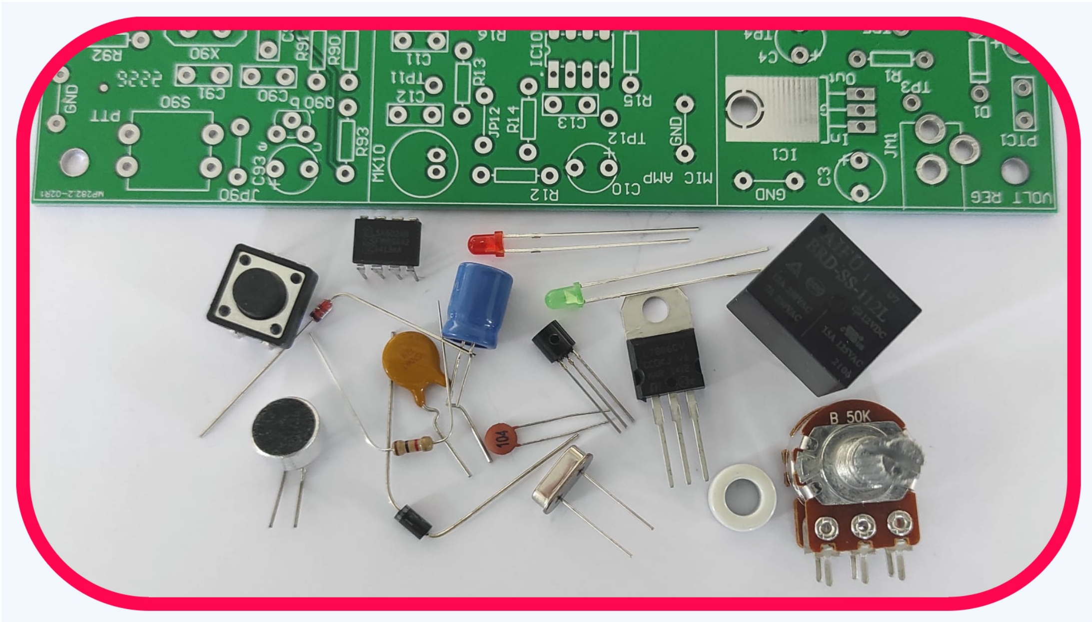
# ANALOG ELECTRONICS

**Fourth Edition (rev. 7.8)**




**Hayrettin Köymen**
**Abdullah Atalar**

**Bilkent University**

Typeset by LaTeX, published by Bilkent University (Cert. No: 51344) and printed by Meteksan A. Ş. (Cert. No: 46519) Ankara, Turkey.

First Edition, rev. 3.5 (May 2005)

Second Edition, rev. 4.1 (August 2014), rev. 4.2 (January 2015), rev. 4.3 (February 2017)

Third Edition, rev. 5.2.2 (September 2021)

Fourth Edition, rev. 7.8 (November 2023)

ISBN: 978-605-9788-47-2

# Contents

# Preface

Electronic Engineering is a wonderful profession. It is a synthesis of many diverse subjects such as applied mathematics, probability theory and physics. However, the most exciting part of Electronic Engineering is that it combines the knowledge in these well-established fields with the sense of achievement in designing and eventually constructing an actual instrument.

Electronic Engineering education is undertaking the challenge of teaching young individuals how to start from basic sciences and end up in finished, working, real-life, touchable instruments.

This book is one result of the curriculum renovation activities in the Electrical and Electronics Engineering Department of Bilkent University. Everybody involved in EE education knows a fundamental problem of introducing the "engineering" part relatively late during the education period. The new curriculum aims at introducing and developing engineering skills at an early stage.

As part of the new curriculum, a new introductory analog electronics course is introduced to the first semester of the second year. This course had to have a solid experimental emphasis. The course had to serve as the first circuit theory course as well. A great book by D.B. Rutledge, The Electronics of Radio, is an excellent response to the above considerations. The Electronics of Radio is a text for a course lasting two quarters. This book is an effort to provide similar material for a 14-week semester course.

RF electronics courses had always been senior year courses, if not graduate courses. It was always a problem to develop the lab skills of a student beyond audio frequencies. The students were not exposed to components, materials, etc., and information on materials available to the designer, timely, during their undergraduate education. Following observations were critical when the structure of the new course was determined:

1. Learning electronics, both theory and practice, demands the enthusiasm of 2nd-year students. Only after being exposed to electronics and practical circuit theory at the very beginning, students can choose their specialization accurately.

2. Electronic parts, which can work at HF are now available at a very low cost. This matter is particularly important in undergraduate courses from the "laboratory work cost" point of view.

The course is structured on a scenario of constructing an HF radio transceiver. Topics in analog electronics in the range of 100 Hz to 30 MHz are covered. The block diagram concept is introduced and used. Passive electronic components ($R$, $L$, $C$, diode, crystals, etc.), bipolar-junction-transistors and integrated circuits, as active devices, are discussed. Filters, power supplies, audio amplifiers, speakers, microphones, radio amplifiers, oscillators, mixers, intermodulation, and antennas are progressively introduced towards the construction of the transceiver. Minimum mathematical background and definitions (such as the solution of first-order differential equations and phasors) are introduced only when necessary. All terminology and jargon are introduced. A PCB of the transceiver is provided with the course kit and components.

Another aspect of the course is that every student must possess a soldering iron, de-soldering pump, a multimeter, a scientific calculator and a set of hand tools. Possessing such electronics-specific tools improves the ties and commitment to the discipline and the enthusiasm to learn electronics. Compared to EE education thirty years ago, the students suffer an identity problem today. Thirty years ago, every engineering student had to have a set of drafting tools and a slide rule right at the beginning of freshman. Computers replaced these today. Computers, however, are anonymous. Students of almost every discipline use computers. Computers are not specific to electronics students.

These considerations made this book different from more conventional electronics textbooks. The book tells the story of making a transceiver and introduces various concepts and other information only when necessary. In other words, the related topics in a subject are not, generally, collected together in the same section in this book. They are given at the relevant stages of the transceiver construction.

Acknowledgements:

Many people contributed both to design the course and to the course material. The author wishes to acknowledge the tremendous effort that Müjdat Balantekin had put into this work. The laboratory material for this course would have never been realized without the support provided by Ergün Hırlakoğlu, İsmail Kır and Ersin Başer. One hundred students of the class of 2001 provided invaluable feedback. The author acknowledges their contributions, effort and positive energy. Prof. A. Altıntaş, Prof. A. Atalar, Prof. B. Özgüler, Dr. T. Reyhan, Dr. S. Topçu, Dr. E. Tın, Prof. C. Yalabık, of Bilkent University and E. Ceyhan of ERE Corp. made many suggestions and critically reviewed the text. Finally, the author is indebted to N. Özönder of Telmek Corp. and B. Arıkan of Arıkan Elektronik, for an immaculate TRC-10 instrument tray and PCB.

Hayrettin Köymen
August 25, 2002
Ankara

## Preface to the Fourth Edition

The fourth edition of the book uses a substantially modified version of TRC-10 and it is now called TRC-11. It has a fixed frequency operation at 27.00 MHz corresponding to a wavelength of 11 m. Its receiver can operate, while the transmitter is turned on. This property enables the full testing of the transceiver without a need for a second transceiver. TRC-11 includes an automatic-gain-control circuit and LEDs are added as signal level and power indicators.

The first chapter is a summary of basic concepts of electronic communication. The chapter also has short descriptions of the basic building blocks of the transceiver.

The second chapter introduces most of the components existing in TRC-11: voltage and current sources, resistors, capacitors, and inductors. After the explanation of Kirchhoff's laws, the chapter presents time-domain solutions of first-order *RC* and *RL* circuits.

The third chapter is about frequency-domain solution of circuits excited by sinusoidal signals using the phasor notation. Transfer functions of linear circuits with any order are studied. Thévenin and Norton equivalent circuits are introduced, and the superposition principle is described. Since TRC-11 contains an operational amplifier, it is investigated in some detail. Many practical operational amplifier circuits are given.

The fourth chapter introduces two nonlinear devices: Diodes and bipolar junction transistors. Piecewise linear analysis methods of first-order circuits containing diodes are given. Terminal characteristics of bipolar transistors are introduced. Then, the biasing and small-signal analysis of transistors are given.

Being a transceiver, TRC-11 has also tuned circuits. Chapter five is about parallel and series *RLC* circuits. Limitations of real inductors are discussed and simple transformer equivalent circuit is given. The non-ideal behavior of real inductors is discussed in some detail.

Filters in a receiver play a crucial role. The filter design is introduced in chapter six. Since TRC-11 has a crystal filter, crystal filter design is also discussed.

Since TRC-11 uses amplitude modulation, a demodulation technique using diodes are explained in chapter seven. This chapter also describes automatic gain control system using a PIN diode.

TRC-11 uses the superheterodyne concept, which requires mixers and oscillators for frequency conversion. The eighth chapter has a brief discussion on the operation principles of a mixer as a building block. A few types of oscillators are discussed.

The ninth chapter is an introduction to antennas, radiation impedance and propagation. Basic antenna types like monopole and dipole antennas are described.

The final chapter is an introduction to the operation principles of semiconductor diodes and bipolar-junction-transistors. For this purpose, a simplified theory of semiconductor devices are given.

Several examples are added at the end of each chapter to ease understanding the concepts.

Experimental work associated with each chapter has a corresponding preliminary work, which should be completed before the students appear at the laboratory.

Appendices contain the schematics and datasheets of components used in TRC-11, troubleshooting hints, and answers to selected problems.

To simplify mounting and soldering, all components in TRC-11 are old-fashioned leaded components and integrated circuits have DIP packages.

We are thankful to Emine Sarıtaş, Veli Tayfun Kılıç, and Metin Kayabaşı who pointed out the errors in the second, third, and fourth editions.

Abdullah Atalar
July 2023
Ankara

# Chapter 1

# SIGNALS AND COMMUNICATIONS

## 1.1 Analog and Digital Electronics

In the modern world, electronics is everywhere. Especially, the mobile revolution increased the number of electronic devices and our dependence on them. A block diagram of a modern electronic device is composed of digital and analog parts as shown in Fig. 1.1. Since the real world is not digital, the interface between the world and electronic devices has to be through analog components. Analog sensors, like an electronic compass, an electronic thermometer or an acceleration sensor, are used as input devices. Analog transducers, like a microphone, an antenna, a buzzer, a loudspeaker, or a motor are used to convert energy from one form to another. Typically, input sensors and transducers are connected to analog circuitry, before their signals are converted to digital form by analog-to-digital (A/D) converters. After digital processing, the signals are converted to the analog world by digital-to-analog (D/A) converters. These signals are fed to analog circuitry for final connection to the analog world.

This text deals only with analog electronics. Analog electronics requires a good knowledge of physics, mathematics and circuit theory. Students of this course must have taken basic physics and mathematics courses. This text provides a beginner-level circuit theory. We believe that best learning occurs not by listening to a lecture or reading a text, but rather by *doing*, *experimenting* or *building* things. This textbook aims to provide a learning environment of analog electronics by guiding students to build a working communication device composed of analog parts only with no digital parts. Even though this is not the modern approach, we think it is very instructive for learning analog electronics.


Figure 1.1: A modern electronic device.

## 1.2 Electronic communications

Electronics are commonly used to send and receive signals, such as sound and vision, between parties. A transmitter converts signals into a form, so that they can be transmitted in the air as part of the electromagnetic spectrum. They are received by a receiver, where they are converted back to the original form. Two communicating parties can be quite far from each other, so the term *telecommunication* is used to describe this form of communication.

This book is structured around the building and testing a transceiver (inspired by a great book by Rutledge [1]), TRC-11, operating with a 11-meter wavelength in the amateur band (27 MHz). The name is generic: TRC stands for transceiver and 11 indicates that it works with the 11-meter wavelength. In the following, we describe the basic concepts and main blocks of a transmitter and receiver while keeping the mathematics as little as possible.

## 1.3 Voltage, Current and Frequency

The two variables in any electrical circuit are voltage, $v(t)$, and current, $i(t)$. Both of these variables can be time varying or constant. Voltages and currents that do not change with respect to time are called DC voltages and currents, respectively. The acronym DC 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

$$v(t) = V_1 \cos(\omega t + \theta_v) \tag{1.1}$$

$$i(t) = I_1 \cos(\omega t + \theta_i) \tag{1.2}$$

$V_1$ and $I_1$ are called the amplitudes or the peak values 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), named after German physicist Heinrich Rudolf Hertz (1857–1894). $\theta$ is the *phase* angle of the waveform, expressed in radians or degrees. These waveforms are periodic, which means that it is a repetition of a fundamental form every $T$ seconds. $T$ is called the *period* of the waveform with $T = 2\pi/\omega = 1/f$ seconds (sec).

Quite often, sinusoidal waveforms are referred to by their *peak* amplitudes or *peak-to-peak* amplitudes. The peak amplitude of $v(t) = V_1 \cos(\omega t)$ is $V_1$ volts peak (or $\mathrm{V}_p$) and peak-to-peak amplitude is $2V_1$ volts peak-to-peak (or $\mathrm{V}_{pp}$).

We can see that a DC voltage is in fact a sinusoid with $\omega$=0 rps. Sinusoidal voltages and currents with non-zero frequency are commonly referred to as AC voltages and currents. The acronym AC comes from *alternating current.*

## 1.4 Wavelength

Electromagnetic waves travel at the speed of light, $c = 3.0 \times 10^8$ m/sec in the air (or vacuum). This speed is the same as the speed of light in air, since light

| Light waves | Frequency | Wavelength |
|---|---|---|
| Infrared (IR) | 300 GHz–429 THz | 1 mm–0.7$\mu$m |
| Visible light | 429 THz–750 THz | 0.7$\mu$m–0.4 $\mu$m |
| Red light | 429 THz | 0.700 $\mu$m |
| Green light | 549 THz | 0.546 $\mu$m |
| Blue light | 688 THz | 0.436 $\mu$m |
| Ultraviolet (UV) | 750 THz–30 PHz | 0.4 $\mu$m–0.3 nm |

Table 1.1: Frequencies and wavelengths of light waves. Refer to page 310 for the definition of unit prefixes.

is also an electromagnetic wave at a much higher frequency. The wavelength of a wave can be written in terms of its speed, $c$, and its frequency, $f$, as

$$\boxed{\lambda = \frac{c}{f}} \tag{1.3}$$

where $\lambda$ is the wavelength in meters. Table 1.1 shows the frequency and wavelength of light waves.

Radio waves used in electronic communication are also electromagnetic waves. Table 1.2 lists some commonly used frequencies and the corresponding wavelengths.

| Radio waves | Frequency | Wavelength |
|---|---|---|
| AM radio band | 540 kHz–1630 kHz | 556 m–184m |
| Short-wave radio band | 5.9 MHz–26.1 MHz | 50.8 m–11.5 m |
| Toy radio control | 27 MHz | 11.1 m |
| TRC-11 | 27 MHz | 11.1 m |
| TV Channels 2–6 | 54 MHz–88 MHz | 5.56 m–3.41 m |
| FM radio band | 88 MHz–108 MHz | 3.41 m–2.78 m |
| TV channels 7–13 | 174 MHz–216 MHz | 1.72 m–1.39 m |
| TV channels 14–70 | 470 MHz–806 MHz | 64 cm–37 cm |
| Cellular phone (GSM-900) | 880 MHz–960 MHz | 34 cm–31 cm |
| GPS | 1575 MHz | 19.0 cm |
| Cellular phone (GSM-1800) | 1710 MHz–1880 MHz | 17.5 cm–16.0 cm |
| Cordless phone (DECT) | 1880 MHz-1900 MHz | 16.0 cm–15.8 cm |
| Wi-Fi/Bluetooth | 2.402 GHz–2.483 GHz | 12.49 cm–12.08 cm |
| Microwave oven | 2.45 GHz | 12.2 cm |
| Satellite TV receiver | 10.7 GHz–12.75 GHz | 2.80 cm–2.35 cm |
| Traffic radar | 24 GHz | 1.25 cm |

Table 1.2: Frequencies and wavelengths of radio waves.

Fig. 1.2 is a diagram showing the *frequency spectrum* with labels given to different bands and the corresponding wavelengths.

The formula of Eq. 1.3 also applies to sound waves, which travel in a medium. Sound waves propagate at a speed of 340 m/s in air and 1500 m/s in water. Therefore, a sound wave at a frequency of 1.0 kHz has a wavelength of 34 cm in air and 1.5 m in water.

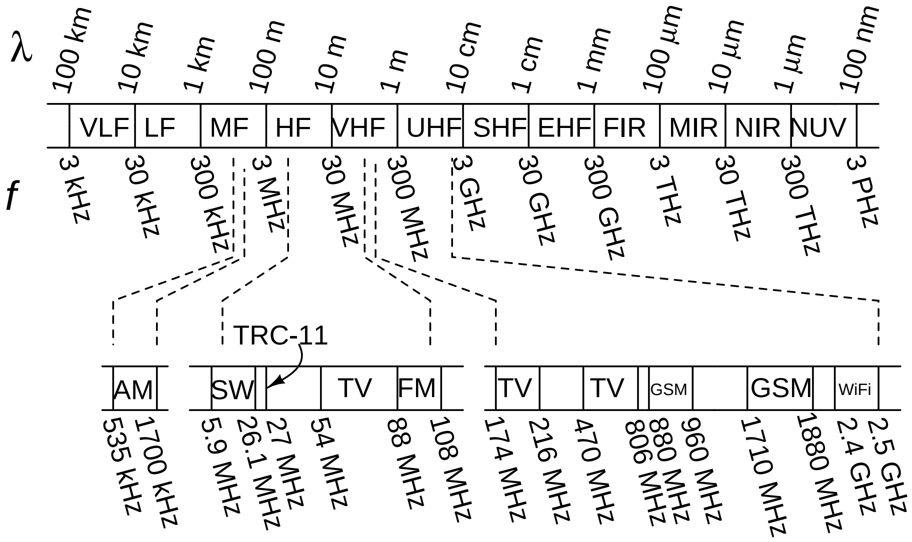



Figure 1.2: The frequency spectrum and its bands: VLF (very low frequency), LF (low frequency), MF (medium frequency), HF (high frequency), VHF (very high frequency), UHF (ultra high frequency), SHF (super high frequency), EHF (extremely high frequency), FIR (far infrared), MIR (mid infrared), NIR (near infrared). Some bands reserved broadcast radio transmission (AM, SW and FM), TV transmission, cellular phone (GSM) and local-area wireless (Wi-Fi) are also shown.

♦ TRC-11 uses electromagnetic waves in the HF band at the frequency of 27.00 MHz with a wavelength of 11.1 m. This is in a frequency range reserved for amateur radio.

## 1.5 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 the most popular. Square wave oscillators are predominantly used in digital circuits to produce time references, synchronization, etc. For example, an electronic wristwatch has an oscillator at a frequency of 32768 Hz, which is easily divided to $2^{15}$ using a 15-stage divide-by-two circuit to generate 1.0000 pulses per second*. *Function generators* are capable of producing a number of periodic signals like sinusoidal, square, triangular and sawtooth waveforms, with frequencies and amplitudes adjustable by the front panel buttons (see for example Fig. 1.3).

We use sinusoidal oscillators in communication circuits for various reasons. In most cases, the oscillators determine the frequency of operation.

*1.0000 expresses the precision of the quantity of one pulse per second. Refer to Appendix B for significant figure notation.

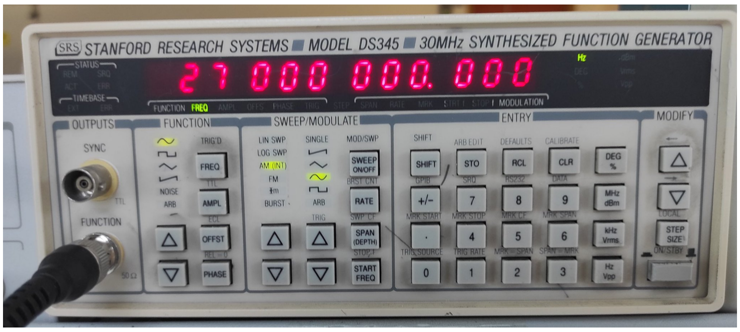



Figure 1.3: SRS DS345 function generator.

A square wave of 2 V peak to peak amplitude with a 1 V offset is depicted in Fig. 1.4. Such a square wave can be represented in terms of sinusoids as a linear combination:

$$s(t) = a_0 + \sum_{n=1}^{\infty} b_n \sin(n\omega t) = 1 + \frac{4}{\pi}\sin(\omega t) + \frac{4}{3\pi}\sin(3\omega t) + \frac{4}{5\pi}\sin(5\omega t)$$
$$+\frac{4}{7\pi}\sin(7\omega t) + \frac{4}{9\pi}\sin(9\omega t) + \dots \quad (1.4)$$

where $a_0$ is the average (or DC) value of $s(t)$, and $b_n$'s are the magnitudes of the harmonics. In this particular case, we have $a_0$=1 and $b_n = (2/n)[1-(-1)^n]$. Note here that

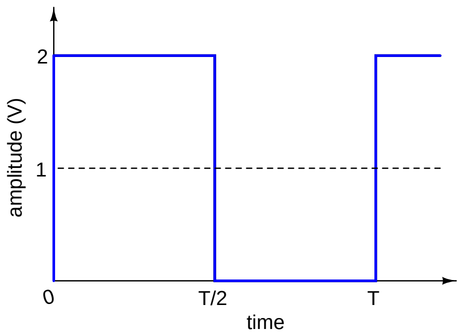



Figure 1.4: A square wave signal

- There are an infinite number of sinusoids in a square wave;
- The frequencies of these sinusoids are only odd multiples of $\omega$, which is a property of square waves with an equal duration of 2's and 0's— we call such square waves as 50% duty cycle square waves;
- The amplitude of sinusoids in the summation decreases as their frequency increases. We refer to the sinusoids with frequencies $2\omega$, $3\omega$, $4\omega$, ..., $n\omega$ as *harmonics* of the fundamental component, $\sin \omega t$.

We can obtain an approximation to a square wave by taking $a_0$, the fundamental, and only a few harmonics into the summation. As we increase the number of harmonics in the summation, the constructed waveform becomes a better representative of the square wave. This successive construction of a square wave is shown in Fig. 1.5: Even with only three terms the square wave is

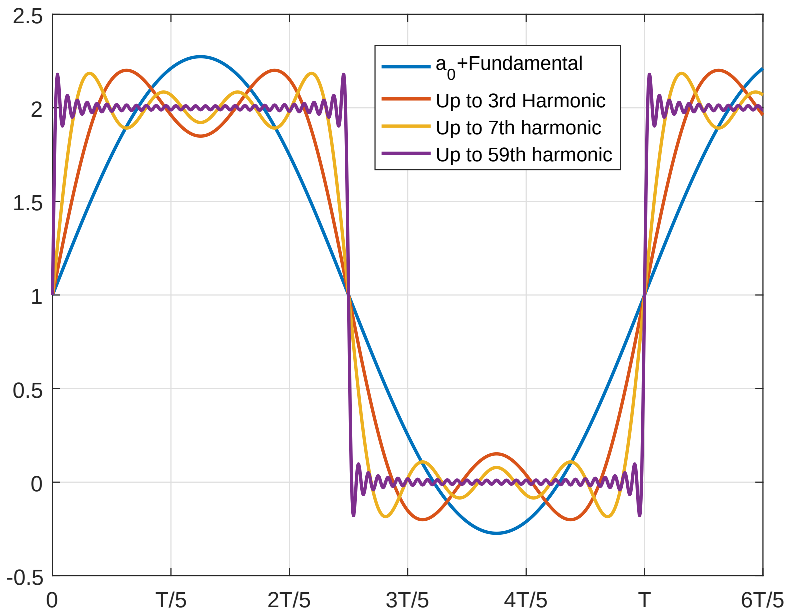



Figure 1.5: Constructing a square wave from harmonics, (a) only $a_0$+ fundamental, (b) all terms up to 3rd harmonic, (c) all terms up to 7th harmonic, (d) all terms up to 59th harmonic.

reasonably well delineated. It looks more like a square wave as the number of added harmonics increase.

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. The spectrum of this square wave is given in Fig. 1.6, which clearly illustrates the frequency components of the square wave. The figure shows that the square wave, being a periodic signal, has energy only at discrete frequencies, more specifically only at the odd harmonics of the fundamental.

- ⧫ TRC-11 has two oscillators, one at the frequency of 12.00 MHz, and the other at 15.00 MHz.

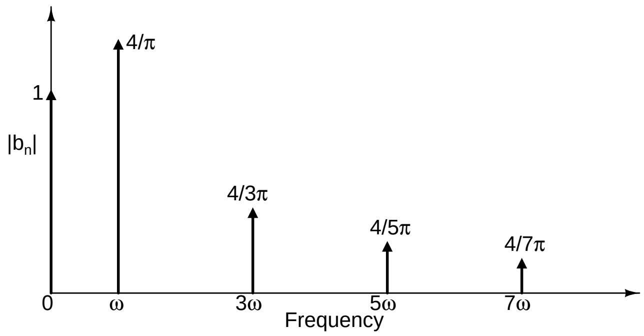



Figure 1.6: The spectrum of the square wave of Fig. 1.4

## 1.6 Modulation

We frequently use electromagnetic waves to transmit information from one place to another. The information, for example, voice or music, must first be converted into an electrical voltage, $v_m(t)$. We, then, convert the electrical signal to an electromagnetic wave to transmit it over some distance to the receiver. The conversion of $v_m(t)$ to an electromagnetic signal occurs by using an antenna. We will see in Chapter 8 that the size of the antenna should be comparable to the wavelength of the signal for efficient conversion.

The wavelength of an electrical voice or music signal is measured in hundreds of kilometers. Using an antenna of that size is obviously not practical. To make the antenna size small, we need to use a much higher frequency sinusoid (called the *carrier*) with a much smaller wavelength to carry the information. In order to transmit voice or music, we need to make one parameter of this carrier sinusoid dependent on the information. Merging the information-carrying signal on a high frequency carrier sinusoid is called *modulation*.

There are three parameters that we can modify in a sinusoid: amplitude, frequency and phase. In the *amplitude modulation* (AM) method, the amplitude of a sinusoid is made dependent on $v_m(t)$. 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,

$$v(t) = V_c \cos(\omega_c t) + v_m(t) \cos(\omega_c t) = V_c \left(1 + \frac{V_m}{V_c} \cos(\omega_m t)\right) \cos(\omega_c t) \quad (1.5)$$

$v_m(t)$ is called the *modulating signal*. In AM, the maximum peak variation of $|v_m(t)|$ must always be less than $V_c$, otherwise some parts of $v_m(t)$ get lost. $V_c[1+(V_m/V_c)\cos(\omega_m t)]$ part in AM signal is called the *envelope*. An AM signal is depicted in Fig. 1.7(b).

The depth of modulation is determined by the maximum value of the normalized modulation signal $|v_m(t)/V_c|$. The *modulation index*, $m$, is defined as

$$m = \left|\frac{V_m}{V_c}\right|_{max} \quad (1.6)$$

If $m = 1$, AM signal is said to have 100% modulation.

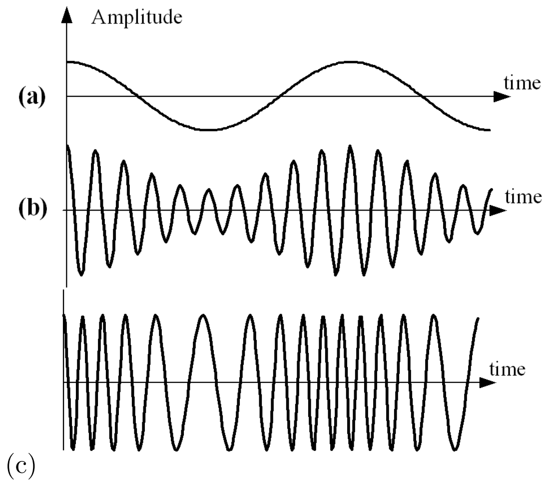



Figure 1.7: (a) Modulating signal, $v_m$, (b) AM modulated signal, (c) FM modulated signal.

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

$$v(t) = V_c \cos\left(\omega_c t + k_f \int v_m(t)dt\right) = V_c \cos\left(\omega_c t + \beta V_m \sin(\omega_m t)\right), \qquad (1.7)$$

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 instantaneous frequency of the carrier signal around the carrier frequency, $\omega_c$, according to the variation of modulating (information) signal, while the envelope of the signal stays constant. An FM modulated signal is shown in Fig. 1.7(c).

Long wave and middle wave radio broadcasting are done by AM, and radio broadcasting in the 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.

♦ TRC-11 employs amplitude modulation with a carrier frequency of 27.000 MHz.

## 1.7 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 a few tens of fW (1 femtoWatt=$1\times10^{-15}$W). This power level corresponds to a few $\mu$V (microvolt, $\mu$=$10^{-6}$) into a 50 $\Omega$ resistance, 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 an *amplifier*.

Amplifiers relate the signal at their input and their output by a gain. We are usually interested in two types of gain, voltage gain and power gain. We

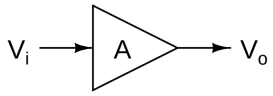



Figure 1.8: An amplifier with an input voltage of $V_i$ and an output voltage of $V_o$.

| dB | −3 | 0 | 3 | 6 | 7 | 10 | 13 | 16 | 20 | 30 | 40 |
|---|---|---|---|---|---|---|---|---|---|---|---|
| $A$ | 0.70 | 1.0 | 1.4 | 2.0 | 2.2 | 3.2 | 4.5 | 6.3 | 10 | 32 | 100 |
| $G$ | 0.50 | 1.0 | 2.0 | 4.0 | 5.0 | 10 | 20 | 40 | 100 | 1000 | 10000 |

Table 1.3: dB conversion table

denote voltage gain by $A$ and power gain by $G$:

$$A = \frac{V_o}{V_i} \text{ and } G = \frac{P_o}{P_i} \tag{1.8}$$

An amplifier with a voltage gain of $A$ is shown as a block diagram in Fig. 1.8. Voltage gain or power gain are unitless quantities. We may use decibels (dB) to describe the amount of gain. We can express the gain expression above in decibels as,

$$\boxed{A_{dB} = 20\log_{10}\left(\frac{V_o}{V_i}\right), \text{ and } G_{dB} = 10\log_{10}\left(\frac{P_o}{P_i}\right)} \tag{1.9}$$

where the logarithm function is with respect to base 10. The coefficient 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 the 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 a power gain of $G = 4$. In decibels, the value of both $A_{dB}$ and $G_{dB}$ is 6 dB. Also, note that 3 dB corresponds to a power gain of 2 and a voltage gain of $\sqrt{2}$.

Decibel notation can also used to define absolute levels. For example, 0.5 milliwatt of power is expressed in decibels as −3 dBm. Here, "m" denotes that this value is relative to 1 milliwatt. Similarly, 20 Watts can be expressed as 43 dBm. Another way of writing absolute levels in decibels is to directly write what it is relative to. For example, we can write 32 $\mu$V as "30 dB re $\mu$V". Some easy-to-remember approximate dB values are given in Table 1.3.

- ⧫ TRC-11 has six amplifiers: One DC amplifier for automatic-gain-control and for driving a light-emitting-diode, two audio (20 Hz–20 kHz) amplifiers, two amplifiers at 15 MHz, and one amplifier at 27 MHz.

## 1.8 Mixers

We frequently want to shift the frequency of the information carrying sinusoid. For transmission purposes, we want to increase the frequency. The process of

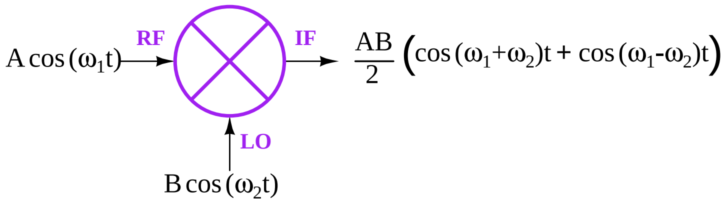



Figure 1.9: A mixer multiplying two sinusoidal signals at different frequencies.

moving the frequency to a higher frequency is called *up-conversion*. When the frequency of the received signal must be reduced, we perform a *down-conversion*. For up-conversion and down-conversion operations, we use *mixers*.

Mixers are basically analog multiplier circuits. They have two input ports and one output port, labeled as RF, LO, and IF. The output signal is the product of the two input signals. Suppose, for example, that the inputs of a mixer have two sinusoidal signals, $A\cos(\omega_1 t)$ and $B\cos(\omega_2 t)$, as shown in Fig. 1.9. The output signal is the product signal: $AB\cos(\omega_1 t)\cos(\omega_2 t)$. Using trigonometric identities, we can write the output signal as

$$\frac{AB}{2}\left(\cos(\omega_1+\omega_2)t+\cos(\omega_1-\omega_2)t\right) \tag{1.10}$$

Hence the output signal is at the sum, $\omega_1+\omega_2$, and at the difference, $\omega_1-\omega_2$, frequencies. If we need an up-conversion, we can select the sum frequency at the higher frequency. On the other hand, if we need a down-conversion, we select the difference frequency. For this selection operation, we need filters.

We discuss mixers in more detail in Chapter 7 on p. 264.

♦ TRC-11 utilizes one mixer to convert 27 MHz to 15 MHz in its receiver.

## 1.9 Filters

We employ *filters* to eliminate unwanted components of a signal and keep the components we like. Most commonly used filters are classified as low-pass, high-pass, and band-pass types.

As the name implies, a low-pass filter (LPF) allows the signals below a specific frequency to pass through the filter and *attenuates* (decrease their amplitude) the signals of higher frequency. This threshold frequency is called *cut-off* or *corner frequency*. This behavior of an LPF is demonstrated in Fig. 1.10: The upper schematic demonstrates the removal of the sinusoidal component above the corner frequency. In the lower schematic, where the input has two sinusoids at two different frequencies, the higher frequency sinusoidal signal is rejected at the output.

A high-pass filter (HPF) has the opposite function: It passes signals of high frequency, while stopping low-frequency signals. (See Fig. 1.11.)

A band-pass-filter (BPF) passes signals within a specified frequency range. It eliminates signals outside this band. Suppose we wish to keep the fundamental component of the square wave of Fig. 1.4 at $\omega$ and eliminate the rest of the components. For this purpose, we can use a band-pass-filter centered at $\omega$. The

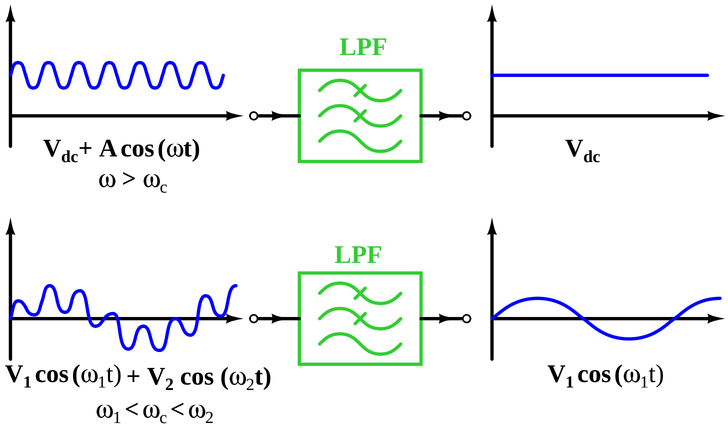



Figure 1.10: Demonstration of the function of a low-pass-filter with a corner frequency of $\omega_c$.

upper diagram in Fig. 1.12 demonstrates this action. The bandwidth is of the filter must be sufficiently small to attenuate the nearest harmonics. Referring to Fig. 1.5, the amplitude of the fundamental component is $4/\pi \approx 1.27$. Conversely, if we use a band-pass-filter centered at $5\omega$, we pick the 5th harmonic of the square wave with a peak amplitude of $4/(5\pi) \approx 0.25$ as shown in the lower diagram of Fig. 1.12.

The filtering effect is not abrupt, but it is gradual. The signal components and noise beyond cut-off frequency are not entirely eliminated, but attenuated more and more as their frequencies are further away from cut-off. A detailed discussion of filters can be found in Chap. 5.

- ♦ TRC-11 has many filters: A crystal band-pass-filter at 15 MHz, a band-pass-filter at 27 MHz, and a number of low-pass and high-pass-filters.

## 1.10 Transmitter and receiver

A conceptual block diagram of a radio transmitter converting sound waves into electromagnetic waves is depicted in Fig. 1.13. A sound wave in air at 1.0 kHz has a wavelength of 34 cm. It is probably generated by a human mouth, having a size comparable to this wavelength. The sound wave is converted into an electrical signal using a microphone, having a size also comparable to this wavelength. The small-signal output of the microphone is amplified by an amplifier. Since the wavelength of the electrical signal (300 km) is too long to be transmitted by a reasonable size antenna, the signal is modulated on a carrier at 27 MHz using a modulator, where the wavelength is reduced to 11.1 m. The signal at 27 MHz is conveniently transmitted by a reasonable length antenna in the form of an electromagnetic wave.

A block diagram of the corresponding receiver is given in Fig. 1.14. Electromagnetic waves received by the antenna are converted to an electrical signal at the same frequency. Since the signal amplitude is small, it is first amplified by an amplifier. The signal is then demodulated from the carrier to obtain the

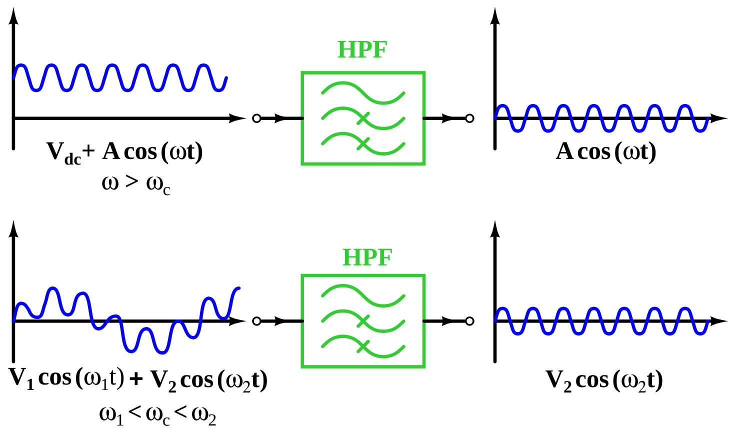



Figure 1.11: Demonstration of the function of a high-pass-filter with a corner frequency of $\omega_c$.

original signal. The signal is fed to a speaker (with a size comparable to the sound wavelength) to generate the sound waves. The sound waves are probably heard by a human ear, whose size is also comparable to wavelength.

Since our transmitter is not the only one in the area, the receiver antenna receives many signals from different sources. Although it is not shown in the simplified diagram, filtering the antenna input signal that rejects all unwanted signals is necessary for a clean reception.

## 1.11 TRC-11

Transceivers are wireless transmitters (TX) and receivers (RX) combined in a single instrument. TRC-11 (see Fig. 1.15) is a transceiver operating in the 28 MHz amateur band, where a license for transmission is not necessary if the output power is kept below a specific limit. Hence TRC-11 transmitter output power is intentionally kept low, not to violate local electromagnetic radiation regulations. On the other hand, the receiver sensitivity is very good, providing communication over some distance.

TRC-11 utilizes the *superheterodyne principle*, which is used by most modern radio receivers (for example, those in mobile phones) today. Superheterodyne receiver systems use a *frequency down-conversion* mechanism of a mixer driven by an oscillator: The incoming AM modulated signal is mixed with a constant amplitude sinusoidal wave of a different frequency generated by a *local oscillator*. The mixer output is an AM signal at a lower and fixed frequency known as intermediate frequency (IF), where the signal is more easily amplified in a narrow-band amplifier chain. If a different frequency input signal is desired, it is sufficient to change the frequency of the local oscillator, while the frequency of narrow-band IF amplifier remains unchanged. The same principle can be used to *up-convert* a low frequency AM signal to its higher frequency version for transmission purposes. More discussion on the superheterodyne principle can be found in p. 275.

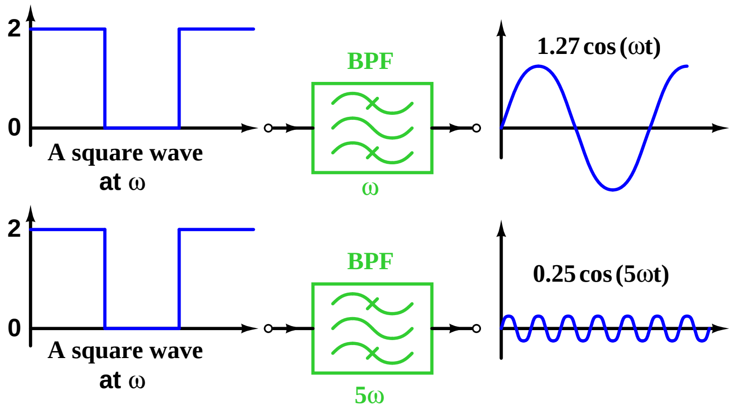



Figure 1.12: A band-pass-filter centered at $\omega$ filtering the fundamental component (upper) and a band-pass-filter centered at $5\omega$ filtering the 5th harmonic (lower) out of a square wave with frequency $\omega$.

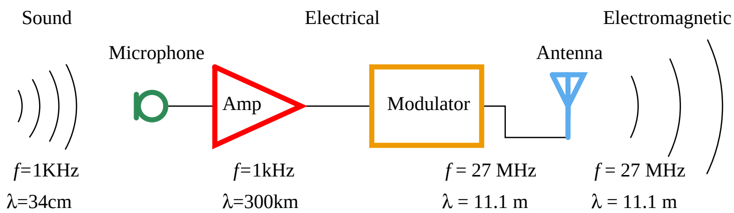



Figure 1.13: A conceptual block diagram of a radio transmitter converting sound waves into electromagnetic waves.

In its transmitter, TRC-11 does *not* use the up-conversion method. Instead, the transmission frequency is generated directly by an oscillator. The signal generated at the transmission frequency is amplitude-modulated and amplifier to be fed to the antenna.

A block diagram of TRC-11 is shown in Fig. 1.16. A low frequency audio input signal to the transmitter is amplified by an audio amplifier. The amplified signal is then used to amplitude modulate a 27.00 MHz sinusoidal signal generated by an oscillator. This 27.00 MHz AM signal is amplified by a radio-frequency (RF) amplifier and then fed to the antenna for transmission into air. A switch (T/R switch) is used to select the transmit or receive mode for the antenna.

A small amplitude 27.00 MHz AM signal picked by the antenna is fed to mixer that acts like a down-converter. The mixer uses a 12.00 MHz local oscillator as the transmitter. The mixer's output has two signals: The difference frequency at 27.00-12.00=15.00 MHz and the sum frequency at 27.00+12.00= =39.00 MHz. 15.00 MHz signal is the desired IF frequency: It is filtered by a 15.00 MHz narrow-band crystal band-pass-filter, providing the good selectivity of the receiver. The resulting signal is amplified by a high-gain IF amplifier chain. The amplitude demodulator block strips the AM signal of its carrier and generates the original audio signal. This signal is then fed to a loudspeaker (or

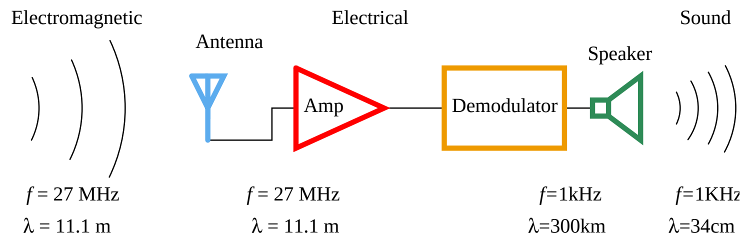



Figure 1.14: A conceptual block diagram of a radio receiver converting electromagnetic waves into sound waves.

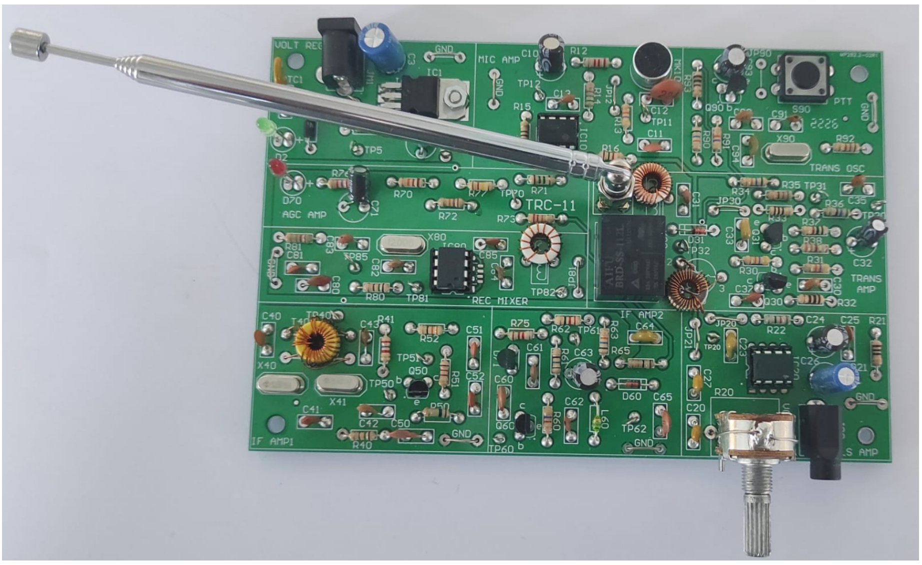


Figure 1.15: Photo of a completed TRC-11 (rev. 7.1)

earphone) amplifier to drive the loudspeaker (or earphone) generating the audio signal. The output of the amplitude demodulator is also used for the automatic gain control circuit, which reduces the gain of the IF amplifier chain and hence prevents a saturation if the input signal is too strong.

If the antenna also picks a neighboring signal at 26.95 MHz, this signal will be down-converted to 26.95-12.00=14.95 MHz using a local oscillator of 12.00 MHz. Since the IF filter is strictly at 15.00 MHz, the 14.95 MHz signal will be rejected. Therefore, very selective TRC-11 receiver only amplifies signals at 27.00 MHz.

Although it is possible to build a transceiver with modern complex integrated circuits in a much smaller area, for the purpose of learning and ease of soldering, TRC-11 is intentionally built from many discrete, inexpensive, —and some old-fashioned— components.

In the following chapters, we will study all blocks of TRC-11 starting from the voltage regulator unit. More information about these subjects can be found

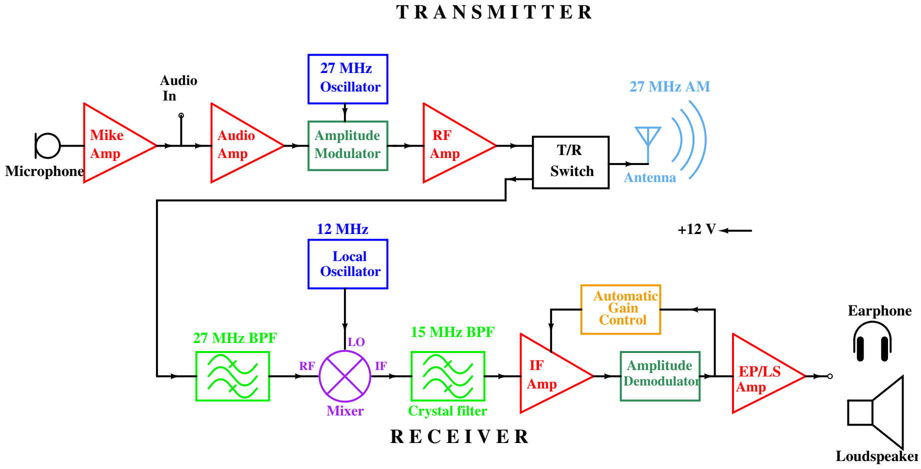



Figure 1.16: A block diagram of TRC-11.

in a book by Nahin [2] or in any yearly edition of the Handbook for Radio Communications published by the American Radio Relay League [3].

## 1.12 Problems

1. Find the wavelengths of sound waves in air at frequencies of 20 Hz and 20 kHz. (20 Hz and 20 kHz are accepted to be lowest and highest audible sound frequencies for most humans.)

2. Find the wavelength of your favorite FM radio station.

3. The input signal of an amplifier is given by $0.01\cos(\omega_o t)$. The output signal is measured to have peak-to-peak voltage of 1 V. Find the voltage gain of this amplifier in dB.

4. The 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 a sum of sinusoids. Plot the magnitude of the individual sine wave components with respect to frequency if $\omega_s$=1000 rps and $\omega_o$=5000 rps, as in Fig. 1.6.

5. Let $s(t) = \cos(\omega_s t) + 2\cos(2\omega_s t) + 3\cos(3\omega_s t) + 4cos(4\omega_s t)$, 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 $s(t)$ with respect to frequency.

6. 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?

7. Show that
$$\tan(A + B) = \frac{\tan(A) + \tan(B)}{1 - \tan(A)\tan(B)}$$

8. Construct a square wave with three and five components and calculate the mean square error, using a computer tool of your choice, a spreadsheet, MATLAB, etc. Mean-square error, $MSE$, between two periodic functions, $f(t)$ and $g(t)$ is given by
$$MSE = \sqrt{\frac{1}{T}\int_0^T (f(t) - g(t))^2 dt}$$
where $T$ is the period of the functions.

# Chapter 2

# CIRCUIT THEORY PRIMER

Circuits are composed of resistors, capacitors, inductors, semiconductor devices, integrated circuits, energy sources, and many other components.* We design circuits employing these components to process electrical energy to perform a particular function. Circuits may contain a large number of components. Algebra and differential equations are the tools that are used to both define the functions of elements and their interrelations. The mathematics of circuit analysis and synthesis, models, and set of rules developed for this purpose is altogether called *circuit theory*. Circuit theory is one of the fundamental tools of electrical engineering [4]. Although this textbook is not a circuit theory textbook, basic rules of circuit analysis are presented. However, you should refer to more comprehensive textbooks [5] to learn the circuit theory.

## 2.1 Electrons

Atoms of some materials, notably metals or acids, allow the movement of the electrons readily. We classify such materials as *conductors*. Gold and aluminum are excellent conductors, while iron and lead are not. Some materials, such as niobium-titanium alloy, exhibit perfect conductivity, known as *superconductivity* below a critical and relatively low temperature. On the other hand, atoms of *insulators* do not allow the electrons to move at all. Dry wood, porcelain, quartz, and rubber are good insulators. Materials, such as germanium and silicon, whose electron conductivity falls midway between good conductors and good insulators, are known as *semiconductors*. The addition of a small percentage of foreign atoms into semiconductors, which is known as *doping*, changes the conductivity properties dramatically. Semiconductor devices [6] provide the enabling technology of the information age.

Electronics are all about controlling electrons. To quantify the movement of electrons in a circuit, we use a number of terms:

- **Charge** represented by $Q$ is used to measure the number of electrons.

*ARRL Handbook of Radio Communications [3] has comprehensive information on electronic components

It has the units of *coulombs* (unit symbol **C**), named after the French physicist Charles-Augustin de Coulomb (1736–1806). He is known for developing the Coulomb's law. Since an electron has a negative charge, $6.241\times10^{18}$ electrons make up a charge of $-1$ C. Equivalently, the charge of one electron is $-1.602\times10^{-19}$C. Positive charges may also exist. An electrolyte may have positively charged ions in addition to negatively charged ions. For example, salty water has positive $Na^+$ and negative $Cl^-$ ions.

- **Current** measures the flow rate of charged particles, represented by the letter $I$. If 1 C of charge moves in one second, it is called one **ampere** (unit symbol **A**), named after French physicist André-Marie Ampère (1775–1836) known for developing Ampere's law. Current is a directional quantity. The current direction is the same as the flow direction of positive charges. Hence, $6.241\times10^{18}$ electrons moving left create a current of 1 A towards the right. The current in a circuit can be measured using an *ammeter*.

- **Voltage** quantifies the electrical potential difference between two points in a circuit. It measures the desire for charges to move from one place to another. It has the unit **volt** (unit symbol **V**), named after Italian physicist Alessandro Volta (1745–1827) , who invented the first chemical battery. 1 V of voltage can deliver 1 J (joule) of energy to 1 C of charge ($E = QV$). The potential difference between the two points is measured using a *voltmeter*.

- **Resistance** defines the degree to which a conductor opposes the electric current through it. The unit of resistance is one **ohm** (unit symbol is Greek letter capital omega Ω), named after German physicist Georg Simon Ohm (1789–1854). A good conductor like a copper wire has a very low resistance; the electrons flow freely through it. Water is a relatively poor conductor of current, so it has a higher resistance. Insulators like glass or ceramics have very high resistance, with negligible current through them. *Resistivity* represented by the Greek letter $\rho$ is an intrinsic property of a material that quantifies how strongly that material opposes the flow of current. The resistivity of some common materials is listed in Table 2.1. The unit of resistivity is Ω-cm and it defines the resistance between the opposing faces of one cubic centimeter of the material.

  The resistance of a material with a resistivity of $\rho$, a cross-sectional area of $A$ and a length of $l$ is given by

  $$R = \frac{\rho l}{A} \tag{2.1}$$

  Electrical components called *resistors* have a predefined level of resistance, represented by the symbol $R$. Their typical values range from 1 mΩ to 1 GΩ (G means $10^9$). The resistance of a component can be measured using an *ohmmeter*.

### 2.1.1 Water flow analogy

Since electrons are not visible, it is helpful to make an analogy to hydraulic systems to understand the concepts.

| | Material | $\rho$, resistivity ($\Omega$-cm) |
|---|---|---|
| Superconductors | Niobium-titanium (below 11°K) | 0 |
| Conductors | Silver | $1.59 \times 10^{-6}$ |
| | Copper | $1.68 \times 10^{-6}$ |
| | Gold | $2.44 \times 10^{-6}$ |
| | Aluminum | $2.65 \times 10^{-6}$ |
| | Tungsten | $5.6 \times 10^{-6}$ |
| | Iron | $9.7 \times 10^{-6}$ |
| | Lead | $22 \times 10^{-6}$ |
| Semiconductors | Doped germanium | 0.001 – 0.2 |
| | Pure germanium | 47 |
| | Doped silicon | 0.01 – 0.5 |
| | Pure silicon | $2 \times 10^{5}$ |
| | Doped gallium-arsenide | 0.1 – 2 |
| | Pure gallium-arsenide | $1 \times 10^{8}$ |
| Insulators | Glass | $1 \times 10^{9} - 1 \times 10^{13}$ |
| | Fused quartz | $7.5 \times 10^{17}$ |

Table 2.1: Resistivity of some common materials at 20°C.

- **Charge** is equivalent to the quantity or volume of water (e.g., in units of liters). We note that in hydraulic systems the volume is only positive hence the equivalent of negative charge does not exist.
- **Current** is equivalent to the flow rate of water (e.g., in units of liters/sec)
- **Voltage** is analogous to the pressure difference between two points in the hydraulic system.
- **Resistance** is created in water pipes due to friction between water and the pipe's inner surface. A relatively wide pipe has a very low resistance. The resistance of the pipe increases as the pipe diameter is reduced. A hydraulic resistor can be created by a constriction in the bore of the pipe (see Fig. 2.10). For example, a water tap is analogous to an adjustable resistor.

## 2.2 Energy Sources

All circuits consume energy in order to work. In electronic circuits, energy sources are either in the form of voltage sources or current sources.

### 2.2.1 Voltage source

An ideal voltage source can provide a defined voltage across its terminals regardless of the amount of current drawn from it. This means that the ideal voltage source is capable of providing infinite amount of energy (a single ideal voltage source could have solved the world energy crisis if it had existed). Energy sources of infinite capacity are not available in nature. The concept of ideal source, however, is essential and instrumental in the analysis of circuits. It is not allowable to short-circuit a voltage source, since it creates a contradiction. The DC voltage source symbol is shown in Fig. 2.1(a). The characteristics of the DC voltage source in the form of an $I-V$ plot are shown in Fig. 2.1(b), where the voltage is constant at $V_1$ independent of the current amplitude and direction flowing through it. A voltage source may also have a time-dependent voltage value. In that case, we use the symbol shown in Fig. 2.1(c).

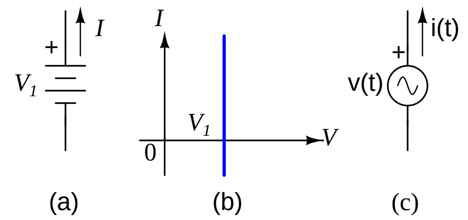



Figure 2.1: (a) Symbol of DC voltage source of value $V_1$, (b) $I-V$ characteristics of a DC voltage source of value $V_1$, (c) symbol of AC voltage source of value $v(t)$.

The value of a DC voltage source can be measured by a DC *voltmeter*. Common AC voltmeters can measure the voltage of an AC voltage source correctly if the voltage is sinusoidal. If the voltage is not sinusoidal, an *oscilloscope* should be utilized, which shows the voltage waveform as a function of time.

A voltage source is analogous to a huge reservoir of water at a certain height. This reservoir provides a constant pressure regardless of water drawn from it. As long as the water height is not reduced, the pressure stays the same.

An alkaline battery sold in the supermarket is almost like a DC voltage source with $V_1$=1.5 V. Similarly, one cell of a nickel-cadmium (Ni-Cd) rechargeable battery approximates a DC voltage source with a nominal voltage of 1.2 V. Lithium-ion batteries commonly used in mobile phones are also rechargeable batteries with a nominal voltage of 3.7 V. Lead-acid battery used in cars is composed of 6-cells, each cell with a nominal voltage of 2 V. The voltage of a battery drops, if a high current is drawn from it. So, none of these batteries is an ideal voltage source.

### 2.2.2 Current source

An ideal current source can provide a set current value whatever the voltage across its terminals may be. Similar to the voltage source, it can provide infinite amount of energy. Therefore, it is only a mathematical representation. While it is possible to short-circuit a current source, it is *not allowed* to be open-circuited since it creates a contradiction. The symbol for a current source is depicted in Fig. 2.2(a). The

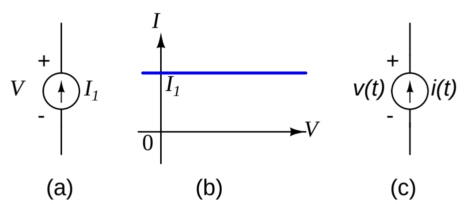



Figure 2.2: (a) Symbol for a DC current source of value $I_1$, (b) $I-V$ characteristics of a DC current source of value $I_1$, (c) symbol for an AC current source of value $i(t)$.

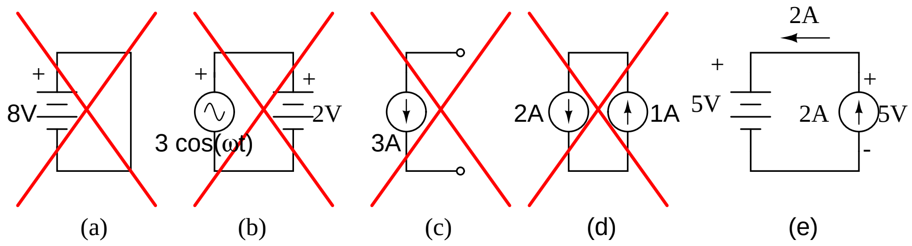



Figure 2.3: (a) Not allowed: a short-circuited voltage source, (b) not allowed: two voltage sources with different values in parallel, (c) not allowed: an open-circuited current source, (d) not allowed: two current sources with different values in series (e) allowed: a current source in parallel with a voltage source.

$I-V$ characteristic of the DC current source of value $I_1$ is shown in Fig. 2.2(b). It provides a current of $I_1$ regardless of the amount or direction of the voltage across it. Current sources with time varying current values are shown with the same symbol as in Fig. 2.2(c).

A current source is equivalent to a constant flow pump. It provides the same water flow rate regardless of the pressure necessary to do it.

A battery charger is almost like a DC current source. It provides the same DC current during the charging period regardless of the voltage of the battery being charged. A battery charger is not an ideal current source: It is allowable to open-circuit a battery charger (no battery connected).

### 2.2.3 Prohibited circuits

Connection of ideal voltage and current sources in certain ways can create contradictions. Those circuits are not allowed.

Fig. 2.3 shows a number of examples. Fig. 2.3(a) and (b) are prohibited circuits where a voltage source is short-circuited or two voltage sources of different values are connected in parallel. Such circuits create a contradiction, and they are prohibited. In Fig. 2.3(c), a current source is open-circuited. The current source insists on pushing a 3 A current while the open-circuit does not allow it. This is a contradiction. In (d), two current sources of different values are connected in parallel. One would like to push 2 A, while the other insists on 1 A creating a contradiction. In (e), a 5 V voltage source is in parallel with a 2 A current source. The current in the loop is set by the current source at 2 A, while the voltage across the current source is 5 V as determined by the voltage source. There is no contradiction in this connection.

### 2.2.4 Power and Energy

The *instantaneous power* delivered by a source can be defined as

$$\boxed{p(t) = v(t)i(t)} \tag{2.2}$$

with the directions of voltages and currents as shown in Figs. 2.1 and 2.2. If the value of $p(t)$ is positive, power is delivered by the source. On the other hand, if

$p(t)$ is negative, it means the power is absorbed by the source.

If the signals are periodic with period $T$, then we can define an *average power*, $P$, delivered by a source as

$$\boxed{P = \frac{1}{T}\int_0^T v(t)i(t)\,dt} \tag{2.3}$$

Power is measured in units of *watts* (W), named after Scottish scientist James Watt (1736–1819). For example, if a light bulb in a 3 V flashlight draws 100 mA, the power delivered by the batteries is $P = 3 \times 0.1 = 0.3$ W.

Energy is the work done by an electrical source in given time duration. It is defined as

$$\boxed{E(T) = \int_0^T p(t)dt} \tag{2.4}$$

Energy is measured in *joules* (J), named after English physicist James Prescott Joule (1818–1889). For example, a 12 V car battery of 80 Ampere-hours capacity stores an energy of $E = 12 \times 80 \times 3600 = 3.4 \times 10^6 = 3400$ kJ, while a Ni-Cd battery of 1.2 V with a 600 mAh capacity stores energy of $E = 1.2 \times 0.6 \times 3600 = 2.6$ kJ.

Suppose the voltage across an element, $v(t)$, is given by a *sinusoid* at the radial frequency of $\omega$ as

$$v(t) = V_1 \cos(\omega t + \theta_v) \tag{2.5}$$

and the current, $i(t)$, through it is given similarly by

$$i(t) = I_1 \cos(\omega t + \theta_i) \tag{2.6}$$

where $\theta_v$ and $\theta_i$ are the *phases* of the voltage and current, respectively. We can calculate the instantaneous power, $p(t)$, delivered to it as

$$p(t) = v(t)i(t) = V_1 I_1 \cos(\omega t + \theta_v) cos(\omega t + \theta_i) \tag{2.7}$$

or

$$p(t) = \frac{V_1 I_1}{2}\cos(\theta_v - \theta_i) + \frac{V_1 I_1}{2}\cos(2\omega t + \theta_v + \theta_i) \tag{2.8}$$

In the case of a resistor, the current and voltage have the same phase ($\theta_v = \theta_i$), and hence we can write the power delivered to a resistor as

$$p(t) = \frac{V_1 I_1}{2} + \frac{V_1 I_1}{2}\cos(2\omega t + 2\theta_v) \tag{2.9}$$

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

The *average power*, $P$, is the average value of $p(t)$ in Eq. 2.8 integrated over one cycle:

$$P = \frac{1}{T}\int_0^T p(t)dt = \frac{V_1 I_1}{2}\cos(\theta_v - \theta_i) \tag{2.10}$$

For a resistive load, the voltage and current phases are the same. Hence we have $\theta_v = \theta_i$, and the average power is

$$P = \frac{V_1 I_1}{2} \tag{2.11}$$

We note that if the element is such that the phase difference between the voltage across and current through is $\theta_v - \theta_i = 90^o$, $P$ is zero. Inductors and capacitors are such elements with no power dissipation.

