ .
- Given a series RLC circuit with  $R = 10 \Omega$ ,  $L = 0.5 \mu\text{H}$  and  $C = 220 \text{ pF}$ , calculate the resonant, lower and upper half power frequencies.
- Show that the resonant frequency  $\omega_0$  of an RLC series circuit is the geometric mean of  $\omega_1$  and  $\omega_2$ , the lower and upper half power (cut-off) frequencies.
- The circuit in Figure 4.18 (a) is a parallel connection of a capacitor and a coil where the coil resistance is  $R_L$ . Find the resonant frequency of the circuit. What is the condition that no resonance occurs?

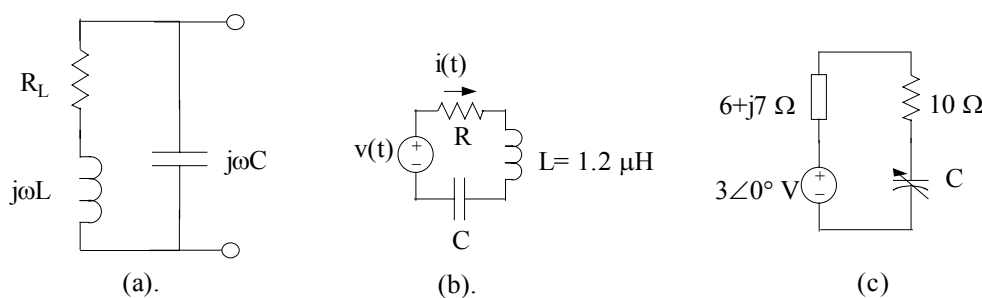


Figure 4.18. Circuit for (a) problem 7, (b) problem 10 and (c) problem 11.

- Show that  $Q = \omega_0 L / R = f_0 / \text{BW}$  for a series RLC circuit.
- Find the  $Q$  of the series circuit with  $R = 20 \Omega$ ,  $C = 47 \text{ pF}$  and  $L = 2 \mu\text{H}$ .
- In the series circuit of Figure 4.18(b), the instantaneous voltage and current are  $v(t) = 1.5 \sin(2\pi \times 10^7 t + 30^\circ) \text{ V}$  and  $i(t) = 10 \sin(2\pi \times 10^7 t + 30^\circ) \text{ mA}$ . Find  $R$  and  $C$ .

11. In the series circuit of Figure 4.18(c), the impedance of the source is  $6+j7$  and the source frequency is 10 MHz. At what value of  $C$  will the power in  $10\ \Omega$  resistor be a maximum? What is this maximum power deliverable to  $10\ \Omega$  resistor.
12. In the parallel circuit of Figure 4.19(a), determine the resonant frequency if  $R = 0\ \Omega$  and  $R = 1\ \Omega$ . Compare them to resonant frequency when  $R = 100\ \Omega$ .
13. In the parallel circuit in Figure 4.19(b) find the resonant frequency  $f_0$ .

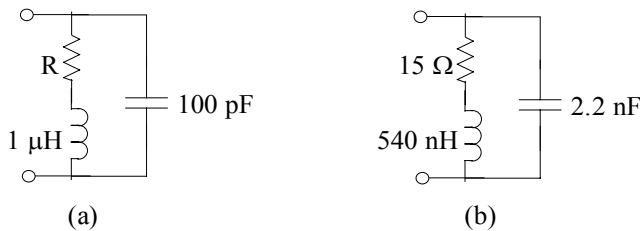


Figure 4.19. Circuit for (a) problem 12 and (b) problem 13

14. Refer to Figures 1.11 and 1.13. There is a lower sideband signal component at  $f_{IF} - f_{VFO}$ , at the output of TX Mixer. Calculate the approximate value of the attenuation this signal experiences compared to the upper side band component, after it is filtered by parallel RLC circuit of Figure 4.4.
15. Calculate the branch voltages, i.e. voltages across R58, L2 and C69, of the series RLC circuit in Figure 4.8 if  $V_s = V\angle 0$ , at the resonance frequency. Write down the time waveform expressions for these voltages.
16. Calculate the same branch voltages in Exercise 15, at upper and lower 3-dB frequency of the series circuit. Write down the time waveform expressions for these voltages.
17. Calculate the d.c. resistance of a 0.2 mm diameter wire of length 10cm. Calculate its skin effect cut-off frequency. Calculate the approximate copper loss of an inductor made of this wire at 29 MHz.
18. Both capacitors and inductors suffer from a *parasitic* effect called *self-resonance*. In case of capacitors, the two leads with which the capacitor is connected to the circuit causes small parasitic inductance in series with the capacitance. This is shown in Figure 4.20. Any piece of wire has an inductance. This inductance is given as

$$L(\mu\text{H}) = 0.002 b [\ln(2b/r) - 0.75]$$

where  $L$  is inductance in  $\mu\text{H}$ ,  $b$  is the length of wire in cm and  $r$  is the radius of the wire in cm.

Assume we have a 1nF capacitor with leads 1 cm each, made of 0.8 mm diameter wire. Calculate  $L_s$ , which is the sum of the two parasitic inductances due to each lead. Calculate the self-resonance frequency of this capacitance. At this frequency

the capacitor appears like a short circuit. As the frequency is further increased, the capacitor exhibits an inductive reactance!

The full equivalent circuit of a capacitance is given in Figure 4.20(c), for the sake of completeness. The parallel resistance  $R_p$  models the loss in the dielectric material from which the capacitor is made off, and series resistor  $R_s$  represents the sum of copper loss in the leads and the losses at the lead contacts. Usually  $R_p$  is very high and  $R_s$  is very low, and both of them can be ignored. The main parasitic effect is the self-resonance.

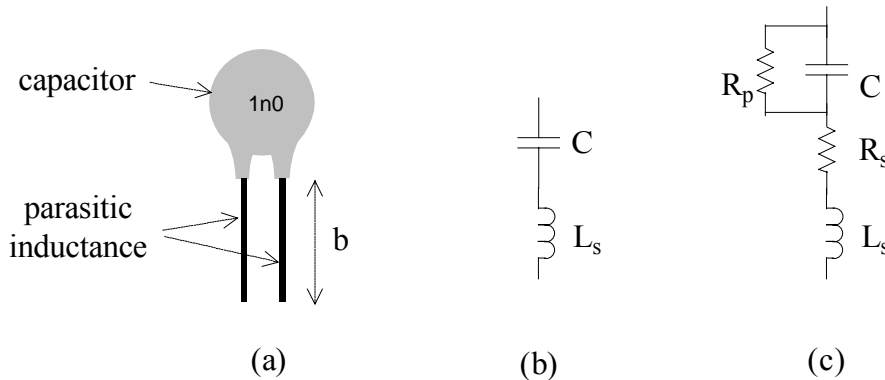


Figure 4.20 (a) Capacitor, (b) Capacitor model with parasitic inductance, (c) Full high frequency model

19. The main parasitic effect in inductances is the *inter-winding capacitance*. There is a capacitance between each turn in an inductance. The value of this capacitance depends on various parameters like the physical distance between each turn, the wire diameter, size of the turn, etc. This is shown in Figure 4.21. The distributed capacitive coupling between windings can be modeled as a parallel parasitic capacitance,  $C_p$ , as shown in Figure 4.21(b). The series resistance is the total loss of the inductor.

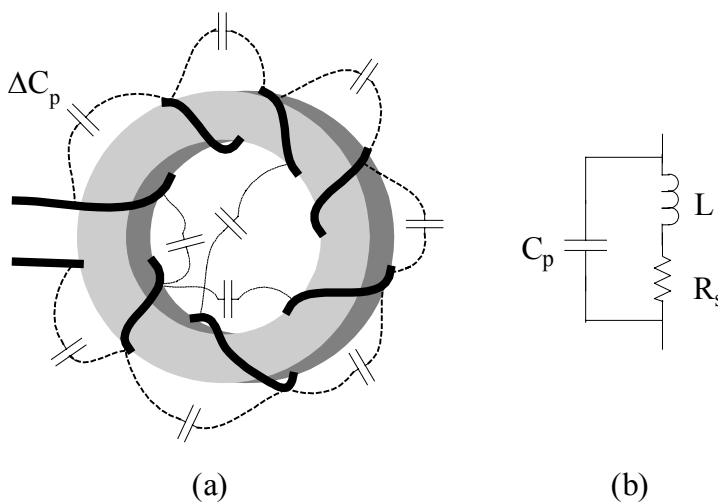


Figure 4.21 (a) Inductor, (b) Inductor model with parasitic capacitance

$C_p$  is a small capacitance, usually less than a pF, but it can be effective in the frequencies of interest, particularly if the inductance value is large.

An inductor made by winding 32 turns on a core, which has  $A_L$  of 20 nH/turn<sup>2</sup>. the inductance is measured to be 20  $\mu$ H at 1 MHz. The impedance of the inductor is purely resistive at 40 MHz. Assuming that the Q of the inductor is larger than 10 and  $A_L$  is constant up to this frequency (which is not a good assumption since the permeability of most materials fall with increasing frequency), calculate the winding capacitance.

20. What must be the length of a 900 nH air core inductor if it has 21 turns and its diameter is 5 mm?
21.  $A_L$  of the T25-10 toroidal core is given as 1.9 nH/N<sup>2</sup>, where N is number of turns. Find the number of turns required to make a 615 nH inductor using T25-10.
22. An inductor made by winding 7 turns on T20-7 toroid yields 0.13  $\mu$ H with a Q of 102 at 30 MHz. Find the approximate value of  $A_L$  for T20-7 and the equivalent series core loss resistance of this inductor at 30 MHz.
23. Consider the all-pass probe compensation circuit given in Figure 4.22. Show that the equivalent impedance  $Z_{eq}$  becomes approximately equal to the parallel connection of  $C_p$  and  $R_p$ , when the probe is compensated ( $R_p = 9 R_T$  for X10 probe).

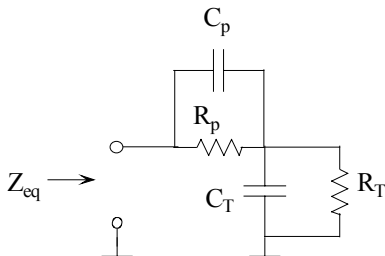


Figure 4.22 Input impedance of probe compensation circuit

24. The RF amplifier given in Figure 4.23(a) is designed using an OPAMP. The open loop voltage gain of the OPAMP is found to be

$$A_{ol}(\omega) = 5 \times 10^5 / (1 + j\omega/\omega_1)$$

from the data sheet, where  $\omega_1$  is 100 rps. What is the voltage gain of the non-inverting amplifier at 400 KHz? What is its gain at 1 KHz?

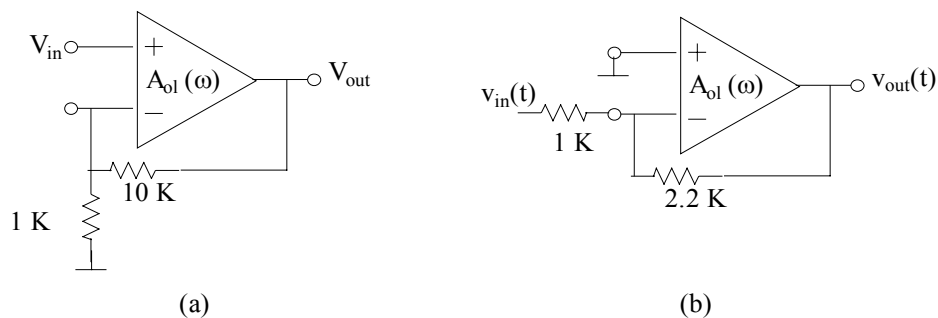


Figure 4.23. Circuit for (a) problem 24 and (b) problem 25.

25. Same OPAMP is used in inverting amplifier configuration, as shown in Figure 4.23(b). What is the output voltage  $v_{out}(t)$  when the input is  $v_{in}(t) = 2 \cos[(2\pi \times 10^6)t + 30^\circ]$  volts? What is  $v_{out}(t)$  when the input is  $v_{in}(t) = 2 \cos[(2\pi \times 10^3)t + 30^\circ]$ ?
26.  $v_i(t)$  in Figure 4.24 is given as  $v_i(t) = A_1 \cos[(2\pi \times 16 \times 10^6)t] + A_2 \cos[(2\pi \times 8 \times 10^6)t]$ , and assume that the output  $v_o(t) = B_1 \cos[(2\pi \times 16 \times 10^6)t + \theta_1] + B_2 \cos[(2\pi \times 8 \times 10^6)t + \theta_2]$  is obtained. Find the value of capacitance which maximizes the ratio  $|B_1/B_2|$ ? What are  $|B_1/A_1|$  and  $|B_2/A_2|$  for this value of  $C$ ?

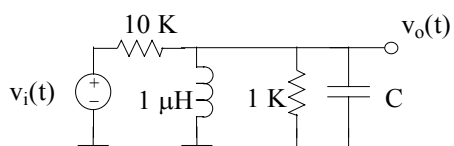


Figure 4.24

## Chapter 5 : FILTERS

Parallel and series tuned circuits filter electronic signals. Their filtering performance is only determined by the quality factor of the circuit. Quite often it is necessary to have filters with improved performance compared to what tuned circuits can offer.

The theory underlying design of electronic filters is formidable. We confine our discussion of filters to the extent necessary to implement the filters used in TRC-10, in this chapter.

### 5.1. Motivation

Any electronic filter can be visualized as a block between a source (input) and a load (output). This is depicted in Figure 5.1. What is expected from this is to maintain the signal components at *wanted* frequencies, and eliminate the ones at *unwanted* frequencies, as much as possible.

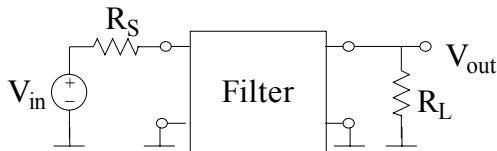


Figure 5.1 Electronic filter

Consider for example the RC LPF given in Figure 5.2(a), which is similar to the ones we used in Chapter 2.

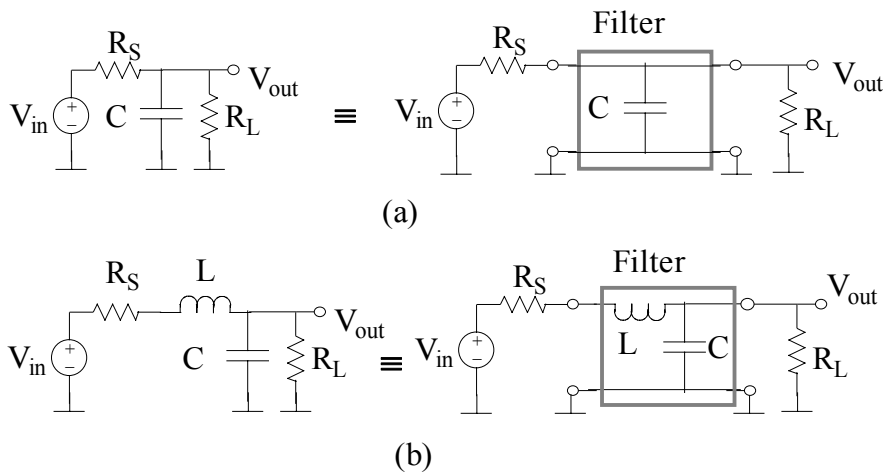


Figure 5.2 LPF (a) one element, (b) two elements

The 3-dB cut-off frequency of such a filter is,

$$\omega_c = 1/C(R_S // R_L)$$

and its transfer function is

$$H_1(\omega) = [R_L / (R_S + R_L)] / (1 + j\omega / \omega_c).$$

We can get a similar filtering effect, if we use a series inductance instead of a parallel capacitance. If we use both a series inductance and a parallel capacitance in the filter block, we expect a better filtering function. Indeed the filter in Figure 5.2(b) has a transfer function

$$H_2(\omega) = 1 / [(1+R_S/R_L) - \omega^2 CL + j\omega(L/R_L + C R_S)].$$

This expression looks like a tuned circuit transfer function and it is not easy to immediately recognize it as a LPF. However for the special case of  $R_S = R_L$ , and with a choice of  $C$  and  $L$  to yield *maximally flat* behavior in the pass-band (we discuss this later in this chapter),  $H_2(\omega)$  becomes

$$H_2(\omega) = 1 / \{2[1 - (\omega/\omega_c)^2 + j\sqrt{2} \omega/\omega_c]\},$$

where  $\omega_c$  is  $\sqrt{2/LC}$ , for this circuit. Under the same condition  $R_S = R_L$ ,  $H_1(\omega)$  is

$$H_1(\omega) = 1 / [2(1 + j\omega/\omega_c)].$$

The magnitudes of both transfer functions are plotted with respect to frequency in Figure 5.3.

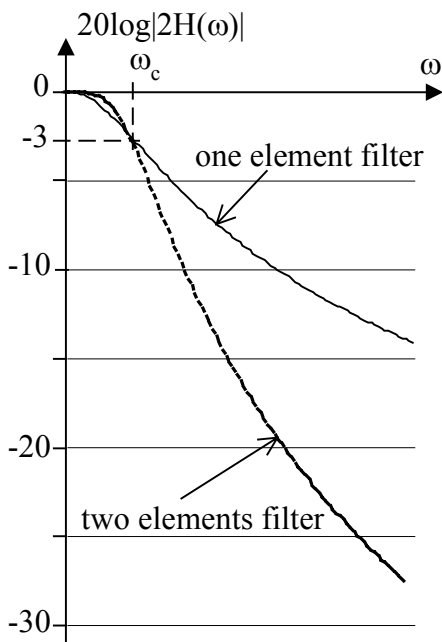


Figure 5.3 Transfer function of two LPFs, (a) one element, (b) two elements

Note that the filter with two elements is superior to the single element one in two respects:

1. Suppression of signal components at frequencies higher than  $\omega_c$  is significantly improved;
2. The signal components with frequencies less than  $\omega_c$  are better preserved, or less distorted.

How many filter elements must be used? Which kind of elements must be used? Or, what must be the values of the elements?

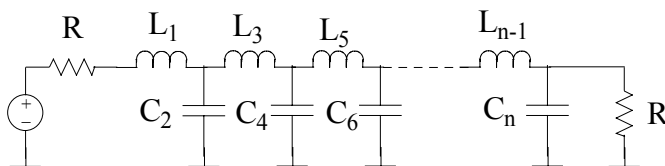
Modern filter theory addresses these questions by a systematic filter design technique.

## 5.2. Polynomial filters

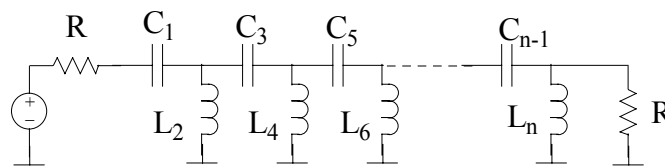
Modern filter theory maps the desired transfer function of a filter to the properties of a class of polynomials like Chebyshev polynomials, elliptic polynomials, etc. We use maximally flat, or *Butterworth filters* in TRC-10, which are also polynomial filters.

The basic building blocks in polynomial filters are LPF and HPF. BPF and band-stop filters are derived from these blocks.

The circuit morphology on which polynomial filters are based is of *ladder* type, as shown in Figure 5.4.



(a)



(b)

Figure 5.4 Ladder filters (a) LPF, (b) HPF

The parallel elements in the LPF configuration of Figure 5.4(a) are capacitors, and series elements are inductors. At low frequencies, inductors provide a low impedance path to the input signal to the output, while capacitors maintain high impedance to ground, and hence low loss. At high frequencies on the other hand capacitor impedance is low, and therefore there is a loss in the signal at every node. Inductors, on the other hand, have high impedance, and the division effect at each node on the signal is increased. In the HPF ladder in Figure 5.4(b), series and parallel elements are interchanged compared to LPF, thus yielding exactly an opposite function.

### 5.2.1. Butterworth filters

Butterworth filter seeks to have a power transfer function of

$$P_L/P_A = 1 / [1+(\omega/\omega_c)^{2n}]$$



where  $\omega_c$  is the 3-dB cut-off frequency,  $n$  is the number of elements (capacitors and inductors),  $P_L$  is the power delivered to load and  $P_A$  is the available power at the source.  $P_L/P_A$  is plotted versus frequency for different number of elements,  $n$ , in Figure 5.5. As  $n$  increases the power transfer function approaches to an ideal LPF.

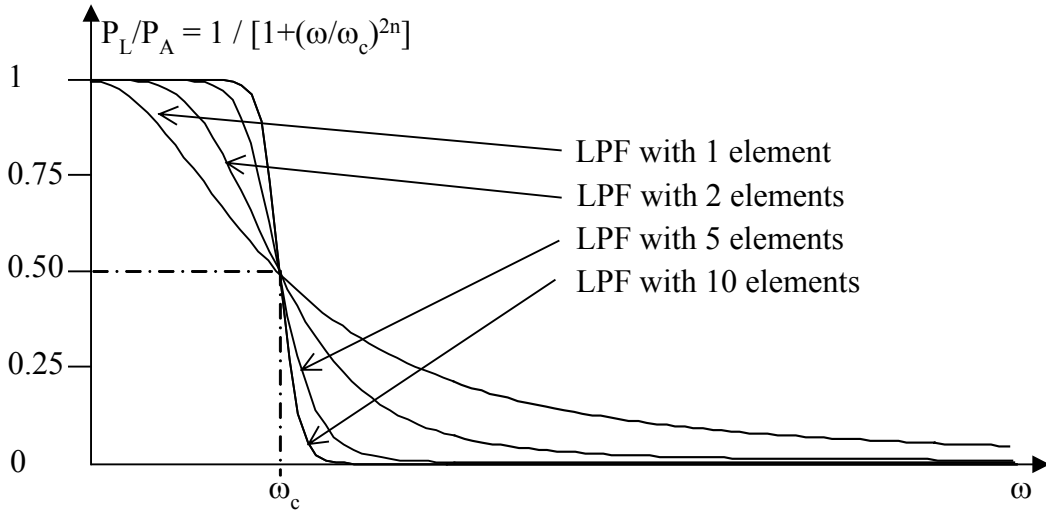


Figure 5.5 Butterworth LPF power transfer function

The number of elements in a filter can be odd or even and the ladder filter can start with a series or a parallel component. Usually configurations with fewer inductors are preferred, for constructional simplicity.

The component values for Butterworth filters, *normalized* with respect to the termination impedance and cut-off frequency, are provided in tables. Normalized coefficient are correct reactance and susceptance values in  $\Omega$ , for  $1 \Omega$  source and load resistance and for a cut-off frequency of  $\omega_c = 1$  rad/sec. A table of these coefficients, up to eight elements, is given in Table 5.1.

Using this table for designing a filter is straightforward. Since the coefficients are normalized component values, we must de-normalize them for given termination resistance and cut-off frequency.

n	b <sub>1</sub>	b <sub>2</sub>	b <sub>3</sub>	b <sub>4</sub>	b <sub>5</sub>	b <sub>6</sub>	b <sub>7</sub>	b <sub>8</sub>
1	2.0000							
2	1.4142	1.4142						
3	1.0000	2.0000	1.0000					
4	0.7654	1.8478	1.8478	0.7654				
5	0.6180	1.6180	2.0000	1.6180	0.6180			
6	0.5176	1.4142	1.9319	1.9319	1.4142	0.5176		
7	0.4450	1.2470	1.8019	2.000	1.8019	1.2470	0.4450	
8	0.3902	1.1111	1.6629	1.9616	1.9616	1.6629	1.1111	0.3902

Table 5.1 Normalized Butterworth filter coefficients

For example, consider a LPF with three elements depicted in Figure 5.6(a). The filter has two parallel capacitors and one series inductor. The source and load resistances of this filter are  $50\ \Omega$ . Assume that we want to have a cut-off frequency of 20 MHz.

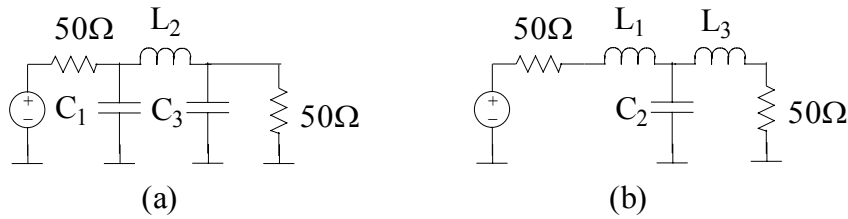


Figure 5.6 Three element LPFs

The filter coefficients are  $b_1=1$ ,  $b_2=2$  and  $b_3=1$  from Table 5.1 and  $\omega_c$  is  $1.257\ E8$  rad/sec, or 20 MHz. The first element is a parallel element and its susceptance is  $B_1 = \omega_c C_1$ . The normalized value of  $B_1$ ,  $B_1/(1/50\Omega)$ , is equal to the first coefficient  $b_1$ . Hence

$$\begin{aligned} B_1 &= b_1/50\Omega \\ &= 20\ \text{mS}, \end{aligned}$$

and the capacitance  $C_1$  becomes

$$\begin{aligned} C_1 &= B_1/\omega_c \\ &= 159\ E-12\ \text{F} \\ &\text{or } 159\ \text{pF}. \end{aligned}$$

The second element is a series element and therefore its normalized reactance  $X_2 = \omega_c L_2$ , is related to the second coefficient as

$$X_2/50\Omega = b_2.$$

Hence  $X_2$  becomes

$$X_2 = 100\Omega$$

and  $L_2$  at 20 MHz becomes

$$L_2 = X_2/\omega_c = 796\ \text{nH}.$$

The last element is again a parallel capacitance with a normalized value equal to that of  $C_1$ , hence  $C_3 = C_1 = 159\ \text{pF}$ . With these component values, this filter becomes a *third order Butterworth LPF* with a 3-dB cut-off at 20 MHz.

We can have the same characteristics with the filter shown in Figure 5.6(b), with same normalized values. In this configuration, the first component is a series element. In this case  $X_1$  is  $\omega_c L_1$  and

$$X_1/50\Omega = b_1 = 1,$$

yielding

$$L_1 = X_1/\omega_c = 398 \text{ nH.}$$

Again, since  $b_3=b_1=1$ ,  $L_3=L_1=398 \text{ nH}$ . The second element is a parallel capacitance. Normalized susceptance  $B_2$ ,  $B_2/(1/50\Omega)$ , is equal to  $b_2$  and  $C_2$  is

$$\begin{aligned} C_2 &= B_2/\omega_c = 2(20 \text{ mS})/(1.257 \text{ E8 rad/sec}) \\ &= 318 \text{ pF.} \end{aligned}$$

### Generalization

Assume that resistances equal to  $R_o$  terminates the ladder filter on both sides. Let  $G_o=1/R_o$ . The algorithm of filter design using tables is as follows:

- If the  $i^{\text{th}}$  element in the ladder is a *series* element, then its normalized absolute reactance is equal to the  $i^{\text{th}}$  coefficient in the table, i.e.  $|X_i|/R_o = b_i$ .
  - ◆ If the *series* element is an *inductance*, then  $L_i = X_i/\omega_c = b_i R_o/\omega_c$ ;
  - ◆ If the *series* element is a *capacitance*, then  $C_i = 1/\omega_c |X_i| = 1/\omega_c b_i R_o$ ;
- If the  $i^{\text{th}}$  element in the ladder is a *parallel* element, then its normalized absolute susceptance is equal to the  $i^{\text{th}}$  coefficient in the table, i.e.  $|B_i|/G_o = b_i$ , or  $|B_i| = b_i/R_o$ ,
  - ◆ If the *parallel* element is a *capacitance*, then  $C_i = B_i/\omega_c = b_i/\omega_c R_o$ ;
  - ◆ If the *parallel* element is an *inductance*, then  $L_i = 1/\omega_c |B_i| = R_o/\omega_c b_i$ .

The coefficients for Butterworth filters up to 8 elements are given in Table 5.1. The coefficients for filters with more elements can be obtained from the following relation:

$$b_i = 2\sin[(2i-1)\pi/2n]$$

where  $i$  is the position of the element and  $n$  is the total number of elements in the filter, i.e.  $i = 1, 2, \dots, n$ .

### 5.2.2. Harmonic filter

In case of TRC-10, we have a TX output amplifier which is expected to deliver about 10 mW to the antenna, at 29 MHz. Although the amplifier is a linear amplifier, there is distortion on the RF signal when the OPAMP is driven at its limits. Distortion means that there are signal components clustered around harmonics of 30 MHz.

Radio-communication regulations are very specific about such harmonic emission. Regulations specify the limitation above 30 MHz as follows:

*“In transmitters with 25 W or less mean output power, spurious emissions must be at least 40 dB below the mean power of fundamental emission and never greater than 25  $\mu$ W, but need not be reduced further than 10  $\mu$ W.”*

Signals produced by transmitters at frequencies other than the transmission frequency are called *spurious emissions* (like harmonics of the carrier), which must be suppressed. 30 dB below 10 mW is 10  $\mu$ W. The harmonic emissions of TRC-10 must not exceed 10  $\mu$ W.

The TX amplifier is a linear amplifier and the carrier waveform at the output is predominantly sinusoidal. The data sheets of LM7171 reveal that we must be able to get an output signal with about 3% distortion at full power. If we allow a safe margin by assuming 10% distortion, then the spurious signals are at least 20 dB below the mean output power to start with. Part of this spurious power (actually a small part in TRC-10) is at second harmonic, at about 58 MHz. If a filter to suppresses the second harmonic by about 10 dB, the higher harmonics will be suppressed more. To be on the safe side, let us design a filter which suppresses the second harmonic by 20 dB.

Any design starts with a requirement specification. The harmonic filter specification for TRC-10 can be derived from above discussion. We need a LPF just before the antenna, which must have 20 dB attenuation at 58 MHz, it must not attenuate 29.7 MHz at any significant level (for example at most 1 dB) and it must have a minimum number of elements.

Let us design a Butterworth filter and write the first two requirements quantitatively to see what we have for third requirement:

$$10\log(P_L/P_A)|_{f=29.7\text{ MHz}} = -10\log[1+(29.7E6/f_c)^{2n}] = -1\text{ dB}$$

$$\Rightarrow (29.7E6/f_c)^{2n} = 0.259$$

$$\Rightarrow n(7.473)-n \log f_c = -0.293$$

and

$$10\log(P_L/P_A)|_{f=58\text{ MHz}} = -20\text{ dB} \quad \Rightarrow n(7.763)-n \log f_c = 0.998.$$

Solving the two equations simultaneously for  $n$  and  $f_c$ , we obtain  $n = 4.45$  and  $f_c = 34.6\text{ MHz}$ .

Number of elements in the filter must be larger than 4. We take  $n = 5$ . In order to keep the number of inductors low, we shall choose the ladder configuration with three parallel capacitors and two series inductors as in Figure 5.7.

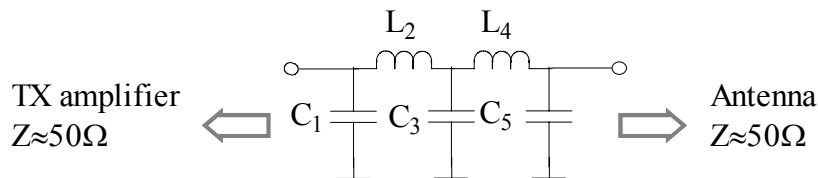


Figure 5.7 Harmonic filter

The filter coefficients are  $b_1 = b_5 = 0.618$ ,  $b_2 = b_4 = 1.618$  and  $b_3 = 2$ , from Table 5.1. Then,

$$C_1 = C_5 = 0.618/\omega_c(50\ \Omega)$$

and

$$C_3 = 2/\omega_c(50 \Omega).$$

At 34.6 MHz,  $C_1$  is 56.9 pF. The capacitor values must be standard values. We must choose the cut-off frequency to yield standard values. Hence we choose  $C_1 = C_5 = 56$  pF.

$C_3$  becomes 181.2 pF  $\Rightarrow C_3 = 180$  pF. With these capacitor values  $f_c$  becomes approximately 35.2 MHz. We cannot be very precise about matching the capacitor values, the cut-off frequency or the termination resistance. The capacitors have tolerances, which exceed this mismatch easily.

The inductors are,

$$L_2 = L_4 = 1.618 (50 \Omega)/\omega_c \approx 370 \text{ nH}.$$

Let us check the attenuation at few frequencies before we finish the design:

$$10\log(P_L/P_A)|_{f=29.7 \text{ MHz}} = -0.72 \text{ dB},$$

$$10\log(P_L/P_A)|_{f=29 \text{ MHz}} = -0.58 \text{ dB},$$

$$10\log(P_L/P_A)|_{f=58 \text{ MHz}} = -21.7 \text{ dB},$$

$$10\log(P_L/P_A)|_{f=87 \text{ MHz}} = -39.3 \text{ dB}.$$

To calculate attenuation at harmonics, we can use the approximation,

$$10\log(P_L/P_A)| = -10\log[1+(\omega/\omega_c)^{2n}] \approx -10\log(\omega/\omega_c)^{2n} = -20n \log(\omega/\omega_c).$$

The attenuation provided by the filter is  $-20n$  dB per decade increase of frequency.

We use the filter designed above in TRC-10 as output harmonic filter. The components of this filter are C77, C78, C79, L6 and L7.

### 5.2.3. Butterworth HPF

HPF ladder uses inductors as parallel elements and capacitors as series elements. The power transfer function for Butterworth HPF is

$$P_L/P_A = 1 - 1 / [1+(\omega/\omega_c)^{2n}] = (\omega/\omega_c)^{2n} / [1+(\omega/\omega_c)^{2n}].$$

Consider the fourth order filter given in Figure 5.8. The normalized values are  $b_1 = b_4 = 0.7654$  and  $b_2 = b_3 = 1.8478$ .

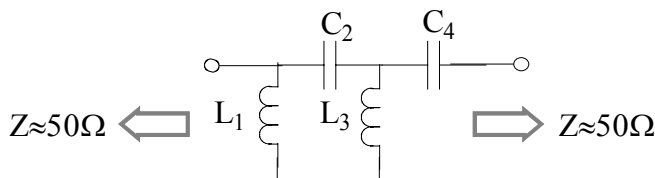


Figure 5.8 4-element HPF

If for example  $f_c$  is 20 MHz, the component values of the circuit in Figure 5.8 are

$$\begin{aligned} L_1 &= (50 \Omega)/\omega_c b_1 = 520 \text{ nH}, \\ C_2 &= 1/\omega_c b_2(50 \Omega) = 86 \text{ pF}, \\ L_3 &= (50 \Omega)/\omega_c b_3 = 215 \text{ nH}, \\ C_4 &= 1/\omega_c b_4(50 \Omega) = 208 \text{ pF}. \end{aligned}$$

Note that we could have chosen the 4-element ladder starting with a series capacitor instead of a parallel inductor. In that case the first element value is equal to  $C_4$  above and the resulting filter is the same the one in Figure 5.8 flipped horizontally. This shows that we can interchange the input and output of these filters. This property is called *reciprocity*.

#### 5.2.4. Butterworth BPF

BPF design starts with the design of a LPF with a 3 dB frequency,  $\omega_c$ , equal to the 3 dB bandwidth,  $\Delta\omega$ , of the BPF. Once LPF is designed, every series element (and every parallel element) is series (parallel) tuned out at an appropriate frequency, to obtain the BPF. In other words, a parallel resonant LC circuit replaces every parallel element and a series resonant LC circuit replaces every series element. The center frequency,  $\omega_{\text{center}}$ , of a BPF is placed at the arithmetic mean of the 3 dB frequencies,  $\omega_1$  and  $\omega_2$ , such that

$$\omega_{1,2} = \omega_{\text{center}} \pm \Delta\omega/2.$$

The resonance frequency  $\omega_0$  is, however, the geometrical mean of the two 3 dB frequencies. Design specifications of a BPF usually include a center frequency and BW.  $\omega_1$  and  $\omega_2$  are calculated first, and then the resonance frequency  $\omega_0$  is determined.

Conversion from LPF to BPF is depicted in Figure 5.9, where a BPF, with 3 dB BW of  $\Delta\omega = \omega_2 - \omega_1$  and a center frequency of  $(\omega_1 + \omega_2)/2$ , is derived from a LPF. In this filter, the resonance frequency is

$$\begin{aligned} \omega_0 &= \sqrt{\omega_1 \omega_2} \\ &= (L_1 C_1)^{-1/2} = (L_2 C_2)^{-1/2}. \end{aligned}$$

The out-of-band suppression of this filter follows the suppression properties of the LPF equivalent. In other words, if the low pass equivalent is of  $n^{\text{th}}$  order, the power transfer function of the BPF is,

$$P_L/P_A = 1 / \{1 + [(\omega - \omega_0)/\omega_c]^{2n}\},$$

or

$$P_L/P_A = 1 / \{1 + [(\omega - \omega_0)/\Delta\omega]^{2n}\}.$$

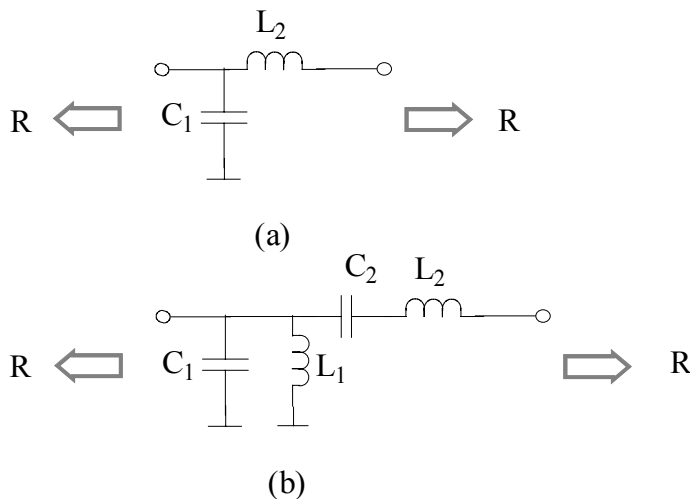


Figure 5.9 (a) LPF with  $\omega_c = \Delta\omega$ , (b) BPF obtained by tuning out  $C_1$  with  $L_1 = 1/\omega_0 C_1$  and  $L_2$  with  $C_2 = 1/\omega_0 L_2$

The wanted signal at the output of RX mixer is an AM signal with center frequency of 16 MHz and a BW of 6 KHz. This signal must be filtered out in order to avoid other products of the mixer to reach IF amplifiers and the detector. We need a 6 KHz BW BPF there. The frequency selectivity of this filter is very high, hence the center frequency and the resonance frequency can be considered to be equal (i.e. the arithmetic mean and the geometrical mean of  $\omega_1$  and  $\omega_2$  are almost equal).

The output resistance of SA602A (or SA612A) is 1.5 K. If we take R as 1.5 K, then  $C_1$  and  $L_2$  becomes

$$C_1 = b_1/\omega_c R = (1.4142)/[(2\pi \times 6E3)(1.5E3)] = 25 \text{ nF}$$

$$L_2 = b_2 R/\omega_c = [(1.4142)(1.5E3)]/[(2\pi \times 6E3)] = 56.3 \text{ mH.}$$

These are the values of the LC LPF with 6 KHz BW. The component values to tune these out at 16 MHz are

$$L_1 = 1/\omega_0^2 C_1 = 4 \text{ nH}$$

$$C_2 = 1/\omega_0^2 L_2 = 0.0018 \text{ pF.}$$

A BPF with above component values has the desired BW at 16 MHz, but such a filter cannot be realized. The required value of  $L_2$  is too high for a RF inductor to successfully implement at 16 MHz and values of  $L_1$  and  $C_2$  are too low.

We may expect an improvement if we decrease the termination resistance. We discuss methods of impedance transformation in the next section. Assume that output impedance of mixer IC is transformed to a low standard value like 50  $\Omega$ . The component values of BPF become

$$C_1 = 750 \text{ nF}$$

$$L_2 = 1.9 \text{ mH}$$

$$L_1 = 0.133 \text{ nH}$$

$$C_2 = 0.053 \text{ pF.}$$

All of the component values are out of range for good circuit design at 16 MHz, now (remember the parasitic lead inductance for capacitors; it is very effective on 750 nF at 16 MHz). Unrealistic component values are the result of very stringent requirement on the Q of the BPF. We require 0.04 % BW. Such filter problems are addressed by using special devices in electronic design. We use a *crystal ladder filter* in TRC-10 as IF filter, which we discuss later in this chapter.

### 5.3. Impedance matching

We often face the problem of converting impedance to another level, without any power loss or at least minimum power loss. One such case is the filtering problem we discussed in the previous section. Another one, and the most common one, is to have maximum power transfer to the load from a source. Impedance transformation is a fundamental topic in electronics and there is a wealth of information on it. The techniques that we use in TRC-10 are discussed below.

#### 5.3.1. Transformers

Transformers are most common impedance transformation devices. The ideal transformer of Section 2.6 is depicted in Figure 5.10 again, using phasor notation.

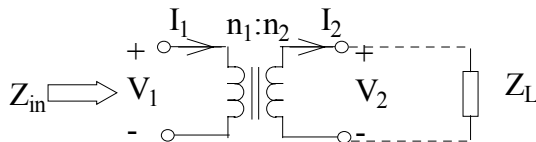


Figure 5.10 Ideal transformer

The primary and secondary voltage and current relations in an ideal transformer are

$$V_2/V_1 = n_2/n_1$$

and

$$I_2/I_1 = n_1/n_2,$$

where  $n_2/n_1$  is the ratio of number of turns in the secondary winding to the number of turns in the primary, which is commonly called the *transformer turns ratio*.