### 2.2.5 Root-mean-square (rms)

Alternating current (AC) signals are usually specified in *root-mean-square* (rms) quantities. A periodic voltage waveform, $v(t)$, or a periodic current waveform, $i(t)$, with a period $T$ have rms values of

$$\boxed{V_{rms} = \sqrt{\frac{1}{T}\int_0^T v^2(t)\,dt}} \quad \text{and} \quad \boxed{I_{rms} = \sqrt{\frac{1}{T}\int_0^T i^2(t)\,dt}} \tag{2.12}$$

If the voltage across and current through an electrical element are sinusoidal with

$$v(t) = V_1 \cos\left(\frac{2\pi}{T}t + \theta_v\right) \quad \text{and} \quad i(t) = I_1 \cos\left(\frac{2\pi}{T}t + \theta_i\right) \tag{2.13}$$

where $V_1$ and $I_1$ are the peak amplitudes, and $T$ is the period. The *rms* value of $v(t)$, $V_{rms}$, is found after integration operation as

$$V_{rms} = \sqrt{\frac{1}{T}\int_0^T v^2(t)\,dt} = \sqrt{\frac{V_1^2}{T}\int_0^T \cos^2\left(\frac{2\pi}{T}t + \theta_v\right)\,dt} = \frac{V_1}{\sqrt{2}} \tag{2.14}$$

where $T$ is the period o the sine wave. Similarly, rms value of a sine wave current, $i(t)$, is

$$I_{rms} = \sqrt{\frac{1}{T}\int_0^T i^2(t)\,dt} = \sqrt{\frac{I_1^2}{T}\int_0^T \cos^2\left(\frac{2\pi}{T}t + \theta_i\right)\,dt} = \frac{I_1}{\sqrt{2}} \tag{2.15}$$

The average power dissipated on that element is given by

$$P = V_{rms} I_{rms} \cos(\theta_v - \theta_i) \tag{2.16}$$

We note that an AC voltmeter measures the rms (not the peak) value of the voltage across its terminals, assuming that the voltage is sinusoidal. Similarly, an AC ammeter measures the rms current flowing through it. Since the ratio of the peak value and to the rms value is $\sqrt{2}$ for a sinusoidal signal, a 220 $V_{rms}$ sinusoidal line voltage has a peak value of $220\sqrt{2}$=311 V. Using Eq. 2.16, we deduce that a $P$=60 W light bulb operating at the line voltage of 220 $V_{rms}$ draws a current of $60/220 = 0.27$ $A_{rms}$, since $\theta_v = \theta_i$ for the light bulb.

#### Example 1

The voltage across a 10 $\Omega$ resistor, $v_R(t)$, is triangular with a period $T$ as shown in Fig. 2.4. Find the average power, $P_{av}$, dissipated in the resistor.

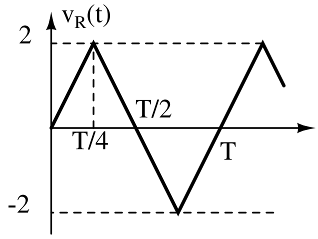



Figure 2.4: Voltage waveform for Example 1.

**Solution**

Due to symmetry, the rms voltage, $V_{Rrms}$, of the resistor can be found by integrating over only a quarter cycle (between 0 and $T/4$):

$$V_{Rrms} = \sqrt{\frac{1}{T}\int_0^T v_R(t)^2(t)\,dt} = \sqrt{\frac{4}{T}\int_0^{T/4}\left(\frac{2t}{T/4}\right)^2 dt} = \sqrt{\frac{4}{T}\left.\frac{64t^3}{3T^2}\right|_0^{T/4}} = \frac{2}{\sqrt{3}}$$

(The rms value of a triangular waveform is $1/\sqrt{3}$ times its peak value.) Hence the average power dissipated in the resistor is

$$P_{av} = \frac{V_{Rrms}^2}{R} = \frac{2^2}{3}\frac{1}{10} = 0.133\ W.$$

---

### 2.2.6 Real-life sources

Real sources deviate from ideal sources in only one aspect. The voltage or current supplied by a real source has a dependence on the amount of current drawn from it. For example, a battery has an internal resistance. When connected to a circuit, its terminal voltage decreases by an amount proportional to the current drawn from it. This is depicted in Fig. 2.5(a). Although it is not recommended, it is possible to short-circuit the terminals, since $R_S$ limits the current. When

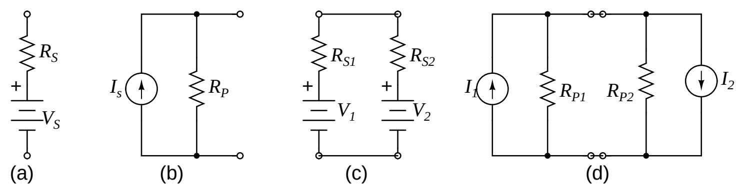



Figure 2.5: (a) Equivalent circuit of a real-life voltage source with a voltage $V_S$ and an internal resistance of $R_S$, (b) equivalent circuit of a real-life current source with a current of $I_S$ and a parallel resistance of $R_P$, (c) an allowed circuit: two real voltage sources of different values in parallel, (d) an allowed circuit: two real current sources of different values in series.

there is no current drawn from the battery, the voltage across the terminals is

$V_o$. When a *load resistance*† is connected to this battery, the voltage across the battery terminals is no longer $V_o$, since there is a voltage drop across $R_S$.

A real-life current source is given in Fig. 2.5(b). It has a parallel resistance of $R_P$. It is possible to leave it open-circuited without causing a contradiction because the current $I_o$ can go through the resistor $R_P$.

Fig. 2.5(c) and (d) show allowed circuits since the connections do not create a contradiction due to the presence of resistors.

### 2.2.7 Power line

Power line voltages differ from country to country, but there are only a few standards. Line voltages are 120 $V_{rms}$ or 220 $V_{rms}$, where all voltages are specified in *rms* (The definition of rms voltage is given in Eq. 2.26). The power line in most of Europe is 50 Hz/220 $V_{rms}$, while it is 60 Hz/120 $V_{rms}$ in America.

Electrical 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. The voltage level is either 6.3 $kV_{rms}$ (=6300 $V_{rms}$) or 13.8 $kV_{rms}$ 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 154 $kV_{rms}$ or 380 $kV_{rms}$. 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 $kV_{rms}$ medium voltage (MV), in the vicinity of the area (may be a town, village, 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 207 to 244 $V_{rms}$) in the close vicinity of the consumer. All this step-up and step-down is done by using power transformers.

We are accustomed to seeing the electric energy coming out of the household systems 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

$$v_1(t) = V_p \sin(\omega t) \tag{2.17}$$

then, it is always accompanied by two other related components

$$\begin{aligned} v_2(t) &= V_p \sin(\omega t + 120^o) \\ v_3(t) &= V_p \sin(\omega t + 240^o) \end{aligned} \tag{2.18}$$

This is necessitated by the economics of the technology employed in electromechanical power conversion. A three-phase system is more economical than an equivalent single-phase system because it uses less conductor material to transport the same power. The three-phase system was invented by Nikola Tesla, a Serbian American scientist (1856–1943), eliminating the DC system then promoted by Thomas Edison (1847–1931) as a result of *War of Currents*. These three phases of line supply are distributed to the consumers such that all three phases are evenly loaded as much as possible.

†A load resistance could be the resistance of an element that performs a useful function, such as a flash light bulb, a heater or a cooling fan.

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-to-line* voltage, e.g., between $v_1(t)$ and $v_2(t)$, is

$$\Delta v(t) = v_1(t) - v_2(2) = \sqrt{3}V_p \sin(\omega t - 30^o) \tag{2.19}$$

The potential difference between the phases is, therefore, $\sqrt{3}$=1.73 times larger than any one of phase voltages with respect to neutral. The line-to-line voltage level is 381 $V_{rms}$ for a phase voltage of 220 $V_{rms}$. The last step-down from MV to low voltage (LV) is depicted in Fig. 2.6. Note that there is no neutral for

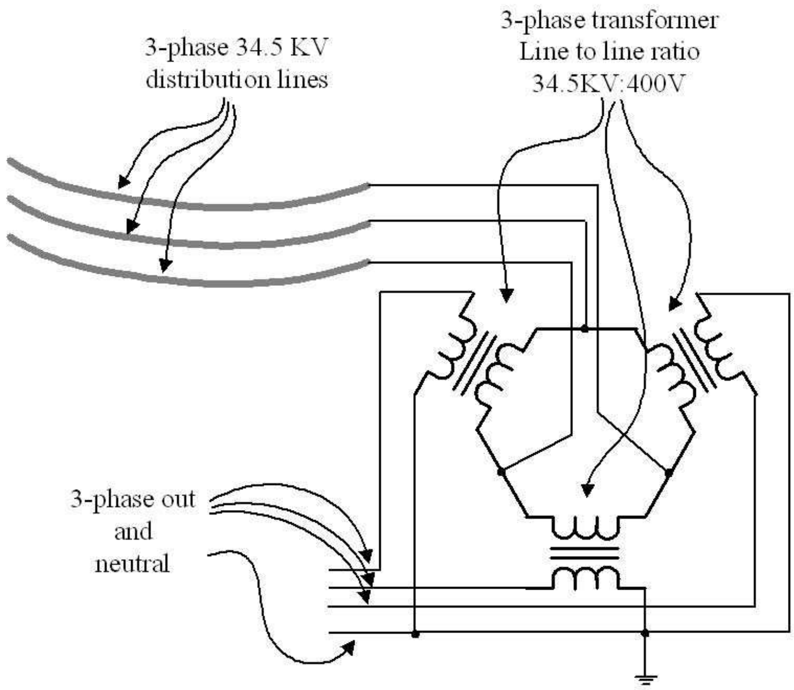



Figure 2.6: A 3-phase MV to LV transformer

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 is 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 the earth. A separate line connected to the earth is also distributed since most household and professional equipment require a separate earth connection for safety. Neutral is the return path of the current we draw from the line. Chassis of household equipment or electrical devices are connected to the earth line. We do not expect any significant current on the earth connection. If there is a significant current, there may be a leakage problem in the electrical system or the equipment.

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 Fig. 2.6. The three-phase line-to-line voltage of 34.5 kV MV is connected to the primary windings of a three-phase transformer, which is connected in a $\Delta$ 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 the earth, while there is no earth connection on the primary. The voltage transformation ratio in these transformers is always stated as the ratio of line-to-line voltages (i.e., the potential difference between the phases) of primary and secondary windings. However, the physical turns ratio of primary and secondary windings correspond to 34.5 kV to 220 V.

Three 220 V live lines, neutral and earth are distributed in the buildings through a few distribution panels. Precautions against excessive current are taken at each panel. This reduces the fire risk in the building and is not helpful to avoid electric shock. One can get electric shock either by touching both live and neutral simultaneously, or by touching live while having contact with the ground.

Building floors have a connection to a 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/she gets 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 difference between the line and neutral currents, and when it exceeds 30 mA, it breaks the circuit. This decreases the severity of the shock.

## 2.3 Kirchhoff's Circuit Laws

### 2.3.1 Kirchhoff's Voltage Law (KVL)

Kirchhoff's Voltage Law (KVL) states that algebraic sum of voltages around any loop should be zero. If $V_n$'s represent the voltages in a loop with $N$ voltages, we have

$$\sum_{n=1}^{N} V_n = 0 \tag{2.20}$$

We note that the individual $V_n$'s could be positive or negative depending on the directions. This law is a direct result of *conservation of energy.*

This law is named after German physicist Gustav Robert Kirchhoff (1824–1887).

Fig. 2.7 shows two example circuits. The *reference directions* for voltages are assigned arbitrarily by the positions of the + signs. We note that the individual voltages can be positive or negative depending on the direction assignment. For the first circuit in (a), from KVL we can write

$$\sum_{n=1}^{N} V_n = -V_V - V_R + V_I = 0$$

While going clockwise around the loop, we use a plus sign in KVL equation if the first-encountered sign of a component voltage is plus, a negative sign otherwise.

Using the same convention for the second circuit in (b), the KVL equations for the three loops are:

$$\begin{aligned} -V_S + V_1 + V_2 - V_3 &= 0 \\ -V_S + V_1 - V_4 + V_5 - V_3 &= 0 \\ -V_2 - V_4 + V_5 &= 0 \end{aligned}$$

We do not have three independent equations. The third equation can be obtained by subtracting the first one from the second.

### 2.3.2 Kirchhoff's Current Law (KCL)

From the *conservation of charge*, Kirchhoff's current law states that the sum of currents flowing into any node (A *node* is a point in the circuit where more than two elements are connected together) should be equal to the sum of currents leaving that node. For a node with $N$ branches, it can be written as

$$\sum_{n=1}^{N} I_n = 0 \tag{2.21}$$

where the branch currents flowing into the node have a negative sign, and the branch currents leaving the node have a positive sign.

Fig. 2.8 shows two example circuits.[‡] In the first circuit of (a), we choose directions for currents arbitrarily. $I_2$ can be found from Ohm's Law as $I_2$=

[‡]To simplify the notation, we use 1000 Ω=1KiloΩ=1K and 1,000,000 Ω=1MegaΩ=1 M.

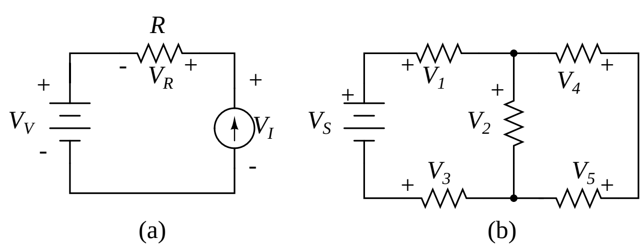



Figure 2.7: (a) An example circuit with three components and one loop, (b) an example circuit with six components and three loops

=5 V/1 kΩ=5 mA. KCL at node A can be written as

$$\sum_{n=1}^{N} I_n = -I_1 + I_2 + I_3 = -I_1 + 5 \text{ mA} + 3 \text{ mA} = 0$$

Hence, we find $I_1$=8 mA.

In the second circuit of Fig. 2.8(b), we see that the current sources determine the currents $I_1 = 5\mu$A (micro, $\mu = 10^{-6}$) and $I_4 = 2\mu$A (1.5 V voltage source and 2 MΩ resistor do not have any influence). Moreover, from Ohm's law, we determine $I_2$=6 V/1M=6 $\mu$A. From the KCL at node B, we write

$$\sum_{n=1}^{N} I_n = I_1 + I_2 + I_3 - I_4 = 5\mu\text{A} + 6\mu\text{A} + I_3 - 2\mu\text{A} = 0$$

Hence, we find $I_3 = -9\mu$A. Since it is a negative quantity, the actual current direction for $I_3$ is upwards.

## 2.4 Resistors and Ohm's Law

The relation between the voltage across a resistor, $V$, and its current, $I$, is governed by Ohm's law (see Fig. 2.9):

$$\boxed{V = RI} \tag{2.22}$$

where $R$ is the *resistance* of the resistor. In a given resistor, more current flows, if more voltage is applied.

In the water flow analogy illustrated in Fig. 2.10(a), more water will flow through a pipe if the pressure across the pipe increases. A variable resistor is analogous to a water tap (Fig. 2.10(b)).

### Example 2

Let us determine the resistance of a 1 mm diameter copper wire of 100 m length. Using Eq. 2.1 and Table 2.1 we write

$$R = \frac{\rho l}{A} = \frac{(1.68 \cdot 10^{-8} \ \Omega\text{-m})(100 \text{ m})}{\pi (0.5 \cdot 10^{-3})^2 \text{ m}^2} = 2.14 \ \Omega \tag{2.23}$$

---

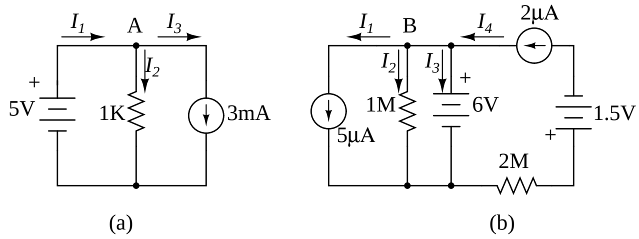



Figure 2.8: (a) An example circuit with three components, (b) an example circuit with six components

r[ht]

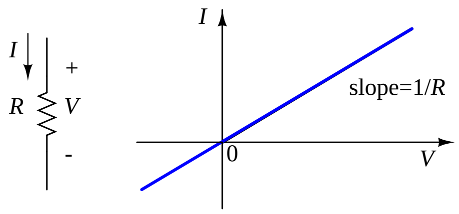



Figure 2.9: Voltage $V$ across a resistor $R$ results in a current of $I$ through it.

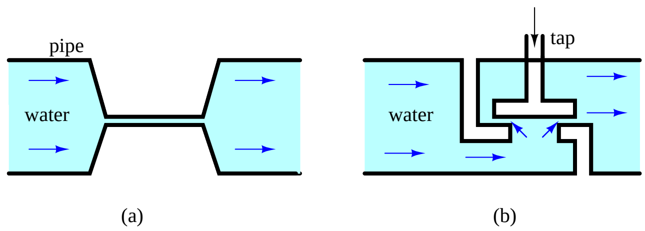



Figure 2.10: (a) Resistor analogy: a water pipe with a constriction, (b) Variable resistor analogy: a water tap

### 2.4.1 Power dissipation in resistors

Resistors *dissipate* energy. Energy dissipation means that all the electrical energy applied to them gets converted into heat energy. As we increase the power delivered to a resistor, it warms up. The *instantaneous power* consumed (or dissipated) on a resistor can be found from

$$p(t) = v(t)\, i(t) = \frac{v^2(t)}{R} = i^2(t)\, R \tag{2.24}$$

If the applied signals are *periodic*, the *average power*, $P$, can be found using an average over a period $T$:

$$P = \frac{1}{T}\int_0^T v(t)i(t)dt = \frac{1}{R}\frac{1}{T}\int_0^T v^2(t)dt = R\frac{1}{T}\int_0^T i^2(t)dt \tag{2.25}$$

From Eq. 2.12, we have

$$V_{rms}^2 = \frac{1}{T}\int_0^T v^2(t)dt \quad \text{and} \quad I_{rms}^2 = \frac{1}{T}\int_0^T i^2(t)dt \tag{2.26}$$

and hence we can simplify the expressions in Eq. 2.25. The average power dissipated on $R$ is

$$\boxed{P = \frac{V_{rms}^2}{R} = I_{rms}^2 R} \tag{2.27}$$

in unit of watts.

Real-life resistors have power dissipation limit. If this limit is exceeded, a resistor may get destroyed.

**Example 3**

Consider the circuit of Fig. 2.11. Find the power delivered/dissipated by the sources and the resistor. Check that the total power delivered is equal to total power dissipated. Note that the reference directions for resistor and for sources are different.

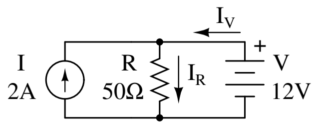



Figure 2.11: Circuit for Example 3

**Solution**

The voltage across the resistor is determined by the voltage source. The current through the resistor is $I_R = V/R = 12/50 = 0.24$ A. Hence the power dissipation in the resistor is

$$P_R = VI_R = \frac{12}{0.24} = 2.88 \text{ W}$$

The voltage across the current source is also determined by the voltage source. The power delivered by the current source is

$$P_I = VI = 12 \times 2 = 24 \text{ W}$$

The current through the voltage source is determined by KCL: $I_V = I_R - I = 0.24 - 2 = -1.76$ A. The power delivered by the voltage source is

$$P_V = VI_V = 12 \times (-1.76) = -21.12 \text{ W}$$

Since $P_V$ is negative, the power is not delivered but rather absorbed by the voltage source. The total power delivered is equal to the total power dissipated:

$$P_I + P_V = 24 - 21.12 = 2.88 = P_R$$

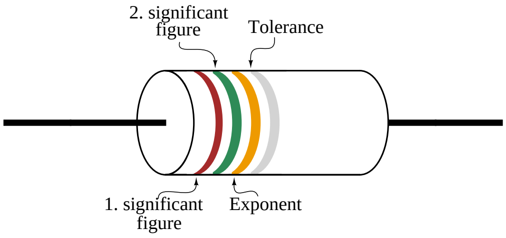
The power delivered by the current source is partly dissipated in the resistor and the remaining part is absorbed by the voltage source.

---

### 2.4.2 Resistor color codes

The resistors that we use in electronics are made of various materials: Carbon composition, metal film, metal oxide, etc. Most abundant are carbon resistors. Most resistors have a color code around them to indicate resistance values. The resistance is expressed in terms of a sequence of colored bands on the resistor body. The color codes are given in Table 2.2. Resistors with 5% and 10% tolerance have 4-band color codes. Hence, a 100 Ω resistor with 10% tolerance is marked as brown-black-brown-silver, and a 4.7 kΩ resistor with 5% tolerance is marked as yellow-violet-red-gold.

The resistors with 10% tolerance are available in standard values with the ratio of consecutive values about 1.2. The two *significant figures* (see Appendix B for an explanation of significant figures) of standard resistor values are:




| Color | Significant figure | Multiplier | Tolerance |
|---|---|---|---|
| Black | 0 | $\times 10^0$ | |
| Brown | 1 | $\times 10^1$ | $\pm$ 1% |
| Red | 2 | $\times 10^2$ | $\pm$ 2% |
| Orange | 3 | $\times 10^3$ | |
| Yellow | 4 | $\times 10^4$ | |
| Green | 5 | $\times 10^5$ | $\pm$ 0.5% |
| Blue | 6 | $\times 10^6$ | |
| Violet | 7 | $\times 10^7$ | |
| Gray | 8 | $\times 10^8$ | |
| White | 9 | $\times 10^9$ | |
| Gold | | $\times 10^{-1}$ | $\pm$ 5% |
| Silver | | $\times 10^{-2}$ | $\pm$ 10% |

Table 2.2: Resistor color codes. The resistor shown above has a 4-band code: 15 kΩ with 10% tolerance.

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

On the other hand, the resistors with 5% tolerance can be found in the following values:

10, 11, 12, 13, 15, 16, 18, 20, 22, 24, 27, 30, 33, 36, 39, 43, 47, 51, 56, 62, 68, 75, 82 and 91.

◆ TRC-11 uses 34 resistors with 10% tolerance in the range 10 Ω to 1 MΩ all with a power dissipation limit of 0.25 W.

Average power dissipation limits of resistors are specified by the manufacturer. Typically, the size of the resistors determines the power limit. Fig. 2.12 shows leaded resistors of different power ratings. Most commonly used leaded resistors found in the lab can dissipate an average power up to 1/4 W. For example, a 100 Ω resistor with a 1/4 W limit can handle a maximum DC voltage of 5 V.

Resistors with 0.5%, 1%, or 2% tolerance with three significant figures are also available, albeit at a higher cost. Such resistors have 5-band color codes, where the first three bands show the three significant figures.

Modern electronic circuits use surface-mount-devices (SMD). These smaller components do not have leads and have a smaller power rating.

Variable resistors are also used frequently. They are typically built by a sliding contact on a carbon film. High power variable resistors are called *rheostats*,

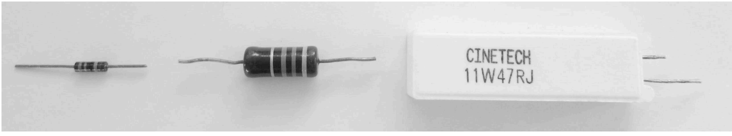



Figure 2.12: Leaded resistors with 1/4W, 1W, and 11W power ratings.

built by a sliding wiper on a resistance wire. A *potentiometer* or *pot* for short is a three-terminal resistor with the sliding contact being the third terminal. *Trimpots* are smaller potentiometers to be mounted on printed circuit boards (PCB). They are meant to be used for only a few adjustments over their lifetime (see Fig. 2.13).

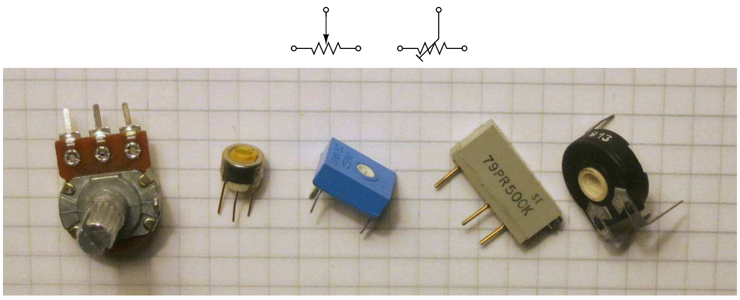


Figure 2.13: The symbol of a potentiometer (top left), of a trimpot (top right). Photos of a potentiometer (bottom left) and trimpots of different types. Multi-turn trimpots have high degrees of accuracy (second from the bottom right).

♦ TRC-11 has one potentiometer for volume control.

### 2.4.3 Series connected resistors

Consider two resistors connected in series as shown in Fig. 2.14(a). From KCL, the current through the resistors is the same. From KVL the total voltage across the two resistors, $V$, is equal to the sum of voltages across each resistor:

$$V = V_1 + V_2 = R_1 I + R_2 I = (R_1 + R_2) I \tag{2.28}$$

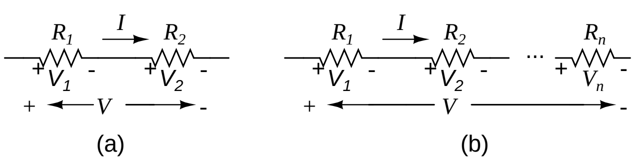



Figure 2.14: (a) Two series-connected resistors, (b) $n$ series-connected resistors

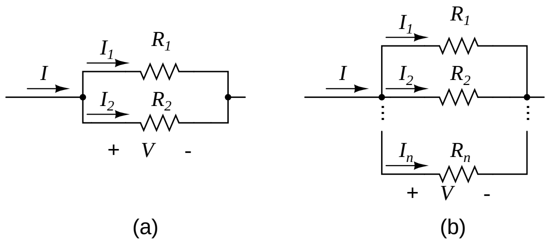



Figure 2.15: (a) Two parallel-connected resistors, (b) $n$ parallel-connected resistors

where we used Ohm's law for each resistor. Therefore, in a series connection of two resistors, the total resistance is equal to the sum of the resistances:

$$\boxed{R = R_1 + R_2} \tag{2.29}$$

Series connected resistors are used frequently as a *voltage divider*. The voltage, $V_1$, across $R_1$ can be written in terms of $V$ as

$$\boxed{V_1 = \frac{R_1}{R_1 + R_2} V} \tag{2.30}$$

If there are $n$ resistors in series as in Fig. 2.14(b), the voltage across the resistor $R_1$ can be found from

$$\boxed{V_1 = \frac{R_1}{R_1 + R_2 + \ldots + R_n} V} \tag{2.31}$$

Since the voltage dividers are very common, it is worth learning the formula above.

When $n$ resistors are connected in series, the total resistance can be found easily:

$$\boxed{R = R_1 + R_2 + \cdots + R_n} \tag{2.32}$$

### 2.4.4 Parallel connected resistors

When two resistors are connected in parallel as shown in Fig. 2.15(a), the voltage, $V$, across every one of them is the same, but each one has a different current passing through it. Using KCL and Ohm's law, we write

$$I = I_1 + I_2 = \frac{V}{R_1} + \frac{V}{R_2} = (\frac{1}{R_1} + \frac{1}{R_2})V \tag{2.33}$$

For a parallel connection of two resistors, the total resistance is given by

$$\boxed{R = R_1 \parallel R_2 = \left(\frac{1}{R_1} + \frac{1}{R_2}\right)^{-1} = \frac{R_1 R_2}{R_1 + R_2}} \tag{2.34}$$

Two parallel-connected resistors are often used as a *current divider*. The current $I_1$ in resistor $R_1$ can be written in terms of the total current $I$ as

$$\boxed{I_1 = \frac{R_2}{R_1 + R_2} I} \tag{2.35}$$

Note that unlike Eq. 2.30, the resistance in the numerator refers to the resistor in the *other* branch.

When $n$ resistors are connected in parallel as shown in Fig. 2.15(b), the total resistance can be found from

$$\boxed{R = R_1 \parallel R_2 \parallel \cdots \parallel R_n = \left(\frac{1}{R_1} + \frac{1}{R_2} + \cdots + \frac{1}{R_n}\right)^{-1}} \tag{2.36}$$

For $n$ resistors in parallel, the current divider formula becomes

$$\boxed{I_1 = \frac{R_2 \parallel R_3 \parallel \ldots \parallel R_n}{R_1 + (R_2 \parallel R_3 \parallel \ldots \parallel R_n)} I} \tag{2.37}$$

where the numerator contains the parallel combination of all resistors in the *other* branches.

### 2.4.5 Resistive circuits

Electrical circuits can have resistors connected in all possible configurations. Consider, for example, the circuit given in Fig. 2.16(a). Two resistors are connected in series, which are then connected in parallel to a third resistor. The

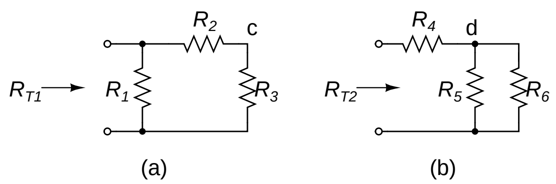



Figure 2.16: Examples of resistive circuits

equivalent resistor, $R_{T1}$, can be found as the parallel combination of $R_1$ with $R_2 + R_3$ using Eq. 2.29 and 2.34:

$$R_{T1} = R_1 \parallel (R_2 + R_3) = \frac{R_1(R_2 + R_3)}{R_1 + R_2 + R_3}$$

In Fig. 2.16(b), two resistors are first connected in parallel, then connected in series with another resistor. The equivalent resistor $R_{T2}$ can be found similarly as

$$R_{T2} = R_4 + (R_5 \parallel R_6) = R_4 + \frac{R_5 R_6}{R_5 + R_6}$$

## 2.5 Analysis of electrical circuits: Nodal analysis

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 *nodal analysis* or *node-voltage method.* A *node* is a point in the circuit where more than two elements are connected together. For example, d in Fig.2.16(b) is a node, but c in Fig. 2.16(a) is not.

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

1. Select a common node with the most possible branches so that all other node voltages are defined with respect to this node. Call this node the *ground node.*
2. Define the voltage difference between all other nodes and the ground node as the unknown node voltages.
3. Write down the KCL at each node, expressing the branch currents in terms of node voltages and sources. (Not to get confused, write the branch currents always as leaving the node except when there are current sources.) If there is a voltage source between two nodes, write down the KVL between those two nodes.
4. Solve the equations obtained in step 3 simultaneously.
5. Find all branch currents and voltages in terms of node voltages.

**Example 4**

Consider the circuit in Fig. 2.17. Let us analyze this circuit to find all element

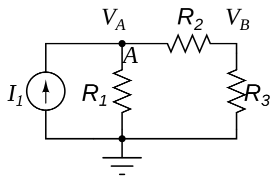



Figure 2.17: Example 4 for nodal analysis

voltages and currents using nodal analysis procedure:

1. Assign the ground symbol to the bottom node.

[§]The other method of circuit analysis is called *mesh analysis.* In this method, first, the currents around the loops in the circuit are defined as mesh currents. 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. Nodal analysis, on the other hand, yields algebraically simpler equations in most electronic circuits. So, mesh analysis is not recommended.

2. We have a single node, $V_A$, in this example. Define the node voltage $V_A$ as the voltage difference between node A and the ground node. $V_B$ is not a node since only two branches join.

3. Write KCL at node A (currents flowing in=currents flowing out) while writing the currents in terms of the node voltage $V_A$:

   Current through $R_1$ (flowing out) + Current through $R_2 + R_3$ (flowing out)-Current source current (flowing into the node) =0

$$\frac{V_A}{R_1} + \frac{V_A}{R_2 + R_3} - I_1 = 0$$

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

$$V_A = \frac{R_1(R_2 + R_3)}{R_1 + R_2 + R_3} I_1$$

5. Determine the branch currents in terms of $V_A$. The current flowing out through $R_1$ is $V_A/R_1$. The current flowing out through $R_2$ and $R_3$ is $V_A/(R_2 + R_3)$. The voltage across $R_3$ can be found from the voltage divider relation of Eq. 2.30:

$$V_B = V_A \frac{R_3}{R_2 + R_3} = \frac{R_1 R_3}{R_1 + R_2 + R_3} I_1$$

Hence, all currents and voltages in the circuit are determined.

As an alternate method, we can use the current divider equation of Eq. 2.35 to find the current in the resistor $R_1$ and multiply it with $R_1$ to find the voltage $V_A$:

$$V_A = \left( \frac{R_2 + R_3}{R_1 + R_2 + R_3} I_1 \right) R_1 = \frac{R_1(R_2 + R_3)}{R_1 + R_2 + R_3} I_1$$

To find the voltage $V_B$, we first find the current flowing in $R_2$ and $R_3$ using the current divider equation and multiply with $R_3$.

$$V_B = \left( \frac{R_1}{R_1 + R_2 + R_3} I_1 \right) R_3 = \frac{R_1 R_3}{R_1 + R_2 + R_3} I_1$$

---

**Example 5**

Consider the circuit of Fig. 2.18 containing three sources and five resistors. We apply the nodal analysis procedure:

1. Assign the ground symbol to the bottom node.

2. We have three nodes in this example. We assign $V_a$, $V_b$, and $V_c$ as node voltages.

3. We note that $V_a$ is already known as $V_a = 8$V. Therefore, we need to write only two equations. Write KCL at nodes b and c as

$$\frac{V_b - 8}{3\text{K}} + \frac{V_b}{5\text{K}} + \frac{V_b - V_c}{1\text{K}} - 6 \text{ mA} = 0$$
$$\frac{V_c - 8}{2\text{K}} + \frac{V_c}{4\text{K}} + \frac{V_c - V_b}{1\text{K}} + 10 \text{ mA} = 0$$

   We note that (V, k$\Omega$, and mA) is a consistent unit set.

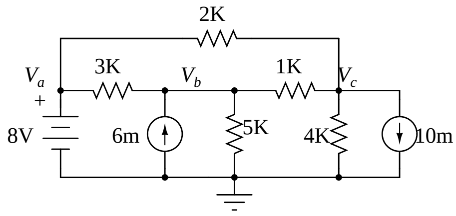



Figure 2.18: Example 5 for nodal analysis

4. Solve the two equations simultaneously to find $V_b$=550/101=5.4 V and $V_c = -32/101$=0.32 V. Note that it does not make sense to provide more than two or three digits of accuracy as answers when the accuracy of resistors is only 5 or 10%. Refer to Appendix at page 309 on significant figures.

5. Any branch current can be easily found since the node voltages are determined. For example, the current in the 1K resistor from left to right is

$$\frac{V_b - V_c}{1K} = \frac{582}{101} = 5.8 \text{ mA}$$

---

**Example 6**

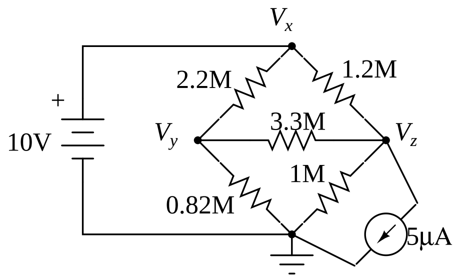



Figure 2.19: Example 6 for nodal analysis

1. Assign the ground symbol to the bottom node.

2. We assign $V_x$, $V_y$, and $V_z$ as node voltages.

3. $V_x$ is already known as $V_x = 10$V. We need to write only two equations. Write KCL at node y and z as

$$\frac{V_y - 10}{2.2\text{M}} + \frac{V_y - V_z}{3.3\text{M}} + \frac{V_y}{0.82\text{M}} = 0$$
$$\frac{V_z - 10}{1.2\text{M}} + \frac{V_z - V_y}{3.3\text{M}} + \frac{V_z}{1\text{M}} + 5\mu\text{A} = 0$$

We note that (V, MΩ, and $\mu$A) is also a consistent unit set.

4. Solve the two equations simultaneously to find $V_y$=2.6 V and $V_z$=1.9 V.

5. Since the node voltages are known, any branch current can be easily found. For example, the current in the 3.3 M resistor from left to right is

$$\frac{V_y - V_z}{3.3\text{M}} = 0.20\ \mu\text{A}$$

Note that we *cannot* use the current divider or voltage divider equation to solve this circuit.

---

## 2.6 Capacitors

Capacitors are built from two conductor plates separated by a thin insulator, as shown in Fig. 2.20. When a voltage of $V$ is applied across the plates, electrical charges $+Q$ and $-Q$ accumulates at the plates. The charges remain there even after the voltage is removed. Hence they act like charge storage devices. In this respect, they resemble voltage sources. However, they cannot supply constant voltage for a long time since they have a finite reservoir of charge. The ratio

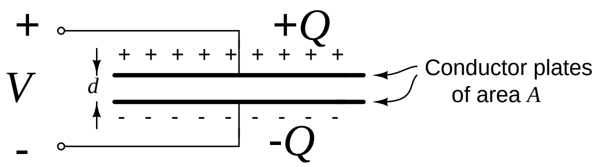



Figure 2.20: Structure of a capacitor built with two conducting plates separated by an insulator.

of the charge, $Q$, stored in a capacitor to the voltage, $V$, applied across it is constant

$$\boxed{C = \frac{Q}{V}} \tag{2.38}$$

where $C$ is the *capacitance* of the capacitor, and is measured in farads (F), named after English physicist Michael Faraday (1791–1867). Hence, the charge stored in a capacitance is proportional to the voltage across it (in contrast to a resistor, where the current is proportional to the voltage across it).

The capacitance of a capacitor built from two parallel conducting plates of area $A$ and separated by a distance $d$ is given by

$$\boxed{C = \frac{\epsilon_o \epsilon_r A}{d}} \tag{2.39}$$

where $\epsilon_o = 8.85 \times 10^{-12}$ F/m is the *permittivity* of free space and $\epsilon_r$ is a unitless quantity showing the *relative permittivity* of the insulator used between the plates. Table 2.6 lists the relative permittivities of some materials. The symbols of non-polarized, polarized, and variable capacitors are depicted in Fig. 2.21.

| Material | $\epsilon_r$, relative permittivity |
|---|---|
| Air | 1.00 |
| Teflon | 2.1 |
| Mylar | 3.1 |
| Paper | 3.85 |
| Pyrex glass | 4.7 |
| Silicon | 11.7 |
| Barium strontium titanate | 500 |

Table 2.3: Relative permittivity of some materials at 20°C .

**Example 7**

Find the capacitance of a capacitor built by a paper dielectric with conductor planes of 3 cm×10 cm. The paper has a thickness of 0.1 mm.

$$C = \frac{\epsilon_o \epsilon_r A}{d} = \frac{(8.85 \times 10^{-12})(3.85)(0.03 \times 0.1)}{0.1 \times 10^{-3}} = 1020\ pF. \tag{2.40}$$

---

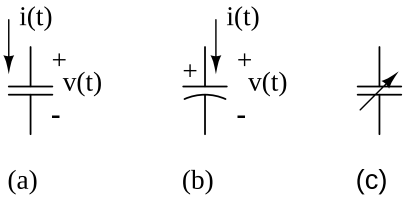



Figure 2.21: Symbols for capacitors: (a) Non-polarized, (b) polarized (c) variable

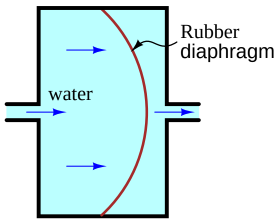



Figure 2.22: Capacitor analogy: Water tank with a rubber diaphragm.

In the water-flow analogy, the capacitor is analogous to a water tank with an elastic rubber diaphragm in the middle (see Fig. 2.22). When water is forced into the tank from one side, the diaphragm stretches and the pressure increases (analogous to increased voltage). As pressure is increased, it is possible to feed more water in the tank (analogous to increased charge). A larger capacitor is analogous to a tank with a larger cross-section, hence a larger capacity. It needs

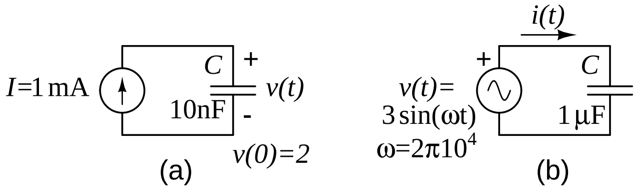



Figure 2.23: (a) A current source driving a capacitor, (b) an AC voltage source across a capacitor.

more water to increase the pressure. On the other hand, if the diaphragm is made stiffer, the capacitance of the tank is reduced.

If we let a current, $i(t)$, of arbitrary time waveform, pass through a capacitor, the amount of charge accumulated on the capacitor within a time interval, 0 to $t_1$, is given as

$$\Delta Q = \int_{t=0}^{t_1} i(t)dt \tag{2.41}$$

If $i(t)$ is a DC current, $I_{DC}$, then the charge accumulated on the capacitor is simply $\Delta Q = I_{DC}t_1$. For example, a DC current of 1 mA accumulates a charge of 1nC (nano=$10^{-9}$) on a capacitor in 1 $\mu$s. This charge generates 100 mV across the capacitor of 10 nF. More generally, this relation is expressed as

$$Q(t) = Q(0) + \int_{\xi=0}^{t} i(\xi)d\xi \tag{2.42}$$

where $\xi$ is the dummy variable of integration. $Q(0)$ refers to the initial charge on the capacitor at the time instant $t = 0$. Using Eq. 2.38, we can relate the voltage across a capacitor and the current through it

$$\boxed{v(t) = v(0) + \frac{1}{C}\int_{\xi=0}^{t} i(\xi)d\xi} \tag{2.43}$$

where $v(0) = Q(0)/C$ is the voltage of the capacitor at $t$=0. If we differentiate both sides of this equation with respect to $t$, the constant term, $v(0)$, disappears and we obtain

$$\frac{d}{dt}v(t) = \frac{1}{C}i(t) \quad \text{or} \quad \boxed{i(t) = C\frac{d}{dt}v(t)} \tag{2.44}$$

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

**Example 8**

Referring to Fig. 2.23(a) and using Eq. 2.43, the voltage of the capacitor driven by the current source is given by

$$v(t) = v(0) + \frac{1}{C}It = 2 + \frac{1}{10 \times 10^{-9}}10^{-3}t = 2 + 10^5 t \tag{2.45}$$

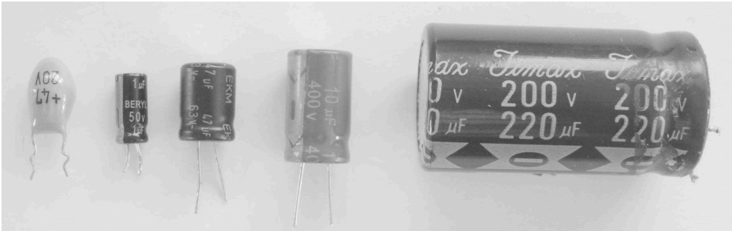



Figure 2.24: Leaded capacitors (left-to-right): A tantalum capacitor, 47 $\mu$F, 20 V; electrolytic capacitors 1 $\mu$F, 50 V; 47 $\mu$F, 63 V; 10 $\mu$F, 400 V; and 220 $\mu$F, 200 V

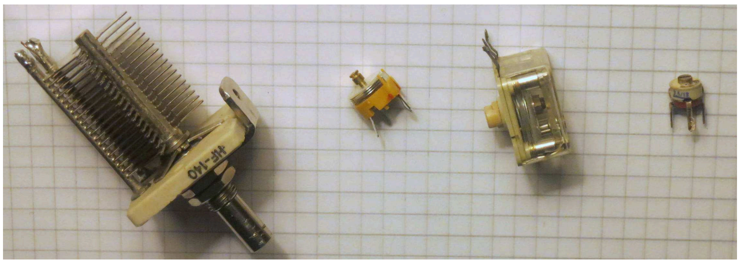


Figure 2.25: Variable capacitors of different types.

The current in the capacitor of Fig. 2.23(b) can be found using Eq. 2.44:

$$i(t) = C\frac{d}{dt}(3\sin(\omega t)) = 3(1 \cdot 10^{-6})(2\pi 10^{4})\cos(\omega t) = 0.19\cos(\omega t) \tag{2.46}$$

---

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 smaller than 0.5 $\mu$F ($\mu$=micro=$10^{-6}$)[¶] are of this type. However, as the capacitance values become large, it is less costly to use capacitors, which have polarity preferences, like electrolytic or tantalum capacitors. Fig. 2.24 is a photo of leaded tantalum and electrolytic capacitors with different voltage ratings. For these capacitors the voltage must always remain in the same polarity indicated on the capacitor.

Capacitors are typically available in the range 1 pF (pico=$10^{-12}$) to 10 mF (milli=$10^{-3}$) range. Capacitors also have voltage ratings. The voltage across it should be kept below the maximum voltage rating. The capacitors with 10% tolerance are available in the following standard values:

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

For values less than 1 nF, variable capacitors are also available (see Fig. 2.25).

Fig. 2.26 depicts surface-mount ceramic, electrolytic and variable capacitors of different values.

[¶]For other unit prefixes refer to Appendix B on page 310

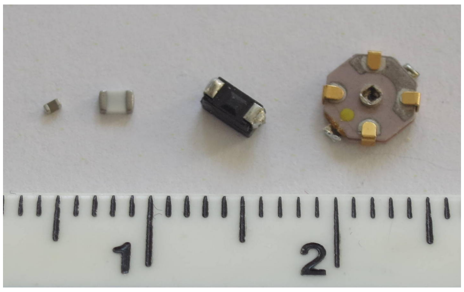



Figure 2.26: Surface-mount capacitors (left-to-right): 0402 package, 0804 package, 1206 package, variable

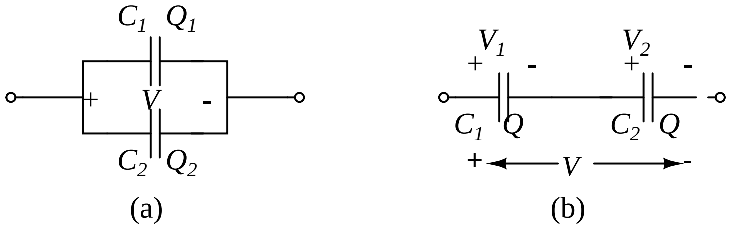



Figure 2.27: Two capacitors (a) in parallel, (b) in series

♦ TRC-11 has 29 unpolarized (10 pF to 220 nF), and nine polarized (1 $\mu$F to 220 $\mu$F).

### 2.6.1 Capacitors in parallel

When two capacitors are connected in parallel, as shown in Fig. 2.27(a), they have the same voltage, $V$, across them. If $Q_1$ and $Q_2$ are the charges accumulated on these two capacitors, then the total charge is the sum of these charges. Using Eq. 2.38 we find

$$Q = Q_1 + Q_2 = C_1 V + C_2 V = (C_1 + C_2)V \tag{2.47}$$

When capacitors are connected in parallel, the total capacitance, $C$, is given by

$$\boxed{C = C_1 + C_2} \tag{2.48}$$

In the water tank analogy, obviously, the tank capacities are added when two tanks are connected in parallel as shown in Fig. 2.28(a).

### 2.6.2 Capacitors in series

When two capacitors are connected in series, as shown in Fig. 2.27(b), they have to have the same charge, $Q$, stored in them. Otherwise, charge neutrality is not satisfied. From Eq. 2.38, this charge corresponds to voltages $V_1 = Q/C_1$ and $V_2 = Q/C_2$. Hence, the total voltage is

$$V = V_1 + V_2 = \frac{Q}{C_1} + \frac{Q}{C_2} = Q\left(\frac{1}{C_1} + \frac{1}{C_2}\right) \tag{2.49}$$

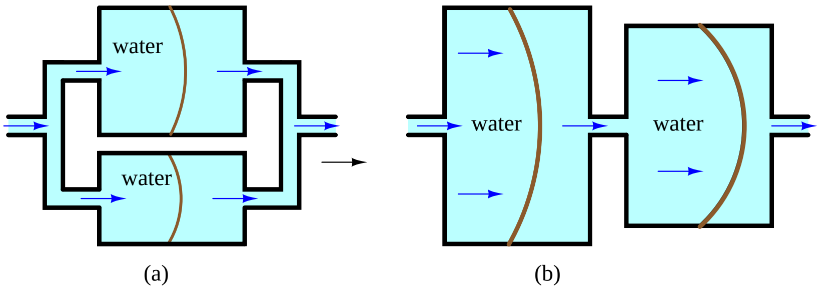



Figure 2.28: Water tanks with the diaphragms (a) in parallel, (b) in series

When two capacitors are connected in series, the total capacitance, $C$, is

$$\boxed{C = \left(\frac{1}{C_1} + \frac{1}{C_2}\right)^{-1}} \quad \text{or} \quad \boxed{C = \frac{C_1 C_2}{C_1 + C_2}} \tag{2.50}$$

We note that the total capacitance is less than either one of them.

Series connected capacitors are frequently used as an *AC voltage divider*. Since capacitors do not consume power, such dividers are preferable to resistive voltage dividers of p. 34. Referring to Fig. 2.27(b) the voltage across $C_1$ can be written as

$$\boxed{V_1 = \frac{C_2}{C_1 + C_2} V} \tag{2.51}$$

Note the difference between this equation and Eq. 2.30.

In the water-flow analogy, the series connection (see Fig. 2.28(b)) of two tanks causes tank capacity to drop since both diaphragms need to be pushed (equivalent to a stiffer diaphragm) to get water into the first tank.

### 2.6.3 Energy stored in a capacitor

Unlike resistors, the capacitors do not dissipate energy. They can only store energy. The energy, $E$, stored in a capacitor can be written as

$$\boxed{E = \frac{1}{2} C V^2} \tag{2.52}$$

For example, a 1000$\mu$F capacitor with a voltage of 24 V stores energy of 0.29 J.

## 2.7 *RC* Circuits

When a resistor, $R$, is connected to a charged capacitor, $C$, in parallel, as in Fig. 2.29(a), the circuit voltages become a function of time. Assume that initially capacitor is charged to $V_0$ volts (it has $Q = CV_0$ coulombs stored charge). At $t$=0 we connect the resistor, $R$. Using KCL and Ohm's law, we write‖

$$i(t) = C\frac{dv(t)}{dt} = -\frac{v(t)}{R} \tag{2.53}$$

‖Since the current $i(t)$ is leaving the resistor in the opposite direction to the voltage $v(t)$, we must use a negative sign for $v(t)/R$.

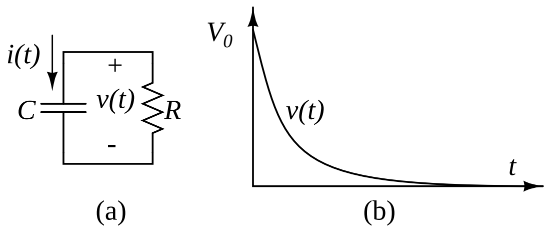



Figure 2.29: (a) RC circuit, (b) voltage as a function of time

or

$$\frac{dv(t)}{dt} + \frac{v(t)}{RC} = 0 \tag{2.54}$$

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

$$v(t) = V_0 e^{-t/RC} \quad \text{for} \quad t \geq 0. \tag{2.55}$$

$v(t)$ is plotted in Fig. 2.29(b). The above expression tells us 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 $\tau = RC$. $\tau$ is called *time constant* and has units of time (1Ω× 1F=1sec). The current flowing in the capacitor is

$$i(t) = C\frac{dv(t)}{dt} = C\frac{d}{dt}V_0 e^{-t/RC} = -\frac{V_0}{R}e^{-t/RC} \quad \text{for} \quad t \geq 0. \tag{2.56}$$

The value of the current is found negative, indicating that the current flows in the direction opposite to the one shown in the figure.

The **voltage of a capacitor** is always a **continuous function of time** (A sudden jump in capacitor voltage is not possible. Possible only mathematically, when an impulse current of infinite amplitude and infinitesimally short duration is applied.) On the other hand, the current of a capacitor *can* be a discontinuous function of time.

Note that we used a *sign convention* while writing Eq. 2.53: *The sign of the voltage on an element must be chosen positive if the positive terminal is the one where the current enters the element.* In this circuit, the current is chosen in the direction entering to $C$ at its positive terminal (top) (as in Fig. 2.21), thus the sign of the capacitance equation of Eq. 2.53 must be positive. On the other hand, the current leaves the positive terminal for the resistor (unlike the current in Fig. 2.9); hence we must use a negative sign for Ohm's law.

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. We must carefully stick to the above convention when writing down the equations of the elements or KVL and KCL equations. Otherwise, our results are not correct.

The magnitude of the current in the above circuit is at its maximum, $V_0/R$, initially, and decreases towards zero as time passes. This is expected as the voltage across the capacitor similarly decreases.

**Water flow analogy of $RC$ circuit**

Consider the water flow analogy of $RC$ circuit shown in Fig. 2.30. In the be-

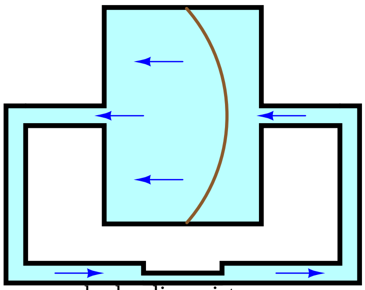



Figure 2.30: Water flow analogy of RC circuit: water tank feeding a hydraulic resistor

ginning, the diaphragm is highly stretched, and the pressure is the greatest. At that time, the water flow will be the fastest. As the water flows, the diaphragm relaxes and the pressure reduces. The flow rate approaches zero when the diaphragm relaxes fully. The energy stored in the stretched diaphragm is dissipated in the hydraulic resistor.

## 2.8 Analysis of first-order $RC$ circuits

RC circuits containing just one capacitor are called *first-order circuits*. The problem can be expressed as a first-order differential equation as in Eq. 2.54 of the previous section. The solution of this differential equation for any voltage or current variable in the circuit is always an exponential of the form given in Eq. 2.55 or 2.56. Because of its simplicity, the solution for any voltage or current can be written without writing the differential equation. For the solution of a first-order $RC$ circuit, we can use the following procedure:

1. Kill the sources: Place a short-circuit for the voltage sources, and remove (or open-circuit) the current sources. Find the equivalent resistance, $R_{eq}$, across the capacitor.
2. Write the time constant as $\tau = R_{eq}C$
3. Using KVL, KCL or nodal analysis, find the initial value, $v_i$ or $i_i$ (voltage or current), of the desired quantity, by substituting a voltage source with a value equal to the initial voltage of the capacitor.
4. Using nodal analysis, find the final value, $v_f$ or $i_f$, of the desired quantity by substituting an open circuit for the capacitor.

5. Write the solution for the desired voltage or current variable as

$$v(t) = v_f + (v_i - v_f)e^{-t/\tau} \quad \text{or} \quad i(t) = i_f + (i_i - i_f)e^{-t/\tau} \tag{2.57}$$

Note that when we substitute $t$=0, we get $v(0) = v_i$ and $i(0) = i_i$, consistent with step 3. Likewise, as $t \to \infty$, $v(t)$ approaches to $v_f$, and $i(t)$ approaches to $i_f$, consistent with step 4.

If a circuit contains two or more independent capacitors, the circuit is no longer a first-order circuit. Obtaining the time-domain solution of such circuits is more difficult, and we do not deal with such circuits. A circuit simulator can be used for that purpose. See page 300 for a tutorial on time-domain solutions of circuits using LTSpice.

**Example 9**

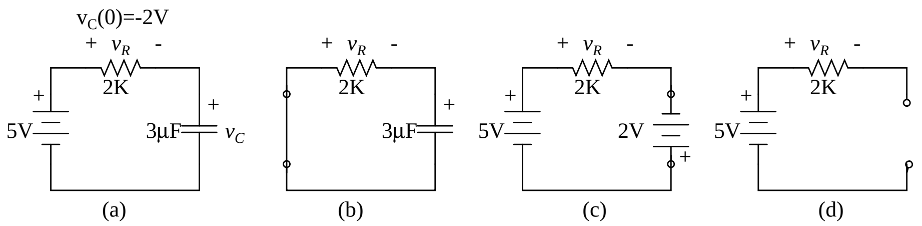



Figure 2.31: Example 9 for first-order RC circuit analysis

Consider the simple circuit of Fig. 2.31(a) with a single capacitor. The initial value of the capacitor voltage is given at $t = 0$. Suppose we would like to determine the voltage across the resistor, $v_R(t)$ in the polarity shown. Apply the procedure:

1. Kill the voltage source: Place a short-circuit in its place as depicted in Fig. 2.31(b). The equivalent resistance across the capacitor is $R_{eq}$=2K
2. Write the time constant as $\tau$ =(2 k$\Omega$)(3 $\mu$F)=6 ms
3. Substitute a voltage source with a value equal to the initial voltage of the capacitor as in Fig. 2.31(c). From KVL we find $v_{Ri}$=7 V.
4. Substitute an open circuit for the capacitor as in Fig. 2.31(d). Since there is no current in 2 k$\Omega$ resistor, $v_{Rf}$=0
5. Write the solution for the desired voltage as

$$v_R(t) = 0 + (7 - 0)e^{-t/6\ m} = 7e^{-t/6\ m}$$

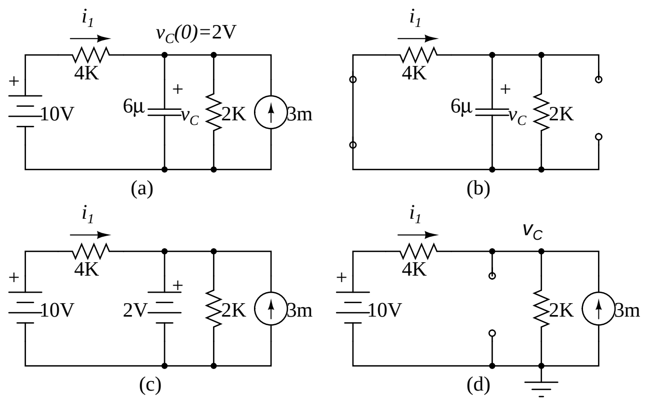



Figure 2.32: Example 10 for first-order RC circuit analysis

**Example 10**

Consider the circuit of Fig. 2.32(a) with a single capacitor. The initial value of the capacitor voltage is given at $t = 0$. We would like to determine the current through the 4 kΩ resistor, $i_1(t)$ in the direction shown with two significant figures. Apply the procedure:

1. Kill the sources: Place a short-circuit for the voltage source and an open-circuit for the current source as in Fig. 2.32(b). The equivalent resistance across the capacitor is 2 kΩ in parallel with 4 kΩ. From Eq. 2.34: $R_{eq}$=(2K)‖(4K)=(2K 4K)/(2K+4K)=1.3 kΩ

2. Write the time constant as $\tau$ =(1.3 kΩ)(6 $\mu$F)=8.0 ms

3. Substitute a voltage source with a value equal to the initial voltage of the capacitor as in Fig. 2.32(c). From KVL and Ohm's law we find $i_{1i}$=(10−2)/4K=2.0 mA.

4. Substitute an open circuit for the capacitor as in Fig. 2.32(d). Writing the node equation at $v_C$:

$$\frac{v_C - 10}{4\text{K}} + \frac{v_C}{2\text{K}} - 3 \text{ mA} = 0$$

   We find $v_C$=22/3=7.3 V. Hence $i_{1f}$=(10-22/3)/4K=2/3=0.67 mA.

5. Write the solution for the desired current as

$$i_1(t) = \frac{2}{3} + (2 - \frac{2}{3})e^{-t/8m} = 0.67 + 1.3e^{-t/8m} \text{ mA}$$

A MATLAB code to plot $i_1(t)$ is given below. The corresponding graph is shown in Fig. 2.33 with the tangent to the curve drawn at $t$=0. Note that the tangent line intersects the final value line $i_{1f}$ =0.67 mA at $t = \tau$=8 ms.

```
% MATLAB code to draw i1(t)
clear all % clear all variables in MATLAB
```

```
t=0:0.01:40; % t in milliseconds, t is a vector
i1=2/3+4/3*exp(-t/8); % MATLAB performs an array operation
plot(t,i1,'LineWidth',2) % plot with a linewidth of 2
grid on % to plot the grid lines
xlabel('t (ms)') % to place the x-label on the plot
ylabel('i_1 (mA)') % to place the y-label
title('Example 5') % to place a title on the graph
hold on
axis([0 40 0 2]); % define the axes limits
y=-1/6*t+2; %calculate the tangent to curve at t=0
plot(t,y,'--') % draw the tangent with a dashed line at t=0
y1=0*t+2/3;
plot(t,y1,'--') % draw the asymptote at infinity.
legend(['i_1(t)'],['Tangent at t=0'],['Asymptote at infinity'])
```

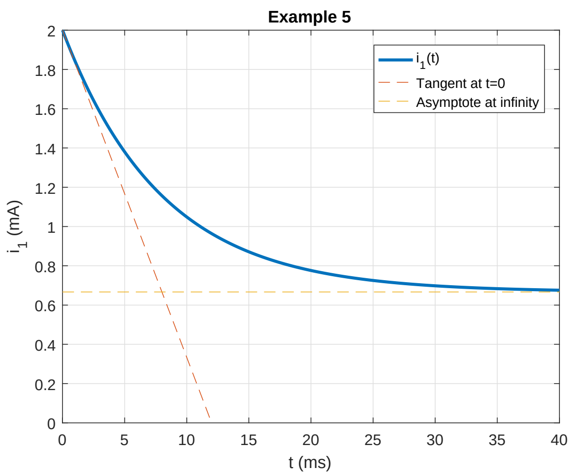



Figure 2.33: Example 10: Plot of $i_1(t)$.

---

**Example 11**

Consider the circuit of Fig. 2.34(a) with a single capacitor. The initial value of the capacitor voltage is given at $t = 0$. We would like to determine the current through the 4 kΩ resistor, $i_2(t)$, in the direction shown. Apply the procedure:

1. Kill the sources: Place a short-circuit for the voltage source and an open-circuit for the current source as in Fig. 2.34(b). The equivalent resistance across the capacitor is:

$$R_{eq} = (2\text{K}) \parallel (3\text{K}) + (4\text{K}) \parallel (1\text{K}) = \frac{2\text{K }3\text{K}}{2\text{K} + 3\text{K}} + \frac{4\text{K }1\text{K}}{4\text{K} + 1\text{K}} = 2\text{K}$$

2. Write the time constant as $\tau$ =(2 kΩ)(1 $\mu$F)=2 ms

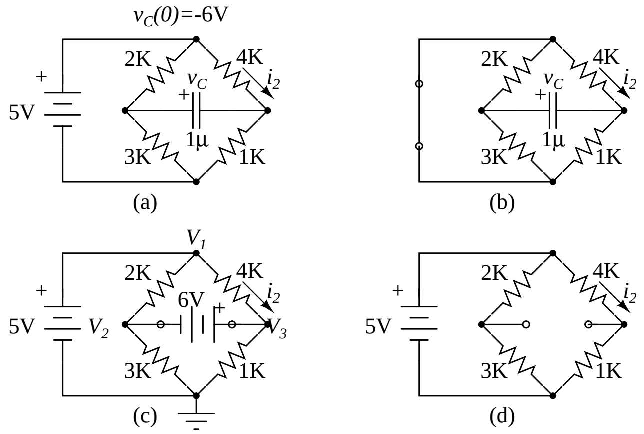



Figure 2.34: Example 11 for first-order RC circuit analysis

3. Substitute a voltage source with a value equal to the initial voltage of the capacitor as in Fig. 2.34(c). Use nodal analysis to solve the circuit:
   - Assign the ground node to the bottom.
   - Assign nodes $V_1$, $V_2$, and $V_3$. $V_1$=5 V is already known. So, we need two equations:
   - $V_3$ is easily written in terms of $V_2$. Then, write KCL at one of the nodes:

$$\begin{array}{ll} V_3: & V_3 = V_2 + 6 \\ V_2: & \frac{V_2-5}{2K} + \frac{V_2}{3K} + \frac{V_3-5}{4K} + \frac{V_3}{1K} = 0 \end{array}$$

   - Solve the equations to find $V_2 = -9/5 = -1.8$ V and $V_3 = 21/5 = 4.2$ V
   - Desired quantity $i_{2i} = (V_1 - V_3)/4$K=1/5=0.2 mA
4. Substitute an open circuit for the capacitor as in Fig. 2.34(d). Find $i_2$: $i_{2f}$=5 V/5K=1 mA
5. Write the solution for the desired current as

$$i_1(t) = 1 + (\frac{1}{5} - 1)e^{-t/2m} = 1 - 0.8e^{-t/2m} \text{ mA}$$

---

## 2.9 Inductors

When a current flows through a wire, a magnetic flux is generated around the wire. Reciprocally, if a conductor is placed in a time-varying magnetic field, a voltage is induced in it. From the electrical circuits point of view, this phenomenon introduces the circuit element, the *inductor*. Inductors are magnetic energy storage elements. Inductor is characterized by its *inductance*, measured in *Henries* (H), named after American scientist Joseph Henry (1797–1878). 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 an inductor (Fig. 2.35(a)). The terminal relations of an inductor is given as

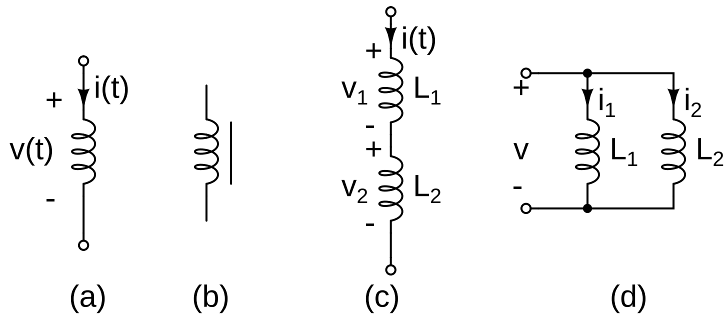



Figure 2.35: (a) Inductor symbol, (b) inductor symbol with a core, (c) inductors in series, and (d) inductors in parallel.

$$\boxed{v(t) = L\frac{di(t)}{dt}} \quad \text{or} \quad \boxed{i(t) = i(0) + \frac{1}{L}\int_0^t v(\xi)d\xi} \tag{2.58}$$

where $L$ is the inductance in Henries, $i(t)$ and $v(t)$ are current through, and voltage across the inductor. $i(0)$ is the current of the inductor at $t$=0.

We note that voltage is proportional to the time derivative of current in an inductor. If $i(t)$ is a DC current, its derivative is zero, and hence the voltage induced across the inductor is zero. Alternatively, if we apply a DC voltage across an inductor, the current increases linearly (we can only do this for a short time; otherwise, the current through the inductor can be very large).

> The **current of an inductor** is always a **continuous function of time**. Except when a short-pulse voltage of infinite magnitude is applied, the inductor current cannot have a sudden jump. On the other hand, the inductor voltage *can* be a discontinuous function of time.

Inductors are available in the range 1 nH (nano=$10^{-9}$) to 1H. 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 a higher permeability compared to air. This material is referred to as *core*, and such inductors are symbolized by a bar next to the inductor symbol, as shown in Fig. 2.35(b). Fig. 2.36 and 2.37 depict photos of different types of inductors.

- ♦ TRC-11 has three inductors. They are used to form tuned circuits in conjunction with capacitors.

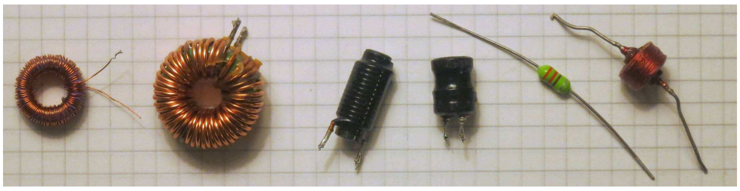


Figure 2.36: Photo of different leaded inductors.

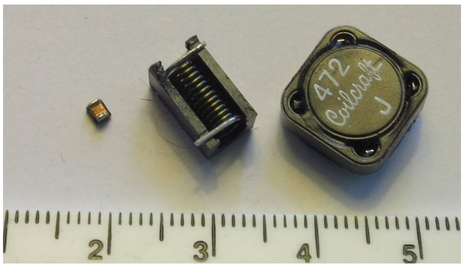



Figure 2.37: Various surface-mount inductors

### 2.9.1 Inductors in series and parallel

The series and parallel combination of inductors are similar to resistor combinations, as can be understood from the terminal relation above. For series-connected inductors, as in Fig. 2.35(c), the currents are the same, the voltages add up. Therefore, the total inductance for $n$ inductors in series is

$$\boxed{L = L_1 + L_2 + \cdots + L_n} \tag{2.59}$$

whereas for parallel-connected inductors (Fig. 2.35(d)), voltages are the same and the currents add up:

$$\boxed{L = \left(\frac{1}{L_1} + \frac{1}{L_2} + \cdots + \frac{1}{L_n}\right)^{-1}} \tag{2.60}$$

### 2.9.2 Energy stored in an inductor

Like capacitors, the inductors do not dissipate energy. They can only store the energy. The energy, $E$, stored in an inductor is determined by the current through the inductor

$$\boxed{E = \frac{1}{2}LI^2} \tag{2.61}$$

For example, a 500 $\mu$H inductor carrying 1 A stores energy of 0.3 mJ.

## 2.10 $RL$ circuits

Suppose a resistor, $R$, is connected in parallel with an inductor, $L$, as in Fig. 2.38(a). Initially, the inductor current is equal to $i_L(0) = I_i$. Since $R$


Figure 2.38: RL Circuit.

and $L$ are connected in parallel, they have the same terminal voltage:

$$v = Ri_R = L\frac{di_L}{dt} \tag{2.62}$$

and from KCL we have

$$i_L(t) = -i_R(t) \tag{2.63}$$

Therefore,

$$L\frac{di_L(t)}{dt} + Ri_L(t) = 0 \tag{2.64}$$

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

$$i_L(t) = I_i e^{-t/\tau} \tag{2.65}$$

where the time constant is $\tau = L/R$.

**Water-flow analogy of $RL$ circuit**

Consider the water-flow analogy of $RL$ circuit shown in Fig. 2.39. At the

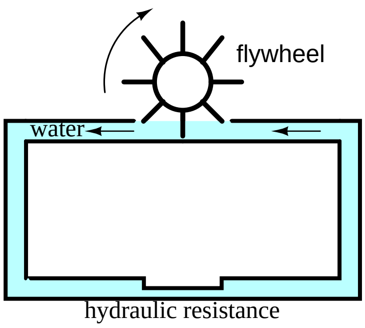



Figure 2.39: Water flow analogy of $RL$ circuit: flywheel feeding a hydraulic resistor

beginning, the flywheel with a moment of inertia is turning fast and the flow rate is the greatest. At that time, the pressure across the hydraulic resistor is the greatest. As the water flows, the flywheel slows down and the pressure reduces. The flow rate becomes zero, when the flywheel stops. The energy stored in the flywheel is dissipated in the hydraulic resistor.

## 2.11 Analysis of first-order $RL$ circuits

Like RC circuits, RL circuits containing just one inductor are also called *first-order circuits*. The problem can be expressed as a first-order differential equation. Just like the first-order RC circuits, the solution for any voltage or current can be written without writing the differential equation. We use the following procedure:

1. Kill the sources: Place a short-circuit for the voltage sources, and remove (open-circuit) the current sources. Find the equivalent resistance, $R_{eq}$, across the inductor.
2. Write the time constant as $\tau = L/R_{eq}$
3. Using KVL, KCL or nodal analysis, find the initial value, $v_i$ or $i_i$ (voltage or current), of the desired quantity, by substituting a current source with a value equal to the initial current of the inductor.
4. Using nodal analysis, find the final value, $v_f$ or $i_f$, of the desired quantity by substituting a short circuit for the inductor.
5. Write the solution for the desired voltage or current variable as

$$v(t) = v_f + (v_i - v_f)e^{-t/\tau} \quad \text{or} \quad i(t) = i_f + (i_i - i_f)e^{-t/\tau} \tag{2.66}$$

**Example 12**

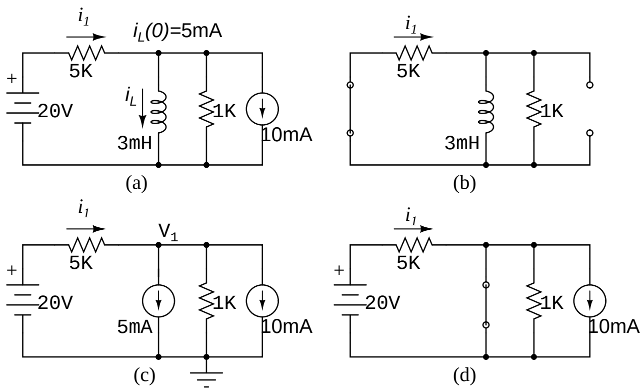



Figure 2.40: Example 12 for first-order $RL$ circuit analysis

Consider the circuit of Fig. 2.40(a) with a single inductor. The initial value of the inductor current is given at $t = 0$. We would like to determine the current through the 5 k$\Omega$ resistor, $i_1(t)$ in the direction shown. Apply the procedure:

1. Kill the sources: Place a short-circuit for the voltage source and an open-circuit for the current source as in Fig. 2.40(b). The equivalent resistance

across the inductor is 5 k in parallel with 1 k. From Eq. 2.34:

$$R_{eq} = (5\text{K})//(1\text{K}) = \frac{5\text{K} \cdot 1\text{K}}{5\text{K} + 1\text{K}} = 0.83\text{K}$$

2. Write the time constant as $\tau$ =3 mH/(0.83 k$\Omega$)=3.6 $\mu$s

3. Substitute a current source with a value equal to the initial current of the inductor as in Fig. 2.40(c). Find the voltage at node $V_1$, using nodal analysis:

$$\frac{20 - V_1}{5\text{K}} = 5\ mA + \frac{V_1}{1\text{K}} + 10\ mA$$

So, $V_1$=$-$55/6=$-$9.2 V. From KVL and Ohm's law we find $i_i(0) = i_{1i} = (20 - (-9.2))/5\text{K} = 5.8$ mA.

4. Substitute a short-circuit for the inductor as in Fig. 2.40(d). From KCL we find the current through the resistor at $i_1(\infty) = i_{1f}$: $i_{Lf}$=20/5K=4.0 mA.

5. Write the solution for the desired current as

$$i_1(t) = 4.0 + (5.8 - 4.0)e^{-t/3.6\mu} = 4.0 + 1.8e^{-t/3.6\mu}$$

$i_1(t)$ is plotted in Fig. 2.41 where the tangent to the curve at $t$=0 is shown. It intersects $i_1$=4.0 mA line at $t = \tau = 3.6$ ms.

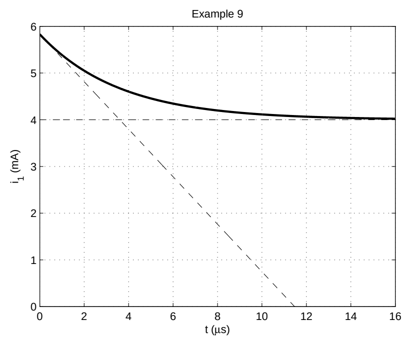



Figure 2.41: $i_1(t)$ of Example 12

---

**Example 13**

Consider the circuit of Fig. 2.42(a) with a single inductor. We would like to find the voltage across the inductor. The initial value of the inductor current is *not* given. Instead, we know that the voltage source $V_S$ is 4 V for $t < 0$. So, the inductor current must have reached its final value for $t < 0$. First, apply the part of the procedure to find $i_L(0^-)$ (it is the final value of the inductor current when $V_S$=4 V). Since the inductor current must be continuous, we must have $i_L(0^-) = i_L(0^+)$.

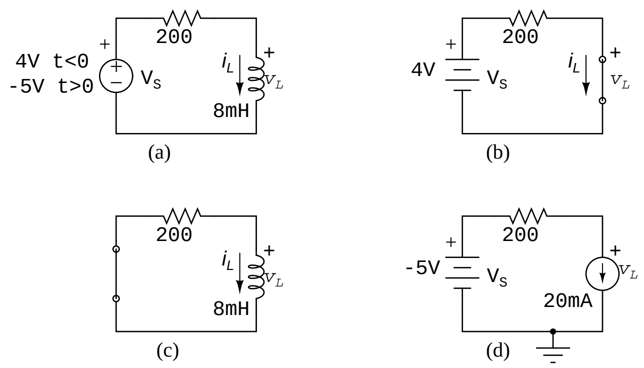



Figure 2.42: Example 13 for first-order RL circuit analysis

1. Substitute a short-circuit for the inductor as in Fig. 2.42(b) while $V_S = 4$V. We find $i_L(0^-)$=4/0.2K=20 mA. We also know that $v_L(t)$=0 for $t < 0$.

Hence, the initial value of the inductor current at $t = 0$ is $i_L(0^+)$=20 mA. Even though we are not asked about $i_L(t)$, we need to find it. Because it is *the* continuous variable while the input voltage is changing. Now, apply the full procedure to find $v_L(t)$ for $t > 0$ when $V_S$=−5 V:

1. Kill the source: Place a short-circuit for the voltage source as in Fig. 2.42(c). The equivalent resistance across the inductor is 200 Ω.

2. Write the time constant as $\tau$ =8 mH/(200 Ω)=40 $\mu$s

3. Substitute a current source with a value equal to the initial current of the inductor as in Fig. 2.42(d). Since the current through the resistor is 20 mA, we can find the voltage $v_L$ easily: $v_L$=−5−(0.2K)(20 mA)=−9 V. This is the initial value of $v_L$.

4. Substitute a short-circuit for the inductor. Hence, the final value of inductor voltage $v_L(\infty)$=0.

5. Write the solution for the desired voltage as

$$v_L(t) = -9e^{-t/40\mu}$$

   $v_L(t)$ is plotted in Fig. 2.43. Note that the inductor voltage is discontinuous at $t$=0, while the inductor current *is* continuous.

---

## 2.12 Ideal Transformer

Transformers are two or more magnetically *coupled* inductors. The circuit symbol of an ideal transformer with two windings (i.e., two inductors) is given in Fig. 2.44.

The windings in a transformer are referred to as *primary winding* and *secondary winding*. Transformers transform the voltage and current amplitudes

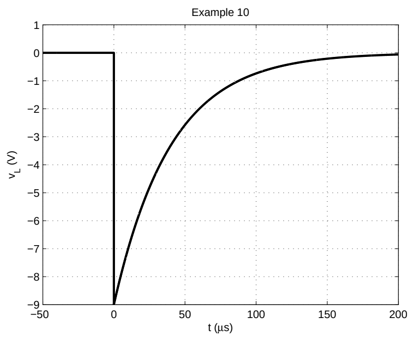



Figure 2.43: $v_L(t)$ of Example 13

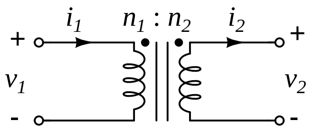



Figure 2.44: Transformer symbol showing voltage and current directions.

that appear 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* $n_2/n_1$. The relations in an ideal transformer are as follows:

$$\boxed{\frac{v_2}{v_1} = \frac{n_2}{n_1}} \quad \text{and} \quad \boxed{\frac{i_1}{i_2} = \frac{n_2}{n_1}} \tag{2.67}$$

**Water-flow analogy of transformer**

Fig. 2.45 depicts the water-flow analogy of a transformer. Two flywheels with different radii are connected to the same shaft and rotate at the same speed. The lower pipe with a larger cross-section has a higher flow rate than the upper pipe, while the pressure in the upper pipe is higher than the pressure in the lower pipe.

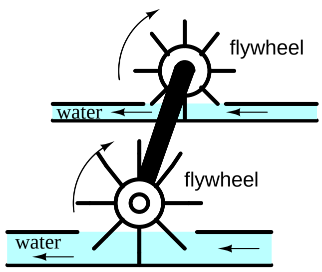



Figure 2.45: Water flow analogy of a transformer.

♦ TRC-11 uses two transformers operating at RF frequencies.

## 2.13 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.

### 2.13.1 Varistors

Varistors are nonlinear resistors made of ceramic-like materials like sintered zinc oxide or silicon carbide. The $I-V$ characteristics of a varistor are depicted in Fig. 2.46, together with its symbol. When the voltage across the varistor is

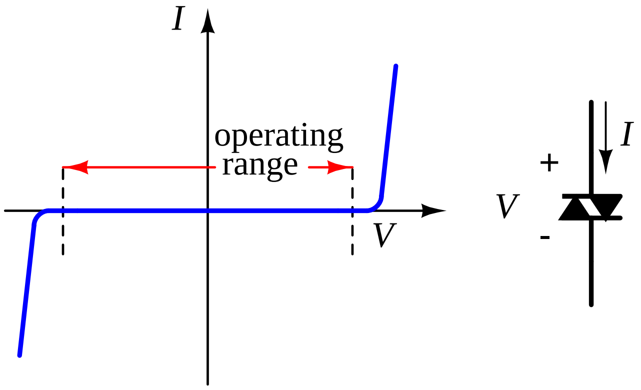



Figure 2.46: Varistor characteristics and symbol

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

### 2.13.2 Thermistors

A thermistor is a resistor whose resistance depends on the temperature. There are two kinds: Negative-temperature coefficient (NTC), resistance decreases as temperature rises; positive-temperature-coefficient (PTC), resistance increases as temperature rises.

#### PTC thermistor

We consider a PTC thermistor as a *resettable fuse*. PTC is in low-resistance state at room temperature, but its resistance increases abruptly with increasing temperature beyond a specified limit (reference temperature). The current through the PTC under normal operating conditions is sufficiently low. At this current level, the power dissipated by the PTC is low enough, such that the PTC temperature does not exceed the reference temperature. When a short-circuit or a high current condition occurs, the current through the PTC increases. The power dissipation in the PTC causes an increase in the temperature beyond the reference temperature, and PTC trips to high resistance state, hence limiting the current flowing in the circuit. To speed up the temperature rise, such PTC's are usually covered by a heat insulator. When the short-circuit or high-current condition is removed, PTC cools down and returns to its normal low-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.

#### NTC thermistors

NTC thermistors can be used as an *inrush current limiter* device in power supply circuits. Since the power supply capacitor is initially discharged, a very large current can flow in the rectifier diodes and this large transient current may destroy the diodes. If NTC thermistors are connected in series with the diodes, they present a high resistance at the initial turn-on, while they are cold. This prevents the large inrush current, saving the diodes. As currents flow through them, they heat up and become much lower resistance. Therefore, they do not dissipate a significant power in the steady-state. It is common to put a heat preserving cover around them to keep them hot and hence low resistance.

NTC thermistors are also used as temperature sensors in many applications. The measured resistance can be converted to temperature if the resistance versus temperature characteristics is known.

### 2.13.3 Circuit protection

An overvoltage protection circuit typically has the form shown in Fig. 2.47(a) or (b). The circuit shown in Fig. 2.47(a) operates as follows: $VR_1$ and $PTC_1$ are

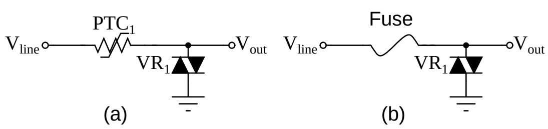



Figure 2.47: Over-voltage protection circuits: Using (a) a PTC and a varactor, (b) a fuse and a varactor

chosen such that, when there is no over-voltage, the voltage across $VR_1$ is in the normal range and the current through $PTC_1$ is less than $I_N$. In this case, $PTC_1$ exhibits a low resistance, and $VR_1$ exhibits a very high resistance. When an over-voltage occurs on the line voltage (for example, due to a flash of lightning in a thunderstorm), the voltage across $VR_1$ increases beyond its operating range. The current through $VR_1$ increases rapidly due to the nonlinear nature of the varistor resistance. This current passes through $PTC_1$ and warms the PTC up. When this current exceeds $I_S$, $PTC_1$ switches to high impedance mode isolating the line from the output. To speed up the warming and hence the response time, PTC's are usually placed in a thermally insulating jacket. When the overvoltage condition is over, PTC cools down and returns to its original low resistance state. This kind of protection circuits is always present in the line-voltage inputs of most modern power supplies and power adapters.

Note that a PTC placed in series with a circuit can also be used as an over-current protection circuit. If the current in the circuit exceeds the predetermined level, for example, due to an accidental short-circuit, PTC heats up increasing its resistance, limiting the current.

The circuit shown in Fig. 2.47(b) has a *fuse* instead of a PTC. A fuse is a metal wire placed in a glass tube with metal caps on both ends. Fuse metal melts when too much current flows through it, thereby interrupting the current. Glass allows a visual inspection of the fuse. Unlike PTC, when a fuse blows, it does not recover. It has to be replaced with a new fuse.

⧫ TRC-11 utilizes one PTC as a resettable fuse.

## 2.14 Electromechanical Switches

Switches are electrical components that can connect or disconnect paths. Most common switches are electromechanical switches that contain movable electrical contacts. In a manual electromechanical switch, an isolated knob helps operate the switch.

Switches are classified according to their different characteristics. The most common type of mechanical switches are

- Single-pole single-throw (SPST)
- Single-pole double-throw (SPDT)
- Double-pole single-throw (DPST)
- Double-pole double-throw (DPDT)

Their symbols are shown in Fig. 2.48, which also describes their electrical differences. In a double-pole or a multi-pole switch, two or more switches operate in tandem.

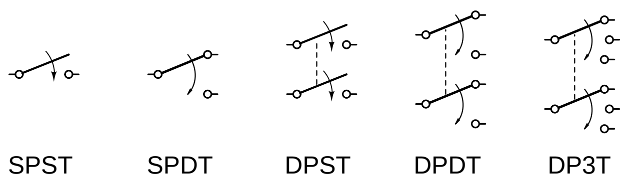



Figure 2.48: Symbols of different kind of electromechanical switches.

Switches are also classified according to their mechanical behavior.

- Toggle switch: Manually activated single- or double-throw switch with two stable states.

- Push-button switch: Manually activated single- or double-throw switch with one stable state. The other state of the switch is active only when a mechanical force is present, as in the case of doorbell switch or reset button switch of a computer.

- Slide switch: One or more throw switch with a manually activated slider.

The switches may also be activated electrically through an electromagnet. Such a switch is called *relay* (see Fig. 2.49), which is activated with a current through a solenoid. Most relays are non-latching type, in which a spring retracts the switch to its original position when the solenoid current is lost. Latching relays operate like toggle switches. A solenoid current toggles the state of the switches. Photos of some relays are depicted in Fig. 2.50.

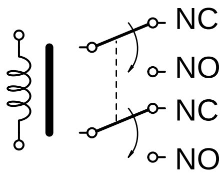



Figure 2.49: Symbol of a relay with a DPDT switch with normally-open (NO) and normally-closed (NC) contacts.

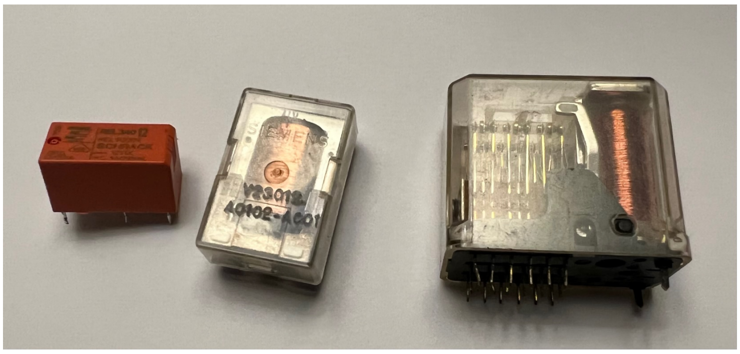


Figure 2.50: Different relays from left to right: SPST, DPDT, 6PDT.

⧫ TRC-11 utilizes one double-pole push-button switch and one non-latching SPDT relay.

## 2.15 Examples

### Example 14

Using the current divider formula, find the current $I_1$ in Fig. 2.51 with the reference direction shown.

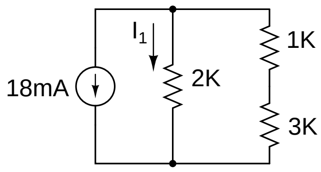



Figure 2.51: Circuit for Example 14.

#### Solution

Using the current divider formula of Eq. 2.35, we find

$$I_1 = -18\frac{1+3}{1+3+2} = -12 \text{ mA}$$

Note that the negative sign arises due to direction of current source versus the reference direction of $I_1$.

---

### Example 15

Using nodal analysis find the voltage $V_1$ in Fig. 2.52. Then determine the current $I_1$ in the reference direction shown.

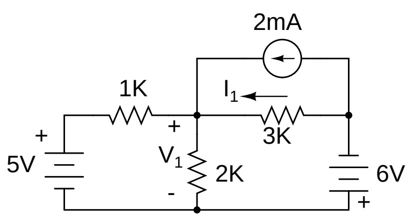



Figure 2.52: Circuit for Example 15.

#### Solution

We define the bottom wire as ground and write the node equation at $V_1$ as ({V, mA, kΩ} is a consistent unit set)

$$\frac{V_1 - 5}{1} + \frac{V_1}{2} + \frac{V_1 - (-6)}{3} - 2 = 0$$

Solving $V_1$

$$V_1\left(1+\frac{1}{2}+\frac{1}{3}\right)=5-\frac{6}{3}+2=5 \quad\Rightarrow\quad V_1=2.72 \text{ V}$$

Now $I_1$ can be found:

$$I_1=\frac{-6-V_1}{3}=\frac{-6-2.72}{3}=-2.91 \text{ mA}$$

---

## Example 16

In Fig. 2.53, find the value of the voltage source, $V$, such that $I_1$=0.

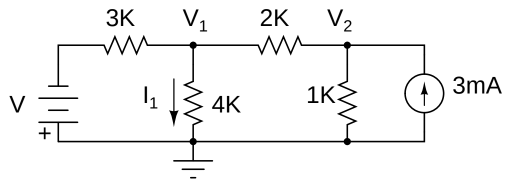



Figure 2.53: Circuit for Example 16.

### Solution

We define the bottom wire as ground and write the node equations at $V_1$ and $V_2$

$$\frac{V_1-(-V)}{3}+\frac{V_1}{4}+\frac{V_1-V_2}{2}=0$$
$$\frac{V_2-V_1}{2}+\frac{V_2}{1}-3=0$$

Rearranging the equations

$$V_1\left(\frac{1}{3}+\frac{1}{4}+\frac{1}{2}\right)-V_2\left(\frac{1}{2}\right)=-\frac{V}{3}$$

$$-V_1\left(\frac{1}{2}\right)+V_2\left(\frac{1}{2}+1\right)=3$$

Multiplying the first equation by 3 and adding to the second one, we eliminate $V_2$ to get

$$V_1\left(\frac{39}{12}-\frac{1}{2}\right)=-V+3$$

To make $I_1$, we must have $V_1$=0, hence

$$-V+3=0 \quad\Rightarrow\quad V=3 \text{ V}$$

---

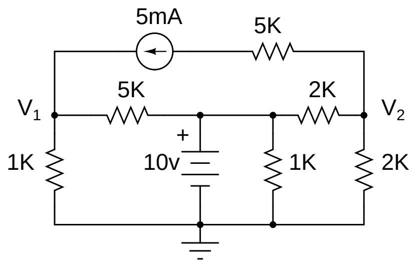



Figure 2.54: Circuit for Example 17.

## Example 17

Find $V_1 - V_2$ in the circuit given in Fig. 2.54.

### Solution

Apply nodal analysis method to find $V_1$ and $V_2$. KCL at $V_1$ node gives

$$\frac{V_1}{1K} + \frac{V_1 - 10}{5K} - 5 \text{ mA} = 0 \text{ or } V_1 = \frac{35}{6} = 5.8 \text{ V}$$

KCL at $V_2$ node gives

$$\frac{V_2}{2\text{K}} + \frac{V_2 - 10}{2\text{K}} + 5 \text{ mA} = 0 \text{ or } V_2 = 0$$

Therefore, $V_1 - V_2 = 5.8$ V.

---

## Example 18

Assuming that $v_C(0)$=2 V, find $i_1(t)$ in Fig. 2.55(a).

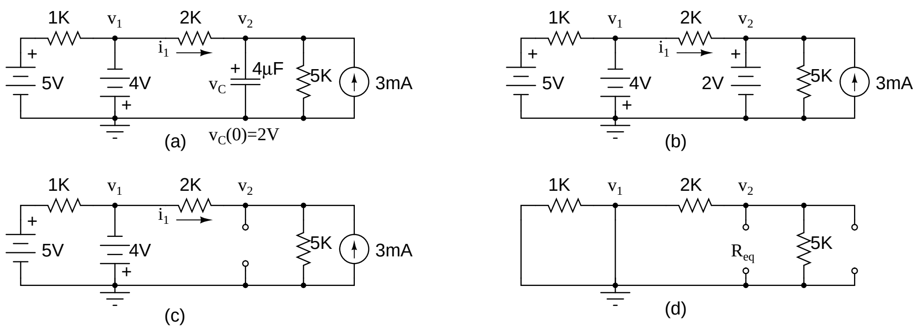



Figure 2.55: Circuit for Example 18.

**Solution**

For $t$=0 solution, we replace the capacitor with a voltage source of 2 V as in Fig. 2.55(b), and find $i_1(0)$

$$i_1(0) = \frac{v_1 - v_2}{2} = \frac{(-4) - 2}{2} = -3 \text{ mA}$$

For $t=\infty$ solution, we replace the capacitor with an open circuit as shown in Fig. 2.55(c) and apply nodal analysis to find $v_2(\infty)$

$$\frac{v_2 - v_1}{2} + \frac{v_2}{5} - 3 = \frac{v_2 - (-4)}{2} + \frac{v_2}{5} - 3 = 0 \;\;\Rightarrow\;\; v_2(\infty) = \frac{10}{7}$$

Therefore, the current $i_1(\infty)$ is found as

$$i_1(\infty) = \frac{v_1 - v_2}{2} = \frac{-4 - 10/7}{2} = -\frac{19}{7}$$

We find the equivalent resistance, $R_{eq}$, across the capacitor by killing all sources as depicted in Fig. 2.55(d)

$$R_{eq} = 5 \parallel 2 = \frac{5 \times 2}{5 + 2} = \frac{10}{7} \text{ K}\Omega$$

Hence the time constant, $\tau$, is $\tau$=(10/7)K$\Omega$ $\times$ (4 $\mu$F)=(40/7) ms. Now, we can write the solution for $i_1(t)$ as

$$i_1(t) = i_1(\infty) + (i_1(0) - i_1(\infty))e^{-t/\tau} = -\frac{19}{7} + \left(-3 + \frac{19}{7}\right) e^{-t/(40/7)} \text{ mA}$$

where $t$ is in ms. Note that 5 V voltage source and its 1 K series resistance has no effect on the value of $i_1(t)$.

---

## Example 19

Assuming that $i_L(0)$=20 mA, find $v_1(t)$ in Fig. 2.56(a).

**Solution**

For $t$=0 solution, we replace the inductor with a current source of 20 mA as in Fig. 2.56(b), and write the nodal equation at $v_A$ as

$$\frac{v_A - 10}{0.15} + 20 + \frac{v_A}{0.1} + 25 = 0$$

We find $v_A$=1.3 V. Therefore, $v_1(0)$=10−1.3=8.7 V. For $t=\infty$ solution, we replace the inductor with a short circuit as shown in Fig. 2.56(c) and find $v_2(\infty)$ easily as

$$v_1(\infty) = 10 - 0 = 10 \text{ V}$$

We find the equivalent resistance, $R_{eq}$, across the inductor by killing all sources as depicted in Fig. 2.56(d)

$$R_{eq} = 0.15 \parallel 0.1 = 0.056 \text{ K}\Omega$$

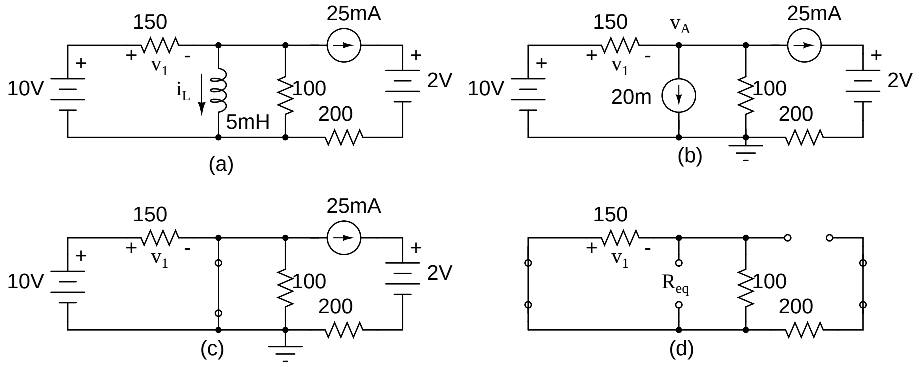



Figure 2.56: Circuit for Example 19.

Hence the time constant, $\tau$, is $\tau$=5 mH/0.056 K$\Omega$=89.3 $\mu$s. Now, we can write the solution for $v_1(t)$ as

$$v_1(t) = v_1(\infty)+(v_1(0)-v_1(\infty))e^{-t/\tau} = 10+(8.7-10)e^{-t/89.3} = 10-1.3e^{-t/89.3} \text{ V}$$

where $t$ is in $\mu$s.

---

## Example 20

In Fig. 2.57(a), find the capacitor voltage, $v_C(t)$, and current, $i_C(t)$, for $t > 0$. We have $v_C(0)$=7 V and the switch S is closed at 25 ms.

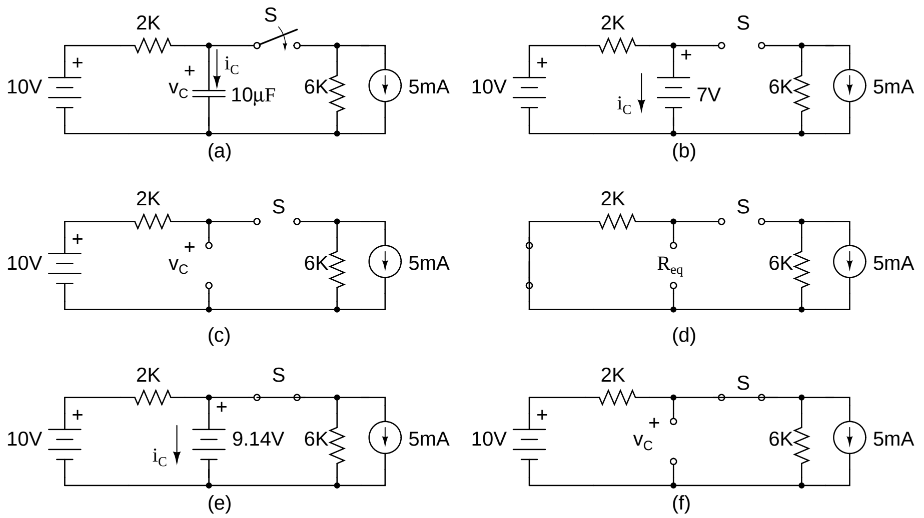



Figure 2.57: Circuit for Example 20.

**Solution**

We need the find the capacitor voltage $v_C$(25 ms) to act as the initial condition for the circuit when S is closed. We have switch S open for $0 < t < 25$ ms. For $t$=0 solution, we replace the capacitor with a voltage source of 7 V as in Fig. 2.57(b), and find

$$i_C(0) = \frac{10-7}{2} = 1.5 \text{ mA}$$

For $t$=$\infty$ solution (we assume that the switch S is still open!), we replace the capacitor with an open circuit as shown in Fig. 2.57(c) and find $v_C(\infty)$ easily as

$$v_C(\infty) = 10 \text{ V}$$

We find the equivalent resistance, $R_{eq1}$, across the capacitor by killing all sources as depicted in Fig. 2.57(d)

$$R_{eq1} = 2 \text{ K}\Omega$$

Hence the time constant, $\tau_1$, for this time duration is $\tau_1$=(2 KΩ)(10 $\mu$F)=20 ms. Now, we can write the solution for $v_C(t)$ and $i_C(t)$ during $0 < t < 25$ ms as

$$v_C(t) = v_C(\infty) + (v_C(0) - v_C(\infty))e^{-t/\tau_1} = 10 + (7-10)e^{-t/20} = 10 - 3e^{-t/20} \text{ V}$$

$$i_C(t) = i_C(\infty) + (i_C(0) - i_C(\infty))e^{-t/\tau_1} = 0 + (1.5-0)e^{-t/20} = 1.5e^{-t/20} \text{ mA}$$

where $0 < t < 25$ is in ms. Now, we can find the initial condition for the case S is closed.

$$v_C(25) = 10 - 3e^{-25/20} = 9.14 \text{ V}$$

The capacitor voltage will be preserved after the switch is closed. This is not the case for the capacitor current. We find the capacitor current just before the switch S is closed, $t$=25−, as

$$i_C(25-) = 1.5e^{-25/20} = 0.43 \text{ mA}$$

At the moment S is closed, we have the circuit as shown in Fig. 2.57(e). We find the capacitor current at this moment, $t$=25+, from Fig. 2.57(e) by writing the node equation as

$$\frac{9.14-10}{2} + i_C + \frac{9.14}{6} + 5 = 0 \;\;\Rightarrow\;\; i_C(25+) = -6.09 \text{ mA}$$

For $t$=$\infty$ solution while S is closed, we open circuit the capacitor as depicted in Fig. 2.57(f), and write the node equation to find $v_C(\infty)$ as

$$\frac{v_C - 10}{2} + \frac{v_C}{6} + 5 = 0 \;\;\Rightarrow\;\; v_C(\infty) = 0$$

The equivalent resistance, $R_{eq2}$ and the time constant, $\tau_2$, for $t > 25$ ms is found as

$$R_{eq2} = 2 \parallel 6 = 1.5 \text{ K}\Omega \;\text{ and }\; \tau_2 = (10\ \mu)(1.5 \text{ K}) = 15 \text{ ms}$$

Therefore, we can write the capacitor voltage and current as

$$v_C(t) = 0 + (9.14-0)e^{-(t-25)/\tau_2} = 9.14e^{-(t-25)/15}$$

$$i_C(t) = 0 + (-6.09-0)e^{-(t-25)/\tau_2} = -6.09e^{-(t-25)/15}$$

for $25 < t < \infty$ in ms. The capacitor voltage and current is plotted in Fig. 2.58. Note that the capacitor voltage is always a continuous function, while the capacitor current may be discontinuous.

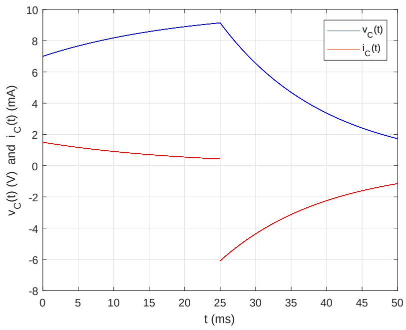



Figure 2.58: $v_C(t)$ and $i_C(t)$ for Example 20.

## Example 21

Suppose that the switch in Fig. 2.59 is open for $t$ <0, and closed at $t = 0$. Find $i_L$ and $v_1$ as a function of time for $-\infty < t < \infty$.

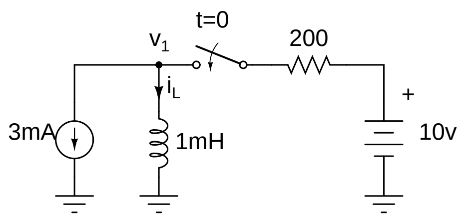



Figure 2.59: Circuit for Example 21.

### Solution

The circuit contains only one inductor; it is a first-order $RL$ network. Hence we can use the method given in Section 2.11.

1. For $t > 0$ the switch is closed. Killing the sources: The voltage source is short-circuited, the current is open-circuited. In this case, the resistance seen by the inductance is 200 Ω.

2. The time constant of the network is $\tau = L/R = 10^{-3}/(200)$=5 $\mu$s.

3. For $t < 0$, $i_L(t) = -3$ mA, $v_1(t) = 0$V. At $t = 0$, the switch is closed, and $i_L$ should be continuous: $i_L(0^+) = i_L(0^-) = -3$ mA. At $t = 0^+$, KCL at $v_1$ node implies that there is no current in 200 Ω resistor. Therefore, $v_1(0^+)$=10 V.

4. Short-circuit the inductor. We find $v_1(\infty) = 0$ and $i_L(\infty) = \frac{10}{200} - 3 =$ 47 mA

5. We write the solutions for $t > 0$ as

$$v_1(t) = 0 + (10 - 0)e^{-t/5\mu} \quad \text{and} \quad i_L(t) = 47 + (-3 - 47)e^{-t/5\mu} \text{ mA}$$

## Example 22

Considering the circuit in Fig. 2.60, at $t = 0$, the switch is opened, and 1 kΩ resistor is in the circuit. Find the voltage $v_o(t)$.

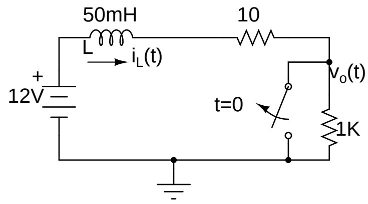



Figure 2.60: Circuit for Example 22.

**Solution**

1. 1 kΩ resistor is shorted for $t < 0$, while 10 Ω resistor is in the circuit. We have $v_o(t) = 0$ for $t < 0$. The current $i_L(0^-)$ just before the switching, can be found by short-circuiting the inductor: $i_L(0^-) = 12/10 = 1.2$ A.

2. Since the inductor current is continuous, the inductor current just after the switching, $i_L(0^+) = 1.2$ A. Hence $v_o(0^+) = 1.2 \cdot 1000 = 1200$ V!

3. At $t = \infty$, we short-circuit the inductor. We determine $i_L(\infty) = 12/1010 = 11.9$ mA and $v_o(\infty) = 11.9$ V.

4. The time constant is found as $\tau = 0.050/1010 = 49.5\mu$s.

5. We write the voltage, $v_o(t)$ for $t > 0$ as

$$v_o(t) = 11.9 + (1200 - 11.9)e^{-t/\tau}$$

which is plotted in Fig. 2.61.

Note that this circuit generates a 1200 V pulse from a 12 V battery.

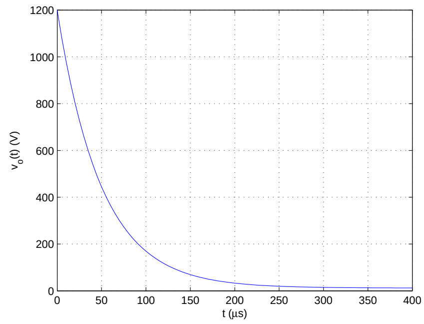



Figure 2.61: $v_o(t)$ as a function of time.

## 2.16 Problems

(Answers of most problems are in p. 311.)

1. Find the values and tolerances of resistors with the following color codes:
   (a) green-blue-red-silver
   (b) orange-white-yellow-silver
   (c) violet-green-brown-gold
2. Find the *average* power delivered or dissipated by the sources and resistor in Fig. 2.62. Make sure that the average power delivered is equal to the average power dissipated.

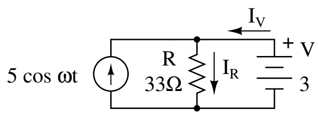



Figure 2.62: Circuit for Prob. 2.

3. Find the *rms* value of the periodic voltage waveform shown in Fig. 2.63.

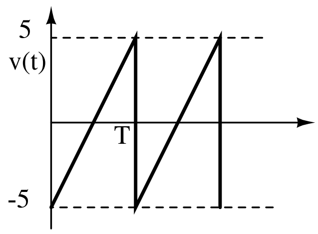



Figure 2.63: Voltage waveform for Prob. 3.

4. Find $R_{eq}$ in the circuits given below (two significant figures, in Ω, K or M as appropriate):
5. Write down the *rms* values (two significant figures) of following voltage or current waveforms:
   (a) 10 cos(1000$t$) V (b) 1.4 sin($314t + 30^o$) A (c) 28 cos($\omega t + \theta$) V
6. What are the frequencies of the waveforms in Problem 5 (three significant figures)?
7. A rechargeable Li-ion (lithium-ion) battery of a mobile phone has a nominal voltage of 3.7 V and a capacity of 650 mA-hr. How long can a charged battery supply energy to a 330 Ω resistor at its rated voltage? Assume that the internal resistance is very small compared to 330 Ω. What is the total energy (in joules) delivered to the resistor?

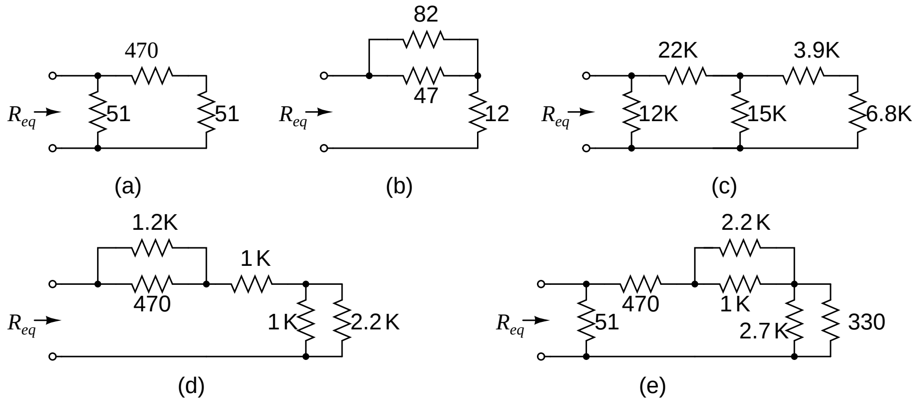



Figure 2.64: Resistor networks for Prob. 4.

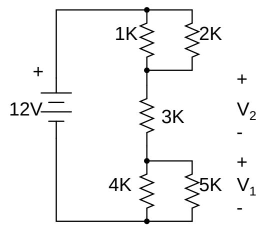



Figure 2.65: Circuit for Prob. 8.

8. Using the voltage divider formula, find the voltages $V_1$ and $V_2$ in Fig. 2.65.

9. Determine the current $I_1$ in Fig. 2.66. Note that {V, $\mu$A, M$\Omega$} is consistent unit set.

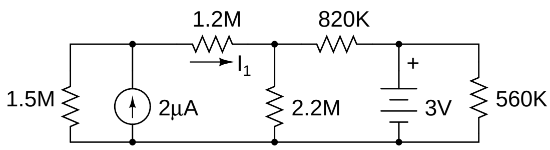



Figure 2.66: Circuit for Prob. 9.

10. Find the marked variables using nodal analysis of the circuits in Fig. 2.67 (three significant figures). Check your results using LTSpice.

11. In Fig. 2.68, find and plot $i_L(t)$ and $v_R(t)$ with $i_L(0)$=30 mA. Initially S is closed, at $t_1$=40 $\mu$s, S is opened. Hint: You need to find the inductor current, $i_L(t_1)$ to act as the initial condition for the circuit when S is opened.

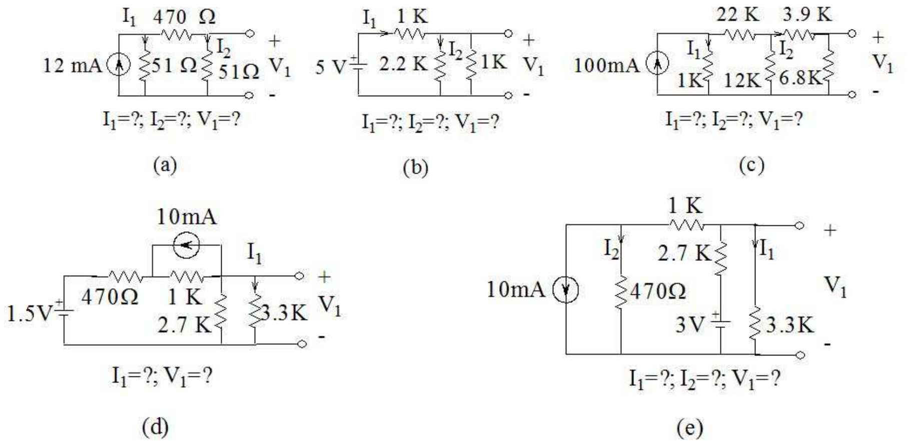



Figure 2.67: Circuits for Prob. 10.

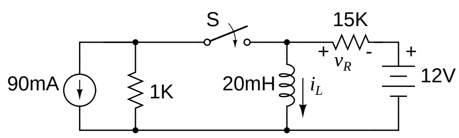



Figure 2.68: Circuit for Prob. 11.

12. Find and plot $i_L(t)$ in Fig. 2.69 with $i_L(0)$=5 mA.

13. Consider the circuits in Fig. 2.70(a) and (b). Both circuits are driven by a step current source $i_S(t)$, shown in Fig. 2.70(c). Find and sketch $i_C(t)$, $v_C(t)$, $i_L(t)$, and $v_L(t)$. Assume that $v_C(0) = 0$ and $i_L(0) = 0$. Verify your results with LTSpice.

14. Find the resistance of 500 m copper wire of diameter 0.1 mm.

15. Steel reinforced aluminum wires are used in long-distance high-voltage overhead lines (HVOHL). "954 ACSR" wire (954 tells the type of conductor and ACSR stands for "Aluminum Conductor-Steel Reinforced") has a cross-section of 485 mm$^2$ and a resistance (per unit length) of 0.059 Ω/km. HVOHL are carried by transmission line poles separated by approximately 400 m, on average. Considering that the wire is made of aluminum predominantly, calculate the mass of the three-phase line between two poles. Note that aluminum has a density of 2.7. Calculate the power loss if 32 MW of power is carried over 200 km at a 380 KV$_{rms}$ 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$_{rms}$? Calculate the mass for this case.

16. Find the ratio of $R_1$ to $R_2$ such that the output of the resistive voltage divider of the circuit given in Fig. 2.71(a) is $V_{out} = -8$ V . Find a pair of

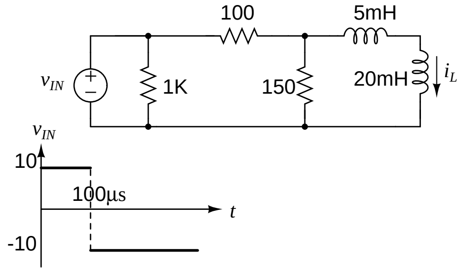



Figure 2.69: Circuit for Prob. 12.

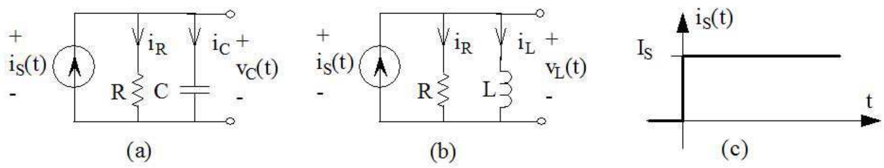



Figure 2.70: Circuits for Prob. 13.

5% tolerance standard resistor values (given in p. 32) for $R_1$ and $R_2$ such that the above ratio is satisfied as much as possible. What is the percent error?

17. Consider the AC voltage divider formed by two capacitors of Fig. 2.71(b). Find a pair of standard 10% tolerance capacitor values (given in p. 42) to obtain $v_{out}(t) = 5\sin(\omega t)$. Find the maximum and minimum $v_{out}$ values considering the tolerance of the capacitors.

18. Find the maximum capacitance of an air variable capacitance (the left-most capacitor in Fig. 2.24), with plates in the form of a half-circle of radius 13 mm. There are 18 plates on the fixed side, and 18 plates on the moveable side. The air gap between the plates is 0.6 mm. Note that there are 35 parallel plate capacitors in this structure.

19. For the circuit shown in Fig. 2.72, the voltage source is as shown in the same figure. (a) Find $i_L(0^-)$ and $v_L(0^-)$. ($0^-$ denotes the time right before $t$=0). (b) Find $i_L(0^+)$ and $v_L(0^+)$ ($0^+$ denotes the time right after $t$=0). (c) Find and plot $v_L(t)$ for $-\infty < t < \infty$. Check your results with LTSpice.

20. Consider the amplifier given in Fig. 2.73, which has an 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 $R_{in} = 10R_L$? In dB?

21. Considering the circuit given in Fig. 2.74, find and plot the current in the inductor, $i_L(t)$, as a function of time between $0 < t < 10$ $\mu$s.

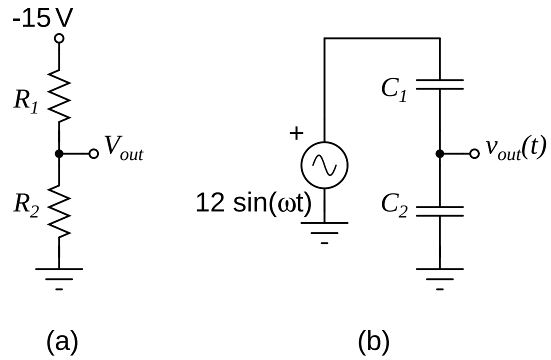



Figure 2.71: Circuits for Prob. 16 and 17

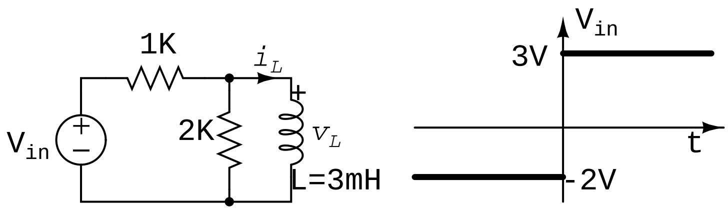



Figure 2.72: Circuit for Prob. 19

22. For the circuit of Fig. 2.75, find and plot the voltage of the capacitor, $v_C(t)$, as a function of time between $0 < t < 30$ ms.

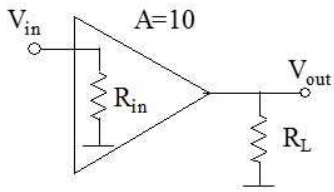



Figure 2.73: Circuit for Prob. 20

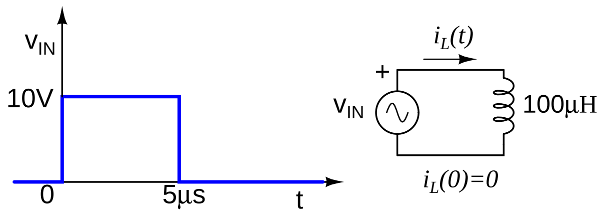



Figure 2.74: Circuit for Prob. 21

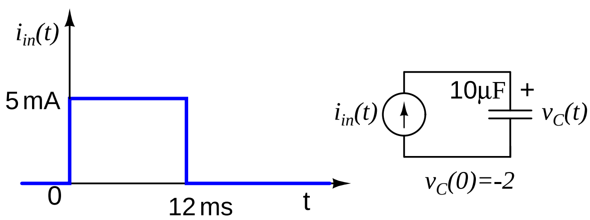



Figure 2.75: Circuit for Prob. 22

# 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-11 are discussed in this chapter. The mathematical tools necessary to analyze circuits used in TRC-11 are also developed.

English electrical engineer Oliver Heaviside (1850–1925) adapted complex numbers to analyze electrical circuits. Complex numbers are very important in Electrical Engineering, so we give a brief summary of complex numbers here.

## 3.1 Complex numbers

The equation $x^2 + 1 = 0$ has no real roots. Solution of this equation can be written as $x = \pm\sqrt{-1}$. To handle such problems, we use the complex number system.
A complex number $z$ has the rectangular form

$$z = a + jb \quad \text{where} \quad j = \sqrt{-1} \tag{3.1}$$

and $a$ and $b$ are real numbers. $j$ is called the *imaginary unit* and has the property of $j^2 = -1$.* $a$ is the *real part* and $b$ is the *imaginary part*:

$$\mathcal{Re}\{z\} = a \quad \text{and} \quad \mathcal{Im}\{z\} = b \tag{3.2}$$

This complex number can be shown as a vector in the complex plane as demonstrated in Fig. 3.1(a). 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. If 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, while $0 - j6$ is an imaginary number.
The complex conjugate of a complex number $z = a + jb$ is $z^* = a - jb$.

*Electrical Engineers prefer to use the symbol $j$ rather than $i$, since the symbol $i$ is reserved for current.

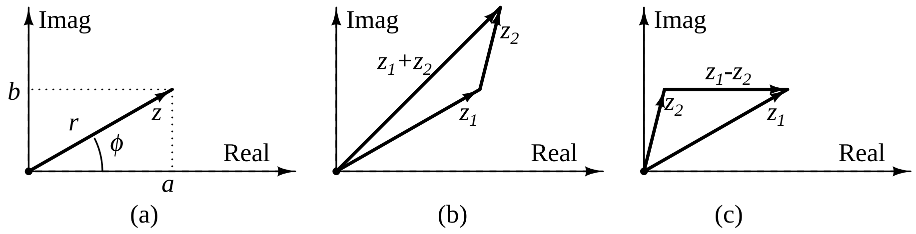



Figure 3.1: (a)Vector showing the complex number $z = a + jb$, (b) addition of two complex numbers, (c) subtraction of two complex numbers.

The algebra of complex numbers is the same as the algebra of real numbers with $j^2$ replaced by $-1$:
Addition (see Fig. 3.1(b)):

$$z_1 + z_2 = (a + jb) + (c + jd) = (a + c) + j(b + d)$$

Subtraction (see Fig. 3.1(c)):

$$z_1 - z_2 = (a + jb) - (c + jd) = (a - c) + j(b - d)$$

Multiplication:

$$z_1 z_2 = (a + jb)(c + jd) = (ac - bd) + j(bc + ad)$$

Division:

$$\frac{z_1}{z_2} = \frac{a + jb}{c + jd} = \frac{ac + bd}{c^2 + d^2} + j\frac{bc - ad}{c^2 + d^2}$$
$$\frac{1}{z_2} = \frac{1}{c + jd} = \frac{c}{c^2 + d^2} - j\frac{d}{c^2 + d^2}$$

Absolute value:

$$|a + jb| = \sqrt{a^2 + b^2}$$

If $z = a + jb$, then

$$zz^* = |z|^2 = a^2 + b^2$$

We also have

$$|z_1 z_2| = |z_1||z_2|$$

and

$$\left|\frac{z_1}{z_2}\right| = \frac{|z_1|}{|z_2|} \quad \text{if } z_2 \neq 0$$

There is a relation between sinusoids and exponential function as follows:

$$e^{j\phi} = \cos\phi + j\sin\phi \tag{3.3}$$

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

$$\cos\phi = \mathcal{Re}\left\{e^{j\phi}\right\} \tag{3.4}$$

or

$$\cos\phi = \frac{e^{j\phi} + e^{-j\phi}}{2} \tag{3.5}$$

and

$$\sin\phi = Im\left\{e^{j\phi}\right\} \tag{3.6}$$

or

$$\sin\phi = \frac{e^{j\phi} - e^{-j\phi}}{2j} \tag{3.7}$$

in turn. The magnitude of this exponential function is

$$|e^{j\phi}| = 1 \tag{3.8}$$

regardless of the value of the argument $\phi$.

If $z = a + jb$ we can write the complex number $z$ in trigonometric form as

$$z = r(\cos\phi + j\sin\phi) \tag{3.9}$$

the exponential form as (see Fig. 3.1)

$$z = re^{j\phi} \tag{3.10}$$

and the polar form as

$$z = r\angle\phi \tag{3.11}$$

$$r = \sqrt{a^2 + b^2} \ \text{ and } \ \phi = \tan^{-1}\left(\frac{b}{a}\right) \tag{3.12}$$

With $z_1 = r_1 e^{j\phi_1}$ and $z_2 = r_2 e^{j\phi_2}$, we write the product as

$$z_1 z_2 = r_1 r_2 e^{j(\phi_1 + \phi_2)}$$

or

$$z_1 z_2 = r_1 r_2 \angle(\phi_1 + \phi_2)$$

the division as

$$\frac{z_1}{z_2} = \frac{r_1}{r_2} e^{j(\phi_1 - \phi_2)}$$

or

$$\frac{z_1}{z_2} = \frac{r_1}{r_2} \angle(\phi_1 - \phi_2)$$

Clearly, the multiplication and division are easily performed in exponential or polar forms, while the addition and subtraction are easier in rectangular form.

## 3.2 Phasors

In Electrical Engineering we frequently deal with sinusoidal signals; sometimes we add them, sometimes we subtract them from each other. Consider two sinusoids at the same frequency $\omega$, but with a differing amplitude and phase. The sum of these sinusoids can be written as

$$A\cos(\omega t + \theta_1) + B\cos(\omega t + \theta_2) =$$
$$= \sqrt{A^2 + B^2 - 2AB\cos(\theta_2 - \theta_1)} \quad \cos\left(\omega t + \tan^{-1}\left(\frac{A\sin\theta_1 + B\sin\theta_2}{A\cos\theta_1 + B\cos\theta_2}\right)\right) \tag{3.13}$$

Just the sum of two sine waves at the same frequency is a rather clumsy equation. It is obviously difficult to manipulate equations of sine waves using the time-domain notation above.

We can simplify a great deal if we use the Euler's formula. From Eq. 3.4 we can write

$$A\cos(\omega t + \theta_1) = \mathcal{Re}\{Ae^{j(\omega t+\theta_1)}\} = \mathcal{Re}\{Ae^{j\omega t}e^{j\theta_1}\} \tag{3.14}$$

similarly

$$B\cos(\omega t + \theta_2) = \mathcal{Re}\{Be^{j(\omega t+\theta_2)}\} = \mathcal{Re}\{Be^{j\omega t}e^{j\theta_2}\} \tag{3.15}$$

To simplify the notation, we can get rid of the $e^{j\omega t}$ term since the frequency is common. To simplify further we can also get rid of the real part operator $\mathcal{Re}$:

$$A\cos(\omega t + \theta_1) \;\Rightarrow\; Ae^{j\theta_1} \tag{3.16}$$

$$B\cos(\omega t + \theta_2) \;\Rightarrow\; Be^{j\theta_2} \tag{3.17}$$

Hence, we represent a sine wave with a complex number with no time dependence. The magnitude of the complex number represents the amplitude of the sine wave and the phase of the complex number corresponds to the phase of the sine wave. It is also possible to write the complex numbers in polar coordinates as $A\angle\theta_1$ or $B\angle\theta_2$. We call this notation as *phasor* notation.

We can easily return to the time-domain notation by taking the real part of the phasor multiplied by $e^{j\omega t}$.

$$\mathcal{Re}\{Ae^{j\theta_1}e^{j\omega t}\} = A\cos(\omega t + \theta_1) \tag{3.18}$$

$$\mathcal{Re}\{Be^{j\theta_2}e^{j\omega t}\} = B\cos(\omega t + \theta_2) \tag{3.19}$$

With this phasor notation, the sum of the two sine waves can be found easily by adding two complex numbers:

$$A\cos(\omega t + \theta_1) + B\cos(\omega t + \theta_2) \;\Rightarrow\; A\angle\theta_1 + B\angle\theta_2 \tag{3.20}$$

Since adding complex numbers is best done in rectangular form, we write

$$A\angle\theta_1 + B\angle\theta_2 = (A\cos\theta_1 + jA\sin\theta_1) + (B\cos\theta_2 + jB\sin\theta_2) =$$

$$= (A\cos\theta_1 + B\cos\theta_2) + j(A\sin\theta_1 + B\sin\theta_2) \tag{3.21}$$

We can convert the final result to time-domain by multiplying this complex number by $e^{j\omega t}$ and taking its real part to get the same result given in Eq. 3.13. Obviously, adding two complex numbers is much easier than dealing with the trigonometric identities of Eq. 3.13.

**Example 1**

Below are some examples of conversion from time-domain to phasor notation.

$$5\cos(\omega t + 23^o) \;\Rightarrow\; 5e^{j23^o}$$

$$3.2\sin(\omega t) = 3.2\cos(\omega t - \pi/2) \;\Rightarrow\; 3.2e^{-j\pi/2} = -3.2j$$

$$-10\cos(\omega t) \;\Rightarrow\; -10$$

$$
\begin{aligned}
10\cos(\omega t + 12^o) + 8\cos(\omega t - 76^o) \;\;\Rightarrow\;\; & 10e^{j12^o} + 8e^{-j76^o} = \\
= 10(\cos 12^o + j\sin 12^o) + 8(\cos(-76^o) + j\sin(-76^o)) & = \\
= 9.78 + j2.08 + 1.93 - j7.76 & = 11.71 - j5.68
\end{aligned}
$$

in the last example, we used Eq. 3.9 to convert from the polar form to rectangular form.

In the following examples, we use Eq. 3.18 to convert the phasors to time domain:

$$7e^{-j12^o} \;\;\Rightarrow\;\; 7\cos(\omega t - 12^o)$$

$$6 + j6 = 6\sqrt{2}e^{j\pi/4} \;\;\Rightarrow\;\; 6\sqrt{2}\cos(\omega t + \pi 4)$$

$$3 - j4 = 5e^{j\tan^{-1}(-4/3)} = 5e^{-j53.1^o} \;\;\Rightarrow\;\; 5\cos(\omega t - 53.1^o)$$

In the last example, we first used the formulas of Eq. 3.12 to convert the rectangular form complex number to the exponential form.

---

### 3.2.1 Derivative operator

Let us find what the derivative operator does in phasor domain:

$$\frac{d}{dt}A\cos(\omega t + \theta) = -A\omega\sin(\omega t + \theta) \;\;\Rightarrow\;\; -\omega A e^{j(\theta - \pi/2)} = j\omega A e^{j\theta} \tag{3.22}$$

Hence, the derivative operator in the time-domain corresponds to a multiplication by $j\omega$ in the phasor domain.

### 3.2.2 Integration operator

Let us find what the integration means in phasor domain:

$$\int A\cos(\omega t + \theta) = \frac{A}{\omega}\sin(\omega t + \theta) \;\;\Rightarrow\;\; \frac{A}{\omega}e^{j(\theta - \pi/2)} = \frac{1}{j\omega}Ae^{j\theta} \tag{3.23}$$

Hence, the integration operation in the time-domain corresponds to a division by $j\omega$ in the phasor domain.

### 3.2.3 Resistor with sinusoidal excitation

If a resistance has sinusoidal voltage or current then we can use phasors. Ohm's law in the time domain is similar in the phasor domain:

$$v_R = Ri_R \;\;\Rightarrow\;\; V_R = RI_R \tag{3.24}$$

Therefore, the resistance $R$ in phasor domain is unchanged.

### 3.2.4 Capacitor with sinusoidal excitation

For a capacitor with sinusoidal voltage or current, we can write

$$i_C = C\frac{d}{dt}v_C \;\;\Rightarrow\;\; I_C = j\omega C V_C \text{ or } V_C = \frac{1}{j\omega C}I_C \tag{3.25}$$

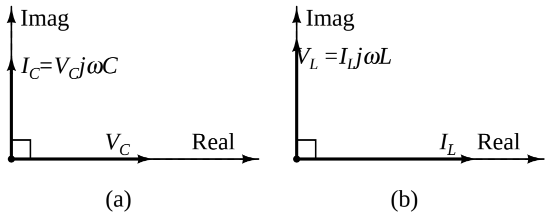



Figure 3.2: Current and voltage phasors for a capacitor (a) and for an inductor (b).

Note that the resulting phasor equation is like Ohm's law with $1/(j\omega C)$ replacing $R$. The current and voltage phasors for a capacitor are demonstrated in Fig. 3.2(a).