If a load impedance is connected across the secondary terminals, as shown in Figure 5.10, secondary terminal voltage and current must satisfy the relation

$$V_2/I_2 = Z_L.$$

On the primary side, we have

$$Z_{in} = V_1/I_1.$$

Substituting  $V_2$  and  $I_2$  we obtain,

$$Z_{in} = V_1/I_1 = (n_1/n_2)V_2/(n_2/n_1)I_2$$



$$= (n_1/n_2)^2 V_2/I_2$$

$$= (n_1/n_2)^2 Z_L.$$

The load impedance appears as  $(n_1/n_2)^2 Z_L$  across the primary terminals. By choosing appropriate turns ratio, we can convert  $Z_L$  virtually to any value.

### 5.3.2. Real transformers

Transformers are two (or more) coils wound on the same core. A transformer is shown in Figure 5.11.

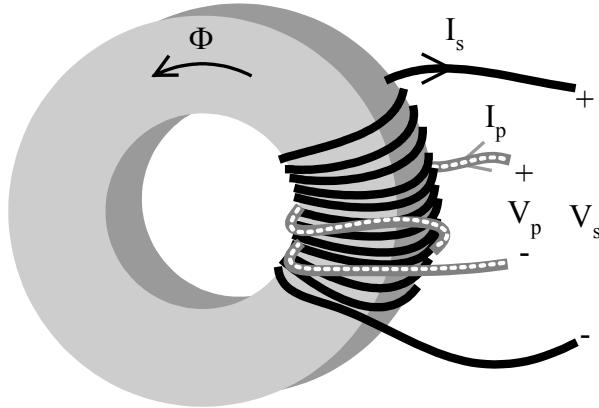


Figure 5.11 Real transformer

The flux generated in the core by two currents  $I_p$  and  $I_s$  is

$$\Phi = n_p A_L I_p - n_s A_L I_s$$

where  $n_p$  and  $n_s$  are the number of turns in primary and secondary windings respectively,  $A_L$  is the inductance constant of the core and  $I_p$  and  $I_s$  are primary and secondary currents, respectively. Both windings are wound in the same sense. If the currents in both windings have the same direction, they generate a flux in the same direction. By convention, we show the *primary current flowing into the transformer* and the *secondary current out of the transformer*. Therefore the total flux generated by two currents is the difference between primary and secondary. Using this equation, we can write  $I_p$  as

$$I_p = (n_s/n_p) I_s + \Phi/n_p A_L.$$

The first term is the relation between the primary and secondary currents in ideal transformer and is called the *transformer current*. The second term is called the *magnetizing current* and it is related to the finite value of the primary inductance (note that this inductance is implicitly assumed infinite in ideal transformer).

The same flux induces voltages across the primary and secondary windings. The induced primary and secondary voltages, on the other hand, are

$$V_p = j\omega n_p \Phi$$

and

$$V_s = j\omega n_s \Phi,$$

as in the case of inductance. Using these relations, we can relate two voltages as

$$V_p/V_s = n_p/n_s.$$

Also substituting  $\Phi = V_p/j\omega n_p$  into expression for  $I_p$ , we obtain

$$I_p = (n_s/n_p)I_s + V_p/j\omega n_p^2 A_L.$$

$n_p^2 A_L$  is the inductance of the primary winding,  $L_p$ . Hence we can write  $I_p$  as

$$I_p = (n_s/n_p)I_s + V_p/j\omega L_p.$$

The two equations relating the primary and secondary terminal voltages and currents can be shown in form of an equivalent circuit comprising an ideal transformer and a parallel inductance as in Figure 5.12. Note that in this model, if the primary inductance is indefinitely large, then its susceptance is zero and it can be removed from the equivalent circuit. The value of  $L_p$  depends on number of turns and the core, and can be calculated as discussed in Chapter 4.

There are two approaches to design RF transformers. If the requirements are such that we need the functions of a transformer in a relatively small bandwidth compared to a center frequency, we design *narrow-band* or *resonant transformers*. The idea is simple: if we tune out  $L_p$  by a parallel capacitor across the primary terminals (or equivalently the secondary terminals), we end up with only an ideal transformer left in the circuit. This is suitable only over a frequency range limited to the  $Q$  of the parallel tuned circuit. In this case the value of  $L_p$  need not be very large, but is chosen to provide the necessary bandwidth.

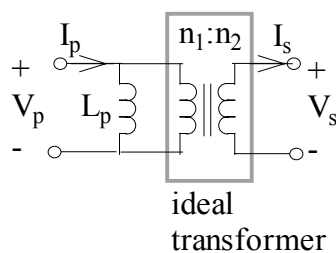


Figure 5.12 Equivalent circuit of a lossless real transformer

Second approach is *wide-band transformer* design. In this case we require that  $|j\omega L_p|$  is significantly larger than the effective impedance that appears across it over the frequency range of interest. Since the transformers are usually employed for converting and matching resistances,  $L_p$  is usually chosen such that its reactance at the lower end of the frequency band is more than *four* times the effective resistance across it. Both narrow-band and wide-band designs are given in Figure 5.13.

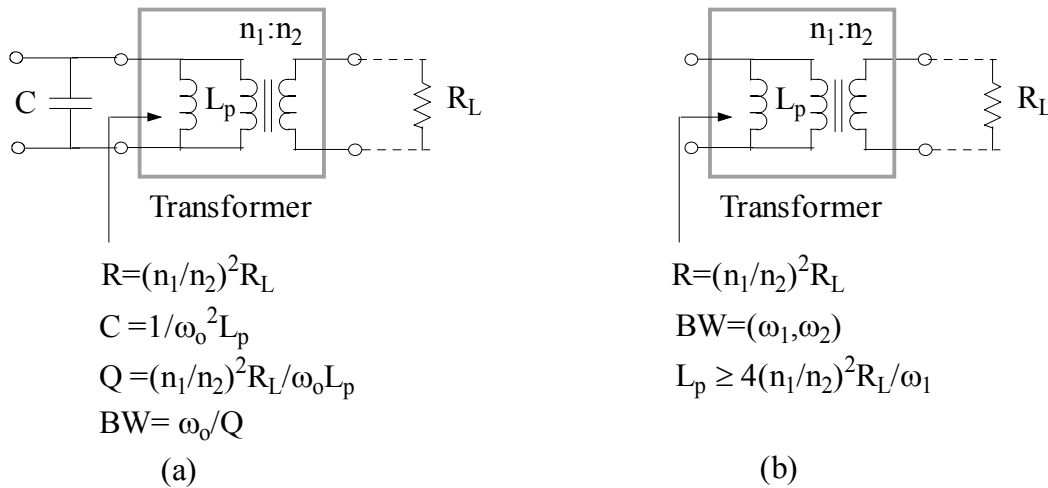


Figure 5.13 (a) Narrow-band transformer, (b) Wide-band transformer

There are two fundamental assumptions in the model of Figure 5.12:

1. There are no core or copper losses;
2. All the flux is confined within the core such that it intersects both coils.

We discussed how losses are handled in inductors in Section 4.4. We include losses related to  $L_p$  in the model similarly. Usually core losses are shown as a parallel equivalent resistance in transformer models and copper losses as series.

Whether the flux is confined in the core or not is unimportant in making inductors. In transformers, however, having a large the portion of the flux shared by both coils is critical. Assuming that all the flux is confined in the core is the equivalent to assuming that the permeability of the core material is infinitely large. However, the cores that can be used at high end of HF are usually of moderate permeability materials. Hence as a coil generates a flux, a large part of it remains in the core but some flux escapes to the surrounding medium. We call this part of flux *leakage*. Leaked flux contributes to the inductance of the generating coil, but it does not contribute to the induced voltage on the other coil. The other coil does not intersect the leaked flux. The part of flux intersected by both coils is called *linkage*. It is due to linkage that the transformer action exists.

Although the equivalent circuit which models flux leakage is more involved, it is possible to reduce it to the equivalent circuit given in Figure 5.14.  $L_p$  is divided into two parts,  $k^2 L_p$  and  $(1-k^2)L_p$ , where  $k^2 L_p$  represents the linkage and  $(1-k^2)L_p$  represents the leakage. The parameter  $k$  is called the *coupling coefficient*. In a transformer with low leakage,  $k$  is close to 1. Note that the coupling coefficient modifies the transformation ratio to  $kn_1/n_2:1$ , which is lower compared to turns ratio  $n_1/n_2:1$ .

$R_s$  and  $R_p$  represent the copper and core losses in Figure 5.14, respectively. All the current that flow into the primary winding experiences the copper losses. Hence  $R_s$ , representing this loss, is in series with the entire circuit. The leakage flux is part of flux, which leaks to the environment and not maintained in the core. The core losses are mainly associated with the linkage, hence  $R_p$  is connected in parallel with the inductance, which represents linkage.

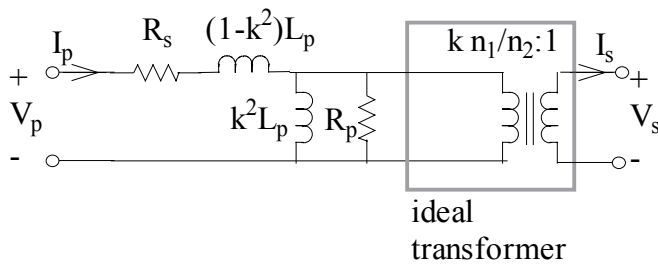


Figure 5.14 Equivalent circuit of a real transformer.

5.3.3. Matching by resonant circuits

The simplest way of narrow band impedance matching is using series RLC circuits. We discussed the equivalence of the series and parallel RLC circuits and amplification property of tuned circuits, in Section 4.3. Same property provides a means of matching. Figure 4.7 is repeated here as Figure 5.15, for convenience.

The resistance  $R_s$  in series with the inductance is transformed into a resistance  $(Q^2+1)R_s$  in parallel with the capacitance, as shown in Figure 5.15. There is a transformation ratio of  $Q^2+1$ . This transformation is valid within the 3 dB bandwidth of the tuned circuit. This impedance transforming LC circuit is called an *L-section*. The same circuit transforms a parallel resistance  $R_p$  into a series one with a value  $R_p/(Q^2+1)$ .

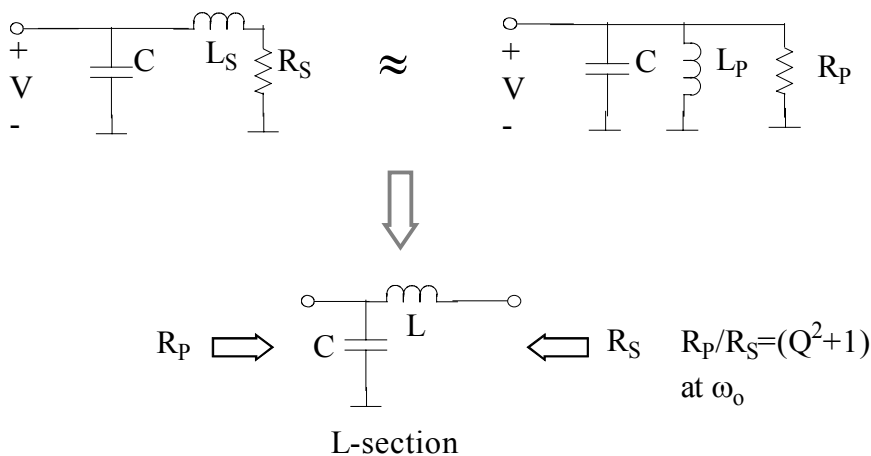


Figure 5.15 Narrow band impedance matching using L-section filters

5.3.4. Impedance inverters

Impedance inverters are narrow-band impedance transformers. An impedance inverter circuit is depicted in Figure 5.16. The circuit has a “T” form with two equal series reactances and a parallel reactance of same magnitude but opposite sign.

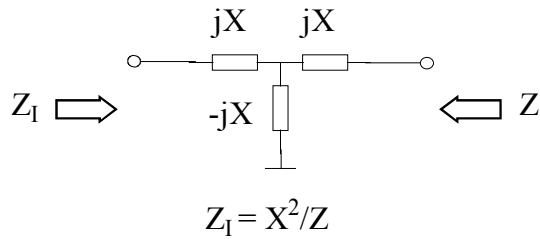


Figure 5.16 Impedance inverter

When an impedance  $Z$  is connected to one end of an inverter, the impedance seen at the other end becomes

$$Z_i = jX + 1/[-1/jX + 1/(jX + Z)]$$

$$= X^2/Z.$$

This useful circuit is used for many matching and filtering purposes. It can convert a series resonant circuit into a parallel resonant circuit. Assume that  $Z$  is the impedance of a series resonant RLC circuit, i.e.

$$Z = j\omega L + R + 1/j\omega C.$$

$Z_i$  becomes

$$Z_i = X^2/(j\omega L + R + 1/j\omega C),$$

and the admittance is

$$Y_i = 1/Z_i = (j\omega L + R + 1/j\omega C)/X^2$$

$$= j\omega L/X^2 + R/X^2 + 1/j\omega CX^2.$$

This is the admittance of a parallel RLC circuit with a *capacitance* of value  $L/X^2$ , *conductance* of  $R/X^2$  and an *inductance* of  $CX^2$ .

#### 5.4. Crystal filters

Tuned circuits made of inductors and capacitors cannot meet very tight frequency selectivity requirements. The  $Q$  of such circuits is limited at about 100. Mechanical devices made of PZT ceramics and quartz crystals are used in such applications. PZT ceramics are *electrostrictive* and quartz crystals are *piezoelectric* materials, respectively. They can convert electric field applied on them into mechanical vibration within their body. Mechanical filter devices are usually made by evaporating electrodes on small plates of these materials. Applying a voltage across these electrodes produces an electric field in the material. Device converts this field into mechanical vibration at the same frequency of the applied voltage. The plate dimensions define various *modes of mechanical resonance*. When the frequency of the applied field matches to one of these resonance frequencies, the impedance that

appears across the electrodes makes a dip. The frequency of resonance is very stable and the losses are very small, limited to frictional activity during particle motion.

#### 5.4.1. Quartz crystals

Quartz crystals have the highest possible Q. A Q value of 100,000 is common in quartz crystals. The circuit symbol of a crystal (XTAL) is given in Figure 5.17(a). There are many modes of resonance in a XTAL and each mode has a fundamental resonance frequency and its *overtones*. We are interested in the fundamental resonance frequency of the only one mode of vibration. The equivalent circuit of the XTAL in the vicinity of this frequency is also given in Figure 5.17(b).

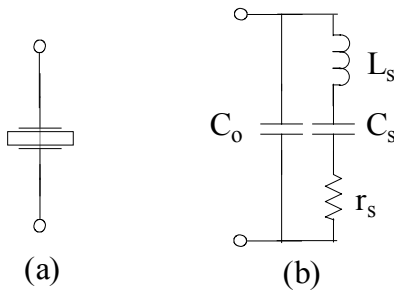


Figure 5.17 Quartz crystal (a) Circuit symbol and (b) Equivalent circuit

This equivalent circuit models the impedance at the (electrical) terminals of the XTAL. The mechanical properties of quartz and the dimensions of the plate, however, determine all three of the series circuit elements.  $L_s$  is proportional to the mass of the plate and  $C_s$  is determined by the *compliance* of quartz. Resistance  $r_s$  models the friction losses during vibration. The only inherently electrical component is  $C_0$ , which is the capacitance between the electrodes.

A *motional inductance*  $L_s$  of about 10 mH can be obtained easily with a small sized quartz plate.

The model in Figure 5.17(b) has two resonance frequencies:

$$f_s = 1/[2\pi(L_s C_s)^{1/2}]$$

and

$$f_p = 1/\{2\pi[L_s C_s C_0/(C_s + C_0)]^{1/2}\}.$$

$f_s$  is the resonance frequency of the series  $r_s L_s C_s$  branch and the impedance decreases down to  $r_s$  at this frequency (in parallel with  $j2\pi f_s C_0$ ).  $f_p$ , on the other hand, is the parallel resonance frequency, where the inductance  $L_s$  resonates with the series combination of  $C_s$  and  $C_0$ , i.e.,  $C_s C_0/(C_s + C_0)$ .

The motional element  $L_s$  is in the range of few mH to over 10 mH for XTALs at 16 MHz.  $C_s$  is in the order of fF (femto Farad,  $10^{-15}$  Farad) and  $r_s$  is in the range of few ohms to few tens of ohms.  $C_0$  is an electrical component, the *clamp capacitance*, and it is usually a few pF. We are interested in the frequencies in the vicinity of series

resonance. At  $f_s$ ,  $|j2\pi f_s C_o|$  is very small compared to  $1/r_s$ . Hence the total terminal impedance is  $Z_s \cong r_s$ . The only effect of  $C_o$  is to shift the series resonance slightly.

Quartz crystals are usually employed as they are supplied, i.e., with the highest possible  $Q$ . Timing circuits in watches, computers, or fixed frequency oscillators in communication equipment require very sharp filter characteristics at a precise resonant frequency.

We use quartz crystals for two different purposes in TRC-10. The 16 MHz clock module is a XTAL oscillator. It is an electronic circuit with an integrated quartz crystal in its case. We discuss this module in Chapter 7. We also use two quartz crystals to make a very narrow band IF filter with center frequency of 16 MHz.

#### 5.4.2. IF ladder filter

A 16 MHz XTAL has a typical bandwidth of about 100-200 Hz. We need 6 KHz BW at IF stage. A single XTAL can provide a limited attenuation of a single element filter (we view it as a BPF derived from a LP equivalent). We use two XTALs to make a Butterworth ladder filter.

Consider the circuit in Figure 5.18. Each XTAL provides a series resonance circuit. We need one series and one parallel resonant circuit, in order to employ techniques we use in designing Butterworth filters. We employ an impedance inverter to invert one of the series resonant circuits into a parallel resonant circuit, as in Section 5.3.4. We choose the external circuit parameters  $X$ ,  $R_1$  and  $R_2$  appropriately, to satisfy the bandwidth requirement.

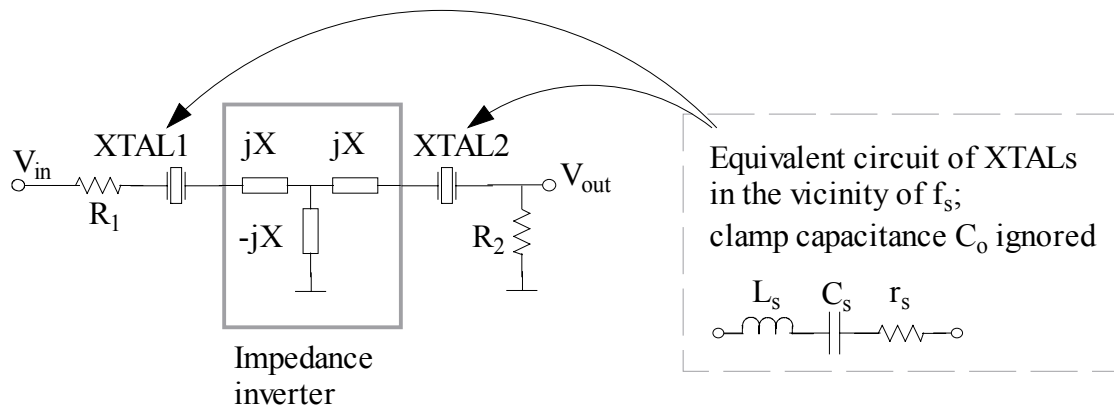


Figure 5.18 Two-crystal ladder filter

The inverter inverts the series circuit provided by XTAL2 and  $R_2$  into a parallel tuned circuit. The equivalent circuit of the resulting ladder filter is shown in Figure 5.19(a).

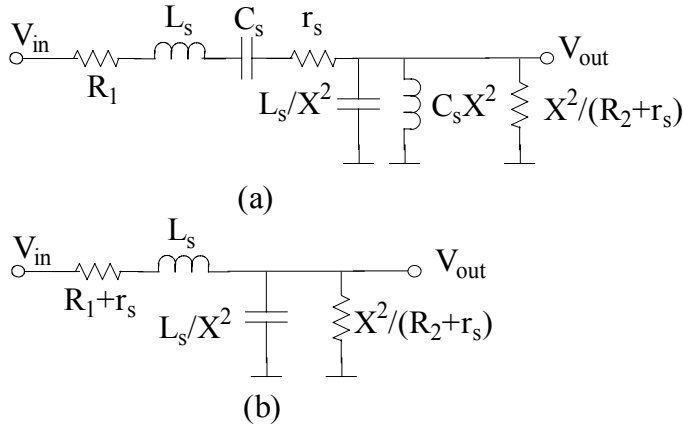


Figure 5.19 (a) Equivalent circuit of two-crystal ladder BPF and (b) LP equivalent of the filter

We must set all circuit parameters such that the filter becomes a second order Butterworth BPF. As we discussed in Section 5.2.4, we first consider the LP equivalent of this filter, which is given in Figure 5.19(b). Hence

$$R_o = R_1 + r_s = X^2 / (R_2 + r_s),$$

$$L_s = 1.4142 R_o / \omega_c,$$

and

$$L_s / X^2 = 1.4142 / \omega_c R_o$$

at the cut-off frequency  $\omega_c$  of the LPF for  $n=2$ , where  $R_o$  is the termination resistance. The cut-off frequency of the LP equivalent is 6 KHz.  $L_s$  and  $C_s$  are motional circuit elements and are fixed. Above relations specify  $X$  and  $R_o$  (hence  $R_1$  and  $R_2$ ) uniquely as

$$R_o = X = R_1 + r_s = R_2 + r_s = \omega_c L_s / 1.4142.$$

Once  $X$ ,  $R_1$  and  $R_2$  are specified as such, the parallel inductance  $C_s X^2$  and series capacitance  $C_s$  tunes out  $L_s / X^2$  and  $L_s$  respectively, producing a BPF with BW of  $\omega_c$  at the series resonance frequency of the XTAL.

A 16 MHz XTAL with  $L_s = 6$  mH requires  $X$ ,  $R_1$  and  $R_2$  to be

$$X = R_o = R_1 + r_s = R_2 + r_s = 160 \Omega,$$

for a BW of 6 KHz.



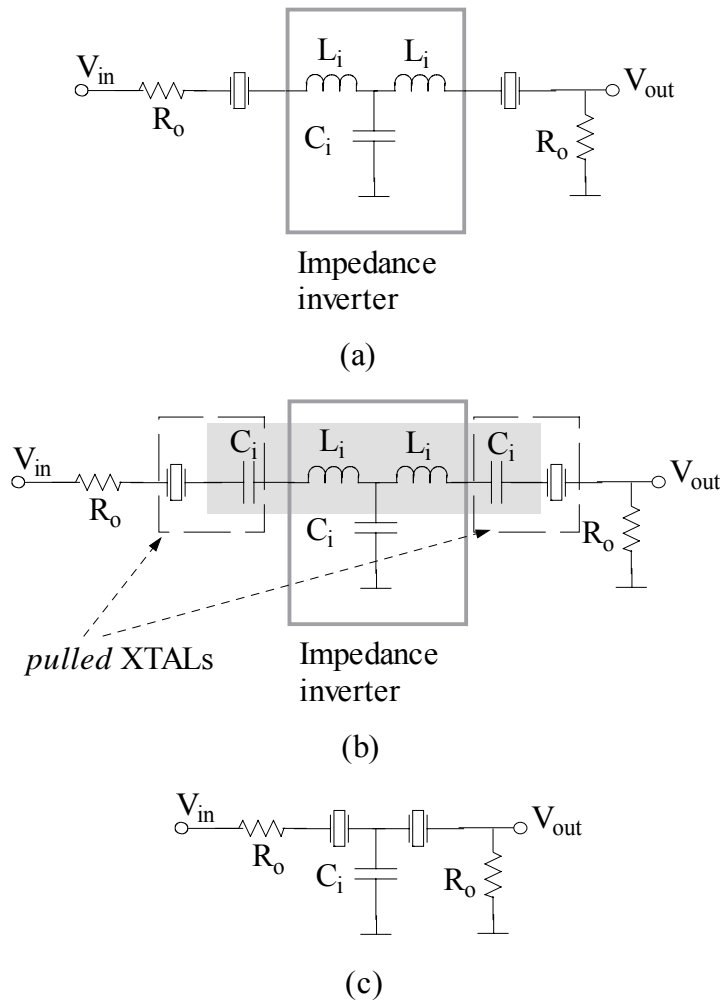


Figure 5.20 Impedance inverter for two-crystal ladder filter, (a) Proper inverter, (b) Inverter inductors tuned out by series capacitors, modifying the series XTAL branches, (c) Actual circuit with only one capacitor

Now let us examine how we shall realize the inverter. The inverter is expected to have a parallel capacitor ( $C_i$ ) and two series inductors ( $L_i$ ), providing  $-jX$  and  $jX$  at  $f_s$ , respectively. The circuit is depicted in Figure 5.20(a). Here the inverter circuit components are related as

$$X = 2\pi f_s L_i = 1/2\pi f_s C_i.$$

However we can tune the series inductors at  $f_s$  by including series capacitors  $C_i$ , into the circuit as shown in Figure 5.20(b). The series capacitor modifies the XTAL circuit only slightly, because the value of this capacitance is always few orders of magnitude larger than the motional capacitance of the XTAL. The effect is a slight increase in the series resonance frequency of the XTAL. This effect is called *pulling the frequency of the XTAL*. We are effectively left with an inverter and two *pulled* XTALs. The center frequency of the filter is now slightly higher than  $f_s$ .

Series connected  $C_i$  and  $L_i$  is simply a short circuit in the vicinity of  $f_s$ . If we replace all series  $C_i$  and  $L_i$  in the shaded area in Figure 5.20(b) by a short circuit, the circuit

behavior is left unchanged. The resulting circuit implementation is given in Figure 5.20(c).

The bandwidth requirements and the XTAL parameters determine the termination resistances. When we use XTAL filters, we must employ appropriate impedance transformation techniques to provide the proper termination to the filter.

### 5.5. Bibliography

There are excellent texts on filter theory and filter implementations. However the treatment of the topic at introductory electronics textbooks is usually very weak. In *The Electronics of Radio* filters are discussed excellently and in a detail, which is not available in other introductory electronics textbooks.

*The ARRL Handbook* has a very good chapter on filters (chapter 16).

There are very good application notes on HF transformer design in Philips Semiconductors and Ferroxcube web sites, [www-us6.semiconductors.com](http://www-us6.semiconductors.com) and [www.ferroxcube.com](http://www.ferroxcube.com), respectively.

A lot of information is available on every aspect of crystals. There is concise application information and data in PTI's web site, [www.piezotech.com](http://www.piezotech.com).

### 5.6. Laboratory Exercises

#### *Harmonic filter*

1. The harmonic filter of TRC-10 is given in Figure 5.21.

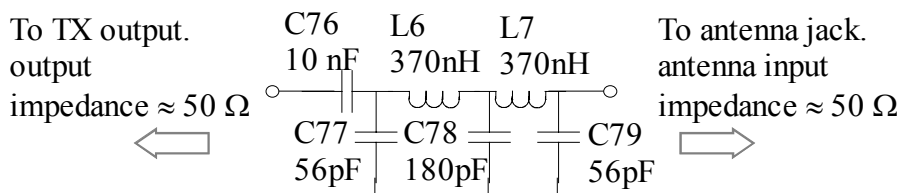


Figure 5.21 Harmonic filter

This filter is designed in Section 5.2.2. C76 is a 10 nF d.c. decoupling capacitor. The inductors of the filter, L6 and L7, are air-core inductors. Measure the diameter of the *coil former*, preferably using a compass. We shall use enameled wire of 0.5 mm diameter. Calculate the number of turns required for 370 nH using the formula given in Section 4.4.1. Make sure that the length of the coil does not exceed the spacing between the connection holes on the PCB. Check your calculations carefully.

Estimate the total wire length necessary for one coil. Add an extra 10 cm for easy manipulation and cut a piece of wire of the total length. Wind the coil of calculated number of turns on the coil former tightly. Stretch the coil to the calculated length, making sure that the coil is in good helical shape and wound

tightly on the coil former. Fix the winding on the coil former using a piece of tape (otherwise it springs out). Trim the leads and strip the enamel. Cover the leads by solder. (You can measure the inductance using LCR meter or a network analyzer. Ask the assistant or the lab technician to help you for measurement.) Using appropriate heat resistant glue, fix the winding, and remove the tape.

This is an air-core inductor and it does not have any core losses. Calculate the copper loss and the Q of the inductor at 29 MHz. Install this inductor as L6 and solder. Make L7 similarly, install and solder. Install C76, C77, C78 and C79 and solder them.

2. Use a very short piece of insulated cable as a jumper to connect the free end of C76 and the attenuator following R61 (better still connect free ends of C76 and 47 $\Omega$  resistor in the attenuator together- the attenuator circuit you constructed in Exercise 4.15), by-passing the TX/RX switching components.

The output section of the measurement circuit is given in Figure 5.22. This position of the cable from antenna jack is the correct position, but we must temporarily connect it using ground plane construction technique for this exercise. We have to remove the PCB several times to mount remaining components and we do not want this cable on our way.

Signal generator input is connected to C64 and the input part of the measurement set up is exactly as in Figure 4.16. Set the signal generator frequency to 28.85MHz and its voltage to the level you recorded in Exercise 7 in Chapter 4.

3. Measure second and third harmonic components. Compare the power of these components to the ones measured without harmonic filter, after making the correction for attenuator. Does the harmonic emission of TRC-10 satisfy regulatory requirements?

Estimate the attenuation provided by the harmonic filter at the second and third harmonic frequencies of 29 MHz. Compare this attenuation with the calculated ones.

4. Switch the power and the equipment off. Disconnect the spectrum analyzer and the signal generator. De-solder RG58/U you connected in Exercise 4.8. **Keep the 51  $\Omega$  and 1.5 K resistors (ground plane constructed) connected to C64.** You need them in TX/RX circuit experiments in Chapter 6.

Install R61 in its place. De-solder the attenuator and the antenna jack cable connection in gray brackets in Figure 5.22. De-solder the jumper. Remove and clean the remaining solder from the PCB, using de-soldering pump.

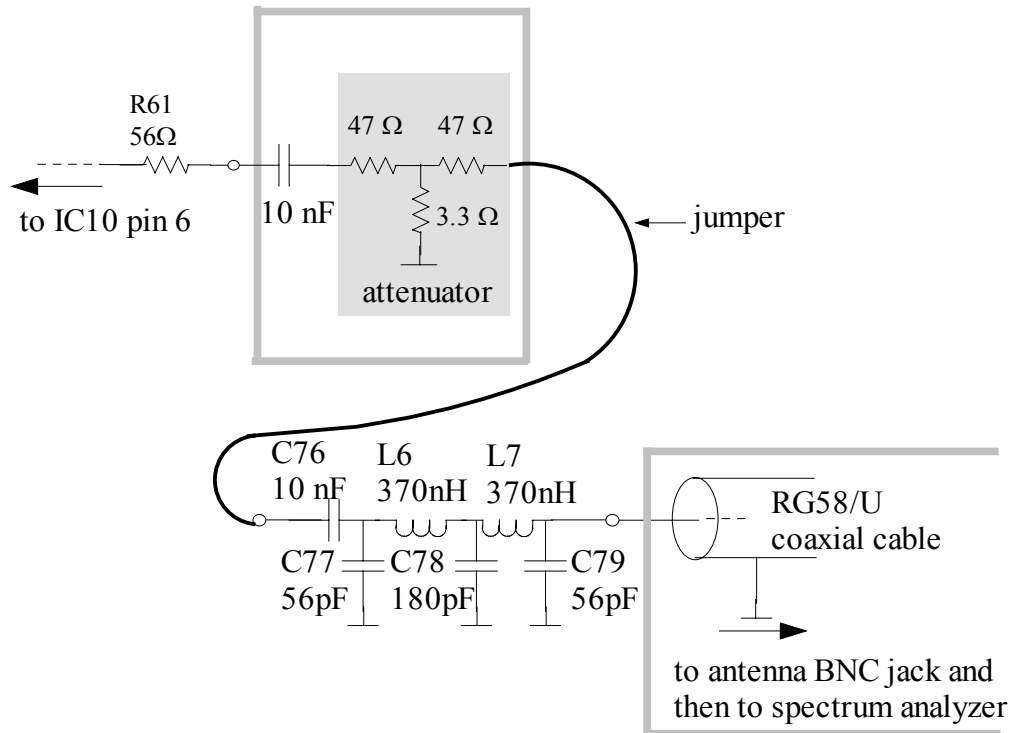


Figure 5.22 Harmonic measurement set up

### Crystals

- IF of TRC-10 is 16 MHz. XTALs at this frequency can be found abundantly in all component suppliers. The tricky thing about such XTALs is, on the other hand, in order to make best use of lower cost and for convenience, we acquire these crystals from local component stores, in this particular case from *Konya Sokak*. Usually the only thing that is known about these XTALs is their nominal series resonance frequency  $f_s$ , and the value of the pulling capacitance that must be used to obtain this frequency. The proper way of acquiring XTALs is to order them to the manufacturer with clearly delineated specifications, in order to get most reliable and repeatable results. This way of acquisition is the most expensive way also.

It is sufficient to have the  $f_s$  of two XTALs within few hundred Hz in TRC-10, since the bandwidth of the IF filter is 6 KHz. We must make sure that both crystals are of same make and come from the same production batch, in order to maintain this. Then their  $f_s$  are within at most few hundred Hz, and this is close enough for our purposes in this application.

We shall measure the  $f_s$  and  $Q$  of the XTAL in this exercise and then characterize the relevant XTAL parameters. The measurement set up is given in Figure 5.23.

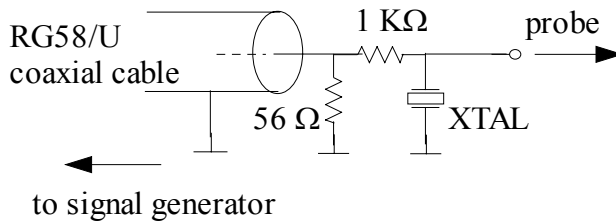


Figure 5.23 XTAL measurement set up

Set up the circuit in Figure 5.23. on a small piece of copper clad PCB, using ground plane construction technique. Connect the cables to the signal generator and the oscilloscope. Switch the power and the equipment on.

1 KΩ resistor is much larger than the expected series resistance of the crystal, which is less than 100 Ω. We use 1 KΩ resistor, to convert the signal source into a current source. We use 56 Ω resistor to terminate the coaxial cable connected to the signal generator. Adjust the signal generator to deliver 2 V pp sinewave at 16 MHz to 56 Ω resistor, and set it up so that you can change the frequency at 1 KHz intervals. Now the crystal is driven approximately by 2 mA pp, or 1 mA p, current at frequencies in the vicinity of  $f_s$ . Scan the frequency range between 15.990 MHz and 16.005 MHz. As we change the frequency we shall observe a minimum on the voltage across the crystal, because the crystal has a series resonance. At the frequency where you observe a dip on the voltage, decrease the scan step to 10 Hz. Find the frequency of minimum voltage. This frequency is  $f_s$ . Measure and record this frequency and this minimum voltage. Now calculate the series resistance  $r_s$ .

Shift the frequency and observe that the voltage increases. Record the frequency on both sides of  $f_s$  where the voltage is  $\sqrt{2}$  times the minimum you recorded. These two frequencies corresponds to the situation where reactance of the crystal is equal to  $r_s$ , hence the difference between them is  $-3$  dB BW. Calculate the series Q of the crystal.

Calculate  $L_s$  and  $C_s$  from  $f_s$ ,  $r_s$  and Q.

Most of the XTALs supplied in the kit for this course have  $L_s$  of about 5.5 mH. The design of the IF filter is based on this of value for  $L_s$ . Your measurement technique is likely to yield a less accurate result for  $r_s$ , which affect the calculated value of  $L_s$ . However, in case if your measurement results deviate from this value of  $L_s$  considerably, first check your measurements and calculations, and then contact the instructor for a discussion of your measurements.

Compare this value of  $L_s$  to other inductances in TRC-10. Calculate the number of turns necessary on T37-7 core to get this inductance value.

Can you think of a similar method to measure  $C_o$ ?

Desolder the cable and the XTAL. Desolder and remove the resistors.

**IF filter**

6. Following the procedure explained in Section 5.4.2, calculate  $R_o$  and  $X$  of the ladder filter, using the value of  $L_s$  you have found. Calculate the inverter capacitance value.
7. The IF filter of TRC-10 is given in Figure 5.24. Start building the filter by placing the crystals on its wide side on the marked rectangle on copper surface. Solder the case, at the top of the crystal, to the copper by using ample amount of heat and solder. This is necessary to keep the large metal surface of the crystal grounded. Otherwise it will be left floating, and will provide a stray path for capacitive coupling of various signals in the TRC-10 circuits to relatively high impedance nodes as unwanted signals.

The termination impedance for this filter is given as  $150 \Omega$ . This is the correct standard valued resistor for XTALs with 5.5 mH equivalent inductance. Also, when 68 pF is used for C100, it yields a reactance of  $150 \Omega$  at 16 MHz. Should the value of C100 be different than 68 pF, the only effect is to have a different termination resistance and hence a different BW and some mismatch.

Calculate the  $R_o$  and  $C_i$  for the XTAL at hand and determine the standard values for R90 and C100. connect and solder C100.

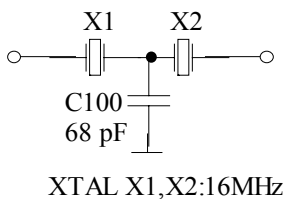


Figure 5.24 IF filter

**IF Transformer**

8. The RX mixer is a SA602A (or SA612A) IC. The output impedance of this chip is approximately 1.5 K. It has two  $180^\circ$  out-of-phase outputs. We use both outputs of this chip constructively to get four times more power.

Consider the equivalent circuit in Figure 5.25(a). In this circuit two out-of-phase outputs of the mixer chip are modeled as two voltage sources  $V_{IFo1}$  and  $V_{IFo2}$ , respectively, with equivalent 1.5 K source resistances.  $V_1$  becomes  $V_{IFo1} - V_{IFo2}$ , and the two out-of-phase outputs are constructively added at the transformer primary terminals.

The other function of the transformer is to convert the total source impedance of 3 K ( $1.5 \text{ K} + 1.5 \text{ K}$ ) to approximately  $150 \Omega$  at its secondary, which is the assumed termination impedance of the IF filter. This requirement sets the transformer ratio to

$$n_1^2/n_2^2 = 3000/150 \Rightarrow n_1/n_2 \approx 4.5.$$

If the termination impedance  $R_o$ , hence  $R_{90}$ , of the IF filter is different than  $150\Omega$ , the turns ratio, and the number of turns, must be chosen accordingly.

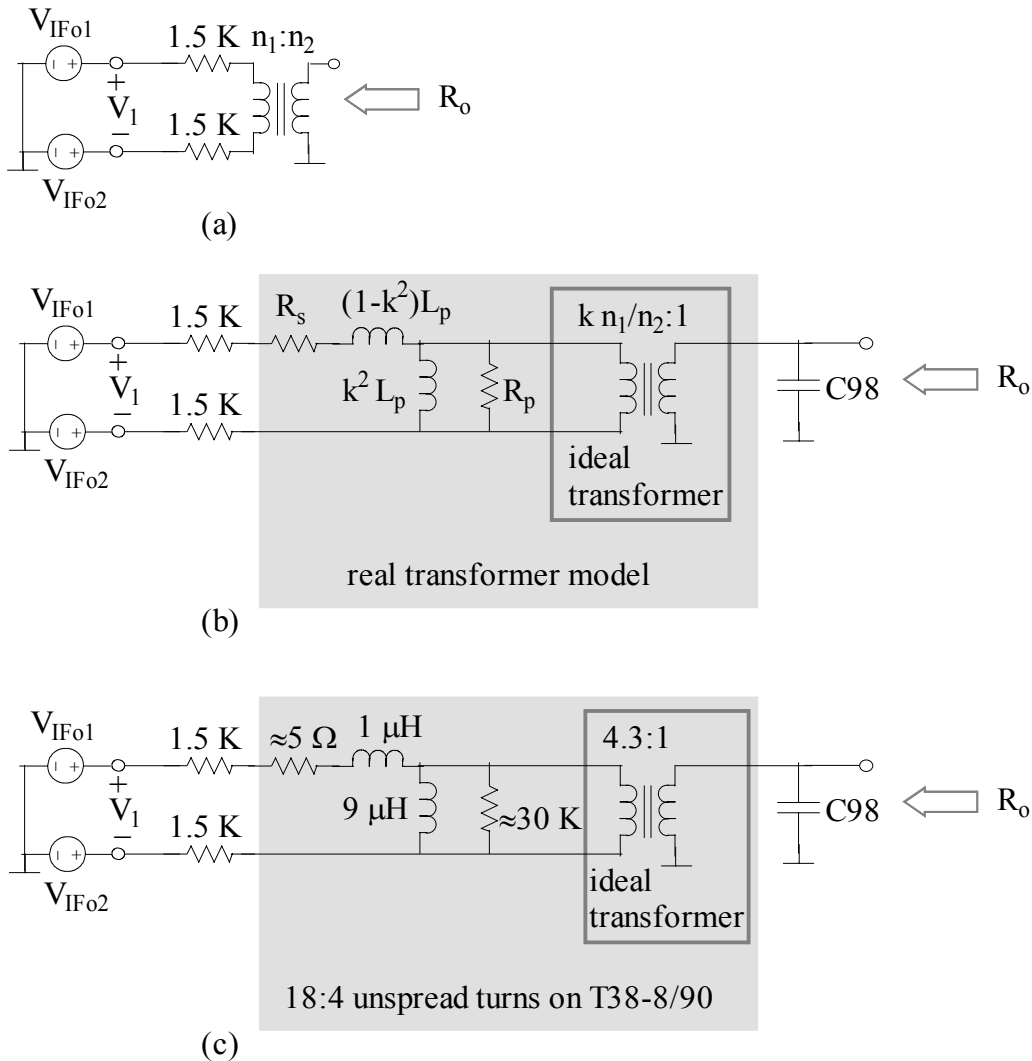


Figure 5.25 Impedance matching between RX mixer and IF filter (a) with an ideal transformer, (b) with a real tuned transformer model and (c) with the typical values of a transformer with 18:4 turns on a T38-8/90 toroid core.