We note that real-life capacitors behave like that given in Eq. 3.25 only up to a frequency limit (SRF) † due to the inductance of the capacitor leads or internal connections. Above this frequency limit, the capacitor acts like a small inductor! Since small capacitors have higher SRF, it is recommended to choose the smallest capacitor that will satisfy the requirements. To extend the frequency range of capacitors, often small capacitors with higher SRF is placed in parallel with larger capacitors.

### 3.2.5 Inductor with sinusoidal excitation

Similarly, an inductor with a sinusoidal voltage or current is specified in phasor domain as

$$v_L = L\frac{d}{dt}i_L \quad \Rightarrow \quad V_L = j\omega L I_L \tag{3.26}$$

In this case, the phasor equation is also like Ohm's law with $j\omega L$ replacing $R$. The current and voltage phasors for an inductor are shown in Fig. 3.2(b).

Real-life inductors behave like that given in Eq. 3.26 only up to a frequency limit (SRF)‡ due to interwinding capacitance. Above this frequency limit, the inductor acts like a small capacitance. Therefore, a designer should choose the smallest inductor value satisfying the requirements.

## 3.3 Linear circuits

*Linearity* is a fundamental concept in circuit analysis. Consider the block diagram in Fig. 3.3. A circuit is called linear if it satisfies the following relation:

If input signals $x_1(t)$ and $x_2(t)$ (voltage or current) yield the output signals $y_1(t)$ and $y_2(t)$, respectively, then a linear combination of inputs, $ax_1(t)+bx_2(t)$ yields the same combination of the individual outputs, $ay_1(t) + by_2(t)$, where $a$ and $b$ are real numbers.

†SRF is the self-resonance frequency (SRF). It can be found in the data sheets of capacitors.

‡The self-resonance frequency of inductors can be found in the data sheets of off-the-shelf inductors or it can be measured for an in-house manufactured inductor.

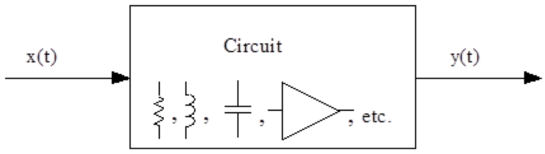



Figure 3.3: A linear circuit block with an input signal $x(t)$ and an output signal $y(t)$.

Consider a circuit formed by a single resistor. If the input signals to the resistor are currents $i_1(t)$ and $i_2(t)$, and the output signals $v_1(t)$ and $v_2(t)$ is the voltages developed across the resistor, then we have

$$v_1(t) = Ri_1(t) \text{ and } v_2(t) = Ri_2(t) \tag{3.27}$$

If we apply a combination of two inputs $ai_1(t) + bi_2(t)$, then the total voltage developed across the resistor is

$$v(t) = R[ai_1(t) + bi_2(t)] = aRi_1(t) + bRi_2(t) = av_1(t) + bv_2(t) \tag{3.28}$$

Hence, a resistor is a linear circuit element.

Consider an inductor. If the input signals to the inductor are currents $i_1(t)$ and $i_2(t)$, and the output signals $v_1(t)$ and $v_2(t)$ are the voltages developed across the inductor, then we have

$$v_1(t) = L\frac{d}{dt}i_1(t) \text{ and } v_2(t) = L\frac{d}{dt}i_2(t) \tag{3.29}$$

If we apply a combination of two inputs $ai_1(t) + bi_2(t)$, then the total voltage developed across the resistor is

$$v(t) = L\frac{d}{dt}[ai_1(t) + bi_2(t)] = aL\frac{d}{dt}i_1(t) + bL\frac{d}{dt}i_2(t) = av_1(t) + bv_2(t) \tag{3.30}$$

Hence, an inductor is also a linear circuit element.

Similarly, a capacitor and a transformer are also linear elements.

However, an ideal diode is *not* a linear element. We can prove this using a counter example: Suppose the input signals to an ideal diode are currents $i_1 = 2$ mA and $i_2 = 1$ mA, and the output signals $v_1$ and $v_2$ is the voltages developed across the diode, from Eq. 4.1 we have

$$v_1 = 0 \text{ and } v_2 = 0 \tag{3.31}$$

We apply a combination of two inputs with $a = -5$ and $b = 1$, or $-5i_1 + i_2 = -9$ mA, then the total voltage developed across the ideal diode should also be 0. Obviously, this is not true, because the voltage is zero only for positive currents. Therefore, an ideal diode is not a linear circuit element.

### 3.3.1 Steady-state solution of linear *RLC* Circuits with sinusoidal excitation

Circuits composed of *RLC* components are linear. If the excitations in these circuits are sinusoidal, we can use phasors to simplify solution of such circuits.

We are not limited to first-order circuits with just one capacitor or one inductor. We can deal with any number of capacitors and inductors.

The phasor method is not able to find the *transient* solution that occurs right after the sinusoidal signal is applied. Rather, it can find the *steady-state* solution long after the sinusoidal signal is applied and transients have disappeared. Note that this method is applicable only when the excitation to the circuit sinusoidal. Moreover, the circuit should not contain any nonlinear elements like diodes.

To find the *steady-state* solution of linear circuits with *sinusoidal* excitation, we use the following procedure:

1. Replace inductors with $j\omega L$ and capacitors with $1/(j\omega C)$
2. Replace voltage or current source with the corresponding phasor.
3. Solve the circuit using nodal analysis.
4. If needed, convert the desired quantities to the time-domain

We cannot use this method for circuits containing non-sinusoidal sources, it applies only to circuits with sinusoidal excitation. Note also that initial values of capacitor voltages or inductor currents do not play a role in this procedure since we find the *steady-state* solution. The procedure is best understood with the following examples.

**Example 2**

Consider the circuit shown in Fig. 3.4(a) and find the voltage across the capacitor.

Since the excitation is sinusoidal and the circuit contains only linear elements we can use the procedure above:

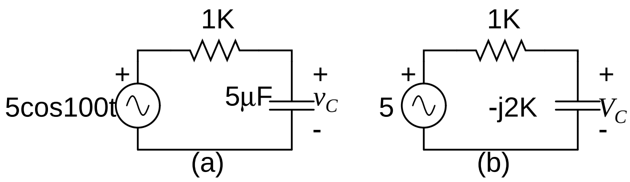



Figure 3.4: Example 2 for the phasor solution of an RC circuit

1. Replace the capacitor with $1/(j\omega C) = 1/(j100 \times 5 \times 10^{-6}) = -j2000 = -j2\text{K}$
2. Replace the voltage source with the phasor 5.
3. Referring to Fig. 3.4(b), the voltage phasor across the capacitor can be found using nodal analysis

$$\frac{V_C - 5}{1K} + \frac{V_C}{-j2\text{K}} = 0$$

Hence

$$V_C = \frac{-j2\text{K}}{1K - j2\text{K}}5 = \frac{-j2(1+j2)}{1+4}5 = 4 - j2 = \sqrt{20}e^{-j26.5^o}$$

4. In time domain, we have $v_C(t) = \sqrt{20}\cos(100t - 26.5^o)$ V

---

**Example 3**

Consider the circuit shown in Fig. 3.5(a) and find the voltage $v_1(t)$.

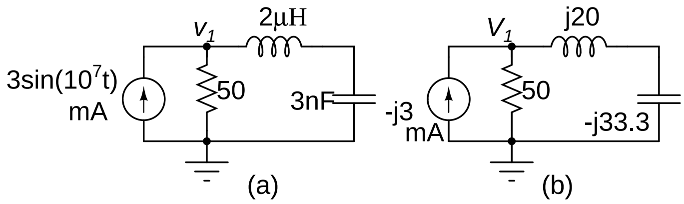



Figure 3.5: Example 3 for the phasor solution of an RLC circuit

We have a linear circuit excited with a sinusoidal signal. We can use the phasor method:

1. Replace the inductor with $j\omega L = j10^7 \times 2 \times 10^{-6} = j20$, and the capacitor with $1/(j\omega C) = 1/(j10^7 \times 3 \times 10^{-9}) = -j33.3$
2. Replace the current source with the phasor $-j3 \times 10^{-3}$.
3. Referring to Fig. 3.4(b), the voltage phasor across the capacitor can be found from nodal analysis:

$$-j3 \times 10^{-3} = \frac{V_1}{50} + \frac{V_1}{j20 - j33.3} = V_1\frac{50 - j13.3}{-j13.3 \times 50}$$

Hence, we find

$$V_1 = \frac{-1.99}{50 - j13.3} = \frac{1.99\angle 180^o}{51.7\angle - 14.9^o} = 0.038\angle 194.9^o$$

where we converted the nominator and the denominator from rectangular form to polar form to simplify complex division.

4. In time domain, we have $v_1(t) = 0.038\cos(10^7 t + 194.9^o)$ V

---

### 3.3.2 Power relation for phasors

The average power, $P$, dissipated on a resistor is given by Eq. 3.32:

$$P = \frac{v_{rms}^2}{R} \tag{3.32}$$

For a sinusoidal signal with $v(t) = V_p \cos(\omega t + \theta)$, we find

$$v_{rms} = \sqrt{\frac{1}{T}\int_0^T V_p^2 \cos^2(\omega t + \theta) dt} = \frac{V_p}{\sqrt{2}} \tag{3.33}$$

Hence, the average power is

$$P = \frac{V_p^2}{2R} \tag{3.34}$$

In phasor notation, the signal $v(t)$ is represented by the phasor $V = V_p e^{j\theta}$ and the average power for a resistor can be written as

$$P = \frac{|V|^2}{2R} = \frac{VV^*}{2R} = \frac{VI^*}{2} = \frac{II^*R}{2} = \frac{|I|^2 R}{2} \tag{3.35}$$

where $I$ is the current phasor and $I = V/R$ or $I^* = V^*/R$.

In general, for circuits where the current and voltage may have phase difference, the power dissipated in the circuit can be written as

$$P = \mathcal{Re}\left\{\frac{VI^*}{2}\right\} = \mathcal{Re}\left\{\frac{V^*I}{2}\right\} \tag{3.36}$$

For a capacitor with $I_C = j\omega CV$, the average power is given by

$$P = \mathcal{Re}\left\{\frac{-Vj\omega CV^*}{2}\right\} = \frac{|V|^2}{2}\mathcal{Re}\left\{-j\omega C\right\} = 0 \tag{3.37}$$

giving zero, consistent with the fact that a capacitor does not dissipate power. It only stores energy. The same condition holds for an inductor.

### 3.3.3 Impedance

We have observed that the definition of phasors allowed us to convert the differential relations in time into algebraic relations in angular frequency.

*Impedance*, $Z$, of a network is defined as the ratio of voltage phasor to current phasor:

$$Z = R + jX = \frac{V}{I} \tag{3.38}$$

$Z$ is in general a frequency dependent complex quantity and has the unit of $\Omega$. Its real part, $R$, is known as the resistance and its imaginary part, $X$, is referred to as *reactance*.

Impedance is convenient to use when components are connected in series: The impedances of components can be added to find the total impedance.

### 3.3.4 Admittance

> The inverse of $Z$ is called *admittance* and it is denoted by $Y$:
>
> $$Y = G + jB = \frac{1}{Z} = \frac{I}{V} \tag{3.39}$$

The unit of $Y$ is Siemens (S), named after German inventor Ernst Werner von Siemens (1816–1892). The unit of the admittance is also commonly referred to as *mho*, and its symbol $\mho$ (an upside-down $\Omega$). The real part of the admittance, $G$, is *conductance* and the imaginary part, shown by $B$, is the *susceptance*.

In circuits with many parallel components, it is more convenient to use admittance rather than impedance: Admittances of parallel circuits can simply be added.

**Example 4**

Let us find the impedance of the series connected circuit of Fig. 3.6(a) and the admittance of the parallel connected circuit of Fig. 3.6(b) at the given frequencies.

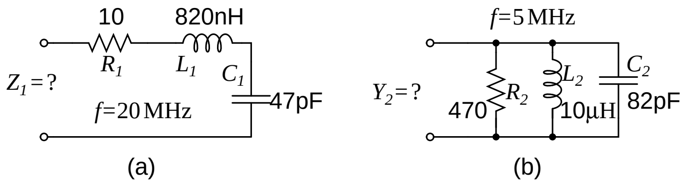



Figure 3.6: Example 4 for impedance and admittance calculation

$$\begin{aligned} Z_1 = R_1 + j\omega L_1 + \frac{1}{j\omega C_1} &= 10 + j(1.26 \cdot 10^8)(820 \cdot 10^{-9}) \\ &+ \frac{1}{j(1.26 \cdot 10^8)(47 \cdot 10^{-12})} = 10 + j103 - j169 = 10 - j66\ \Omega \end{aligned}$$

$$\begin{aligned} Y_2 = \frac{1}{R_2} + j\omega C_2 + \frac{1}{j\omega L_2} &= \frac{1}{470} + j(3.14 \cdot 10^7)(82 \cdot 10^{-12}) \\ + \frac{1}{j(3.14 \cdot 10^7)(10 \cdot 10^{-6})} &== 2.13 \cdot 10^{-3} + j2.58 \cdot 10^{-3} - j3.18 \cdot 10^{-3} = \\ &= 2.13 - 0.61\ \text{m}\mho \end{aligned}$$

---

**Example 5**

Consider the circuit given in Fig. 3.7. The impedance can be found by applying a voltage phasor $V$ and finding $I$ in terms of it:

$$I = \frac{V}{100} + \frac{V}{j\omega(2 \cdot 10^{-6}) + 1/(j\omega(3 \cdot 10^{-9}))}$$

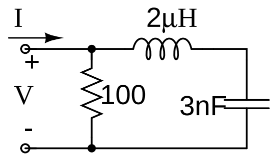



Figure 3.7: Example 5 for impedance and admittance calculation

Hence the impedance is given by

$$Z(\omega) = \frac{V}{I} = \frac{100[(1/3)\cdot 10^9 - \omega^2(2\cdot 10^{-6})]}{(1/3)\cdot 10^9 - \omega^2(2\cdot 10^{-6}) + j\omega 100}$$

the resistive part is

$$R(\omega) = \mathcal{Re}\{Z\} = \frac{100[(1/3)\cdot 10^9 - \omega^2(2\cdot 10^{-6})]^2}{[(1/3)\cdot 10^9 - \omega^2(2\cdot 10^{-6})]^2 + \omega^2 10^4}$$

and the reactance is given by

$$X(\omega) = \mathcal{Im}\{Z\} = \frac{-j\omega 10^4[(1/3)\cdot 10^9 - \omega^2(2\cdot 10^{-6})]}{[(1/3)\cdot 10^9 - \omega^2(2\cdot 10^{-6})]^2 + \omega^2 10^4}$$

Note that the real part of the impedance acts like a frequency dependent resistor. On the other hand, the admittance is found as

$$Y(\omega) = \frac{I}{V} = \frac{(1/3)\cdot 10^9 - \omega^2(2\cdot 10^{-6}) + j\omega 100}{100[(1/3)\cdot 10^9 - \omega^2(2\cdot 10^{-6})]}$$

Hence the conductance is

$$G(\omega) = \mathcal{Re}\{Y\} = \frac{1}{100}$$

and the susceptance is

$$B(\omega) = \mathcal{Im}\{Y\} = \frac{j\omega}{(1/3)\cdot 10^9 - \omega^2(2\cdot 10^{-6})}$$

---

## 3.4 Transfer function

For networks with an input voltage phasor, $V_i$, and an output voltage phasor, $V_o$, we define the ratio of output to input as a transfer function, $H$:

$$H(\omega) = \frac{V_o(\omega)}{V_i(\omega)} \tag{3.40}$$

a complex function of $\omega$. It specifies the output with respect to input for any radial frequency $\omega$.

The magnitude of the transfer function $|H(\omega)|$ is typically plotted as a graph to visualize the frequency response of the network. Decibels (dB) may be the preferred vertical axis to be able to see very small and very large values on the same graph.

$$|H(\omega)|_{dB} = 20 \log_{10} \left| \frac{V_o(\omega)}{V_i(\omega)} \right| \tag{3.41}$$

Moreover, the frequency axis may be plotted as a logarithmic axis to be able to see the response in a wide range of frequencies.

### 3.4.1 Transfer function of first-order circuits

The following procedure can be used to draw the transfer function (in dB) of a *first-order* circuit on a logarithmic frequency scale:

1. Write the complex transfer function, $H(\omega)$, in one of the following three generic forms:

$$A\frac{1}{1+j\omega X}, \quad A\frac{j\omega X}{1+j\omega X}, \quad A\frac{1+j\omega X}{1+j\omega Y} \tag{3.42}$$

2. Draw the low-frequency asymptote of $|H(\omega)|_{dB}$ for $\omega \to 0$.
3. Draw the high-frequency asymptote of $|H(\omega)|_{dB}$ for $\omega \to \infty$.
4. The asymptotes have a slope of either 0 or $\pm 20$ dB/decade.
5. For the first two generic forms, the asymptotes should intersect at the 3-dB corner frequency of

$$\omega_c = \frac{1}{X} \text{ or } f_c = \frac{1}{2\pi X} \tag{3.43}$$

6. For the last generic form, draw the mid-frequency asymptote with a slope of $\pm 20$ dB/decade between the two corner frequencies:

$$f_{c1} = \frac{1}{2\pi X} \quad \text{and} \quad f_{c2} = \frac{1}{2\pi Y} \tag{3.44}$$

7. Draw the transfer function approximately using the asymptotes.

**Example 6**

To find the transfer function of the first-order circuit shown in Fig. 3.8(a), we use the procedure of page 89 and write

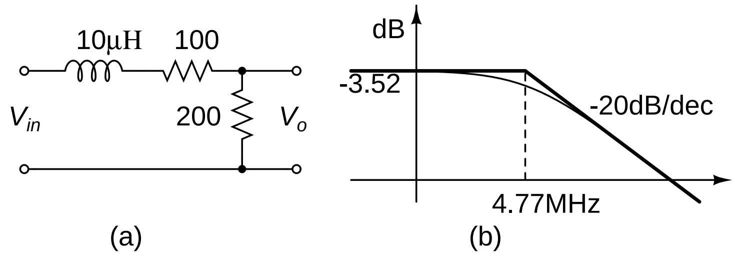



Figure 3.8: (a) A first-order circuit for transfer function evaluation, (b) Transfer function.

1. Using the voltage divider formula, we find

$$H(\omega) = \frac{V_o}{V_{in}} = \frac{200}{200 + 100 + j\omega\, 10^{-5}} = \frac{2}{3} \frac{1}{1 + j\omega\, 10^{-5}/300} \tag{3.45}$$

   which is like the first generic form of Eq. 3.42.

2. The low-frequency asymptote for $\omega \to 0$ is $|H(\omega)|_{dB} = 20 \log_{10}(2/3) = -3.52$ dB with 0 slope.

3. The high-frequency asymptote of $|H(\omega)|_{dB}$ for $\omega \to \infty$ has a slope of $-20$ dB/dec, since the transfer function amplitude drops as the frequency is increased.

4. Since the asymptotes intersect at

$$f_c = \frac{1}{2\pi} \frac{300}{10^{-5}} = 4.77 \text{ MHz}$$

   we can draw the high frequency asymptote accordingly.

5. The transfer function is drawn approximately as in Fig. 3.8(b) using the asymptotes.

---

**Example 7**

To find the transfer function of the first-order circuit shown in Fig. 3.9(a), we use the same procedure:

1. Using the voltage divider formula, we find

$$H(\omega) = \frac{V_o}{V_{in}} = \frac{1K + 1/(j\omega(2.2n))}{22K + 1K + 1/(j\omega(2.2n)))} = \frac{1 + j\omega\,(2.2n)(1K)}{1 + j\omega\,(2.2n)(23K)} \tag{3.46}$$

   The transfer function fits the last generic form of Eq. 3.42.

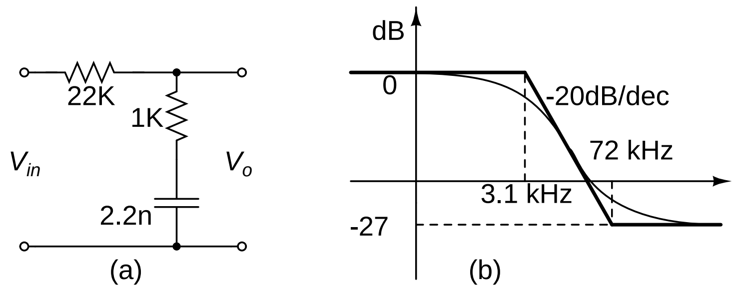



Figure 3.9: (a) A first-order circuit for transfer function evaluation, (b) Transfer function.

2. The low-frequency asymptote for $\omega \to 0$ is $|H(\omega)|_{dB} = 20\log_{10}(1) = 0$ dB with 0 slope.

3. The high-frequency asymptote of $|H(\omega)|_{dB}$ for $\omega \to \infty$ is $|H(\omega)|_{dB} = 20\log_{10}(1/23) = -27.2$ dB with 0 slope, since the transfer function stays constant at high frequencies.

4. Since the asymptotes intersect at

$$f_{c1} = \frac{1}{2\pi}\frac{1}{(2.2n)(23K)} = 3.1 \text{ kHz} \quad f_{c2} = \frac{1}{2\pi}\frac{1}{(2.2n)(1K)} = 72.3 \text{ kHz}$$

   we can draw the mid-frequency asymptote with slope $-20$ dB/dec accordingly.

5. The transfer function is drawn approximately as in Fig. 3.9(b) using the three asymptotes.

---

## 3.4.2 Transfer function of higher-order circuits

The method given in page 89 can not be used for higher order circuits. After finding the transfer function, one may try to find the low- and high-frequency asymptotes. But at mid-frequencies, a numerical evaluation is generally necessary.

LTSpice can also be used to plot transfer function of a linear circuit of a higher order (see page 302 for a tutorial on a second-order circuit).

**Example 8**

Consider the network given in Fig. 3.10. Let us find the transfer function and plot the magnitude of the transfer function. We apply a voltage phasor at the input and find the output phasor using nodal analysis:

$$H(\omega) = \frac{1/(j\omega(3\cdot 10^{-9}))}{10 + j\omega(2\cdot 10^{-6}) + 1/(j\omega(3\cdot 10^{-9}))} = \frac{1}{1-\omega^2(6\cdot 10^{-15}) + j\omega(3\cdot 10^{-9})}$$

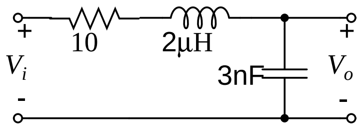



Figure 3.10: Example for transfer function calculation

The low-frequency asymptote is found as $|H(\omega)|_{dB} = 20\log_{10}(1) = 0$ dB. The high-frequencies are dominated by the second term of the denominator with $\omega^2$. Hence the slope of the high-frequency asymptote is $-20 \times 2 = -40$ dB/dec.

The magnitude $|H(\omega)|$ can be plotted in dB scale on a logarithmic frequency axis using the following MATLAB code:

```
% MATLAB code to plot the transfer function
clear all % clear all variables in MATLAB
hold off
fmin=1e5; %minimum frequency in Hz
fmax=2e7; %maximum frequency in Hz
C=3e-9; % Capacitor value in F
L=2e-6; % Inductor value in H
R=10; % resistance in Ohms

f=fmin:fmin:fmax; % Frequency vector
w=2*pi*f; % angular frequency vector
H=1./(j*w*C)./(R+j*w*L+1./(j*w*C)); % MATLAB performs
%                                     an array operation
% Note that we need a "." in front of operators
%                               to perform array operations
Hdb=20*log10(abs(H)); % calculate the magnitude of
%                          transfer function in dB
semilogx(f,Hdb,'LineWidth',2) % plot on a logarithmic x-axis
%                         with a linewidth of 2
grid on % to plot the grid lines
xlabel('f (Hz)') % to place the x-label on the plot
ylabel('|H(\omega)|_{dB}') % to place the y-label
title('Transfer Function Example') % to place a title on the graph
hold on
axis([fmin fmax  -40 10]); % define the axes limits
```

The resulting graph is shown in Fig. 3.11. Note that the magnitude of the transfer function is unity (0 dB) for small frequencies. It is greater than unity (>0 dB) at around 2 MHz, while it is less than 1 (<0 dB) for frequencies more than 3 MHz.

---

## 3.5 Thévenin Equivalent Circuit

A purely resistive linear circuit composed of any number of voltage sources, current sources and resistances with the terminals A and B as in Fig. 3.12(a)

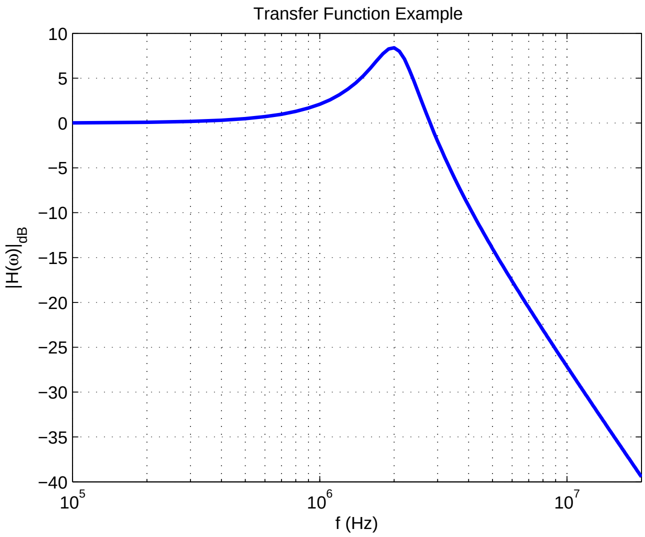



Figure 3.11: Magnitude of the transfer function in dB scale

can be modelled with just two components at the same terminals: A voltage source, $v_{th}$ and a series resistance, $R_{eq}$ as shown in Fig. 3.12(b). This simple model is called the *Thévenin* equivalent circuit, named after the French engineer Léon Charles Thévenin (1857–1926). Since the circuit is linear, its $V-I$

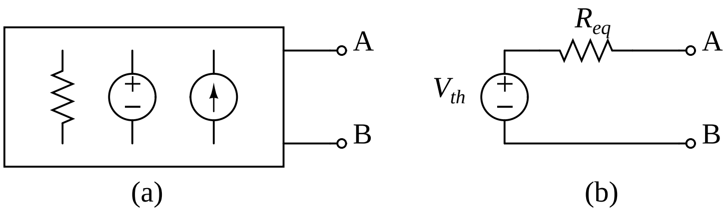



Figure 3.12: Thévenin equivalent circuit

characteristics is a straight line. Thévenin's theorem basically states that the straight line can be modelled with just two parameters: Its slope ($R_{eq}$) and its intersection on the voltage axis ($v_{th}$).

Thévenin equivalent circuit of a black-box composed of any number of resistors/voltage and current sources can be found using the following procedure:

1. Find the voltage between the terminals A and B while those terminals are open-circuited to determine $v_{th}$.
2. Kill the voltage and current sources within the black-box: Short-circuit the voltage sources and open-circuit the current sources. Find

the resistance between the terminals A and B to determine $R_{eq}$.

**Example 9**

Consider the circuit given in Fig. 3.13(a). Let us find the Thévenin equivalent of the circuit inside the dashed box. Applying the procedure:

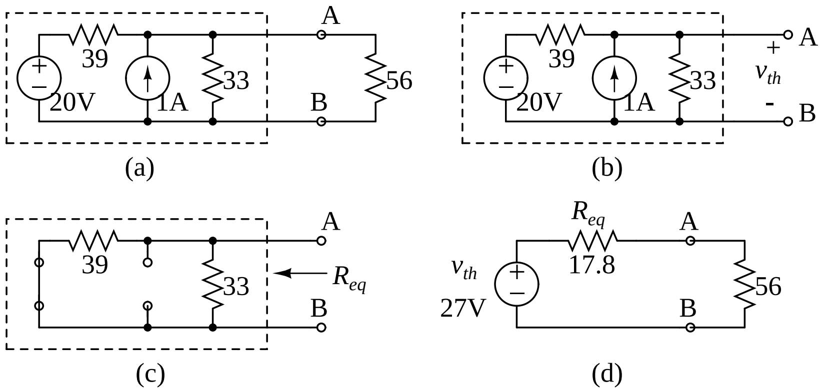



Figure 3.13: Example for Thévenin equivalent circuit

1. Using nodal analysis, we find the open-circuit voltage at terminals A-B with 56 Ω removed (Fig. 3.13(b)):

$$\frac{v}{33} + \frac{v-20}{39} = 1 \quad \text{or} \quad v_{th} = 27\text{V}$$

2. Kill the sources as in Fig. 3.13(c). Find the resistance between the terminals A and B (also with 56 Ω removed): $R_{eq} = 33 \parallel 39 = 33{\cdot}39/(33{+}39) = 17.8\Omega$.

The Thévenin equivalent circuit is shown in Fig. 3.13(d).

### 3.5.1 Thévenin equivalent circuit for $RLC$ networks with sinusoidal excitation

Thévenin's theorem is applicable to linear networks containing any number of resistances, capacitors and inductors as long as the sources are sinusoidal at the same frequency. In this case, the procedure is slightly modified.

For RLC networks with sinusoidal excitation, the Thévenin equivalent circuit is found by

1. Find the voltage phasor between the terminals A and B while nothing is connected to those terminals to determine the phasor $V_{th}(\omega)$.

2. Kill the voltage and current sources within the black-box: Short-circuit the voltage sources and open-circuit the current sources. Find the impedance between the terminals A and B to determine $Z_{eq}(\omega)$.

**Example 10**

Consider the RLC circuit given in Fig. 3.14(a). Since the excitation is sinusoidal, we can find the Thévenin equivalent circuit: Applying the procedure:

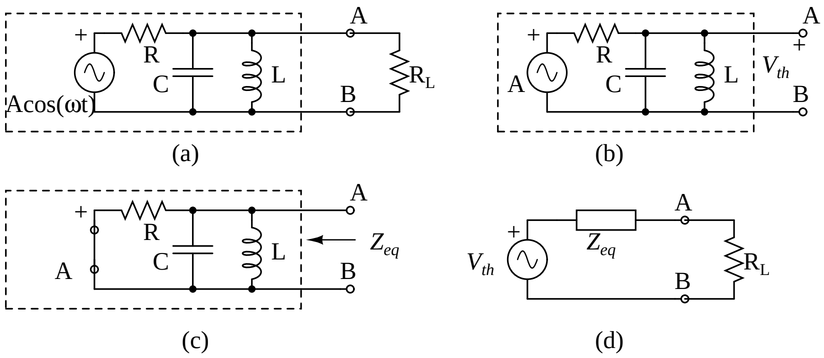



Figure 3.14: RLC circuit example for Thévenin equivalent circuit

1. Using nodal analysis, we find the open-circuit voltage phasor at terminals A-B with $R_L$ removed (Fig. 3.14(b)):

$$\frac{V}{j\omega L} + \frac{V}{1/(j\omega C)} + \frac{V-A}{R} = 0 \quad \text{or} \quad V_{th}(\omega) = \frac{j\omega L}{R - \omega^2 RLC + j\omega L} A$$

2. Kill the voltage source phasor as in Fig. 3.14(c). Find the impedance between the terminals A and B (also with $R_L$ removed):

$$Z_{eq}(\omega) = \frac{j\omega RL}{R - \omega^2 RLC + j\omega L}$$

The Thévenin equivalent circuit is shown in Fig. 3.14(d).

## 3.6 Norton Equivalent Circuit

Dual of the Thévenin equivalent circuit is the *Norton* equivalent circuit, named after American engineer Edward Lawry Norton (1898–1983). In this case, the straight line in $V - I$ characteristics is modelled by its slope and its intersection with the $I$ axis. The equivalent model consists of a current source, $i_N$ and a parallel resistance $R_{eq}$.

The procedure to find the Norton equivalent circuit of a black-box composed of any number of resistors/voltage and current sources is:

1. Find the current flowing between the terminals A and B while those terminals are short-circuited to determine $i_N$.

2. Kill the voltage and current sources within the black-box: Short-circuit the voltage sources and open-circuit the current sources. Find the resistance between the terminals A and B to determine $R_{eq}$.

Clearly, for the same black-box $R_{eq}$ is the same value for Thévenin and Norton equivalent circuits, because they are found in the same way. Since they represent the same straight line we also have $v_{th} = R_{eq} i_N$.

**Example 11**

Consider the same circuit example of Thévenin equivalent circuit in Fig. 3.15(a). Norton equivalent circuit is found by:

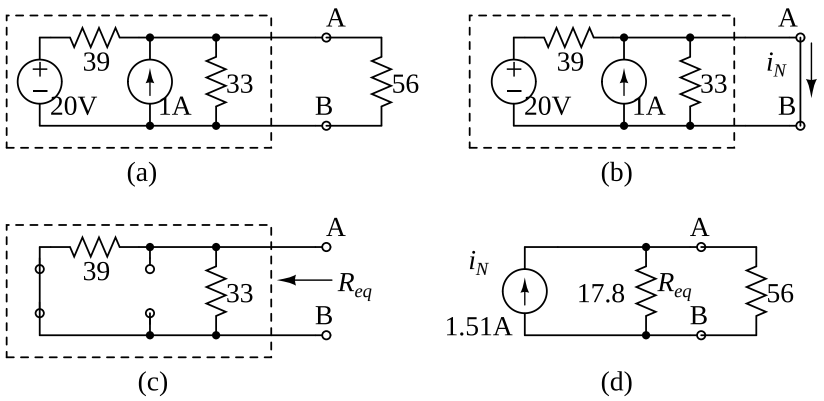



Figure 3.15: Finding the Norton equivalent circuit

1. Using nodal analysis, we find the short-circuit current between the terminals A-B (Fig. 3.15(b)):

$$i_N = 1 + \frac{20}{39} = 1.51\text{A}$$

2. Kill the sources as in Fig. 3.15(c). Find the resistance between the terminals A and B (with 56 Ω removed): $R_{eq} = 33 \parallel 39 = 33 \cdot 39/(33 + 39) = 17.8$.

The Norton equivalent circuit is shown in Fig. 3.15(d). For this example, finding the Norton equivalent circuit was simpler than finding the Thévenin equivalent. If there are parallel elements across the terminals A and B, Norton equivalent circuit should be preferred, since it gets rid of those elements when the terminals are shorted. On the other hand, if there are series elements at the terminals A or B, Thévenin should be preferred. The series elements will be eliminated when the terminals are open-circuited.

### 3.6.1 Norton equivalent circuit for $RLC$ networks with sinusoidal excitation

Norton method is also applicable to linear $RLC$ networks with sinusoidal excitation. The procedure is:

1. Find the short-circuit current phasor between the terminals A and B to determine the phasor $I_N(\omega)$.
2. Kill the voltage and current sources within the black-box: Short-circuit the voltage sources and open-circuit the current sources. Find the impedance between the terminals A and B to determine $Z_{eq}(\omega)$.

**Example 12**

Refer to the RLC circuit given in Fig. 3.16(a) considered earlier. We can find the Norton equivalent circuit as

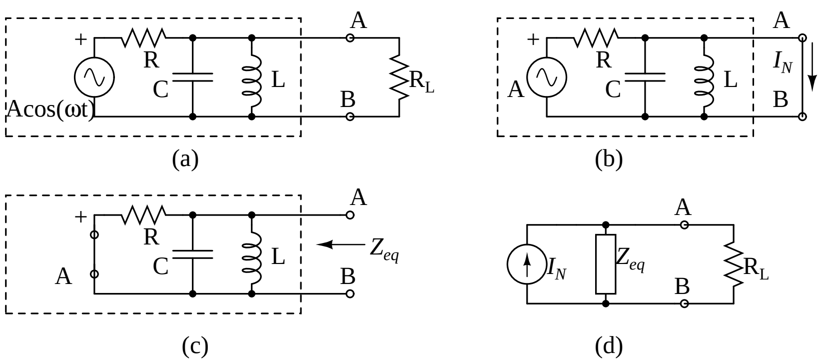



Figure 3.16: Finding the Norton equivalent circuit

1. Since $C$ and $L$ are shorted, we find the short-circuit current phasor at terminals A-B easily (Fig. 3.16(b)):
$$I_N = \frac{A}{R}$$
2. Kill the voltage source phasor as in Fig. 3.16(c). Find the impedance between the terminals A and B (with $R_L$ removed):
$$Z_{eq}(\omega) = \frac{j\omega RL}{R - \omega^2 RLC + j\omega L}$$

The Norton equivalent circuit is shown in Fig. 3.16(d).

### 3.6.2 Using Thévenin and Norton equivalent circuits

Thévenin and Norton equivalent circuits are used extensively in circuit analysis. These circuits provide a very efficient way to simplify the circuit to be analyzed. Whenever a piece of linear circuit is connected to another circuit through two terminals, the equivalent circuit analysis is often the simplest way to understand how the latter is affected. Consider the circuit in Fig. 3.17. As far as Circuit 2 is concerned, every effect of the first circuit is summarized by its equivalent circuit at the interconnection terminals. Once Thévenin equivalent circuit is

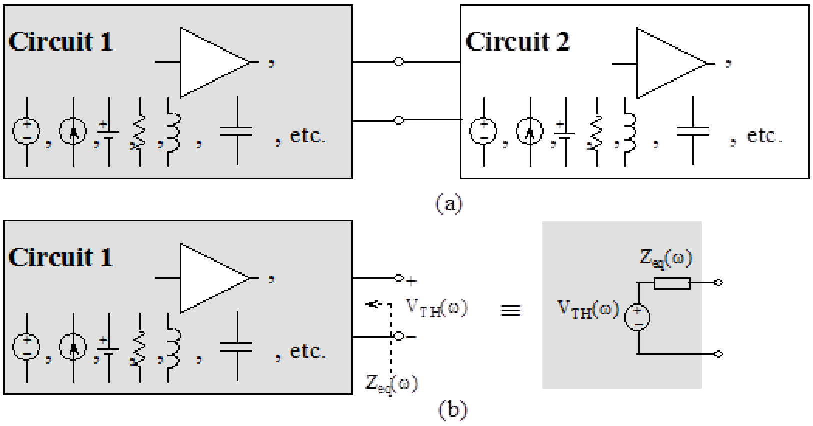



Figure 3.17: (a) Two circuit pieces connected to each other by means of two terminals, (b) Thévenin equivalent of Circuit 1.

obtained, Circuit 1 can be replaced by its equivalent as in Fig. 3.18 and Circuit 2 can be analyzed.

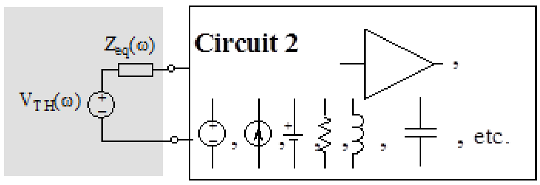



Figure 3.18: Thévenin equivalent of Circuit 1 connected to Circuit 2.

Thévenin and Norton equivalent circuits are equivalent. In other words, open circuit voltage of Norton circuit, $I_N Z_{eq}$, yields $V_{th}$, as discussed above. Hence, $Z_{eq}$ can also be obtained from $V_{th}$ and $I_N$:

$$Z_{eq}(\omega) = \frac{V_{th}}{I_N} \tag{3.47}$$

**Example 13**

Consider the circuit in Fig. 3.19. Voltage across 2.7 kΩ resistor can be found in many different ways.

One of the most effective ways of solving such problems is depicted in Fig. 3.20. First, the Thévenin equivalent of a part of the circuit, which contains the battery, is evaluated. Note that the equivalent voltage is the open circuit voltage across 1.8 kΩ resistor and the equivalent resistor is 1.2K‖1.8K. This equivalent circuit is connected to the remaining part and a Norton circuit is evaluated. To do this we disconnect the part of the circuit at the dotted line

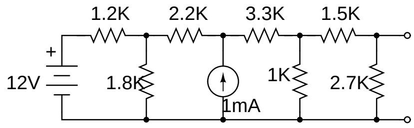



Figure 3.19: Example showing the use of Thévenin and Norton equivalent circuits.

and short it. The short circuit current is the equivalent current and the equivalent resistor is 0.72K+2.2K. In third step, we notice that the equivalent current and 1 mA current source are in parallel. Combining them in a single source of 3.47 mA, we convert the Norton circuit to its Thévenin equivalent. After another Thévenin equivalent conversion, output voltage is obtained as 0.75 V.

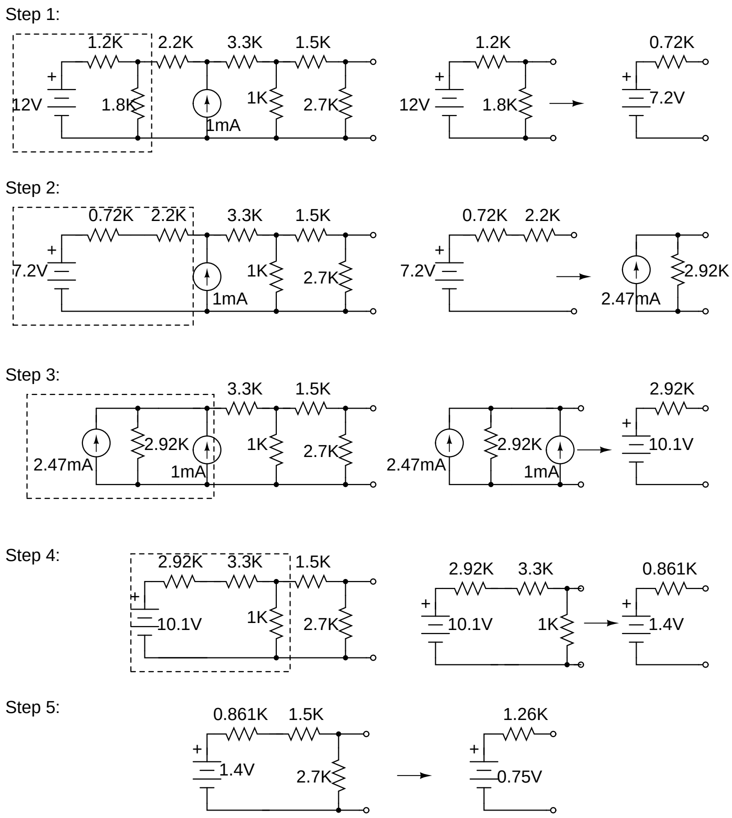



Figure 3.20: Analysis of circuits using equivalent circuits.

## 3.7 Superposition principle

*Superposition principle* is another very useful tool that can be used in the solution of linear circuits. If a circuit has more than one source, the effect of each source can be determined independently, and their responses can be added up to give the overall response. This is a direct result of linearity. For circuits containing an initial value, the initial value should also be considered as a source.

> The procedure for the superposition principle can be summarized as follows:
>
> 1. Kill all sources and initial value except one of them (Killing means that the voltage sources are short-circuited, the current sources are open-circuited and the initial value is zeroed). Find the desired response using an appropriate method.
> 2. Repeat step 1 for all sources and the initial value one-by-one.
> 3. Add the resulting responses to find the overall response.

**Example 14**

Refer to the resistive circuit depicted in Fig. 3.21(a). Use the superposition principle to find the voltage $v_A(t)$.

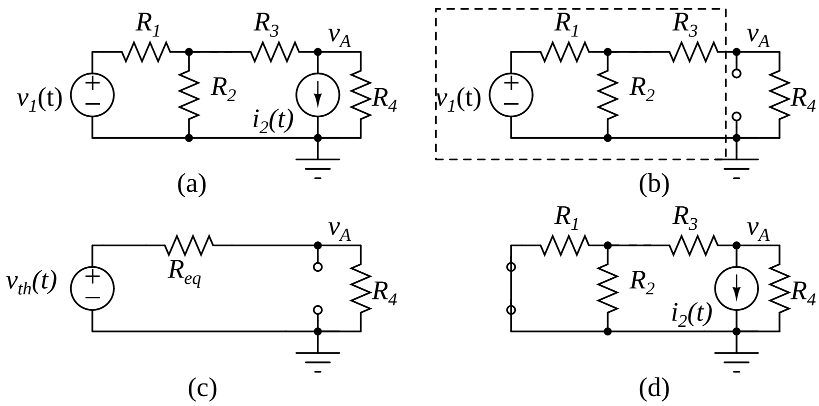



Figure 3.21: Example to use the superposition principle

**Solution**

1. Kill the current source as shown in Fig. 3.21(b). Find the Thévenin equivalent circuit of the circuit in the dashed box. We find $v_{th}$ using the voltage

divider formula (Eq. 2.30) and $R_{th}$ is found by parallel and series resistor combination formulas:

$$v_{th}(t) = \frac{R_2}{R_1 + R_2} v_1(t) \quad \text{and} \quad R_{eq} = \frac{R_1 R_2}{R_1 + R_2} + R_3$$

$v_{A1}(t)$ for the voltage source is found from the voltage divider in Fig. 3.21(c):

$$v_{A1}(t) = \frac{R_4}{R_{eq} + R_4} v_{th}(t) = \frac{R_4}{R_{eq} + R_4} \frac{R_2}{R_1 + R_2} v_1(t)$$

2. Kill the voltage source as in Fig. 3.21(d). We find the total resistance across the current source, $R_T$, and hence the corresponding $v_{A2}(t)$ as

$$R_T = R_{eq} \parallel R_4 = \frac{R_{eq} R_4}{R_{eq} + R_4} \quad \text{and} \quad v_{A2}(t) = -R_T i_2(t)$$

3. The total $v_A(t)$ is found by adding the two results:

$$v_A(t) = v_{A1} + v_{A2} = \frac{R_4}{R_{eq} + R_4} \frac{R_2}{R_1 + R_2} v_1(t) - R_T i_2(t)$$

As an exercise, solve the same circuit, using nodal analysis. As a third alternative, you can use Norton equivalent circuit for the part in dashed lines to get two current sources and two resistors in parallel which are easily combined.

---

**Example 15**

Consider the first-order circuit with two sources and with an initial value given in Fig. 3.22(a). Find the capacitor voltage, $v_C(t)$ using the superposition principle.

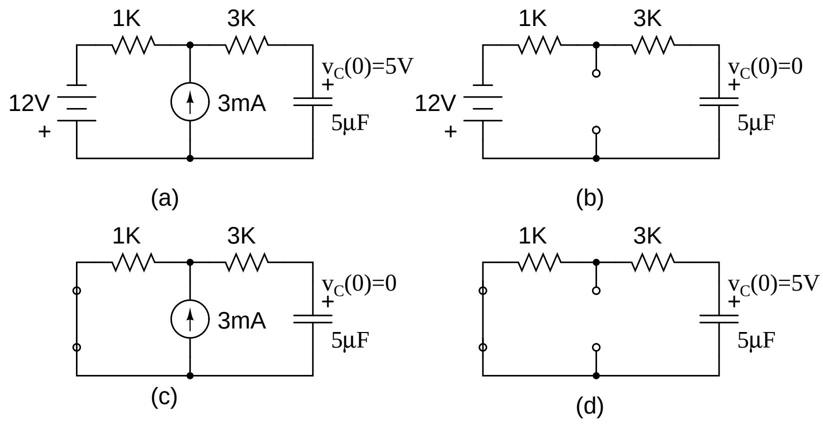



Figure 3.22: Another example to use the superposition principle

**Solution**

We use the superposition principle with two sources and one initial condition.

1. Kill the current source and the initial condition as shown in Fig. 3.22(b). The final value of the capacitor voltage is $v_f$=−12 V. The time constant is $\tau$=(5$\mu$)(1K+3K)=20 ms. The solution for this case is

$$v_c(t) = -12 + (0 - (-12))e^{-t/20m}$$

2. Kill the voltage source and the initial condition as depicted in Fig. 3.22(c). The final value of the capacitor voltage is determined by the 3 mA current flowing in 1K resistor: $v_f$=3 V. The time constant, $\tau$, remains the same. The corresponding solution is

$$v_c(t) = 3 + (0 - 3)e^{-t/20m}$$

3. Kill the voltage source and current source as shown in Fig. 3.22(d). The final value of the capacitor voltage is zero. The time constant is the same. Hence we have

$$v_c(t) = 0 + (5 - 0)e^{-t/20m}$$

4. We add the three equations above to find the requested solution:

$$v_c(t) = -9 + 14e^{-t/20m}$$

---

**Example 16**

Consider the circuit in Fig. 3.23(a) driven by sinusoidal sources. The two sources are at two *different* frequencies, $\omega_1 = 10^3/3$ and $\omega_2 = 10^3$: $v_1(t) = 2\cos(10^3/3t + \pi/6)$ and $i_2(t) = 5\cos(10^3 - \pi/4)$. Find the *steady-state* value of the resistor current, $i_3(t)$ using the superposition principle.

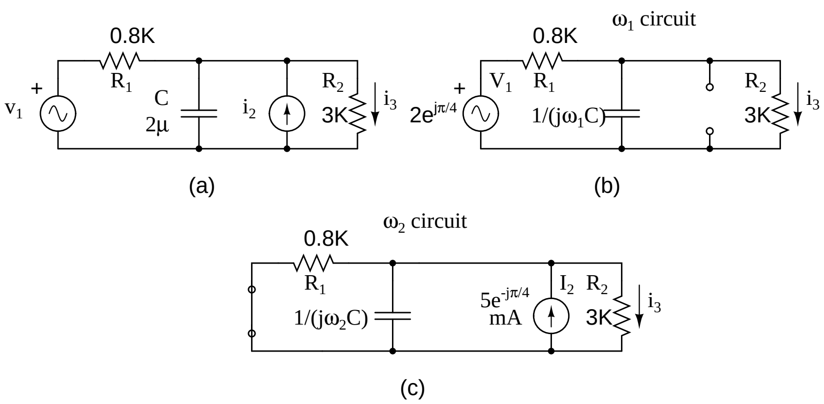



Figure 3.23: Phasor example using the superposition principle

**Solution**

Since the sources are sinusoidal and we need the steady-state solution, we can use phasors.

1. Kill the current source and replace the circuit with phasor equivalents at $\omega_1 = 10^3/3$ as depicted in Fig. 3.23(b). We find the resistor voltage using the voltage divider formula. Then the current is found:

$$I_3 = V_1 \frac{R_2 \parallel (1/j\omega_1 C)}{R_1 + (R_2 \parallel (1/j\omega_1 C)} \frac{1}{R_2} = 0.485e^{j0.125} \text{ mA}$$

2. Kill the voltage source and replace the circuit with phasor equivalents at $\omega_2 = 10^3$ as depicted in Fig. 3.23(c). We find the resistor current using the current divider formula.

$$I_3 = I_2 \frac{R_1 \parallel (1/j\omega_2 C)}{R_2 + (R_1 \parallel (1/j\omega_2 C)} = 0.653e^{-j1.687} \text{ mA}$$

3. We convert the phasor to time domain equivalents and add them to find the steady-state solution of the resistor current

$$i_3(t) = 0.485 \cos(10^3/3t + 0.125) + 0.653 \cos(10^3 - 1.687)$$

---

## 3.8 Amplifier types

Since the voltage and current are the two independent variables of circuits, we can have four possible amplifiers:

1. Voltage amplifier: It amplifies the input voltage to generate an output voltage as shown in Fig. 3.24(a). An ideal voltage amplifier has an infinite input impedance and a zero output impedance. $A$ (a unitless quantity) is the voltage gain of the amplifier.

2. Transimpedance amplifier: It amplifies the input current to generate an output voltage (Fig. 3.24(b)). An ideal transimpedance amplifier has a zero input impedance and a zero output impedance. $R_f$ with the unit of $\Omega$ is known as the *transimpedance* of the amplifier.

3. Transconductance amplifier: It amplifies the input voltage to generate an output current (Fig. 3.24(c)). An ideal transconductance amplifier has an infinite input impedance and an infinite output impedance. $G_f$ (with the unit of Siemens, S) is known as the *transconductance* of the amplifier.

4. Current amplifier: It amplifies the input current to generate an output current (Fig. 3.24(d)). An ideal current amplifier has a zero input impedance and an infinite output impedance. $B$, a unitless quantity, is the current gain of the amplifier.

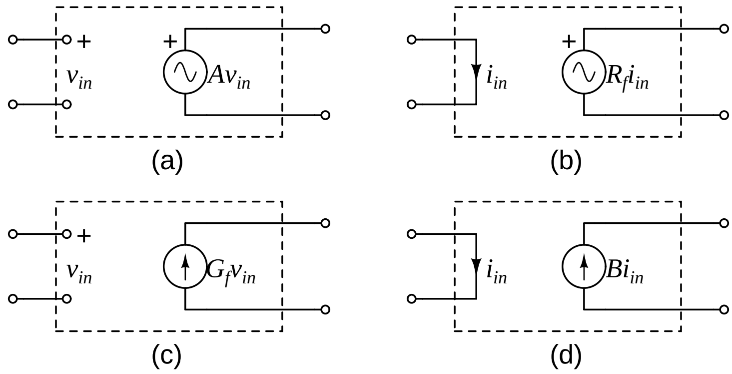



Figure 3.24: (a) Voltage amplifier, (b) transimpedance amplifier, (c) transconductance amplifier, and (d) current amplifier.

## 3.9 Operational amplifiers

*Operational amplifiers* (OPAMP) are versatile building blocks [7] very frequently used in electronic circuits. They can be used to obtain a large variety of functions. The symbol of an OPAMP is shown in Fig. 3.25. $v_1$ and $v_2$ denotes the voltages at + and − inputs of the OPAMP. $+V_C$ and $-V_C$ represent the positive and negative supply voltages.

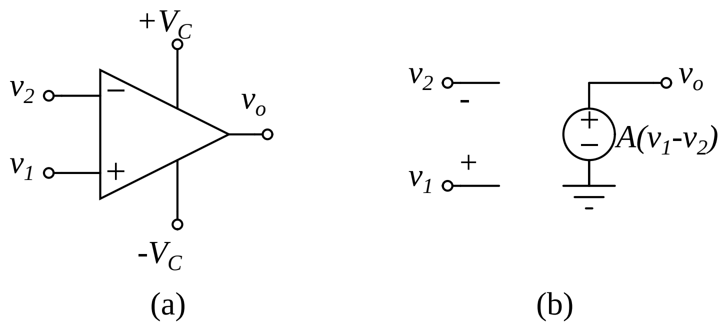



Figure 3.25: (a) OPAMP symbol, (b) an ideal OPAMP equivalent circuit

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 ($A \rightarrow \infty$). The infinite input impedance means that there is no current flowing into the OPAMP. The series output impedance of an ideal OPAMP is zero. The equivalent circuit of an ideal OPAMP is shown in Fig. 3.25(b).

In the ideal OPAMP equivalent circuit, there is an output voltage source with the value of $A(v_1 - v_2)$. Such a source is called a controlled voltage source, because its value is determined by some parameter in the circuit, $v_1 - v_2$ in this case.

All real OPAMPs have a maximum, $V_{max}$, and minimum, $V_{min}$ voltage limit at its output. Typically, $V_{max}$ is slightly smaller than the positive supply voltage $+V_C$, and $V_{min}$ is slightly greater then the negative supply voltage $-V_C$.

We can express the characteristics of a real OPAMP as follows:

$$v_o = \begin{cases} A(v_1 - v_2) & \text{if } \frac{V_{min}}{A} \le v_{in1} - v_{in2} \le \frac{V_{max}}{A} \\ V_{max} & \text{if } v_1 - v_2 > \frac{V_{max}}{A} \\ V_{min} & \text{if } v_1 - v_2 < \frac{V_{min}}{A} \end{cases} \tag{3.48}$$

Typically, $A$ is a large number in the order of $10^5$ or $10^6$. This implies that as long as $V_{min}/A \le v_1 - v_2 \le V_{max}/A$ is satisfied $v_1 - v_2$ is a very small number. Hence we can write

$$v_1 \approx v_2 \quad \text{if} \quad V_{min} \le v_o \le V_{max} \tag{3.49}$$

This approximation simplifies the solution of most OPAMP circuits. In the solution of OPAMP circuits, the output node $v_o$ should be considered as a voltage source, which may provide the needed current. It is fair to assume that there is no current going into the OPAMP at the pins $v_1$ and $v_2$.

### 3.9.1 Inverting amplifier

Consider the inverting voltage amplifier configuration shown in Fig. 3.26(a). We can analyze this circuit assuming $V_{min} \le v_o \le V_{max}$. In this case we have

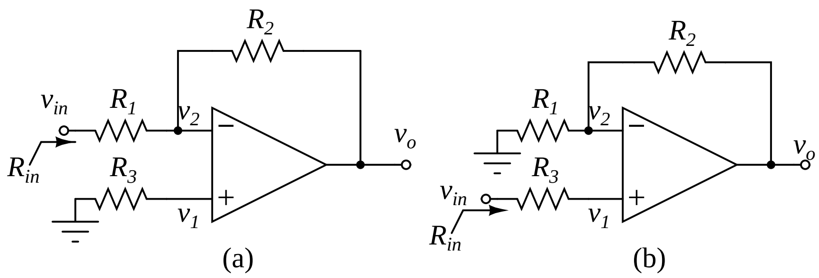



Figure 3.26: (a) Inverting amplifier, (b) Non-inverting amplifier

$v_1 \approx v_2$. Since there is no current through $R_3$ we have $v_1 = 0$. Hence $v_2 = 0$ also. All the current through $R_1$ must flow through $R_2$. We determine the voltage gain, $v_o/v_{in}$, as

$$\frac{v_{in} - v_2}{R_1} = \frac{v_2 - v_0}{R_2} \quad \text{or} \quad \frac{v_o}{v_{in}} \approx -\frac{R_2}{R_1} \tag{3.50}$$

A negative gain value means that the polarity of the output voltage is the inverse of the input voltage. The ratio of the resistors $R_2$ and $R_1$ determine the gain value. $R_3$ has no effect on the gain expression. The input impedance, $R_{in}$, of the amplifier is equal to $R_1$, since $v_2$ is at ground potential. For audio amplifiers, $R_1$ is chosen typically in 1k$\Omega$ to 100 k$\Omega$ range. Even though $R_3$ is not in the gain expression, it is recommended that $R_3$ is chosen equal to the parallel combination of $R_1$ and $R_2$ ($R_1R_2/(R_1+R_2)$) to assure symmetry.

Note that in OPAMP amplifier circuits we provide feedback always to − input of OPAMP. This type of feedback is called *negative feedback*. If we connect the feedback resistor to the + input of the OPAMP, we have a *positive* feedback. With a positive feedback, the output voltage is not stable and can not remain within the range $V_{min} \leq v_o \leq V_{max}$: It is either $V_{max}$ or $V_{min}$.

### 3.9.2 Non-inverting amplifier

Now, refer to the non-inverting voltage amplifier shown in Fig. 3.26(b). Again, we assume $V_{min} \leq v_o \leq V_{max}$. Hence we have $v_1 \approx v_2$. Since there is no current through $R_3$ we have $v_1 = v_{in}$. Hence $v_2 \approx v_{in}$ also. All the current through $R_1$ must flow through $R_2$. We find the voltage gain, $v_o/v_{in}$, as

$$\frac{v_2}{R_1} = \frac{v_o - v_2}{R_2} \quad \text{or} \quad A = \frac{v_o}{v_{in}} \approx 1 + \frac{R_2}{R_1} \tag{3.51}$$

The input impedance, $R_{in}$, of this amplifier is very high (infinity for an ideal OPAMP), since no current flows through $R_3$. Similar to the inverting amplifier, the value of $R_1$ should be chosen in the range 1 kΩ to 100 kΩ.

If we choose $R_2 = 0$ and/or $R_1 = \infty$, then the gain becomes one. Such a circuit is called *unity gain amplifier* or *voltage follower*, and it is commonly used as a *buffer*. 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.

### 3.9.3 Summing amplifier

OPAMPs can be used to add two or more signals. A schematic diagram of a *summing amplifier* is depicted in Fig. 3.27(a). We assume $V_{min} \leq v_o \leq V_{max}$, and use $v_1 \approx v_2$. Since there is no current in $R_4$, there is zero voltage across it. Hence $v_1 = 0$. Writing KCL at node $v_2$:

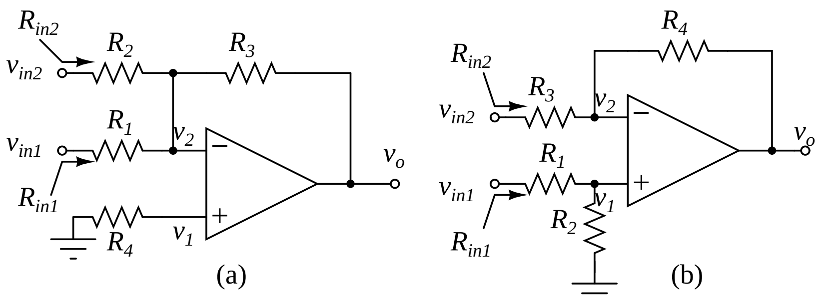



Figure 3.27: (a) Summing amplifier, (b) Difference amplifier

$$\frac{v_2 - v_{in1}}{R_1} + \frac{v_2 - v_{in2}}{R_2} + \frac{v_2 - v_o}{R_3} = 0$$

Since $v_2 \approx v_1 = 0$, we find

$$v_o = -\left(\frac{R_3}{R_1} v_{in1} + \frac{R_3}{R_2} v_{in2}\right) \tag{3.52}$$

If $R_1 = R_2 = R_3$, we get the sum of input signals with inverted polarity: $v_o = -(v_{in1} + v_{in2})$.

The input impedances are given by $R_{in1} = R_1$ and $R_{in2} = R_2$, since $v_2$=0.

**Exercise**

Design an OPAMP circuit to add four signals (with inverted polarity).

### 3.9.4 Difference amplifier

Referring to the *difference amplifier* shown in Fig. 3.27(b), we assume again $V_{min} \leq v_o \leq V_{max}$ and $v_1 \approx v_2$. Using the voltage divider relation, we write

$$v_1 = \frac{R_2}{R_1 + R_2} v_{in1} \quad \text{and} \quad v_2 = \frac{R_4}{R_3 + R_4} v_{in2} + \frac{R_3}{R_3 + R_4} v_o$$

In the second equation, we used the superposition principle for signals $v_{in2}$ and $v_o$. Since $v_1 \approx v_2$, we get

$$v_o = \frac{R_2}{R_1 + R_2} \frac{R_3 + R_4}{R_3} v_{in1} - \frac{R_4}{R_3} v_{in2} \tag{3.53}$$

The input impedances are $R_{in1} = R_1 + R_2$ and $R_{in2} = R_3$ (if $v_{in1} = 0$). Note that $R_{in2}$ depends on $v_{in1}$, as $v_2$ is determined by $v_{in1}$.

Note that if $R_3$=$R_1$+$R_2$, and $R_4$=$R_2$+$R_2^2/R_1$, we have

$$v_o = \frac{R_2}{R_1}(v_{in1} - v_{in2}) \tag{3.54}$$

resulting in a difference amplifier with a gain of $R_2/R_1$ and equal input impedances of $R_{in1}$=$R_{in2}$=$R_1 + R_2$.

**Example 17**

Design a circuit with the transfer characteristics $v_o = 5v_{in} + 3$.

**Solution**

Use a difference amplifier and choose $v_{in1} = v_{in}$, $R_1 = R_2$, $R_4/R_3 = 9$ and $v_{in2} = -1/3$V.

---

### 3.9.5 Transimpedance amplifier

The circuit shown in Fig. 3.28 is a transimpedance amplifier (see page 104) with a current input, $i_{in}$, and a voltage output, $v_o$. If the output voltage is not saturated, we have $v_1 = v_2 = 0$. Hence, the input impedance of this amplifier is zero as ideally needed in a transimpedance amplifier. The output voltage is determined by the feedback resistance, $R_f$: $v_o = -R_f i_{in}$. Therefore, $-R_f$ is the transimpedance of this amplifier.

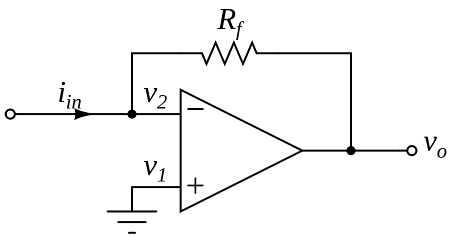



Figure 3.28: Transimpedance amplifier

### 3.9.6 Current source

Consider to OPAMP circuit shown in Fig. 3.29. Let us find the current $I_L$ with the condition $R_4 = R_1R_3/R_2$.

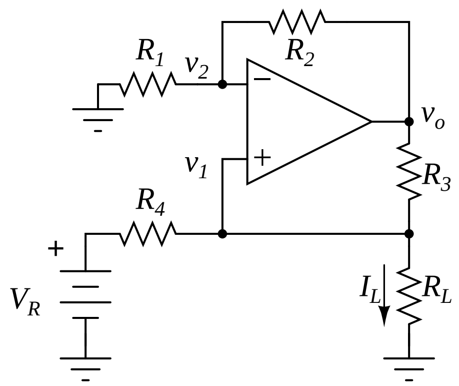



Figure 3.29: Current source

Using the voltage divider relation, we can find the voltage $v_2$ in terms of $v_o$:

$$v_2 = \frac{R_1}{R_1 + R_2} v_o \tag{3.55}$$

Using the node equation at node $v_1$;

$$\frac{v_1 - V_R}{R_4} + \frac{v_1 - v_o}{R_3} + I_L = 0 \tag{3.56}$$

If $v_o$ is not saturated, we have $v_1 = v_2$. Combining these equations, we find

$$\left[\frac{R_2}{R_1 + R_2}\left(\frac{1}{R_3} + \frac{1}{R_4}\right) - \frac{1}{R_3}\right] v_o - \frac{V_R}{R_4} + I_L = 0 \tag{3.57}$$

With the condition $R_4 = R_1R_3/R_2$, the term inside the square brackets vanishes, and we find

$$I_L = \frac{V_R}{R_4} \tag{3.58}$$

$I_L$ is independent of $R_L$, so the resulting circuit acts like a current source.

### 3.9.7 Integrator

An OPAMP can be used to build an *integrator* as depicted in Fig. 3.30(a). We observe that $v_1 = 0$, since there is no current in $R_2$. The current in $R_1$ is equal

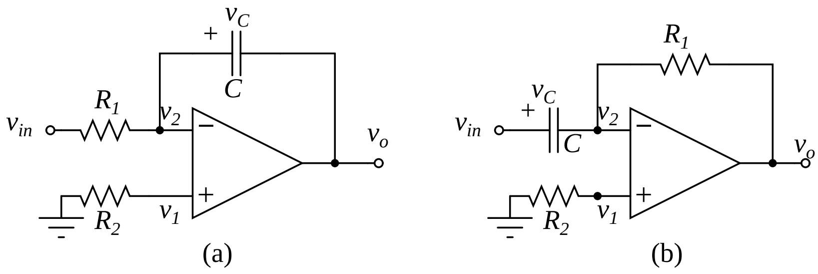



Figure 3.30: (a) Integrator, (b) Differentiator

to the current in $C$:

$$\frac{v_{in} - v_2}{R_1} = C\frac{dv_C}{dt} = C\frac{d(v_2 - v_o)}{dt}$$

If the output voltage, $v_o$, is not saturated ($V_{min} \leq v_o \leq V_{max}$), we can use $v_2 \approx v_1 = 0$. We integrate both sides of the equation above from 0 to $t$ to get

$$v_0(t) = v_o(0) - \frac{1}{R_1 C}\int_0^t v_{in}(\tau)d\tau \tag{3.59}$$

where $v_o(0)$ is the output voltage at $t = 0$.

**Example 18**

Assume that $R_1 = 1$K, $C = 10\mu$F, $v_o(0) = 3$V and $v_{in}$ is a 5 V pulse waveform as shown in Fig. 3.31(a). We find the output voltage as plotted in Fig. 3.31(b).

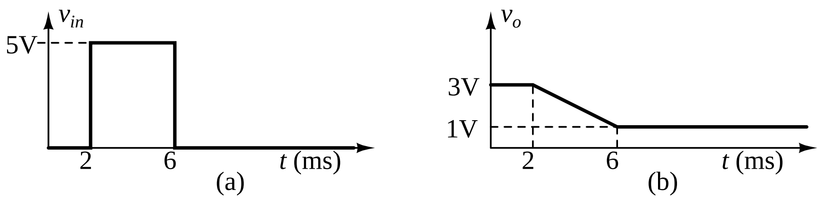



Figure 3.31: (a) Input signal, $v_{in}$, of the integrator, (b) Output signal, $v_o$, of the integrator

---

### 3.9.8 Differentiator

A differentiator is built from an OPAMP as drawn in Fig. 3.30(b). If the output is not saturated, we have $v_2 = 0$ since $v_1 = 0$. The current in $C$ is equal to the current in $R_1$:

$$C\frac{d(v_{in}-v_2)}{dt} = C\frac{dv_{in}}{dt} = \frac{v_2 - v_o}{R_1} = -\frac{v_o}{R_1}$$

Hence

$$v_0 = -R_1 C\frac{dv_{in}}{dt} \tag{3.60}$$

### 3.9.9 First-order low-pass-filter (LPF)

A low-pass-filter passes the signals with frequencies less than a predetermined value, but it attenuates signals with frequencies higher than that value. It is possible to build a first-order low-pass-filter using an OPAMP by modifying the inverting amplifier. Refer to Fig. 3.32(a). For analysis, we use phasors

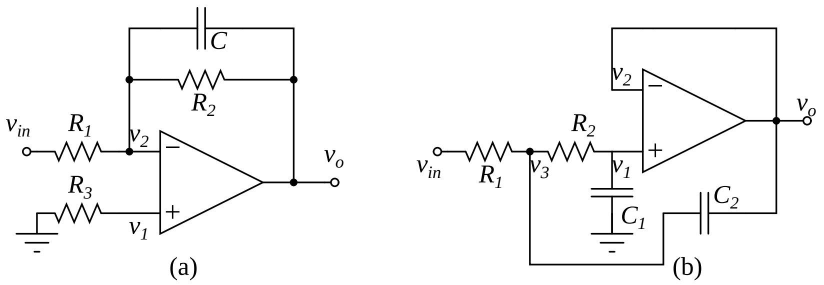



Figure 3.32: (a) Inverting first-order low-pass-filter amplifier, (b) Non-inverting second-order low-pass-filter amplifier

(capital letters) assuming that the input signals are sinusoidal. We know that $V_2 \approx V_1 = 0$. Hence, we write

$$\frac{V_{in}}{R_1} = \frac{-V_o}{R_2} - V_o j\omega C$$

Therefore, the transfer function is given by

$$H(\omega) = \frac{V_o(\omega)}{V_{in(\omega)}} = -\frac{R_2}{R_1}\frac{1}{1 + j\omega R_2 C} \tag{3.61}$$

When $\omega R_2 C = 1$, the magnitude of low-pass-filter term becomes $1/\sqrt{2}$. This frequency,

$$\omega_0 = \frac{1}{R_2 C} \quad \text{or} \quad f_0 = \frac{1}{2\pi R_2 C} \tag{3.62}$$

is known as the *corner frequency* or *cutoff frequency*. In decibels the magnitude is $20\log_{10}(1/\sqrt{2}) = -3$ dB.

The magnitude of the transfer function in decibels is

$$\left|\frac{V_0}{V_{in}}\right|_{dB} = 20\log_{10}\left(\frac{R_2}{R_1}\right) - 10\log_{10}(1 + (\omega/\omega_0)^2) \tag{3.63}$$

The first term is the dB gain of the inverting amplifier. The second term is the low-pass-filter term. We plot this transfer function in Fig. 3.33 for $R_2 = 1$K

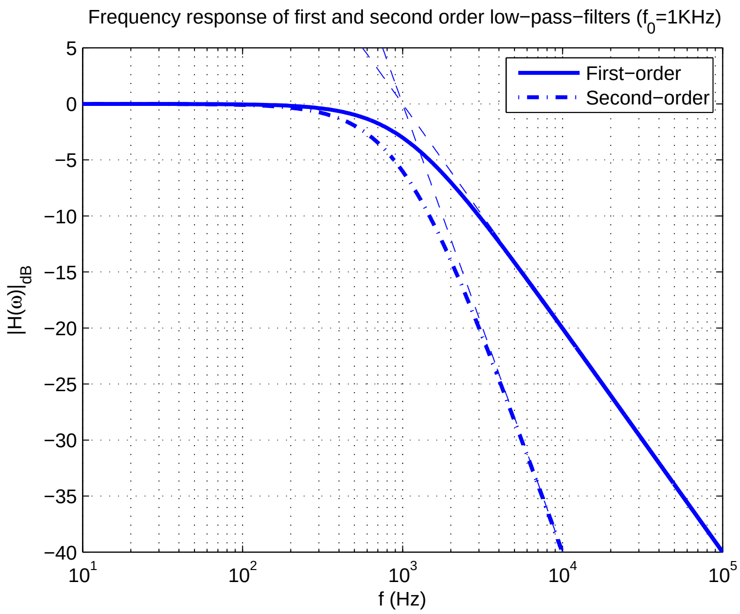



Figure 3.33: Frequency response of first and second-order low-pass-filters

and $C = 0.159\mu$F. For low frequencies when $\omega \ll \omega_0$, the low-pass-filter term approaches to 1 ($20\log_{10}(1) = 0$ dB). On the other hand, at frequencies much higher than the corner frequency, $\omega \gg w_0$, we can assume that $1 + j(\omega/\omega_0) \approx j\omega/\omega_0$. So the magnitude of low-pass-filter term in decibels is

$$\left|\frac{V_0}{V_{in}}\right|_{dB} \approx 20\log_{10}\left(\frac{R_2}{R_1}\right) - 20\log_{10}(\omega/\omega_0) = 20\log_{10}\left(\frac{R_2}{R_1}\right) - 20\log_{10}(f/f_0) \tag{3.64}$$

For $f = 10f_0$, the last term is -20 dB. For $f = 100f_0$, the last term becomes -40 dB. Clearly, every *decade* (factor of 10) increase in frequency causes an additional loss of 20 dB. We express this asymptotic behavior with a slope of $-20$ dB/decade, very convenient notation for logarithmic plots like the graph in Fig. 3.33. The slope can also be expressed in terms of *octaves*. One octave is a factor of two. For $f = 2f_0$, the low-pass-filter term is $-6$ dB. Hence, the slope is $-6$ dB/octave. In Fig. 3.32 the asymptotic line with the slope $-20$ dB/decade (or $-6$ dB/octave) is shown as a dashed line.

### 3.9.10 Second-order low-pass-filter

A second-order low-pass-filter can be obtained using the circuit in Fig. 3.32(b). We perform nodal analysis in phasor domain as follows:

$$\frac{V_3 - V_{in}}{R_1} + \frac{V_3 - V_1}{R_2} + (V_3 - V_o)j\omega C_2 = 0$$

and

$$\frac{V_3 - V_1}{R_2} = V_1 j\omega C_1$$

Since $V_1 \approx V_2 = V_o$, we find the transfer function as

$$\frac{V_o}{V_{in}} = \frac{1}{1 + j\omega(R_1 + R_2)C_1 - \omega^2 R_1 R_2 C_1 C_2} \tag{3.65}$$

We will show in p. 227 that the shape of the response becomes desirable if

$$(R_1 + R_2)^2 C_1 = 2R_1 R_2 C_2 \tag{3.66}$$

Fig. 3.33 also shows the magnitude of the transfer function for this filter for $R_1 = R_2 = 1\text{K}$ and $C_1 = C_2 = 0.159\mu\text{F}$. It has a higher slope ($-40$ dB/decade) above the corner frequency.

## 3.10 Microphone

Microphone is a device that converts sound into an electrical signal. Sometimes, it is abbreviated as mike. Many different types of microphones are available.

- Dynamic microphone: The diaphragm of the microphone is attached to a small movable induction coil, which is positioned in the magnetic field of a permanent magnet. When the sound wave moves the diaphragm, the coil moves in the magnetic field, producing a varying current in the coil through electromagnetic induction. This type of microphone has a low electrical impedance.

- Crystal or piezoelectric microphone: The diaphragm of the microphone applies a pressure to a piezoelectric crystal, which creates an electrical voltage proportional to the applied pressure. This type of of microphone has a high electrical impedance. They are commonly used in electrical guitars, directly contacting the vibrating surfaces.

- Condenser microphone: The basic condenser microphone is a parallel plate capacitor with one of the plates made from a very thin diaphragm (typically metallized mylar film) while the other one is a thick metal plate. A cross-section of this type of microphone is shown in Fig. 3.34 along with its equivalent circuit.

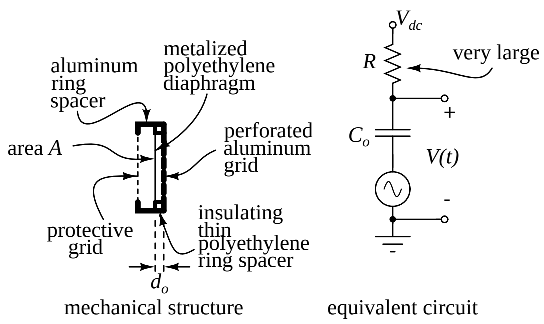



Figure 3.34: Condenser microphone and its equivalent circuit with a bias voltage.

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

$$C_o = \frac{\epsilon_o A}{d_o} \tag{3.67}$$

where $\epsilon_o$ is the permittivity of the free space given by $8.854 \times 10^{-12}$ F/m. $A$ is the area of the plates and $d_o$ is the spacing of the plates. These microphones are used with circuits like the one shown in Fig. 3.34. $C_o$ is charged to a voltage of $V_{dc}$ through a very large resistor $R$. From Eq. 2.38 on page 39, the electric charge, $Q$, between the plates is

$$Q = C_o V_{dc} \tag{3.68}$$

When there is sound incident on the thin diaphragm, sound pressure forces the diaphragm to vibrate back and forth. Suppose that the spacing is large enough so that the electrostatic attraction force between the capacitor plates is negligible compared to the force due to the sound pressure. With that assumption, the spacing of the capacitor plates varies linearly as a function of time: $d(t) = d_o \pm \Delta(t)$, where $\Delta(t)$ is proportional to the sound signal amplitude. Hence, the value of the capacitor becomes also a function of time

$$C(t) = \frac{\epsilon_o A}{d(t)} = \frac{\epsilon_o A}{d_o \pm \Delta(t)} = \frac{C_o d_o}{d_o \pm \Delta(t)} \tag{3.69}$$

Since the resistor $R$ has a large value, the total charge $Q$ on the capacitor remains the same. Therefore, the voltage of the capacitor must be a function of time:

$$V(t) = \frac{Q}{C(t)} = \frac{C_o V_{dc}}{C_o d_o}(d_o \pm \Delta(t)) = V_{dc}\left(1 \pm \frac{\Delta(t)}{d_o}\right) \tag{3.70}$$

The sound is thus converted to the electrical signal $V_{dc}\Delta(t)/d_o$ in a linear manner. For ordinary sound levels, the variation in the membrane displacement $|\Delta(t)|_{max}$ is a small fraction of the gap, $d_o$. Note that the sensitivity of a microphone depends on the size of $d_o$: smaller $d_o$ is, more sensitive the microphone becomes. To preserve linearity, the electrostatic force between the plates should be negligible, hence the capacitor plate separation should not be very small. In most commercial microphones, a compromise between these two requirements is found at about $d_o = 25$ $\mu$m. The expression for $V(t)$ in Eq. 3.70 tells that there is a voltage source $V_{dc}\Delta(t)/d_o$ additive to the capacitor (i.e. in series with the capacitor). Fig. 3.34 also shows the overall equivalent circuit.

The microphones must be connected to amplifiers using well-shielded (sometimes double-shielded) cables to prevent *hum*. Hum is the unwanted low-frequency sound caused by power-line frequencies (50 or 60 Hz) in audio systems. Unshielded cables cause power-line frequencies be capacitively coupled to the sensitive input an amplifier causing an unpleasant noise.

For a microphone with a surface of $A = 1$ cm$^2$, and a gap of $d_o = 25$ $\mu$m, $C_o$ is approximately 35 pF. The capacitance of a typical shielded cable used in audio work is at least 40 pF/m. If we connect the microphone to the amplifier by a cable of capacitance 35 pF, the sensitivity will be nearly halved (see Problem 29). Moreover, the perfectly linear relationship between the sound level and output voltage will be lost. Therefore, a very high input impedance and low capacitance buffer amplifier is always used within the housing of the microphone to isolate the capacitor $C_o$ 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.

- Electret microphone: Microphones used with sound cards of computers and cellular phones, tie-clip microphones, amateur video camera microphones are all electret type. An electret microphone is a special type of condenser microphone. In these microphones, a polymer membrane replaces the vibrating membrane, which is precharged. Polyvinyl fluoride (PVF) polymers (similar to teflon) have a property of keeping a static charge 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 $\mu$m thick, and metal plated (aluminum, nickel or gold) by a technique called *sputtering.* The structure and equivalent circuit of an electret microphone is given in Fig. 3.35.

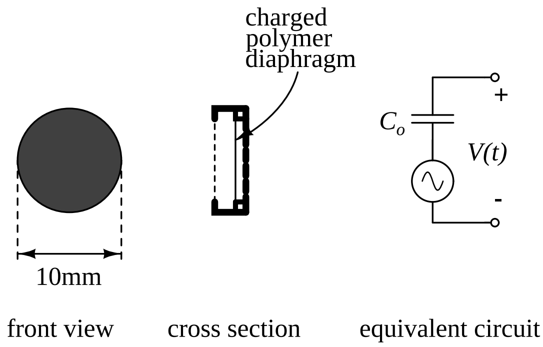



Figure 3.35: Electret condenser microphone

Since the polymer membrane is charged, we do not have to provide an external dc supply to make the microphone work. The electric charge trapped in the membrane, $Q$, is equivalent to $C_oV_{dc}$ in condenser type microphone. This charge enables the microphone to produce the equivalent electrical signal, in the absence of any external voltage. This type of microphone also suffers from capacitive output impedance drawback, and hence a buffer amplifier must be used.

Electret microphones are commercially available as capsules, which contain the microphone, the buffer and internal wiring. The electrical model of such a microphone and its equivalent circuit in is given in Fig. 3.36.

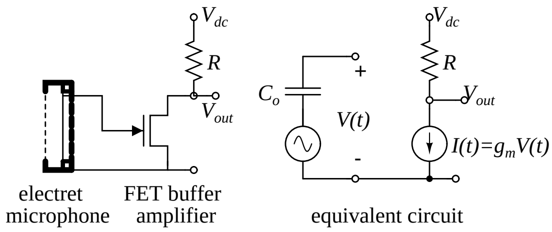



Figure 3.36: Buffered electret condenser microphone and its equivalent circuit.

The microphone housing contains the microphone and a *field effect transistor* (FET), which is used as a buffer amplifier. 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 $V_{dc}$ through a resistor $R$ as shown in the Fig. 3.36 in order to act as a buffer amplifier. FET is a semiconductor device and acts like a *voltage controlled current source*. $V(t)$ is the voltage produced by the microphone proportional to the sound pressure, and it is the controlling voltage for the current source. FET converts its voltage input into a current source output $g_mV(t)$, where $g_m$ is a parameter of FET, called *transconductance*. The terminals of the microphone are these two terminals of the FET. We will 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.

⧫ TRC-11 uses an electret microphone as its audio input.

- Microelectromechanical (MEMS) microphone: It is basically a type of condenser microphone built on silicon. The thin diaphragm of the microphone is manufactured on silicon using MEMS processing technology. Silicon diaphragm has holes to equalize the gap pressure with the external pressure. An integrated buffer amplifier is used for the same reason as condenser microphones. Modern smart phones have several MEMS microphones.

## 3.11 Loudspeaker

A loudspeaker is an electro-acoustic transducer [8] that converts electrical energy into acoustic energy. Unlike electromagnetic energy, acoustic energy requires a presence of matter in the medium to propagate (acoustic signals doe not propagate in vacuum). 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 at the frequency of the signal and at an amplitude proportional to the signal amplitude. The structure of an ordinary loudspeaker is given in Fig. 3.37.

Loudspeakers most commonly have a circular symmetry. The cone section in Fig. 3.37 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

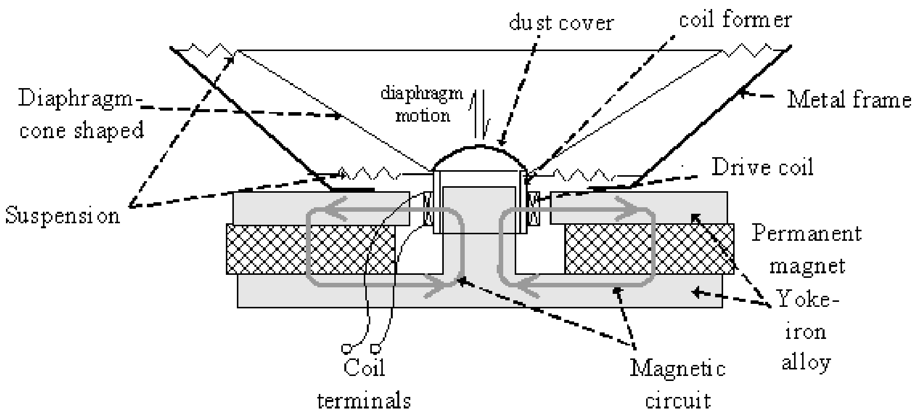



Figure 3.37: Parts of a loudspeaker

by corrugated suspension, both at top and at bottom. This support allows the diaphragm to move easily, but up and down only.

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 Fig. 3.38.

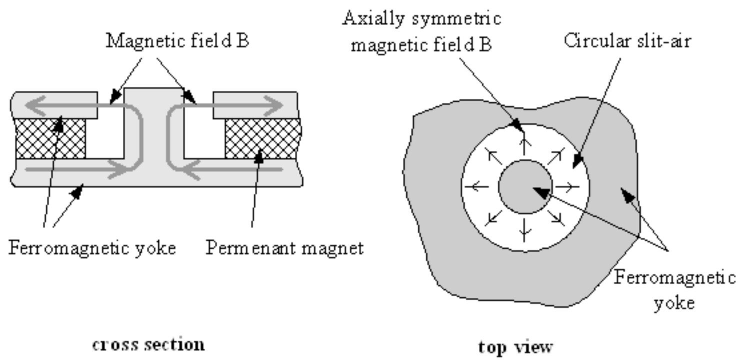



Figure 3.38: Magnetic circuit of a loudspeaker

The source of the magnetic field is the permanent magnet, whose North-South 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 Fig. 3.39.

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

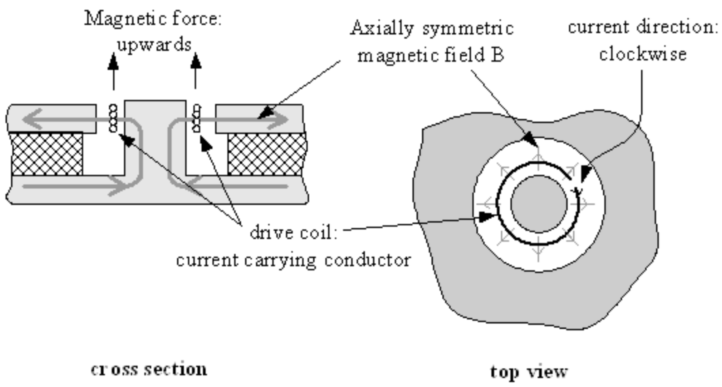



Figure 3.39: Current carrying coil in a loudspeaker

$$F = NIB \tag{3.71}$$

where $I$ is the current in the coil, $B$ is the magnetic field in Tesla 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.

Loudspeakers are specified by their input resistance. 4 Ω, 8 Ω and 16 Ω are standard input resistance values for this type of loudspeakers.

Headphones are a pair of small loudspeakers worn around the head over a user's ears. Earphones are a pair of small loudspeakers that plug into the ear canal.

- ⧫ TRC-11 needs a loudspeaker, headphones or earphones to convert the electrical energy into sound.

## 3.12 Examples

### Example 19

Suppose that the OPAMP shown in Fig. 3.40 is ideal (there is no current into + or − terminals, and the output of OPAMP acts like a voltage source.). Find $V_{out}$ in terms of $V_{in}$.

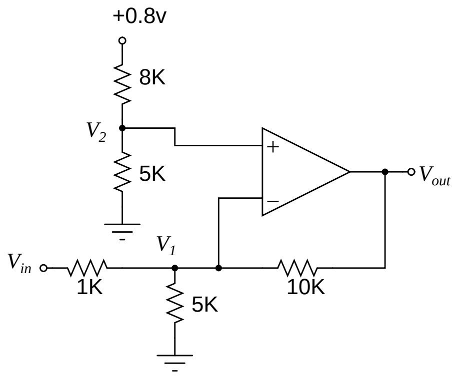



Figure 3.40: Circuit for Example 1.

#### Solution

Since the circuit does not contain any capacitor or inductor, we do not have a differential equation. $V_{out}$ can be expressed in terms of $V_{in}$ algebraically. We write KCL at node $V_1$ as

$$.\frac{V_1 - V_{in}}{1K} + \frac{V_1}{5K} + \frac{V_1 - V_{out}}{10K} = 0$$

or $10(V_1 - V_{in}) + 2V_1 + V_1 - V_{out}$=0 We can find $V_2$ from the voltage divider formula

$$V_2 = \frac{5K}{5K + 8K} 0.8 = \frac{4}{13} \text{V}$$

Assume that the output voltage of OPAMP is not saturated and hence it is in the linear region. From Eq. 3.49, we must have $V_1 = V_2$. Combining equations, we get $V_{out} = -10V_{in} + 4$. We write the solution as

$$V_{out} = \begin{cases} -10V_{in} + 4 & \text{for } V_{min} < -10V_{in} + 4 < V_{max} \\ V_{max} & \text{for } -10V_{in} + 4 > V_{max} \\ V_{min} & \text{for } -10V_{in} + 4 < V_{min} \end{cases}$$

---

### Example 20

For the circuit given in Fig. 3.41, assume that the input voltage $v_{in}(t)$ is a step function, as shown on the right. Assuming $V_{min} = -15$ V and $V_{max} = 15$ V, find $v_{out}(t)$ for all $t > 0$.

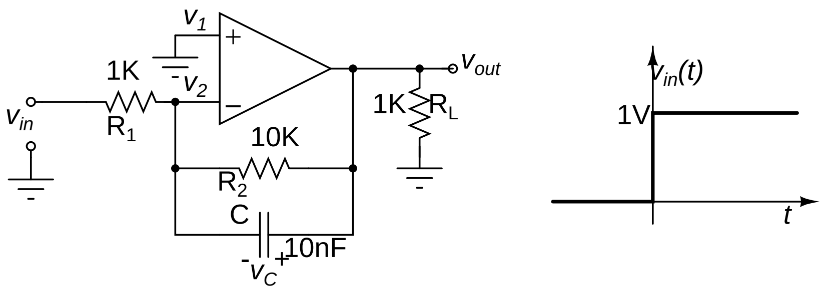



Figure 3.41: Circuit for Example 2 and 3.

**Solution**

This is a first-order $RC$ network. Assume that OPAMP output is not saturated. We can use the time-domain solution method of Section 2.8 on page 46. (We cannot use the phasor method since the input is not a sinusoid.)

1. We kill the source $v_{in}$. In this case, $v_{out} = 0$. Since $v_2 = v_1 = 0$, there is no current in $R_1$. Therefore, the total resistance seen by $C$ is $R_2$ only.
2. The time constant is $\tau = R_2 C = 100\mu$s.
3. $v_{out}(0^-) = -(R_2/R_1)v_{in}(0^-) = 0$ and $v_C(0^-) = v_C(0^+) = 0$V. Hence, $v_{out}(0^+) = 0$V.
4. We open-circuit the capacitor and write
$$v_{out}(\infty) = -\frac{R_2}{R_1}v_{in}(\infty) = -10\text{V}$$
Since $v_1 = v_2 = 0$, $v_C(\infty) = v_{out}(\infty) - v_2(\infty) = v_{out}(\infty) = -10$V
5. Since $V_{min} = -15 < -10$, the OPAMP is not saturated and we write $v_{out}$ as
$$v_{out}(t) = -10 + (0 - (-10))e^{-t/\tau} = -10 + 10e^{-t/100\mu} \text{ for } t > 0$$
This equation is plotted in Fig. 3.42.

---

## Example 21

For the OPAMP circuit of Fig. 3.41, the input signal is $v_{in}(t) = 0.5\cos(2\pi 1000t)$. Assuming $V_{min} = -12$ V and $V_{max} = 12$ V, what is $v_{out}(t)$?

**Solution**

Assume that the OPAMP output voltage is not saturated. Therefore, the circuit is linear. Since the input voltage is sinusoidal, we can use phasors. The input phasor is $V_{in}$=0.5. The transfer function is given by

$$\frac{V_{out}}{V_{in}} = -\frac{R_2 \parallel \frac{1}{j\omega C}}{R_1} = -\frac{R_2}{R_1}\frac{1}{1 + j\omega R_2 C}$$

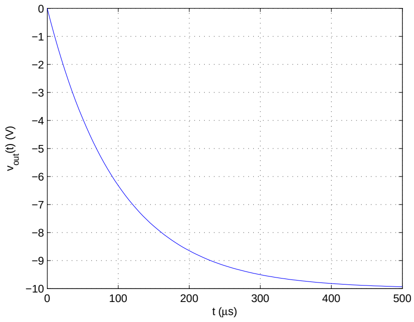



Figure 3.42: $v_{out}(t)$ as a function of time for Example 20.

Hence, the magnitude ratio is

$$\left|\frac{V_{out}}{V_{in}}\right| = \frac{R_2}{R_1}\frac{1}{\sqrt{1+(\omega R_2 C)^2}} = \frac{10}{\sqrt{1+(2\pi 10^3 10^{-4})^2}} = 8.46$$

and the phase difference is

$$\angle\frac{V_{out}}{V_{in}} = \pi - \tan^{-1}(\omega R_2 C) = 2.59\text{rad}$$

Therefore, we write

$$v_{out}(t) = 0.5 \cdot 8.46\cos(2\pi 1000t + 2.59) = 4.23\cos(2\pi 1000t + 2.59)$$

Since $4.23 < 12$, our initial assumption of linearity is verified.

---

## Example 22

What is the transfer function, $V_{out}/V_{in}$, of the circuit given in Fig. 3.43, assuming that the input signal is sinusoidal and the OPAMP output is not saturated.

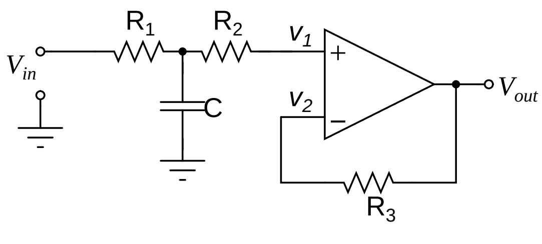



Figure 3.43: Circuit for Example 4.

**Solution**

We use phasors to find the transfer function. From the voltage divider at the input side, we find

$$V_1 = \frac{\frac{1}{j\omega C}}{\frac{1}{j\omega C} + R_1} V_{in} == \frac{1}{1 + j\omega R_1 C} V_{in}$$

$R_2$ is not in the equation since there is no current through it. If the output of OPAMP is not saturated, we have $V_1 = V_2$. Since there is no current through $R_3$, we have $V_{out} = V_2$. Therefore, we have

$$\frac{V_{out}}{V_{in}} = \frac{1}{1 + j\omega R_1 C}$$

---

## Example 23

What are the Thévenin and Norton equivalent circuits of the circuit given in Fig. 3.44(a).

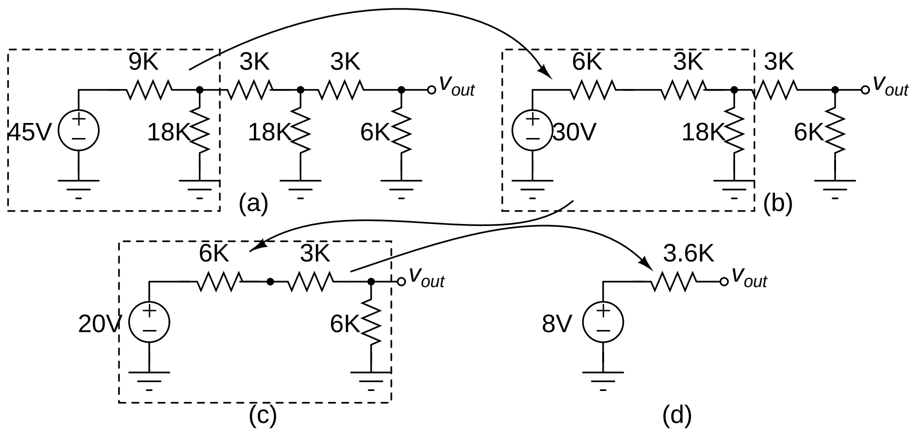



Figure 3.44: Circuit for Example 5.

**Solution**

We first find the Thévenin equivalent of 45 V, 9 kΩ and 18 kΩ resistor (inside the dashed lines): Since 9K‖18K=6K, and 45·18/(18+9)=30 V, we have the equivalent circuit shown in Fig. 3.44(b). One more application gives us the circuit in Fig. 3.44(c). Finally, we have $R_{eq}$=9K‖6K=3.6K and $V_{TH} = 20 \cdot 6K/(6K + 9K) = 8\text{V}$. We can find the Norton current from $I_N = V_{TH}/R_{eq}$=49.4 mA

---

## Example 24

For the circuit in Fig. 3.45, find $v_{out}$ if (a) $R_2/R_1$=2 and $v_{in}$=3 V, (b) $R_2/R_1$=4 and $v_{in}$=3 V, (c) $R_2/R_1$=4 and $v_{in}$=5 V.

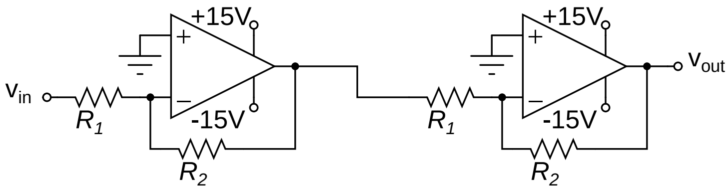



Figure 3.45: Circuit for Example 6.

**Solution**

Both amplifiers are inverting amplifier configurations. We also have $V_{min} = -15$ V and $V_{max} = 15$ V. The gain of one stage is $-R_2/R_1$. We have $V_{max} = 15$V and $V_{min} = -15$V.

(a) $v_{out} = (-R_2/R_1)^2 v_{in} = 4 \cdot 3 = 12$ V

(b) $v_{out} = (-R_2/R_1)^2 v_{in} = 16 \cdot 3 = 48$ V! The second OPAMP is saturated. $v_{out}$=15 V.

(c) $v_{out} = (-R_2/R_1)^2 v_{in} = 16 \cdot 5 = 80$ V! Both OPAMPs are saturated. $v_{out}$=15 V.

---

## Example 25

Consider the circuit given in Fig. 3.46. Find $v_3(t)$.

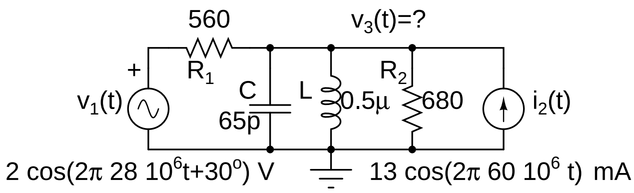



Figure 3.46: Circuit for Example 7.

**Solution**

The linear circuit has two sources at two *different* frequencies. The phasor approach can be used to find the solution when all excitations are at the same frequency. Since we have two different frequencies, we can use the superposition principle to solve the circuit for one excitation at a time:

When the current source, $i_2(t)$, is killed (open-circuited), the voltage source, $v_1(t)$, is the only excitation at $\omega_1 = 2\pi 28 \cdot 10^6$. For this frequency, the capacitor is replaced with $1/j\omega_1 C$, and inductor is replaced with $j\omega_1 L$. The voltage source is represented with the phasor $V_1 = 2\angle 30^o$. At this frequency the voltage phasor $V_3$ is found as

$$V_3 = \frac{2\angle 30^o}{560}\left(\frac{1}{560} + \frac{1}{680} + j\omega_1 C + \frac{1}{j\omega_1 L}\right)^{-1} = 1.1\angle 30^o$$

Since the imaginary part of the expression in parentheses is zero, the angle of the $V_3$ is the same as the angle of $V_1$. In the time domain, we find

$$v_3(t) = 1.1\cos(2\pi 28\cdot 10^6 + 30^o)$$

When the voltage source, $v_1(t)$, is killed (short-circuited), the current source, $i_2(t)$, is the driver at $\omega_2 = 2\pi 60\cdot 10^6$. Hence, the capacitor is replaced with $1/j\omega_2 C$, and the inductor is replaced with $j\omega_2 L$. The current source is represented with the phasor $I_2 = 13$ mA. At this frequency, the voltage phasor $V_3$ is found as

$$V_3 = 13\cdot 10^{-3}\left(\frac{1}{560}+\frac{1}{680}+j\omega_2 C+\frac{1}{j\omega_2 L}\right)^{-1} = 0.67\angle - 80^o$$

In time domain, this is equivalent to $v_3(t) = 0.67\cos(2\pi 60\cdot 10^6 - 80^o)$.

Therefore, the total solution is

$$v_3(t) = 1.1\cos(2\pi 28\cdot 10^6 + 30^o) + 0.67\cos(2\pi 60\cdot 10^6 - 80^o)$$

---

## Example 26

Consider the circuit given in Fig. 3.47(a) with $i_B(t) = 60\cos(2\pi 10^3 t)$ and

$$v_A(t) = \begin{cases} -5 & \text{for } t < 0 \\ +10 & \text{for } t > 0 \end{cases}$$

Find $v_C(t)$.

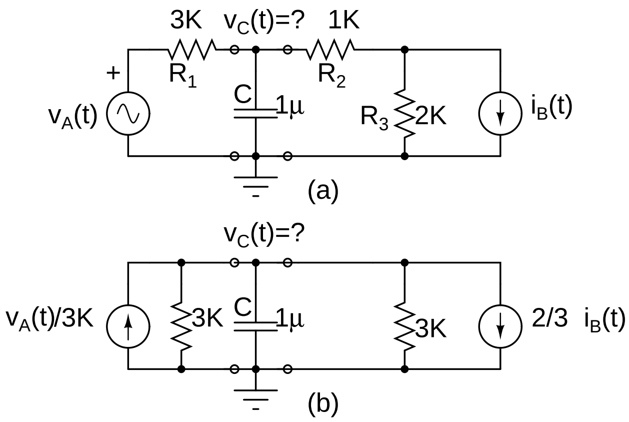



Figure 3.47: (a) Circuit for Example 8, (b) simplified circuit obtained using Norton equivalent circuits.

**Solution**

The linear circuit has two sources, and the superposition principle is applicable. The voltage source is a step function at $t = 0$. Since the circuit is first-order, we can use the method given on page 46 to find $v_C(t)$. On the other hand, the current source is sinusoidal. For this source, the phasor approach can be easily applied.

To simplify the circuit, we find the Norton equivalent of the $v_A(t)$ and $R_1$ as shown on the left-hand side of Fig. 3.47(b). Similarly, the Norton equivalent of $i_B(t)$, $R_2$, and $R_3$ are substituted on the right-hand side.

First, we find $v_C(t)$ due to current source on the left, while the current source on the right is killed. $v_C(0)$ is found by open-circuiting $C$ for $v_A(t) = -5$ with $t < 0$. We get

$$v_C(0) = \frac{-5}{3K}(3K \parallel 3K) = -2.5$$

$v_C(\infty)$ is determined by open-circuiting $C$ while $v_A(t) = +10$ with $t > 0$:

$$v_C(\infty) = \frac{+10}{3K}(3K \parallel 3K) = 5$$

The time constant is equal to $\tau = (1\mu)(3K \parallel 3K) = 1.5$ ms. Therefore the solution for this source is

$$v_C(t) = \begin{cases} -2.5 & \text{for } t < 0 \\ 5 + (-2.5 - 5)e^{-t/1.5\cdot 10^{-3}} & \text{for } t \geq 0 \end{cases}$$

We find $v_C(t)$ due to the sinusoidal current source on the right, while the current source on the left is killed. We replace $C$ with $1/j\omega C$ with $\omega = 2\pi 10^3$, the current source with the phasor $2/3 \cdot 60 = 40$ mA. The phasor $V_C$ is found as

$$V_C = -40 \cdot 10^{-3}\left(\frac{1}{3K} + \frac{1}{3K} + j\omega C\right)^{-1} = \frac{-0.040}{0.00632\angle 84^o} = 6.33\angle 96^o$$

Hence, the total solution is

$$v_C(t) = \begin{cases} -2.5 + 6.33\cos(2\pi 10^3 t + 96^o) & \text{for } t < 0 \\ 5 - 7.5e^{-t/1.5\cdot 10^{-3}} + 6.33\cos(2\pi 10^3 t + 96^o) & \text{for } t \geq 0 \end{cases}$$

---

## Example 27

Consider the 50 Hz high-voltage three-phase circuit given in Fig. 3.48. We have $V_a = 380\angle 0$ kV, $V_b = 380\angle 120^o$ kV, $V_c = 380\angle 240^o$ kV all in rms. Find the rms line currents, $I_1$, $I_2$, $I_3$. Find the total power delivered to load resistances.

**Solution**

The rms load currents can be found easily since we know the voltages across the loads:

$$I_a = \frac{380\text{K}\angle 0}{16\text{K} + j1.41\text{K}} = \frac{380\angle 0}{16.06\angle 5.05^o} = 23.7\angle(-5.05^o) \text{ A}$$

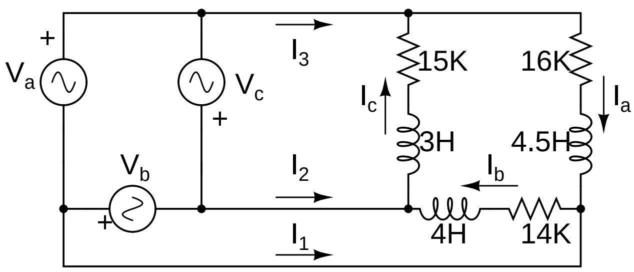



Figure 3.48: Three-phase circuit for Example 9.

$$I_b = \frac{380\text{K}\angle 120^0}{14\text{K} + j1.26\text{K}} = \frac{380\angle 120^o}{14.06\angle 5.13^o} = 27.0\angle(115^o)\ \text{A}$$

$$I_c = \frac{380\text{K}\angle 240^o}{15\text{K} + j0.942\text{K}} = \frac{380\angle 240^o}{15.03\angle 3.60^o} = 25.3\angle(236^o)\ \text{A}$$

The rms line currents can be found from KCL at the nodes:

$$I_1 = I_b - I_a = 27.0\angle(115^o) - 23.7\angle(-5.05^o) =$$
$$= -11.4 + j24.5 - 23.6 + j2.08 = -34.9 + j26.6 = 43.9\angle 143^o\ \text{A}$$

$$I_2 = I_c - I_b = 25.3\angle(236^o) - 27.0\angle(115^o) =$$
$$= -14.0 - j21.1 + 11.4 - j24.5 = -2.61 - j45.6 = 45.6\angle(-93.3^o)\ \text{A}$$

$$I_3 = I_a - I_c = 23.7\angle(-5.05^o) - 25.3\angle(236^o) =$$
$$= 23.6 - j2.08 + 14.0 + j21.1 = 37.6 + j19.0 = 42.1\angle 26.8^o\ \text{A}$$

The power delivered to loads can be found as

$$P = |I_a|^2 16\text{K} + |I_b|^2 14\text{K} + |I_c|^2 15\text{K} = 8.95 + 10.2 + 9.6 = 28.8\ \text{MW}$$

---

## Example 28

Three OPAMP circuit shown in Fig. 3.49 is known as an *instrumentation* amplifier. Find the output voltage $v_o$ in terms of input voltages $v_1$ and $v_2$.

### Solution

Assuming that OPAMPs are not saturated, we have $v_1 = v_5$ and $v_2 = v_6$. Hence $i_a = (v_5 - v_6)/R_a = (v_1 - v_2)/R_a$. Since the same current flows in the neighboring resistors, we have

$$v_3 - v_4 = i_a(2R + R_a) = \frac{v_1 - v_2}{R_a}(2R + R_a) = \left(1 + \frac{2R}{R_a}\right)(v_1 - v_2)$$

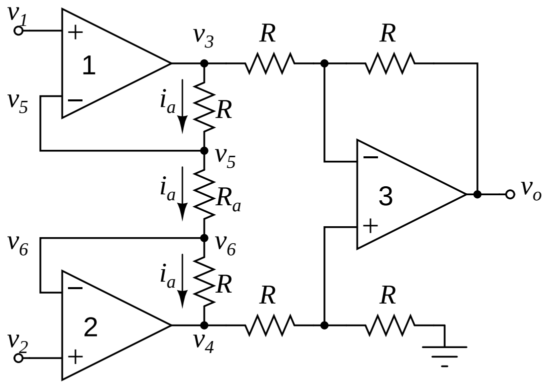



Figure 3.49: Instrumentation amplifier.

The OPAMP #3 is a difference amplifier as in Fig. 3.27(b), with all resistors equal to $R$. Hence the output voltage is given by

$$v_o = \frac{R}{R+R}\frac{R+R}{R}v_4 - \frac{R}{R}v_3 = v_4 - v_3 = \left(1 + \frac{2R}{R_a}\right)(v_2 - v_1)$$

The instrumentation amplifier is a symmetrical difference amplifier with infinite input impedance at both inputs. Moreover, the gain can be set using a single resistor, $R_a$.

## 3.13 Problems

1. Evaluate the following complex number equations. Find the results in rectangular and polar format with three significant digits:
   (a) (7.82-j10.11)(-2.25-j4.11)
   (b) (3.73+j9.32)/(1.28-j2.20)
   (c) $3.44\angle 34^o + 2.99\angle 138^o$
   (d) $1.99 \cdot 10^{-3}\angle(-27^o) - 8.9 \cdot 10^{-4}\angle(-150^o)$
2. Expand $e^{j\theta}$ in the Taylor series, group the real and imaginary parts, and show that the real series corresponds to the expansion of $\cos\theta$ and the imaginary series corresponds to the expansion of $\sin\theta$.
3. Show that capacitance is a linear circuit element.
4. 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.
5. 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 this amplifier is a linear circuit component.
6. Design an LPF using a resistor and an inductor. Find the transfer function for this filter and plot its magnitude with respect to angular frequency.
7. Design an HPF using a resistor and an inductor. Find the transfer function for this filter and plot its magnitude with respect to angular frequency.
8. Find the impedance of the circuits given in Fig. 3.50 at the specified frequency. Write the impedance in polar form, i.e., magnitude and phase (2 significant figures):

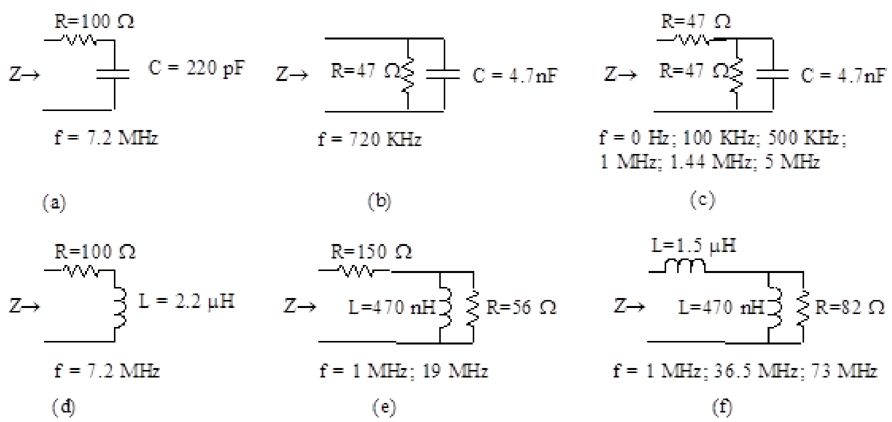



Figure 3.50: Circuits for problem 8

9. The voltage (current) sources in Fig. 3.51 are connected to the circuits in problem 8. The frequencies of the sources are as given in problem 8. Find the current through (voltage across) the sources.

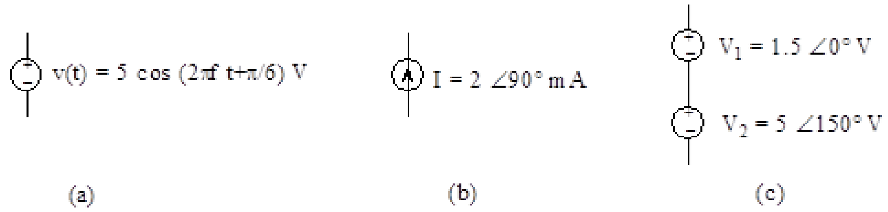



Figure 3.51: Sources for problem 9

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

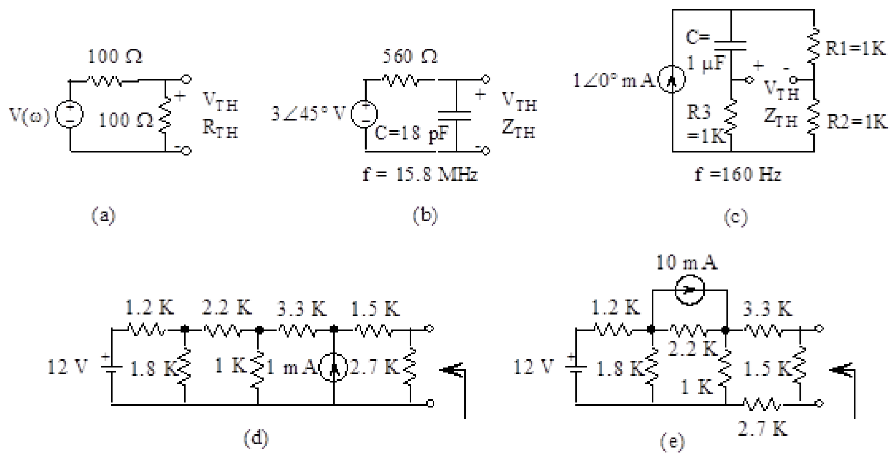



Figure 3.52: Circuits for problem 17

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

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

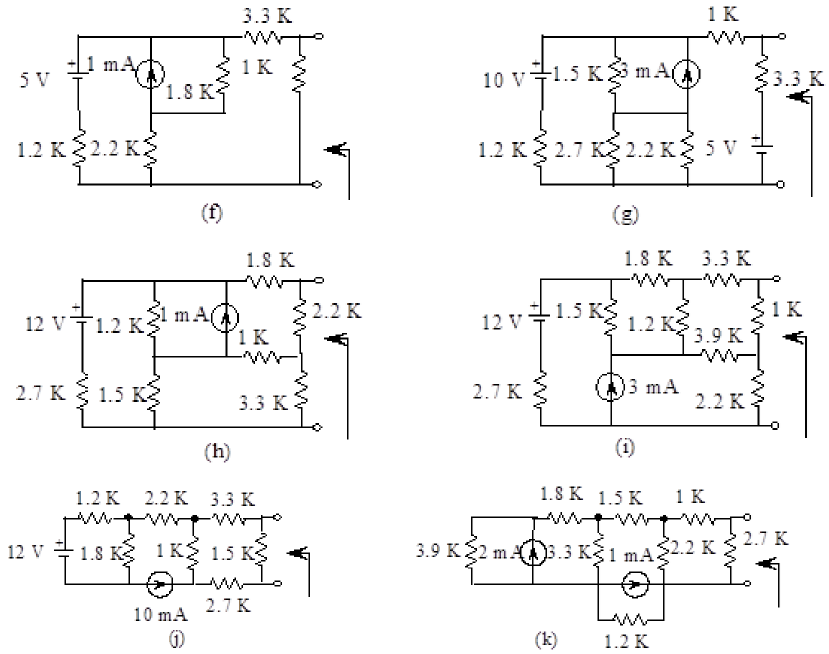



Figure 3.53: (cont.) Circuits for problem 17

12. 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.

13. A series circuit has $R$=1 k$\Omega$ and $C$=120 pF. What is the frequency (not angular frequency) at which the phase difference between the current and voltage is $\pi/4$?

14. A series $RC$ circuit has $C$=470 pF. Find $R$ if the phase difference between current and voltage is $30^o$ at 1 kHz.

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

16. Assume that the voltage and current pairs given in problem 11 are for two-element parallel circuits. Determine the component types and their values.

17. Find and draw the Thévenin equivalent of the circuits given in Figs. 3.52 and 3.53. Express the equivalent voltages and impedances in polar form.

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

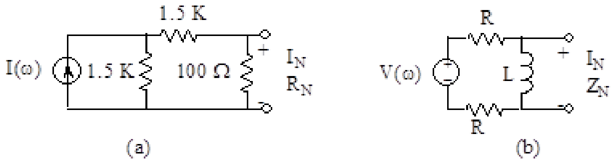



Figure 3.54: Circuits for problem 19

19. Find and draw the Norton equivalent of the circuits given in Fig. 3.54. Express the equivalent currents and impedances in polar form.

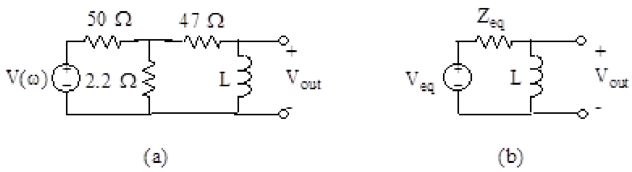



Figure 3.55: Circuits for problem 20

20. Find the $V_{eq}$ and $Z_{eq}$ such that the circuit given in Fig. 3.55(a) can be represented as in (b).

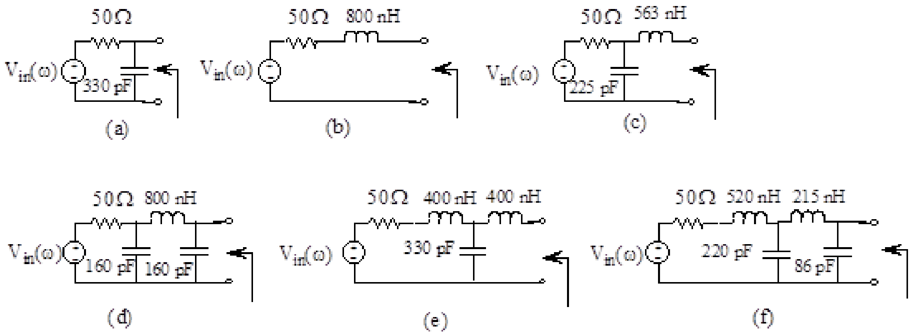



Figure 3.56: Circuits for problem 21

21. Find and draw the Thévenin and Norton equivalents of the circuits in Fig. 3.56 at DC, 20 MHz, and 40 MHz.

22. This problem illustrates how a unity feedback amplifier is used to avoid loading effects. Consider the divider circuit in the Fig. 3.57(a). What is $V_{out}$? Assume we want to apply $V_{out}$ across a 1 kΩ resistor, as shown in part (b). What is $V_{out}$ now? Now assume we place a buffer amplifier between the divider and 1K resistor as in part (c). Find $V_{out}$.

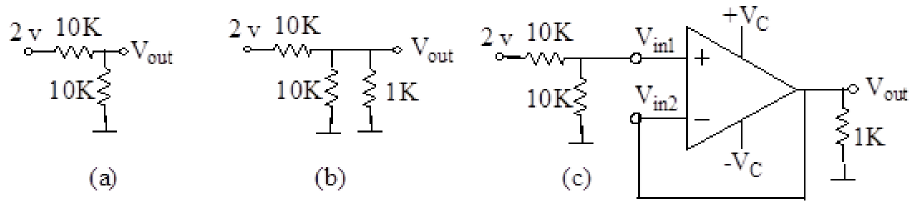
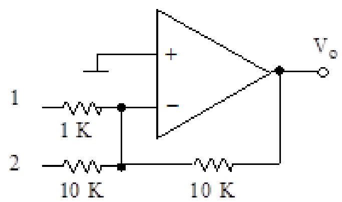



Figure 3.57: Circuits for problem 22

23. Assume there are two signals, $V_1$ and $V_2$. Design a summing amplifier to produce $V_{out} = 2V_1 + 0.5V_2$, using (a) two OPAMPs, and (b) one OPAMP. assuming that OPAMPs are ideal.




| 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 | | |

Figure 3.58: Circuit for problem 24

24. Make a table indicating what terminals to connect to the input signal source or the output to get all possible (different) amplification factors for the circuit in Fig. 3.58. Also, calculate the resulting input impedance and the possible gains, and include them in the table.

25. Find $V_o/V_{in}$ in the OPAMP circuits of Figs. 3.59 and 3.60.

26. Find the transfer functions $V_o(\omega)/V_{in}(\omega)$ for the circuits given in Figs. 3.61 and 3.62.

27. In the circuits of Fig. 3.61 and 3.62, find the asymptotic behavior of the transfer function at low frequencies and high frequencies.

28. Find the transfer function of the circuit given in Fig. 3.63. Is there a frequency at which the gain is zero? Which frequency?

29. Find the voltage output of a condenser microphone of capacitance $C_o$ as a function of diaphragm vibration when a cable of capacitance $C_c$ is connected at its output.

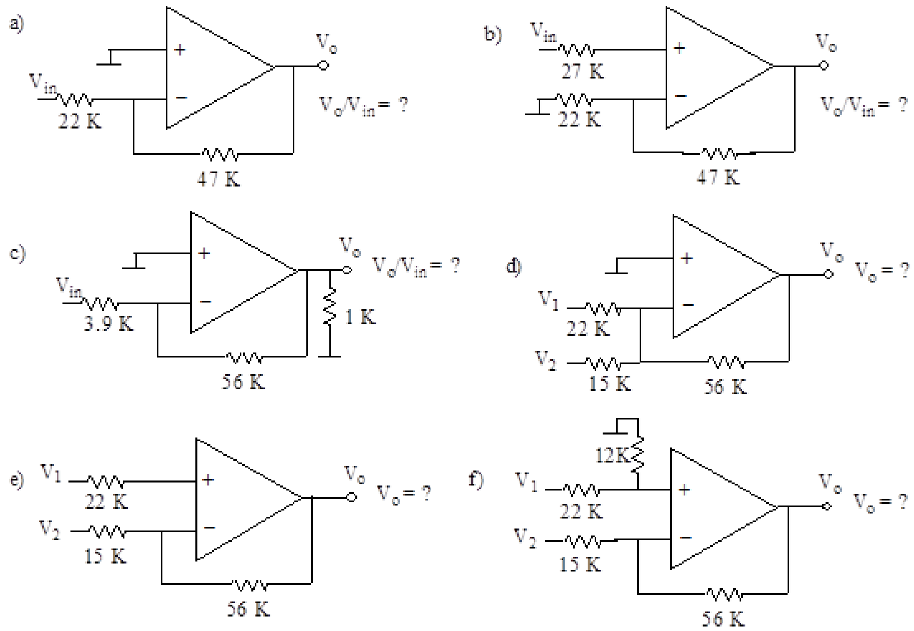



Figure 3.59: Circuits for problem 25

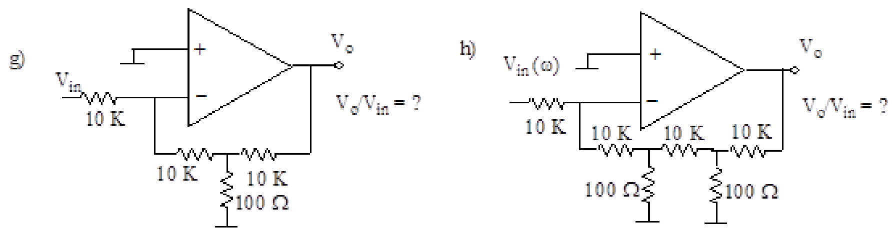



Figure 3.60: (cont.) Circuits for problem 25

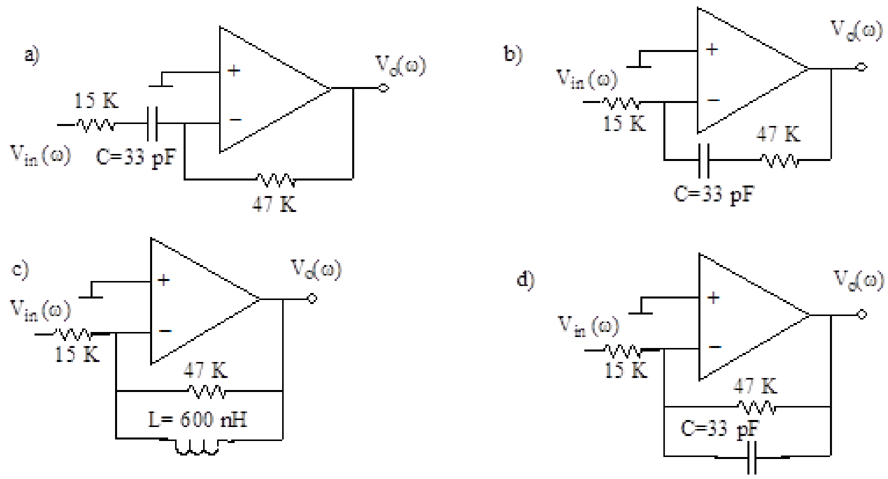



Figure 3.61: Circuits for problems 26 and 27

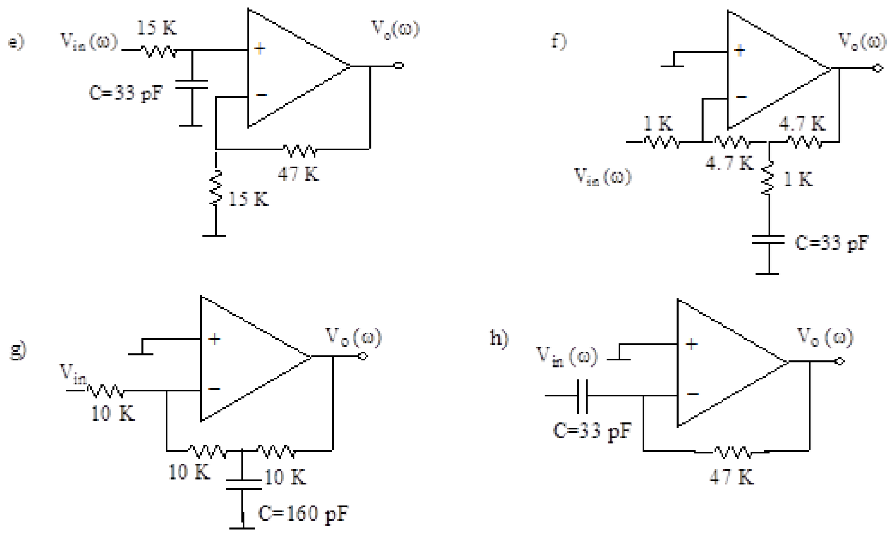



Figure 3.62: (cont.) Circuits for problems 26 and 27

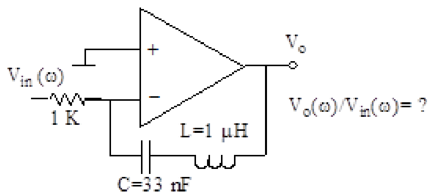



Figure 3.63: Circuit for problem 28

# Chapter 4

# DIODES and BIPOLAR JUNCTION TRANSISTORS

Diodes are important electronic devices used in many applications. Unlike the devices like resistors, capacitors, and inductors, they are nonlinear devices. In the first part of this chapter, we explore the characteristics of diodes as a circuit element. We study the solution of circuits containing diodes. Several important circuit containing didoes are presented.

As seen in the previous chapter, OPAMPs can be used for low-frequency (or audio frequency) amplification. At high frequencies, discrete transistors or integrated circuit amplifiers are used for amplification. At microwave frequencies, MMICs (monolithic microwave integrated circuits) are preferred for the same purpose. In the second part of this chapter, we explore the bipolar junction transistors. After introducing the DC solution of circuits containing a BJT, the amplification property of the circuit is demonstrated.

## 4.1 Diodes

A *diode* is a nonlinear resistor with a symbol shown in Fig. 4.1(a). It carries the current in one direction but not in the other. An *ideal diode* is described with two states: OFF state—it carries no current with a negative voltage across it; ON state—it has no voltage across it with a positive current through it.

An ideal diode characteristics can be written as

$$\begin{aligned} \text{ON state:} \quad & v_D = 0 \;\; \text{if} \;\; i_D \geq 0 \\ \text{OFF state:} \quad & i_D = 0 \;\; \text{if} \;\; v_D < 0 \end{aligned} \tag{4.1}$$

In a graphical form, this can be shown as an $I - V$ characteristic as in Fig. 4.1(b).

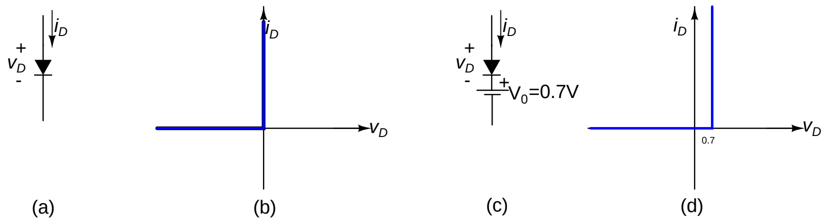



Figure 4.1: (a) Diode symbol and reference directions, (b) $I-V$ characteristics for an ideal diode, (c) Approximate diode equivalent circuit, (d) $I-V$ characteristics for the diode approximate equivalent circuit

### Water flow analogy of a diode

Consider the structure in Fig. 4.2. If water flows from left to right, the flap moves right, and water is allowed to pass. If water comes from the right, the flap blocks the flow, and no water flows to the left. This is analogous to a diode.

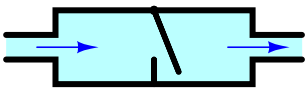


Figure 4.2: Water flow analogy of a diode

### Real diodes

Most of the contemporary diodes are semiconductor devices. They are built by a junction of p-type and n-type semiconductors called *p-n junction diodes*. They are nonlinear resistors, resistance of which depend on the voltage across them. The $I-V$ characteristic of a semiconductor diode can be well approximated by an exponential relation:

$$I_D = I_S(e^{-V_D/\gamma} - 1) \tag{4.2}$$

where $I_D$ and $V_D$ are the current and voltage of the diode. $I_S$ and $\gamma$ are the physical constants related to the material and construction of the diode. The typical I-V characteristic of a silicon p-n junction power diode is given in Fig. 4.3.