The core for the IF transformer T3 is a T38-8/90 toroid, produced by Micrometals. T38 tells that the outer diameter of the core is 0.38 inch (or 9.53 mm) and 8/90 is the iron powder material type. The core materials for transformer applications must have a permeability as high as possible, in order to keep most of the flux within the core. This material has a relative permeability  $\mu_r$  of 35 and yields an  $A_L$  of  $20 \text{ nH/turn}^2$  with its given geometry. What follows is a discussion of how an IF filter is designed. Every design requires an output specification and it is again assumed that the termination is  $150\Omega$ . For other values all relevant component values must be modified accordingly.

The presence of leakage inductance in a transformer reduces the coupling coefficient (see Section 5.3.2), and therefore reduces the effective transformer

ratio to a value below the winding ratio. The model of a real transformer of Figure 5.14 is repeated in Figure 5.25(b), for convenience.

We shall use a winding ratio of 4.5 as calculated above. Considerations on deciding the number of turns for the secondary (or equivalently the primary) of this transformer are as follows:

- The secondary inductance (hence the number of turns) must be as large as possible in order to keep the  $Q$  of the transformer secondary (with a tuning capacitor) low, consequently to avoid tuning problems;
- The inductance must be low enough to have 16 MHz sufficiently below the self-resonance frequency of the primary inductor  $L_p$  (the larger one).

We use 18 turns for primary and 4 turns for secondary windings. First wind the primary using 40 cm long 0.2 mm enameled wire. Wind 18 tight turns on T38-8/90 toroid. *Do not stretch the winding to cover the toroid*, keep it tight. This is useful to minimize the leakage flux since we have small number of turns in secondary winding. Fix the ends of the winding using a piece of tape. Wind 4 *tight* turns of primary *on* the secondary winding *centered on* it, as a second layer. Again, use 0.5 mm wire. Fix the ends using a tape. Trim all four leads, strip the enamel and cover with solder. Replace the pieces of tape by glue to fix the ends.

The inductance observed at the secondary terminals would be  $4^2 \times 20$  nH, or 320 nH, if the winding is stretched. With an unstretched winding this inductance can be up to twice as high. The value of the capacitor C98 to tune out this inductance must be about 160 pF to 200 pF. The resistance that appears across the secondary, on the other hand, is approximately  $150 \Omega / 150 \Omega = 75 \Omega$ , which yields a  $Q$  of 1.2 to 1.35. Hence the BW of the transformation is very large and the value of C98 is not critical at all. We choose it 180 pF standard capacitor value.

Transformers are characterized by a measurement technique called *short and open circuit tests*. This method is developed for low frequency transformers, but we can adopt a version of it here. We want to find  $k$ ,  $L_p$ ,  $R_s$  and  $R_p$  of our transformer. We first measure the impedance across the primary terminals using an RLC meter or a network analyzer at 16 MHz, *with secondary terminals left open circuit*. Then we short the secondary terminals and measure the impedance again. When we carry out this test on transformers with 18:4 turns on T38-8/90 toroid made as described above, the typical measurements are as follows:

open circuit test at 16 MHz:	10 $\mu$ H in series with about 45 $\Omega$ , as equivalent primary impedance, and
short circuit test at 16 MHz:	1 $\mu$ H in series with 5 $\Omega$ , as equivalent primary impedance.

Applying this test on the transformer model in Figure 5.14 or Figure 5.25(b), short circuit test immediately yields

$$R_s = 5 \Omega \text{ and } (1-k^2)L_p = 1 \mu\text{H}.$$



Solving for  $k^2L_p$  and  $R_p$  is somewhat more complicated. Since the  $Q$  of the primary winding is large (more than 20), the equivalent inductance of the primary impedance in open circuit test is  $L_p$ . In other words, the primary admittance in open circuit test is

$$\begin{aligned} Z_p &= R_s + j\omega_{IF}(1-k^2)L_p + 1/[1/R_p + 1/(j\omega_{IF} k^2L_p)] \\ &\approx R_s + R_p / [(R_p/\omega_{IF} k^2L_p)^2 + 1] + j\omega_{IF} L_p \\ &\Rightarrow L_p = 10 \mu\text{H} \end{aligned}$$

where  $\omega_{IF}$  is the IF frequency. The equivalent series resistance of  $45 \Omega$  in open circuit test cannot be measured accurately, since the series reactance to it is very large. However the above relation gives an indication of a quite large  $R_p$  of about  $30 \text{ K}$ .

Coupling coefficient can be found as  $k = 0.95$ . Hence the transformer ratio  $k n_1/n_2:1$  of the ideal transformer in the model becomes  $4.3:1$ . The equivalent circuit parameters of a typical IF transformer is given in Figure 5.25(c).

It is interesting to note that when a  $3 \text{ K}$  resistance is connected to the primary of such a transformer as in Figure 5.25(c), the impedance across the secondary terminals is measured as  $150 \Omega$  in parallel with  $510 \text{ nH}$ . This is very close to what the equivalent circuit predicts. The decrease in transformer ratio combines with the effect of the presence of  $R_p$  to keep the equivalent secondary resistance near  $150 \Omega$ .

Install the transformer T3, paying attention to the correct placement of primary and secondary pair of leads. Solder the leads. Install and solder the tuning capacitor C98, and two d.c. blocking capacitors C94 and C95.

9. The transformer T3 matches the output impedance of the mixer to  $R_o$  of the IF filter. The other terminal of the filter is terminated by R90. Install and solder R90.
10. Install IC12, an LM7171 for the first IF amplifier. Carefully check the pin configuration and the placement details on the PCB. Solder IC12. Install and solder the supply by-pass capacitors and feedback resistors, C102, C103, R91 and R92, respectively. Check all the connections, both visually and using a multimeter (as ohmmeter for connectivity), carefully.

Switch the power on. Using a multimeter check all d.c. voltages at the supplies, input and output terminals. Connecting R90 between the amplifier input and ground provides a d.c. path for the input bias current of the OPAMP. Carry out the d.c. analysis of this amplifier referring to the data sheet, and calculate the maximum d.c. off-set voltage that we may expect at the output of IC12.

Switch the power off.

11. The circuit part comprising IF transformer, IF filter, and IF1 amplifier (first IF amplifier) is given in Figure 5.26. In order to test this circuit, we simulate the balanced the mixer output by the circuit shown in gray bracket. Two  $1.5 \text{ K}$  resistors represent the output impedance of two output terminals of the mixer.

56 Ω terminates the signal generator cable and signal generator voltage simulates  $V_1$  in Figure 5.25.

- Mount and solder two 1.5 K resistors and the 56 Ω resistor, using ground plane construction technique. Solder the free end of coaxial cable with BNC connector for the signal generator connection. Solder a 56 Ω resistor between pin 6 of IC12 and ground plane using ground plane construction technique. Connect the oscilloscope probe to the pin 6 of IC12. Switch the equipment on.

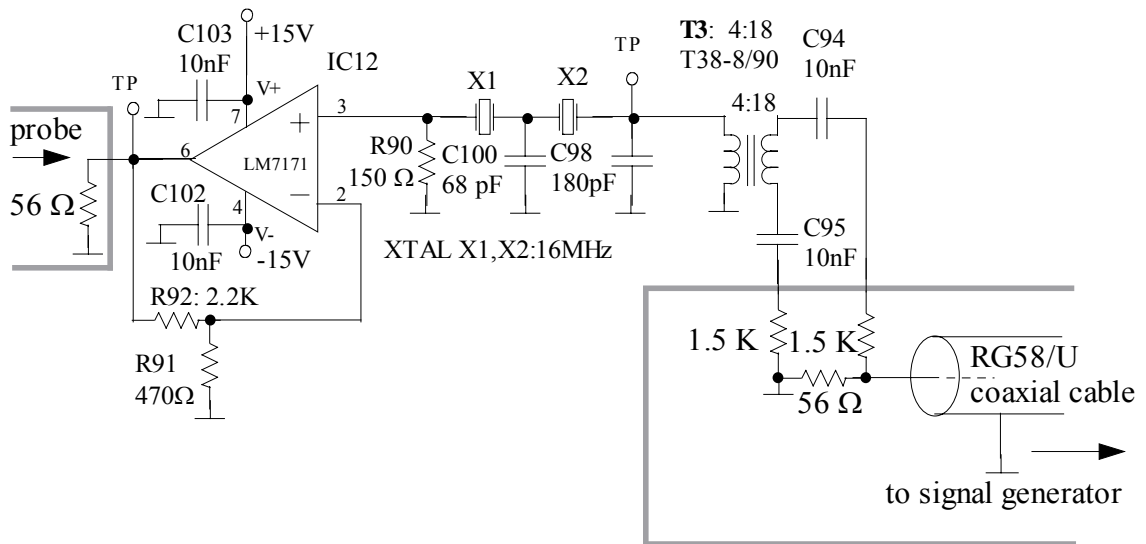


Figure 5.26 IF stage measurement set up

Set the signal generator output to -7 dBm or 0.1 V rms sine wave. Vary the frequency over the pass-band of IF filter around 16 MHz and, find and set it to the frequency where the amplitude of the amplifier output is maximized. This frequency is the center frequency of IF filter. Record it. Measure the magnitude of the peak output voltage amplitude  $V_{out1}$  and record it.

- The approximate equivalent circuit of this part at the center frequency of the IF filter is given in Figure 5.27. Following the method described in Section 4.5.1 and referring to the data sheet of LM7171, first determine the open loop gain of the OPAMP at 16 MHz and then calculate the voltage and power gain of the circuit in Figure 5.27.

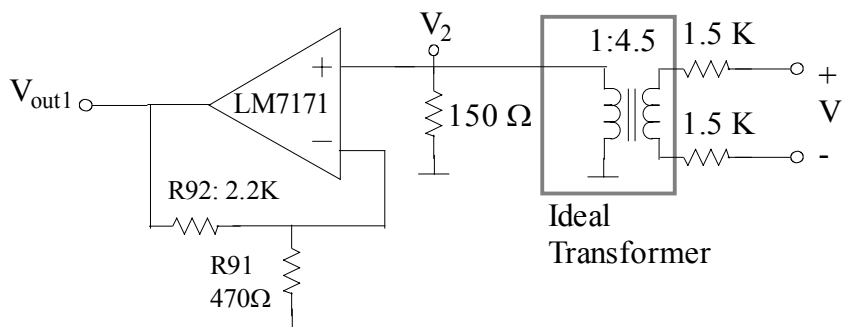


Figure 5.27 Equivalent circuit of IF stage at IF frequency

14. Set the signal generator output voltage amplitude to 1 V rms. Measure the peak to peak output voltage,  $V_{out1}$ , in a frequency range of 100KHz centered at passband. Calculate the loss

$$L = 20\log | V_{out1}/V_1 | \text{ dB}$$

at each frequency, where you can take  $V_1$  as the value displayed on the signal generator (take care to convert rms value to peak value). Plot this loss versus frequency. Vertical axis must be scaled so that it can accommodate 60 dB range. The number of measurements and the frequencies where they are taken must be such that the filter pass-band and stop-band are properly characterised. Find the the -3 dB frequencies and the bandwidth of the IF filter.

Switch the power and equipment off. Desolder and remove the 56  $\Omega$  resistor connected to pin 6 of IC12. Remove the excess solder from the copper surface using your desoldering pump.

15. We need another inductor, L8, for the second IF amplifier, IF2. We use a T50-7 Micrometals toroid core to make L8. You already know what T50-7 means. Find the properties of this core from the data sheet. Calculate the number of turns necessary for an inductance of approximately 8.4  $\mu\text{H}$ . It must be about 38. We use 0.2 mm thick enameled wire to wind the inductor. Calculate the length of wire you must use. Cut that length of wire and make an excellent inductor. **Do not forget to spread the winding evenly around the core.** All the data provided assumes this. Install and solder L8.

L8 and L10 have approximately same inductance. Since the ferrite core has a large inductance constant, we could get away with fewer turns compared to the iron powder core of L8. The price to pay for the convenience is higher core loss.

Install and solder R93 and C104. C104 is a trimmer capacitor. Make sure that you place it such that ground pin corresponds to adjustment pin.

Install IC13, another LM7171. Carefully check the pin configuration and the placement details on the PCB. Solder IC13. Install and solder the supply by-pass capacitors and feedback resistors, C105, C106, R94 and R95, respectively. Check all the connections, both visually and using a multimeter (as ohmmeter for connectivity), carefully.

Install and solder a 100  $\Omega$  resistor between the pin 6 of IC13 and copper surface using ground plane construction technique.

16. The series RLC circuit comprising R93, L8 and C104 is a reasonably high Q circuit. We must tune C104. First calculate the Q. Calculate the expected voltage and power gain between pin 6 of IC12 and pin 6 of IC13, in dB. Record your calculations and their results.

17. Switch the power on. Using a multimeter check all d.c. voltages at the supplies, input and output terminals of IF2 amplifier stage. What is the d.c. path for the bias currents of this OPAMP?

Connect the probe of Channel 1 of the oscilloscope to the pin 6 of IC12 and Channel 2 to pin 6 of IC13. **Make sure that the input circuit is exactly as the same as in Exercise 12 and the input voltage level is set to 0.1 V<sub>rms</sub>, as in Exercise 12**, and the signal generator is connected. Switch the equipment on.

Observing the waveform on Channel 1, make sure that the signal generator frequency is set at the center frequency of the IF filter. Adjust the frequency, if necessary. Decrease the input signal amplitude, if you observe distortion.

Adjust C104 until the amplitude of the signal on Channel 2 is maximized, using the alignment tool. Decrease the input signal amplitude further, if you observe distortion. IF2 amplifier is tuned.

Measure the voltage gain between pin 6 of IC12 and pin 6 of IC13 and record it. Compare it with your calculation. Can you comment on the discrepancies on a quantitative basis?

Measure the overall voltage gain between the signal generator output and IF2 output. Convert it into dB and record it. Calculate the power gain in dB using this measurement and record it.

Switch the power and the equipment off.

Desolder and remove the coaxial cable at the input. Desolder and remove the 100 Ω resistor at the output of IF2. Remove the excess solder from the copper surface using your desoldering pump.

You need the two 1.5 K resistors and the 56 Ω resistor in the exercises of Chapter 6. Leave them on the PCB.

## 5.7. Problems

- Using a spread sheet and using the component values calculated in Section 5.2.1, calculate and plot the transfer response  $20\log|H(\omega)|$  of filters in Figure 5.6 from 1 rad/sec to  $10^8$  rad/sec.
- Find the amplitude and phase of the voltage at the output of the circuit given in Figure 5.28 when input voltage is  $2 \cos(2\pi \times 16 \times 10^6 \times t)$  volts. Find the amplitude of the current flowing through the capacitor.

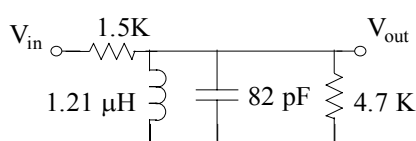


Figure 5.28

- Design a 3<sup>rd</sup> order Butterworth LP filter (i.e. find the values of all reactive components) with minimum number of inductors, whose cutoff frequency is 3 KHz and termination impedance are 1 K.
- If  $V = V_p \angle \theta$  is the phasor of voltage across a resistor R, we know that the power delivered to that resistor is  $V_p^2/(2R)$ , or  $|V|^2/(2R)$ . It is given in Section 5.1 that the transfer function of the circuit in Figure 5.2(b) is

$$V_{out}/V_{in} = H_2(\omega) = 1/\{2[1-(\omega/\omega_c)^2 + j\sqrt{2}\omega/\omega_c]\},$$

For  $R_S=R_L=R$  and for maximally flat response. Now since the available power at the input of this circuit is  $|V_{in}|^2/(8R)$ , show that the ratio of the power delivered to the load  $P_L$  to the available power  $P_O$  is

$$P_L/P_O = 1 / [1+(\omega/\omega_c)^{2 \times 2}],$$

i.e. the power transfer function of a second order Butterworth filter

- Find the amplitude and phase of the voltage at the output of the circuit given in Figure 5.29 when input current is  $10 \cos(1.82 \times 10^8 \times t)$  mA. Find the amplitude of the current flowing through the inductor.

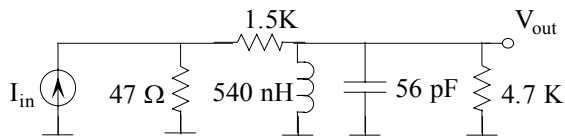


Figure 5.29

- Find the 3-dB cut of frequency of the LPF given in Figure 5.30.

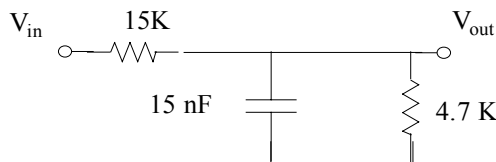


Figure 5.30

- Design a 5<sup>th</sup> order Butterworth LP filter with minimum number of inductors, whose cutoff frequency is 3 KHz and termination impedance are 1 K.
- Design a 3<sup>rd</sup> order Butterworth HP filter with minimum number of inductors, whose cutoff frequency is 3 MHz and termination impedance are 75  $\Omega$ .
- Design a Butterworth LP filter which has an attenuation of at most 1 dB at 16 MHz and an attenuation of at least 20 dB at 32 MHz (i.e. find the minimum number of components, find reactive element values for a filter with at minimum number of inductors, and finally check the attenuation at specified frequencies for your filter).

10. Design a Butterworth BPF with center frequency of 30 MHz and a 3-dB bandwidth of 3 MHz, such that its attenuation at 60 MHz is at least 20 dB.
11. Assume we have an *ideal* transformer with a primary/secondary winding ratio of 1:2. What is the impedance that appears across the primary if a 300  $\Omega$  is connected across the secondary.
12. Assume we wind up a very good *real* transformer with no loss and no leakage with 1:2 primary/secondary winding ratio. The primary inductance is 1  $\mu\text{H}$ . Find the impedance across the primary at 10 MHz when 300  $\Omega$  is connected across the secondary. What is the phase shift between the primary voltage and primary current?
13. Calculate the *transformer current* and the *magnetizing current* for the transformer and load given in Problem 12 when the transformer is driven by a voltage of  $2\angle 0^\circ$  volts at 10 MHz.
14. A real transformer has a primary inductance of 1  $\mu\text{H}$  and a primary/secondary turns ratio of 4:24. Assuming that the transformer has no loss and no leakage, calculate the impedance across the primary at 30 MHz when a 1.8 K load is connected across the secondary. What is the phase shift between the primary voltage and primary current?
15. Find the value of parallel capacitance required across the primary to tune out the primary inductance at 30 MHz, for the transformer and load given in Problem 14. Find the impedance across the primary at 30 MHz. Calculate the phase shift between the primary voltage and primary current when this capacitance is connected across the primary.
16. Find the impedance across the terminals of a XTAL at the parallel resonance frequency. The circuit in Figure 5.17(b) is redrawn below in Figure 5.31 for convenience. (Hint: First find the impedance across series circuit formed by  $L_s$  and  $r_s$  at resonance, and then evaluate the effect of divider formed by  $C_s$  and  $C_o$ .)

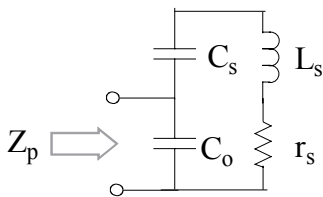


Figure 5.31 Quartz crystal equivalent circuit

17. A 16 MHz crystal is measured to have  $f_s = 15,992,875$  Hz, the parallel resonance frequency  $f_p = 16,024,600$  Hz and peak conductance at  $f_s$   $1/r_s = 135$  mS. The Q of the series resonance is found as 112,000. Calculate  $C_s$ ,  $L_s$ ,  $r_s$  and  $C_o$ .
18. Assume that a 16 pF *pulling* capacitor is connected to the XTAL in problem 17 in series. Calculate the *pulled* frequency.

19. A 16 MHz crystal is measured and the G and B vs frequency plots shown in Figure 5.32 are obtained. Find the equivalent series inductance, capacitance and loss resistance for this XTAL. Calculate  $R_o$  and  $C_i$  of the IF filter if a pair of XTAL's with this characteristics are used. Determine the turns ratio for T3 and the values (standard) of R90, C100 and C98 for this filter. (Ans: 4:15; 180  $\Omega$ ; 47 pF; 180 pF)

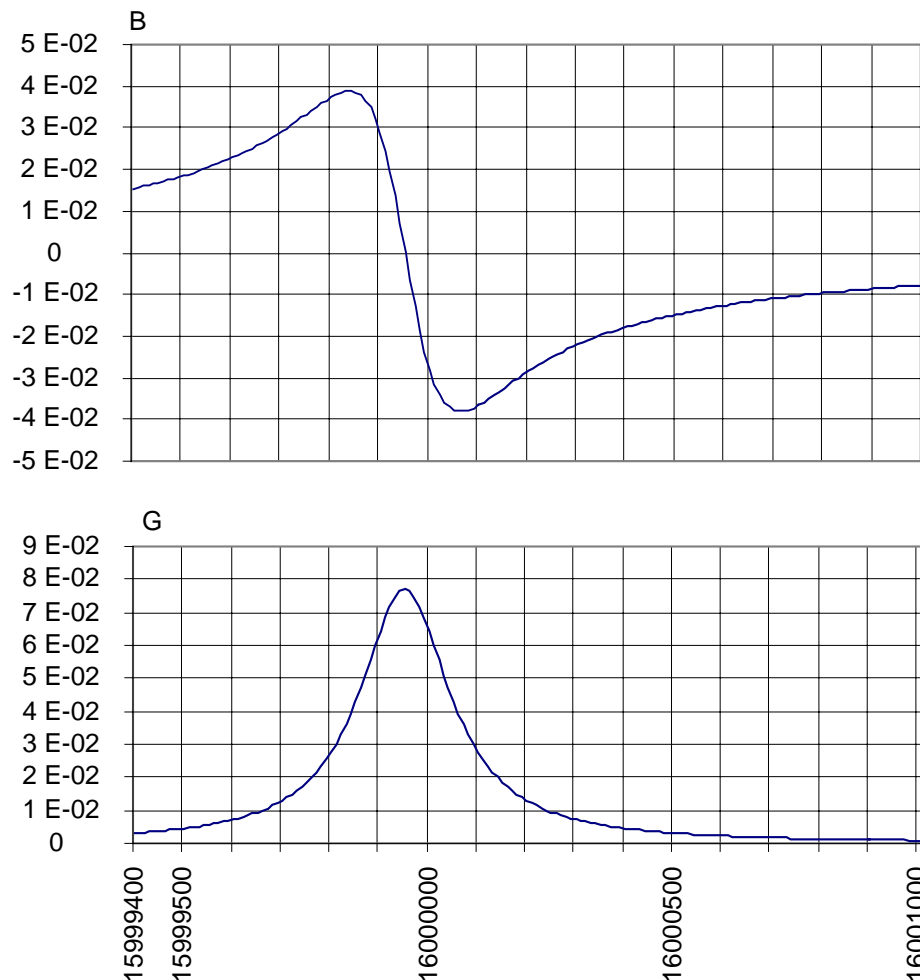


Figure 5.32 XTAL susceptance and conductance vs frequency: vertical axis- siemens; horizontal axis- Hz.

20. Consider the circuit given in Figure 5.28. The voltage across this circuit can be expressed as

$$v_{out}(t) = V_{dc} + V_1 \cos(\omega_0 t + \theta_1) + V_2 \cos(2\omega_0 t + \theta_2) + V_3 \cos(3\omega_0 t + \theta_3) + V_4 \cos(4\omega_0 t + \theta_4) + V_5 \cos(5\omega_0 t + \theta_5) + V_6 \cos(6\omega_0 t + \theta_6) + \dots$$

if  $v_{in}(t)$  is a square wave signal of 2 V pp amplitude, as shown in Figure 1.2, with period  $T = 0.0625 \mu\text{sec}$ . Calculate the amplitude and phase of the first six terms in  $v_{out}(t)$ . How much deviation is there in  $v_{out}(t)$  from a sinusoid?

## Chapter 6 : DIODES IN TELECOMMUNICATIONS

We used diodes to rectify the a.c. voltage and convert it to a d.c. supply voltage in Chapter 2. We employed the *self-operated switch* property of diodes in that application. When the potential across the diode exceeds the threshold voltage, diode becomes almost a short circuit. Otherwise it remains open. This useful property of diodes is exploited in many applications in telecommunications electronics.

### 6.1. Diode detector

The AM signal at the output of the second IF amplifier is

$$v_{IF}(t) = V_{IF} [1 + m(t)] \cos(\omega_{IF} t).$$

The waveform of this signal is given in Figure 1.5(a) (where  $m(t)$  is a sinusoid) and its spectrum for a general modulating signal  $m(t)$  is given in Figure 1.12(b). AM demodulation is separating the information signal  $m(t)$  from  $\cos(\omega_{IF} t)$ . The simplest and oldest method of doing this is called *envelope detection*. In an envelope detector, the signal is first *half wave* rectified and then low pass filtered. We employ the rectification property of diodes in envelope detection.

A half wave rectifier using an ideal diode and its function on  $m(t)$  is shown in Figure 6.1(a). The information signal is assumed to be a sinusoidal test signal

$$m(t) = (V_m/V_{IF})\cos(\omega_m t).$$

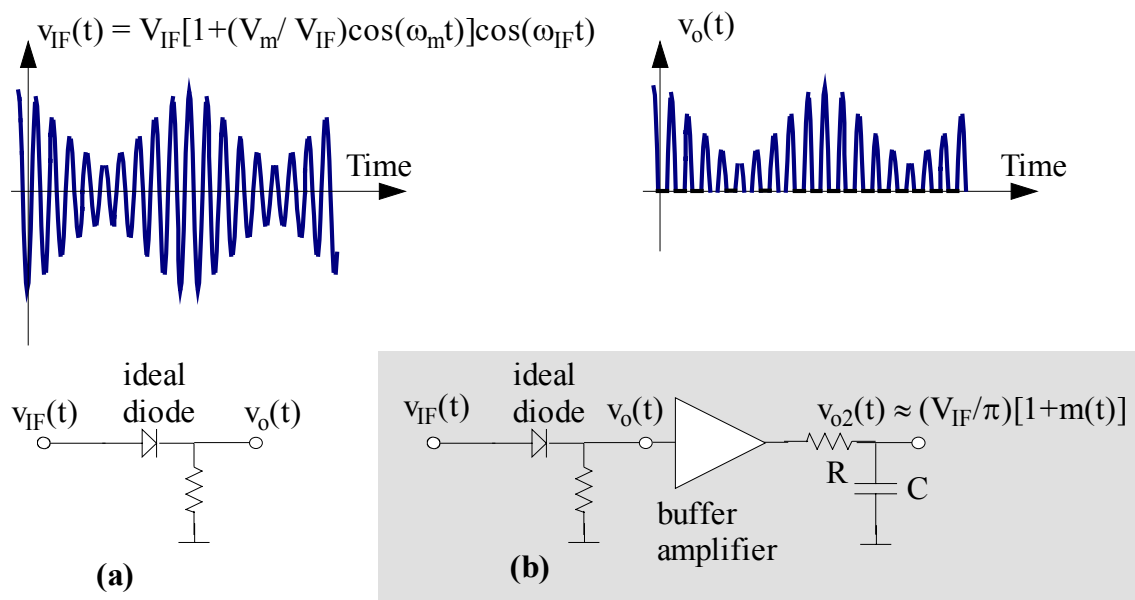


Figure 6.1 (a) Half wave rectifier and waveforms and (b) RC LPF following the rectifier

The IF frequency is always much larger than the modulating signal frequency. Diode rectifies the AM signal and only positive half cycles appear across the resistor. Note



that the mechanism is similar to power rectification problem in Chapter 2, except the amplitude varies with respect to time in this case.

Half wave rectified signal can be written as

$$v_o(t) = V_{IF} [1 + m(t)] \{[\cos(\omega_{IF} t) + |\cos(\omega_{IF} t)|]/2\}.$$

$$s(t) = [\cos(\omega_{IF} t) + |\cos(\omega_{IF} t)|]/2$$

is the mathematical expression for half wave rectified sine wave. The shape of  $s(t)$  is similar to the one in Figure 2.17. This waveform can also be represented as a linear combination of sinusoids, like the square wave in Section 1.2, as

$$\begin{aligned} s(t) &= 1/\pi + (1/2)\cos(\omega_{IF} t) + (2/3\pi)\cos(2\omega_{IF} t) - \\ &\quad (2/15\pi)\cos(4\omega_{IF} t) + (2/35\pi)\cos(6\omega_{IF} t) - \dots \\ &= a_0 + a_1\cos(\omega_{IF} t) + \sum_{n=1}^{\infty} a_{2n}\cos(2n\omega_{IF} t) \end{aligned}$$

where  $a_0$  is the average value of  $s(t)$  and  $a_1$  is the coefficient of the fundamental component. The harmonics in the summation are only even harmonics in this signal. When we substitute this expression for  $s(t)$ ,  $v_o(t)$  becomes

$$v_o(t) = (V_{IF}/\pi)[1 + m(t)] + V_{IF} [1 + m(t)] \times \{a_1\cos(\omega_{IF} t) + \sum_{n=1}^{\infty} a_{2n}\cos(2n\omega_{IF} t)\}$$

Spectral structure of  $v_{IF}(t)$  and  $v_o(t)$  are given in Figure 6.2, for  $m(t) = (V_m/V_{IF})\cos(\omega_m t)$ . Note that the first term in  $v_o(t)$  is  $(V_{IF}/\pi)[1 + m(t)]$  and it is the lowest frequency component in the summation. This term is the information signal (plus a d.c. term) and all we must do is to filter out all the other terms using a LPF.

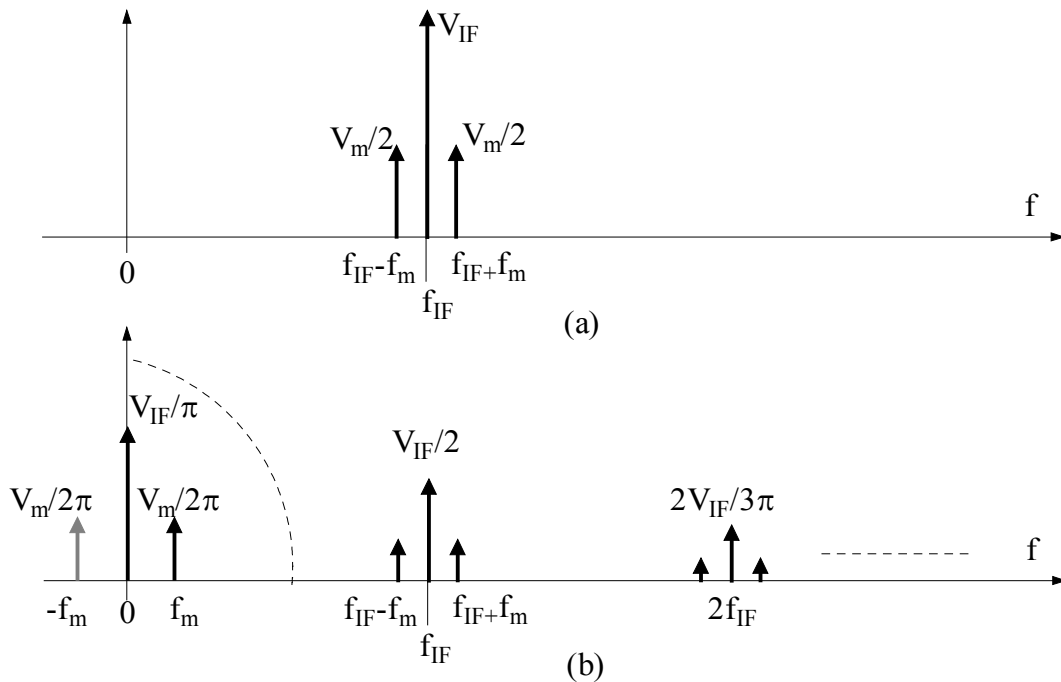


Figure 6.2 Frequency spectrum of (a)  $v_{IF}(t)$  and (b)  $v_o(t)$

Figure 6.1(b) shows how  $(V_{IF}/\pi)[1 + m(t)]$  can be obtained from  $v_o(t)$ . The amplifier serves as a buffer in this circuit and isolates the capacitor of the BPF from the diode. The rectified voltage  $v_o(t)$  appears intact at the output of the amplifier. Amplifier output is like a voltage source and hence  $v_o(t)$  is fed into the RC LPF. With a properly chosen cut-off frequency, LPF eliminates all frequency components except the low frequency ones. This is an *ideal* envelope detector configuration.

The simple envelope detectors are not implemented using a buffer amplifier, in practice. The capacitor of the LPF is connected directly across the diode as shown in Figure 6.3. Without a buffer amplifier, the capacitor loads the diode. When diode is *on*, the capacitor charges up through the *on* resistance of the diode. When it is *off*, the capacitor discharges through parallel resistance  $R$ . The time constants in two cases are different. This situation arises because of the nonlinear nature of the diode.

Voltage across the capacitor  $v_o(t)$  follows the envelope with some ripple as in the case of power rectification. Here we are not allowed to increase the capacitance, hence the time constant  $RC$ , indefinitely.  $RC$  must be chosen such that the capacitor can discharge fast enough and its voltage can follow the minimum slope of the envelope,  $-V_m\omega_m$  (V/sec).

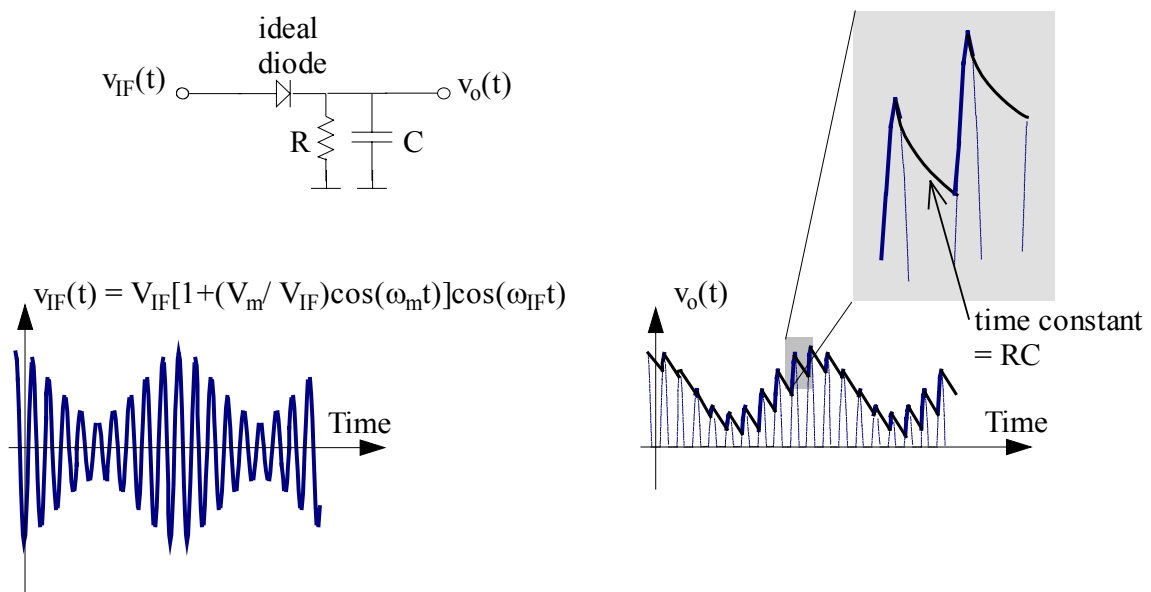


Figure 6.3 Envelope detector

Otherwise the detected envelope signal suffers from what is known as *failure-to-follow distortion* or *diagonal distortion*. This upper limit on  $RC$  causes ripple on the detected waveform, particularly during the up-sloping phases of the envelope, in envelope detectors. This is shown in the shaded box in Figure 6.3.

#### 6.1.1. Real diodes in envelope detectors

We use a fast signal diode 1N4448 in the envelope detector of TRC-10. 1N4448 is a low cost silicon diode, which is very commonly used in low power applications. Real diodes have threshold voltage  $V_o$ , which has adverse effects on the detected signal.

Assume that we replace the ideal diode in Figure 6.1 with a real diode, like 1N4448. To keep the discussion simple, let us model the diode by piece-wise linear model of Figure 2.16(a). The operation of the rectifier of Figure 6.1 with a real diode is shown in Figure 6.4.

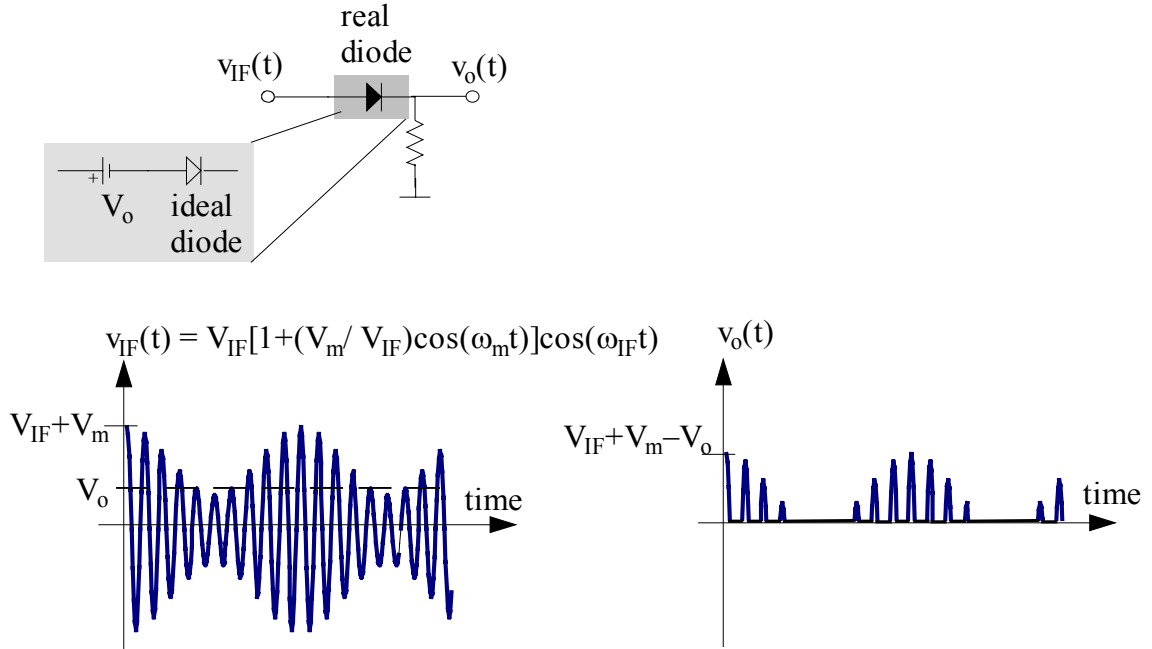


Figure 6.4 Rectifier with a real diode

The diode conducts only after the voltage across its terminals exceeds the threshold voltage  $V_o$ .  $V_o$  can be taken as 0.6 volts approximately for 1N4448, in which case the detected envelope of a  $v_{IF}(t)$  of, for example, 2 V pp amplitude is severely distorted. This is a very important limitation, which is illustrated in Figure 6.4.

We overcome this problem in electronics by few precautions. Consider the circuit in Figure 6.5(a). The d.c. current source sets the average diode current to  $I_{dc}$  at all times. The diode is *on* even when  $v_{IF}(t)$  is zero. This also means that the d.c. voltage across the capacitor  $v_o(t)$  is  $-V_o$ . The output voltage  $v_o(t)$  can follow the envelope down to very low positive input voltage levels.

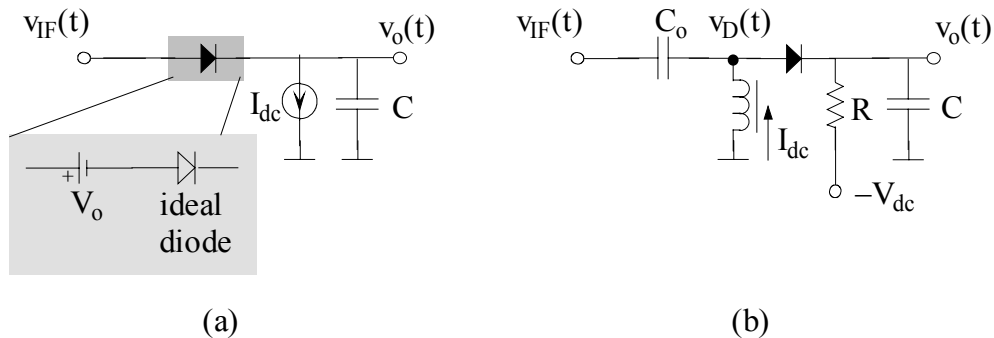


Figure 6.5 Envelope detector with a real diode compensated for  $V_o$ ; (a) equivalent circuit and (b) implementation

This preconditioning of a diode by forcing a d.c. current to flow through it is called *biasing*.

We implement this solution in electronics by the circuit given in Figure 6.5(b). Here the d.c. current source is implemented by a large resistor  $R$  connected to a negative voltage supply,  $-V_{dc}$ . The value of  $I_{dc}$  is  $(V_{dc}-V_o)/R$ . The return path of the d.c. current is the inductor, keeping the d.c. component of  $v_D(t)$  at zero. As far as d.c. circuit is concerned, we must maintain that the d.c. current  $(V_{dc}-V_o)/R$  is insensitive to variations of other parameters in the circuit, such as  $V_o$  and the *on* resistance of the diode. This requirement enforces the choice of a large  $R$  and  $V_{dc} \gg V_o$ . The a.c. circuit must experience a high impedance at this node (the current source terminal impedance is infinite), which also means that  $R$  must be large compared to other prevailing a.c. circuit impedance.

We employ a *large* capacitor  $C_o$  in series with the diode to block the d.c. component of the diode current.  $C_o$  presents virtually zero impedance to  $v_{IF}(t)$  and hence  $v_{IF}(t)$  appears directly across the *large* inductor. All high frequency components of diode current pass through  $C_o$ . Inductor is virtually open circuit for all frequency components at and above  $\omega_{IF}$ .

## 6.2. TX/RX switch

The communication mode of TRC-10 is *simplex*, which means that we use the same frequency channel both for transmission and for reception. During the transmission, TX output amplifier must be connected to the antenna, and the frequency channel must be occupied by our transmitter signal. The antenna must be switched to the receiver input while listening. The TX/RX switch in TRC-10 does this electronic switching.

The switching property of diodes is enhanced when they are used with appropriate biasing. Consider the circuit in Figure 6.6.

Assume that  $v_{in}(t)$  is the RF signal that must be switched on and off the load  $R_o$ . The current source is a d.c. source, and it can take on two levels,  $I_{dc}$  and  $-I_{dc}$ . When we want to switch the diode *off*, we set the current source to  $-I_{dc}$ . The  $I_b$  is set to  $I_{dc}$  in order to switch the diode *on*.

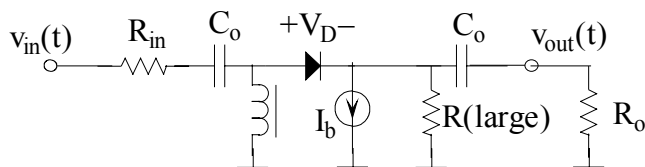


Figure 6.6 A diode switch circuit

The two capacitors block the d.c. current, and forces it to flow either through the diode and the inductor, or through the large resistor  $R$ . On the other hand, the capacitors exhibit low impedance path to RF signals. The inductor provides a short

circuit path for the d.c. current to ground, hence keeping the anode of the diode at zero d.c. potential at all times.

The circuit behavior for  $I_b = -I_{dc}$  is shown in Figure 6.7(a). the current direction is opposite to that of diode, and the only path it can pass is through the large resistor R.  $I_{dc}$  develops a potential of  $I_{dc}R$  across R, which reverse-biases the diode. The diode voltage is

$$v_D(t) = v_{in}(t) - I_{dc}R.$$

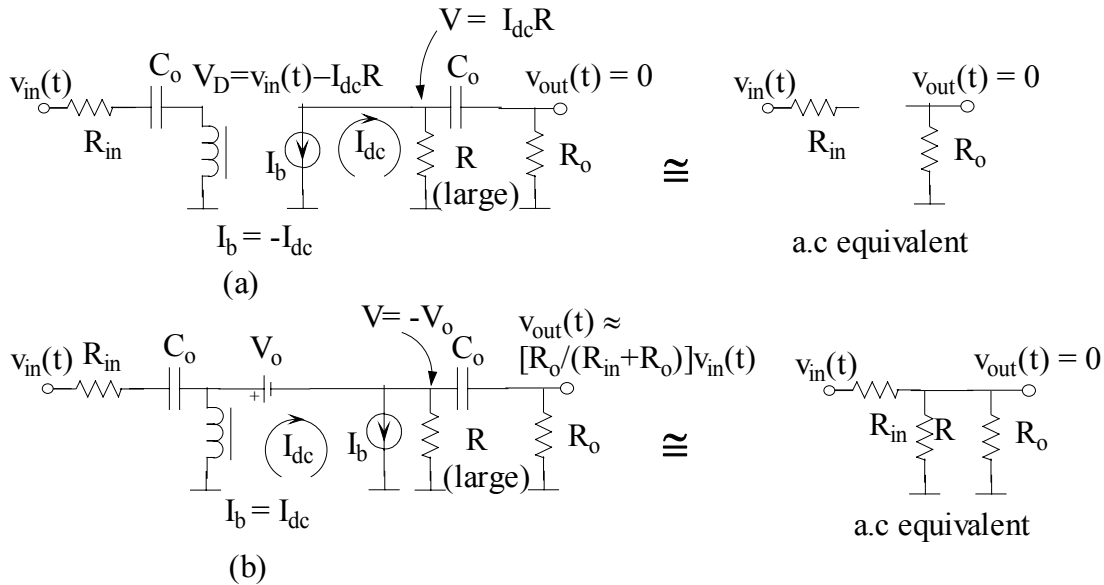


Figure 6.7 Diode switch circuit (a) when  $I_b = -I_{dc}$  and (b) when  $I_b = I_{dc}$

As long as the amplitude of  $v_{in}(t)$  does not exceed  $I_{dc}R$ , the diode is OFF and  $v_{out}(t) \approx 0$ . The value of  $I_{dc}$  and R must be determined to satisfy this condition.

When  $I_b = I_{dc}$  the d.c. current flows through low impedance path of the inductor in series with the diode. The diode is turned on and the input voltage is switched on to load. Diode remains on as long as the minimum current (negative)  $\{[v_{in}(t) - V_o]_{min}\} / (R_{in} + R_o)$  is less than  $I_{dc}$ .

### 6.2.1. Diode biasing

Diode V-I characteristics is useful to discuss the current bias and diode switching on a quantitative basis. Consider a generic case of  $R_{in} = R_o = 50 \Omega$  and the power that must be delivered to the load is 100 mW, in the circuit depicted in Figure 6.6. Assuming that  $v_{in}(t)$  is sinusoidal (this is always a good assumption for narrow band signals like the signals in TRC-10), the amplitude of  $v_{in}(t)$  becomes 6.3 V, i.e.

$$v_{in}(t) = (6.3 \text{ V})\cos(\omega t).$$

In the following example, we shall first find the required bias current  $I_{dc}$  by making certain assumptions and then check whether these assumptions are correct, with  $v_{in}(t)$  as input .

The switching diode is 1N4448. Assuming that the piecewise linear model is valid, the RF current  $i(t)$  through the diode is

$$i(t) = v_{in}(t)/(R_{in}+R_o) = (63 \text{ mA})\cos(\omega t).$$

The total diode current becomes

$$i_D(t) = I_{dc} + i(t) = I_{dc} + (63 \text{ mA})\cos(\omega t).$$

Peak-to-peak swing of  $i_D(t)$  is  $2 \times 63 \text{ mA} = 126 \text{ mA}$ .  $I_{dc}$  must be such that  $i_D(t)$  is always positive. This sets a minimum value of 63 mA for  $I_{dc}$ . Let us choose  $I_{dc} = 65 \text{ mA}$ .

We can apply this current on the V-I characteristics of 1N4448 to find the diode voltage  $v_D(t)$  and output voltage  $v_{out}(t)$ , as shown in Figure 6.8.

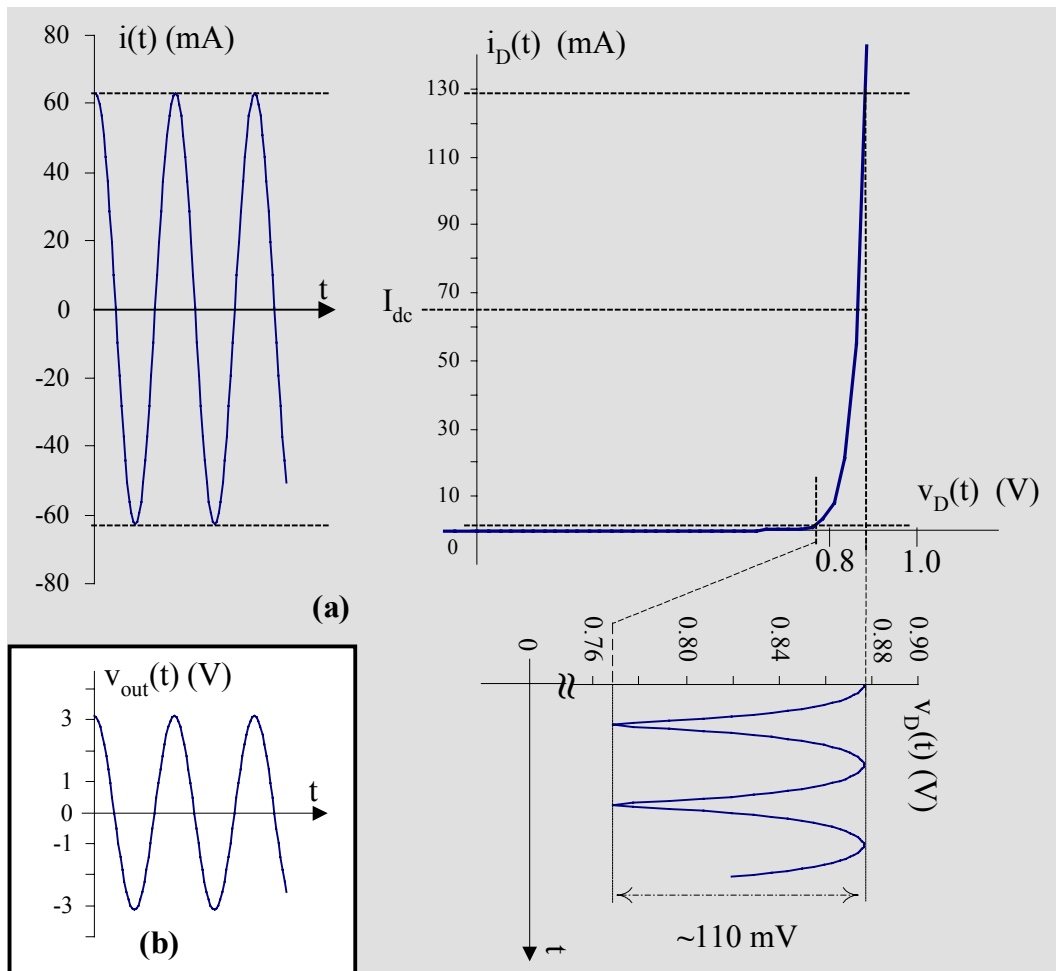


Figure 6.8 Diode biasing (a) diode characteristics and diode voltage and (b) output signal

Once  $I_{dc}$  biases the diode so that the average current through it is 65 mA, whatever the other parameters may be, the diode operates with  $i_D(t)$  confined in a range between 2 mA and 128 mA. This is delineated in Figure 6.8(a). This input current yields a

diode voltage variation between 0.77 V and 0.88 V. The  $v_D(t)$  waveform is severely distorted.

However, if we calculate  $v_{out}(t)$  as

$$v_{out}(t) = v_{in}(t) - i(t)R_{in} - v_D(t)$$

at every point in time, we obtain the waveform shown in Figure 6.8(b). This signal looks like a perfect sine wave, because the distortion in it is limited to the distortion in  $v_D(t)$ .  $v_D(t)$  is a 110 mV p-p signal. The p-p amplitude of  $v_o(t)$  is 6.25 V, which is almost the same as found from *assumed* RF current  $i(t)$ ,  $v_o(t) = i(t) \times R_o = (3.15 \text{ V})\cos(\omega t)$ . The diode and circuit functions like an ideal switch.

It is obvious that the equivalent circuit in Figure 6.7(b) is a very good one. It models the biased diode switch operation perfectly.

### 6.2.2. PIN diode switch

1N4448 performs very well as a switch, as we have seen in the example of Section 6.2.1. Let us now examine the *power budget* of this switch. The RF power delivered to the load is 100 mW. Another 100 mW is dissipated on the source resistance. Hence the switch handles a total of 200 mW RF power.

On the other hand, 1N4448 requires 65 mA bias current to operate. The threshold voltage  $V_o$  of 1N4448 at this average current level is about 0.88 V, as can be seen in Figure 6.8(a). 65 mA passing through 0.88 V makes 57 mW power dissipation.

We are dissipating 57 mW to switch 100 mW of RF power! This 57 mW dissipation can grow to a significantly higher level when the implementation requirements of the current source  $I_b$  is considered.

A better approach is to use *PIN diodes* instead of signal diodes. “PIN” refers to the semiconductor structure of the diode. These diodes have three layers of semiconductor material, **P**-type/**I**ntrinsic (un-doped)/**N**-type, as opposed to two layer **P**/**N** of other diodes like 1N4448.

PIN diodes cannot be switched on and off quickly. Applying a forward voltage or a bias current activates the semiconductor current carrying mechanism in all diodes. When this voltage or current is removed diode stops conduction. PIN diodes require certain amount of time before this current carrying mechanism stops.

MPN3404 (or BA479) is a PIN diode. The datasheet is given in the Appendix. The *minority carrier life-time* of such PIN diodes is few microseconds (4  $\mu$ seconds for BA479). This means that while the diode is on, if we reverse the current or the voltage across it for a brief period sufficiently less than this life-time, diode does not turn off. It remains in conduction. Half a period at 30 MHz is  $1/60 \mu$ s, which is very short compared to minority carrier life-time.

This property of PIN diodes allows us to use them with small bias currents, in power switching applications.

TRC-10 uses two PIN diodes in a configuration shown in Figure 6.9.

R56 is the output resistance of TX output amplifier. C72, C75 and C76 are d.c. blocking capacitors, which isolate the switching circuit d.c. wise and provide a low impedance (virtually zero impedance) path for the RF signals.

S2 is a DPDT manual switch. It has a pair of switches, with independent contacts, operating in parallel. When the center tap of the first switch, S2/1, is connected to +15 V supply, TRC-10 transmits. S2/1 is the one shown in Figure 6.9. The other one, S2/2, is in the 16 MHz oscillator circuit. When the center tap of S2/1 is connected to -15 V, TRC-10 receives. The functions of S2 are:

- During reception:
  1. D8 is turned ON and D7 is turned OFF, antenna is connected to RX;
  2. 16 MHz oscillator module (discussed in Chapter 7) is turned OFF by shutting down its power supply;
- During transmission:
  1. D7 is turned ON and D8 is turned OFF, antenna is connected to TX;
  2. 16 MHz oscillator module is turned ON.

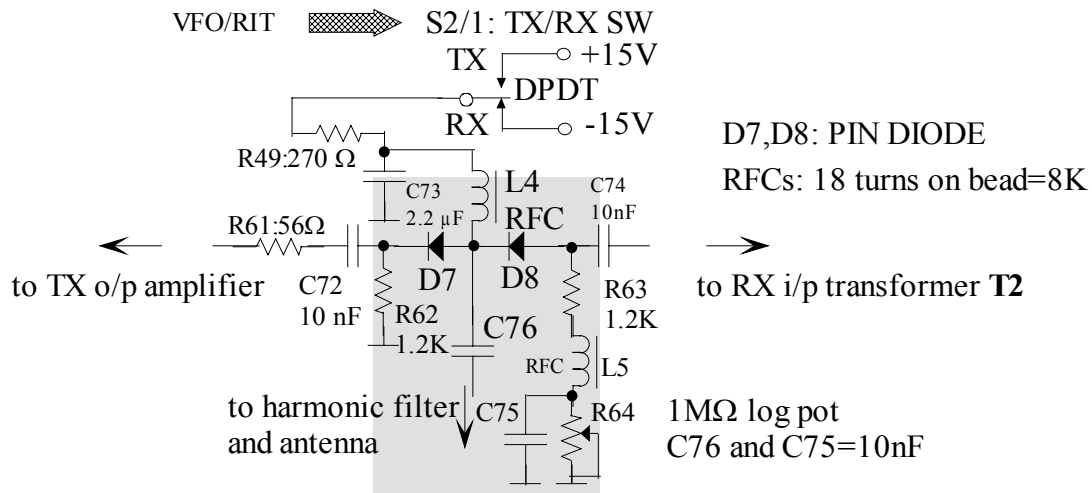


Figure 6.9 TX/RX switch circuit

The part of the circuit shaded in gray has two functions. One function is the *RF gain* adjustment performed by 1 M potentiometer R64. The other function is TX/RX RF switch. RF gain adjustment is discussed in Section 6.2.2. For the following discussion of RF switch function, assume that the center tap of R64 is all the way up and RFC L5 is grounded.

Figure 6.10 gives a clarified view of RF switch circuit.



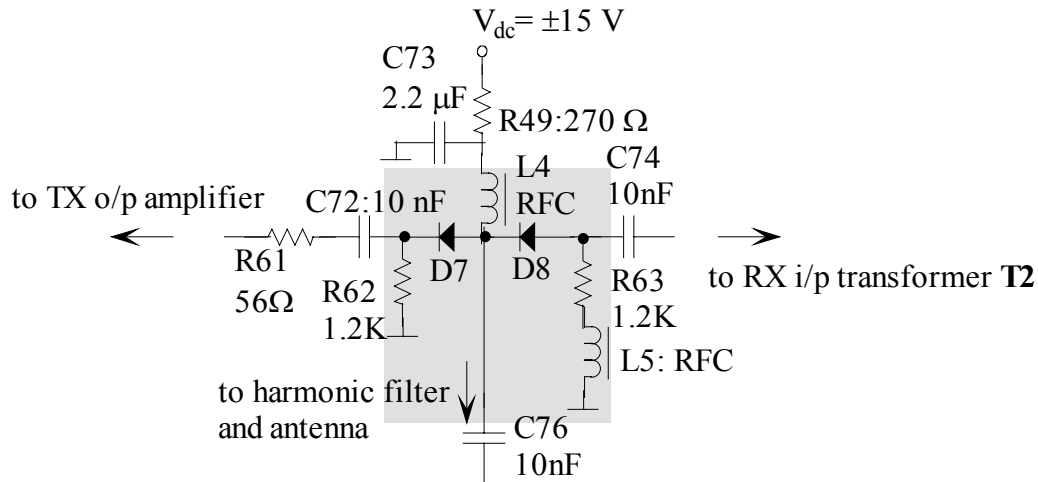


Figure 6.10 TX/RX RF switch circuit

Few mA of bias current is sufficient to keep the PIN diode in conduction. However bias current in diodes has another effect: we can control the value of the ON resistance  $R_D$  of the diode by changing the bias current. The variation of  $R_D$  with respect to bias current is given in the data sheet of MPN3404. ON resistance of the diode is only  $0.7 \Omega$  at  $10 \text{ mA}$  forward current ( $5 \Omega$  for BA479).

For good switching of circuits with termination resistances of about  $50 \Omega$  as in our case,  $R_D$  must be as low as possible. Otherwise a significant voltage drop occurs on the diode. We choose the minimum, approximately  $1 \Omega$  at  $10 \text{ mA}$  of bias current.

When  $V_{dc}$  is set to  $+15 \text{ V}$ , d.c. bias current flows through R49, L4, D7 and R62. L4 is an inductor and short circuit d.c. wise. Similarly C73 is open circuit for the bias current. Hence the bias current is approximately  $(+15 \text{ V})/(R49+R62)$ , or  $10 \text{ mA}$ , ignoring the turn-on voltage  $V_o$  of D7. This current sets the ON resistance of D7 to  $1 \Omega$  and The RF signal at TX output amplifier is connected to harmonic filter through a  $1 \Omega$  resistor.

The d.c. voltage at the node between D7 and D8 is now  $(+15 \text{ V} - V_o) \times [R62/(R49+R62)] + V_o$ , or  $12.4 \text{ V}$ . D8 is completely reverse biased and its only effect in the circuit is a small *junction capacitance* that appears between its terminals. This capacitance is given in the data sheet as  $0.5 \text{ pF}$  approximately. The RX input is effectively isolated from the TX output, which is helpful to protect and avoid saturation at RX mixer input.

When transmitters are switched on to the antenna abruptly, the instant power rise causes emissions at frequencies other than the intended one. This is a kind of spurious emission, which lasts for only few tens of microseconds. They produce clicking sounds at the output of the receivers of other receivers operating at other bands. C73 and R49 are included into the circuit to slow down the initial build up of bias current when TRC-10 is switched to TX. When  $V_{dc}$  is switched to  $+15 \text{ V}$ , C73 charges up from  $-15 \text{ V}$  with a time constant of  $R49 \times C73 = 0.6 \text{ ms}$ . Bias current rises up to  $10 \text{ mA}$  in about  $2.4 \text{ ms}$  (four times the time constant), hence allowing the emitted signal power to increase gradually.

When  $V_{dc}$  is set to  $-15$  V, d.c. bias current flows through L5, R63, D8, L4 and R49. The bias current through D8 is 10 mA, again. The RF signal at the antenna is connected to RX input via a  $5 \Omega$  resistor. D7 is reverse biased and TX output is isolated from the antenna and RX, in this case.

### 6.2.3. RF gain control

Wireless telecommunications have many aspects. One of the implications of the word “tele” is that the communicating parties can be very far away from each other in some occasions and they can be very close to each other (just next door), in others. This type of use imposes a strong requirement on receivers, related to received signal strength. The received signal power from a nearby TX and a far away TX can differ by 80-90 dB, while still remaining within the limits of a receiver. A wireless receiver must be able to cope with received signals of such diverse strength.

Attenuating the strong RF signals just after the antenna is one of the measures to this end. TRC-10 employs a manual RF gain control (rather attenuator). This attenuator makes use of the controllable  $R_D$  property of PIN diode D8, and the potentiometer R64. RF gain control circuit is shown in Figure 6.11.

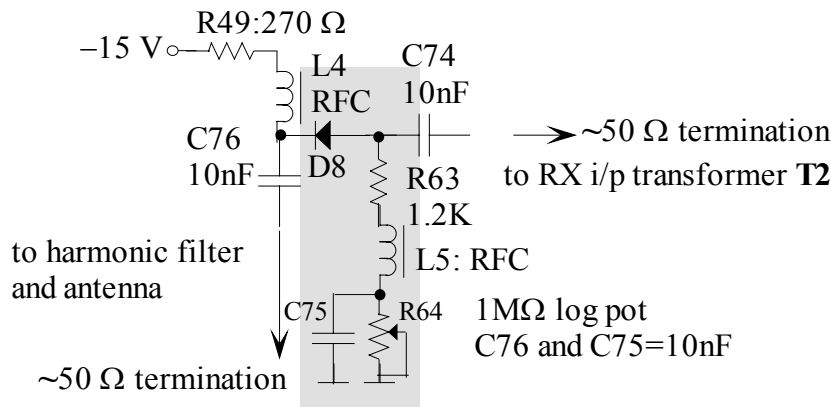


Figure 6.11 RX RF attenuator

C75 is a by-pass capacitor, which provides a very low impedance path for RF signals and by-passes R64 for RF. C75 keeps R64 always out of the RF path, which is particularly useful to decrease the TX signal break through during transmission.

C75 is open circuit for d.c. 1 M potentiometer is in the bias current path during reception. Bias current of D8 during reception is

$$(15 \text{ V} - V_o) / (R64 + R63 + R_D + R49),$$

where  $V_o$  and  $R_D$  are the threshold voltage and the on resistance of D8, respectively.

When R64 is at its minimum, which is zero ohms, the analysis of the circuit is as given in Section 6.2.2. Bias current is about 10 mA and  $R_D$  is about  $1 \Omega$ .

When  $R_{64}$  is at its maximum, 1 M, the bias current is approximately  $(15 \text{ V} - V_o)/(1\text{M})$ , or  $14 \mu\text{A}$ .  $R_D$  at this forward current level is large for PIN diodes, in the order of 10K.

$L5$  is added in series with  $R_{63}$  to increase the RF impedance of the current bias path. Both  $L4$  and  $L5$  provide impedance of about 8 K at 16 MHz.

With above considerations, the RF equivalent circuit the attenuator during reception is given in Figure 6.12.

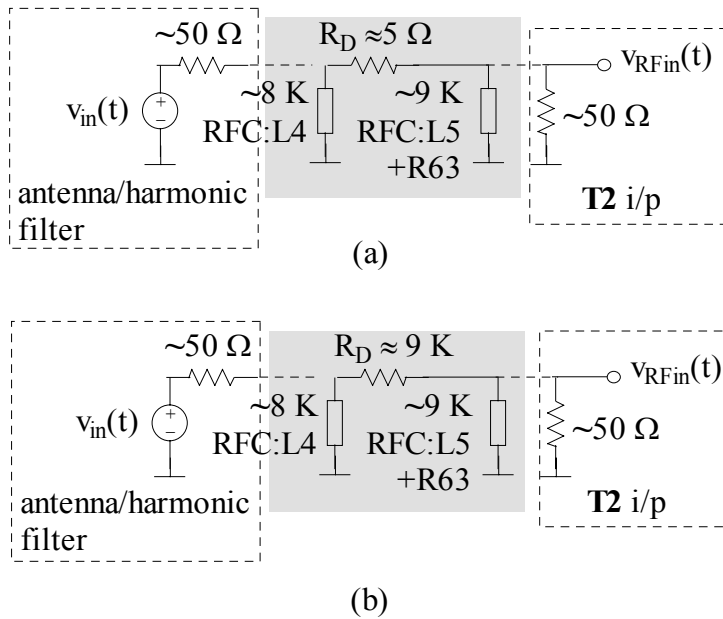


Figure 6.12 Equivalent circuit of RX RF attenuator at 16 MHz; (a) 10 mA bias current and (b)  $14 \mu\text{A}$  bias current

### 6.3. Audio gain control

Referring back to Figure 6.5(b), the output voltage of the envelope detector is

$$v_o(t) = V_{IF} [1 + m(t)] - V_o,$$

with possibly some ripple on it. The LPF formed by  $R$  and  $C$  effectively filters out almost the entire ripple, which is an RF component.  $V_{IF}[m(t)]$  is the detected information signal. We amplify this signal using TDA7052A and feed it to the loudspeaker, as discussed in Chapter 2. The d.c. part  $V_{IF} - V_o$  in  $v_o(t)$  also carries an important information about the communication channel.  $V_{IF}$  is a scaled version of the amplitude of the RF signal delivered by the antenna.

Wireless communication channels have peculiar properties, particularly at HF band. One such effect is called *fading*. The received signal amplitude changes occasionally in time rather slowly, due to propagation mechanisms that take place in HF band. This variation usually has a period more than few tens of seconds. When listening to the receiver output under fading, one gets the feeling that the voice slowly *fades* out and then comes back. This can be quite disturbing.

We use the d.c. component of detected signal to compensate fading effect in TRC-10. “ $V_{IF}$ ” is used to control the gain of TDA7052A, such that as the received signal amplitude *fades out*, the gain of the amplifier is increased. Similarly, as received signal comes back, gain decreases. The loudspeaker amplifier is shown in Figure 6.13.

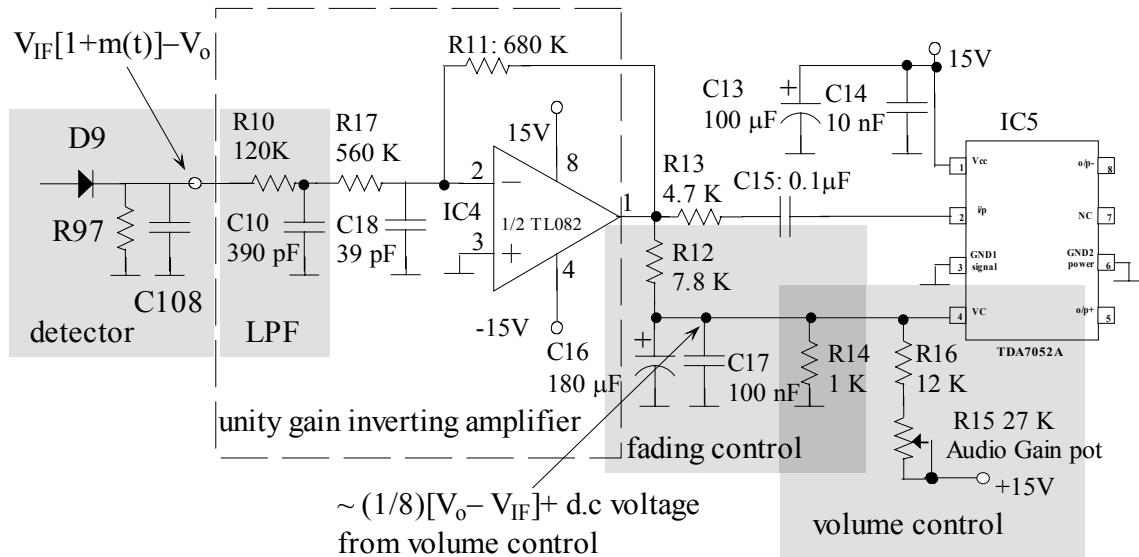


Figure 6.13 Audio gain control circuit

R12 and the parallel combination  $R14 // (R15 + R16)$  together with C16+C17 filters  $v_o(t)$ , to provide approximately  $(1/8) [V_o - V_{IF}]$  only to the pin 4 of TDA7052A. the volume control potentiometer R15 and R16 superimposes a d.c. voltage on  $(1/8) [V_o - V_{IF}]$ , such that potential across pin 4 varies approximately between  $0.4 - (1/8)V_{IF}$  and  $1.1 - (1/8)V_{IF}$  volts d.c. Once the R15 is adjusted for a comfortable volume level, the sound volume remains stable at that level. If the RF signal amplitude fades,  $V_{IF}$  falls and the voltage across pin 4 increases. The gain of TDA7052A increases, in turn.

#### 6.4. Bibliography

The ARRL Handbook has a chapter on modulation and demodulation (Chapter 15) and information on diodes that we discussed in this chapter (p.8.17).

Biasing is a fundamental topic in electronics and every electronics text devotes a significant space for it. Also detection is covered by all communications textbooks. P.J. Nahin's *The Science of Radio* (2<sup>nd</sup> Ed., Springer-Verlag New York, Inc, 2001) is an excellent book on introductory Telecommunications. A classic text, *Communication Circuits: Analysis and Design* by K. K. Clarke and D. T. Hess, gives a very good account of envelope detectors, although it is rather advanced and somewhat seasoned.

## 6.5. Laboratory Exercises

### *Envelope detector*

1. The detector circuit of TRC-10 is given in Figure 6.9.

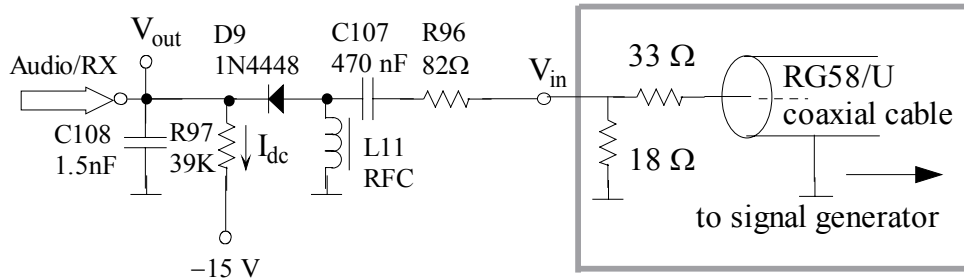


Figure 6.14 TRC-10 envelope detector

Calculate the bias current  $I_{dc}$  in this circuit. Find out the threshold voltage  $V_o$  of 1N4448 for this bias current. You can do this by examining the data sheet of 1N4448. The graph of “ $V_F$  -Forward Voltage” vs “ $I_F$  -Forward Current” reveals this information. Record this voltage. What is the threshold voltage if the bias current is 1 mA? (See problem 1 of this chapter.)

2. Wind 17-18 turns on RFC core to make L11. Take care while winding and stripping the leads, etc. as you did in the previous inductors. Install the components C107, L11, D9, R97 and C108 and solder them. Install R96 such that IC13 side is not connected and solder the C107 side. Check all connections both visually and using an ohmmeter.

Switch the power on. Using your multimeter, measure the threshold voltage  $V_o$ , across the anode and cathode of the diode. Does it agree with what you have found from the data sheet?

Measure the voltage across C108,  $V_{out}$ . How does it compare with  $V_o$ ? Switch the power off.

3. Mount and solder 33  $\Omega$ , 18  $\Omega$  resistors and the coaxial cable, with a BNC connector on one end, to the free end of R96 as shown in Figure 6.14 in gray bracket, using ground plane construction technique. Connect the coaxial cable to the signal generator. Switch the signal generator on. Set the signal generator to AM signal with 10 V p-p amplitude and 50 % modulation, to obtain

$$(2.5V)[1+0.5\cos(2\pi f_m t)]\cos(2\pi f_{IF} t),$$

where  $f_m$  is 1 KHz and  $f_{IF}$  is 16 MHz. Connect the signal generator output also to channel 2 of the scope, using a BNC-T connector and a coaxial cable with BNC connectors on both ends. Observe the waveform on the scope.

Calculate  $V_{in}$ , ignoring the loading of R96.

- Switch the power on. You will hear a high pitch sound. This sound is the detected 1 KHz modulation signal. Adjust the volume pot to a comfortable level.

Connect the probe of channel 1 of the scope to  $V_{out}$ . Adjust the scope to observe approximately 0.8 V p-p 1KHz sine wave. Measure and record the amplitude of  $V_{out}$ , and minimum and maximum ripple on the signal.

Connect a probe to IC6 output. Is there any ripple?

- Decrease the signal generator output amplitude gradually, while both watching IC6 output and listening the sound. Adjust the scope and the volume pot as necessary. Record the input amplitude levels where you can no longer see the signal and where you can no longer hear the signal, separately. Calculate  $V_{in}$  for these levels.
- Set the signal generator output back to 5 V p-p. Vary  $f_m$  between 500 Hz and 10 KHz and measure the IC6 output amplitude. Chose the number of measurements and measurement frequencies adequately. Find the  $-3$  dB frequency. Switch the power and equipment off.
- Remove 33  $\Omega$ , 18  $\Omega$  resistors and the coaxial cable. Remove excess solder. Place the free end of R96 into its place and solder.
- Set up the input part of the circuit in Figure 5.26 (i.e. two 1.5 K and the 56  $\Omega$  resistors, and the coax.), and check that the entire RX/IF section and loudspeaker amplifier chain works. Signal generator output amplitude must be low. 10 mV to 50 mV p-p is sufficient. Describe how you performed this test and record your observations. Do not forget that now you have the very narrow band IF filter on the way. You must tune the signal generator frequency to the center frequency of IF filter. All probe connections must be made to the OPAMP outputs only.

Desolder and remove the two 1.5 K resistors, 56  $\Omega$  resistor and the coaxial cable at the input. Remove the excess solder from the copper surface using your desoldering pump. Install and solder C94 and C95.

### ***TX/RX switching circuit***

- Make the two RFCs L4 and L5, like you made L11. The circuit diagram of TX/RX switching circuit is given in Figure 6.9. Install and solder L4, L5, R49, R62, R63, C73, C74, C75, D7 and D8. Make sure that the polarities of the diodes are correct.

Install and solder the 2-pin PCB jack J21 for 1 M R64. Install and solder the 3-pin PCB jack J22 for S2/1 TX/RX switch.

Check all connections.

- R64 is a 1 M *logarithmic potentiometer*. When its knob is set to approximately the mid point of the range, the resistances on either side are not equal in these *log-pots*. One of them is approximately  $(1 \text{ E}6)^\alpha \Omega$ , where  $\alpha$  is a positive number less

than one. For this pot,  $\alpha$  is about 0.8 and  $(1 - \alpha) \Omega$  is about 70 K, approximately. Find out which side pin is on the high resistance side. Short circuit that pin to the center pin by soldering a small piece of wire to those two pins on the pot.

Cut two 20 cm pieces of insulated construction wire. Strip a 5 mm of insulation from each end. Connect and solder one wire to *low* impedance side pin. Solder the other wire to one of the two shorted pins. Fit a 2-pin PCB plug to the other two leads of the wires.

Fit the plug into the jack J21 and check that the contacts are properly functioning.

Carefully mount R64 on the panel. Fit its knob. Collect and tie the wires together and guide it along the side of the tray tidily, from the panel to the jack.

11. S2 is a DPDT “ON-ON” switch. It has two switches operating in parallel. Each switch has three pins. The normal position of S2 is such that the center (floating) pin is connected to one of the side pins, in each switch. This side pin is RX pin in each switch, so that when the switch is left alone TRC-10 remains in RX mode. When the switch is toggled, center pin contacts the other side pin. This pin is TX pin.

Identify the center pins, the RX pins and TX pins on each switch, using a multimeter.

Cut three 20 cm pieces of insulated wire for each switch (six altogether). Strip a 5 mm of insulation from each end. Solder a wire to each contact of S2. In this exercise, we connect only S2/1. S2/2 switch is connected to the PCB in the exercises of Chapter 7. Collect the three wires from S2/2 together, wind them into a coil and tie them.

Before fitting a 3-pin PCB plug to the wires of S2/1, check the PCB jack J22 pins. +15 pin on the jack must correspond to TX pin on the plug. The pin connected to R49 on the jack must correspond center pin on the plug, and -15 on the jack to RX on the plug. Fit the wires to the plug carefully.

Fit the plug into the jack J22 and check that the contacts are properly functioning.

12. Make sure that C72 is *not* connected yet. Switch the power on. In this exercise, we perform the d.c. test of the switch circuit without any signal in it.

Connect your multimeter across R62 and measure the d.c. voltage across it, when S2 is thrown to TX mode. Calculate the current passing through R62. This is the bias current of D7. Record these values.

Measure the d.c. voltage across D7, with S2 again in TX mode. Record this voltage. From the data sheet of PIN diode, find the threshold voltage and  $R_D$  corresponding to the current you measured. Compare this voltage with the one you measured. Record all your work and observations.

Let S2 be in its normal position (i.e. RX mode). Measure the d.c. voltage across R63 with R64 set to minimum, to midpoint and to maximum. Calculate the bias current of D8 for these three cases. Find out the threshold voltage and  $R_D$  corresponding to these bias currents.

Switch the power off.

13. We assumed that approximately  $50\ \Omega$  terminates all three ports of the TX/RX switch circuit. The RX port provides the input to RX mixer. SA602A has input impedance of 1.5 K, nominally. Actually this input impedance varies significantly with frequency, and we expect it to be less than 1.5 K. T2 transformer is a matching transformer. We use a T25-10 toroid, for this transformer.

Low loss, high permeability transformer materials are not available at 30 MHz. We use a core made of low loss inductor material, mix-10 of Micrometals. This transformer is wound as an *auto transformer* in order to reduce the leakage flux. Figure 6.15 shows the structure of T2.

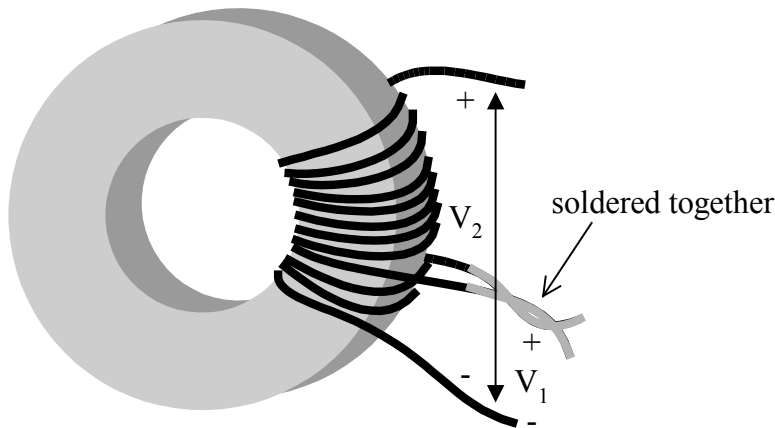


Figure 6.15 T2 auto transformer

The first four turns of the secondary winding and the primary winding are the same, and two coils are electrically connected in this auto transformer. The secondary winding has eight more turns, making a turns ratio of 4:12.

Wind the transformer using an appropriate length of 0.2 mm diameter wire. Wind the turns tightly. You can try a *trifillar* winding to reduce leakage flux, if you wish. Refer to ARRL Handbook for multifillar windings. You can also ask the lab technician to describe how to make it and show you one.

Install and solder T2.

14. The RX input circuit is given in Figure 6.16.



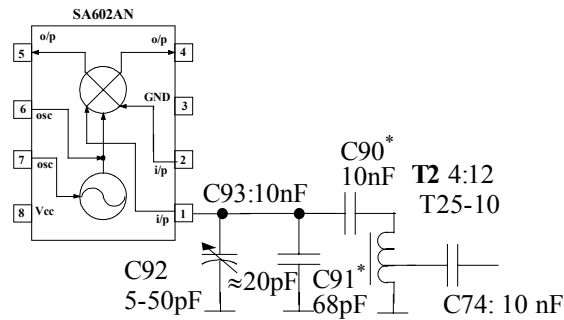


Figure 6.16 RX mixer input circuit

Mix-10 material has a relative permeability of 6. T25-10 core has an inductance constant of  $1.9 \text{ nH/turns}^2$ . With 12 turns secondary winding presents an inductance of about 300 nH. C91 and C92 tune out this inductance at 29 MHz.