$V_D$ 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 $V_0$ ($V_0$=0.7 V for silicon diodes), hence it behaves like a short circuit for these larger voltages. The real diodes have a small *threshold voltage*, $V_0$, across them when they are in the ON state. So a better approximation for silicon diodes can be written as follows (see the model shown in Fig. 4.1(c)):

$$\begin{aligned} \text{ON state:} \quad & v_D = V_o \text{ V} \;\; \text{if} \;\; i_D \geq 0 \\ \text{OFF state:} \quad & i_D = 0 \;\; \text{if} \;\; v_D < V_o \text{ V} \end{aligned} \tag{4.3}$$

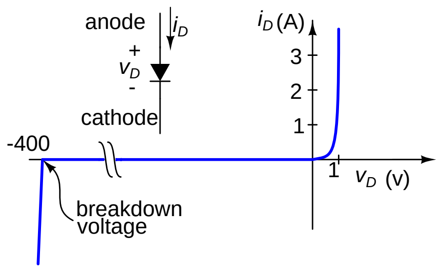



Figure 4.3: $I - V$ characteristic of a real p-n junction power diode

where $V_o$ is approximately 0.7 V. $I - V$ graph for this approximation is shown in Fig. 4.1(d).

Note in Fig. 4.3 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.

### 4.1.1 Schottky diode

Schottky diode is a diode formed by a junction of a semiconductor with a metal. It was named after German physicist Walter Schottky (1886–1976). It has a forward voltage drop of about half that of a p-n junction diode at the same current level. The maximum allowable reverse voltages of Schottky diodes are typically less than 50 V, lower compared to p-n junction diodes. They are used in applications requiring high switching speeds. The symbol of the Schottky diode is shown in Fig. 4.4.

### 4.1.2 Solutions of circuits containing diodes

Since diodes are nonlinear devices, the analysis methods described earlier cannot be used directly. If we approximate a diode using the piecewise linear model, we can use the methods suitable for linear circuits as described below.

> The procedure for the solution of circuits containing one or more diodes is as follows:
>
> 1. Assign ON or OFF state to the diode(s).
> 2. Replace the diode(s) with the equivalent circuit in that state.
> 3. Solve the resulting linear circuit

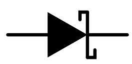


Figure 4.4: Symbol of a Schottky diode.

4. Check if the condition of the assumed state is satisfied for the diode(s).
5. If not, go to step 1 and change the state of the diode(s).

If it is difficult to estimate the state of the diode, it is better to try the OFF state first, since it usually results in a simpler circuit. If there are several diodes, the procedure should be repeated until the conditions of all diodes are satisfied. The procedure is best understood by the examples given below.

**Example 29**

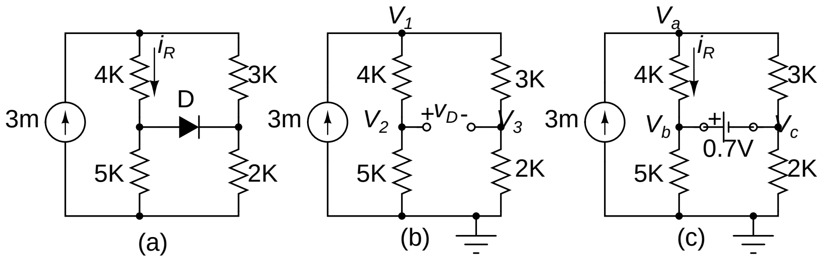



Figure 4.5: Example 29: A circuit containing a diode

We want to find the current $i_R$ in the circuit given in Fig. 4.5 with one diode. Use the approximate model of the diode in Eq. 4.3. Apply the procedure:

1. Assume that the diode is OFF.
2. Replace the diode with an open-circuit (as in Fig. 4.5(b)).
3. Solve the resulting linear circuit:
   - Assign the ground node to the bottom of the circuit.
   - The circuit has only one node: $V_1$. ($V_2$ and $V_3$ are not nodes since they have only two branches.)
   - Write KCL for $V_1$:

$$\frac{V_1}{4\text{K}+5\text{K}} + \frac{V_1}{2\text{K}+3\text{K}} - 3\text{mA} = 0$$

4. Solve to find $V_1$=135/14=9.6 V
5. The voltage across the diode is using the voltage divider relations:

$$v_D = V_2 - V_3 = V_1\frac{5\text{K}}{4\text{K}+5\text{K}} - V_1\frac{2\text{K}}{2\text{K}+3\text{K}} = \frac{3}{2} = 1.5\ V$$

6. Since the diode voltage $v_D$=1.5 V>0.7 V, the OFF condition of Eq. 4.3 is not satisfied: The diode must be ON.

7. Substitute the ON model of the diode (as in Fig. 4.5(c)).

8. Solve the resulting circuit to find $i_R$:

   - Assign the ground node to the bottom of the circuit.
   - The circuit has three nodes: $V_a$, $V_b$, and $V_c$.
   - Write KCL for $V_a$ and $V_b$. $V_c$ can be written in terms of $V_b$ by KVL:

$$\begin{array}{ll} V_a: & \frac{V_a-V_b}{4K}+\frac{V_a-V_c}{3K}-3\text{mA}=0 \\ V_b: & \frac{V_b}{5K}+\frac{V_b-V_a}{4K}+\frac{V_c-V_a}{3K}+\frac{V_c}{2K}=0 \\ V_c: & V_c=V_b-0.7 \end{array}$$

9. Solve to find $V_a$=9.52 V and $V_b$=4.78 V.
   Hence, $i_R$=(9.52-4.78)/4K=1.19 mA

---

**Example 30**

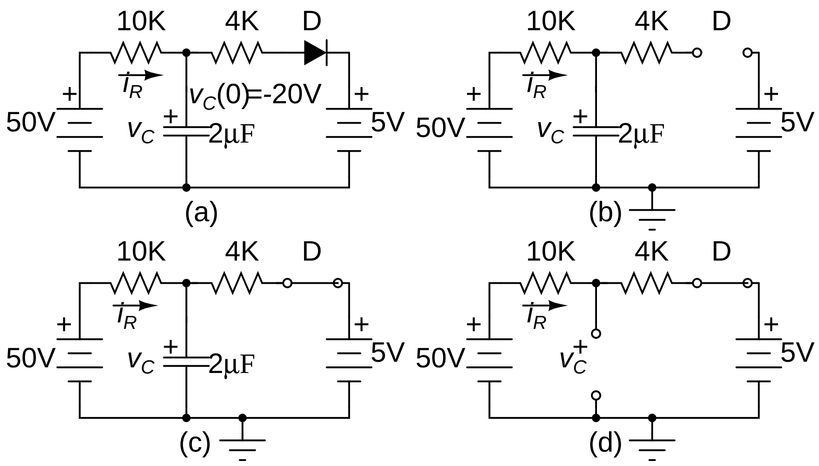



Figure 4.6: Example 30: An RC circuit containing a diode

We want to find the current $i_R$ in the circuit given in Fig. 4.6 with one ideal diode ($V_0$=0). Use the approximate model of the diode in Eq. 4.3. Apply the procedure:

1. Assume that the diode is OFF.

2. Replace the diode with an open-circuit (as in Fig. 4.6(b)).

3. Solve the resulting linear circuit:

   - Assign the ground node to the bottom of the circuit.
   - Solve the resulting linear RC circuit using the procedure in Section 2.8:

- Kill the 50 V voltage source. Find the equivalent resistor across capacitor: $R_{eq}$=10 kΩ.
- The time constant is $\tau$=$R_{eq}$C=10K×2 $\mu$F=20 ms
- The initial value of $v_C(0) = -20$ V. The initial value of $i_R$ is found by substituting a voltage source of $-20$ V in place of the capacitor: $i_R(0) = (50 - (-20))/10K = 7.0$ mA.
- The final value of $v_C(\infty)$=50 V. The final value of $i_R$ is found by substituting an open circuit for the capacitor: $i_R(\infty) = 0$
- Write the solution for $v_C$ as $v_C(t) = 50 + (-20 - 50)e^{-t/20m}$ and for $i_R$ as $i_R(t) = 0 + (7 - 0)e^{-t/20m}$ mA

4. Check that the diode is actually OFF by finding the diode voltage: $v_D = v_C - 5 = 45 - 70e^{-t/20m}$. We see that the diode voltage is negative, $v_D < 0$ (so our assumption of OFF diode is correct), when $70e^{-t/20m} > 45$ or when $t < 20\ln(70/45)$ms=8.84 ms. For $t >$ 8.84 ms we must assume that the diode is ON.

5. Substitute a short-circuit for diode and solve the resulting circuit shown in Fig. 4.6(c).

   - Kill both 50 V and 5 V voltage sources. Find the equivalent resistor across capacitor: $R_{eq}$=10K‖4K=2.86 k.
   - The time constant is $\tau$=2.86K×2 $\mu$F=5.72 ms
   - The initial value of capacitor at the time the diode turns ON is $v_C(8.84\text{m}) = 50 - 70e^{-8.84m/20m}$=5.0 V. The initial value of $i_R$ is found by substituting a voltage source of 5 V in place of the capacitor: $i_R(8.84\text{m}) = (50 - 5)/10K = 4.50$ mA.
   - The final value of $i_R$ is found by open circuiting the capacitor as in Fig. 4.6(d). $i_R(\infty) = (50 - 5)/(10K + 4K)$=3.21 mA. Hence $v_C(\infty)$=50−3.21×10K=17.9 V.
   - Write the solution for $v_C$ for $t$ >8.84 m as

     $$v_C(t) = 17.9 + (5.0 - 17.9)e^{-(t-8.84m)/5.72m}$$

     and for $i_R$ as

     $$i_R(t) = 3.21 + (4.50 - 3.21)e^{-(t-8.84m)/5.72m}$$

6. We check that for $t$ >8.84 m, the diode current (=$i_R$) is positive, so our assumption of the diode being ON is correct.

A MATLAB code to plot $v_C$ and $i_R$ is given below. The resulting plot is shown in Fig. 4.7.

```
% MATLAB code to draw iR and vC of Example 8
clear all hold off
t1=0:0.01:8.84; % define two separate vectors
t2=8.84:0.01:20; %             for two regions
iR1=7*exp(-t1/20); % current for the first region
iR2=3.21+(4.5-3.21)*exp(-(t2-8.84)/5.72); % second region
```

```
vC1=50-70*exp(-t1/20); % voltage for the first region
vC2=17.9+(5-17.9)*exp(-(t2-8.84)/5.72); % second region
[hx,h1,h2]=plotyy([t1 t2],[iR1 iR2],[t1 t2],[vC1 vC2])
% plotyy function draws a graph using left and right axes
grid on xlabel(hx(1),'t (ms)')
ylabel(hx(1),'i_R (mA)') % to define the label for left axis
ylabel(hx(2),'v_C (V)') % label for the right axis
set(h1,'LineStyle','-','LineWidth',2);
set(h2,'LineStyle','--','LineWidth',2);
legend([h1 h2],['i_R'],['v_C']) % to put a legend on the graph
title('Example 8')
```

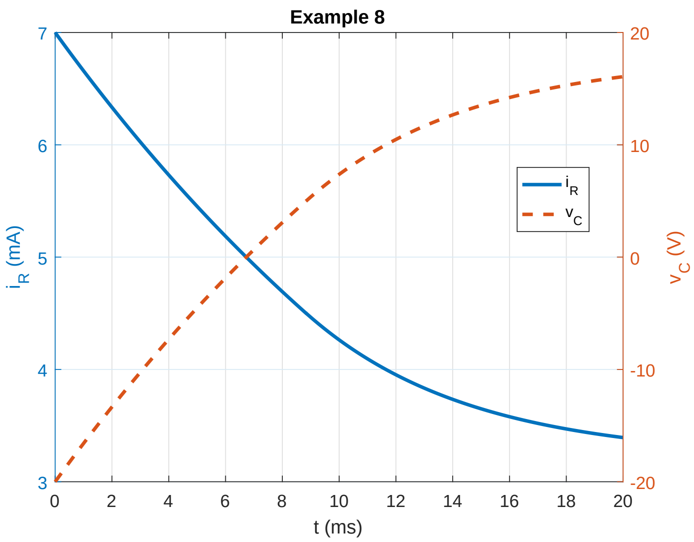



Figure 4.7: Example 30: Plots of $i_R(t)$ and $v_C(t)$.

---

### 4.1.3 Diodes as rectifiers

Diodes are used for many different purposes in electronic circuits. One primary application is *rectification*. Electrical energy is distributed in the 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 for electronic instrumentation. Almost all electronic instruments have power supply sub-system, where AC energy supply is converted into DC voltage supplies in order to provide the necessary energy for the electronic circuits. We first need to rectify the AC voltage to generate a DC voltage.

### 4.1.4 Half-wave rectifier

Consider the circuit depicted in Fig. 4.8(a). There is a current flowing through the circuit in Fig. 4.8(b) during the positive half cycles of the AC voltage, $v_{in}$,

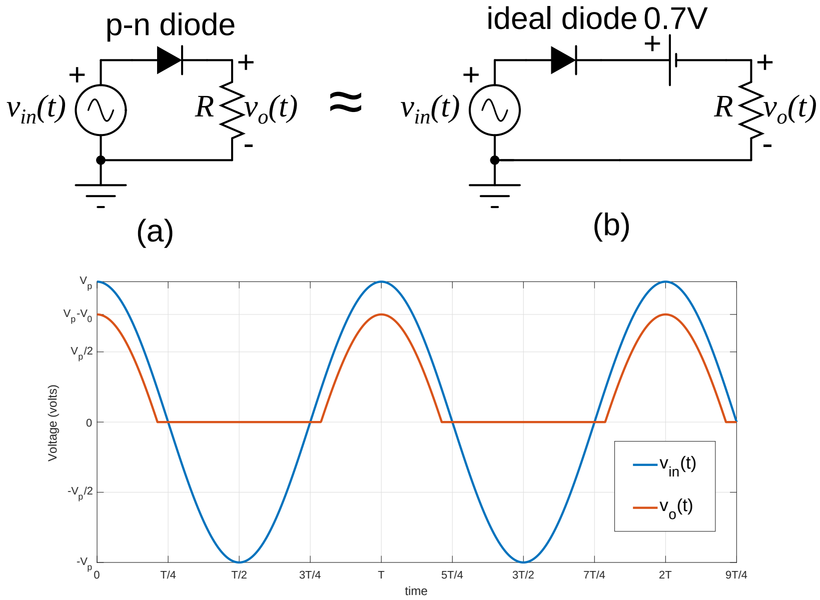



Figure 4.8: (a) Half-wave diode rectifier, (b) the equivalent circuit, (c) the input AC voltage, $v_{in}(t)$, with a peak value, $V_p$; the output voltage, $v_o(t)$, on the load resistor, $R$.

at the input, while it becomes zero during negative half cycles. The current starts flowing as soon as $v_{in}$ exceeds $V_0 = 0.7$V and stops when $v_{in}$ falls below $V_0$. The voltage that appears across the load, $v_o$, is, therefore, sine wave halves as depicted in Fig. 4.8(c). This voltage waveform is neither an AC voltage nor a DC voltage, but it is always positive.

When this circuit is modified by adding a capacitor in parallel with $R$, we obtain the circuit in Fig. 4.9(a), and its equivalent Fig. 4.9(b). The capacitor functions like a filter together with the resistor, to smooth out $v_o(t)$.

Consider the $v_o(t)$ waveform in Fig. 4.9(c) with $RC = 10T$.

- When $v_{in}(t) - V_0$ exceeds $v_o(t)$ (at $t$=$t_r$), the diode starts conducting, and the current through the diode charges up the capacitor.
- The charge up continues until $v_o$ reaches the peak, $V_p - V_0$, at $t$=$T$.
- After the peak voltage is reached, the voltage at the anode of the diode, $v_{in}$, falls below $V_p$ and hence the voltage across the diode, $v_{in} - v_o$, becomes less than $V_o$. The current through the diode ceases flowing. The diode is *reverse biased*.
- Now, the AC voltage source is isolated from the parallel $RC$ circuit, and the capacitor is charged up to $V_p - V_o$. The capacitor starts discharging on $R$ with a time constant of $RC$. If $RC$ is small, as depicted in Fig. 4.9(c) for $RC = T$, the capacitor discharges quickly. If $RC$ is large, the discharge is slow, as shown in the case with $RC = 10T$.

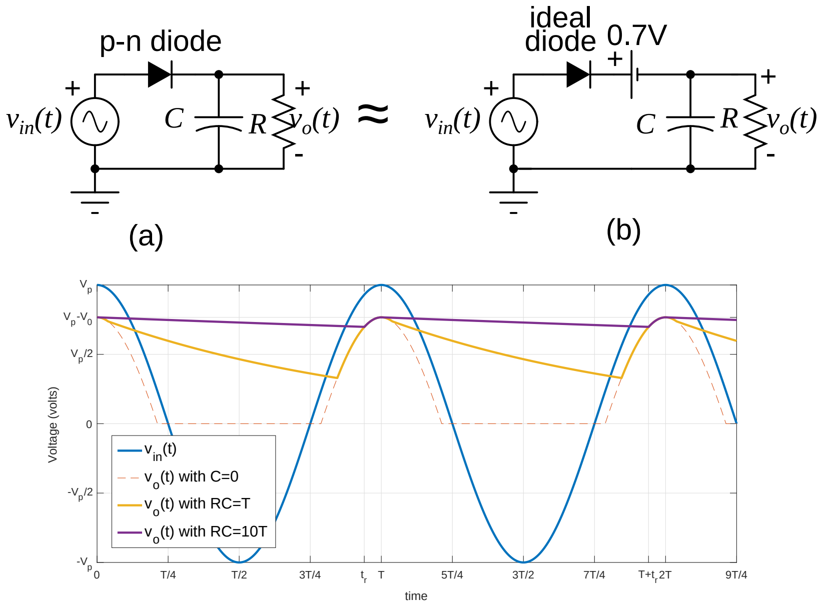



Figure 4.9: (a) Half-wave diode rectifier with $RC$ filter, (b) equivalent circuit and (c) voltage waveforms on load resistor for $RC = T$ and $RC = 10T$.

- As $v_{in}$ increases for the next half-cycle of positive sine wave tip, it exceeds the voltage level to which the capacitor discharged until then, at $t{=}T+t_r$, and the diode is switched on again.
- It starts conducting, and the capacitor is charged up to $V_p - V_0$ all over again ($t{=}2T$).
- The fluctuation in the output voltage is called *ripple*. Clearly, the case for $RC = T$ has a larger ripple than the case for $RC = 10T$.

The waveform obtained in Fig. 4.9(c) is highly irregular, but it is obviously a better approximation to a DC voltage compared to the one in Fig. 4.8(c). We prefer an *electrolytic* capacitor in this circuit, which possesses polarity. Large capacitance values in small physical sizes are available in the electrolytic form. Large capacitance values allow us to have more charge storage for the same voltage level, thus smoother output waveforms with smaller ripple. In this circuit, the voltages that may appear across the capacitor are always positive because of rectification, and hence there is no risk in using such a polarized capacitor type. Note that connecting an electrolytic capacitor in the wrong direction may destroy the capacitor.

**Ripple estimation for the half-wave rectifier**

Consider the $v_o(t)$ waveform with $RC = 10T$ in Fig. 4.9(c). We would like to find the peak-to-peak ripple voltage, $V_r = v_o(0) - v_o(t_r)$. We can write the output voltage waveform mathematically as

$$v_o(t) = \begin{cases} (V_p - V_0)e^{-t/\tau} & \text{for } 0 \le t \le t_r \\ V_p \cos(\frac{2\pi}{T}t) - V_0 & \text{for } t_r \le t \le T \end{cases} \tag{4.4}$$

It is clear that $v_o(0) = V_p - V_0$. If the time constant, $\tau$, is large compared to the period, $T$, $t_r$ is very close to $T$. Using the approximation $t_r \approx T$, we can find $v_o(t_r)$:

$$v_o(t_r) \approx (V_p - V_0)e^{-T/\tau} \quad \text{if} \quad T \ll \tau \tag{4.5}$$

Since $T/\tau \ll 1$, we can use the first two terms of the Taylor series for the exponential function ($e^{-x} \approx 1 - x$ for $x \ll 1$):

$$v_o(t_r) \approx (V_p - V_0)\left(1 - \frac{T}{\tau}\right) \quad \text{if} \quad T \ll \tau \tag{4.6}$$

The peak-to-peak ripple voltage for the half-wave rectifier is given by

$$V_r = v_0(0) - v_o(t_r) \approx (V_p - V_0)\frac{T}{\tau} = (V_p - V_0)\frac{T}{RC} \quad \text{if} \quad T \ll \tau \tag{4.7}$$

If $T \ll \tau$ is not satisfied (as in the case of $RC = T$ in Fig. 4.9(c)), then we have to find the value of $t_r$ using the equation:

$$v_0(t_r) = (V_p - V_0)e^{-t_r/\tau} = V_p \cos(\frac{2\pi}{T}t_r) - V_0 \tag{4.8}$$

Since an analytical solution does not exist, we must use numerical techniques to find the solution.

**Example 31**

Find the peak-to-peak ripple voltage amplitude for the half-wave rectifier when $v_{in}(t) = 25\cos(2\pi 50t)$, $V_0$=0.7 V, $R = 470\ \Omega$ and $C = 1000\ \mu$F.

We have $T = 1/50$Hz=20 ms. $\tau = RC = 470 \cdot 1000 \cdot 10^{-6} = 470$ms. Since $T \ll \tau$, we can use Eq. 4.7:

$$V_r \approx (V_p - V_0)\frac{T}{\tau} = (25 - 0.7)\frac{20}{470} = 1.0\ \text{V}_{pp}$$

This circuit is also solved with LTSpice circuit simulator (see page 300 for a tutorial on LTSpice). The capacitor voltage and diode current obtained after a transient simulation are given in Fig. 4.10. The results show that the peak-to-peak ripple voltage is 0.95 V (our estimate of 1 V is reasonably close). The peak diode current is 1.68 A, while the average resistor current is only 50.5 mA. Note that the diode current exists only during a short period while the capacitor is being charged.

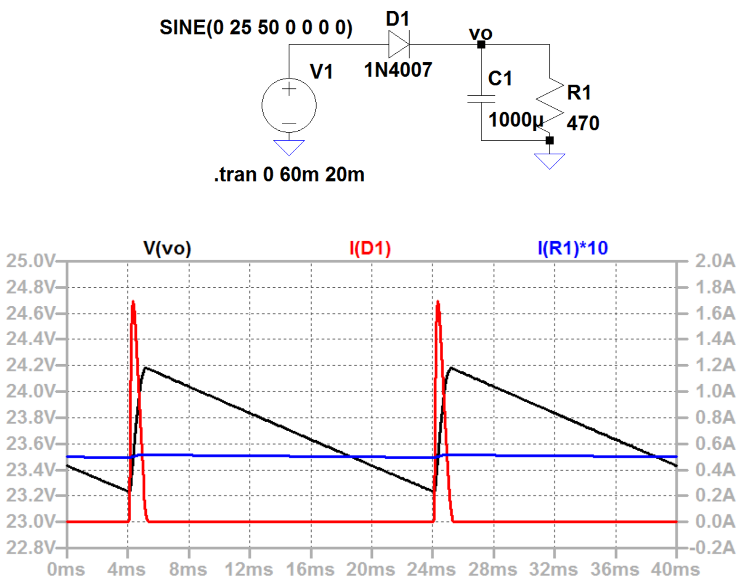



Figure 4.10: LTSpice schematic of the half-wave rectifier and the results of LTSpice simulation for the output voltage, the current in the diode, and the resistor.

### Newton-Raphson method

We can use the Newton-Raphson method (named after English Mathematician and Physicist Isaac Newton (1642–1727) and English Mathematician Joseph Raphson (1648–1725)) the find the zero of a function $f(x)$ in an iterative manner:

$$x_{n+1} = x_n - \frac{f(x_n)}{f'(x_n)} \tag{4.9}$$

where $f'(x)$ is the derivative of the function $f(x)$. We stop the iterations, when $x_{n+1}$ and $x_n$ are very close to each other.

### Example 32

Find the peak-to-peak ripple voltage amplitude for the previous example when $R$=220 Ω and $C = 100\mu$F.

$\tau = RC = 220 \cdot 100 \cdot 10^{-6} = 22$ms. Since $T \ll \tau$ is not satisfied, we must find the intersection point, $t_r$, numerically from

$$v_o(t_r) = (V_p - V_0)e^{-t_r/\tau} = V_p \cos(\frac{2\pi}{T}t_r) - V_0$$

or

$$24.3e^{-t_r/\tau} = 25\cos(\frac{2\pi}{T}t_r) - 0.7$$

To find the value of $t_r$, let us use Newton-Raphson method to find the zero of the function

$$f(t) = 24.3e^{-t/\tau} - 25\cos(\frac{2\pi}{T}t) + 0.7 = 0$$

First, we find the derivative of $f(t)$:

$$f'(t) = -\frac{24.3}{\tau}e^{-t/\tau} + 25\frac{2\pi}{T}\sin(\frac{2\pi}{T}t)$$

Therefore, the iteration formula is

$$t_{n+1} = t_n - \frac{24.3e^{-t/\tau} - 25\cos(\frac{2\pi}{T}t) + 0.7}{-\frac{24.3}{\tau}e^{-t/\tau} + 25\frac{2\pi}{T}\sin(\frac{2\pi}{T}t)}$$

We begin by an initial estimate of $t_1 = 0.9T$=18 ms. We find in the consecutive iterations, $t_2$=16.3, $t_3$=16.6, $t_4$=16.6. Hence, $t_r$=16.6 ms and the peak-to-peak ripple voltage is $V_r = 24.3(1-e^{-t_r/\tau})$=12.9 V. With the approximate expression of Eq. 4.7, we get 22 V, not very close to the correct value.

A MATLAB program to perform the Newton-Raphson iterations is given below:

```
% MATLAB program to demonstrate
% Newton-Raphson method
clear all
T=1/50; % period
R=220; % resistance value
C=100e-6; % capacitance value
tau=R*C; % time constant
Vp=25; V0=0.7;

error=1e-6; % error for stop condition
t=0.9*T; % initial estimate
condition=true; % initialize condition

while condition % repeat while condition is true
f=(Vp-V0)*exp(-t/tau)-Vp*cos(2*pi*t/T)+V0;
fprime=-(Vp-V0)/tau*exp(-t/tau)+Vp*2*pi/T*sin(2*pi*t/T);
tnew=t-f/fprime; % Newton-Raphson iteration step
condition=(abs(tnew-t)>error); % check condition
t=tnew; % get ready for new iteration
end

ripple=(Vp-V0)*(1-exp(-t/tau)) % result
rippleEst=(Vp-V0)*T/tau % approx result
```

---

### 4.1.5 Full-wave rectifier

If we have two identical AC sources, we can use both halves of the cycle to get a full-wave rectifier, as depicted in Fig. 4.11(a).

- In the positive half-cycle of $v_{AC}$, $D_1$ conducts, and $D_2$ is OFF. While the energy is supplied by the upper AC source, the capacitor $C$ gets charged to the peak value minus the voltage drop across the diode: $V_p - V_0$.

- In the negative half-cycle, the lower AC source provides the current since $D_2$ is conducting, and $D_1$ is OFF.

As shown in Fig. 4.11(b), the capacitor is charged in the same direction to the same peak value: $V_p - V_0$. Thus, the capacitor is charged to the peak value twice in one cycle, reducing the amplitude of ripple.

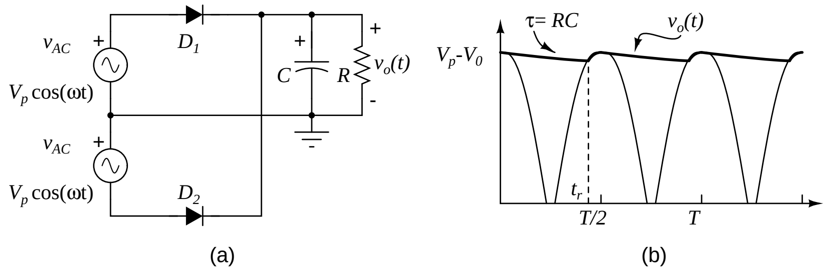



Figure 4.11: (a) A full-wave rectifier utilizing two identical AC sources, (b) the voltage $v_o(t)$ across the capacitor.

### Ripple estimation for the full-wave rectifier

For the full-wave rectifier, the period is halved compared to the half-wave rectifier. If the time constant, $\tau = RC$, is much greater than half the period of the sine wave, we estimate the intersection of the exponential with the sine wave as $t_r \approx T/2$. Hence, we can use the modified form of Eq. 4.7.

The peak-to-peak ripple for the full-wave rectifier is

$$V_r \approx (V_p - V_0)\frac{T/2}{\tau} = (V_p - V_0)\frac{T}{2RC} \quad \text{if} \quad T/2 \ll \tau \tag{4.10}$$

### Two full-wave rectifiers

It is possible to combine two full-wave rectifiers using two identical AC sources to generate positive and negative DC voltages. As shown in Fig. 4.12(a), this configuration makes use of both halves of both sources.

- In the positive half-cycle of $v_{AC}$, $D_1$ and $D_4$ conduct while the other two diodes are OFF. The upper AC source charges $C_1$ to $V_p - V_0$ through $D_1$, and the lower AC source charges $C_2$ to $-(V_p - V_0)$ through $D_4$.
- In the negative half-cycle, $D_2$ and $D_3$ are conducting. The lower source charges $C_1$ through $D_2$, and the upper source charges $C_2$ through $D_3$ to the same peak value.

We get a positive DC voltage as well as a negative DC voltage. Obviously, the ripple expression is the same as a full-wave rectifier.

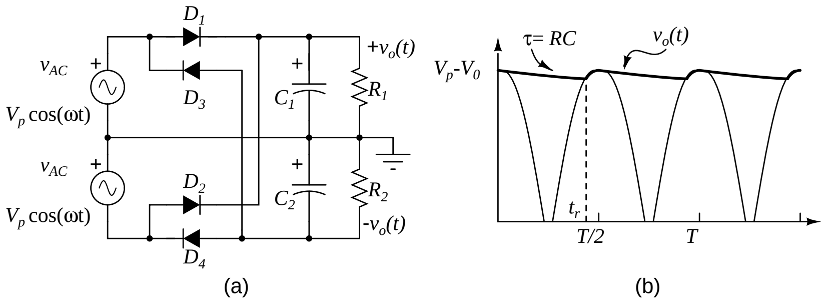



Figure 4.12: (a) Two full-wave rectifiers utilizing two identical AC sources, (b) the voltage $\pm v_o(t)$ across the load resistors.

### 4.1.6 Bridge rectifier

A common way of rectifying a *single* AC voltage is to use four diodes instead of one, as shown in Fig. 4.13. We utilize the negative half cycles as well as positive ones of the same AC source. The four-diode configuration is called a bridge, and the circuit is called a *bridge rectifier*. Four diodes are widely available commercially in a single package for use in bridge rectifiers.*

- When $v_{AC}$ is in its positive phase, $D_2$ and $D_4$ conduct, and current flows through $D_2$, the capacitor, and $D_4$ until the capacitor is charged up to the peak value, $V_p - 2\ V_0$. The peak voltage for $v_L$ is less than the one in a single diode case because the charging voltage has to overcome the threshold voltage of two diodes instead of one.
- During the negative half-cycles, $D_1$ and $D_3$ conduct, and the capacitor is thus charged up in the negative phase as well. Since the capacitor is charged twice in one cycle of $v_{AC}$ the ripple in the waveform of Fig. 4.13(c) is nearly the same as that in the full-wave rectifier.

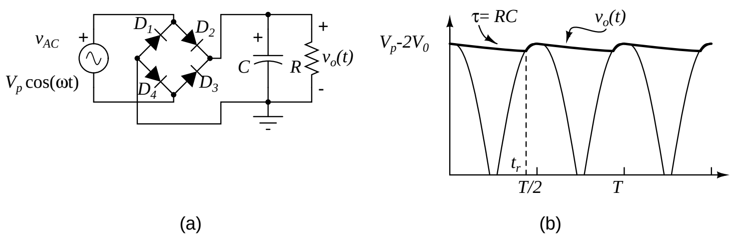



Figure 4.13: (a) Bridge rectifier, (b) rectified output voltage without capacitor (thin curve), and filtered output voltage (thick curve).

*The same four-diode bridge configuration can also be used for the two full-wave rectifiers of Fig. 4.12(a).

**Ripple estimation for bridge rectifier**

For the bridge rectifier, the period is the same as the full-wave rectifier. However, the peak voltage is $V_p - 2V_0$ rather than $V_p - V_0$. If the time constant, $\tau = RC$, is much greater than half the period of the sine wave, we can use the modified form of Eq. 4.10.

The peak-to-peak ripple for the bridge rectifier is

$$V_r \approx (V_p - 2V_0)\frac{T/2}{\tau} = (V_p - 2V_0)\frac{T}{2RC} \quad \text{if} \quad T/2 \ll \tau \tag{4.11}$$

**Example 33**

Find the peak-to-peak ripple voltage amplitude for the bridge rectifier when $v_{AC}(t) = 25\cos(2\pi 50t)$, $R = 470\Omega$ and $C = 1000\mu\text{F}$.

From Eq. 4.11:

$$V_r \approx (V_p - 2V_0)\frac{T/2}{\tau} = (25 - 1.4)\frac{10}{470} = 0.5\text{V}_{pp}$$

The ripple voltage is half of the ripple for the half-wave rectifier circuit of p. 143 with the same $R$ and $C$ values.

---

## 4.1.7 Zener diodes as voltage sources

Zener diodes are p-n junction diodes with well-defined and relatively small breakdown voltages. They are used as a DC voltage reference in the vicinity of breakdown voltage as shown in Fig. 4.14. The symbol for the zener diode is also depicted in the same figure. It is named after its inventor, Clarence Melvin Zener, an American physicist (1905–1993).

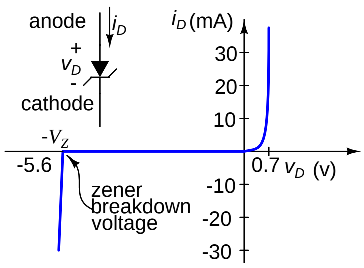



Figure 4.14: A 5.6V zener diode and its characteristics.

$I - V$ characteristics of a zener diode can be expressed as

$$v_D = \begin{cases} 0.7 \text{ V (ON)} & \text{if } i_D > 0 \\ -V_Z \text{ (Zener)} & \text{if } i_D < 0 \end{cases} \qquad i_D = 0 \text{ (OFF)} \quad \text{if } -V_Z < v_D < 0.7 \tag{4.12}$$

When a zener diode is used in a circuit given in Fig. 4.15(a), a reverse diode current

$$I = -I_D = \frac{V_S - V_Z}{R} \tag{4.13}$$

, flows through the diode as long as $V_S > V_Z$. $V_o = V_Z$ appears across the diode *independent* of the value of $V_S$ as long as $V_S > V_Z$. We call this action "voltage regulation". On the other hand, if $V_S$ is less than $V_Z$, the diode is no longer in the breakdown region, and behaves like an open circuit. In that case, we have $V_o = V_S$, and *no* voltage regulation.

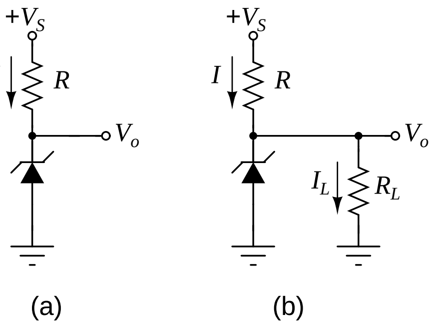



Figure 4.15: Zener diode in a voltage reference circuit.

Assume that a load resistor $R_L$ is connected across the zener diode, as shown in Fig. 4.15(b). If the zener diode is in the breakdown region and a reverse current flows, we have $V_o = V_Z$. In this case, the current through $R_L$ is

$$I_L = \frac{V_Z}{R_L} \tag{4.14}$$

To satisfy this condition, we must have

$$I = \frac{V_S - V_Z}{R} > I_L \quad \text{or} \quad V_Z < \frac{R_L}{R + R_L} V_S \tag{4.15}$$

Therefore, $V_o = V_Z$ is independent of the value of $V_S$, and voltage regulation is achieved and as long as Eq. 4.15 is satisfied. We note that under this condition, the zener diode dissipates a power of

$$P_Z = V_Z(I - I_L) = V_Z \left( \frac{V_S - V_Z}{R} - \frac{V_Z}{R_L} \right) \tag{4.16}$$

which must be less than the allowed power dissipation rating of the zener diode.

If the condition in Eq. 4.15 is not satisfied, the zener diode remains *off*. The output voltage is determined by the voltage divider formed by $R$ and $R_L$:

$$V_o = \frac{R_L}{R + R_L} V_S \tag{4.17}$$

and there is no voltage regulation.

### Water flow analogy of a zener diode

Fig. 4.16 demonstrates the water flow analogy of a zener diode. In the left figure, the water flows from left to right, the larger flap opens. If the water comes from the right with small pressure (as in the middle figure), both flaps stay closed. If the pressure from the right is sufficiently high, the smaller flap with a spring opens, as shown in the right figure.

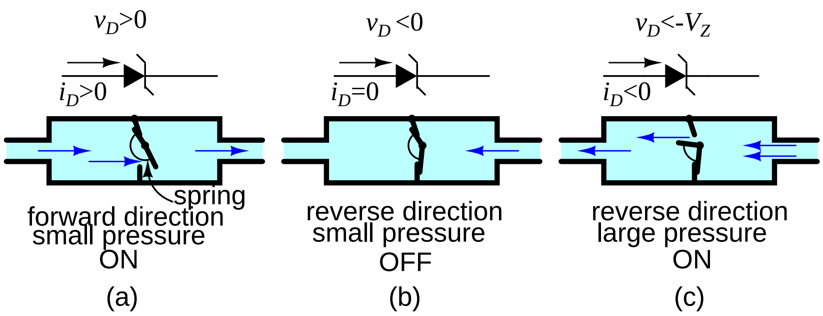



Figure 4.16: Water flow analogy of a zener diode: (a) Current/water flows in the forward direction, (b) current/water does not flow in the reverse direction, (c) current/water flows in the reverse direction with sufficient voltage/pressure.

### Example 34

Design a 5.6 V zener diode regulator circuit for a voltage source $V_S$, which varies between 10 V to 13 V (for example, due to ripple). Suppose that we have a 5.6 V zener diode, which can at most dissipate $P_{Zmax}$=300 mW. Find the smallest load resistor while the regulation is still performed. For a good regulation the minimum zener current should be 1 mA.

The maximum current that the zener diode can carry is found from its power dissipation limit:

$$-I_D = \frac{P_{Zmax}}{V_Z} = \frac{0.3\text{W}}{5.6\text{V}} = 0.053 \text{ A} = 53 \text{ mA}$$

Referring to Fig. 4.15, we choose the value of $R$ using the no-load condition (i.e., when there is no $R_L$) and under the maximum $V_S$.

$$R = \frac{V_{Smax} - V_Z}{-I_D} = \frac{13 - 5.6}{0.053} \approx 140 \ \Omega$$

Let us choose the next largest standard resistor value: $R$=150 Ω. With this $R$, the worst-case load current is supplied when $V_{Smin}$=10 V is applied. The current flowing in $R$ is

$$\frac{V_{Smin} - V_Z}{R} = \frac{10 - 5.6}{150} = 29 \text{ mA}$$

Since we must reserve at least 1 mA for the zener diode itself, we have 28 mA remaining for the load current. This current is only sufficient for a load resistor of

$$R_L = \frac{5.6}{0.028} = 210\ \Omega$$

Hence the zener diode regulator supplies a constant 5.6 V output voltage for load resistors in the range 210 $\Omega < R_L < \infty$, while the input voltage $V_S$ varies between 10 to 13 V. With any $V_S$ and $R_L$ in this range, the zener diode current is such that the output voltage is 5.6 V. Any excess current coming from the source side through $R$ flows in the zener diode, and the zener diode dissipation is less than 300 mW limit.

---

Zener diodes are useful as voltage regulators only when the load current demand is low. For higher current needs, integrated-circuit voltage regulators must be utilized.

### 4.1.8 LED

Light-emitting-diode (LED) is a special semiconductor diode that emits light when a current flows. The symbol and package polarity of an LED are shown in Fig. 4.17(a) and (b). Since it is a diode, it carries current only in one direction. The color of the light is determined by the type of semiconductor used in the fabrication of the diode. The energy bandgap of the semiconductor determines the wavelength of the emitted photons. LEDs exist in different colors of the visible spectrum as well as infrared and ultraviolet regions.

The voltage drop across an LED is higher than 0.7 V of a regular silicon diode. Red-colored LED's have a voltage drop of 1.6 to 1.7 V. The voltage drops of the infrared LEDs are lower while those of green LEDs are higher. Blue LEDs have even higher voltage drops. The current-voltage characteristic of an LED is similar to that of a silicon diode. The current increases exponentially as a function of voltage, meaning that a small change in voltage can cause a large change in current. Not to exceed the maximum current rating of LEDs, they are typically driven by a resistor in series that limits the current. Referring to

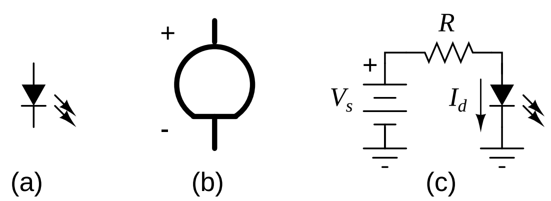



Figure 4.17: (a) LED Symbol, (b) LED package showing the polarity, (c) LED drive circuit.

Fig. 4.17(c), the value of the series resistor, $R$, should be chosen as

$$R = \frac{V_s - V_o}{I_d} \tag{4.18}$$

where $I_d$ is the desired current through the LED and $V_o$ is the voltage drop across the LED ($V_o \approx 1.6$ V for a red LED).

The light conversion efficiency of LEDs is higher than that of incandescent lamps. Modern power LEDs are being used for illumination purposes, replacing the conventional lamps. A typical LED lamp contains many LEDs connected in series to increase the light output. The schematic of a low-power LED lamp is given in Fig. 4.18. The current flowing through the LEDs is in the 30 mA to 200 mA range. The voltage to drive the LED is obtained by a bridge rectifier and an electrolytic capacitor $C_2$. To reduce the 220 $V_{rms}$ line voltage to the lower voltage necessary to drive the LEDs, a high-voltage capacitor, $C_1$, is used in series with the bridge rectifier. Since a capacitor does not dissipate any energy, it is a low-cost and energy-saving solution to reduce the voltage. A small resistor in series is used to limit the current and as a protection to reduce the risk of fire in case the series capacitor fails.

- ⧫ TRC-11 has six diodes of different types:
  - 1N4001, silicon p-n junction power diode (1) suitable for rectifying applications at frequencies below 500 Hz.
  - 1N4148, silicon p-n junction signal diode (2): suitable for low power signal applications at frequencies below 50 MHz.
  - MPN3404, PIN diode (1): a special purpose diode acting as a current-controlled variable resistor at frequencies above 1 MHz.
  - NTE3019, green and red light-emitting-diodes, LED (2)
- ⧫ TRC-11 utilizes one integrated-circuit as a linear voltage regulator to generate +6 V from +12 V.

## 4.2 Bipolar Junction Transistor (BJT)

One of the most common transistors is the *bipolar junction transistor* or BJT. It is a semiconductor device invented by William Shockley, Walter Brattain, and

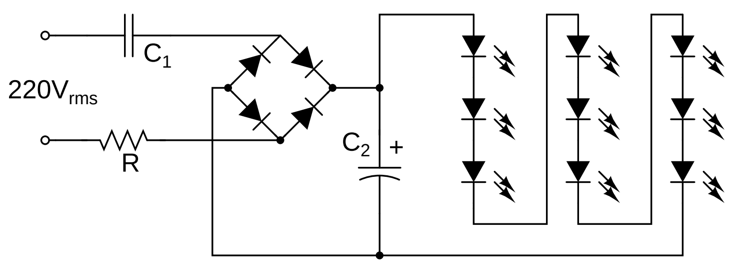



Figure 4.18: Schematic of a low-power LED lamp.

John Bardeen in 1947. A BJT is essentially two p-n junction diodes connected back-to-back, sharing either p or n regions. If the diodes share the p region, the resulting BJT is called *NPN* type. If the n-region is the shared region, the resulting BJT is of *PNP* type. The shared region of the diodes is called the *base*. The other two terminals of a BJT are called *emitter* and *collector*.

The symbols of the two types of BJTs are shown in Fig. 4.19. The arrow in the symbol is always in the emitter terminal and indicates the direction of the emitter current, $I_E$.

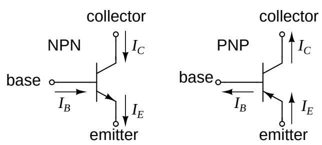



Figure 4.19: BJT symbols and current directions.

### 4.2.1 States of a BJT

The different operating states of a BJT can be summarized in Table 4.1:

| State | Emitter-base junction | Collector-base junction |
|---|---|---|
| Cutoff (OFF) | reverse-biased | reverse-biased |
| Active (ACT) | forward-biased | reverse-biased |
| Rev.-active (REVACT) | reverse-biased | forward-biased |
| Saturation (SAT) | forward-biased | forward-biased |

Table 4.1: States of a BJT in terms of the junction bias voltages.

In the cutoff state, no current flows in the emitter-base junction ($I_B$=0) and no current flows in the collector-base junction ($I_C$=0). The collector acts like an open-circuit. In the active state, we have the transistor action and high current gain results. In this state, Eqs. **??** and **??** are valid. The collector acts like a current-controlled current source. In the reverse active mode, the roles of emitter and collector are interchanged, and a low current gain results. In the saturation mode, the collector current is not determined by Eq. **??**. Hence Eqs. **??** and **??** are not valid. In that case, the collector acts like a voltage source of value $V_{Sat}$. and external circuitry determines the collector current.

Since the current gain is much smaller, the reverse-active region is not a preferred operation mode, and it should be avoided except in very rare circumstances. In this text, from this point on, the reverse-active state will not be dealt with.

We consider only the three states, cutoff (OFF), active (ACT), and saturation (SAT) states, as summarized in Table 4.2 for an NPN BJT.

In this table, $V_0$ is the turn-on voltage of the base-emitter diode, and it is in the range 0.6 to 0.8 V. $V_{Sat}$ is the collector-emitter saturation voltage, and it is in the range 0.1 to 0.3 V for small BJTs.

The states of a PNP BJT is similar as shown in Table 4.3. The polarities of base-emitter and collector-emitter voltage are changed.

| State | $V_{BE}$ | $V_{CE}$ | $I_B$ | $I_C$ |
|---|---|---|---|---|
| OFF | $< V_0$ | $> 0$ | 0 | 0 |
| ACT | $V_0$ | $> V_{Sat}$ | $I_B > 0$ | $\beta I_B$ |
| SAT | $V_0$ | $V_{Sat}$ | $I_B > 0$ | $< \beta I_B$ |

Table 4.2: The common three states of a NPN BJT.

| State | $-V_{BE} = V_{EB}$ | $-V_{CE} = V_{EC}$ | $I_B$ | $I_C$ |
|---|---|---|---|---|
| OFF | $< V_0$ | $> 0$ | 0 | 0 |
| ACT | $V_0$ | $> V_{Sat}$ | $I_B > 0$ | $\beta I_B$ |
| SAT | $V_0$ | $V_{Sat}$ | $I_B > 0$ | $< \beta I_B$ |

Table 4.3: The common three states of a PNP BJT.

We note that the current gain $\beta$ of a BJT varies in a wide range (e.g., 180 to 460) even for the same brand and the same model transistor. $\beta$ is also dependent on the value of the collector current, temperature, and age of the BJT.

## 4.3 DC Analysis of BJT circuits

BJT is a nonlinear device and it can be in one the three states. Simple DC models for different states of an NPN BJT are given in Fig. 4.20.

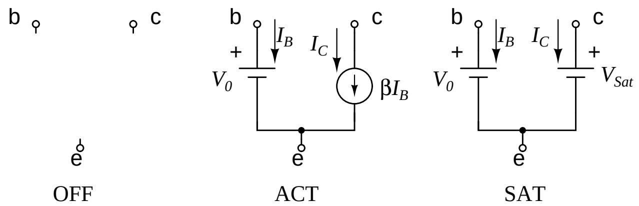



Figure 4.20: Models for an NPN BJT to be used in DC analysis.

Simple DC models for different states of a PNP BJT are given in Fig. 4.21.

The procedure to find the state of a NPN BJT is similar to the procedure to find the state of diodes as described earlier:

1. Assume a probable state for NPN BJT (start with ACT)
2. Substitute the DC model of the state.
3. Perform a DC analysis of the corresponding linear circuit.
4. Check if the conditions of the assumed state are satisfied:
   - For ACT, $I_B > 0$ and $V_{CE} > V_{Sat}$.

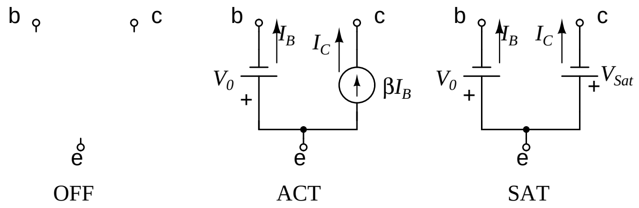



Figure 4.21: Models for an PNP BJT to be used in DC analysis.

- For SAT, $I_B > 0$ and $I_C < \beta I_B$.
- For OFF, $V_{BE} < V_0$, $V_{BC} < 0$.

If the conditions of the assumed state is satisfied, the solution is valid. Otherwise, one should try another state until the conditions of that state is satisfied.

The procedure for a PNP transistor is similar, with some quantities changing sign:

1. Assume a probable state for PNP BJT (start with ACT)
2. Substitute the DC model of the state.
3. Perform a DC analysis of the corresponding linear circuit.
4. Check if the conditions of the assumed state are satisfied:
   - For ACT, $I_B > 0$ and $-V_{CE} = V_{EC} > V_{Sat}$.
   - For SAT, $I_B > 0$ and $I_C < \beta I_B$.
   - For OFF, $-V_{BE} = V_{EB} < V_0$, $-V_{BC} = V_{CB} < 0$.

   If the conditions of the assumed state is satisfied, the solution is valid. Otherwise, one should try another state until the conditions of that state is satisfied.

**Example 1**

Find the range of values of $R_C$ such that the NPN BJT in the circuit of Fig. 4.22 stays in ACT region. We have $V_{CC}$=8 V, $V_0$=0.7 V, $R_B$=220 KΩ, $V_{Sat}$=0.2 V and $\beta$=120.

**Solution**

Assume that the NPN BJT is in the ACT state. We find $I_B$=$(V_{CC}-V_0)/R_B$=(8−0.7)/220=0.033 mA. Hence $I_C$=$\beta I_B$=3.98 mA. To be in the active region, we

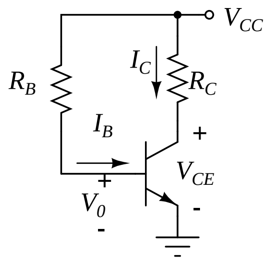



Figure 4.22: An NPN BJT circuit.

must have $V_{CE}$=$V_{CC} - R_C I_C > V_{Sat}$. Hence $R_C < (V_{CC} - V_{Sat})/I_C$=1.95 KΩ.

---

**Example 2**

Find the range of values of $R_B$ such that the PNP BJT in the circuit of Fig. 4.23 stays in ACT region. We have $V_{CC}$=6 V, $V_0$=0.7 V, $R_C$=1.8 KΩ, $V_{Sat}$=0.2 V and $\beta$ is in the range 80 to 130.

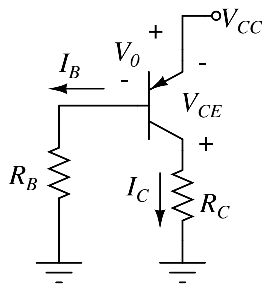



Figure 4.23: A PNP BJT circuit.

**Solution**

Assume that the PNP BJT is in the ACT state. We must have $-V_{CE} > V_{Sat}$=0.2 V. Since $-V_{CE}$=$V_{EC}$=$V_{CC}-R_C I_C$=$6-1.8I_C$ >0.2 V. Hence $I_C$ <3.22 mA. For the worst case, we use $\beta$=130. $I_B$ <3.22/130=0.025 mA. Since $I_B$=$(V_{CC} - V_0)/R_B$=$(6 - 0.7)/R_B$ <0.025 mA. We find $R_B >$ 214 KΩ. Note that a smaller value of $\beta$ will result in a smaller $I_C$, and hence guaranteeing the active state condition.

---

## 4.4 Biasing of BJTs

BJTs are normally kept in the active state to generate a current gain. Other states do not generate a current gain. Applying certain currents and voltages to a BJT to keep its active state is called *biasing*.

Biasing of BJTs must be carefully done. It is important to design biasing circuits with a small dependence on the value of $\beta$ since the current gain, $\beta$, of a BJT can vary wildly from device to device.

### 4.4.1 Simple Base Bias

The simplest biasing arrangement is shown in Fig. 4.24. The base current is

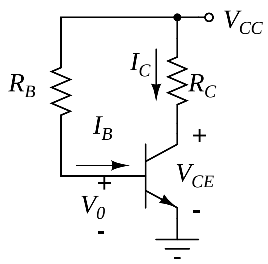



Figure 4.24: Biasing an NPN transistor using a base resistor.

determined from

$$I_B = \frac{V_{CC} - V_o}{R_B} \tag{4.19}$$

To analyze the circuit, we assume that the BJT is in ACT region. So we have $I_C = \beta I_B$. The collector voltage is determined by the the collector current and the collector resistance, $R_C$:

$$V_{CE} = V_{CC} - R_C I_C \tag{4.20}$$

where $\beta$ is the current gain of the BJT. To check our assumption of BJT being in ACT region, we must check that $V_{CE} > V_{Sat}$.

Otherwise, the BJT is in SAT region, and we have $V_{CE} = V_{Sat}$ and

$$I_C = \frac{V_{CC} - V_{Sat}}{R_C} \tag{4.21}$$

**Example 3**

Let us consider the schematic in Fig. 4.24 with $R_B$=470 KΩ, $R_C$=1 KΩ, $V_{CC}$=12 V, $V_0$=0.7 V, and $V_{Sat}$=0.2 V, while $\beta$ varies from 180 to 460 device to device for a particular transistor.

We assume that the BJT is in ACT state. With $\beta$=180 from Eq. 4.19 we find $I_B$=24 $\mu$A, from Eq. **??** $I_C$=4.3 mA, and from Eq. 4.20, we have $V_{CE}$=7.7 V. Since 7.7 > 0.2 V our ACT state assumption is valid.

For $\beta$=460, we find $I_C$=11 mA, $V_{CE}$ = 0.94 V. Since 0.94 > 0.7, the BJT is (luckily) still in the ACT state.

---

**Example 4**

Let us consider the same circuit above with $R_C$=1.2 KΩ.

We assume that the BJT is in ACT state. With $\beta$=180 from Eq. 4.20, we have $V_{CE}$=6.8 V. Since $6.8 > 0.2$ V our ACT state assumption is valid.

For $\beta$=460, we find $V_{CE} = -1.3$ V. Since $-1.3 < 0.7$, the BJT is *not* in the ACT state. It is in the SAT state. We have $V_{CE} = V_{Sat} = 0.2$ V and from Eq. 4.21, $I_C$=9.8 mA$\neq \beta I_B$.

---

### 4.4.2 Base Bias with Emitter Resistor

As the example above shows, $V_{CE}$ varies a lot as a function of $\beta$ and it is difficult to guarantee that the transistor operates in ACT state while $\beta$ of the transistor varies in a large range. A better circuit can be obtained by adding a resistor in the emitter, as shown in Fig. 4.25. Since $V_{CC} = R_B I_B + V_0 + R_E I_E$ from KVL

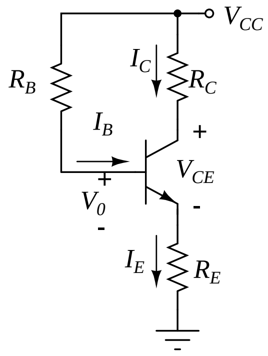



Figure 4.25: Biasing an NPN transistor using a base resistor and an emitter resistor.

and $I_E = (\beta + 1)I_B$, the base current can be found from

$$I_B = \frac{V_{CC} - V_0}{R_B + (\beta + 1)R_E} \tag{4.22}$$

and assuming the transistor in ACT region, we find $V_{CE}$ as

$$V_{CE} = V_{CC} - R_C I_C - R_E I_E \tag{4.23}$$

Again we need $V_{CE} > V_{Sat}$ for the verification of BJT ACT state operation.

If the transistor is found to be in SAT state, then we have $V_{CE} = V_{Sat}$ and use nodal analysis to find the emitter voltage:

$$\frac{V_E}{R_E} - \frac{V_{CC} - V_0 - V_E}{R_B} - \frac{V_{CC} - V_{Sat} - V_E}{R_C} = 0 \tag{4.24}$$

We can determine the collector current from

$$I_C = \frac{V_{CC} - V_{Sat} - V_E}{R_C} \tag{4.25}$$

**Example 5**

Let us consider the schematic in Fig. 4.25 with $R_B$=560 KΩ, $R_C$=1 KΩ, $R_E$=470 Ω, $V_{CC}$=12 V, $V_0$=0.7 V, and $V_{Sat}$=0.2 V while $\beta$=180–460.

First, we assume the transistor to be ACT. From Eq. 4.22, we have $I_B$=17 $\mu$A (for $\beta$=180) or $I_B$=14 $\mu$A (for $\beta$=460). From Eq. 4.23 we have $V_{CE}$=7.4 V (for $\beta$=180) or $V_{CE}$=2.1 V (for $\beta$=460). The variation in $V_{CE}$ is smaller in this case.

### 4.4.3 Conventional Bias Circuit

An even better circuit can be built by adding one more resistor in the biasing circuit (see Fig. 4.26(a)). To analyze the circuit, we first find the Thévenin

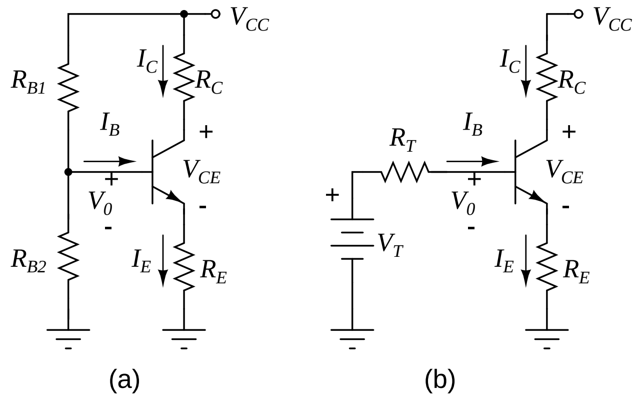



Figure 4.26: (a) Conventional biasing arrangement of an NPN transistor using two base resistors and an emitter resistor, (b) the Thévenin equivalent circuit of the base resistors.

equivalent circuit of the two base resistors and the supply voltage as shown in Fig. 4.26:

$$V_T = \frac{R_{B2}}{R_{B1} + R_{B2}} V_{CC} \quad \text{and} \quad R_T = \frac{R_{B1} R_{B2}}{R_{B1} + R_{B2}} \tag{4.26}$$

We assume ACT state for BJT and using KVL, we can find the base current as

$$I_B = \frac{V_T - V_0}{R_T + (\beta + 1) R_E} \tag{4.27}$$

$V_{CE}$ can be found from Eq. 4.23.

**Example 6**

Let us consider the schematic in Fig. 4.26(a) with $R_{B1}$=18 KΩ, $R_{B2}$=4.7 KΩ, $R_C$=1 KΩ, $R_E$=470 Ω, $V_{CC}$=12 V, $V_0$=0.7 V, and $V_{Sat}$=0.2 V while $\beta$=180–460. Find the range of $V_{CE}$ values.

**Solution**

First, we assume the transistor to be ACT. From Eq. 4.26, we have $V_T$=2.5 V and $R_T$=3.7 KΩ. From Eq. 4.27, we get $I_B$=20 $\mu$A (for $\beta$=180) or $I_B$=8.1 $\mu$A (for $\beta$=460). From Eq. 4.23 we have $V_{CE}$=6.6 V (for $\beta$=180) or $V_{CE}$=6.5 V (for $\beta$=460). Since $V_{CE} > V_{Sat}$=0.2 V, BJT is ACT for the whole range of $\beta$. Clearly, the variation in $V_{CE}$ is much smaller with this biasing circuit.

---

**Example 7**

Let us consider the PNP BJT in the schematic of Fig. 4.27(a) with $R_{B1}$=33 KΩ, $R_{B2}$=8.2 KΩ, $R_C$=2.2 KΩ, $R_E$=680 Ω, $V_{CC}$=15 V, $V_0$=0.7 V, and $V_{Sat}$=0.2 V while $\beta$=90–180. Find the range of $V_{CE}$.

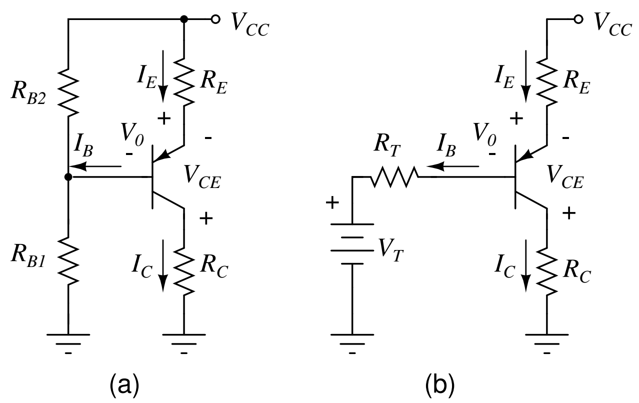



Figure 4.27: (a) Conventional biasing arrangement of a PNP transistor using two base resistors and an emitter resistor, (b) the Thévenin equivalent circuit of the base resistors.

**Solution**

First, we assume the transistor to be ACT. We have $V_T$=$(R_{B1}/(R_{B1}$+$R_{B2})V_{CC}$=12 V and $R_T$=$R_{B1} \parallel R_{B2}$=6.57 KΩ. From

$$I_B = \frac{V_{CC} - V_T - V_0}{R_T + (\beta + 1)R_E}$$

For $\beta$=90, $I_B$=0.0336 mA, $I_C$=3.02 mA, and for $\beta$=180, $I_B$=0.0177 mA, $I_C$=3.19 mA. Since $-V_{CE} = V_{CC} - R_E I_E - R_C I_C$, we find $I_E$=3.05 mA, and $-V_{CE}$=5.90 V for $\beta$=90; $I_E$=3.21 mA, and $-V_{CE}$=5.79 V. Since $-V_{CE} > V_{Sat}$=0.2 V, the PNP BJT is ACT for the whole range of $\beta$.

---

### 4.4.4 Bias Circuit with Collector Feedback

In some cases, it is not desirable to have an emitter resistance. In such cases, we can use the biasing circuit shown in Fig. 4.28. For the analysis of this circuit

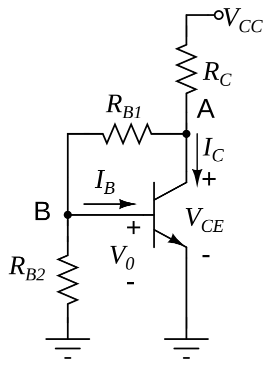



Figure 4.28: Biasing an NPN transistor using two base resistors with collector feedback.

we assume that the BJT is ACT state and write the nodal equations at nodes A and B:

$$\frac{V_{CE} - V_{CC}}{R_C} + I_C + \frac{V_{CE} - V_0}{R_{B1}} = 0 \tag{4.28}$$

$$\frac{V_0}{R_{B2}} + \frac{I_C}{\beta} + \frac{V_0 - V_{CE}}{R_{B1}} = 0 \tag{4.29}$$

since $I_B$=$I_C/\beta$. Solving for $I_C$ and substituting in the other equation, we get

$$\left(\frac{\beta+1}{R_{B1}} + \frac{1}{R_C}\right) V_{CE} = \left(\frac{\beta+1}{R_{B1}} + \frac{\beta}{R_{B2}}\right) V_0 + \frac{V_{CC}}{R_C} \tag{4.30}$$

to determine $V_{CE}$. There is no need to check that $V_{CE} > V_{Sat}$, since $V_{CE} > V_0 > V_{Sat}$ at all times. So, this biasing arrangement can never go wrong as long as $R_{B2}$ is not selected too small. Indeed, if we ignore the small terms in Eq. 4.30 we find

$$V_{CE} \approx \frac{R_{B1} + R_{B2}}{R_{B2}} V_0 \quad \text{if} \quad R_C > \frac{R_{B1}}{\beta} \tag{4.31}$$

We note that the BJT in this biasing arrangement can never be in the SAT state since $V_{CE} > V_0$ to get a positive $I_B$. On the other hand, one should make sure that

$$\frac{R_{B1} + R_{B2}}{R_{B2}} V_0 < V_{CC} \tag{4.32}$$

otherwise, the transistor is in the OFF state.