SA602A has a d.c. bias at its both inputs. This bias voltage must be guarded. C90 is a d.c. blocking capacitor performing this task.

Install C90, C91 and C92, taking care to place the adjustment pin of C92 to ground side. Solder them. Check all contacts.

15. Install and solder C72. Check its connections. Solder a  $51 \Omega$  resistor and the free end of the cable from antenna jack across C79, again using ground plane construction technique. Connect the coaxial cable, with BNC plugs on each end, between the antenna jack and a channel of the oscilloscope.

The 29 MHz section of the transmitter is complete. The circuit is given in Figure 6.17.

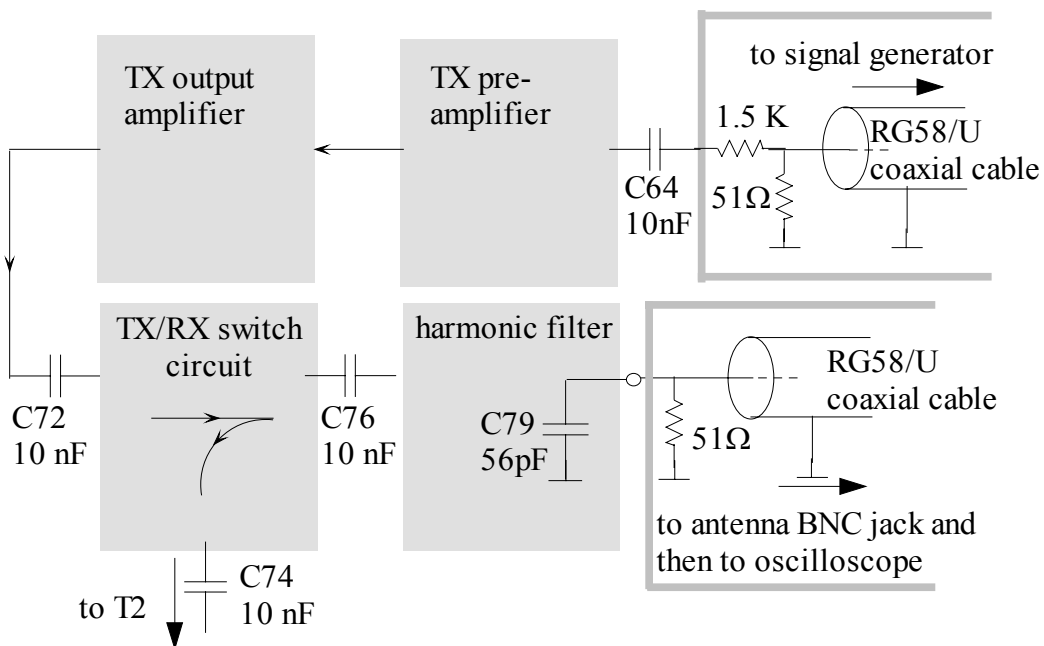


Figure 6.17 TX 29 MHz section

Solder the RG58/U cable for signal generator connection. Note that the 1.5 K and 51  $\Omega$  resistors are already on the board, left from Exercises 4.16 and 5.8.

Set the signal generator for a 29 MHz sine wave output with 50 mV p-p amplitude. Throw S2 to TX mode and watch the signal on the scope. Check that the entire 29 MHz section works. Estimate the gain of the overall section. Retune C65 and C69 if necessary.

Switch the power and the equipment off. Disconnect the oscilloscope and the signal generator. Desolder both cables, **but leave all ground plane constructed components and their connections on the board.**

## 6.6. Problems

- Find the current  $I$  in the circuit of Figure 6.18(a). Also find the threshold voltage for this current. (Hint: You must refer to the data sheet of 1N4448. The solution requires iteration. Start by assuming that the threshold voltage of the diode is e.g. 0.7 V or 0 V, then calculate  $I$ . Find the threshold voltage for this value of  $I$  from data sheet and re-calculate  $I$  for new threshold voltage. Continue until convergence, i.e. the difference between two successive  $I$  values is less than e.g. 5%)

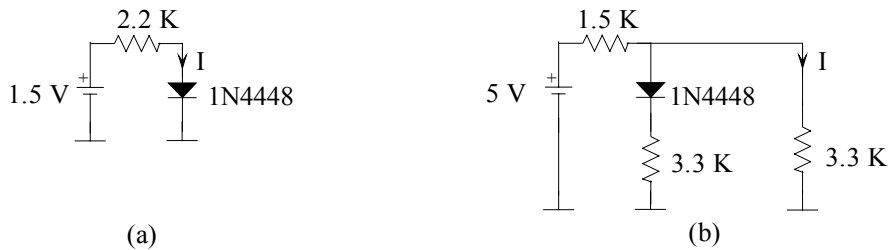


Figure 6.18. Circuit for (a) problem 1 and (b) problem 2

- Find current  $I$  in the circuit of Figure 6.18(b). (Hint: First find the Thevenin equivalent circuit across the diode terminals and find the current through the diode as in problem 1)
- Determine and sketch the waveform at the output for the circuits given in Figure 6.19. Assume that the diode is ideal.

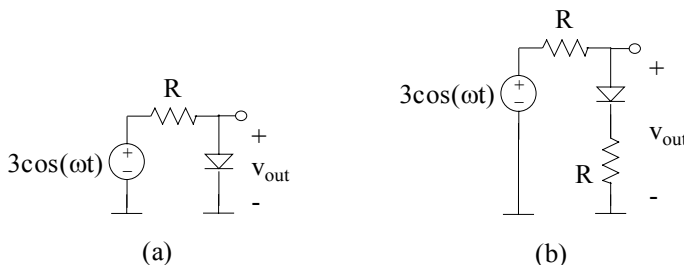


Figure 6.19 Circuits for problem 3

- $v(t)$  is  $0.5 \cos(\omega t)$  V in Figure 6.20(a). Assume that the diode can be modeled by the piecewise linear model of Figure 2.16(a), with  $V_o = 0.6$  V. What is  $v_{out}(t)$  if

$I = 0$  A? Find the minimum value of  $I$  for which there is an undistorted replica of  $v(t)$  at the output. Find  $v_{out}(t)$  for this value of  $I$ . What is the value of  $I$  such that time varying part of  $v_{out}(t)$  is exactly half wave rectified (but scaled, of course) form of  $v(t)$ ? (Hint: First find the Thevenin equivalent circuit, comprising both sources, across the detector circuit)

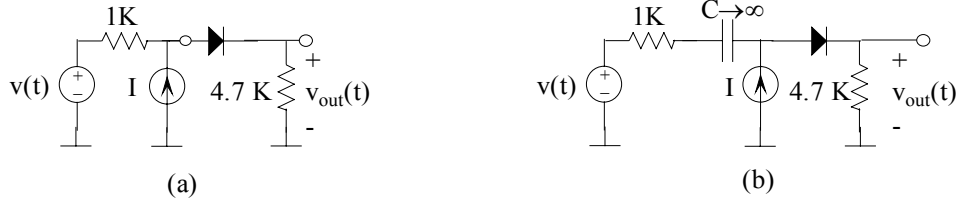


Figure 6.20 Circuit for (a) problem 4 and (b) problem 5

5. Repeat problem 4 for the circuit given in Figure 6.20(b).
6. If  $RC = 1/\omega_m$ , in the circuit in Figure 6.3, estimate the maximum ripple on detected envelope if the modulation index is 100%? Assume  $\omega_m \ll \omega_{IF}$ . What is the minimum ripple? Is there any failure-to-follow distortion?
7. Calculate the attenuation with 10 mA and 14  $\mu$ A bias current level in Figure 6.12, in dB. What is the *dynamic range*, i.e. the ratio between the two, of this attenuator?

## Chapter 7 : FREQUENCY CONVERSION

Mixers convert the frequency band of signals. The audio signal is first converted from *base-band* up to 16 MHz in TRC-10, and then to amateur band, between 28 MHz and 29.7 MHz. This is referred to as *up conversion*.

The received signal, on the other hand, is *down converted* to 16 MHz, where it is filtered and detected.

### 7.1. Amplitude modulators

Amplitude modulation is also a frequency conversion operation. Modulated signal is obtained right after the microphone amplifier in TRC-10. A diode and a tuned circuit perform amplitude modulation. Consider the circuit given in Figure 7.1.

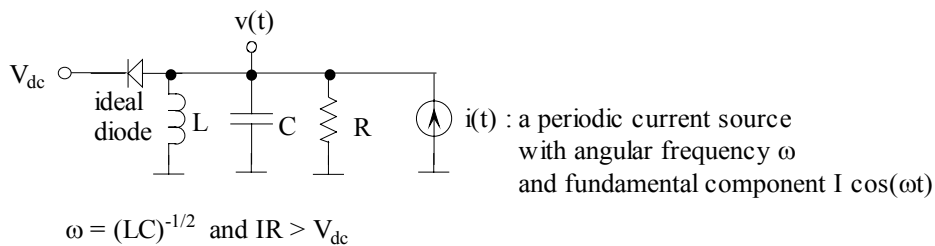


Figure 7.1 A circuit for amplitude modulation

A parallel tuned circuit is driven by a periodic current source and it is connected to a d.c. voltage source through a diode. In this circuit the voltage across the tuned circuit is always

$$v(t) = V_{dc} \cos(\omega t)$$

as long as  $IR > V_{dc}$  and the circuit is tuned to  $\omega$ .

Principle of operation is straightforward. We enforce  $IR > V_{dc}$ , so that if there were not any diode, the voltage peak across the tuned circuit would exceed  $V_{dc}$ . Whenever the voltage across the tuned circuit exceeds  $V_{dc}$ , diode conducts. Since the resistance of the d.c. voltage source is zero the peak of the voltage is *clamped* to  $V_{dc}$  (i.e. it cannot be more than  $V_{dc}$ ).

On the other hand, the voltage that can appear across the tuned circuit can only be in sinusoidal form, if the Q of the circuit is reasonably high. The d.c. current component flows through the inductor and harmonics flow through the capacitor, generating insignificant voltages. Hence the tuned circuit voltage is forced to be a sinusoidal voltage with amplitude of  $V_{dc}$  and frequency of  $\omega$ .

This problem belongs to a family of circuit theory problems, usually called as “nonlinear loading of narrowband circuits”. Exact solution of this problem, given above, can be analytically found by applying a circuit theory technique called *harmonic balance*. We leave the discussion of harmonic balance to advanced texts.

Now if we add a slowly varying (compared to  $\omega$ ) audio signal component to  $V_{dc}$  as in Figure 7.2, we obtain an amplitude modulator. OPAMP output impedance is very low and it behaves like a voltage source providing  $V_{dc}[1+m(t)]$ . Here,  $IR$  must be larger than the maximum value of  $V_{dc}[1+m(t)]$ , of course.

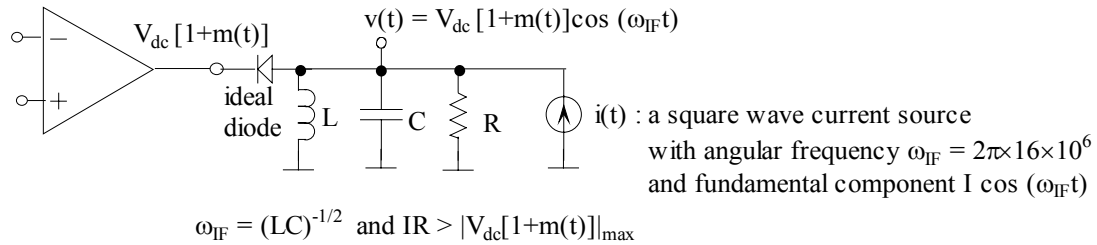


Figure 7.2 Amplitude modulator

### 7.2. Mixers in telecommunication circuits

#### 7.2.1. Switch mixer

Converting the frequency of a signal requires multiplication operation. Consider a simple mixer comprising a switch in Figure 7.3.

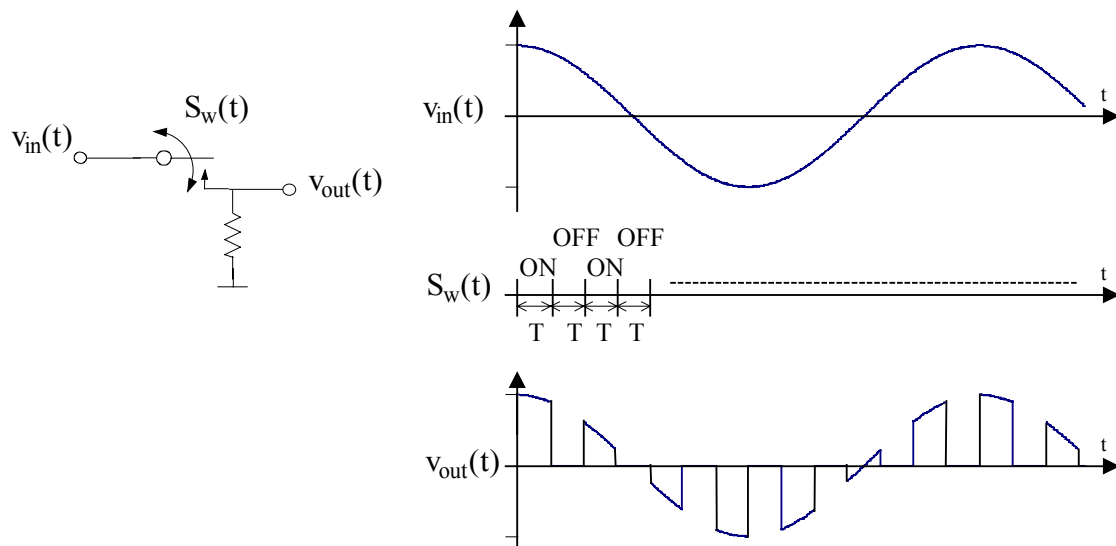


Figure 7.3 Switch mixer

Switch remains closed for  $T$  seconds and the opens and remains open for  $T$  seconds periodically. When the switch is closed,  $v_{out}(t)$  is an exact replica of  $v_{in}(t)$  in this circuit. When switch is open,  $v_{out}(t)$  is zero.

This operation is equivalent to multiplying  $v_{in}(t)$  by a square wave like the one in Section 1.2. This square wave, however, has amplitude of 1 when the switch is closed, and it is zero when the switch is open. Such a square wave can be written as

$$s(t) = a_0 + \sum_{n=1}^{\infty} b_n \sin(n\omega_s t)$$

where  $a_0$  is 0.5 and  $b_n = (1/n\pi)[1 - (-1)^n]$  for this square wave (p-p amplitude of this square wave is 1). The period of the square wave is  $2T$ , hence its fundamental angular frequency  $\omega_s$  is  $\pi/T$ . Assuming that the input signal is a narrow band signal and can be represented by a sinusoid

$$v_{in}(t) = V \cos(\omega_i t),$$

output becomes

$$\begin{aligned} v_{out}(t) &= v_{in}(t) \times s(t) \\ &= \{V \cos(\omega_i t)\} \left\{ 0.5 + \sum_{n=1}^{\infty} b_n \sin(n\omega_s t) \right\} \\ &= 0.5 V \cos(\omega_i t) + \sum_{n=1}^{\infty} b_n V \sin(n\omega_s t) \cos(\omega_i t). \end{aligned}$$

Output signal contains signal components at many frequencies now, as well as a component at the input frequency,  $0.5 V \cos(\omega_i t)$ . For  $n=1$ ,  $b_1$  is  $2/\pi$ . The next coefficient  $b_2$  is zero and  $b_3$  is  $2/3\pi$ . The open form of output signal can be written as

$$\begin{aligned} v_{out}(t) &= 0.5 V \cos(\omega_i t) + (2/\pi) (V) \sin(\omega_s t) \cos(\omega_i t) + (2/3\pi) (V) \sin(3\omega_s t) \cos(\omega_i t) + \dots \\ &= 0.5 V \cos(\omega_i t) + (V/\pi) \sin[(\omega_s + \omega_i)t] + (V/\pi) \sin[(\omega_s - \omega_i)t] + \\ &\quad (V/3\pi) \sin[(3\omega_s + \omega_i)t] + (V/3\pi) \sin[(3\omega_s - \omega_i)t] + \dots \end{aligned}$$

The frequency of the input signal  $v_{in}(t)$  is now shifted by all odd harmonics of the switch frequency. The signal component  $(2/\pi) (V) \sin(\omega_s t) \cos(\omega_i t)$  in  $v_{out}(t)$ , for example, can be viewed as a DSBSC AM signal where  $\omega_s$  is the carrier and  $\omega_i$  is the modulating frequency (or the other way around). The spectra of  $v_{in}(t)$  and  $v_{out}(t)$  are given in Figure 7.4.

If we filter out the  $(2/\pi) (V) \sin(\omega_s t) \cos(\omega_i t)$ , or equivalently

$$(V/\pi) \sin[(\omega_s + \omega_i)t] + (V/\pi) \sin[(\omega_s - \omega_i)t],$$

from  $v_{out}(t)$ , we obtain a *frequency converted* version of base band signal  $v_{in}(t)$ , converted to  $\omega_s$ . All frequency components of  $v_{in}(t)$  are shifted by  $\omega_s$  in frequency.

The circuit in Figure 7.3 *mixes*  $v_{in}(t)$  and  $s(t)$ . This circuit can easily be implemented, using the techniques we discussed in Chapter 6.

Note that a component at input signal frequency is also present in  $v_{out}(t)$ . In some mixer configurations, a component at switch fundamental frequency is also present.

Such mixers are called *unbalanced* mixers. It is quite undesirable to have either one of these components at the output, because their presence complicates the filtering.

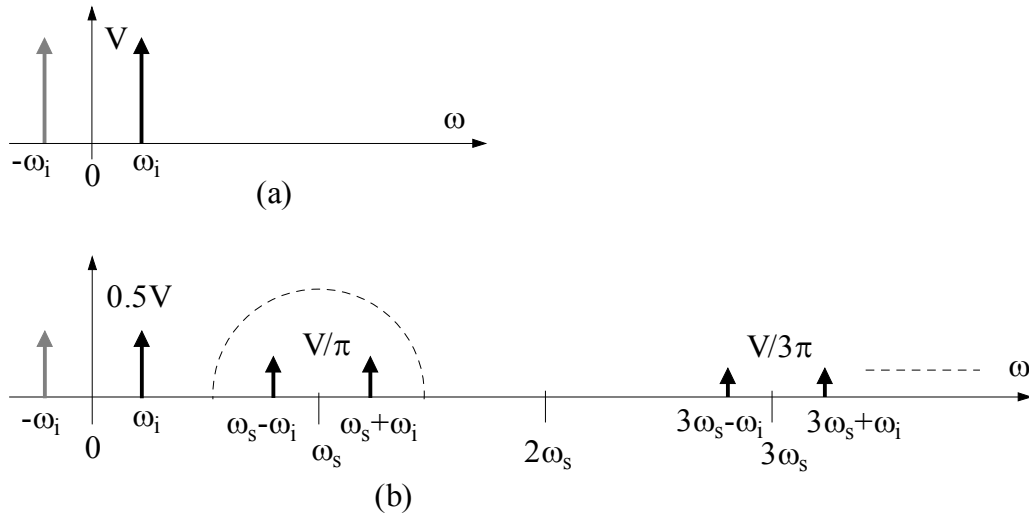


Figure 7.4 Frequency spectrum of (a)  $v_{in}(t)$  and (b)  $v_{out}(t)$

7.2.2. Double-balanced mixers

Mixers with configurations that provide an output free from any component at input frequencies ( $\omega_i$  and  $\omega_s$  in the case for the circuit in Section 7.2.1) are called *double-balanced* mixers. A double-balanced mixer circuit is shown in Figure 7.5.

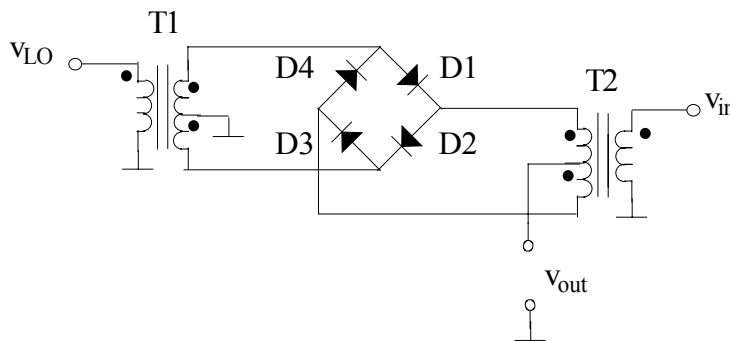


Figure 7.5 Double-balanced mixer

$v_{LO}$  is the periodic signal (usually square wave) which provides the mixing frequency, like  $s(t)$  in Section 7.1.1. It is denoted as  $v_{LO}$ , because usually a *local oscillator* within the communication device generates this signal.  $v_{in}$  and  $v_{out}$  are the signal to be mixed and mixed output, respectively.

The dots on the transformer winding show the coupling polarity of primary and secondary voltages. Voltages on the dotted side of the windings have the same phase in each transformer.

This mixer works by switching the route of  $v_{in}$  to  $v_{out}$ . When the square wave of  $v_{LO}$  is in positive phase, D1 and D2 conduct. In this case the secondary current in T2 passes through the upper secondary winding of T2, D2 and lower secondary winding of T1

to ground.  $v_{out}$  is the voltage across the upper secondary winding of T2 in this case. Note that the dotted side of this winding is on the grounded side for this route.

When the square wave is in negative phase, D3 and D4 conduct. The secondary current follows the path provided by lower secondary winding of T2, D3 and upper secondary winding of T1 to ground.  $v_{in}$  couples to  $v_{out}$  across the lower secondary winding of T2. In this case un-dotted terminal of the winding is grounded. The phase of  $v_{out}$  in this route is exactly 180 out of phase with the one in the previous route.

The function of the circuit in Figure 7.5 can be modeled as multiplying  $v_{in}$  by a square wave of 1's and -1's, rather than 1's and zeros. The average value of this kind of *symmetric* square wave is zero and hence the d.c. term in its expansion does not exist.

**7.3. Analog multipliers**

A more sophisticated type of double-balanced mixing employs *analog multipliers*. Certain electronic circuit configurations provide means of multiplying two analog signals. *Gilbert cell* is a commonly used analog multiplier configuration. The input/output relations in a Gilbert cell circuit are modeled in Figure 7.6(a).

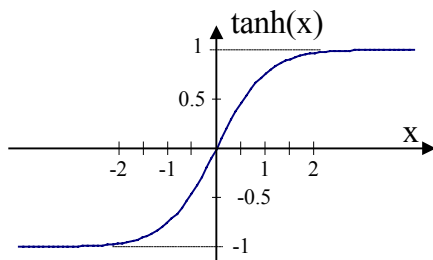
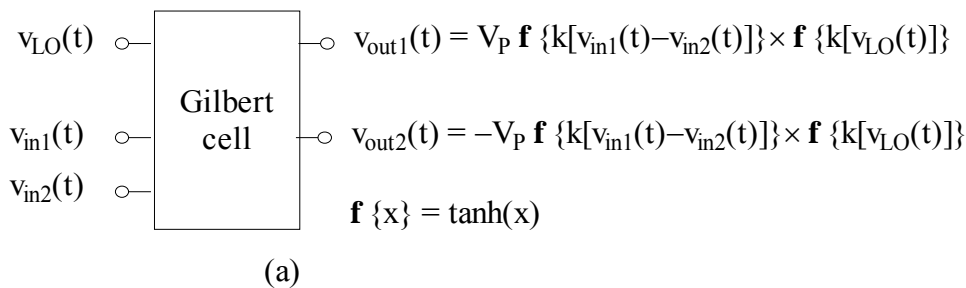


Figure 7.6 (a) Block diagram of *Gilbert cell* and (b)  $\tanh(x)$

The output of a Gilbert cell is a product of a function of the difference of two input signals and the same function of the local oscillator signal. The function is *hyperbolic tangent*, or  $\tanh(\cdot)$ , given as

$$\tanh(x) = \frac{e^x - e^{-x}}{e^x + e^{-x}}$$



The variation of  $\tanh(x)$  is depicted in Figure 7.6 (b). Gilbert cell is an electronic circuit and obviously it scales the input voltages in order to convert them into the argument of the  $\tanh(\cdot)$  function. The scaling factor is  $k$  and has a unit of  $(\text{volt})^{-1}$ .

Note that when  $x$  is small,  $\tanh(x)$  is linear. Indeed for  $|x| < 0.3$ ,  $\tanh(x) \approx x$ . Therefore for low amplitude levels of  $v_{in1} - v_{in2}$  and  $v_{LO}$ ,  $v_{out1}(t) \approx V_p k^2 [v_{in1}(t) - v_{in2}(t)] [v_{LO}(t)]$ . A direct *analog multiplication* can be performed.

The block diagram of a typical application of using an analog multiplier as a double balanced mixer is given in Figure 7.7 (a).

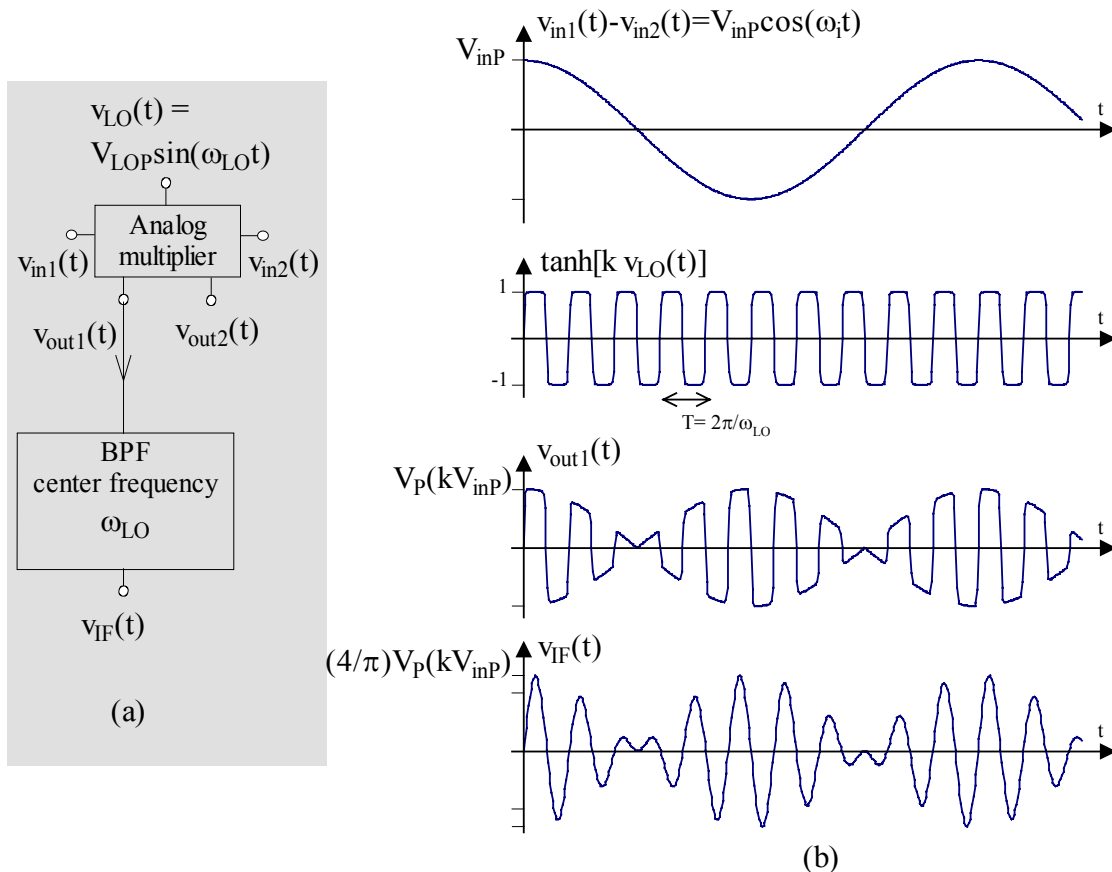


Figure 7.7 Analog multiplier and IF filter outputs with large  $v_{LO}$  amplitude (a) block diagram and (b) waveforms.

Base band signals (such as audio, for example)  $v_{in1}(t)$  and  $v_{in2}(t)$  and a sine wave from the local oscillator  $v_{LO}(t)$  are applied as inputs to the analog multiplier. The output of the multiplier is filtered using a BPF centered at the local oscillator frequency. IF signal  $v_{IF}$  is obtained at the filter output.

The amplitude of  $v_{in1}(t)$  and  $v_{in2}(t)$  is normally kept low in order to avoid distortion. A low amplitude sine wave is assumed in the waveforms given in Figure 7.7 (b).

When analog multipliers are used as double-balanced mixers for frequency conversion, a high amplitude sinusoidal  $v_{LO}(t)$  signal is employed. Then  $\tanh[kv_{LO}(t)]$  is almost like a square wave with  $\pm 1$  levels, as depicted in Figure 7.7 (b).

With such choices for  $v_{in1}(t) - v_{in2}(t)$  and  $v_{LO}(t)$ , the output becomes

$$v_{out1}(t) \approx V_P k(v_{in1} - v_{in2}) \left\{ \sum_{n=1}^{\infty} b_n \sin(n\omega_s t) \right\} = V_P \sum_{n=1}^{\infty} b_n \sin(n\omega_s t) \{k(v_{in1} - v_{in2})\}.$$

The signal  $v_{out1}(t)$  is given in Figure 7.7 (b). When this signal passes through the BPF, all harmonics of  $\tanh[kv_{LO}(t)]$  are eliminated and we obtain the IF signal

$$v_{IF}(t) \approx b_1 V_P (kV_{inP}) \cos(\omega_i t) \sin(\omega_s t).$$

The fundamental component amplitude in this square wave is  $b_1 = 4/\pi$  (and is larger than 1), like the one in Chapter 1. Therefore the p-p amplitude of  $v_{IF}(t)$  is  $b_1$  times larger than the p-p amplitude of  $v_{out1}(t)$ .

Gilbert cell analog multipliers are difficult to implement at frequencies above few hundred MHz. SA602A is a very good integrated circuit implementation of Gilbert cell, which operates very well up to UHF. Such implementations are active circuits and they provide gain as well as analog multiplication.

### 7.3.1. Conversion gain

An important parameter in the evaluation of mixers is *conversion gain*. Conversion gain is defined as

$$G_c = P_o/P_i$$

where  $P_o$  is the total power delivered to a matched load at output and  $P_i$  is, again, the total available power at the input. This expression is similar to the gain of an amplifier, except here the input and output frequencies are different.

## 7.4. Oscillators

We need oscillators, i.e. periodic signal generators, for frequency conversion. Any telecommunication receiver contains at least one oscillator. We use one fixed frequency oscillator and one variable frequency oscillator (VFO) in TRC-10.

Fixed frequency oscillator is an IC module, which produces a square wave at 16 MHz.

The VFO in TRC-10 is implemented using an OPAMP and discrete elements. VFO provides a sinusoidal signal. Its frequency can be changed between 12 MHz and 13.7 MHz.

## 7.4.1. Oscillator concept

We discussed *feedback* in Section 3.4 and noted that we always use negative feedback for amplification. In oscillators, we need *positive feedback*.

Consider the simple circuit given in Figure 7.8 (a), where a current source is connected to a *very high Q* parallel tuned circuit. We assume that the current source  $\bar{i}(t)$  contains many sinusoidal components at all frequencies and its amplitude is very small. Actually *noise* in electronics is such a signal. Only the current component in the vicinity of  $\omega_0$  generates a voltage  $v_1(t)$  across the tank circuit. The amplitude of  $v_1(t)$  is very small also. We expect to observe  $v_1(t)$  as

$$v_1(t) \approx V_1 \cos(\omega_0 t),$$

where  $\omega_0$  is  $(LC)^{-1/2}$  and  $V_1$  is very small. If an amplifier is connected to this node,  $v_1(t)$  is amplified as shown in Figure 7.8 (b).

Supply voltage levels limit the output voltage swing of amplifiers. When the amplified signal amplitude  $AV_1$  approaches to supply voltage  $V^+$ , output waveform gets distorted. We obtain a clipped waveform. The amplifier is said to be *saturated*. If input voltage amplitude increases further, the output waveform approaches to a square wave.

At this stage, let us assume that the gain is not large enough to saturate the amplifier.

When a *feedback* path to the *positive input* is provided by means of a resistor  $R_2$  as shown in Figure 7.8 (c),  $v_1(t)$  is modified. Initially, an additive sample from output increases  $v_1(t)$  to

$$v_1(t) \approx V_1 \cos(\omega_0 t) + [R_1/(R_1+R_2)] \times (AV_1) \cos(\omega_0 t).$$

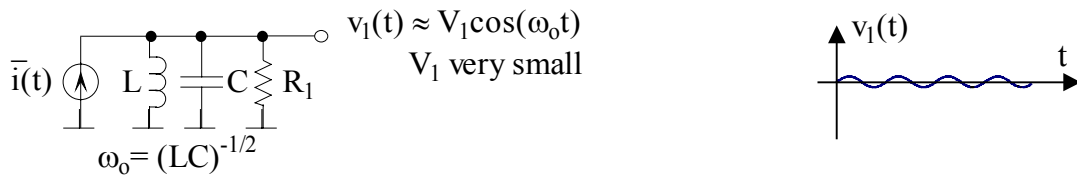
The additive component has an amplitude of  $[R_1/(R_1+R_2)] \times (AV_1)$  which is much larger than the signal directly created by  $\bar{i}(t)$ . This component alone drives the amplifier deep into saturation, since  $A \times [R_1/(R_1+R_2)] \times (AV_1) \gg V^+$ . We immediately have a square wave at the output as  $v_2(t)$ . The p-p amplitude of this wave is twice the saturation voltage, which is usually slightly less than  $V^+$ . This waveform is also shown in Figure 7.8 (c).

When we have a square wave at the output, the feedback signal is also a square wave. However, the tank circuit filters the fundamental component out of this square wave, producing a  $v_1(t)$  as

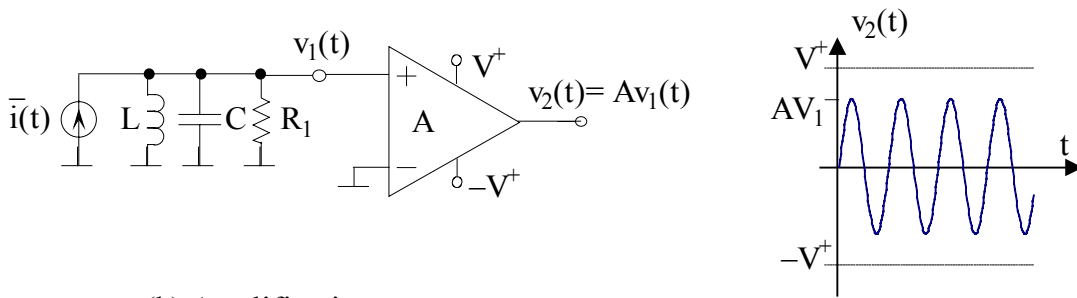
$$v_1(t) \approx V_1 \cos(\omega_0 t) + [R_1/(R_1+R_2)] \times b_1(V^+) \cos(\omega_0 t).$$

Here,  $b_1 = 4/\pi$  is the coefficient of the fundamental component in a square wave. Since the output of the amplifier cannot change any more, the circuit operation is stabilized with a square wave at its output. The tank circuit determines the frequency of this signal.

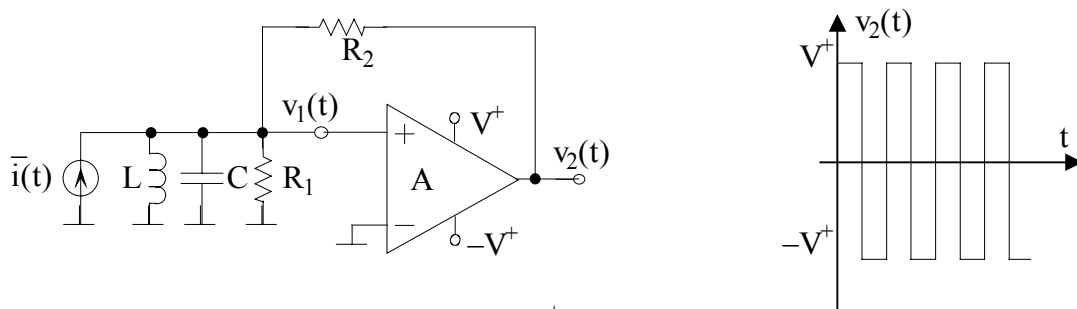
The input signal  $v_1(t)$  is predominantly the feedback signal. If we remove the current source from the circuit, the output is still a square wave. Neither  $v_2(t)$  nor  $v_1(t)$  are affected, as shown in Figure 7.8 (d).



(a) Tank circuit driven by a wide band current source



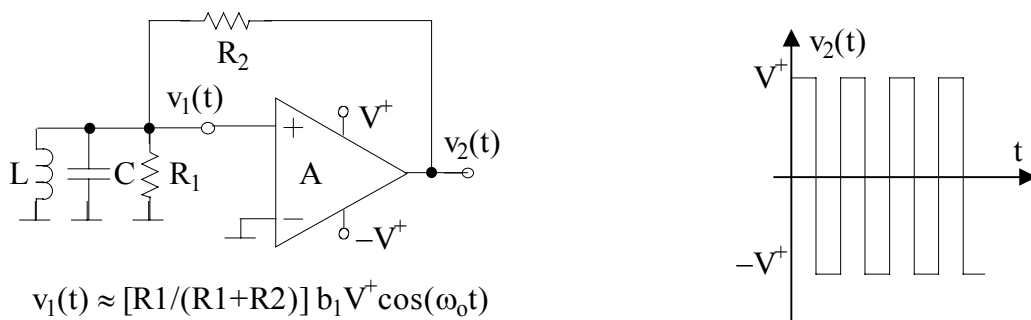
(b) Amplification



$$v_1(t) \approx V_1 \cos(\omega_0 t) + [R_1 / (R_1 + R_2)] b_1 V^+ \cos(\omega_0 t)$$

$$[R_1 / (R_1 + R_2)] b_1 V^+ \gg V_1$$

(c) Positive feedback



$$v_1(t) \approx [R_1 / (R_1 + R_2)] b_1 V^+ \cos(\omega_0 t)$$

(d) current source removed

Figure 7.8 Oscillator concept

Practically we never include the current source to start the oscillation, because it is not necessary. There is *always* noise in electronic circuits. Also we do not need to remove the current source, because we cannot. It is *always* there.

It is possible to deduce from the above discussion that if

$$[R_1/(R_1+R_2)]A=1,$$

we have a sustained oscillation, once it starts. In this case the output waveform is sinusoidal and amplifier works in linear region all the time. This condition, i.e. the product of amplifier gain and the feedback ratio being unity is *oscillation criterion*.

In the circuit of Figure 7.8, we have the amplitude limiting mechanism of saturating amplifier. Since the fundamental component of the saturated output is  $b_1 V^+$ , input amplitude is always  $[R_1/(R_1+R_2)] b_1 V^+$ . As long as the gain is large enough to keep the amplifier in saturation with this input, oscillation is sustained. The larger gain of the amplifier helps oscillations to start easily. The feedback ratio  $[R_1/(R_1+R_2)]$  and gain  $A$  must be such that

$$[R_1/(R_1+R_2)]A \times (V^+) > V^+ \quad \Rightarrow \quad [R_1/(R_1+R_2)]A > 1,$$

in order that oscillation starts up in this circuit.

There are many subtle subjects in the theory of oscillators, such as *frequency* and *amplitude stability*, *phase jitter*, etc. We leave these topics to advanced texts.

#### 7.4.2. Frequency control

The frequency determining parameters are  $C$  and  $L$  in the oscillator circuit of Figure 7.8 (d). We must change the value of one of these components, if we want to vary the frequency. It is possible to use either a variable capacitor or a variable inductor.

One of the simplest ways of changing the capacitance is to use a semiconductor device called *varactor diode* (also called *varicap diode* and *tuning diode*). The symbol of the varactor diode is shown in Figure 7.9 (a). Varactor diodes are used with a reverse voltage bias on them. The capacitance across the cathode and anode depends on the level of reverse bias voltage. The variation of the diode capacitance with respect to reverse diode voltage is given in Figure 7.9 (b).

The circuit in Figure 7.9 (c) delineates the way a varactor diode is used in a circuit. The potentiometer  $R_{\text{tune}}$ , connected between two supply voltages, control the reverse voltage bias on the diode. The capacitance  $C_o$  blocks the d.c. bias voltage on the diode. The large series resistor only serves to isolate the tuned circuit elements from  $R_{\text{tune}}$ , so that the  $Q$  of the resonant circuit remains high. The series combination of diode capacitance and  $C_o$  appears across the tank circuit. The total capacitance of the tank circuit becomes

$$C + C_D C_o / (C_D + C_o).$$

Adjusting the potentiometer can now vary the resonance frequency of the tank circuit.

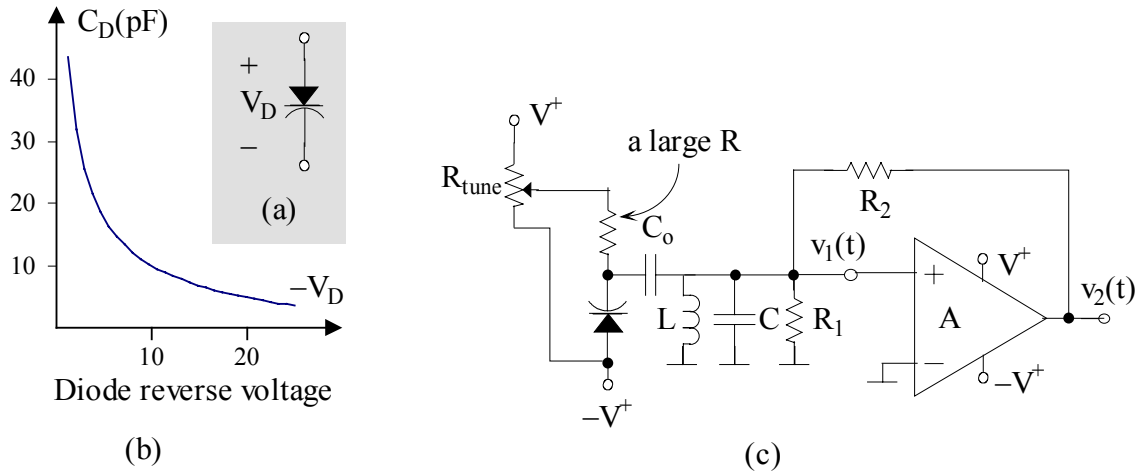


Figure 7.9 Varactor diode (a) symbol, (b) capacitance variation and (c) use in the circuit

#### 7.4.3. Phase considerations in oscillators

We assumed that the OPAMP is ideal in Section 7.4.1 when discussing the concept of oscillation. There, we implicitly assumed that the amplifier provides a gain without any effect on the phase of the signal. However, when OPAMPs are used for RF amplification, the open loop transfer function must be considered in calculating the gain, as discussed in detail in Section 4.5.1. The LM7171 based non-inverting TX amplifier analyzed in Section 4.5.1 turned out to have a voltage gain of  $5.1 \angle -31^\circ$  between the positive input pin 3 and output. If the OPAMP had indefinitely large open loop gain as in the case of an ideal OPAMP, the overall gain of the amplifier would be  $5.7 \angle 0^\circ$  instead of  $5.1 \angle -31^\circ$ . This phase shift does not affect the performance of the TX amplifier functionality.

A similar phase shift, however is critical if we use the same amplifier in an oscillator. The concept of oscillation relies on the consideration that a constructive addition of the *fed back signal* and the available signal takes place at the positive input terminal, as depicted in Figure 7.8 (c). The sum of addition is maximum when the phases of both signals match. Hence the oscillator configuration in Figure 7.8, seeks the frequency where the phases of the output and that of the input are the same. In other words, there must be no phase difference between the output and positive input at the oscillation frequency.

We use a series capacitor, C140, in the positive feed back loop of the oscillator amplifier in TRC-10, instead of a resistor only. The value of this capacitor must be chosen such that the phase shift provided by this series combination compensates for the shift in the amplifier in the vicinity of the required oscillation frequency.

## 7.5. Bibliography

Many texts are available on mixers. However the treatment of the topic at introductory electronics textbooks is usually very weak. In *The Electronics of Radio* mixing and SA602A are discussed at an excellent level and detail.

P. R. Gray, P.J. Hurst, S. H. Lewis and R.G. Meyer (Wiley 2001) discuss Gilbert cell in *Analysis and Design of Analog Integrated Circuits, 4th Edition*, in Chapter 10.

Oscillators are extensively discussed in literature. A very good account of OPAMP oscillators is given in D. L. Terrell's *OPAMPS: Design, Application, & Troubleshooting*, Prentice Hall (1992).

## 7.6. Laboratory Exercises

### *16 MHz oscillator*

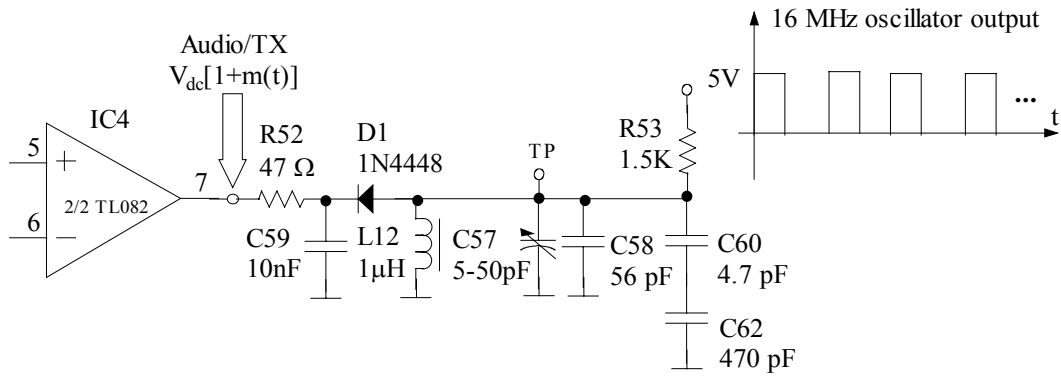
1. Study the data sheet of 16 MHz oscillator module. Note that the power supply circuit of this module is connected to +15 V through TX/RX switch. We want this oscillator to operate only when we are transmitting. Therefore we switch its power off during reception. Install and solder D5 and C66. Install and solder the C66 end of R55. Using a piece of wire, make a soldered jumper connection from the free end of R55 to +15 V supply. Switch the power on. Measure and record the d.c. voltage across D5. It must be about 5 V. Switch the power off.
2. Install the IC8 module taking care of correct pin positions, and solder. Switch the power on.
3. Connect the oscilloscope probe to output pin, and observe the square wave at 16 MHz generated by the module. Measure and record the p-p amplitude (must be about 4-5 V average amplitude). Measure and record the frequency using a frequency counter.

### *Amplitude modulator*

4. Amplitude modulator of TRC-10 is given in Figure 7.10.

The operation concept of this circuit is given in Section 7.1. There are few deviations from the circuit of Figure 7.2, in this circuit:

- a. The 16 MHz oscillator and a 1.5 K resistor, which couples the output voltage of oscillator to the circuit, replace the current source.
- b. Parallel capacitance is about 100 pF and is a combination of C57, C58, C60 and series C62. We need a voltage division between C60 and C62 to have a proper signal amplitude at IC7 input.
- c. D1 is a real diode rather than an ideal one.
- d. There is a R52-C59 section between the audio signal and the modulator.



L12: 18 turns on T37-7

Figure 7.10 Amplitude modulator of TRC-10

Calculate the Norton equivalent of the circuit formed by 16 MHz oscillator output and R53. Re-draw the tuned circuit after replacing this part of the circuit by the Norton equivalent and replacing C57, C58, C60 and C62 by their equivalent capacitance. What is the resonant frequency? What is the Q of the circuit? What is the maximum amplitude of the square wave current that can flow into the tank circuit? What is the maximum amplitude of the voltage that can appear across the tuned circuit if there were not any diode?

5. A good voltage source output is required at the analog side, for proper operation of this modulator. This requirement also includes a very low output impedance. (recall that the output impedance of an ideal voltage source is zero!). Analog voltage is presented at the output of a TL082 OPAMP. TL082 has very low output impedance at low frequencies. What is it at 1 KHz (approximately)? This circuit has current components at 16 MHz and at its overtones. We do not have any data at these frequencies on TL082, but we know that the output impedance can be considerably high (How?). Hence the output impedance of TL082 by means of R52 and C59 to such a form that it presents reasonably low impedance at all frequencies.

Calculate the Thevenin equivalent circuits Both the voltage and the impedance seen at the diode side at 1 KHz, 16 MHz and 48 MHz.

6. Diode D1 is a 1N4448 signal diode. It is not ideal. We know that if a diode is not ideal, we may suffer certain problems when the voltage across it is small. We also discussed that a possible solution is to bias it.

The rigorous analysis of this modulator circuit (which is not difficult but is definitely beyond the scope of this book) shows that there is always some bias current which improves the diode performance in this circuit. Hence consider D1 ideal, as long as the modulation index is not close to 100%.

7. Install and solder R52, R53, C57, C58, C59, C60, C62 and D1. Wind 18 careful and loving turns on a T37-7. Install and solder L12 after carefully trimming and soldering the leads.



8. Check the connections both visually and using a multimeter. Make sure that there are no short circuits and all your solders are well. Making good solder joints are particularly important in RF circuits.
9. Compensate both of your probes.
10. Connect the probe of channel 1 at the output of the microphone amplifier (an appropriate place) and the probe of channel 2 across L12. Switch on the power. Set potentiometer R31 to somewhere at mid-range. Adjust R28 such that approximately 1.2 V d.c. is observed on channel 1.

Now tune the tank circuit to the frequency of the 16 MHz oscillator by adjusting C57. You must observe a good (no distortion) sine wave across the tuned circuit (on channel 2) when it is properly tuned. Measure and record the amplitude of the sine wave. Compare this amplitude with the d.c. voltage at the output of microphone amplifier.

Switch off the power.

11. Connect the signal generator to the input of microphone amplifier as in Figure 3.28. Connect the probe of channel 1 across D3 and D4.
12. Switch on the power. Adjust the signal generator to a sine wave of 1 KHz frequency. Increase the amplitude sufficiently, such that a clipped sine wave of approximately 0.5-0.6 V peak amplitude appears across D3 and D4.

Connect channel 1 probe back to the output of microphone amplifier output. Adjust the scope settings so that you can observe the clipped sine wave on channel 1. Adjust R31 so that you observe 0.6 V peak a.c. voltage superimposed on a 1.2 V d.c., on channel 1 at the output of amplifier.

Keep the oscilloscope triggered by the signal on channel 1 and adjust the scope settings so that you can observe the signal on channel 2 connected across L12. It must be an approximately 4 V p-p amplitude AM signal at 16 MHz. Re-adjust C57 for maximum swing, if necessary.

Adjust R28 for 50% amplitude modulation. Record the peak and valley amplitude. (You may need to refresh your knowledge on AM given in Chapter 1). Switch the power off.

Since the probe of channel 2 was connected across the tank circuit during tuning, the probe tip capacitance  $C_p$  (please refer back to Laboratory exercise 3 of Chapter 3) also appears in parallel with the tank circuit. Hence the tank circuit is tuned to a lower frequency than what is required when the probe is not connected. We shall therefore adjust C57 after installing TX mixer.

13. What is the expected frequency content of this signal when the modulating signal is a pure sine wave? How many sine waves are there in the modulated signal? What are they?

**VFO**

14. VFO of TRC-10 generates sine wave at frequency range of 12-13.7 MHz, approximately. VFO is an OPAMP oscillator circuit, which uses a LM7171 as the gain element. It has a basic oscillator part and a frequency variation sub-circuit. The *variable frequency* property is provided by two parallel varactors and their d.c. bias circuits. The basic oscillator circuit is depicted in Figure 7.11.

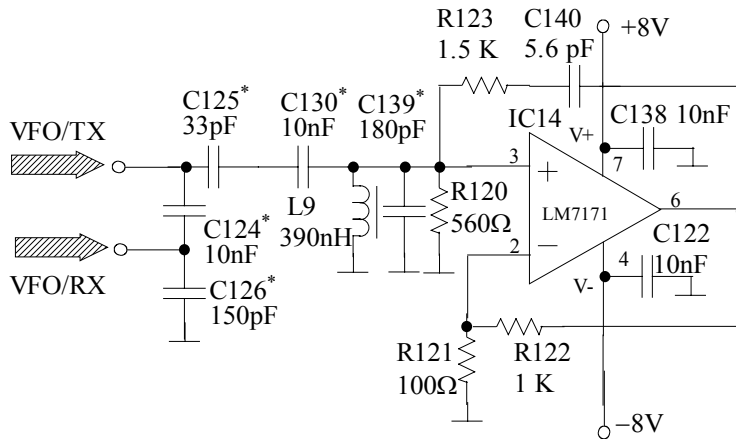


Figure 7.11 Basic oscillator circuit

The gain providing part is the OPAMP with resistive feedback in Figure 7.11. This part is shown in Figure 7.12, separated from the rest of the VFO circuit.

R121 and R122 are the resistors of the feedback block. C138 and C122 are supply by-pass capacitors.

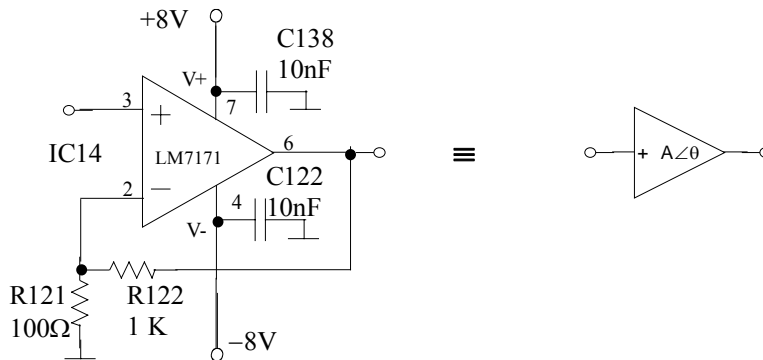


Figure 7.12 Gain block of VFO

15. Calculate the gain  $A$  and its phase  $\theta$  at 16 MHz for this amplifier, making use of the data in the datasheet.
16. The parts of VFO basic circuit are shown in Figure 7.13.

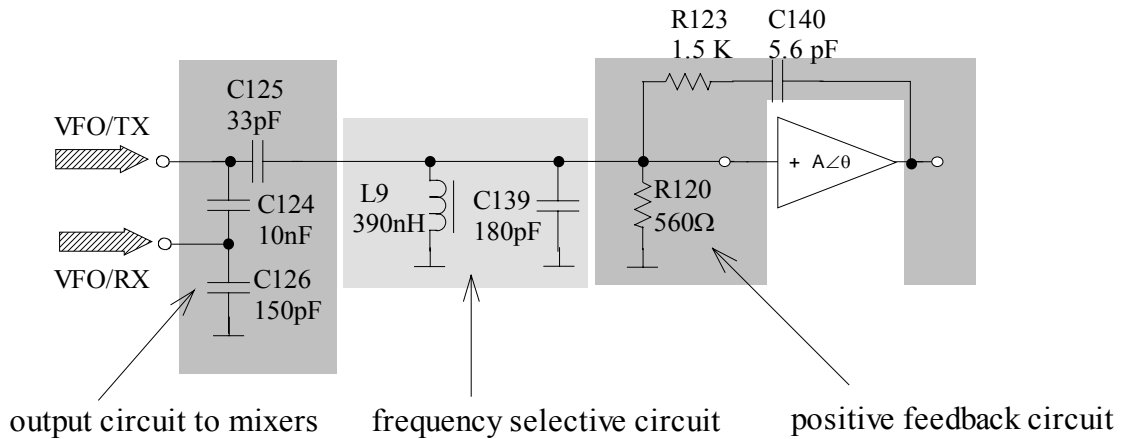


Figure 7.13 Blocks in oscillator circuit

The output circuit must both attenuate the oscillator output to that level required by SA602A. Also it must decouple the circuits d.c. wise.

The output of the OPAMP is a square wave. It is fed back by R120, R123 and C140 filtered into a sine wave across the tuned circuit. Series C140 compensates for the phase  $\theta$  calculated in 15. What must be the exact value of C140 so that the voltages at pins 2 and 3 of the OPAMP have the same phase?

Calculate the positive feedback ratio.

17. The frequency selective circuit has an inductor L9 and a parallel capacitor C139. However other capacitors in the feed back circuit and the output circuit also affect the resonance. The equivalent parallel capacitance can be calculated from Figure 7.14. C130 is a large d.c blocking capacitor used in varactor circuit.

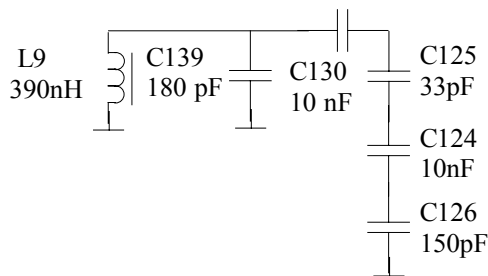


Figure 7.14 All capacitors in parallel with L9

The equivalent parallel capacitance is approximately 250 pF. This capacitance sets the resonance frequency to about 16 MHz with 390 nH. Any other capacitance that appears in parallel with C139 pulls this frequency downwards. There is additional parallel capacitance due to PCB connections of this circuit.

18. Install and solder IC14, taking care of correct placement of the pins. Install and solder the supply by-pass capacitors C122 and C138. Install and solder R120, R121 and R122. Do not install R123 and C140 yet.

19. Switch the power on. Measure the supply voltages and make sure that they are all right. Measure the d.c. levels of the OPAMP output, and both inputs, which must be zero nominally (remember that the presence of input bias currents and offset voltage can set the input to few mV and output to few hundred mV d.c., as discussed in Chapter 3). Switch the power off.
20. We use a T37-7 core for L9. Calculate the number of turns necessary for L9. Wind the inductor, install and solder. Install and solder C124, C125, C126, C130, C139, C140 and R123. Check the joints.
21. Switch the power on. Connect the probe of the oscilloscope across the OPAMP output. The lead of R123 (or R122) connected to pin 6 may be a better place to hook up the probe. Observe the square wave signal. Measure and record its amplitude and its approximate period. Measure the frequency of this square wave.
- It is possible to observe the sine wave across the tank circuit with the oscilloscope. We can measure the sine wave across the VFO/RX and VFO/TX pigtailed (i.e. across C126 and C124), safely. The capacitive loading created by the probe is insignificant when compared with the large capacitor C126. Measure and record the amplitude of these sine wave signals. Switch the power off.
22. Frequency variation for VFO is maintained by two parallel varactors in a single case, KV1360NT. The d.c. reverse bias voltage is adjusted appropriately to change the total capacitance of two diodes, which are connected in parallel with C130 through a large (d.c. blocking) capacitor C130. The varactor circuit of VFO is shown in Figure 7.15. The only function of C130 is to isolate the d.c. bias voltage of varactor diodes from the tank circuit. Calculate the reactance of C130 at 16 MHz. Can this reactance be ignored when compared with the reactance of varactor diode?

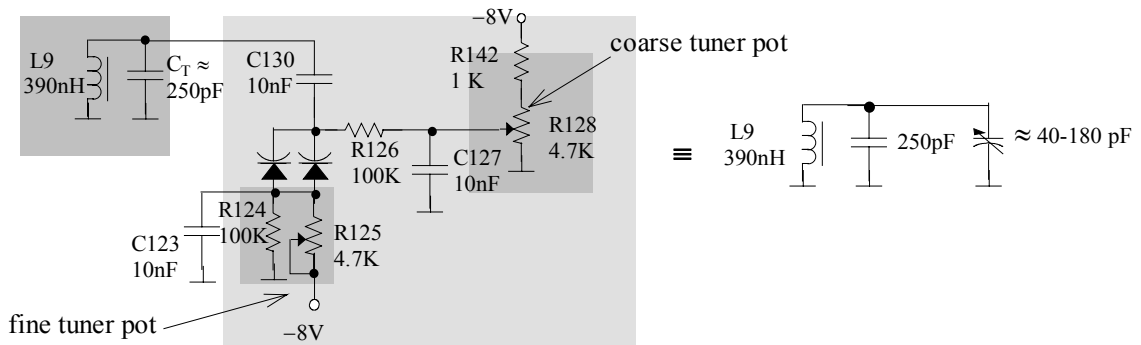


Figure 7.15 Tuner circuit

Calculate the d.c. potential range that can be produced at the cathode of the diodes by the *coarse tuner potentiometer*, and the potential range at the anode by *fine tuner potentiometer*. C123 and C127 are large capacitors to provide a stable d.c. bias voltage for the varactors.

The approximate variation of the capacitance of KV1360NT with respect to reverse bias voltage is given in the datasheet in the appendix. Determine the tuning range of total varactor capacitance in this circuit.

23. Install and solder C123, C127, R126, and R124. Install and solder the PCB connectors J24 and J25. Install and solder the varactor diodes D10, taking care of correct polarity. Check the connections.
24. R125 is a panel potentiometer for fine tuning. Short the middle pin and one of the side pins of R125. Cut two 15 cm pieces of wire and solder them to the respective pins of R125. Crimp the contacts for PCB jack to their other ends. Mount R125 on the panel, using a pair of pliers. Use anti-slip washer. Take care not to round the nut. Fit an *enumerated skirt* and a knob on the pot swindle. Fit the jack.
25. Install and solder R142. R128 is a panel potentiometer for coarse tuning. Cut three 15 cm pieces of wire and solder them to the respective pins of R128. Crimp the contacts for J24 PCB jack to their other ends. Mount R128 on the panel similar to R125. Again fit an enumerated skirt and a knob on the pot swindle. Fit the jack J24.
26. Hook up a frequency counter to the R122 leg connected to pin 6 of LM7171. Switch on the power. Find the tuning range of the oscillator. We want the tuning range to cover 12-13.7 MHz range. If this range is not covered, modify L9 for proper frequency adjustment.
27. Set fine adjustment pot to minimum. Set each step of the enumerated skirt of *coarse tuner* potentiometer to a pointer, and measure the corresponding frequency using the counter. Make a record of your measurements by filling the following table.

0	1	2	3	4	5	6	7	8	9

### **Mixers**

28. We use three SA602A integrated circuits produced by Philips, as mixers in TRC-10. SA602A has a Gilbert cell made up of transistors. SA602A is a very popular and low-cost analog multiplier. It can be used in applications up to 500 MHz.

There is also SA612A, which is almost exactly the same as SA612A, with slightly relaxed specifications. As far as TRC-10 is concerned, SA612A is a direct replacement for SA602A. The data sheet of SA602A is given in Appendix D.

Examine the data sheet carefully. The following discussion is based on the information given in the data sheet.

As in the case of an OPAMP, internal circuit configuration is not relevant here. All we need to know is how to use this IC in our circuits. The information about the power considerations and the input/output specifications of SA602A as a

block is sufficient for our purposes. The block diagram and pin configuration of SA602A is given in Figure 7.16.

SA602A has an internal amplifier for oscillator. In other words, a local oscillator can also be directly implemented within this mixer. This amplifier input is pin 7 and oscillator output is pin 6. However we use external oscillators in TRC-10. The external oscillator signal is applied through pin 6. We do not use pin 7 at all, and this pin is *not connected* (NC) in our circuits.

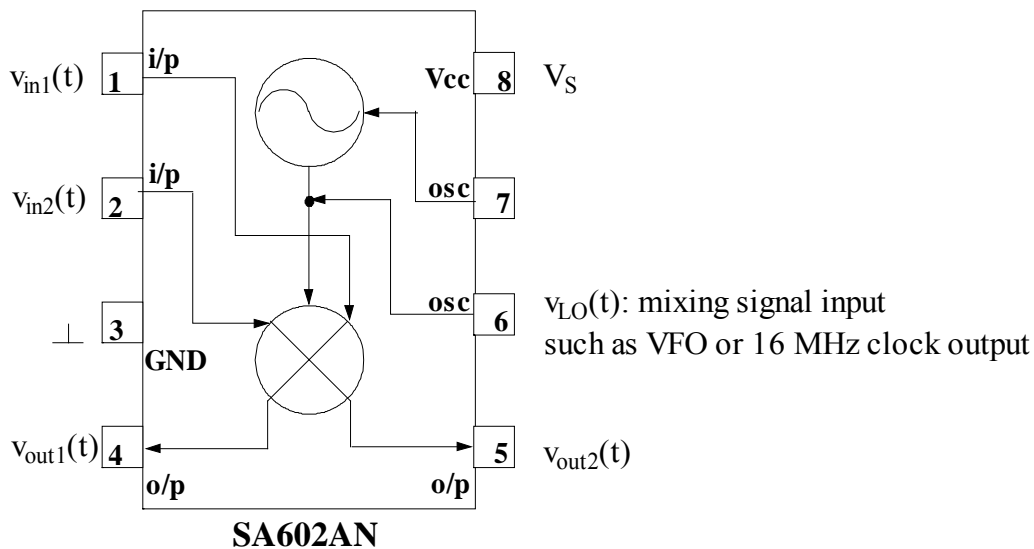


Figure 7.16 SA602A block diagram

SA602A operates from a single supply voltage and supply pin is pin 8. The ground pin is pin 3.

SA602A has two input pins. Either one can be used. Input pins have a bias voltage of about 1.4 V d.c. The external circuit must not disturb this voltage level. A d.c. blocking capacitor in series with the input pin is necessary.

When SA602 is operated from a single input, the other input pin must be very effectively by-passed to ground, with capacitors.

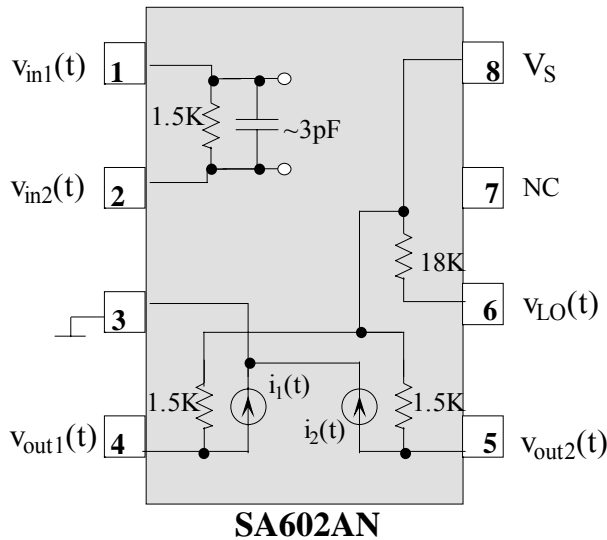
We can also use both input pins of SA602A *differentially*. The voltage difference between the two pins is used in mixing, in that case.

When used as a mixer local oscillator must be at least 200 mV p-p, and must not exceed 300 mV. 200 mV p-p amplitude a square wave shaped internal  $v_{LO}(t)$  signal, as we discussed in Section 7.3.

The two input terminals have impedance of approximately 1.5 K resistance in parallel with about 3 pF capacitance. The impedance of the local oscillator input terminal (between pin 6 and 3) is about 18 K.

An equivalent circuit of output terminals is given in Figure 7.17. The two outputs of SA602A are connected to the supply pin through 1.5 K resistors. The output

signals are supplied by symmetrical *current sources*  $i_{in1}(t)$  and  $i_{in2}(t)$ . The current sources have a d.c. component of 0.8 mA, as well as the signal components described in Section 7.3 and delineated in Figure 7.17. The signal component amplitude is also 0.8 mA. The scaling constant  $k$  is  $(52 \text{ mV})^{-1}$ , approximately.



$$i_1(t) \approx (0.8 \text{ mA}) \{1 + \tanh[k(v_{in1} - v_{in2})] \times \tanh[kv_{LO}]\}$$

$$i_2(t) \approx (0.8 \text{ mA}) \{1 - \tanh[k(v_{in1} - v_{in2})] \times \tanh[kv_{LO}]\}$$

$$k \approx 1/(2 \times 26 \text{ mV})$$

Figure 7.17 Equivalent circuit of SA602A input and output

The circuit configuration and the d.c. component in the current sources yields a d.c. voltage level of  $V_S - (0.8 \text{ mA}) \times (1.5 \text{ K}) = V_S - 1.2 \text{ V}$  approximately. External circuits must not disturb this voltage.

The specifications of SA602A are applicable when the IC terminals are properly terminated. The external circuit must provide 1.5 K resistance to output pins.

### ***UP-conversion to 29 MHz with SA602A***

29. Install and solder IC7, the TX mixer, which converts 16 MHz signal to 28-29.7 MHz, taking care of correct pin alignment. Make sure that pin 8 corresponds to 8 V supply pad.
30. Switch the power on. Using a multimeter, measure and record the d.c. supply voltage at pin 8, d.c. voltages at two input pins and d.c. voltages at two output pins. Compare them with the expected values given in Exercise 28. If you cannot observe these voltages at respective pins, then there must be something wrong in your circuit.
31. Check the connections both visually and using a multimeter. Make sure that there are no short circuits and all your connections are properly soldered. Making good solder joints are particularly important in RF circuits.

32. Install and solder C61 and C63. Desolder the ground plane constructed resistors connected to C64, 51  $\Omega$  and 1.5 K. Install and solder the free lead of C64.
33. Calculate the voltage amplitude that appears across pin 1 of IC7, using the values of C60 and C62, and the measurement made in Exercise 12.
34. Connect the probe of channel 1 the output of the microphone amplifier. Switch on the power on. Adjust the signal generator to a sine wave of 1 KHz frequency. Adjust the amplitude such that a clipped sine wave of approximately 1.0V pp appears across D3 and D4.

Connect the probe of channel 2 across the pin 6 of IC9 (rather on TP on R58). This is a comfortable point to connect probes. You must observe a few hundred mV amplitude AM signal at approximately 29 MHz. Re-adjust C57 for maximum amplitude.

If necessary, re-tune C65 for maximum amplitude on channel 2 (*this should not be necessary if you have not played with C65 after it is tuned*).

Re-adjust R28 for 50% amplitude modulation, if necessary. Record the peak and valley amplitude.

35. Connect the probe of channel 2 to the output of IC 10 (possibly at IC10 side of R61), instead of the output of IC9. Observe the AM signal.

Both oscillators must be active and connected to respective mixers in order that the AM signal is generated. Make sure that the jumper connection for +15 V supply made in Exercise 1, is still connected.

Adjust C69 for maximum amplitude, if necessary. This must be an AM signal of approximately 6 V peak without distortion. Adjust R28 again if necessary for maximum amplitude. Measure and record the peak amplitude at IC 10 output and the modulation index at this amplitude.

36. Remove the supply jumper to 16 MHz oscillator. Place the free end of R55 into the hole and solder. Now the 16 MHz oscillator can be active only when the S2 is thrown to TX.

### ***RX mixer***

37. All the input and output circuits of the RX mixer are already installed and used. Install and solder IC11 (another SA602A). Take care of correct placement. Install the supply bypass capacitor C96, and input bypass capacitor C93.
38. Repeat Exercise 30 for IC11.
39. We need a very weak (low power) signal at the antenna input to test the RX chain. The signal generator output does not provide such low level signals. We must use an attenuator at the antenna input. We use the circuit given in Figure 7.18.



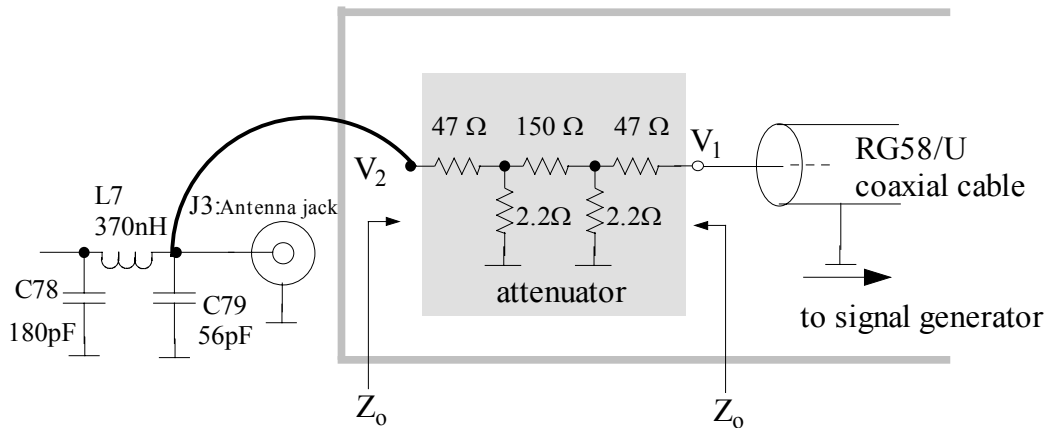


Figure 7.18 Signal generator connection for RX sensitivity measurement

Show that the impedance  $Z_0$ , at both ends of this attenuator is approximately  $50 \Omega$ . Calculate the attenuation from  $V_1$  to  $V_2$  in dB, considering that the attenuator is terminated by  $50 \Omega$  at antenna jack side.

40. The circuit enclosed in gray bracket must be mounted using ground plane construction technique. The cable connecting the signal generator is the coaxial cable with a BNC on one end. Find a clear spot on the PCB near the harmonic filter and construct the circuit in gray bracket. Connect attenuator output to J3-C79-L7 node using a very short piece of insulated wire or connect the free lead of  $47 \Omega$  directly.
41. Set the signal generator to 50% *internal AM modulation* (this produces an AM signal at the output with 1 KHz modulating signal), and the carrier frequency to 29 MHz. Set the output to 20 dBm.
42. Connect the Channel 1 probe to IC12 output (R93 is a good place to connect) and Channel 2 probe to Audio/RX pigtail (across C108). Switch on the power.
43. Set the RF gain pot to maximum gain and audio gain pot to a medium position.
44. Initially you will not see any signal at all. This is because your VFO is off tuned. Using the coarse tuning pot of VFO try to tune to 29 MHz. As you pass 29 MHz, you will hear the whistle out of the speaker. Use the fine tuner pot to tune the VFO to exactly 29 MHz (or maximum sound level).
45. Measure and record the peak and valley amplitude at the output of IC12, and the amplitude and the frequency of the signal across C108. Using this data, calculate the composite gain (conversion gain + amplification) up to the output of IC12.
46. Adjust C92 for maximum amplitude on Channel 1, and C104 for maximum amplitude on Channel 2, if necessary.

47. Gradually decrease the signal generator output until you cannot hear any thing. Record this signal generator output in dBm. Calculate the power delivered to the antenna input in dBm. This is the *sensitivity* of your receiver (subjectively, at least).
48. Install J20 and solder. Make the connections for S2/2, install and solder. Remove the jumper from +15 bus to pin 1 of IC5, you connected in Exercise 3.13.
49. Switch the power off. Disconnect the signal generator and the probes. De-solder and remove the ground plane constructed circuit.

You now have a working transceiver, TRC-10! You need an antenna.

### 7.7. Problems

1. Calculate the forward gain of the amplifier and the feedback ratio in the VFO circuit at the VFO frequency.
2. What is  $v(t)$  in the circuit given in Figure 7.1, if  $IR < V_{dc}$ ?
3. Consider the circuit in Figure 7.19(a). Find the transfer function  $H(\omega) = V_2(\omega)/V_1(\omega)$ . Determine the frequency at which  $\angle H(\omega) = 0$ , when  $R_3 = R_4$  and  $C_3 = C_4$ . What is  $|H(\omega)|$  at that frequency?
4. The circuit shown in Figure 7.19(b) is called the *Wien-bridge oscillator*. The circuit given in Figure 7.19(a) provides the positive feedback. The circuit oscillates at the frequency  $\omega_0$  where  $\angle H(\omega_0) = 0$ , if the condition  $(1+R_2/R_1)H(\omega_0) > 1$  is satisfied. What is the minimum value of  $R_2/R_1$  so that the circuit can oscillate, if  $R_3 = R_4$  and  $C_3 = C_4$ ?

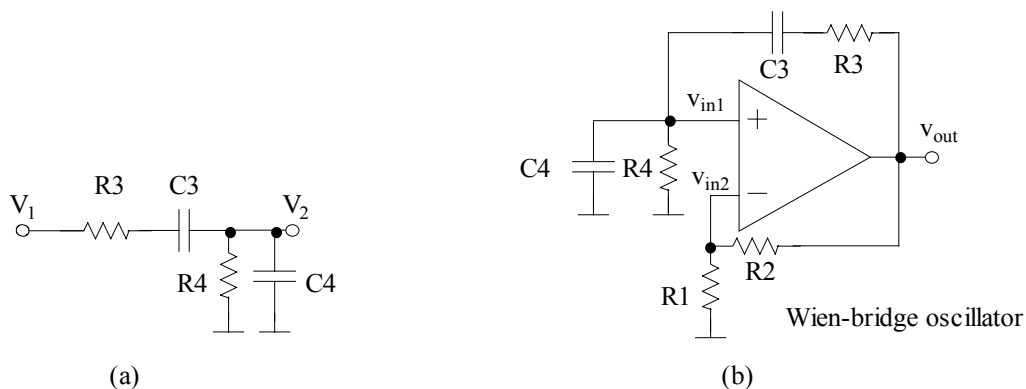


Figure 7.19 Circuits for (a) problem 3 and (b) problem 4.



## Chapter 8 : ON THE AIR

Radio waves are combinations of electric and magnetic fields and therefore they are usually called electromagnetic waves. Antennas convert electrical energy into electric and magnetic fields for transmission, and electromagnetic energy to electrical energy for reception.

Antennas are combinations of pieces of conductors of specific lengths (sometimes area) and shapes. There are many different types of antennas for numerous applications. A fundamental antenna form is a *dipole* antenna. We use a dipole in TRC-10.

Antennas belong to the class of devices called *transducers*. Transducers convert one form of energy into another. Loudspeaker, for example, is a transducer which converts electrical energy into sound (or mechanical) energy, and microphone converts sound into electrical energy. Antennas provide transduction between electrical and electromagnetic energy.

What follows in this chapter is a descriptive theory of electromagnetics and antennas.

### 8.1. Antenna concept

In electronic circuits, capacitors, inductors, other components and their interconnections are small compared to the wavelength at the frequency used. We defined wavelength  $\lambda$  as

$$\lambda = (3.0 \text{ E}+8 \text{ m/sec})/f$$

where  $3.0 \text{ E}+8 \text{ m/sec}$  is the velocity of electromagnetic waves (and also light) in free space,  $f$  is frequency in cycles/second (Hz) and wavelength is in meters. Wavelength can also be interpreted as the distance electromagnetic wave travels in one full cycle.

When the circuit dimensions are small compared to the wavelength, most of the electromagnetic energy generated by the circuit is confined to the circuit. It is either conserved for the desired purpose or converted to heat. When the dimensions of the components or the interconnections become large (e.g. comparable to the wavelength) part of the energy escapes into space in form of electromagnetic waves. This part of the energy used in the circuit appears as lost (or “dissipated”) energy to the circuit, whereas it provides a source for the electromagnetic waves in space.

Antennas are devices, which makes use of this conversion-and-escape mechanism to produce radio waves as efficiently as possible.

#### 8.1.1. Radiation from an antenna

A dipole antenna is made up of two pieces of conductor wires or poles aligned together with a small isolating gap in between. A dipole is shown in Figure 8.1. The generation of electromagnetic waves by a dipole is also shown in the same figure, conceptually.

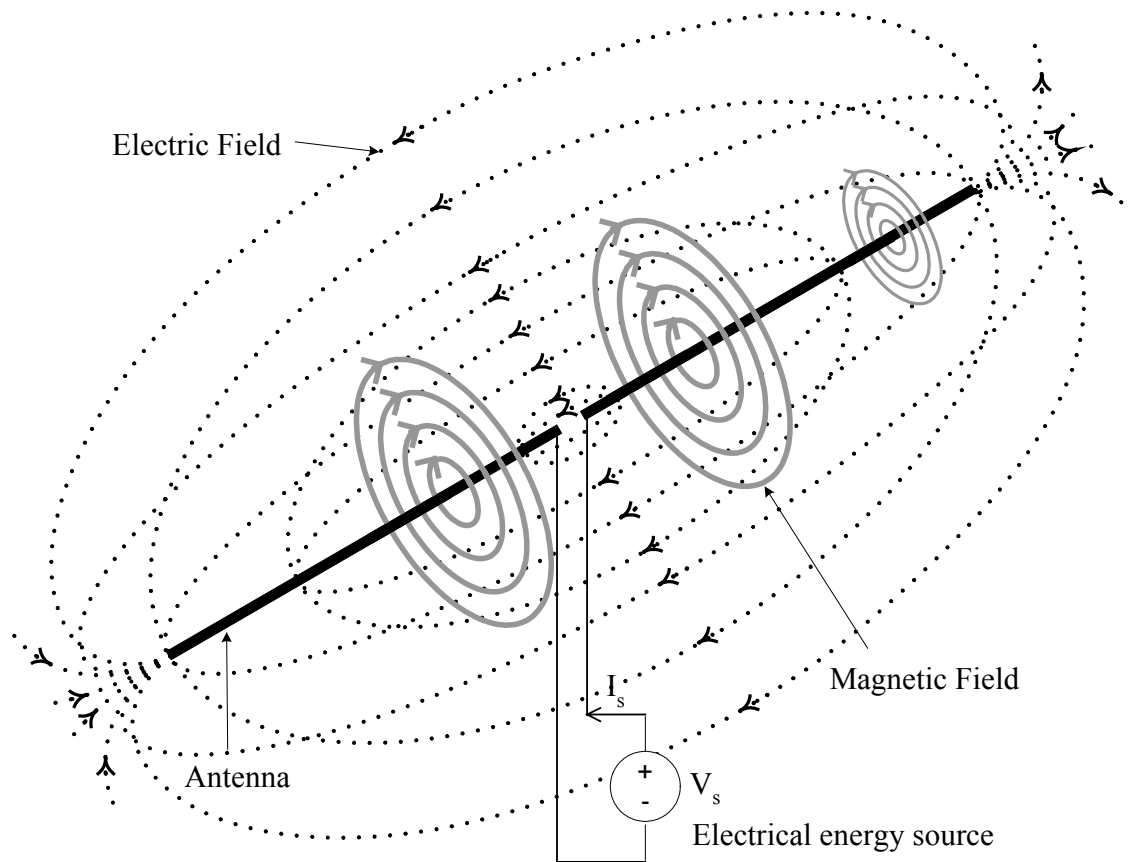


Figure 8.1 Dipole antenna and radio wave generation concept

When a voltage source  $V_s$  is connected to these two conductors across the gap, as shown in Figure 8.1, an electric field between the entire surfaces of two conductors is produced due to the potential difference between them. This field is time varying when the source is a non d.c. source. Its frequency is equal to the frequency of the voltage source. Electric field is denoted by letter  $\mathbf{E}$  and it has units of V/m.

The electric field extends to the entire space, but its strength decreases as the observation distance from the dipole increases. The electric field is strongest near the gap.