**Example 8**

Consider the schematic in Fig. 4.28 with $R_{B1}$=33 KΩ, $R_{B2}$=4.7 KΩ, $R_C$=1 KΩ, $V_{CC}$=12 V, and $V_0$=0.7 V while $\beta$=180–460.

From Eq. 4.30, we have $V_{CE}$=6.6 V (for $\beta$=180) or $V_{CE}$=6.0 V (for $\beta$=460). The variation in $V_{CE}$ is small in spite of the large variation in $\beta$.

## 4.5 Small-signal BJT Model

Small-signal models are useful to determine the gain of a BJT amplifier. The model is called *small-signal* because it is valid when the applied signals are small, and the BJT stays in the same state throughout the operation. A BJT in the OFF state can be modelled as an open-circuit at all terminals, as shown in Fig. 4.29. In the ACT state the transistor is modelled as a resistor between base and emitter and as a current source between collector and emitter. In the SAT state, the transistor is modelled with a short-circuit between collector and emitter. For ACT and SAT states, the B-E junction is modeled as a resistor whose value is determined by the DC current, $I_B$, through the base-emitter junction. The same models are valid for NPN and PNP transistors.

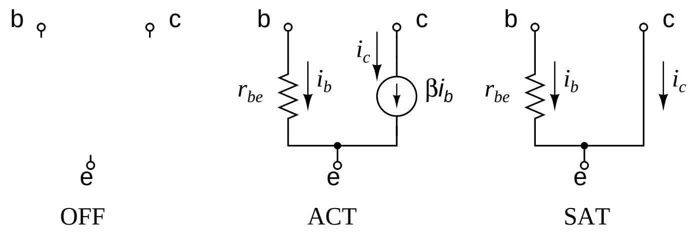



Figure 4.29: Small-signal model of a BJT (NPN or PNP) in the OFF, ACT, and SAT states.

$$r_{be} = \frac{kT}{qI_B} \tag{4.33}$$

where $k$ is the Boltzmann constant ($k = 1.38 \times 10^{-23}$ J/°K), $T$ is the temperature of the B-E junction in Kelvins, $q$ is the charge of an electron ($q = 1.6 \times 10^{-19}$ coul), and $I_B$ is the DC B-E junction current in Amperes. As the base current, $I_B$, gets larger, the small signal resistance, $r_{be}$ becomes smaller. At room temperature ($T = 300$°K) we have

$$\frac{kT}{q} \approx 0.0259 \text{ V} \tag{4.34}$$

Therefore

$$r_{be}(\Omega) \approx \frac{0.0259}{I_B(\text{A})} = \frac{25.9}{I_B(\text{mA})} \tag{4.35}$$

The small-signal analysis of a NPN (or PNP) BJT circuit in the ACT state can be summarized as follows:

1. Perform a DC analysis of the circuit by open-circuiting capacitors, short-circuiting inductors, and killing the AC small-signal sources.
2. Find the DC base current, $I_B$, and the DC collector-emitter voltage, $V_{CE}$.

3. If $I_B > 0$ and $V_{CE} > V_{Sat}$ ($-V_{CE} > V_{Sat}$ for PNP), the transistor is in the ACT (active) state. In that case, substitute the small-signal model of the transistor. Use the DC base current, $I_B$, in calculating $r_{be}$ value.

4. Keep the AC small-signal sources, and kill the DC sources.

5. If there are capacitors whose reactances are much smaller than the resistors in the circuit, they can be shorted. If there are inductors whose reactances are much larger than the resistors in the circuit, they can be open-circuited.

6. Perform a small-signal AC analysis of the circuit. If you have capacitors or inductors left, use their phasor equivalents.

Our notation is as follows: DC quantities are shown by capital letters with capital subscripts, small-signal AC quantities are denoted by lower-case letters with lower-case subscripts. DC plus AC quantities are shown with lower-case letters with capital subscripts: For example, $i_B$ (DC + AC) $=I_B$ (DC) $+i_b$ (AC).

Using the small-signal models, we can easily find the voltage gain of a BJT amplifier as exemplified below.

**Example 9**

Consider the BJT amplifier shown in Fig. 4.30(a) with $V_{CC}$=12 V, $V_0$=0.7 V, $R_B$=470 KΩ, and $R_C$=1 KΩ with $\beta$=180–420. From the DC analysis of the circuit, we know that the BJT is in the ACT state for all $\beta$ values. Hence we can use the small-signal model of Fig. 4.29. Assuming that DC-block capacitors, $C_1$ and $C_2$, are sufficiently large at the frequency of the input sinusoidal source, find the small-signal voltage gain of this amplifier ($v_{out}/v_{in}$). The DC-block capacitors are there to prevent the bias circuit to be affected by the input voltage source or output load that may be present.

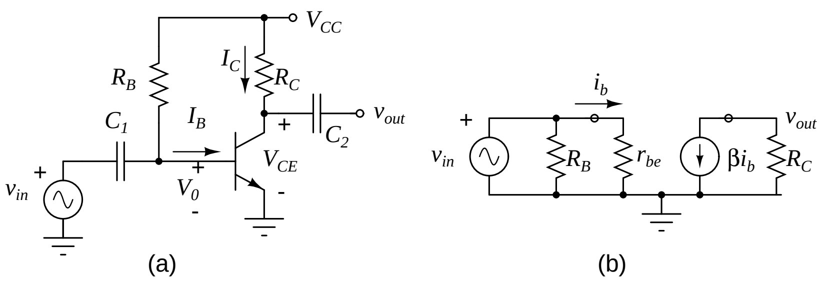



Figure 4.30: (a) A simple BJT amplifier, (b) The small-signal model of the amplifier.

We first determine the DC current flowing through the B-E junction:

$$I_B = \frac{V_{CC} - V_0}{R_B} = \frac{12 - 0.7}{470\text{K}} = 0.024 \text{ mA} \tag{4.36}$$

Then we determine the value of the small-signal resistance $r_{be}$:

$$r_{be} = \frac{25.9}{0.024} = 1077\ \Omega \tag{4.37}$$

The capacitors $C_1$ and $C_2$ act like short circuits at the operating frequency. We kill DC the power supply $V_{CC}$, hence it acts like a ground. We can draw the small-signal model of the amplifier as in Fig. 4.30(b). In this schematic all voltages and currents are small-signal AC voltages and currents and no DC voltage exists. The small-signal ac current $i_b$ is given by

$$i_b = \frac{v_{in}}{r_{be}} \tag{4.38}$$

Therefore, the small-signal output voltage is found as

$$v_{out} = -i_c R_C = -\beta i_b R_C = -\beta \frac{v_{in}}{r_{be}} R_C \tag{4.39}$$

Hence the voltage gain, $A_v$, is

$$A_v = \frac{v_{out}}{v_{in}} = -\beta \frac{R_C}{r_{be}} = \begin{cases} -180\frac{1}{1.077} = -167 & \text{for} \quad \beta = 180 \\ -460\frac{1}{1.077} = -427 & \text{for} \quad \beta = 460 \end{cases} \tag{4.40}$$

Note that the negative sign in front of the gain value signifies that there is a $180^o$ phase change from input to output. As $\beta$ varies from device to device, the voltage gain will also vary in this case.

---

**Example 10**

Let us find the small-signal voltage gain of the amplifier shown in Fig. 4.31(a) with $R_{B1}$=33 KΩ, $R_{B2}$=4.7 KΩ, $R_C$=1 KΩ, $V_{CC}$=12 V, $V_0$=0.7 V, and $\beta$=180–460. We know that the BJT is in ACT state, so we can use the small-signal model of Fig. 4.29. DC-block capacitors, $C_1$ and $C_2$ are sufficiently large to act like short-circuits at the operating frequency.

From the DC analysis of the circuit, we have $I_B$=0.029 mA, $r_{be}$=0.89 K (for $\beta$=180) and $I_B$=0.013 mA, $r_{be}$=1.99 K (for $\beta$=460). We write the node equation at node A as

$$\frac{v_{out}}{R_C} + \frac{v_{out} - v_{in}}{R_{B1}} + \beta i_b = 0 \tag{4.41}$$

We also have

$$i_b = \frac{v_{in}}{r_{be}} \tag{4.42}$$

Hence we find the voltage gain as

$$A_v = \frac{v_{out}}{v_{in}} = -\frac{\beta/r_{be} - 1/R_{B1}}{1/R_C + 1/R_{B1}} \approx -\beta \frac{R_C \parallel R_{B1}}{r_{be}} \tag{4.43}$$

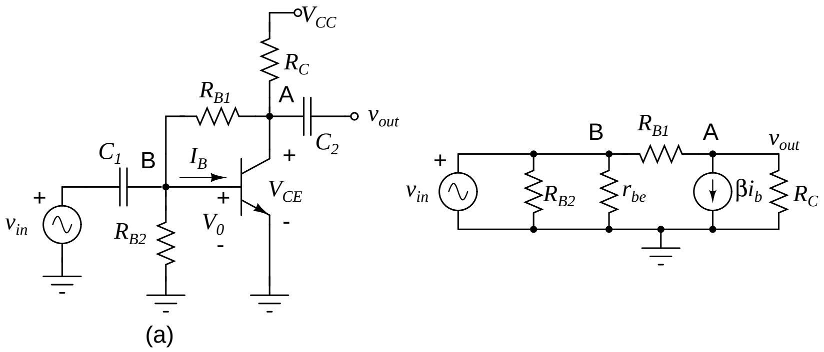



Figure 4.31: (a) A BJT amplifier with collector feedback, (b) The small-signal model of the amplifier.

or

$$A_v = \begin{cases} -196 & \text{for} \quad \beta = 180 \\ -224 & \text{for} \quad \beta = 460 \end{cases} \tag{4.44}$$

Note that the gain variation is small in spite of the large variation in $\beta$.

---

We note that if the BJT is in SAT or OFF state, no voltage gain should be expected. Therefore, a good DC biasing design is essential to get a small-signal voltage gain from a BJT.

**Example 11**

Find the gain of the circuit given in Fig. 4.32(a) with $R_B$=10 K, $R_C$=1 K, and $V_{CC}$=12 V.

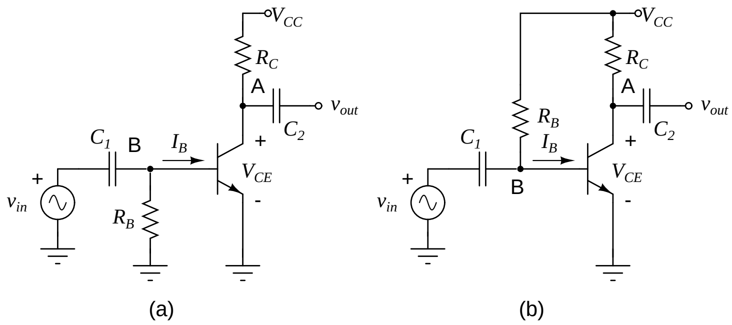



Figure 4.32: BJT circuits for examples.

In this circuit, the bias circuit is faulty. It is missing a resistor that supplies a positive voltage to the base. Since there is no current through $R_B$, we have $V_B$=0 and $I_B$=0. Since $V_B < V_0$, the BJT is in the OFF state. $I_C$=0 and $V_{CE}$=$V_{CC}$. In this case, the small-signal gain is zero.

---

**Example 12**

Find the gain of the circuit given in Fig. 4.32(b) with $R_B$=10 K, $R_C$=1 K, $V_0$=0.7 V, $V_{Sat}$=0.2 V, $\beta$=150, and $V_{CC}$=12 V.

We find the DC base current, $I_B$ as

$$I_B = \frac{12 - 0.7}{10} = 1.13 \text{ mA} \tag{4.45}$$

Assuming that the BJT is in ACT state, we have

$$I_C = \beta I_B = 150 \times 1.13 = 169.5 \text{ mA} \tag{4.46}$$

Hence the collector-emitter voltage is

$$V_{CE} = V_{CC} - I_C R_C = 12 - 169 \times 1 = -157 \text{ V !} \tag{4.47}$$

Since $V_{CE} < V_{Sat}$, the transistor is in the SAT state, and the small-signal voltage gain is zero.

---

**Example 13**

Find the gain of the PNP BJT circuit given in Fig. 4.33(a) with the same values of bias circuit example at page 160: $R_{B1}$=33 KΩ, $R_{B2}$=8.2 KΩ, $R_C$=2.2 KΩ, $R_E$=680 Ω, $V_{CC}$=15 V, $V_0$=0.7 V, and $V_{Sat}$=0.2 V while $\beta$ varies in the range 90–180. $C_1$ and $C_2$ are large capacitors and their effect can be neglected for the gain calculation.

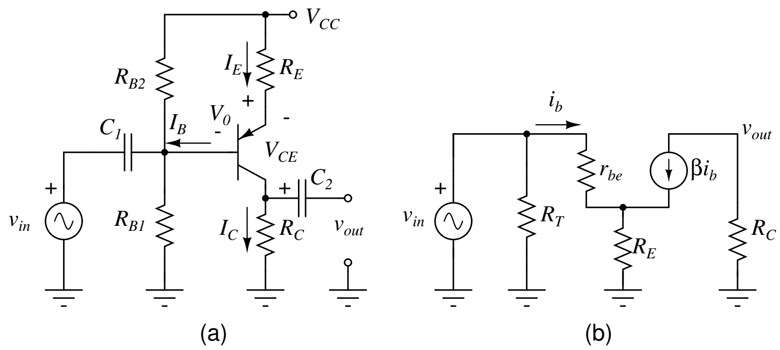



Figure 4.33: (a) PNP BJT amplifier circuit, (b) Equivalent circuit for gain calculation.

**Solution**

The bias circuit was analyzed earlier in page 160: For $\beta$=90, $I_B$=0.0336 mA, and for $\beta$=180, $I_B$=0.0177 mA. Hence we determine the value of $r_{be}$ for the range of $\beta$: For $\beta$=90, $r_{be}$=0.77 KΩ, and for $\beta$=180, $r_{be}$=1.46 KΩ. The small-signal equivalent circuit is shown in Fig. 4.33(b). Here, $R_T$ stands for the parallel combination of the bias resistors, $R_{B1}$ and $R_{B2}$. Since $v_{in}$=$r_{be}i_b + R_E(\beta+1)i_b$, we have

$$i_b = \frac{v_{in}}{r_{be} + (\beta+1)R_E} \tag{4.48}$$

For $\beta$=90, $i_b$=$v_{in}$/62.6 K, and for $\beta$=180, $i_b$=$v_{in}$/124 K. Since $v_{out}$=$-R_C\beta i_b$, the gain $v_{out}/vin$ is $-3.16$ for $\beta$=90, and $-3.19$ for $\beta$=180. The presence of the emitter resistor, $R_E$, reduces the gain, but it makes it almost independent of the value of $\beta$.

---

♦ TRC-11 uses four NPN BJTs and one PNP BJT.

## 4.6 Examples

### Example 14

Find $V_o$ versus $V_{in}$ for the circuit shown in Fig. 4.34, assuming that the diode is ideal.

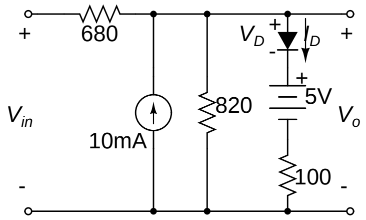



Figure 4.34: Circuit for Example 14.

**Solution**

1. Assume that the diode is OFF. We can solve the remaining part using nodal analysis:

$$\frac{V_o - V_{in}}{0.68\text{K}} + \frac{V_o}{0.82\text{K}} - 10 = 0 \text{ or } V_o = 0.547V_{in} + 3.72$$

   This equation is valid as long as $V_o < 5$ or when $V_{in} < 2.34$ V. We verify that the diode is OFF:

$$V_D = V_o - 5 < 0$$

2. For $V_{in} \geq 2.34$ V, the diode is ON. In this case, we write the nodal equation as

$$\frac{V_o - V_{in}}{0.68\text{K}} + \frac{V_o}{0.82\text{K}} - 10 + \frac{V_o - 5}{0.1\text{K}} = 0 \text{ or } V_o = 0.116V_{in} + 4.728$$

   We can verify that for $V_{in} \geq 2.34$ V, the diode has a positive current through it:

$$I_D = \frac{V_o - 5}{0.1\text{K}} = 1.16V_{in} - 2.71 \geq 0 \quad \text{for} \quad V_{in} \geq 2.34 \text{ V}$$

---

### Example 15

Find the inductance current, $i_L(t)$, and the voltage $v_o(t)$ as a function of time. Suppose that the initial inductance current, $i_L(0)$=0, and the diodes have a turn-on voltage of 0.7 V.

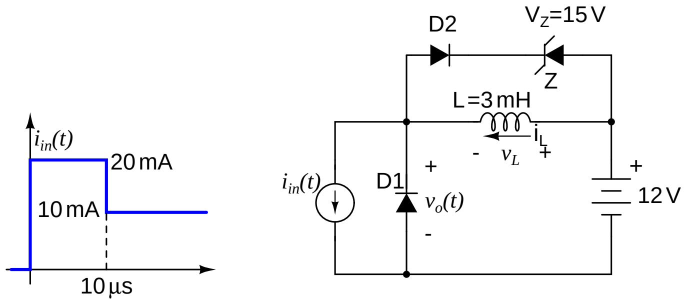



Figure 4.35: The circuit for Example 15.

**Solution**

- We have $i_{in}(0^+) = 20$ mA. Since $i_L(0+) = 0$, the current can only flow through D1. D2 and Z remains OFF.
- $v_o(0^+) = -0.7$ V and $v_L(0^+) = 12 - (-0.7) = 12.7$ V.
- Initially, the current in the inductance ramps up linearly: $i_L(t) = (1/L)v_L t = (1/3 \cdot 10^{-3})12.7t = 4.2 \cdot 10^3 t$. This continues as long as D1 is ON.
- When $i_L(t_1) = i_{in}(t_1) = 20$ mA, D1 turns OFF. We find $t_1 = 4.7$ $\mu$s. At this moment $i_L(t_1) = 20$ mA.
- For $t_1 < t < t_2 = 10$ $\mu$s, the inductor current is constant at 20 mA, so $v_L(t) = 0$ and $v_o(t) = 12$ V during this period.
- $i_{in}(t_2^+) = 10$ mA and $i_L(t_2^+) = 20$ mA. The extra inductance current of 10 mA can only flow through D2 and Z. Hence, D2 and Z turn ON. We have $v_L(t_2+) = -15.7$ V and $v_o(t) = 27.7$ V.
- Starting from 20 mA, the inductor current ramps down, since the voltage across it is negative: $i_L(t) = i_L(t_2) + (1/L)v_L(t - t_2) = 0.02 - (1/3 \cdot 10^{-3})15.7(t - t_2) = 0.02 - 5.2 \cdot 10^3(t - t_2)$. This continues as long as $i_L(t) > i_{in}(t)$.
- When $i_L(t_3) = i_{in}(t_3) = 10$ mA, $i_{in}$ is sufficient to cover the inductance current. We find $t_3$ as $0.02 - 5.2 \cdot 10^3(t - t_2) = 0.01$, or $t_3 = 15.2$ $\mu$s.
- For $t > t_3$, the inductor current is constant at 10 mA, so $v_L(t) = 0$ and $v_o(t) = 12$ V during this time.

The waveforms are plotted in Fig. 4.36.

---

**Example 16**

Find the output voltage of the half-wave rectifier circuit given in Fig. 4.37 in terms of given parameters ($\omega = 2\pi/T$). Assume that $L$ is very large and the diodes are ideal.

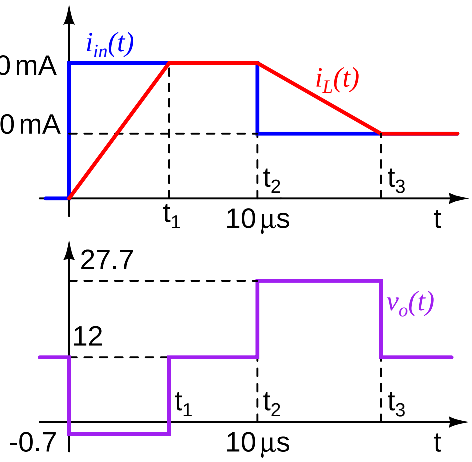



Figure 4.36: The solutions for Example 15.

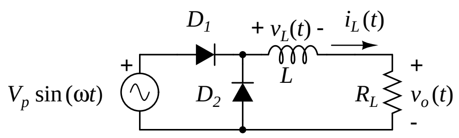



Figure 4.37: The circuit for Example 16.

**Solution**

Since the inductance is very large, we assume that the inductance current, $i_L(t) = I_L$, is a constant, and the output voltage is $v_o(t) = R_L I_L$ is also a constant. In the steady-state, the average voltage across the inductor should be zero. Otherwise, the inductance current may increase indefinitely. Since $I_L > 0$, one of the diodes should be turned on to carry that current. When the input voltage $V_p \sin \omega t$ is greater than zero, $D_1$ turns on. When the input voltage is negative, $D_2$ turns on. Hence we write the inductor voltage as

$$v_L(t) = \begin{cases} V_p \sin(\omega t) - v_o & \text{if } V_p \sin(\omega t) > 0 \\ -v_o & \text{if } V_p \sin(\omega t) < 0 \end{cases}$$

Let us find the average value of $v_L(t)$ by finding the integral over one complete cycle:

$$\frac{1}{T}\int_0^T v_L(t) = \frac{1}{T}\left(\int_0^{T/2} (V_p \sin(\frac{2\pi}{T}t) - v_o)dt + \int_{T/2}^T (-v_o)dt\right) = \frac{V_p}{\pi} - \frac{v_o}{2} - \frac{v_o}{2}$$

Hence we find

$$v_o(t) = \frac{V_p}{\pi}$$

Note that the half-wave rectifier circuit of Fig. 4.9 using a capacitor as the voltage smoothing element has a much larger output voltage of $v_o(t) = V_p$. So, the circuit of Fig. 4.37 is very rarely used.

**Example 17**

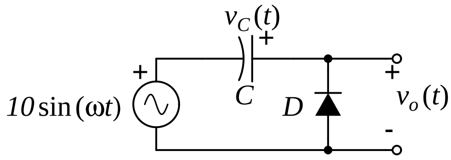



Figure 4.38: The voltage clamper circuit for Example 17.

Assuming that the initial capacitor voltage, $v_C(0) = 0$, find the output voltage, $v_o(t)$, in the *voltage clamper* circuit of Fig. 4.38. Assume that the diode has a turn-on voltage of 0.6 V.

**Solution**

In the first positive half cycle of the input, the diode remains off, and the capacitor voltage remains at zero. Hence we have $v_o(t) = 10\sin(\omega t)$ during this time. In the second half-cycle when $10\sin(\omega t_1) < -0.6$, the diode turns on. We have $v_o(t) = -0.6$ V. The capacitor voltage charges to the peak voltage of $v_c = 10 - 0.6 = 9.4$ V when $10\sin(\omega t_2) = -10$, just like the half-wave rectifier. For $t > t_2$, we have

$$v_o(t) = \begin{cases} 10\sin(\omega t) & \text{for } 0 < t < t_1 \\ -0.6 & \text{for } t_1 < t < t_2 \\ 10\sin(\omega t) + 9.4 & \text{for } t > t_2 \end{cases}$$

For $t > t_2$, we have a shifted sine wave, whose negative peaks are *clamped* to $-0.6$ V as shown in Fig. 4.36.

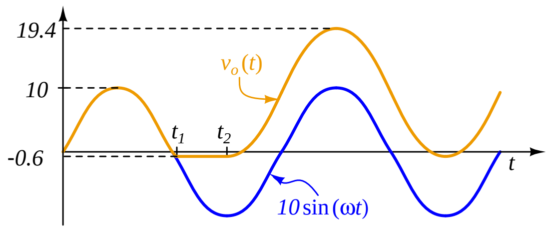



Figure 4.39: The input and output voltages of voltage clamper of Example 17.

**Example 18**

Assuming that the initial capacitor voltages, $v_{C1}(0) = v_{C2} = 0$, find the output voltage, $v_o(t)$, in the *voltage doubler* circuit of Fig. 4.40. Assume that the diodes have a turn-on voltage of 0.6 V.

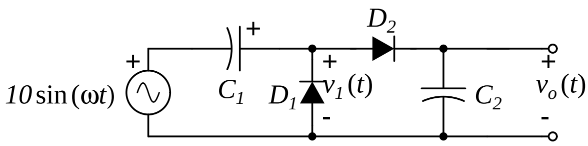



Figure 4.40: The voltage doubler circuit for Example 18.

**Solution**

$C_1$ and $D_1$ form a voltage clamper as investigated in Fig. 4.38. $D_2$ and $C_2$ form a half-wave rectifier of Fig. 4.9. After one and a quarter cycles, the output voltage becomes nearly double the peak voltage of the input voltage, as shown in Fig. 4.41.

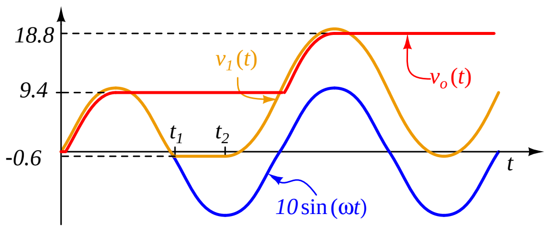



Figure 4.41: The input and output voltages of voltage doubler of Example 18.

---

**Example 19**

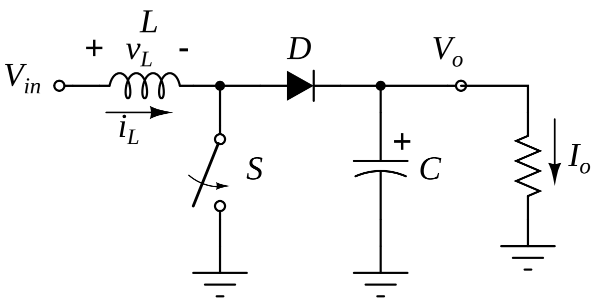



Figure 4.42: Boost converter circuit of Example 19.

Referring to Fig. 4.42, suppose that switch is turning on and off at a frequency of $1/T$. D is a Schottky diode with a forward voltage drop of $V_F$. $C$ is very large. The output current is sufficiently large so that the current $i_L$ remains positive at all times. With $V_{in} < V_o$, find the duty cycle $(t_{on}/T)$ in the steady-state.

**Solution**

Since $C$ is very large, the output voltage $V_o$ is assumed to be constant without any ripple. In the first part of the cycle, when the switch is turned on, the inductor has a constant voltage of $V_{in}$ across it. Using the equation of the inductor

$$v_L = L\frac{di_L}{dt}$$

we find that the current of the inductance increases linearly starting from an initial positive value of $I_i$:

$$i_L(t) = \frac{V_{in}}{L}t + I_i$$

At $t = t_{on}$, the change in the current is

$$\Delta i_L = \frac{V_{in}}{L}t_{on}$$

In the second part of the cycle, when the switch is turned off, the current of the inductance continues to flow. During that time, the voltage across the inductor becomes $v_L = V_{in} - (V_F + V_o) < 0$. Since this is a negative and constant value, the current of the inductance decreases linearly. The change in the current during the time interval $T - t_{on}$ is given by

$$\Delta i'_L = \frac{(V_{in} - V_F - V_o)}{L}(T - t_{on})$$

In the steady-state, the increase in the current during the first part of the cycle ($t_{on}$) should be equal to the decrease in the current during the second part of the cycle ($T - t_{on}$). We must have $\Delta i_L = -\Delta i'_L$. Hence we have

$$\frac{V_{in}}{L}t_{on} = -\frac{(V_{in} - V_F - V_o)}{L}(T - t_{on})$$

or

$$\frac{t_{on}}{T} = \frac{V_F + V_o - V_{in}}{V_F + V_o} \tag{4.49}$$

Since the average of the inductor current should be equal to the load current, we also have

$$I_o = I_i + \Delta i_L/2$$

---

**Example 20**

If a circuit has a nonlinear element, like a diode, we can still use Thèvenin or Norton equivalent circuit method for the linear parts of the circuit. In this case, the nonlinear element must be left out of the dashed-box as demonstrated in this example.

Consider the circuit given in Fig. 4.43(a) where $v_1(t)$ is an arbitrary function of time. Let us find the Thévenin equivalent of the resistive parts of the circuit: Applying the procedure:

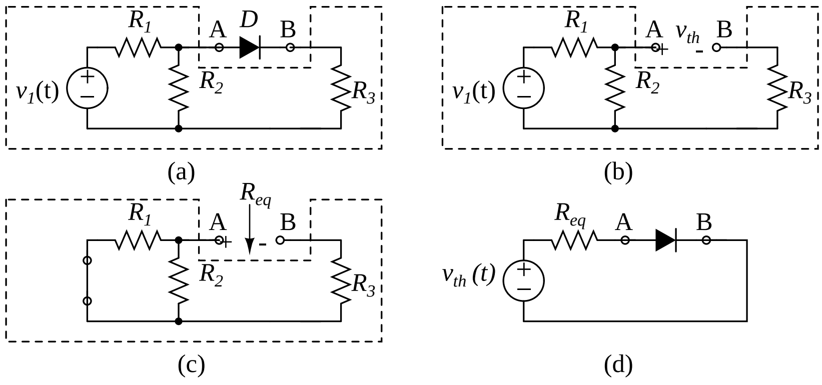



Figure 4.43: Example for Thévenin equivalent circuit for a circuit containing nonlinear element

1. Using the voltage divider formula of Eq. 2.30, we find the open-circuit voltage at terminals A-B with diode removed (Fig. 4.43(b)):

$$v_{th}(t) = \frac{R_2}{R_1 + R_2} v_1(t)$$

Note that $R_3$ does not have an effect here, since there is no current through it.

2. Kill the voltage source as in Fig. 4.43(c). Find the resistance between the terminals A and B (also with diode removed):

$$R_{eq} = \frac{R_1 R_2}{R_1 + R_2} + R_3$$

The resulting simpler circuit is shown in Fig. 4.43(d).

---

## Example 21

For the NPN BJT circuit shown in Fig. 4.44(a), we have $R_1$=22 KΩ, $R_2$=8.2 KΩ, $R_3$=120 Ω, $V_{CC}$=12 V, $\beta$=150, $V_0$=0.7 V for BJT and diodes, and $V_{Sat}$=0.2 V. Find the value of $R_c$ to set $V_{CE}$=4 V. With this value of $R_c$, what is $V_{CE}$ if $\beta$=300?

### Solution

We assume that the BJT is in ACT state since $V_{CE} = 4 > V_{Sat}$=0.2 V. We find the Thevenin equivalent circuit (Fig. 4.44(b)) with

$$.V_T = V_{CC}\frac{R_2}{R_1 + R_2} = 3.26 \text{ V} \text{ and } R_T = \frac{R_1 R_2}{R_1 + R_2} = 5.97 \text{ K}\Omega$$

From KVL at the base side going to ground via emitter path:

$$V_T = I_B R_T + V_0 + 2V_0 + R_3 I_E$$

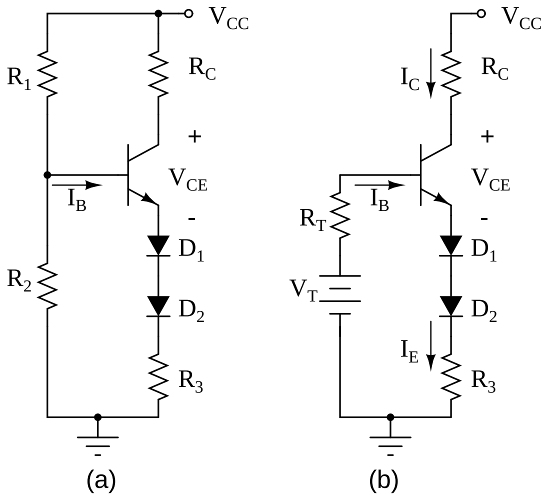



Figure 4.44: Circuit for Example 21.

Since $I_E = (\beta+1)I_B$, we write

$$I_B = \frac{V_T - 3V_0}{R_T + (\beta+1)R_3} = 0.048 \text{ mA}$$

Hence $I_C = \beta I_B = 7.21$ mA and $I_E = (\beta+1)I_B = 7.26$ mA. From KVL at the collector side going to ground via emitter path

$$V_{CE} = V_{CC} - R_C I_C - 2V_0 - R_3 I_E = 12 - 7.21R_C - 1.4 - 0.12 \times 7.26 = 4.0$$

Therefore, $R_c$=794 Ω.

If $\beta$ is changed to 300, we assume BJT is ACT and we get

$$I_B = \frac{3.26 - 2.1}{5.97 + (300+1)0.12} = 0.0276 \text{ mA}$$

With $I_C = \beta I_B = 8.26$ mA and $I_E = (\beta+1)I_B = 8.29$ mA, we get

$$V_{CE} = V_{CC} - R_C I_C - 2V_0 - R_3 I_E = 12 - 7.21 \times 0.794 - 1.4 - 0.12 \times 8.29 = 3.05$$

Since $V_{CE} > 0.2$ V, BJT is indeed ACT.

---

## Example 22

Find and plot $v_C(t)$ for the circuit shown in Fig. 4.45 in the interval $0 < t < 2$ ms. We have $V_{CC}$=24 V, $v_C(0) = 0$ V, $C$=150 nF, $R_1$=82 KΩ, $R_2$=390 Ω, $\beta$=220, $V_0$=0.7 V, $V_{sat}$=0.2 V, and $v_{IN}$=3.3 V for $t > 0$.

### Solution

Since $v_{IN} > 0.7$ V, the BJT is not OFF. We write for the base circuit:

$$v_{IN} = R_1 I_B + V_0 + R_2 I_E = R_1 I_B + V_0 + R_2(\beta+1)I_B$$

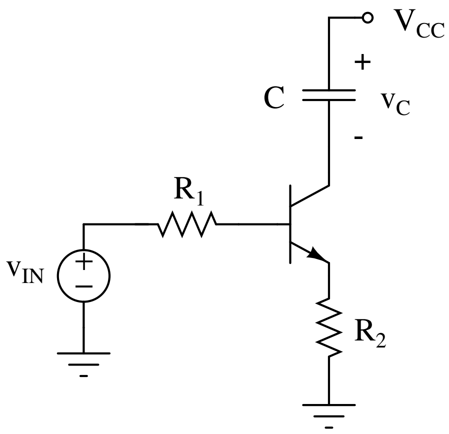



Figure 4.45: Circuit for Example 22.

Hence

$$I_B = \frac{v_{IN} - V_0}{R_1 + R_2(\beta + 1)} = \frac{3.3 - 0.7}{82 + 0.39 \times 221} = 0.0155 \text{ mA}$$

Now, assume that the BJT is ACT at $t = 0$. Hence $I_C = \beta I_B = 3.40$ mA and $I_E = (\beta + 1)I_B = 3.42$ mA. At $t = 0$ we have $V_{CE} = V_{CC} - v_C(0) - R_2 I_E = 24 - 0 - 0.39 * 3.42 = 22.7$ V. Since $V_{CE} = 22.7 > V_{sat} = 0.2$ V, the BJT is ACT at $t = 0$. The capacitor is charged with a constant current of $I_C = 3.40$ mA. Since

$$I_C = C\frac{dv_C}{dt}$$

integrating both sides, we have

$$v_C(t) = v_C(0) + \frac{I_C}{C}t = 0 + \frac{3.40 \times 10^{-3}}{150 \times 10^{-9}}t = 2.27 \times 10^4 t$$

The collector to emitter voltage of BJT is given by

$$V_{CE}(t) = V_{CC} - v_C(t) - R_2 I_E = 22.7 - 2.27 \times 10^4 t$$

So, $V_{CE}$ decreases linearly with time. The BJT becomes SAT when $V_{CE} = V_{sat} = 0.2$ V at some $t_1 > 0$:

$$V_{CE}(t_1) = 22.7 - 2.27 \times 10^4 t_1 = 0.2$$

or $t_1 = 1$ ms. What happens after $t > t_1$? Consider the equivalent circuit shown in Fig. 4.46. We have a first order circuit with an exponential solution with $v_C(t_1) = 22.7$. We find

$$v_C(\infty) = V_{CC} - V_{sat} - v_{IN}\frac{R_2}{R_1 + R_2} = 24 - 0.2 - 3.3\frac{0.39}{82 + 0.39} = 23.8$$

and the time constant is $\tau = C(R_1 \parallel R_2) = 58.2$ $\mu$s. Therefore

$$v_C(t) = 23.8 + (22.7 - 23.8)e^{-(t-t_1)/58.2\mu} \text{ for } t > t_1$$

$v_C(t)$ is plotted in Fig. 4.47 for both the ACT (the voltage increases linearly) and the SAT regions of BJT (the voltage increases exponentially).

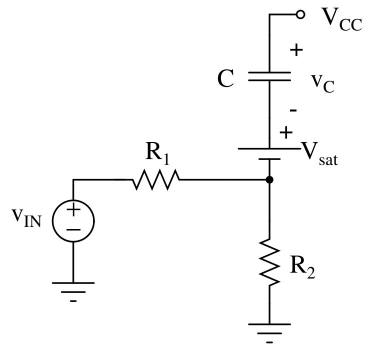



Figure 4.46: Circuit for Example 22 after the BJT became SAT.

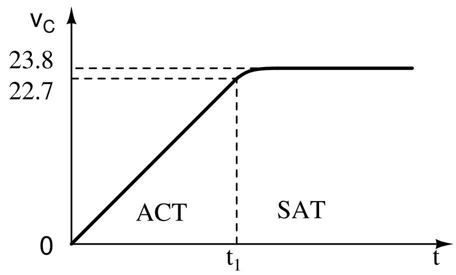



Figure 4.47: $v_C(t)$ for Example 22.

## Example 23

For the NPN BJT circuit shown in Fig. 4.48(a), find the small-signal gain $A_v = v_c/e_in$. We have $V_{BB}$=3.3 V, $R_1$=47 KΩ, $R_2$=1 KΩ, $V_0$=0.7 V, $\beta$=85, $V_{CC}$=15 V, $V_{Sat}$=0.2 V.

### Solution

We first make a DC analysis by assuming that the BJT is ACT.

$$I_B = \frac{V_{BB} - V_0}{R_1} = \frac{3.3 - 0.7}{47} = 0.055 \text{ mA}$$

Hence, $I_C$=4.7 mA. We find $V_{CE} = V_{CC} - I_C R_2$=15−4.7×1=10.3 V. Since $10.3 > 0.2$, ACT assumption is valid.

We draw the small-signal equivalent circuit as in Fig. 4.48(b) by killing the DC sources, $V_{BB}$ and $V_{CC}$, and with

$$r_{be} = \frac{0.0259}{0.055 \text{ mA}} = 0.468 \text{ K}\Omega$$

We find the small-signal base current, $i_b$, as

$$i_b = \frac{e_{in}}{R_1 + r_{be}} = \frac{e_{in}}{47.47}$$

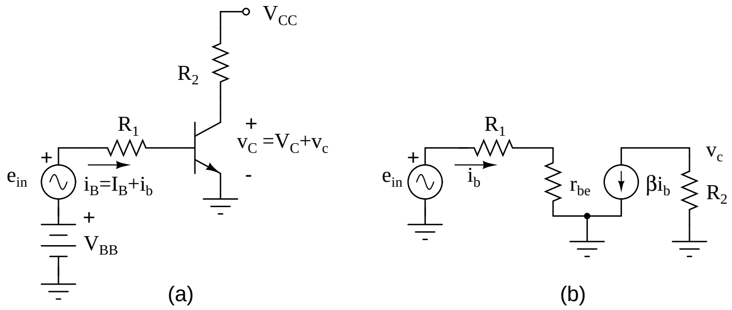



Figure 4.48: Circuit for Example 23.

The small-signal collector voltage is

$$v_c = -\beta i_b R_2 = -85\frac{e_{in}}{47.47}1 = -1.79e_{in}$$

Therefore, the small-signal gain is $A_v$=$-1.79$.

## 4.7 Problems

1. For the circuit shown in Fig. 4.49, the initial voltage of the capacitor is $v_C(0)$=0 V, and the threshold voltage of the diode, $V_o$=0.7 V. Find $v_C(t)$ for $t > 0$ and plot it. Make sure that you specify the value of $v_C(2\text{ ms})$ (It is the variable that stays continuous at discontinuities.)

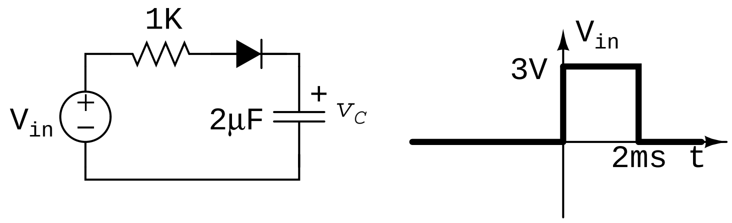



Figure 4.49: Circuit for Prob. 1

2. For the circuit of Fig. 4.50, the switch S is closed for $0 < t < 5$ $\mu$s, and then it is kept open. The initial value of the inductor current is $i_L(0) = 0$. Find and plot the current of the inductor, $i_L(t)$, as a function of time between $0 < t < 15$ $\mu$s. Assume that the diode has a forward voltage drop of 0.7 V, when it is ON.

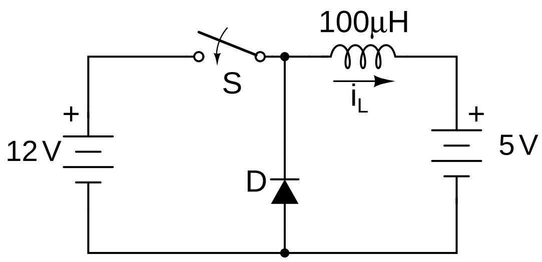



Figure 4.50: Circuit for Prob. 2

3. Find the output voltage, $v_o(t)$, in the circuit (Greinacher multiplier) of Fig. 3, assuming that the turn-on voltages of diodes are 0.6 V. (Hint: First find the voltage, $v_1(t)$.)

4. Determine the states of the BJT and collector to emitter voltages, $V_{CE}$ in the circuit of Fig. 4.26 at p. 159 with resistor values $R_{B1}$=15 KΩ, $R_{B2}$=3.9 KΩ, $R_C$=820 Ω, $R_E$=390 Ω and with parameters $V_0$=0.7 V, $V_{Sat}$=0.2 V, $V_{CC}$=12 V, and

   (a) $\beta$=100

   (b) $\beta$=200

   (c) $\beta$=400

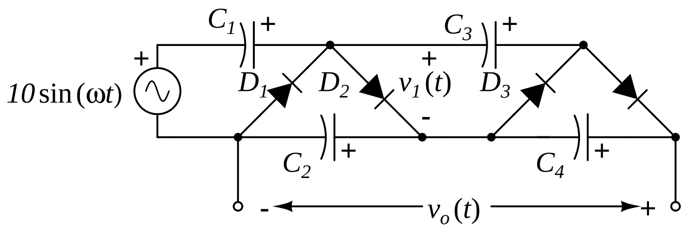



Figure 4.51: Greinacher multiplier circuit of Prob. 3

5. Determine the states of BJT and collector to emitter voltages, $V_{CE}$ in the circuit of Fig. 4.28 at p. 161 with resistor values $R_{B1}$=22 KΩ, $R_{B2}$=3.3 KΩ, $R_C$=1.5 KΩ, and with the parameters $V_0$=0.7 V, $V_{Sat}$=0.2 V, $V_{CC}$=12 V, and
   (a) $\beta$=100
   (b) $\beta$=200
   (c) $\beta$=400
6. Find the state of the BJT and the voltage, $V_C$, in the circuit of Fig. 4.52 with resistor values $R_{B1}$=2.2 KΩ, $R_{B2}$=27 KΩ, $R_E$=100 Ω, $R_C$=1.5 KΩ, and with the parameters $V_0$=0.7 V, $V_{Sat}$=0.2 V, $V_{CC}$=12 V, and

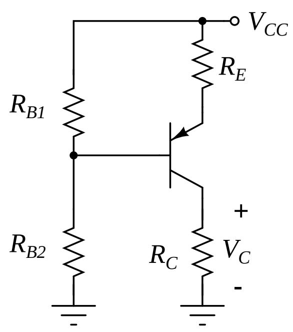



Figure 4.52: Circuit for problem.

   (a) $\beta$=70
   (b) $\beta$=150
   (c) $\beta$=250
7. Determine the state of the BJT and the collector to emitter voltage, $V_{CE}$ in the circuit of Fig. 4.53(a) with resistor values $R_{B1}$=15 KΩ, $R_{B2}$=3.9 KΩ, $R_{B3}$=56 KΩ, $R_C$=820 Ω, $R_E$=390 Ω and with the parameters $V_0$=0.7 V, $V_{Sat}$=0.2 V, $V_{CC}$=12 V and $\beta$=220.
8. Determine the collector voltage, $V_C$ in the circuit of Fig. 4.53(b) with resistor values $R_B$=120 KΩ, $R_C$=1.5 KΩ, $R_E$=220 Ω and with the parameters $V_0$=0.7 V, $V_{Sat}$=0.2 V, $V_{CC}$=6 V, $V_{EE}$=−6 V, and $\beta$=300.

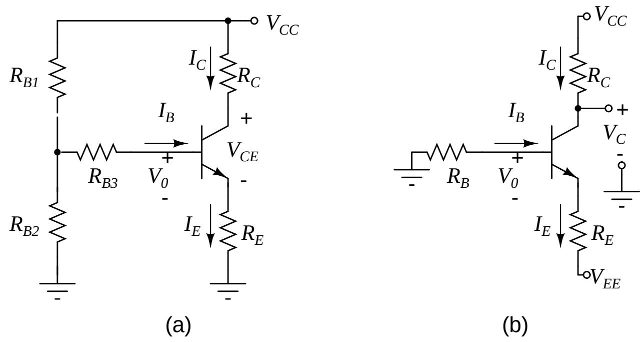



Figure 4.53: Circuits for problems.

9. Determine the state and the collector to emitter voltage, $V_{CE}$ in the circuit of Fig. 4.54 with resistor values $R_{B1}$=100 KΩ, $R_{B2}$=8.2 KΩ, $R_{C1}$=3.9 KΩ, $R_{C2}$=2.2 KΩ, and with the parameters $V_0$=0.7 V, $V_{Sat}$=0.2 V, $V_{CC}$=15 V and $\beta$=200.

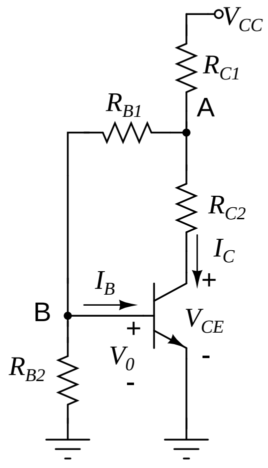



Figure 4.54: Circuit for problem.

10. Find the small-signal voltage gain, $A_V$, of the circuit of Fig. 4.30 at p. 163 with $R_B$=560 K, $R_C$=820 Ω, $V_0$=0.7 V, $V_{Sat}$=0.2 V, $V_{CC}$=15 V and $\beta$=150. First, find the base current and the state of the transistor.

11. Find the small-signal voltage gain, $A_V$, of the circuit of Fig. 4.31 at p. 165 with $R_{B1}$=27 K, $R_{B2}$=3.9 K, $R_C$=680 Ω, $V_0$=0.7 V, $V_{Sat}$=0.2 V, $V_{CC}$=15 V and $\beta$=150. First, find the base current and the state of the transistor.

12. We have a NPN BJT with parameters $V_0$=0.7 V, $V_{Sat}$=0.2 V while $\beta$=180–460. Design a BJT amplifier with a small-signal voltage gain of $-100 \pm 5$ using a supply voltage of 15 V.

13. Find the small-signal voltage gain of the circuit of Fig. 4.55(a) with $I_B$=40 $\mu$A, $\beta$=150, $R_C$=1 K, $R_E$=220 Ω. Assume that the transistor is in ACT state.

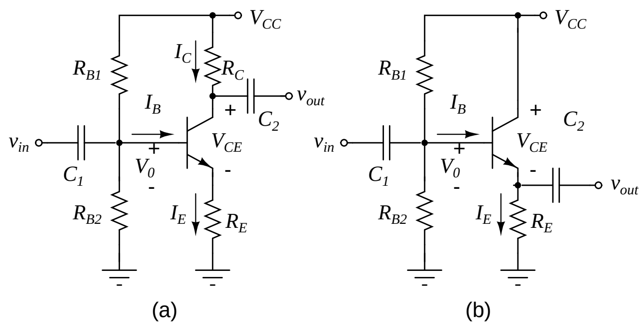



Figure 4.55: Circuits for problems.

14. Find the small-signal voltage gain of the circuit of Fig. 4.55(b) with $I_B$=40 $\mu$A, $\beta$=150, $R_E$=560 Ω. Note that BJT in this circuit cannot be in SAT state.

# Chapter 5

# TUNED CIRCUITS

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

## 5.1 Parallel $RLC$ circuit

Consider the circuit in Fig. 5.1(a). A capacitor and an inductor are connected in parallel and are driven by a current source. $I$ is the current phasor $I_p\angle 0$ of a sinusoidal source signal $i(t) = I_p \cos\omega t$ at an arbitrary frequency $\omega$. The

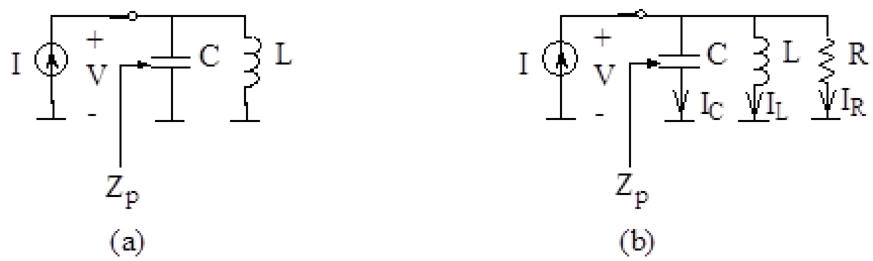



Figure 5.1: Parallel tuned circuit

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

$$Z_p(\omega) = \left(j\omega C + \frac{1}{j\omega L}\right)^{-1} = \frac{j\omega L}{1-\omega^2 LC} \tag{5.1}$$

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

$$\omega^2 LC = 1 \;\;\Rightarrow\;\; \omega = \frac{1}{\sqrt{LC}}. \tag{5.2}$$

This means that if the angular frequency is adjusted to this special angular frequency, the parallel $LC$ circuit acts as an open-circuit. This phenomenon is called *resonance*, and the frequency

$$\boxed{f_o = \frac{1}{2\pi\sqrt{LC}}} \tag{5.3}$$

$f_o$ is called the *resonance frequency.* A resonating circuit formed by a capacitor and an inductor in parallel is called a *parallel tuned* circuit. This equation can be reduced to the following formula to simplify calculations

$$\boxed{f_o^2(\mathrm{MHz}^2) = \frac{25330}{L(\mu\mathrm{H})C(\mathrm{pF})}} \tag{5.4}$$

This circuit is analogous to a water tank with diaphragm connected to a flywheel in the water-flow analogy illustrated in Fig. 5.2. The water flows in the direction

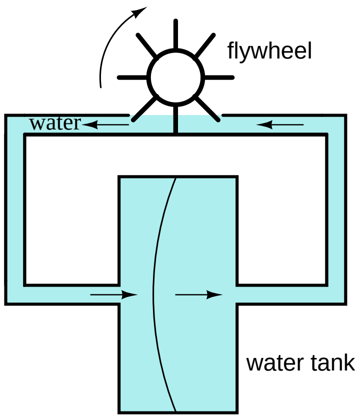



Figure 5.2: Water-flow analogy of $LC$ circuit: Water tank feeding a flywheel.

shown as the diaphragm pushes the water. The flywheel gains speed, and it takes over when the diaphragm is released. The flywheel pushes the water, and the diaphragm is stretched in the other direction. When the flywheel stops, the diaphragm pushes the water in the other direction. Hence, the back-and-forth movement of the water continues at the system's resonance frequency. The resonance frequency is determined by the moment of inertia of the flywheel and the stiffness of the diaphragm.

$LC$ circuit of Fig. 5.1(a) does not contain any loss. To introduce a loss, we should add a resistance to the circuit. Consider a slightly modified circuit given in Fig. 5.1(b). The impedance now becomes

$$Z_p(\omega) = \left(\frac{1}{R} + j\omega C + \frac{1}{j\omega L}\right)^{-1} = \frac{j\omega LR}{R + j\omega L - \omega^2 RLC} = \frac{j\omega L}{j\omega L/R + (1-\omega^2 LC)} \tag{5.5}$$

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

$$Z_p(\omega_o) = \frac{j\omega_o L}{j\omega_o L/R + 0} = R \tag{5.6}$$

Hence, the imaginary part of the impedance vanishes at resonance. As a result, the voltage across the circuit at $\omega = \omega_0$ is

$$V_p = I_p R \tag{5.7}$$

where $V_p$ is the output voltage phasor. In the time domain, this is $V_p \cos \omega_o t = I_p R \cos \omega_o t$. This means that the current through the resistor is exactly equal to the input current at resonance frequency $\omega_o$.

At the resonance, the current phasor in the capacitor branch, $I_C$ is given by

$$I_C = \frac{V_p}{1/j\omega_o C} = \frac{I_p R}{1/j\omega_o C} = I_p(j\omega_o RC) \tag{5.8}$$

If the factor $\omega_o RC$ is greater than one, we have $|I_C| > I_p$. The capacitor current can be larger than the current supplied by the source! In the time domain representation, we find

$$i_C(t) = \mathcal{Re}\{I_p j\omega_o RC e^{j\omega_o t}\} = -I_p \omega_o RC \sin \omega_o t = -V_p \omega_o C \sin \omega_o t \tag{5.9}$$

Similarly, we can find the current in the inductor branch, $I_L$ as

$$I_L = \frac{V}{j\omega_o L} = \frac{IR}{j\omega_o L} = I(-j\omega_o RC) = -I_C \tag{5.10}$$

Hence, the inductor current in the time domain is

$$i_L(t) = I_p \omega_o RC \sin \omega_o t = V_p \omega_o C \sin \omega_o t \tag{5.11}$$

A phasor diagram at the resonance is shown in Fig. 5.3(a).

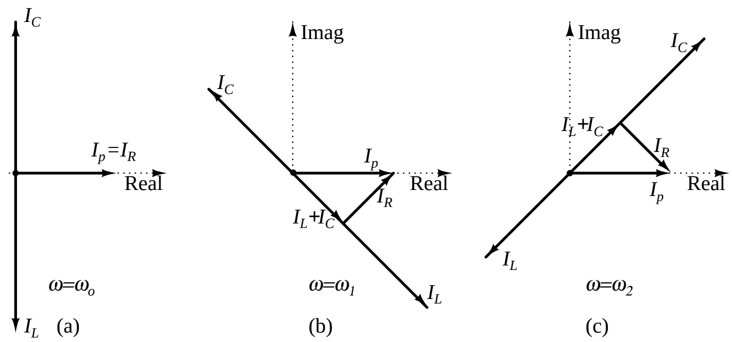



Figure 5.3: Phasor diagrams of parallel $RLC$ circuit at different frequencies.

At the resonance, the inductor current has the same magnitude as the capacitor current, but it is $180^o$ out of phase. That is why the parallel combination of $LC$ acts like an open-circuit: The current through the inductor cancels the current through the capacitor.

The resonance is an interesting phenomenon!

### 5.1.1 Energy in a parallel $RLC$ circuit

At resonance, the voltage across the capacitor is given by $v_C(t) = V_p \cos\omega_o t = I_p R \cos\omega_o t$. Hence the instantaneous stored energy in the capacitor is given by (see Eq. 2.52)

$$E_C(t) = \frac{1}{2} C v_C^2(t) = \frac{1}{2} C V_p^2 \cos^2\omega_o t \tag{5.12}$$

The peak stored energy, $E_{Cp}$, is reached when $\cos\omega_o t = 1$

$$E_{Cp} = \frac{1}{2} C V_p^2 = \frac{1}{2} C (I_p R)^2 \tag{5.13}$$

The inductor current $i_L(t)$ is given by Eq. 5.11. Hence the stored energy in the inductor is given by (see Eq. 2.61)

$$E_L(t) = \frac{1}{2} L i_L^2(t) = \frac{1}{2} L V_p^2 (\omega_o C)^2 \sin^2\omega_o t = \frac{1}{2} C V_p^2 \sin^2\omega_o t \tag{5.14}$$

The peak stored energy, $E_{Lp}$, is reached when $\sin\omega_o t = 1$

$$E_{Lp} = \frac{1}{2} C V_p^2 = \frac{1}{2} C (I_p R)^2 \tag{5.15}$$

It is interesting to observe that the total stored energy, $E_s$, is

$$E_s = E_C(t) + E_L(t) = \frac{1}{2} C V_p^2 (\cos^2\omega_o t + \sin^2\omega_o t) = \frac{1}{2} C V_p^2 = E_{Cp} = E_{Lp} \tag{5.16}$$

Therefore, the total stored energy is always constant. The energy is being transferred between the capacitor and inductor back and forth.

### 5.1.2 Quality factor

The quality factor, $Q$, of a resonator is defined at the resonance frequency as

$$Q = 2\pi \frac{\text{The total energy stored at resonance}}{\text{Energy lost per cycle}} = 2\pi \frac{E_s}{E_d} \tag{5.17}$$

where $E_s$ is the total energy stored in the lossless elements, and $E_d$ is the energy lost to dissipative elements at the resonance frequency. $Q$ is a dimensionless parameter. A higher $Q$ indicates a low rate of energy loss relative to the stored energy in the resonator.

A pendulum suspended from a high-quality bearing and oscillating in vacuum has a high $Q$, while a pendulum immersed in a liquid has a low $Q$. Resonators with high $Q$ factors ring longer. For example, a crystal wine glass rings for a long time when pinged, indicating that it is a resonator with a high $Q$ factor. $Q$ factor is approximately equal to the number of cycles of such a ring.

### 5.1.3 Quality factor of a parallel $RLC$ circuit

For a parallel $RLC$ circuit, the total energy stored in the resonator at the resonance frequency is given by (see Eq. 5.16)

$$E_s = E_{Cp} = \frac{1}{2} C V_p^2 \tag{5.18}$$

The power dissipated by the resistor is given by

$$P_d = \frac{1}{2}\frac{V_p^2}{R} \tag{5.19}$$

Hence in one cycle ($T_o = 1/f_o$ is the period at resonance frequency) the energy dissipated is

$$E_d = T_o P_d = T_o \frac{1}{2}\frac{V_p^2}{R} = \frac{1}{2f_o}\frac{V_p^2}{R} \tag{5.20}$$

Therefore, the quality factor of a parallel $RLC$ circuit is

$$\boxed{Q = 2\pi\frac{E_s}{E_d} = 2\pi f_o RC = \omega_o RC = \frac{R}{\omega_o L}} \tag{5.21}$$

In Eq. 5.21 the last equality is reached from $\omega_o C = 1/\omega_o L$. To have a large $Q$ factor, the energy stored can be increased by a large $C$, and the dissipated energy can be reduced by a large $R$.

After a little complex algebra, Eq. 5.5 can be written as,

$$Z_p(\omega) = \frac{R}{1 + jQ(\omega/\omega_o - \omega_o/\omega)} \tag{5.22}$$

in terms of $R$, $Q$, and $\omega_o$. We can see that when $\omega = \omega_o$, $|Z_p(\omega)|$ reaches a maximum at $Z_p(\omega_o) = R$ 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 the parallel $RLC$ circuit, normalized to $R$, versus frequency is given in Fig. 5.4. This impedance is calculated for an $RLC$ circuit with a $Q$ of 10. While $R_p(\omega)$

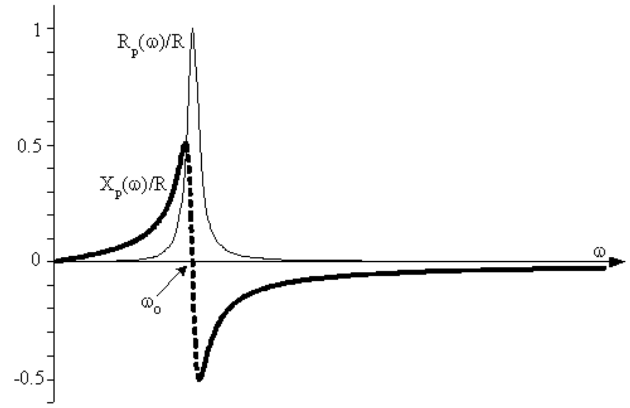



Figure 5.4: Normalized real and imaginary parts of $Z_p$.

reaches $R$ at resonance, $X_p(\omega)$ becomes zero.

There are two angular 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 $X_p(\omega_2) = -R/2$. We have

$$\boxed{\omega_1 = \sqrt{\frac{1}{LC} + \left(\frac{1}{2RC}\right)^2} - \frac{1}{2RC}} \quad \text{and} \quad \boxed{\omega_2 = \sqrt{\frac{1}{LC} + \left(\frac{1}{2RC}\right)^2} + \frac{1}{2RC}} \tag{5.23}$$

For $Q > 3.5$, these frequencies are approximately given by

$$\boxed{f_1 \approx f_o\left(1 - \frac{1}{2Q}\right)} \quad \text{and} \quad \boxed{f_2 \approx f_o\left(1 + \frac{1}{2Q}\right)} \tag{5.24}$$

where the approximation is good within 1%. The phasor diagrams showing the current phasors at these frequencies are given in Fig. 5.3(b) and (c). The difference between these frequencies is

$$\Delta f = f_2 - f_1 = \frac{f_o}{Q} \tag{5.25}$$

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. From Eq. 5.23, we can see that the angular resonance frequency, $\omega_o = 1/\sqrt{LC}$, is the geometrical mean of $\omega_1$ and $\omega_2$:

$$\boxed{\omega_o = \sqrt{\omega_1 \omega_2}} \quad \text{or} \quad \boxed{f_o^2 = f_1 f_2} \tag{5.26}$$

The variation of the magnitude of $Zp(\omega)$ with respect to angular frequency is given in Fig. 5.5. In this figure, the magnitude of impedance for a circuit with

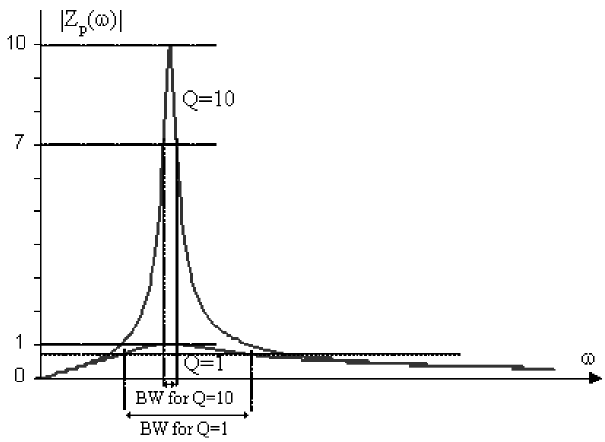



Figure 5.5: $|Z_p(\omega)|$ of two parallel tuned circuits with $Q$=1 and $Q$=10.

$Q$=10, as in Fig. 5.4, 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 $\omega_o$, but the parallel resistance of the high-$Q$ circuit is 10 times larger than the other one. Note that $|Z_p(\omega)|$ is $R/\sqrt{2} = 0.7R$ at $\omega = \omega_1$ and $\omega_2$.

The ratio of the voltage magnitudes generated across the circuit in Fig. 5.1(b) at resonance and $\omega_1$ (or $\omega_2$) is

$$\frac{|V(\omega_o)|}{|V(\omega_1)|} = \frac{|Z_p(\omega_o)|}{|Z_p(\omega_1)|} = \sqrt{2} \tag{5.27}$$

and

$$20\log_{10}\left|\frac{V(\omega_o)}{V(\omega_1)}\right| = 20\log_{10}\sqrt{2} = 10\log_{10}2 = 3\text{dB} \tag{5.28}$$

This is why the bandwidth $\Delta f = f_2 - f_1$ is called 3-dB BW.

**Example 1**

Consider $RLC$ circuit as in Fig. 5.1(b) with $C = 100$ pF and $R$=500 Ω. Find the value of $L$ to resonate the circuit at 28 MHz. Find the quality factor, $Q$, and determine $I_C$, $I_L$, and $I_R$ at resonance if $I_p$=1 mA. Find the 3 dB frequencies.

From Eq. 5.4 we find

$$L(\mu\text{H}) = \frac{25330}{f_o^2(\text{MHz}^2)C(\text{pF})} = \frac{25330}{28^2 \cdot 100} = 0.323\ \mu\text{H}$$

Using Eq. 5.21 we determine

$$Q = \frac{R}{2\pi f L} = \frac{500}{2\pi 28 \cdot 10^6 \cdot 0.323 \cdot 10^{-6}} = 8.80$$

From Eq. 5.8 and 5.21, we can find the capacitance current

$$I_C = I_p(j\omega_o RC) = jI_p Q = j1 \cdot 8.8 = j8.8\ \text{mA}$$

From Eq. 5.10, we determine the inductance current as

$$I_L = -I_C = -j8.80\ \text{mA}$$

The resistor current is

$$I_R = I_p = 1.00\ \text{mA}$$

Note that KCL is satisfied with $I_p = I_R + I_C + I_L$ $(1.00 = 1.00 + j8.80 - j8.80)$ while $|I_C|$ and $|I_L|$ are $Q$ times higher than the source current, $I_p$!