The two conductor surfaces constitute a *distributed* capacitance across the terminals at which the voltage source is applied. This phenomenon is depicted in Figure 8.2(a). The current  $I_s$  supplied by the source, leaks from one conductor to the other along the length of the conductor, through the capacitive path. The current amplitude decreases as we move along the conductor. Current diminishes at the tip of the conductor. A typical current distribution along the antenna is given in Figure 8.2(b).

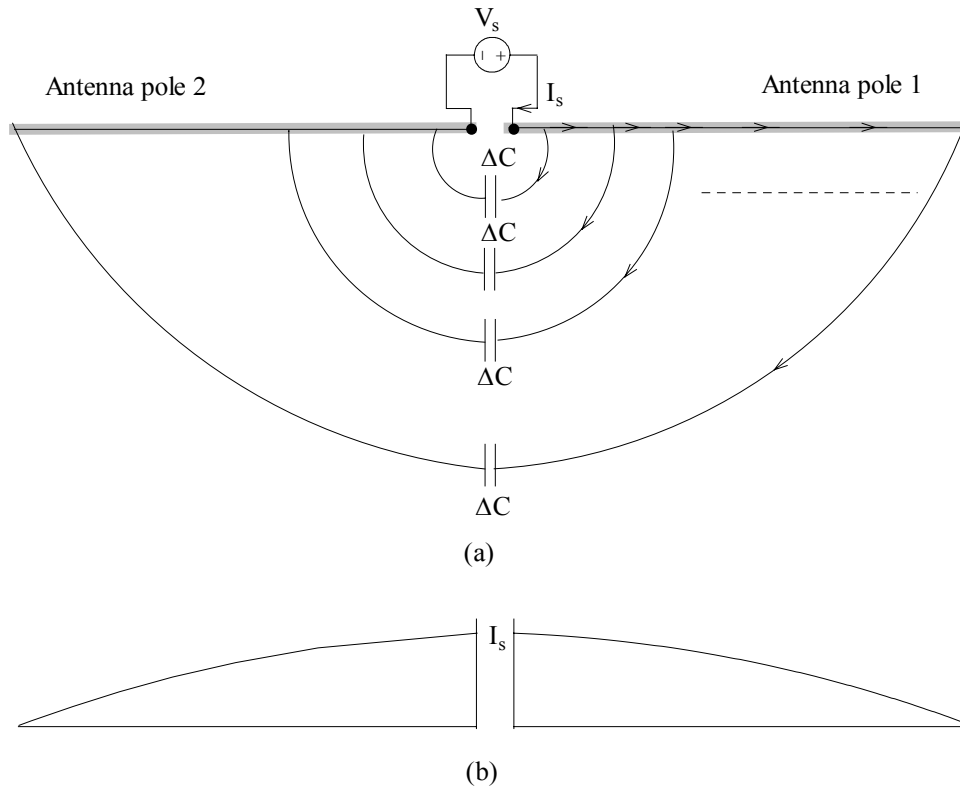


Figure 8.2 (a) Distributed capacitance on antenna, and (b) current amplitude distribution along antenna

The current along the antenna produces a magnetic field shown in Figure 8.1. Again, the frequency of the magnetic field is the same as that of the voltage source. Letter  $\mathbf{H}$  denotes the magnetic field and it has units of A/m. Electric and magnetic fields have magnitude and direction and hence they can be modeled as vectors.

During propagation in space, some components of  $\mathbf{E}$  and  $\mathbf{H}$  vectors decay very quickly away from the dipole. Only *orthogonal* components (mutually perpendicular components) of the electric and magnetic fields are maintained during propagation at far away distances. When we observe the electromagnetic wave emanating from a dipole at a large distance, the equal phase surfaces (the surface defined by electric and magnetic waves which have the *same* phase; the fields which have the same phase must have left the antenna at the same instant- phase is like age), called *wavefront*, appear like concentric spheres. The center of these spheres, which is called the *phase center*, coincides with the center of the isolating gap in the dipole. This is shown in Figure 8.3.

The direction of propagation in this figure is outward from the center. The electromagnetic wave generated at some instant gets away from the dipole in all directions, at a speed of  $3.0 \text{ E}+8 \text{ m/sec}$ .

Now let us take a closer look at the field shown on the patch over the spherical surface, in Figure 8.3. If the radius of the sphere is very large compared to the rectangular patch (which is a very realistic assumption for practical antenna discussions), the patch approximately defines a planar surface. We can define a

Cartesian plane on which electric field coincides with x-axis and magnetic field coincides with y-axis. This is shown in Figure 8.4.

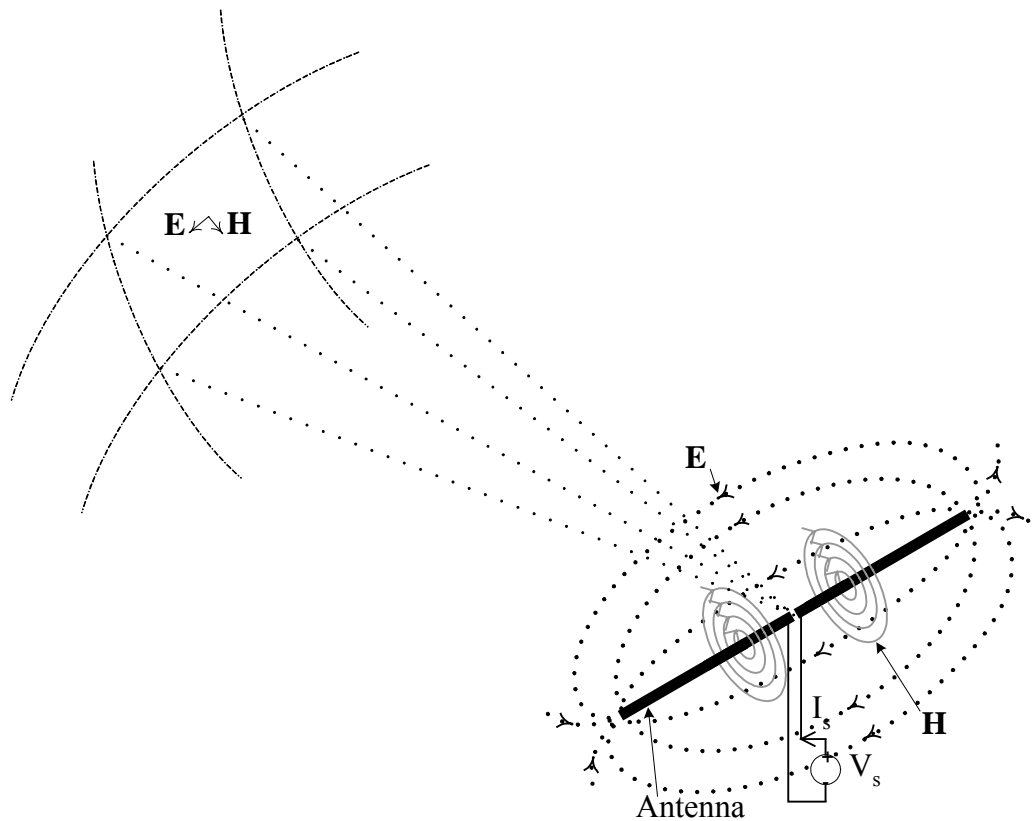


Figure 8.3 Radio wave far away from the source dipole

With  $E_x$  and  $H_y$  are phasors of the electric and magnetic fields respectively, and  $\mathbf{a}_x$  and  $\mathbf{a}_y$  are unit vectors in x and y directions, we can write the two field vectors as

$$\mathbf{E} = E_x \mathbf{a}_x$$

and

$$\mathbf{H} = H_y \mathbf{a}_y$$

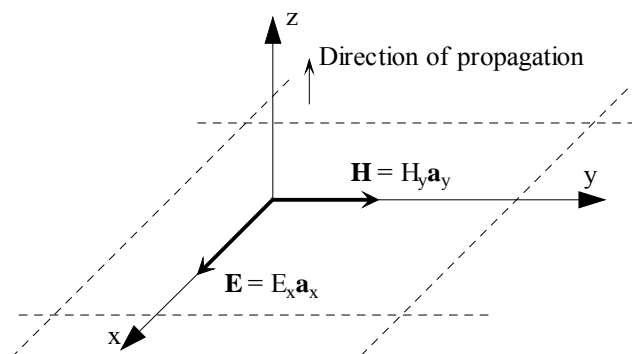


Figure 8.4 Orthogonal electric and magnetic fields and the direction of propagation

$E_x$  and  $H_y$  in an electromagnetic wave propagating in space are related to each other as

$$E_x/H_y = \eta_0 = (\mu_0/\epsilon_0)^{1/2}.$$

Here  $\mu_0$  and  $\epsilon_0$  are the permeability and the permittivity of the free space (and air), respectively. Their values are

$$\mu_0 = 4\pi \times 10^{-7} \text{ H/m}$$

and

$$\epsilon_0 = 8.85 \text{ E-12 F/m.}$$

$\eta_0$  is called the wave impedance and can be calculated as

$$\eta_0 \approx 377 \Omega \approx 120\pi \Omega.$$

As a matter of fact, the speed of light is also related to permeability and permittivity of free space as

$$c = 1/(\mu_0\epsilon_0)^{1/2}.$$

There is a continuous flow of energy in the direction of propagation in electromagnetic waves. This power flow is quantified in terms of *power density* of the electromagnetic wave. The average power density in z direction of the wave depicted in Figure 8.4, is given in terms of the product of the electric field phasor and the complex conjugate of magnetic field phasor as

$$P_{em} = \text{Re} \{E_x H_y^* / 2\} = |E_x|^2 / (2\eta_0).$$

The unit of power density is  $\text{W/m}^2$ . The total power over an area can be calculated by integrating  $|E_x|^2 / (2\eta_0)$  over that area.

Antenna, which radiates in all directions with equal preference is called *omnidirectional* antenna. For example, a dipole whose length is very small compared to the wavelength can be approximated as an omnidirectional antenna. The electric field is uniformly distributed over the spherical equal phase surface, in such antennas. At a distance  $r$  from the antenna, the power density is uniform over the sphere and is given as

$$P_{em} = P_o / (4\pi r^2)$$

where  $P_o$  is the power delivered to the antenna and  $4\pi r^2$  is the area of the sphere (conservation of energy). Hence the electric field at that distance becomes

$$|E_x| = [2\eta_0 P_o / (4\pi r^2)]^{1/2} = (60 P_o)^{1/2} / r$$



where  $P_0$  is in watts and  $r$  is in meters. For example, a transmitter delivering 10 mW to an omnidirectional antenna generates electric field strength of 0.8 mV/m at 1 km distance, in space.

### 8.1.2. Receiving antenna

Antennas are reciprocal devices. They behave similarly in reception. When a dipole is exposed to an electromagnetic wave whose electric field is aligned with the antenna, a voltage is developed across the isolating gap. A receiving antenna in an electromagnetic wave is shown in Figure 8.5.

The open circuit voltage between the two conductors is the potential difference between them. The potential of each conductor is the potential at its mid-point. The dipole has a length of  $l$  meters (where  $l$  is very small compared to wavelength). Hence the received voltage is approximately given as

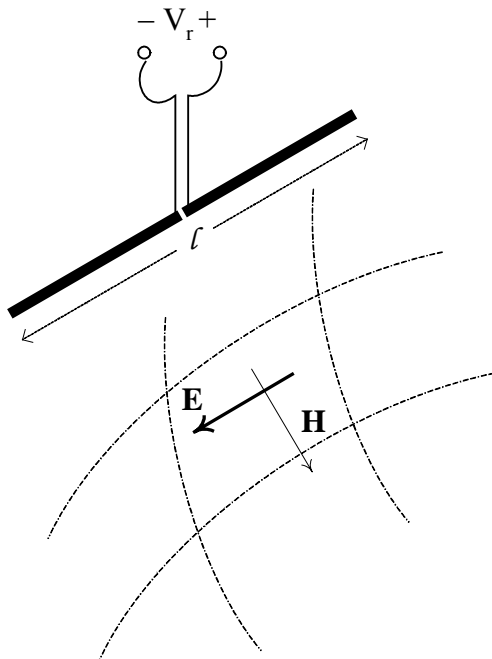


Figure 8.5 Receiving dipole

$$V_r = E_x(l/2).$$

For example, the open circuit voltage developed across a 1.2 m long dipole is 0.48mV when the incident field is 0.8 mV/m.

## 8.2. Antenna impedance

The energy radiated from an antenna appears as lost energy to the circuit, which drives it. As far as the circuit is concerned, the antenna is not different than a piece of circuit with a resistive component, which converts the same amount of energy into heat. It is customary to associate the energy radiated from an antenna by a resistance. This resistance is called *radiation resistance*.

## 8.2.1. Self impedance

The radiation resistance of a short dipole of length  $\ell$ , far away from other conductors, is given approximately in  $\Omega$  as

$$R_r \approx 80\pi^2(\ell/2)^2/\lambda^2 = 20\pi^2(\ell/\lambda)^2.$$

The effective capacitance between the two conductors depends on the diameter of the conductor, as well as its length. This capacitance can be calculated for a cylindrical conductor of radius  $a$  as

$$C_d = \pi\epsilon_0(\ell/2)/[\ln(\ell/2a)-1],$$

where both  $\ell$  and  $a$  are in meters.

These equivalent values allow us to model a short dipole by means of an equivalent circuit given in Figure 8.6.

The radiation resistance (at 29 MHz) and the equivalent capacitance of a 1.2 m long dipole with 2.5 cm diameter are  $2.7 \Omega$  and  $5.8 \text{ pF}$ , respectively.

## 8.2.2. Mutual impedance

Presence of conductors in the vicinity of an antenna affect its radiation impedance. Conductor pieces, ground, which is a lossy conductor are all effective on antennas radiation impedance. Estimation of this effect is a rather involved process, and usually requires the spatial positioning of these non-antenna related conductors to be accounted by numerical techniques.

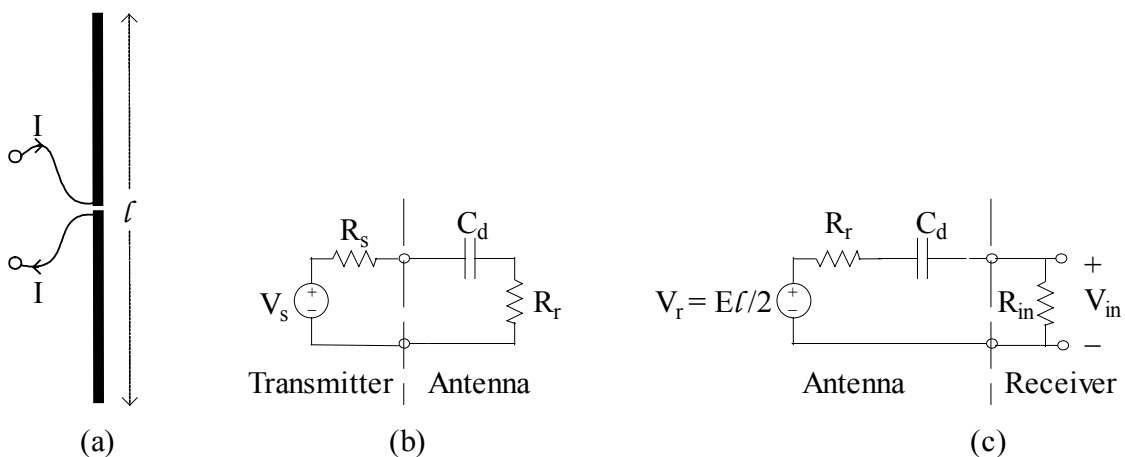


Figure 8.6 (a) Short dipole, (b) equivalent circuit of transmitting antenna, and (c) equivalent circuit of receiving antenna

### 8.3. Half wave dipole

A dipole antenna whose length is a half wavelength exhibits a resonance (actually  $0.48 \lambda$  long dipole). We leave the rigorous analysis of this antenna to more advanced texts on antennas. However, we can discuss the resonance concept for a  $\lambda/2$  dipole from the circuit theory point of view.

Lengths of the poles in a  $\lambda/2$  dipole are considerably larger than that of short dipoles. A  $\lambda/2$  dipole at 29 MHz is 5.2 m long. The inductance of the current path along the poles becomes significant due to this larger length. The Figure 8.2(a) is redrawn for a  $\lambda/2$  dipole in Figure 8.7. The distributed inductance and capacitance combines to produce an equivalent series resonance at  $f = (3.0E+8)/\lambda$  Hz, where wavelength  $\lambda$  is in meters.

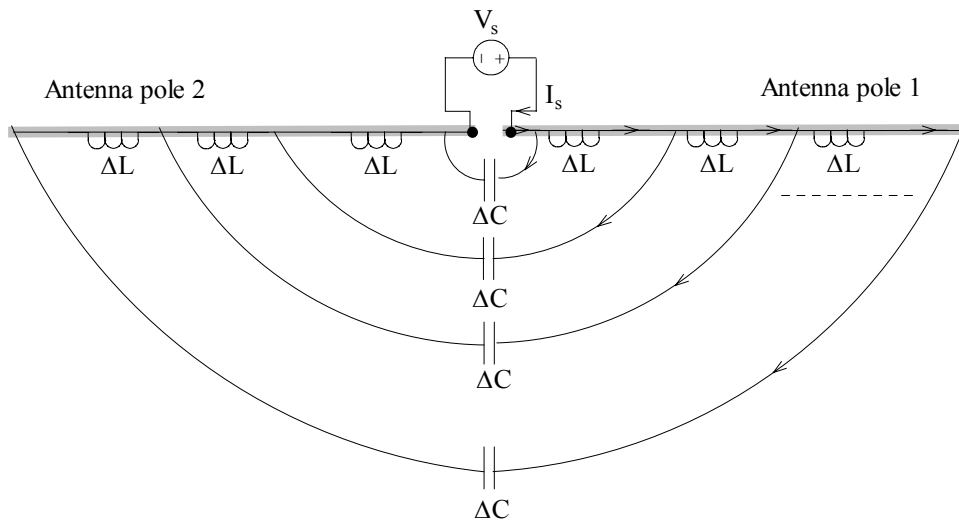


Figure 8.7 Half wave dipole interpreted as a distributed LC circuit

A half wave dipole has the properties of a transmission line as far as input impedance is concerned. The resonance can be modeled as a series resonance with  $73 \Omega$  resistance for a half wave dipole in space. The conducting objects in its near vicinity affect the antenna impedance, as in form of mutual impedance. Since the antenna cannot be far away from ground, the mutual impedance effects pull this resistance to about  $50 \Omega$ .

The bandwidth of half wave dipoles are significantly larger compared to what can be obtained in short dipoles. The quality of resonance at this resonance depends on the diameter of the conductor.  $Q$  decreases as the diameter increases.  $Q$  is less than 6 for an antenna of 2.5 cm diameter at 29 MHz.

### 8.4. Monopole antenna

*Monopole* or *whip* antennas are probably the most commonly encountered antennas. Car radio antennas mobile phone antennas are all this type of antennas. Monopole antennas are derived from dipoles, by eliminating half of the dipole using a reflective *ground plane*. This is shown in Figure 8.8.

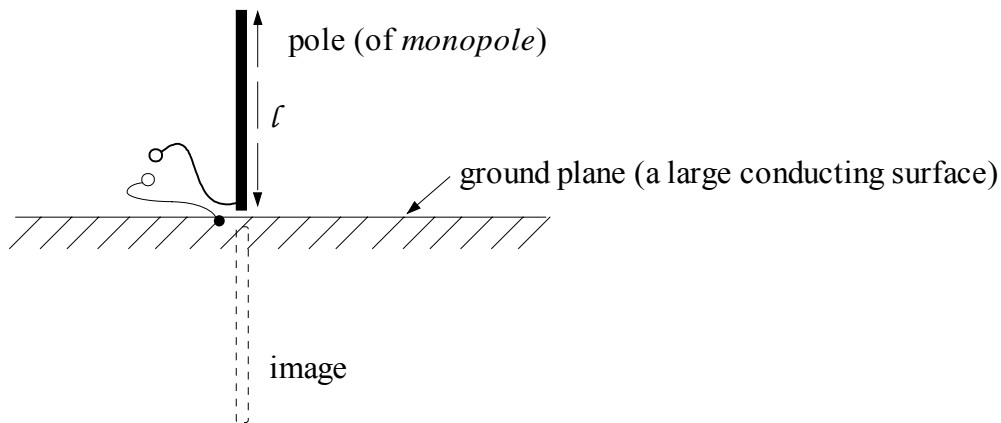


Figure 8.8 Monopole antenna

Conducting surfaces behave like mirrors to electromagnetic waves. As a direct result of this physical property, a pole of length  $\ell$  placed on a conducting surface behaves like a dipole of length  $2\ell$  electromagnetically. Electrical connections are between the pole and the ground plane. The radiation resistance of a short monopole of length  $\ell$  is, however,

$$R_r \approx 40\pi^2(\ell/\lambda)^2,$$

which is twice as much as the resistance of a dipole of length  $2\ell$ . The radiated power in an antenna is

$$P_r = R_r I^2 / 2.$$

For the same input current  $I$ , a dipole (of length  $2\ell$ ) radiates a total power  $P_r$  to the entire space, while a monopole (of length  $\ell$ ) radiates only to *half* space, hence a total power of  $P_r / 2$ . Therefore the radiation resistance of monopole is twice as much as the radiation resistance of an equivalent dipole.

Similarly a monopole antenna of length  $\lambda/4$  placed on a large conducting surface behaves like a half wave dipole.

The effective capacitance between the conductor (of length  $\ell$  and radius  $a$ ) and the ground plane in a monopole antenna is given by

$$C_m = 2\pi \epsilon_0 \ell / [\ln(\ell/a) - 1],$$

where  $C_m$  is in farads and both  $\ell$  and  $a$  are in meters.

In fixed stations where the antennas are installed in places like roofs of buildings, it is possible to simulate a ground plane to an acceptable level by properly designed conducting frame. It is almost never possible to have a large planar conducting surface (compared to the wavelength), on which an antenna can conveniently be placed in mobile stations. The radiation resistance of a monopole is always determined by the mutual impedance of the ground reference in such systems. This can be limited to the dimensions of the casing of a hand-set in case of mobile phones.

Nevertheless, the expression given above is a good approximation to the radiation impedance.

### 8.5. Antenna feeders

Antenna is an electromagnetic transducer, which must be supplied with electrical energy through its electrical terminals. This electrical connection is called *feeder*. Antennas are classified in two groups as *balanced* and *unbalanced*, as far as their electrical input are concerned.

Dipoles are balanced antennas. The input current  $I$  of the antenna flows only through the impedance determined by the antenna parameters  $R_r$  and  $C_d$ , as shown in Figure 8.6. Now if we feed this antenna using an unbalanced feeder like a coaxial cable, we affect the equivalent circuit observed at the antenna terminals. This is depicted in Figure 8.9. One pole is connected to the center conductor of the coaxial line and the other is connected to the shield (the outer conductor) and hence to ground in this case, as shown in Figure 8.9(a). Since there is a direct electrical connection between one pole and the shield, the unbalanced capacitance  $C_u$  between the other pole and the shield (it is not balanced by the a similar capacitance between the shield and directly connected pole to it) cause part of the supplied current  $I_{un}$  to flow through a path which is not between the poles. Hence while the current fed to the isolated pole is  $I$ , the current that returns from the other pole is  $I - I_{un}$ . The two *antenna currents* become unbalanced.

The shield of the coaxial line behaves like an extension of the lower pole and also contributes to the radiated energy in an uncontrolled manner. This part also has an uncontrolled radiation resistance  $R_u$ , which depends on the length, position and shape of the coaxial line. The equivalent circuit of the resulting antenna system is given in Figure 8.9(b).

Another way of looking at this combination is that the shield of the coaxial line, together with the pole connected to it, serves as the ground reference for the isolated pole. This converts the dipole into a monopole with a very poorly designed ground plane.

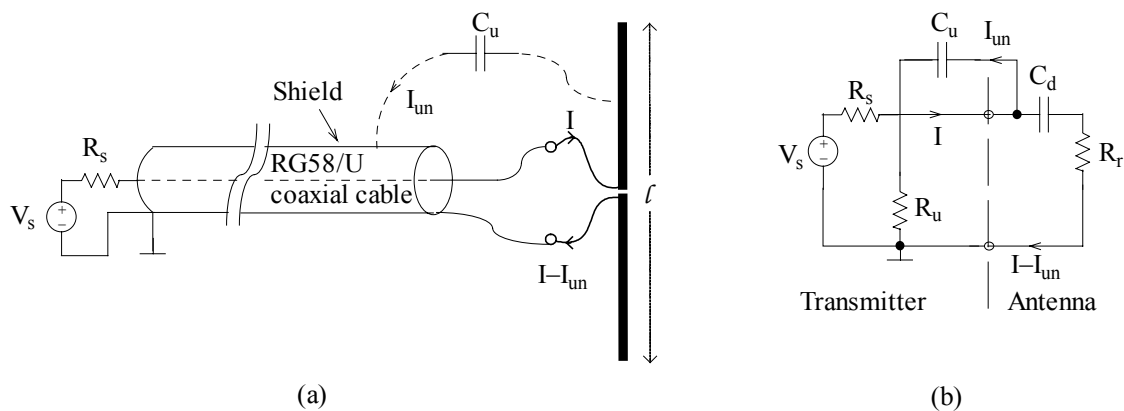


Figure 8.9 (a) Dipole (balanced) driven by an unbalanced feeder and (b) equivalent circuit

Such a feeding configuration is very undesirable for dipoles. Dipoles must be driven by appropriate means, which suits to their balanced nature.

Considerations are different for monopoles. Monopole antennas have a single isolated element and operate with ground reference to start with, as shown in Figure 8.8. The capacitive component in the radiation impedance is between the pole and the ground plane, anyway. They are unbalanced antennas. When the coaxial line feeder is connected to the antenna the center conductor is connected to the pole and the *grounded* shield is connected to the ground plane. This combination is proper for a monopole.

### 8.5.1. Balanced-unbalanced transformation

A straightforward way of driving balanced antennas is to use a balanced feed line like *twin-lead transmission lines*. This approach, of course, requires a balanced transmitter output. These are used extensively in HF for outdoor antenna systems.

The other approach is to convert balanced antenna input into unbalanced load and use coaxial lines. This is more common and is also applicable at higher frequencies. We use *balanced-to-unbalanced* transformation, or *balun* in short, in TRC-10 antenna feeder.

The goal of using a balun is to minimize the unbalanced current component in the antenna. Consider the circuit given in Figure 8.10(a). When large series impedance like a large inductance is inserted, this large impedance isolates the pole and the shield at the radiation frequency. The stray capacitance between this pole and the shield now becomes significant also. The equivalent circuit of this case is given in Figure 8.10(b).

The current that flows into the antenna is now balanced. The effect of the shield of the coaxial cable decreased down to the effect of any conductor in the vicinity of the antenna, and it contributes only to the mutual impedance marginally ( $C_u/2$ ).

On the other hand, large series impedance also impedes the current to the antenna radiation resistance. Coupling any power to the antenna becomes difficult.

There are numerous very clever implementations of baluns. The large inductance is often implemented as a winding of a transformer. A typical balun feeder is given in Figure 8.10(c). This kind of balun is called *current balun*. The turns ratio of the transformer is 1:1 ( $n:n$  in the figure to indicate that actual inductance is large), and therefore  $V_2 = V_1$ . We can write the voltage across the coaxial line  $V_t$  in terms of the voltage delivered to antenna,  $V_a$ , we obtain

$$V_t = V_1 + V_a - V_2.$$

Since  $V_1 = V_2$ , the voltage that appears across the coaxial line is the antenna potential,  $V_t = V_a$ . Antenna impedance (both *mutual* and *self* impedance) appears across the coaxial line, but through a very large series impedance imposed by the windings of the balun transformer.

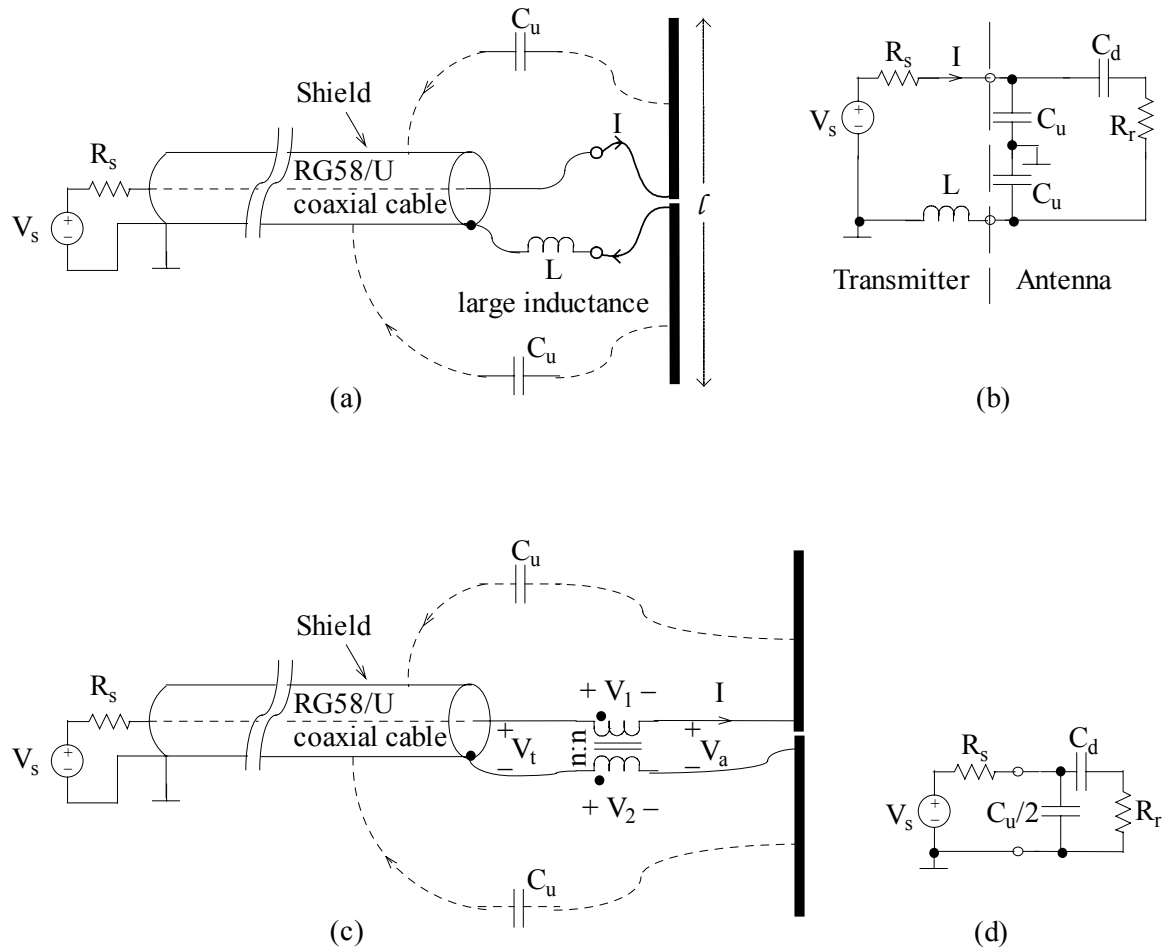


Figure 8.10 Inserting a large series impedance minimizes unbalance current; (a) connections and (b) equivalent circuit, (c) current balun and (d) equivalent circuit of current balun feeder

If the antenna is connected to the line by means of an isolating transformer, the balanced-unbalanced conversion is also maintained. The transformer can also serve to convert the antenna input resistance into nominal termination impedance, like  $50 \Omega$ . Quite often the antennas are  $\lambda/2$ -dipoles or  $\lambda/4$ -monopoles and impedance conversion is not necessary.

The transmitter output of TRC-10 is unbalanced and its output resistance is  $50 \Omega$ . therefore it can drive a  $\lambda/4$ -monopole by a simple connection of a coaxial line. A  $\lambda/4$ -monopole at 29 MHz requires a 2.5 meter long pole and ground plane wiring. For laboratory testing, we use a portable dipole of 1.2 meter length.

### 8.5.2. TRC-10 short dipole feeder

The equivalent circuit of a short dipole of length 1.2 meter at 29 MHz comprises series connected resistance and capacitance. The diameter of the poles is 2.5 cm. The values of radiation resistance and capacitance are calculated in Section 8.2.1 as  $2.7 \Omega$  and  $5.8 \text{ pF}$ , respectively. Hence radiation impedance becomes

$$Z_a = 2.7 - j950 \Omega$$

at 29 MHz, approximately. The reactive part must be tuned out at the frequency of transmission, in order to make most use of available power. The inductance needed for this purpose is about 5.2  $\mu\text{H}$ . Tuning out the capacitance is the first step. The radiation resistance must then be matched to the TX output impedance, which is 50  $\Omega$ . We also need a transformer. The resulting feeder circuit is depicted in Figure 8.11.

The series resonant circuit has a very high  $Q$  ( $\approx 350$ ). It is not possible to have a tuning inductance virtually without any loss at this frequency. The loss resistance of the inductor is in series with the radiation resistance and decreases the efficiency of radiation in this configuration. Another complication is the fact that, a very narrow 3-dB bandwidth of the antenna makes optimum power transfers over the entire 28-29.7 MHz range impossible without extra tuning.

If we consider the case when the dipole length is longer, for example 2.4 meters, the life becomes simpler. For such a dipole, radiation resistance and the capacitance are 10.5  $\Omega$  and 10 pF approximately. The radiation resistance is now significantly higher and, consequently, the series loss resistance of the tuning inductor is less effective. Also the  $Q$  of the antenna is lower, at about 50, and hence, tuning is much simpler.

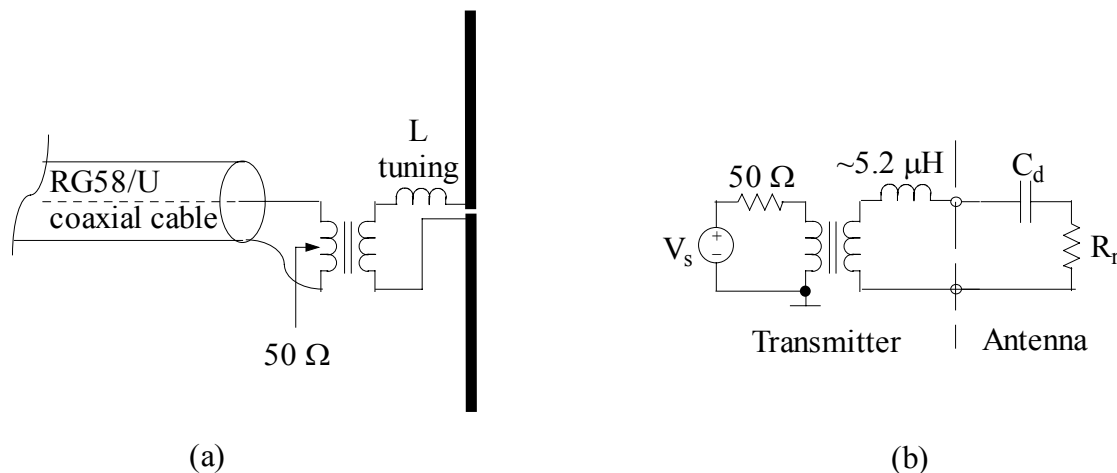


Figure 8.11 Short dipole feeder of TRC-10; (a) connections and (b) equivalent circuit

If we can maintain the length of the dipole at its  $\lambda/2$  resonance length, then the feeding becomes a simple balun problem, free from bandwidth and matching constraints.

We neither live in houses of five meter ceiling height, nor we work in such environment. Our gadgets, including antennas, have to live with our shortcomings. Hence the test antenna length is limited to 1.2 meters. On the other hand, as we experience in the exercises of this chapter, two wishful parties who want to communicate using a pair of TRC-10 can do so even with un-tuned feeders, *for testing purposes*. Mother nature is on our side!



## 8.6. Using amateur frequency bands

National regulation authorities tightly regulate transmission at any frequency. You are allowed to use the transmitters you built in amateur bands, provided your transmitter satisfies the emission requirements. The equipment that can be used in other bands must have “type approval”.

The operators using transmitters are also required to have amateur licenses. If you intend to use your TRC-10 for purposes other than the requirements of this course, you must contact your national regulatory agency and obtain an amateur license.

## 8.7. Bibliography

Electromagnetic theory is the founding subject in electronics. A good understanding of electromagnetic theory is essential for good electronic engineering. Antennas and propagation builds on electromagnetic theory and is a fundamental field in electronic engineering. There are many texts on these subjects with a lot of detailed analysis of almost every aspect.

As far as handling simple practical antenna problems with almost no electromagnetic theory background, *The ARRL Antenna Handbook* is a good source. Also Chapter 20 of *The ARRL Handbook* provides a concise overview of such problems.

## 8.8. Laboratory Exercises

1. Construct the dipole using two aluminum pipes (of 2.5 cm diameter and 60 cm length) and 10 cm long polyamide stud. Place the antenna handle in between as shown in Figure 8.12. and fix the structure using two screws.
2. The total antenna capacitance is somewhat higher than the dipole capacitance because of mutual impedance, as we discussed in Section 8.5.1. Hence tuning coil can be less than 5.2  $\mu\text{H}$ . Allowing few pF for mutual capacitance, a tuning coil of about 4  $\mu\text{H}$  must be sufficient.

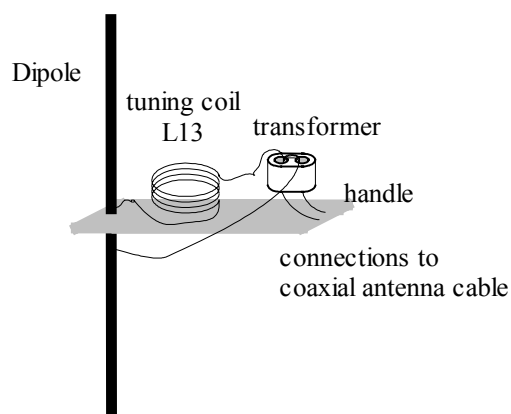


Figure 8.12 Antenna connections

The coil former for tuning inductor L13 is a piece of plastic tube of 16 mm diameter. Cut grooves on the coil former so that you can fix one end of the winding after tuning. Cut about a meter of 0.5 mm diameter enameled wire and wind 17 tight turns on this coil former so that the length of the coil is 1 cm. Fix the winding by means of a tape. Strip enamel on both ends and cover them by solder completely. Fix one end of the inductor to the screw on one of the poles, by making a loop around the screw. Tighten the screw. Fix this end of the coil on the coil former and fix the coil former on the antenna handle using glue. Work the other end into the groove such that the winding is tightly fitted on the cylindrical surface and its length is fixed.

Remove the tape. You can now increase the length of the coil by moving the end in the groove. This provides a means of tuning the antenna. This coil is your *antenna tuner*.

The loss resistance of the coil is in series with the radiation resistance and comparatively larger than the radiation resistance. The best way to find out the total impedance is to measure it. However it is not a straightforward laboratory procedure to measure an antenna radiation resistance at 30 MHz. Instead we shall estimate the loss resistance, assuming that the Q of the coil is about 70.

We can assume the coil loss resistance  $r_s$  as approximately  $10 \Omega$ .

3. The transformer core BLN1728-6 has  $18 \text{ nH/turn}^2$  inductance constant, when the winding is applied from one hole to the other. This shape of core enables low leakage transformers even with low permeability materials, which can be used at higher frequencies. Wind 10:5 winding on BLN1728-6. Strip enamel on all four ends and cover them by solder completely. Solder one lead of the 5-turn secondary to the free end of the coil. Connect other lead to the remaining pole. Again make a loop around the screw.

The approximate equivalent circuit of the antenna is given in Figure 8.13. The magnetizing inductance of the transformer is referred to secondary in this circuit. The impedance  $Z_s$  has two resonance frequencies. The series resonance is near

$$\omega_s = (L_{13}C_d)^{-1/2}.$$

The parallel resonance is in the vicinity of

$$\omega_p = [(L_{13}+L_s)C_d]^{-1/2},$$

and is lower than the series resonance. We use the series resonance of this circuit.

Solder the free end of the coaxial cable (the other end is terminated by a BNC) across the primary terminals of the transformer.

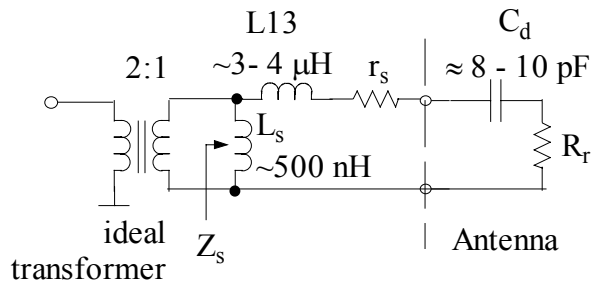


Figure 8.13 Equivalent circuit of the antenna and the feeder.

4. Connect your antenna to your TRC-10 using RG58/U coaxial cable and BNC connector. Switch the power on. Try to tune to the frequency of the test transceiver, which transmits a continuous tone signal. You can extend the length of the tuning inductor to tune your antenna precisely at this reception frequency. Check if you can improve the reception (for example, does the received continuous tone sound increase?) by tuning the antenna.
5. Switch TRC-10 to transmit mode and check if the test receiver receives your signal.
6. Make the microphone connection to the circuit by connecting J12. Find a partner and try to meet at a frequency and communicate. You can use calibration table you produced for coarse tuner in Chapter 7. You may have to re-tune the tuning inductor for a better signal.

### 8.9. Problems

1. Assume we want to use a dipole antenna, which has a radiation resistance of  $10 \Omega$  with TRC-10. Calculate the total length of the antenna. Calculate the antenna capacitance  $C_d$  if the diameter of the poles is 1 cm. Calculate the inductor required to tune this capacitor at 29 MHz. What is the Q of this antenna?
2. Calculate the power delivered to  $R_r$  in Figure 8.13 at the series resonance. Assume that the transmitter shown in Figure 8.11(b) drives the transformer primary and the peak amplitude  $V_s$  of the source is 4.5 V.
3. For the transmitted signal in problem 2, calculate the distance at which the received open circuit signal is  $100 \mu\text{V}$  peak, assuming that the receiver is also a TRC-10 with a similar antenna and feeder.
4. Assume that we do not use L13 at all, but tune  $C_d$  by an appropriately chosen  $L_s$ . Calculate the value of  $L_s$  required to tune out  $C_d$  of TRC-10 antenna, in a parallel resonance. Calculate the power delivered to  $R_r$  in Figure 8.13 at the parallel resonance. Assume that the transmitter as shown in Figure 8.11(b) drives the transformer primary and the peak amplitude  $V_s$  of the source is 4.5V.

5. A short dipole of length 20 cm is used as the antenna of a mobile set at 150 MHz. What is the open circuit voltage generated at a receiving antenna, when another set transmits 2 W at 3 km distance?

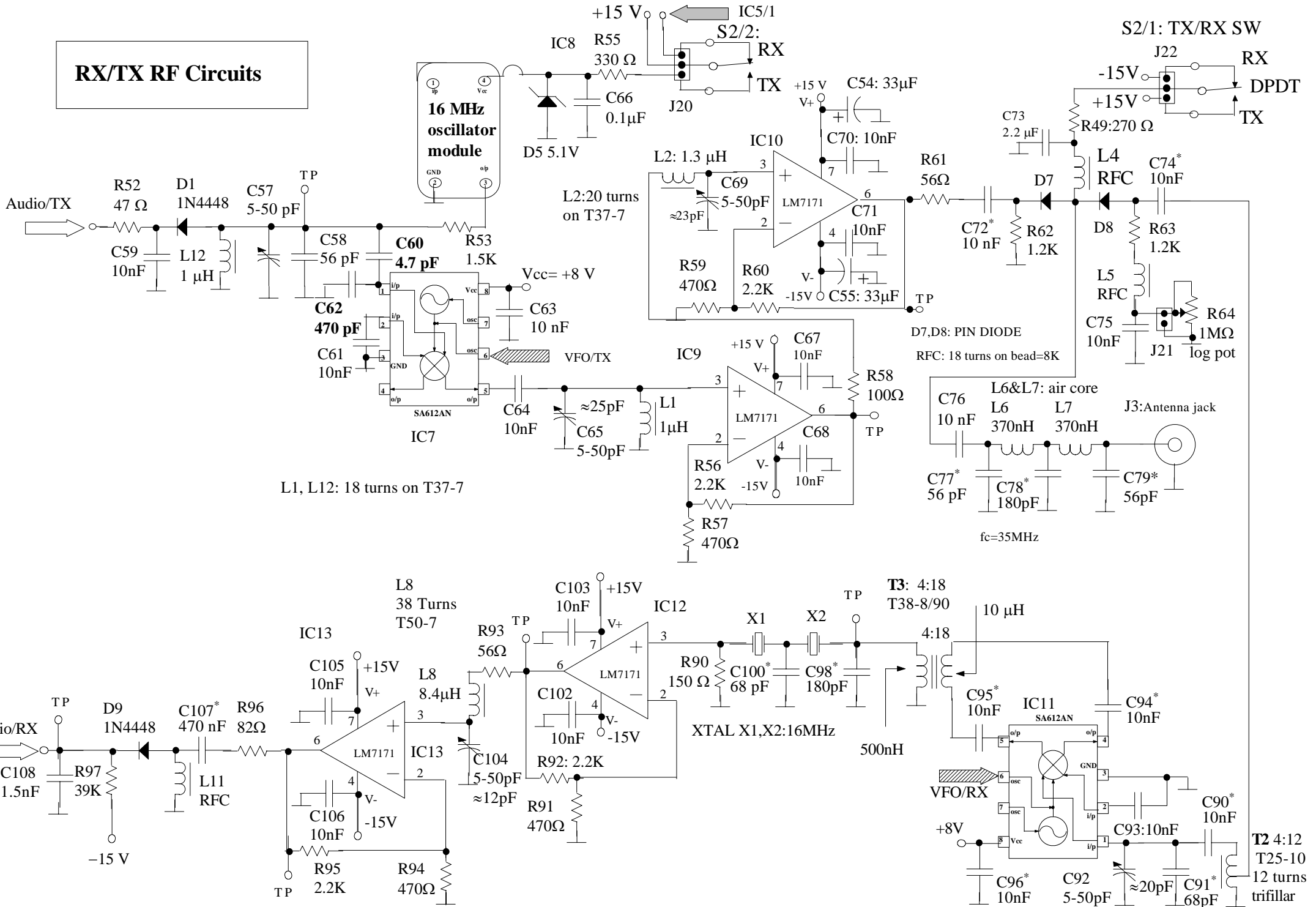


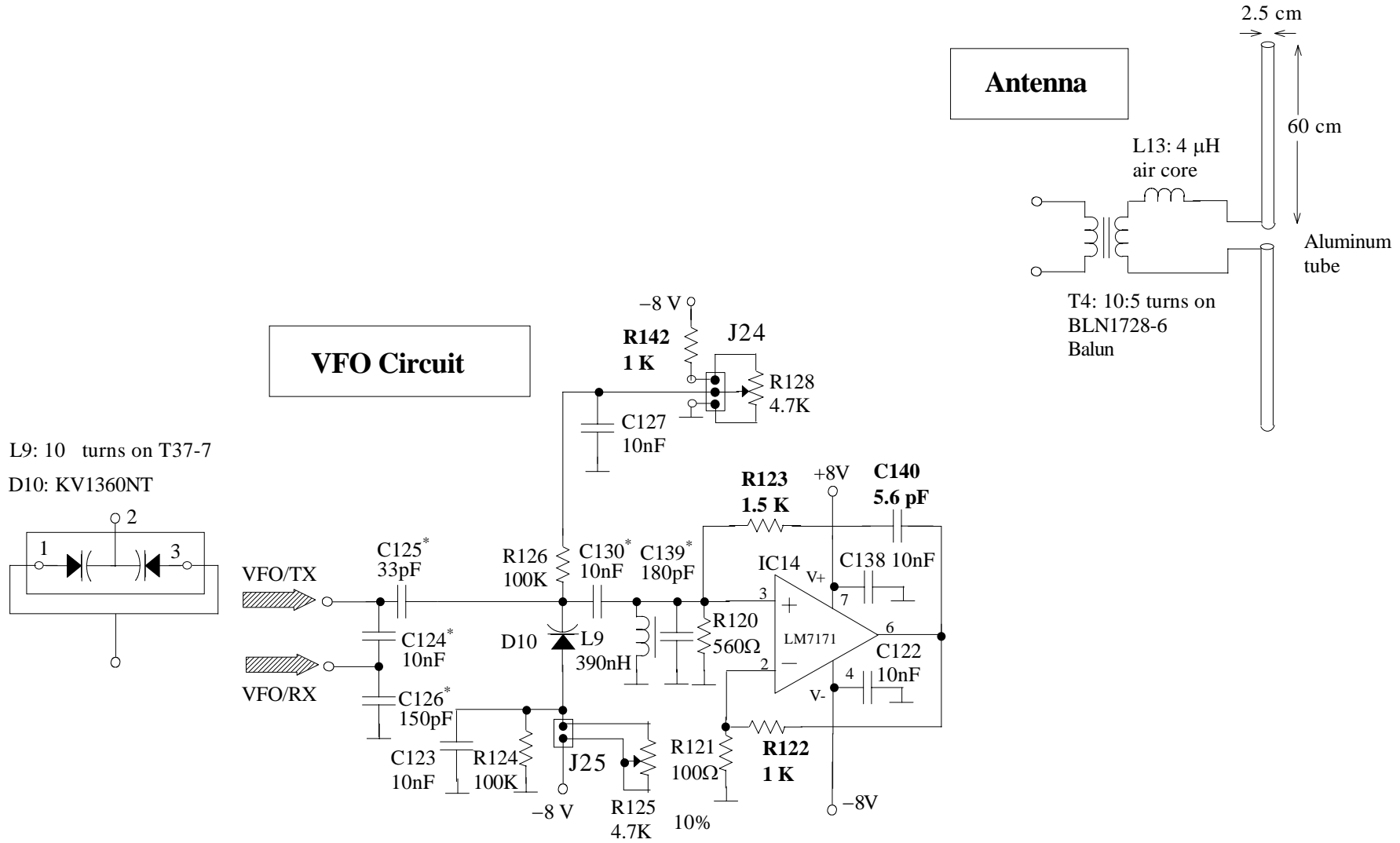
# Appendix A

## Circuit Diagram and PCB Layout

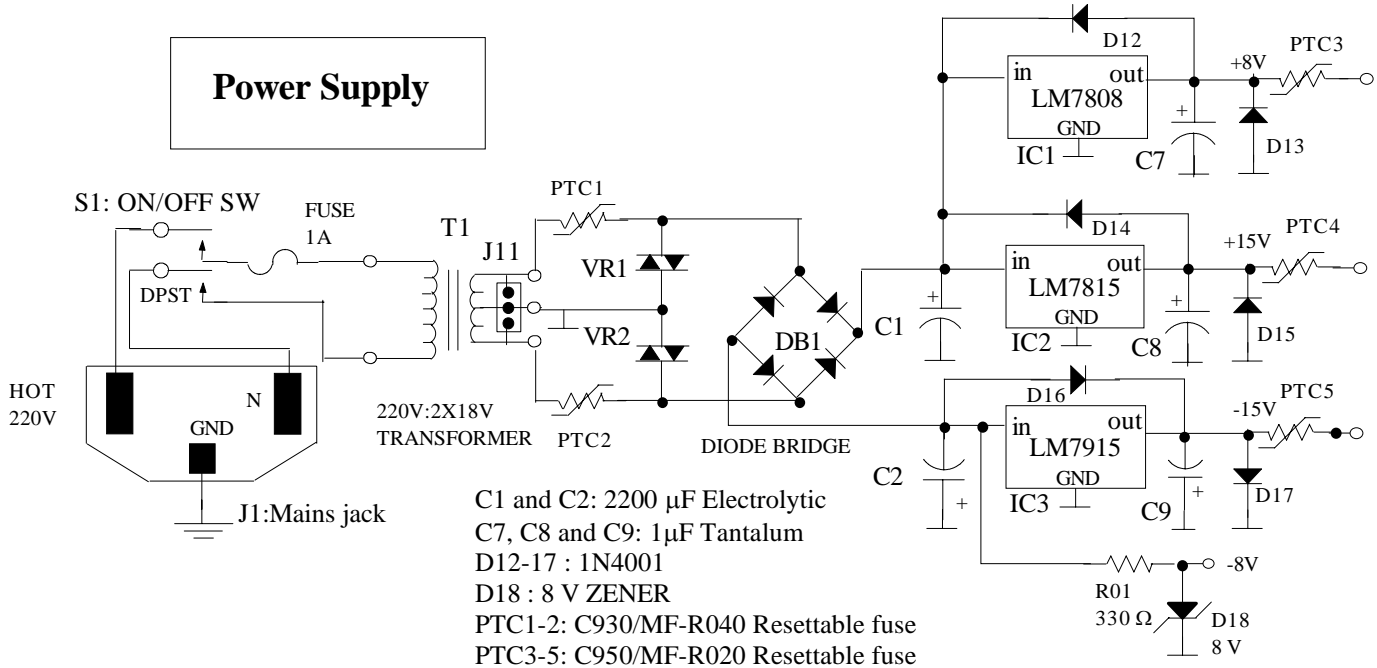
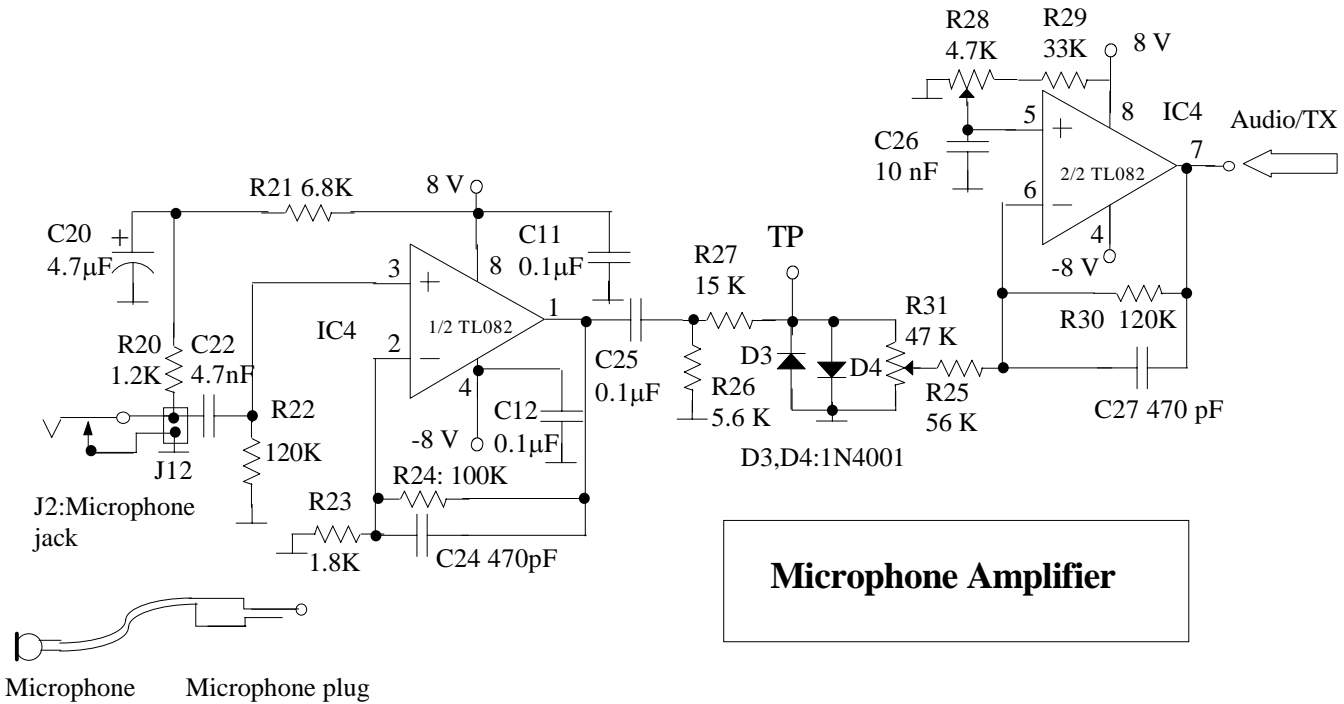
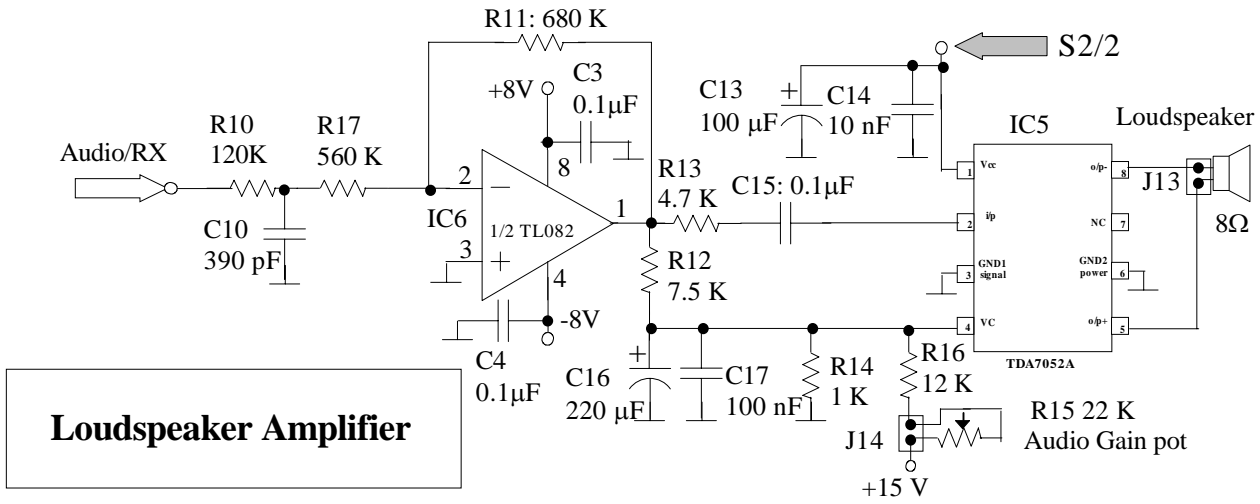
1. RX/TX circuits
2. VFO
3. Antenna
4. Loudspeaker amplifier
5. Microphone amplifier
6. Power supply
7. PCB layout
8. TRC-10

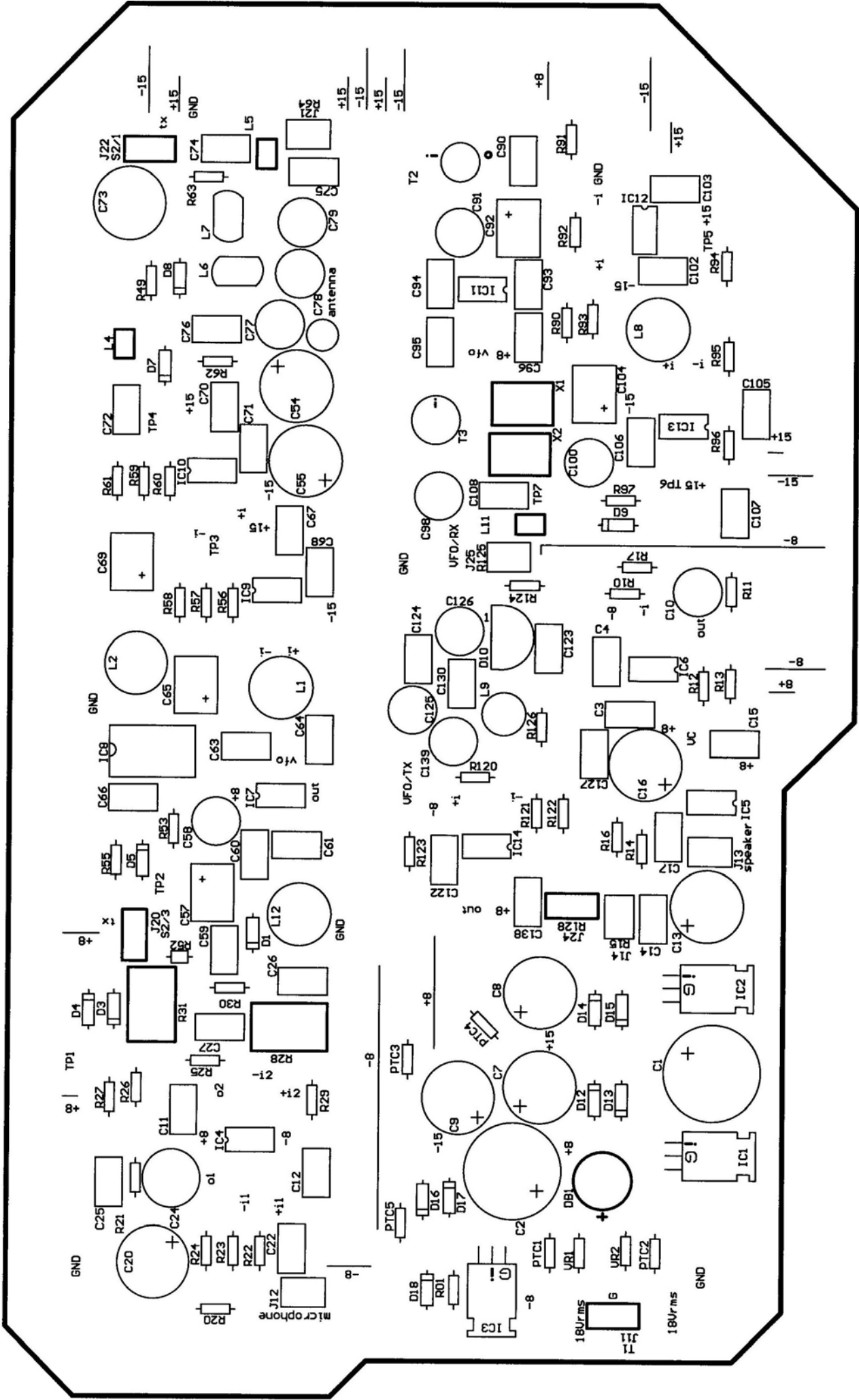
# RX/TX RF Circuits





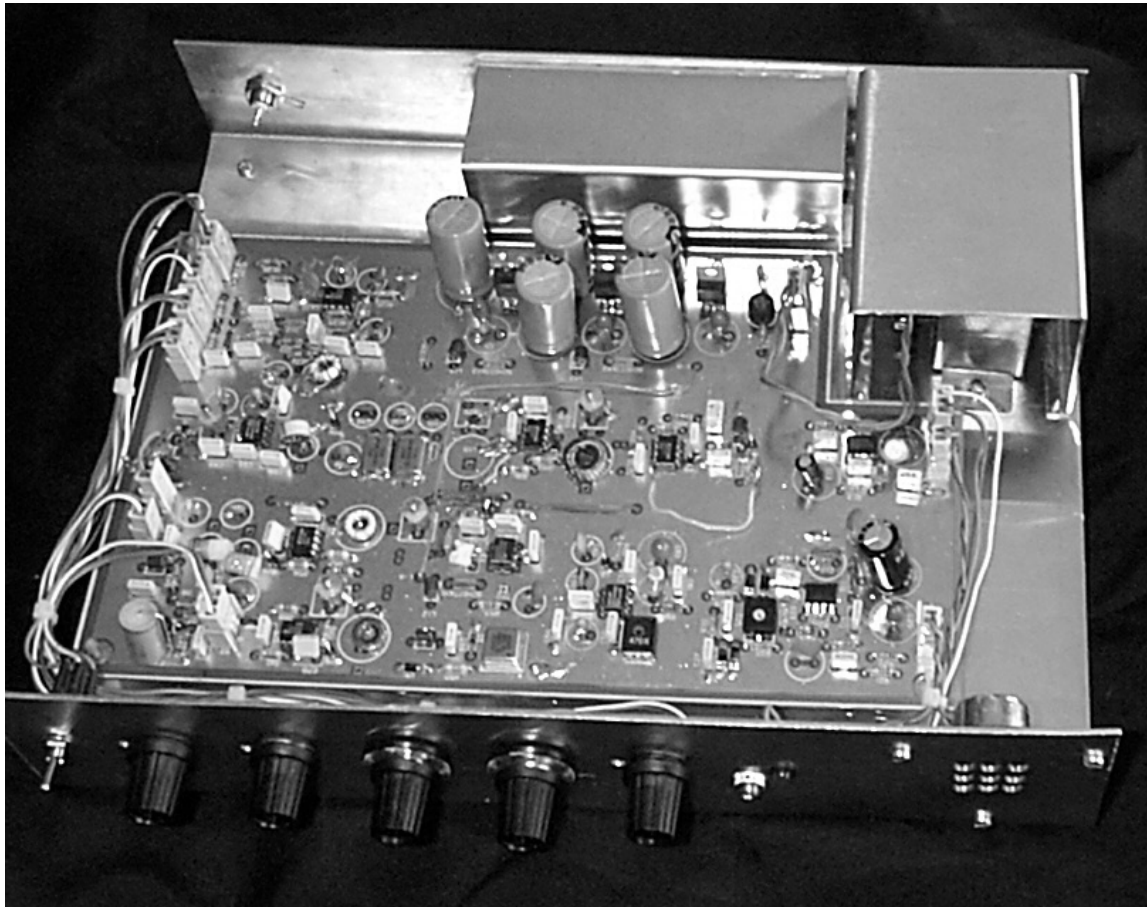




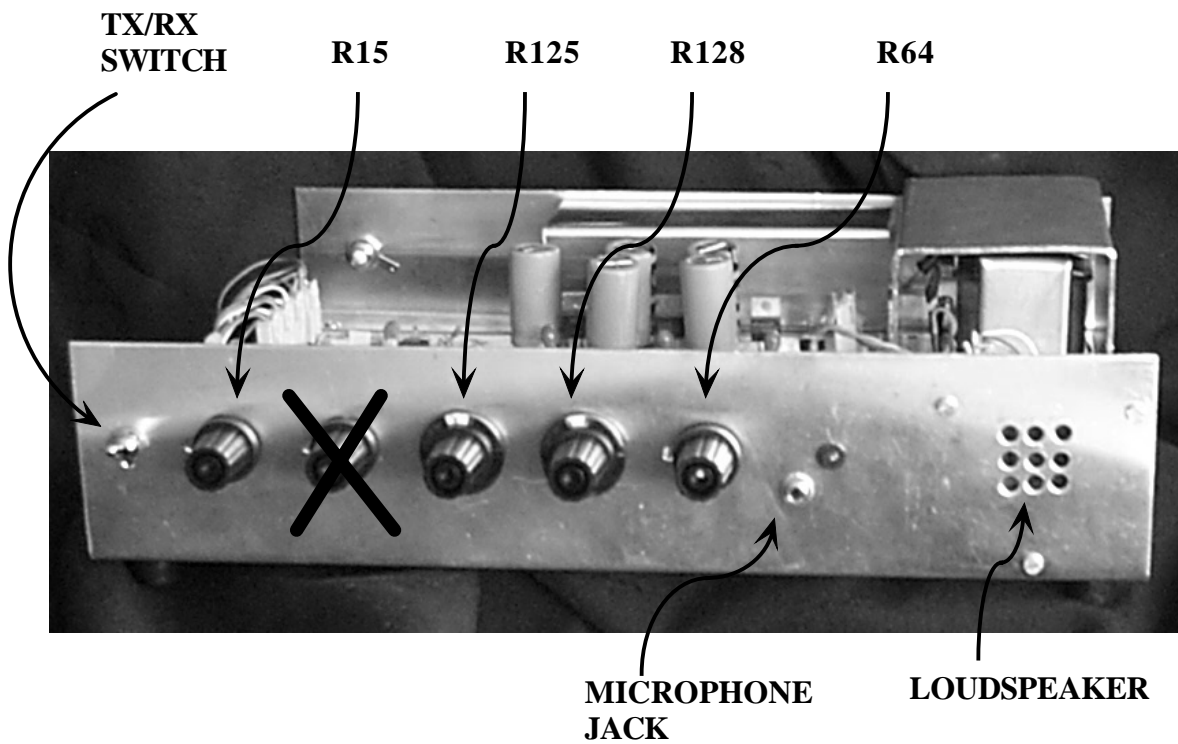


# TRC-10

## TOP VIEW



## FRONT VIEW



## **Appendix B**

### List of Components

**TRC-10 LIST OF COMPONENTS****RESISTORS**

R01	330	$\Omega$		
R10	120	K		
R11	680	K		
R12	7.5	K		
R13	4.7	K		
R14	1	K		
R15	22	K	PANEL LIN POT	
R16	5.6	K		
R17	560	K		
R20	1.2	K		
R21	6.8	K		
R22	120	K		
R23	1.8	K		
R24	100	K		
R25	56	K		
R26	5.6	K		
R27	15	K		
R28	4.7	K	TRIMPOT	
R29	33	K		
R30	120	K		
R31	47	K	TRIMPOT	
R49	270	$\Omega$		
R52	47	$\Omega$		
R53	1.5	K		
R55	150	$\Omega$		
R56	2.2	K		
R57	470	$\Omega$		
R58	100	$\Omega$		
R59	470	$\Omega$		
R60	2.2	K		
R61	56	$\Omega$		
R62	1.2	K		
R63	1.2	K		
R64	1	M	PANEL LOG POT	KNOB
R90	150	$\Omega$		
R91	470	$\Omega$		
R92	2.2	K		
R93	56	$\Omega$		
R94	470	$\Omega$		
R95	2.2	K		
R96	82	$\Omega$		
R97	39	K		
R120	560	$\Omega$		
R121	100	$\Omega$		
R122	1	K		
R123	1.5	K		
R124	100	K		
R125	4.7	K	PANEL LIN POT KNOB, SKIRT	
R126	100	K		
R128	4.7	K	PANEL LIN POT KNOB, SKIRT	
R142	1	K		

**CAPACITORS**

C01	2200	μF	ELECTROLYTIC (35V)
C02	2200	μF	ELECTROLYTIC (35V)
C03	0.1	μF	
C04	0.1	μF	
C07	1	μF	TANTALUM
C08	1	μF	TANTALUM
C09	1	μF	TANTALUM
C10	390	pF	
C11	0.1	μF	
C12	0.1	μF	
C13	100	μF	ELECTROLYTIC (35V)
C14	10	nF	
C15	0.1	μF	
C16	220	μF	ELECTROLYTIC (35V)
C17	0.1	μF	
C20	4.7	μF	TANTALUM
C22	4.7	nF	
C24	470	pF	
C25	0.1	μF	
C26	10	nF	
C27	470	pF	
C54	33	μF	TANTALUM (35V)
C55	33	μF	TANTALUM (35V)
C57	5-50	pF	VARIABLE CAP
C58	56	pF	
C59	10	nF	
C60	4.7	pF	
C61	10	nF	
C62	470	pF	
C63	10	nF	
C64	10	nF	
C65	5-50	pF	VARIABLE CAP
C66	0.1	μF	
C67	10	nF	
C68	10	nF	
C69	5-50	pF	VARIABLE CAP
C70	10	nF	
C71	10	nF	
C72	10	nF	
C73	2.2	μF	
C74	10	nF	
C75	10	nF	
C76	10	nF	
C77	56	pF	
C78	180	pF	
C79	56	pF	
C90	10	nF	
C91	68	pF	
C92	5-50	pF	VARIABLE CAP
C93	10	nF	
C94	10	nF	
C95	10	nF	
C96	10	nF	

C98	180	pF	
C100	68	pF	
C102	10	nF	
C103	10	nF	
C104	5-50	pF	VARIABLE CAP
C105	10	nF	
C106	10	nF	
C107	0.47	μF	
C108	1.5	nF	
C122	10	nF	
C123	10	nF	
C124	10	nF	
C125	33	pF	
C126	150	pF	
C127	10	nF	
C130	10	nF	
C138	10	nF	
C139	180	pF	
C140	5.6	pF	

**INDUCTORS**

L1	1	μH	T37-7	18T
L2	1.3	μH	T37-7	20T
L4	RFC		BEAD	18T
L5	RFC		BEAD	18T
L6	370	nH	AIR	7T
L7	370	nH	AIR	7T
L8	8.4	μH	T50-7	38T
L9	390	nH	T37-7	10T
L11	RFC		BEAD	18T
L12	1	μH	T37-7	18T
L13	4	μH	AIR	17T
T2		04:12	T25-10	
T3		04:18	T38-8/90	
T4	BALUN	10:05	BLN1728-6	

**CRYSTALS**

X1	9GL	16 MHZ	MEC
X2	9GL	16 MHZ	MEC

**SEMICONDUCTORS**

DB1		DIODE BR
D1	1N4448	
D3	1N4001	
D4	1N4001	
D5	ZENER	5.1V
D7	MPN3404	PIN
D8	MPN3404	PIN
D9	1N4448	
D10	KV1360NT	VARICAP
D12	1N4001	
D13	1N4001	
D14	1N4001	

D15	1N4001	
D16	1N4001	
D17	1N4001	
D18	ZENER	8V
PTC1	C930/MF-R040	
PTC2	C930/MF-R040	
PTC3	C950/MF-R020	
PTC4	C950/MF-R020	
PTC5	C950/MF-R020	
VR1	30V varistor	
VR1	30V varistor	

**INTEGRATED CIRCUITS**

IC1	LM7808	
IC2	LM7815	
IC3	LM7915	
IC4	TL082	
IC5	TDA7052A	
IC6	TL082	
IC7	SA612AN	
IC8	CO12100-16	VF150
IC9	LM7171	
IC10	LM7171	
IC11	SA612AN	
IC12	LM7171	
IC13	LM7171	
IC14	LM7171	

**POWER**

T1	MAINS TR.	2X220:18	15W
S1	DPDT	2X3 CONT	ON/OFF SW
FUSE	0.2	A	DELAYED
FUSE HOLDER			
J1		MAINS JACK	
POWER CABLE			

**AUDIO**

MICROPHONE		ELECTRET
SPEAKER	8Ω	
J2		MIC JACK
MIC PLUG		
MIC CABLE		

**MECHANICAL**

PCB		
TRAY		
CABLE TIE	10 EACH	
BOX		
POT KNOB		PUSH ON
HEADER	1x12	MALE
PCB JACK PAIR	2 PINS	4 EACH
PCB JACK PAIR	3 PINS	7 EACH
pcb JACK PINS		27 EACH
STUD	4 EACH	



S2	DPDT MOMENTARY	ON-ON;100 DP 2T1	2X3 contact
POT CAP	RED		
POT CAP	BLUE		
POT CAP	YELLOW		
POT KNOB	PUSH ON		
POT KNOB	PUSH ON		
POT KNOB	PUSH ON		
POT KNOB	PUSH ON		
KNOB SKIRT			
KNOB SKIRT			
<b>ANTENNA</b>			
J3	BNC		ANT JACK
ANT CABLE	2M		
BNC T			
RG58+2BNC	1.2M		
RG58+BNC	ANT PLUG		
2x60CM AL PIPE			
10 CM POLYAMIDE			
HANDLE			
<b>TRC-10 TRAY</b>			
<b>TRC-10 PCB</b>			

# Appendix C

## Data sheets

1. Integrated circuits
  - 1.1 LM7800
  - 1.2 LM7900
  - 1.3 TL082
  - 1.4 TDA7052A
  - 1.5 LM7171
  - 1.6 SA602A
  - 1.7 CO12100 (16 MHz clock module)
  - 1.8 VF150 (16 MHz clock module)
2. Diodes
  - 2.1 1N4001
  - 2.2 1N4448
  - 2.3 KV1360NT
  - 2.4 MPN3404
  - 2.5 BA479
3. Cores
  - 3.1 Ferrite cores
    - 3.1.1. RFC core
  - 3.2 Iron-powder cores
    - 3.2.1. T37-7
    - 3.2.2. T50-7
    - 3.2.3. T38-8/90
    - 3.2.4. T25-10
    - 3.2.5. BLN1728-6
4. Circuit protection devices
  - 4.1 PTC

# **Appendix D**

## Complex Numbers

## Complex Numbers and Complex Algebra

$x^2+1=0$  has no real roots. Solution of this equation is  $x = \pm\sqrt{-1}$ . To handle such situations we use complex number system.

A complex number has the form  $\mathbf{a+jb}$ , where  $j = \sqrt{-1}$  and  $\mathbf{a}$  and  $\mathbf{b}$  are real numbers.  $j$  is called the *imaginary unit* and has the property of  $j^2 = -1$ . If the complex number  $z = a+jb$ , then  $a$  is the *real part* and  $b$  is the *imaginary part* of  $z$ , and they are denoted as

$$a = \text{Re}\{z\} \text{ and}$$

$$b = \text{Im}\{z\}.$$

The symbol  $z$ , which can stand for any complex number is called *complex variable*.

Two complex numbers  $a+jb$  and  $c+jd$  are equal if and only if  $a = c$  and  $b = d$ . Real numbers are a subset of complex numbers, with imaginary part equal to zero. If on the other hand, the real part of a complex number is zero then it is called an imaginary number. For example  $2+j0$  and  $-11+j0$  are real numbers 2 and  $-11$  respectively, while  $0-j6$ , or  $-j6$  briefly, is an imaginary number.

The *complex conjugate* of a complex number  $z = a+jb$  is  $z^* = a-jb$ .

The algebra of complex numbers is the same as the algebra of real numbers with  $\mathbf{j^2}$  replaced by  $\mathbf{-1}$ :

Addition  $(a+jb)+(c+jd) = a+jb+c+jd = (a+c)+j(b+d)$

Subtraction  $(a+jb)-(c+jd) = a+jb-c-jd = (a-c)+j(b-d)$

Multiplication  $(a+jb)(c+jd) = ac+jda+jbc+j^2bd = (ac-bd)+j(bc+ad)$   $\{j^2 \text{ replaced by } -1\}$

Division

$$\frac{a + jb}{c + jd} = \frac{(a + jb)(c - jd)}{(c + jd)(c - jd)} = \frac{ac - jad + jbc - j^2bd}{c^2 - j^2d^2} = \frac{(ac + bd) + j(bc - ad)}{c^2 + d^2} = \frac{ac + bd}{c^2 + d^2} + j \frac{bc - ad}{c^2 + d^2}$$

similarly  $(c+jd)^{-1} = (c-jd)/(c^2+d^2)$ .

Absolute value of a complex number  $a+jb$  is  $|a+jb| = (a^2+b^2)^{1/2}$ . If  $z = a+jb$ , notice that  $zz^* = a^2+b^2$ . For example  $|3-j5| = (9+25)^{1/2} = \sqrt{34}$ .

If  $z_1, z_2, z_3, \dots z_n$ , are complex numbers,

$$|z_1 z_2 z_3 \dots z_n| = |z_1| |z_2| |z_3| \dots |z_n| ;$$

$$\left| \frac{z_1}{z_2} \right| = \frac{|z_1|}{|z_2|}, z_2 \neq 0.$$

**Eulers formula** :  $e^{j\theta} = \cos\theta + j\sin\theta$

## Different representations of complex numbers:

- |                       |                                   |  |
|-----------------------|-----------------------------------|--|
| 1. Rectangular form:  | $z = a+jb$                        | as above   |
| 2. Trigonometric form | $z = r(\cos\theta + j\sin\theta)$ | where $z = a+jb = r(\cos\theta + j\sin\theta)$ ,<br>thus $r = (a^2+b^2)^{1/2}$ and $\theta = \arctan(b/a)$ |
| 3. Exponential form   | $z = r e^{j\theta}$               | where $r$ and $\theta$ are as in 2   |
| 4. Polar form         | $z = r\angle\theta$               | where $r$ and $\theta$ are as in 2   |

Notice that if  $z_1 = r_1 e^{j\theta_1}$  and  $z_2 = r_2 e^{j\theta_2}$ , then

$$z_1 z_2 = (r_1 e^{j\theta_1})(r_2 e^{j\theta_2}) = r_1 r_2 e^{j(\theta_1+\theta_2)}, \text{ directly, or in polar form}$$

$$z_1 z_2 = r_1 r_2 \angle\theta_1+\theta_2. \text{ Similarly}$$

$$z_1/z_2 = (r_1/r_2)e^{j(\theta_1-\theta_2)}, \text{ and}$$

$$z_1/z_2 = (r_1/r_2)\angle\theta_1-\theta_2.$$

For example if  $z_1 = 2+j3$  and  $z_2 = -1-j3$ ,  $z_1 z_2 = (2+j3)(-1-j3) = 7-j9$ ; or first convert into polar:  $z_1 = 3.6\angle 56.3^\circ$  and  $z_2 = 3.16\angle 251.6^\circ$ , and then  $z_1 z_2 = 11.4\angle 307.9^\circ = 7-j9$ .  $z_1/z_2$  can be found as  $z_1/z_2 = 1.14\angle -195.3^\circ = -1.1+j0.3$ .

## Roots of a complex number:

Any complex number  $z = r e^{j\theta}$  can be written as  $z = r e^{j(\theta+2\pi n)}$ , since  $e^{j\theta} = e^{j(\theta+2\pi n)}$ , where  $n = \pm 1, \pm 2, \dots$ . Similarly  $z = r\angle\theta$  can be written as  $z = r\angle\theta+n360^\circ$ . Then

$$z^{1/k} = r^{1/k} e^{j(\theta+2\pi n)/k} \text{ or } z^{1/k} = r^{1/k} \angle(\theta+n360^\circ)/k.$$

Now  $k$  distinct roots can be obtained by assigning  $n = 0, 1, 2, \dots, k-1$ . For example first five distinct roots of 1 can be found as:

$$1 = 1\angle(0+n360^\circ)/5 = 1\angle n72^\circ \Rightarrow \begin{array}{ll} 1^{\text{st}} \text{ root} = \cos 0^\circ + j\sin 0^\circ = 1 & n=0 \\ 2^{\text{nd}} \text{ root} = \cos 72^\circ + j\sin 72^\circ = 0.309 + j0.951 & n=1 \\ 3^{\text{rd}} \text{ root} = \cos 144^\circ + j\sin 144^\circ = -0.809 + j0.588 & n=2 \\ 4^{\text{th}} \text{ root} = \cos 216^\circ + j\sin 216^\circ = -0.809 - j0.588 & n=3 \\ 5^{\text{th}} \text{ root} = \cos 288^\circ + j\sin 288^\circ = 0.309 - j0.951 & n=4 \end{array}$$

## Power of a complex number:

$$z^k = r^k e^{jk\theta} = r^k (\cos k\theta + jsink\theta).$$

## Logarithm of a complex number:

$\ln(z) = \ln(r e^{j\theta}) = \ln(r e^{j(\theta+2\pi n)}) = \ln(r) + j(\theta+2\pi n)$ . The principal value, when  $n = 0$ , is most frequently used.

# **Appendix E**

## **Answers**