From Eq. 5.23 we find

$$f_1 = \frac{1}{2\pi}\left(\sqrt{\frac{10^{18}}{0.323 \cdot 100} + \left(\frac{10^{12}}{2 \cdot 500 \cdot 100}\right)^2} - \frac{10^{12}}{2 \cdot 500 \cdot 100}\right) = 26.45\ \text{MHz}$$

and

$$f_2 = \frac{1}{2\pi}\left(\sqrt{\frac{10^{18}}{0.323 \cdot 100} + \left(\frac{10^{12}}{2 \cdot 500 \cdot 100}\right)^2} + \frac{10^{12}}{2 \cdot 500 \cdot 100}\right) = 29.64\ \text{MHz}$$

As a check we use Eq. 5.26: $26.45 \cdot 29.64 = 784.0 = 28^2$. On the other hand, simpler expressions of Eq. 5.24 give good approximations:

$$f_1 \approx 28\left(1 - \frac{1}{2 \cdot 8.8}\right) = 26.41\ \text{MHz} \quad \text{and} \quad f_2 \approx 28\left(1 + \frac{1}{2 \cdot 8.8}\right) = 29.59\ \text{MHz}$$

---

## 5.2 Series *RLC* circuit

We have a series resonance when a capacitor and an inductor are connected in series. This is depicted in Fig. 5.6(a). The impedance of this circuit is

$$Z_s(\omega) = \frac{1}{j\omega C} + j\omega L = \frac{1 - \omega^2 LC}{j\omega C} \tag{5.29}$$

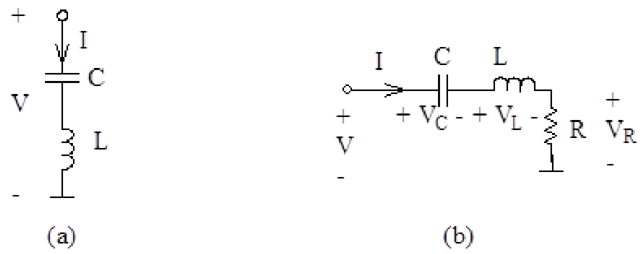



Figure 5.6: (a) Series $LC$ circuit, (b) series $RLC$ circuit.

$Z_s(\omega)$ is zero when $1-\omega^2 LC{=}0$ or $\omega=\omega_o=1/\sqrt{LC}$. This frequency is the series resonance angular frequency.

When we add a series loss element $R$ into the circuit, we obtain the circuit in Fig. 5.6(b). The impedance of the series $RLC$ circuit is

$$Z_s(\omega)=\frac{1}{j\omega C}+j\omega L+R=\frac{1-\omega^2 LC+j\omega RC}{j\omega C} \tag{5.30}$$

At the angular resonance frequency, $\omega_o$, we have $Z_s(\omega_o)=R$. At this angular frequency, the effect of series $L$ and $C$ cancels, hence only $R$ remains. If we apply a voltage of $V_s\cos\omega_o t$ across the $RLC$ circuit, the current is $I_s\cos\omega_o t=(V_s/R)\cos\omega_o t$. Hence, in phasor notation, we write

$$I_s=\frac{V_s}{R}\quad\text{at}\quad \omega=\omega_o \tag{5.31}$$

The voltage phasors across the inductor and capacitor are given by

$$V_L=I_s j\omega_o L=V_s\frac{j\omega_o L}{R}\quad\text{and}\quad V_C=\frac{I_s}{j\omega_o C}=-I_s j\omega_o L=-V_s\frac{j\omega_o L}{R}=-V_L \tag{5.32}$$

Therefore, the voltage across the inductor has the same magnitude but opposite polarity of the voltage across the capacitor. The voltages add up to zero at resonance. Also, notice that the magnitude of the inductor or capacitor voltage can be greater than the applied voltage $V_s$, if the factor $j\omega_o L/R$ is greater than one.

3-dB angular frequencies are given by

$$\boxed{\omega_1=\sqrt{\frac{1}{LC}+\left(\frac{R}{2L}\right)^2}-\frac{R}{2L}}\quad\text{and}\quad\boxed{\omega_2=\sqrt{\frac{1}{LC}+\left(\frac{R}{2L}\right)^2}+\frac{R}{2L}} \tag{5.33}$$

If $\omega_o L\gg R$, we can write the approximate 3-dB frequencies as

$$\boxed{f_1\approx f_o\left(1-\frac{1}{2Q}\right)}\quad\text{and}\quad\boxed{f_2\approx f_o\left(1+\frac{1}{2Q}\right)} \tag{5.34}$$

### 5.2.1 Energy stored in a series $RLC$ circuit

At resonance, the current through the inductor (or $RLC$ circuit) is given by $i_L(t)=I_s\cos\omega_o t=(V_s/R)\cos\omega_o t$. Hence, the instantaneous stored energy in the inductor is given by

$$E_L(t) = \frac{1}{2}Li_L^2(t) = \frac{1}{2}LI_s^2\cos^2\omega_o t \tag{5.35}$$

The capacitor voltage $v_C(t)$ can be found from the phasor given in Eq. 5.32. Hence the stored energy in the capacitor is given by

$$E_C(t) = \frac{1}{2}Cv_C^2(t) = \frac{1}{2}CI_s^2(\omega_o L)^2\sin^2\omega_o t = \frac{1}{2}LI_s^2\sin^2\omega_o t \tag{5.36}$$

Therefore, the total stored energy, $E_s$, is

$$E_s = E_L(t) + E_C(t) = \frac{1}{2}LI_s^2(\cos^2\omega_o t + \sin^2\omega_o t) = \frac{1}{2}LI_s^2 \tag{5.37}$$

Just like the parallel $RLC$ circuit, the energy is being transferred between the capacitor and inductor while the total stored energy is constant.

### 5.2.2 Quality factor of series $RLC$ circuit

For a series $RLC$ circuit, the power dissipated by the resistor is given by

$$P_d = \frac{1}{2}I_s^2 R \tag{5.38}$$

Hence in one period of $T_o$ the energy dissipated is

$$E_d = T_o P_d = T_o\frac{1}{2}I_s^2 R = \frac{1}{2f_o}I_s^2 R \tag{5.39}$$

Using Eq. 5.37, the quality factor of a series $RLC$ circuit is found as

$$\boxed{Q = 2\pi\frac{E_s}{E_d} = \frac{2\pi f_o L}{R} = \frac{\omega_o L}{R} = \frac{1}{\omega_o RC}} \tag{5.40}$$

Note that the $Q$ factor for the parallel circuit given in Eq. 5.21 is the inverse of this factor. To have a large $Q$ factor, we need to increase $L$ to increase the stored energy, and reduce $R$ to reduce the dissipated energy.

The impedance of a series $RLC$ circuit can be written as

$$Z_s(\omega) = R[1 + jQ(\omega/\omega_o - \omega_o/\omega)] \tag{5.41}$$

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

The relation between $\omega_1$, $\omega_2$, and $\omega_o$ is similar to the parallel case as shown in Fig. 5.7.

## 5.3 Equivalence of series and parallel $RLC$ circuits

We often use circuits that do not look like the parallel or series tuned circuit morphologies we discussed above. One very common form is given in Fig. 5.8. The inductors usually possess certain losses, which can be modeled by a series

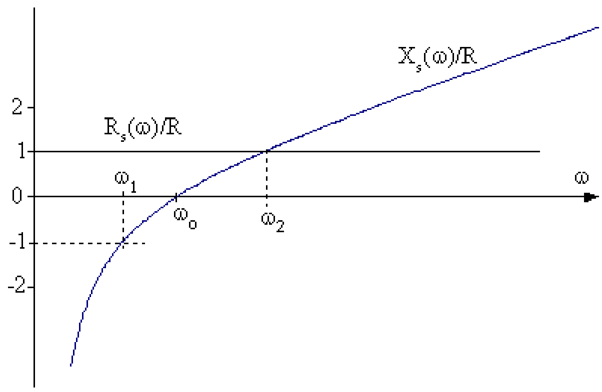



Figure 5.7: Real and imaginary parts of $Z_s(\omega)$ normalized to series resistor $R$.


Figure 5.8: Real and imaginary parts of $Z_s(\omega)$ normalized to series resistance $R$.

resistor. This kind of circuit is called the tank circuit. As far as the resonance is concerned, this circuit can be viewed as a series resonance circuit containing series-connected $C$, $L_S$, and $R_S$. However, we are interested in what appears across the capacitor terminals. The admittance of the tank circuit is

$$Y_T(\omega) = j\omega C + \frac{1}{R_S + j\omega L_S} = \frac{R_S}{R_S^2 + \omega^2 L_S^2} + j\omega \frac{R_S^2 C - L_S + \omega^2 L_S^2 C}{R_S^2 + \omega^2 L_S^2} \quad (5.42)$$

At the resonance, the imaginary part must be zero:

$$R_S^2 C - L_S + \omega_o^2 L_S^2 C = 0 \quad (5.43)$$

This condition yields

$$\omega_o = \left[\frac{1}{L_S C} - \left(\frac{R_S}{L_S}\right)^2\right]^{1/2} = \left(\frac{1}{L_S C}\frac{1}{1 + 1/Q^2}\right)^{1/2} \quad (5.44)$$

If the $Q$ is sufficiently high, the resonance frequency is similar to that of the series $RLC$ circuit. For example, for a $Q$=3.5, the resonance frequency is within 4% of $1/\sqrt{L_S C}$.

The tank circuit is equivalent to a parallel $RLC$ circuit over a certain frequency band. The series inductive branch impedance must be equal to the parallel $L_P R_P$ section impedance at the resonance frequency. For equivalence, we must have

$$R_S + j\omega_o L_S = \left(\frac{1}{R_p} + \frac{1}{j\omega_o L_p}\right)^{-1} = \frac{j\omega_o L_p R_p}{R_p + j\omega_o L_p} \quad (5.45)$$

Equating the real part of the left-hand-side to the real part of the right-hand-side, we find

$$R_S = \frac{R_p(\omega_o L_p)^2}{R_p^2 + (\omega_o L_p)^2} = \frac{R_p}{R_p^2/(\omega_o L_p)^2 + 1} \tag{5.46}$$

Since $R_p/\omega_o L_p = Q$, we have

$$\boxed{R_S = \frac{R_p}{Q^2+1}} \quad \text{or} \quad \boxed{R_p = R_S(Q^2+1)} \tag{5.47}$$

Equating the imaginary parts of Eq. 5.45 to each other, we find

$$\omega_o L_S = \frac{R_p^2 \omega_o L_p}{R_p^2 + (\omega_o L_p)^2} = \frac{\omega_o L_p}{1 + (\omega_o L_p)^2/R_p^2} \tag{5.48}$$

Using $\omega_o L_p/R_p = 1/Q$, we arrive at

$$\boxed{L_S = \frac{L_p}{1+1/Q^2}} \quad \text{or} \quad \boxed{L_p = L_S\left(1 + \frac{1}{Q^2}\right)} \tag{5.49}$$

This equivalence also maintains that the $Q$'s of two circuits are also equal:

$$\boxed{Q = \frac{R_p}{\omega_o L_p} = \frac{\omega_o L_S}{R_S}} \tag{5.50}$$

The equivalence holds at frequencies near the resonance frequency.

**Example 2**

We have an inductor with $L$=0.300 $\mu$H and a series resistor of 1 $\Omega$. Design a resonant circuit using this inductor at 8.00 MHz. Find an equivalent parallel *RLC* circuit.

From Eq. 5.50, with $L_S$=0.300 $\mu$H and $R_S$=1 $\Omega$, we find at $f$=8.00 MHz $Q$=15.1. From Eq. 5.49, we get with $L_S$=0.300 $\mu$H, $L_p$=0.301 $\mu$H. Hence we must have $C$=1314 pF. The parallel resistance, $R_p$ is found from Eq. 5.47 as $R_p$=229 $\Omega$.

---

### 5.3.1 Quality factor of a capacitor

The insulator of a capacitor has a finite resistance. In real capacitor models, this resistance ($R$) is usually included in the model in parallel with the capacitor. The quality factor of a capacitor can be defined as

$$Q(f) = 2\pi \frac{\text{Peak energy stored in the capacitor}}{\text{Energy lost per cycle}} = 2\pi \frac{E_s}{E_d} \tag{5.51}$$

Assuming that $V_p \cos(2\pi f t)$ is the sinusoidal voltage across the capacitor, $Q$ factor as a function of frequency is found as

$$Q(f) = 2\pi \frac{E_s}{E_d} = 2\pi \frac{(1/2)CV_p^2}{T_o(1/2)V_p^2/R} = 2\pi f RC = \omega RC \tag{5.52}$$

The equation above suggests that the $Q$ factor increases as the frequency increases. However, the insulator resistance decreases as the frequency is increased. As a result, capacitors also have an optimum frequency where the quality factor is maximized.

### 5.3.2 Self-resonance in capacitors

Capacitors suffer from a parasitic effect called *self-resonance*. The two leads with which the capacitor is connected to the circuit causes a small parasitic inductance in series with the capacitance. This is shown in Fig. 5.9(a). Any piece of wire has an inductance. This inductance can be calculated from the formula

$$L(nH) = 2b\left[\ln(\frac{2b}{r}) - 0.75\right] \tag{5.53}$$

where $L$ is inductance in nH, $b$ is the length of wire in cm and $r$ is the radius of the wire in cm.

Assume that we have a 100 pF capacitor with leads 1 cm each, made of 0.8 mm diameter wire. $Ls$ is the sum of the two parasitic inductances due to each lead. From Eq. 5.53, we find $L_s$=12.6 nH. The self-resonance frequency of this capacitance (see Fig. 5.9(b)) is 142 MHz. 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 capacitor is

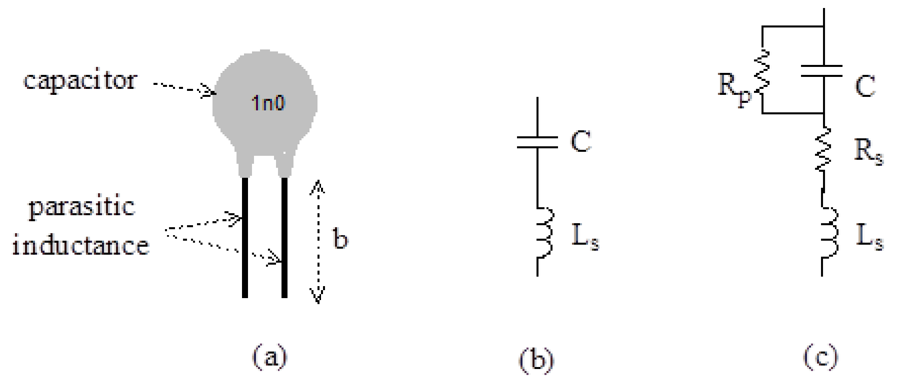



Figure 5.9: (a) A capacitor, (b) the capacitor model with parasitic inductance, (c) the full high frequency model

given in Fig. 5.9(c). The parallel resistance $R_p$ models the loss in the dielectric material from which the capacitor is made up. The series resistor $R_s$ represents the sum of conductor resistance in the leads and the losses at the lead contacts. Usually, $R_p$ is very high and can be ignored.

## 5.4 Real inductors

Inductors store electrical energy in a magnetic field. The 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 Fig. 5.10, the flux is intensified along the solenoid axis. Flux is denoted by $\phi(t)$ and its

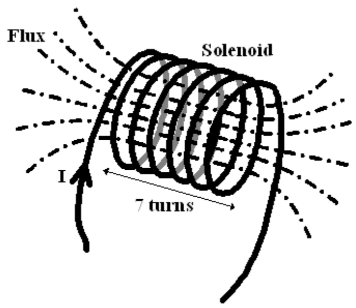



Figure 5.10: An inductor with a 7-turn solenoidal wound conductor.

corresponding phasor by $\Phi$. The flux generated by a single loop is given by

$$\Phi_1 = A_L I \tag{5.54}$$

where $\Phi_1$ is the flux phasor from one turn, $I$ is the current phasor, and $A_L$ is the inductance constant depending on dimensions of the solenoid. The flux, $\Phi$, generated by a solenoid of $N$ turns is

$$\Phi = N\Phi_1 = NA_L I \tag{5.55}$$

The voltage generated across one loop, $v_1(t)$, by this flux is

$$v_1(t) = \frac{d\phi}{dt} \tag{5.56}$$

In phasor form, we write

$$V_1 = j\omega\Phi \tag{5.57}$$

Since there are $N$ loops in the solenoid, we must add all single loop voltages to obtain the total voltage across the solenoid

$$V = NV_1 = j\omega N\Phi = j\omega(N^2 A_L)I \tag{5.58}$$

Remembering that for an inductor, we have $V = j\omega LI$, we conclude that the inductance of the solenoid is

$$\boxed{L = N^2 A_L} \tag{5.59}$$

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 [9–11] used for making inductors and transformers are:

- Air,
- Stacks of laminated steel sheets,
- Various ferrite compounds (cores shaped like rods, beads, toroids, many other forms),
- Powdered iron-based ceramics (similar to ferrites but for higher frequencies).

The laminated steel core is used for mains power transformers. The other three types are used for higher frequencies.

### 5.4.1 Air core inductors

The following formula gives the inductance of an air-core inductor depicted in Fig. 5.11 in terms of its physical dimensions

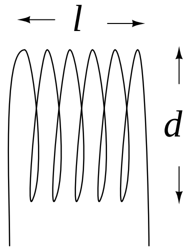



Figure 5.11: An air core inductor with a diameter of $d$ and length $l$.

$$\boxed{L(\mu\text{H}) = \frac{d^2N^2}{46\,d + 102\,l}} \tag{5.60}$$

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

**Example 3**

Let us find the number of turns and the dimensions for a 0.33 $\mu$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., $l = d$. In this case, we have

$$\boxed{L(\mu\text{H}) = \frac{dN^2}{148} \ \text{ if } \ l = d} \tag{5.61}$$

Hence for 0.33 $\mu$H inductor, we find $dN^2$=49 cm-turns$^2$. So for $l = d$ =1 cm, $N$ =7 or for $l = d = 2.5$ mm, $N = 14$ 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, the inductance value reduces.

### 5.4.2 Powdered iron core inductors

A toroidal shape inductor is shown in Fig. 5.12. We can find the inductance of the toroidal inductor from $A_L$ value of the core. For example, Micrometals T37-7 core has an $A_L$ value of 3.2 nH/turn$^2$. On this core, the number of turns necessary for $L = 1\mu$H is $N \approx 18$ turns.

It is possible to change the inductance value by making the turns tight or loose.

The cores are used to obtain high inductance values with smaller turn diameters and a smaller number of turns. This ability of the core material is determined by its *permeability*, $\mu$. The permeability of air is $\mu_o = 4\pi \cdot 10^{-7}$H/m.

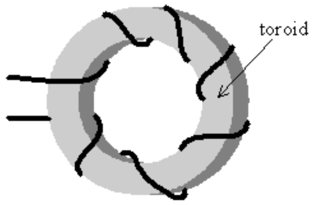



Figure 5.12: An inductor with 7-turn solenoidal wound conductor.

The permeability, $\mu$, of other materials are usually given in terms of *relative permeability*, $\mu_r$, relative to that of air, $\mu = \mu_r \mu_o$. For paramagnetic materials suitable for use as cores, $\mu_r$ is always larger than one. For example, the relative permeability of steel is about $\mu_r$=5000. The relative permeability of ferrite ranges between 50 and a few thousand, and of iron powder materials, between 10 and 50.

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

### 5.4.3 Core loss

Magnetic flux experiences loss in materials. The choice of a particular core material always depends on the amount of loss in the frequency range the inductor is used. Laminated steel sheet cores are useful only at power-line or audio frequencies. At higher frequencies, their loss becomes excessive. Ferrite materials are usable up to thee lower HF range (up to several MHz), and iron powder cores can be used in applications up to the VHF range (up to several tens of MHz). The loss at higher frequencies increases in materials of high permeability. At very high frequencies, the only acceptable core material with no associated loss is air.

The mechanism of this kind of power loss is discussed in texts on electromagnetism extensively. We confine our discussion to its effect in the choice of core materials and the design and modeling of real inductors. There is a trade-off between high $\mu_r$ and low loss.

### 5.4.4 Copper loss

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. The resistance, $R_{dc}$, of the wire at low frequencies is given by

$$\boxed{R_{dc} = \rho \frac{l}{A}} \tag{5.62}$$

where $\rho$ is the resistivity of the wire material (refer to Table 2.1 on page 19), $l$ 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. 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

$$f_{sc} = \frac{0.08}{d^2} \tag{5.63}$$

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

$$R(f) = R_{dc}\sqrt{\frac{f}{f_{sc}}} \quad \text{for} \ \ f > f_{sc} \tag{5.64}$$

The resistance increases about three times when the frequency is increased ten times. Fig. 5.13 shows current distributions in a wire at three different frequencies. Fig. 5.14 depicts the current density in a wire for different frequencies.

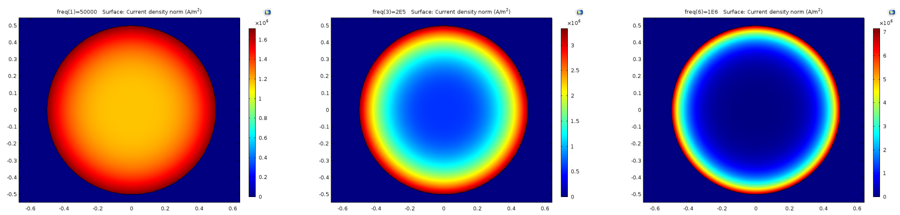


Figure 5.13: Color-coded current distribution in a 1 mm diameter copper wire at 50 kHz, 200 kHz, and 1 MHz. High current: red, low current: blue.

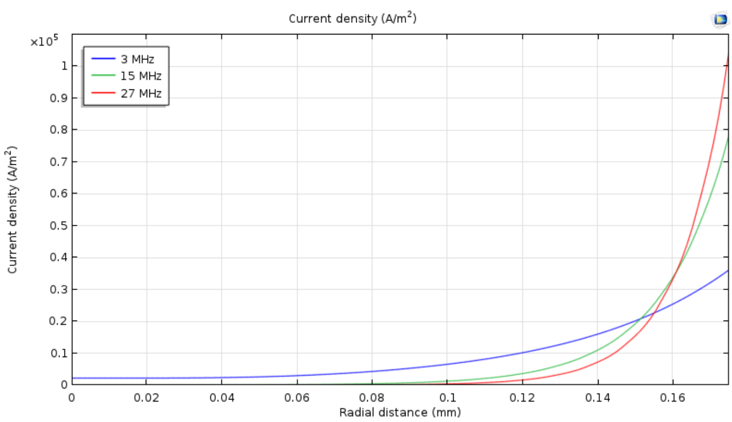



Figure 5.14: Current density in a circular wire with a current of 1 mA as a function of radial distance for different frequencies.

When the wires are next to each other, there is an additional effect called *proximity effect* which causes the current distribution in the wires be affected by the current in the neighboring wires increasing the resistance even further. Fig. 5.15 shows the current distribution in three neighboring 0.35 mm diameter wires at 3 MHz.

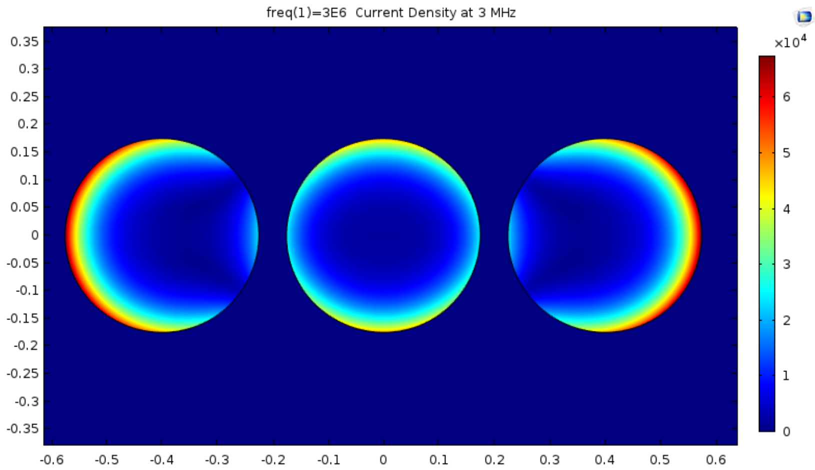



Figure 5.15: Demonstration of proximity effect: Color-coded current distribution in three 0.35 mm diameter copper wires next to each other at 3 MHz. High current: red, low current: blue.

**Example 4**

Let us determine the resistance of 10 cm of 1 mm diameter copper wire at 28 MHz. From 19, copper has a resistivity of $\rho = 1.68 \cdot 10^{-8}$ Ωm. Hence, a 10.0 cm copper wire of 1.00 mm diameter has a DC resistance of 2.14 mΩ. $f_{sc}$ for this wire is found from Eq. 5.63 as 0.08 MHz, or 80 kHz. Then, at 27 MHz $R(f)$ becomes 39.3 mΩ.

---

## 5.4.5 Quality factor of an inductor

Although an ideal inductor is a lossless element, a real inductor has finite loss due to core loss and copper loss. If there were no loss in an inductor, its model would be an inductance only. In real inductor models, a resistor in series with the inductor representing loss is usually included in the model. Since the core loss and copper loss vary with the frequency, the loss resistance in the model is frequency-dependent.

The quality factor of inductor is defined as

$$Q(f) = 2\pi \frac{\text{Peak energy stored in the inductor}}{\text{Energy lost per cycle}} = 2\pi \frac{E_s}{E_d} \tag{5.65}$$

Suppose $I_p \cos(2\pi f)$ is the sinusoidal current flowing in the inductor. $Q$ factor as a function of frequency is found as

$$Q(f) = 2\pi \frac{E_s}{E_d} = 2\pi \frac{(1/2)LI_p^2}{T_o(1/2)RI_p^2} = \frac{2\pi f L}{R} = \frac{\omega L}{R} \tag{5.66}$$

Eq. 5.66 suggests that the quality factor improves as the frequency is increased. This is true if the resistance $R$ remains constant as frequency is changed. However, the equivalent loss resistor of the inductor increases as the frequency is increased due to frequency-dependent skin effect. Moreover, core loss increases as the frequency increases. Consequently, inductors have an optimum frequency where the $Q$ factor is maximized.

Manufacturers usually provide loss data for materials in a variety of ways. One common way is $Q$-graphs with respect to frequency. Such a graph indicates that physically large cores provide higher peak $Q$ than physically small cores. Moreover, the frequency of the highest $Q$ is achieved at low frequencies for large cores, while the small cores peak at higher frequencies.

Typically, single layer winding (as opposed to two layers) is the best to achieve the highest $Q$.

### 5.4.6 Self-resonance in inductors

A parasitic effect in inductances is the *inter-winding capacitance*. There is a capacitance between neighboring turns in an inductance. This is demonstrated in Fig. 5.16(a). The value of this capacitance depends on various parameters like the physical distance between neighboring turns, the wire diameter, and the diameter of the turn. Inductors with smaller wire and coil diameters have smaller inter-winding capacitors. The distributed capacitive coupling between windings can be modeled as a parallel parasitic capacitance, $C_p$, as shown in Fig. 5.16(b). The series resistance, $R_s$, shown in the model is the total loss of the inductor.

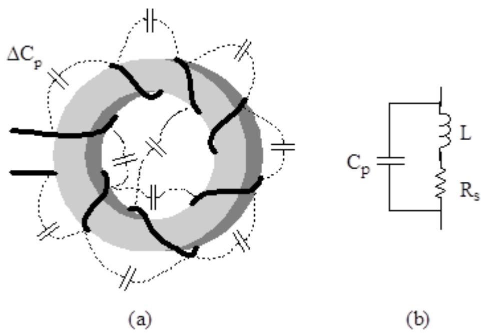



Figure 5.16: (a) An inductor, (b) the inductor model with parasitic capacitance

$C_p$ can be a significant capacitance, and it can be effective in the frequencies of interest, particularly if the inductance value is large and there are multi-layers of winding. The self-resonance frequency, $f_{sr}$, of the inductor is given by

$$f_{sr} = \frac{1}{2\pi L C_p} \text{ for } Q > 10 \tag{5.67}$$

Above the self-resonance frequency an inductor behaves like a capacitor. So the self-resonance frequency defines roughly the highest usable frequency for an inductor.

**Example 5**

Consider an inductor made by winding a single layer of 32 turns on a core with an $A_L$ of 20 nH/turn$^2$. Hence the inductance is 20.5 $\mu$H. The inductor's impedance is found to be purely resistive at 20 MHz. Hence the self-resonance frequency is 20 MHz. Assuming that the $Q$ of the inductor is larger than 10 and $L$ remains at the same value at 20 MHz, we can calculate the inter-winding capacitance as $C_p$=3 pF.

### 5.4.7 RF choke

A *choke* is an inductor used to block higher-frequency current while letting low frequency or DC current pass. If the intended blocking frequencies are RF, the inductor is called *RF choke* or RFC. While the core loss degrades the performance of an inductor used in resonant circuits, it provides useful properties in a choke. A particularly important application area is *radio frequency interference* (RFI) or *electromagnetic interference, EMI* (EMI). The interference of RF signals within the same instrument or between instruments must be avoided. Voltages or currents can couple by electromagnetic means to other parts of the circuits where they are not wanted. Interference of an irrelevant signal in a circuit can cause an instrument to malfunction.

A typical RF interference to the power supply line is shown in Fig. 5.17(a). Here, a circuit is fed by a DC power supply. There is a bypass capacitor,

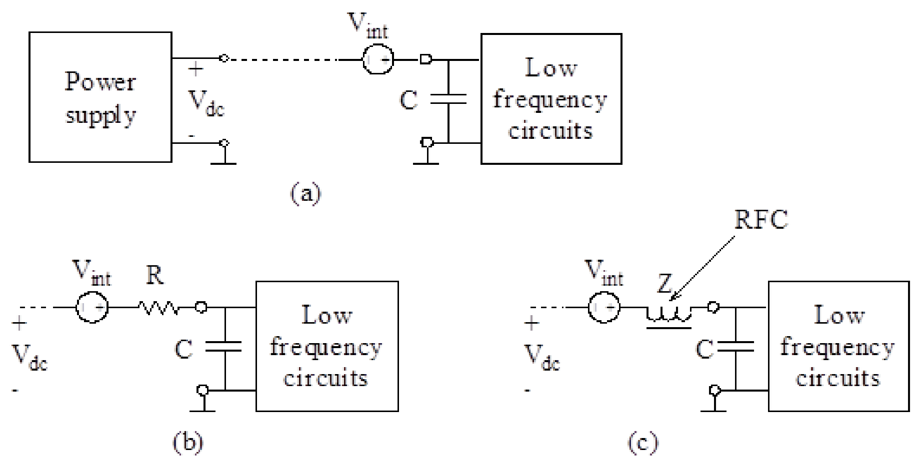



Figure 5.17: (a) An unplanned interference signal $V_{int}$ being applied to the low frequency circuits, (b) adding a series $R$ to form a low-pass-circuit with some loss at DC, (c) adding an RFC to form a low-pass-filter with no loss at DC.

$C$, at the supply terminal of the LF circuit. RF signal couples to the supply line between the power supply and the circuit electromagnetically, producing an interference signal $V_{int}$. While the circuit expects to see $V_{dc}$ only at its terminals, it experiences $V_{dc} + V_{int}$, generating an EMI problem.

One way to decrease $V_{int}$ is to include a series resistance to form an $RC$ LPF, as in Fig. 5.17(b). The DC current drawn by the LF circuit causes a DC voltage drop on $R$ in this case. The DC voltage that appears 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 DC current demand is high or varies considerably.

A better solution is to use an inductor instead of a resistor, as shown in Fig. 5.17(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 DC. Therefore, it need not be a high $Q$ inductor. Such an inductor is an RFC.

A common way of making RFC is to wind a 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 DC, since the core loss of the ferrite at low frequencies is zero. The DC supply voltage, $V_{dc}$, appears at the terminals of the LF circuit.

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

$$Z(\omega) = R_c + j\omega L \tag{5.68}$$

where $R_c$ is the resistance due to core loss. Since the existence of $R_c$ makes $|Z\omega|$ larger, 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 $1/(j\omega C)$:

$$\frac{V_{int}}{1 - \omega^2 LC + j\omega R_c C} \tag{5.69}$$

A modern way to eliminate RF noise is to use a *ferrite bead*, a cylindrical-shaped core of ferrite slipped over a wire. A typical ferrite material made from nickel-zinc alloy used for this purpose might have $\mu_r$=1000, and has a large loss factor above 1 MHz. Computer power cords often have such chokes, consisting of cylindrical ferrites encircling the cords to block noise.

## 5.5 Transformers

Transformers are the most common impedance transformation devices. The ideal transformer of Section 2.12 is depicted in Fig. 5.18 again, using phasor notation. The primary and secondary voltage and current relations in an ideal

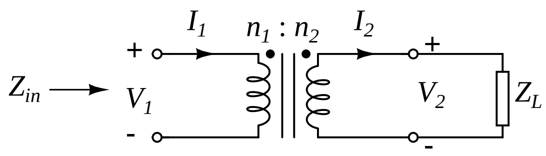



Figure 5.18: Ideal transformer in phasor notation.

transformer are

$$\boxed{\frac{V_2}{V_1} = \frac{n_2}{n_1}} \quad \text{and} \quad \boxed{\frac{I_2}{I_1} = \frac{n_1}{n_2}} \tag{5.70}$$

where $n_2/n_1$ is the ratio of the number of turns in the secondary winding to the number of turns in the primary, commonly called the *transformer turns ratio*.

### 5.5.1 Impedance transformation using an ideal transformer

If a load impedance is connected across the secondary terminals, as shown in Fig. 5.18, the secondary terminal voltage and current must satisfy the relation

$$\frac{V_2}{I_2} = Z_L$$

The impedance seen at the primary side is given by

$$Z_{in} = \frac{V_1}{I_1}$$

Using the two equations above, we obtain

$$Z_{in} = \left(\frac{n_1}{n_2}\right)^2 \frac{V_2}{I_2} = \left(\frac{n_1}{n_2}\right)^2 Z_L$$

The impedance $Z_L$ gets transformed by the $(n_1/n_2)^2$ ratio. Indeed, any shunt (i.e., parallel) or series impedance on the secondary can be transferred to primary side as a shunt or series element provided their value multiplied by $(n_1/n_2)^2$. Similarly, any impedance on the primary side can be transferred to the secondary side through multiplication by the factor $(n_2/n_1)^2$.

♦ TRC-11 has one transformer at 15 MHz, and one transformer at 27 MHz.

**Example 6**

Consider the ideal transformer shown in Fig. 5.19(a). The series impedance $Z_1$ on the primary can be moved to the secondary upon multiplication by $(n_2/n_1)^2$ (see Fig. 5.19(b)). Similarly, the shunt impedance $Z_2$ is moved to the secondary upon multiplication by the same factor as depicted in Fig. 5.19(c). All the circuits shown in the figure are equivalent to each other as far as other circuits are concerned.

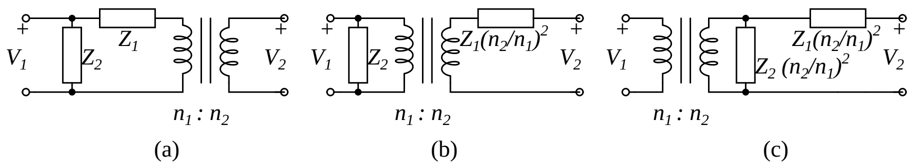



Figure 5.19: Equivalence between the circuits involving an ideal transformer.

---

### 5.5.2 Real transformers

Transformers are two (or more) coils wound on the same core. A transformer is shown in Fig. 5.20. Both windings are wound in the same sense. By convention, we show the primary current, $I_p$, flowing *into* the transformer and the secondary current, $I_s$, *out of* the transformer. Therefore, the total flux generated by two

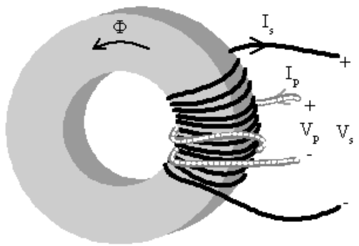



Figure 5.20: A real transformer wound on a toroidal core.

currents is the difference between primary and secondary. The flux, $\Phi$, generated in the core by two currents $I_p$ and $I_s$ is

$$\Phi = n_p A_L I_p - n_s A_L I_s \tag{5.71}$$

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. We can write $I_p$ as

$$I_p = \frac{n_s}{n_p} I_s + \frac{\Phi}{n_p A_L} \tag{5.72}$$

The first term is the relation between the primary and secondary currents in an ideal transformer. It 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 to be infinite in the ideal transformer).

The same flux induces voltages across the primary and secondary windings. The induced primary and secondary voltages, on the other hand, are

$$V_p = j\omega n_p \Phi \text{ and } V_s = j\omega n_s \Phi \tag{5.73}$$

Using these relations, we can relate two voltages as

$$\frac{V_p}{V_s} = \frac{n_p}{n_s} \tag{5.74}$$

Substituting $V_p = j\omega n_p \Phi$ into Eq. 5.72 we obtain

$$I_p = \frac{n_s}{n_p} I_s + \frac{V_p}{j\omega n_p^2 A_L} = \frac{n_s}{n_p} I_s + \frac{V_p}{j\omega L_p} \tag{5.75}$$

where we defined $L_p = n_p^2 A_L$ as the inductance of the primary winding. The two equations relating the primary and secondary terminal voltages and currents can be shown in the form of an equivalent circuit comprising an ideal transformer and a parallel magnetizing inductance, $L_p$, as in Fig. 5.21(a). When the secondary is open-circuited ($I_s = 0$), the equivalent circuit should be the same as the inductor of the primary, $L_p$. Since the primary of the ideal transformer does not carry any current, and the ideal transformer can be removed altogether leaving behind just the inductance of the primary.

Equivalently, $L_p$ can be transferred to the secondary side upon multiplication by $(n_s/n_p)^2$ giving the value $(n_s/n_p)^2 L_p = n_s^2 A_L = L_s$ which is the magnetizing inductance of the secondary (see Fig. 5.21(b)). We should place either the primary or the secondary inductance in the equivalent circuit, but not both.

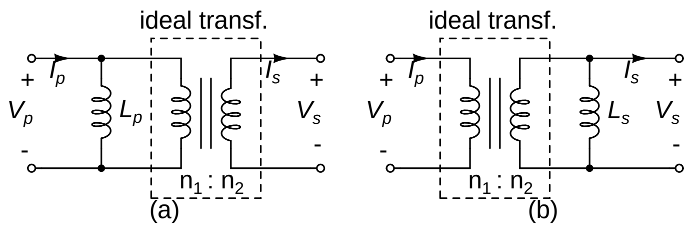



Figure 5.21: Equivalent circuits of a lossless real transformer.

**Coupling coefficient of a transformer**

When the flux generated by the primary winding does not totally go through the secondary winding, the transformer effect is somewhat reduced. The flux lines that does not go through the secondary winding is called *leakage flux*. This effect is defined by the *coupling coefficient* usually shown by $k$. The coupling coefficient is the ratio of flux that is common between two coils to the flux that is due to one coil. When $k = 1$, the coupling between the coils is perfect (an almost impossible case), the equivalent circuit shown in Fig. 5.21 is valid. On the other hand, when $k = 0$, the coils are totally uncoupled, we do not have a transformer, instead we have two separate inductors.

When $0 < k < 1$, the equivalent circuit of the transformer has to be modified as shown in Fig. 5.22. In this figure, $L_1 = (1-k)L_p$ is the *leakage inductance* of

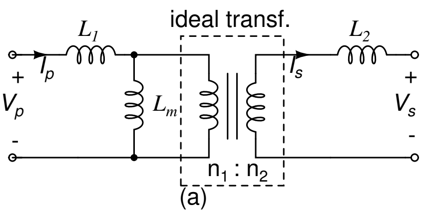



Figure 5.22: Equivalent circuit of a lossless real transformer with $0 < k < 1$.

the primary and $L_2 = (1-k)(n_2/n_1)^2 L_p = (1-k)L_s$ is the leakage inductance of the secondary, while $L_m = kL_p$ is the *magnetizing inductance* of the primary. Equivalently, the magnetizing inductance can be placed in secondary when its value is modified to $(n_2/n_1)^2 L_m$. When the secondary is open-circuited, the primary circuit is equivalent to an inductance of $L_1 + L_m = L_p$ which is just the inductance of the primary winding. Conversely, if the primary is open-circuited, the secondary circuit is equivalent to an inductance of $L_2 + (n_2/n_1)^2 L_m = L_s$ (the inductance of the secondary winding).

**Example 7**

We have a core with $A_L$=10 nH/T$^2$. A transformer is wound with $n_1$=10 and $n_2$=4. Find the equivalent circuit of the transformer if the coupling coefficient between two coils is $k = 0.6$.

**Solution**

We have $L_p = 10^2 \times 10 = 1000$ nH, and $L_s = 4^2 \times 10 = 160$ nH. Hence $L_1 = (1-0.6) \times 1000 = 400$ nH, $L_2 = (1-0.6) \times 160 = 64$ nH, and $L_m = 0.6 \times 1000 = 600$ nH.

---

Fig. 5.23 depicts photos of different size transformers suitable for different frequency bands.

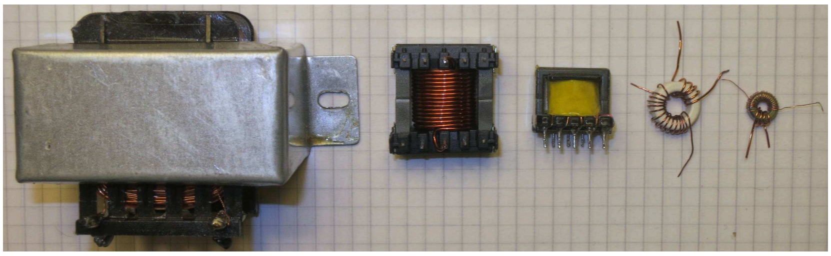


Figure 5.23: Different size transformers.

## 5.6 Tuned amplifiers

When the load resistance of a BJT amplifier is replaced by a parallel RLC circuit, we get a tuned amplifier as shown in Fig. 5.24. In this amplifier, the biasing is done by a conventional bias circuit of Fig. 4.26 at page 159. The emitter resistance has a sufficiently large *bypass* capacitor, $C_E$, not to lower the small-signal gain. The input signal is fed to the base of the BJT through a DC-block capacitance of $C_{c1}$ with a very small reactance at the operating frequency. Similarly, the output voltage is fed to the output from the collector of the BJT through another DC-block capacitance, $C_{c2}$.

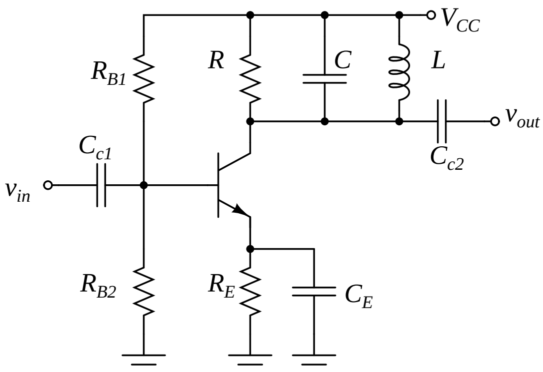



Figure 5.24: Schematic of a tuned BJT amplifier.

The small-signal equivalent circuit of this amplifier is given in Fig. 5.25. In this model, $R_E$ is not present, since the bypass capacitor, $C_E$, can be considered a short-circuit at the operating frequency. The small-signal voltage gain,

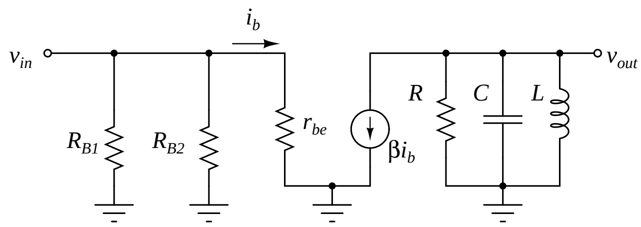



Figure 5.25: Small-signal model of the tuned BJT amplifier.

$A_v$=$v_{out}/v_{in}$, can be written as

$$A_v = -\frac{\beta}{r_{be}} \frac{j\omega L}{(1-\omega^2 LC) + j\omega L/R} \tag{5.76}$$

using the impedance equation of the parallel $RLC$ circuit given in Eq. 5.5. At the resonance frequency, $\omega_o = 1/\sqrt{LC}$, the small-signal gain is given by

$$A_v(\omega_o) = -\frac{\beta}{r_{be}} R \tag{5.77}$$

The small-signal gain of a tuned amplifier with a center frequency of 27 MHz is plotted in Fig. 5.26 for $I_B$=0.02 mA ($r_{be}$=1280 Ω), $R$=300 Ω, $\beta$=65, $C$=100 pF, $L$=0.347 $\mu$H.

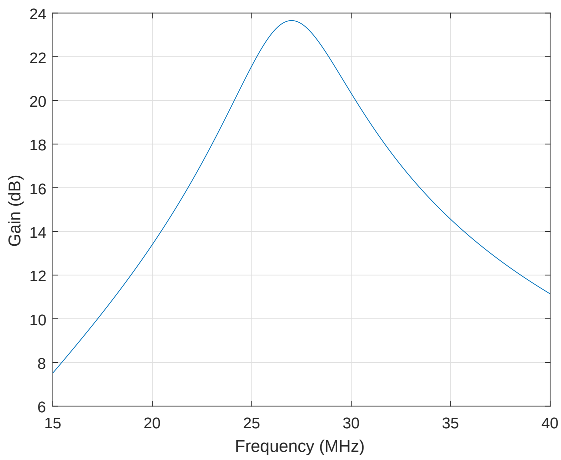



Figure 5.26: Small-signal gain of a tuned BJT amplifier.

## 5.7 High frequency 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 datasheet of LM358, we observe that the gain is 110 dB and constant up to about only 20Hz. Above this frequency, the gain falls at 20 dB/decade as the frequency increases.

### 5.7.1 Gain-bandwidth product

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

$$A_{ol}(\omega) = \frac{A_{dc}}{1 + j\omega/\omega_o} \text{ or } A_{ol}(f) = \frac{A_{dc}}{1 + jf/f_o} \tag{5.78}$$

where $A_{ol}$ is the open-loop gain, $A_{dc}$ is the open-loop gain at DC, and $f_o$ is the 3-dB frequency where gain starts falling. For frequencies higher than $f_o$, the magnitude of the open-loop gain can be approximated as

$$|A_{ol}(f)| \approx A_{dc}\frac{f_o}{f} \text{ for } f > f_o \tag{5.79}$$

The gain drops by 10 times (20 dB) as the frequency increases by 10 times (one decade) for frequencies higher than $f_o$ and eventually reaches unity at a frequency $f_T = A_{dc}f_o$. The frequency $f_T$ is called the *unity-gain bandwidth*.

We use a resistive negative feedback circuit to adjust the gain (called *closed-loop gain*). The unity-gain bandwidth tells us how much bandwidth can be obtained at a given closed-loop gain value since the *gain-bandwidth product* ($GBW$) is a constant, and equal to the unity-gain bandwidth.

$$GBW = G \cdot BW = A_{dc}f_o = f_T \tag{5.80}$$

where $G$ is the closed-loop gain and $BW$ is the 3-dB bandwidth of the amplifier.

We should inquire about the gain-bandwidth product (or unity-gain bandwidth) of an OPAMP to find out suitability at high-frequency amplification. For example, if an OPAMP has a gain-bandwidth product of 4 MHz, it is not suitable for amplifying signals at high frequencies like 27 MHz.

We can calculate the gain more accurately at any frequency using Eq. 5.78. For a non-inverting OPAMP amplifier, as in Fig. 3.26(b) (on page 105), we can write the gain as

$$\frac{V_o}{V_{in}} = \frac{1}{R_1/(R_1 + R_2) + jf/f_T} \tag{5.81}$$

## 5.8 Maximum power transfer

Consider a voltage source represented by phasor $V_{in}$ as shown in Fig. 5.27(a). $R_S$ represents the source resistance. If we connect a load resistor of $R_L$, the current flowing in the circuit is $I_L = V_{in}/(R_S + R_L)$, and the power, $P_L$, delivered to the load $R_L$ is found from the phasor power relation of Eq. 3.35 (page 86) as

$$P_L = \frac{|I_L|^2 R_L}{2} = \frac{1}{2}\frac{|V_{in}|^2 R_L}{(R_S + R_L)^2} \tag{5.82}$$

With a given source of value $V_{in}$ and source impedance $R_S$, the power $P_L$ can be maximized by choosing a suitable $R_L$ value. We find this optimum value by

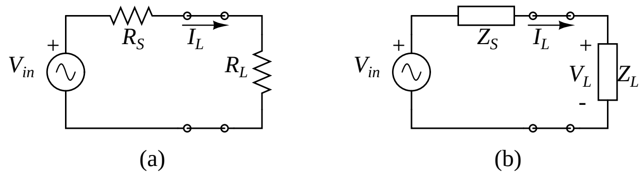



Figure 5.27: Circuits to analyze maximum power transfer.

taking the derivative of $P_L$ with respect to $R_L$ and equating to zero:

$$\frac{dP_L}{dR_L} = \frac{1}{2}|V_{in}|^2 \frac{d}{dR_L} \frac{R_L}{(R_S + R_L)^2} = \frac{1}{2}|V_{in}|^2 \frac{R_S^2 - R_L^2}{(R_S + R_L)^4} = 0 \tag{5.83}$$

The maximum power is achieved when

$$\boxed{R_L = R_S} \tag{5.84}$$

and the maximum power delivered to load resistance is

$$\boxed{P_{Lmax} = \frac{|V_{in}|^2}{8R_S} = \frac{|V_{in}|^2}{8R_L}} \tag{5.85}$$

Now, let us consider the circuit in Fig. 5.27(b), where a source impedance, $Z_S = R_S + jX_S$, (rather than source resistance) is present. In this case, we search for an optimum load impedance, $Z_L = R_L + jX_L$. The current through and the voltage across the load impedance is given by

$$I_L = \frac{V_{in}}{Z_S + Z_L} \quad \text{and} \quad V_L = V_{in} \frac{Z_L}{Z_S + Z_L} \tag{5.86}$$

The power delivered to load can be found from Eq. 3.36 (page 86):

$$P_L = \mathcal{Re}\left\{\frac{V_L I_L^*}{2}\right\} = \mathcal{Re}\left\{\frac{|V_{in}|^2 Z_L}{2|Z_S + Z_L|^2}\right\} = \frac{|V_{in}|^2}{2} \frac{R_L}{|Z_S + Z_L|^2} \tag{5.87}$$

or

$$P_L = \frac{|V_{in}|^2}{2} \frac{R_L}{(R_S + R_L)^2 + (X_S + X_L)^2} \tag{5.88}$$

$P_L$ will be maximized if

$$\boxed{Z_L = Z_S^*} \text{ implying } \boxed{X_L = -X_S} \text{ and } \boxed{R_L = R_S} \tag{5.89}$$

and the maximum power delivered to load impedance is

$$\boxed{P_{Lmax} = \frac{|V_{in}|^2}{8R_S} = \frac{|V_{in}|^2}{8R_L}} \tag{5.90}$$

Since $Z_L = Z_S^*$, this condition is called *conjugate matching*. The condition means that the load reactance is chosen to resonate with the source reactance at the operating frequency to maximize the current and that the load resistance is equal to the source resistance to maximize the power transfer.

Under the conjugate matching condition, the *available power* from the source, $P_A$, is equal to the *power delivered*, $P_L$, to the load. Therefore, the available power is given by

$$P_A = \frac{|V_{in}|^2}{8R_S} \tag{5.91}$$

**Example 8**

Suppose we have a 1000 W audio amplifier to be used in a heavy-metal concert. The output impedance of the audio amplifier, $R_S$ is 8 $\Omega$. But the speaker has a load impedance of $R_L$=4 $\Omega$. Find out how much power we lose by not using a speaker of $R_L$=8 $\Omega$ impedance.

Power ratings of audio amplifiers are specified, assuming $R_L$=$R_S$. From Eq. 5.85, we find $V_{in} = \sqrt{8P_{Lmax}R_S} = \sqrt{8 \cdot 1000 \cdot 8}$=253 V (peak voltage of a sinusoid). Since $R_L$=4 $\Omega$, from Eq. 5.82, we write

$$P_L = \frac{1}{2}\frac{|V_{in}|^2 R_L}{(R_S + R_L)^2} = \frac{1}{2}\frac{253^2 4}{(8+4)^2} = 889 \text{ W}$$

We can deliver 111 W less than we can deliver to a matched load.

## 5.9 Examples

### Example 9

Consider the circuit given in Fig. 5.28. What should $R_L$ value be for the maximum power dissipation in $R_L$? What is the amount of power dissipation?

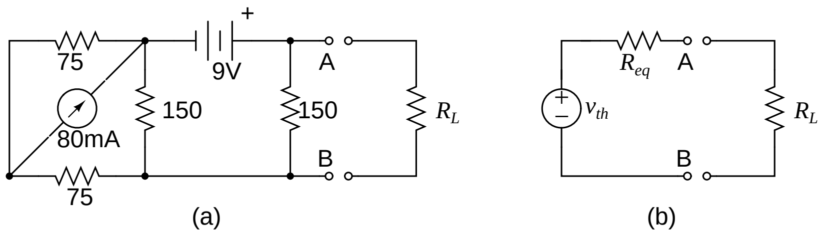



Figure 5.28: (a) Circuit for Example 9, (b) Thévenin equivalent circuit.

**Solution**

First, let us find the Thévenin equivalent circuit between pins A and B. The equivalent resistance $R_{eq}$ can be found by killing the sources (open-circuit the current source and short-circuit the voltage source):

$$R_{eq} = 150 \parallel 150 \parallel (75 + 75) = 50\ \Omega$$

Using Eq. 5.84, we must have $R_L = R_{eq} = 50\ \Omega$ for the maximum power transfer. To find the power transferred, we need to find the Thévenin voltage, $V_{th}$. Since there are two sources, we can use superposition to find $v_{th}$, while $R_L$ is disconnected.

After killing the current source, the open-circuit voltage, $v_{AB}$, is found by the voltage divider:

$$v_{AB} = \frac{150}{150 + (150 \parallel 150)} 9 = \frac{150}{150 + 75} 9 = 6.0\ \text{V}$$

Killing the voltage source, $v_{AB}$ is found using the current divider formula of Eq. 2.35 (see page 35):

$$v_{AB} = \frac{75}{75 + (150 \parallel 150 + 75)} 0.080 \cdot 75 = \frac{75}{75 + 150} 0.080 \cdot 75 = \frac{0.080}{3} 75 = 2.0\ \text{V}$$

and hence $v_{th} = 6.0 + 2.0 = 8.0$ V. The power, $P_L$, dissipated on the resistance $R_L = 50\ \Omega$ is

$$P_L = \left(\frac{v_{th}}{2}\right)^2 \frac{1}{R_L} = \left(\frac{8}{2}\right)^2 \frac{1}{50} = \frac{16}{50} = 0.32\ \text{W}$$

Note that there is an additional factor of 1/2 in Eq. 5.85, since the voltage source is expressed as a phasor.

## Example 10

Given a sinusoidal source with $\omega = 1.25 \cdot 10^8$ shown in Fig. 5.29(a) in dashed lines. Find the values of $n$ and $C_p$ so that the maximum power is transferred to the load resistor, $R_L$. The available core has $A_L$=10 nH/T$^2$. Find the maximum power transferred to the load resistor $R_L$.

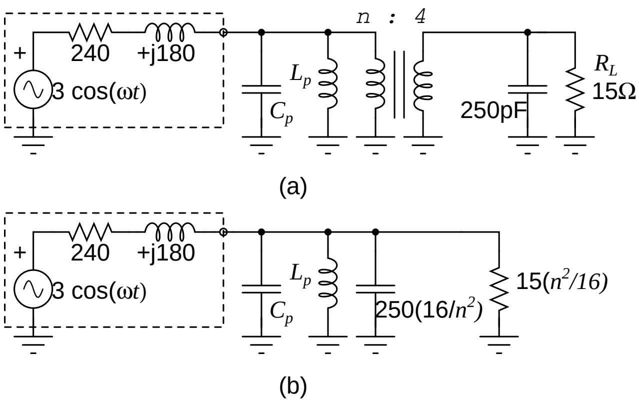



Figure 5.29: (a) Circuit for Example 10, (b) the components on the secondary side transferred to the primary.

### Solution

We can transfer the 250 pF capacitor and the 15 Ω resistance to the primary side as shown in Fig. 5.29(b). The resistance is multiplied by $(n/4)^2$ while the capacitance is divided by $(n/4)^2$. Hence the total shunt resistance of the load is $R_T = 15(n^2/16)$, while the total shunt reactance is the parallel combination of $C_p + 250 \cdot 16/n^2$ (in pF) with the inductance $L_p$. The load seen by the source should be 240−j180 Ω for a maximum power transfer. Hence the load should be a capacitance, $C_T$, in parallel with the resistor, $R_T$. We write the parallel combination as

$$R_T \parallel \frac{1}{j\omega C_T} = \frac{R_T \cdot 1/(j\omega C_T)}{R_T + 1/(j\omega C_T)} = \frac{R_T}{1 + j\omega R_T C_T} = \frac{R_T - j\omega R_T^2 C_T}{1 + \omega^2 R_T^2 C_T^2}$$

For a conjugate match we must have

$$\frac{R_T}{1 + \omega^2 R_T^2 C_T^2} = 240 \quad \text{and} \quad \frac{-\omega R_T^2 C_T}{1 + \omega^2 R_T^2 C_T^2} = -180$$

Solving these two equations simultaneously, we find $R_T = 375$ Ω and $C_T =$ 16 pF. Therefore,

$$R_T = 15\left(\frac{n^2}{16}\right) = 375 \quad \text{or} \quad n = 20$$

Hence we have $L_p = 10$ nH/T$^2 \cdot 20^2 = 4$ $\mu$H. This inductance eliminates a shunt capacitance of value

$$C_r = \frac{1}{\omega^2 L_p} = 16 \text{ pF}$$

Hence

$$C_T = C_p + 250\left(\frac{16}{n^2}\right) - C_r = C_p + 10 - 16 = C_p - 6 \quad \text{pF}$$

Since $C_T = 16$ pF, we find $C_p = 22$pF. The power dissipated on $R_L$ is given by Eq. 5.85

$$P_L = \frac{|V_{in}|^2}{8R_S} = \frac{|3|^2}{8 \cdot 240} = 4.7 \text{ mW}$$

---

## Example 11

Spark plugs used in the ignition system of gasoline motors require 12,000 to 50,000 V to fire. Since most autos have a DC voltage of 12 V, a circuit is necessary to generate the high voltages. Refer to the circuit given in Fig. 5.30(a), where the switch S is normally closed, and it is opened at the time when the ignition is desired. With $L_p$=10 mH and $R = 6\ \Omega$, explain how the high voltages are obtained.

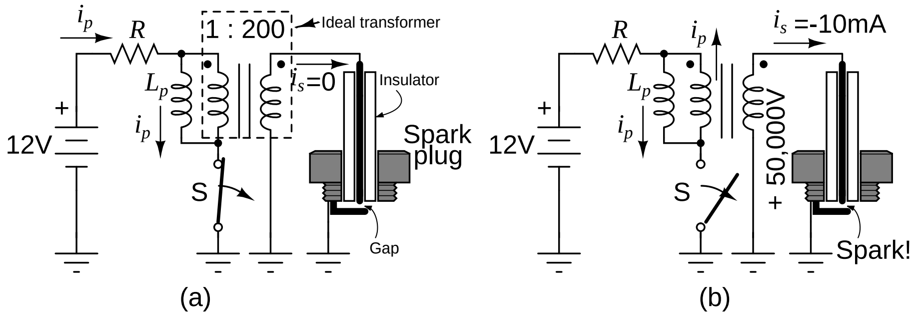



Figure 5.30: (a) Ignition circuit with S closed, (b) ignition circuit right after S is opened.

### Solution

While S is closed, the time constant in the primary side is determined by $\tau = L_p/R = 1.7$ ms. While S is closed for a sufficiently long time (at least $5\tau = 8.3$ ms), a current of

$$i_p = \frac{12}{6\Omega} = 2 \text{ A}$$

flows in $L_p$. Since the spark plug is nonconducting, $i_s = 0$, and hence there is no current in the primary of the ideal transformer. When S is opened (refer to Fig. 5.30(b)), the continuity of inductance current dictates that the same current must flow in the primary of the ideal transformer. Hence a current of $i_s = -i_p(1/200) = -10$ mA flows in the secondary, causing a spark in the spark plug. Since this requires a voltage like 50,000 V, the primary voltage is momentarily $-50,000/200 = -250$ V. When the spark plug is ignited, the voltage at the secondary drops and the current in the secondary decreases exponentially with time.

## Example 12

A tuned BJT amplifier is built using the circuit in Fig. 5.31 with $\beta = 90$, $V_0 = 0.7$ V, $V_{sat} = 0.2$ V, $R_1 = 68$ K, $R_2 = 6.8$ K, $R_3 = 390\ \Omega$, $V_{CC} = 12$ V, $A_L = 8$ nH/T$^2$ (for the transformer core), $n_1 = 20$, $n_2 = 10$, $R = 470\ \Omega$. $C_1$ and $C_2$ are sufficiently large capacitors so that they can be considered short-circuit at the operating frequency. Find the DC base current and the state of the transistor. Find the value of $C$ such that the resonance occurs at $f_0 = 15$ MHz. Determine the small-signal output voltage in terms of $v_{in}$ at 15 MHz.

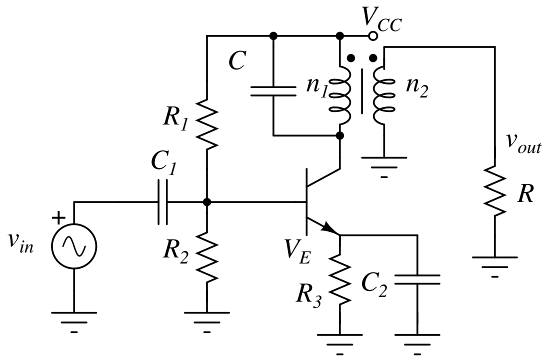



Figure 5.31: Tuned BJT amplifier for Example 12.

### Solution

Thèvenin equivalent circuit of $R_1$, $R_2$ and $V_{CC}$ is found as $R_T = R_1 \parallel R_2 = 68 \parallel 6.8 = 6.2$ K and $V_T = V_{CC}R_2/(R_1 + R_2) = 1.09$ V. Asuming the BJT is ACT, we find the base current as

$$I_B = \frac{V_T - V_0}{R_T + (\beta + 1)R_3} = 0.0094 \text{ mA}$$

Hence $I_E = (\beta + 1)I_B = 0.85$ mA and $V_E = 0.33$ V. Since the inductor the transformer primary is short-circuit at DC, we have $V_{CE} = V_{CC} - V_E = 12 - 0.33 = 11.7$ V and the BJT is ACT. The small-signal equivalent circuit is shown in Fig. 5.32 with

$$r_{be} = \frac{25.9 \text{ mV}}{0.0094 \text{ mA}} = 2.75 \text{ K}$$

and $R_3$ shorted by $C_2$. The primary inductance of the transformer is $L_p = A_L n_1^2 = 3.2\ \mu$H. This can be tuned out with a capacitor of

$$C = \frac{25330}{15^2 \times 3.2} = 35 \text{ pF}$$

The load resistor $R$ is transformed to a value $(n_1/n_2)^2 R$ by the transformer. The small-signal base current is $i_b = v_{in}/r_{be}$. At the resonance $L_p$ is tuned out

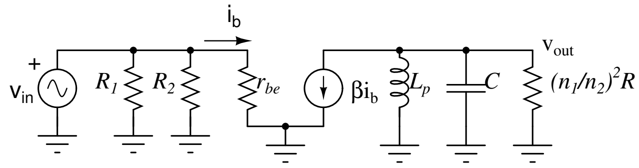



Figure 5.32: The small-signal model of tuned BJT amplifier for Example 12.

with $C$. Hence the small-signal output voltage is

$$v_{out} = -\beta \frac{v_{in}}{r_{be}} \left(\frac{n_1}{n_2}\right)^2 R = -61.5 v_{in}$$

The tuned circuit has a quality factor of $Q = 2\pi f_0 C (n_1/n_2)^2 R = 6.2$. Hence the 3-dB frequencies are at $f_1 = f_0 - f_0/(2Q) = 13.8$MHz and $f_2 = f_0 + f_0/(2Q) = 16.2$ MHz. The BJT will be ACT as long as the peak voltage of $v_{out}$ is less than $V_{CC} - V_E = 11.7$ V at $f_0$.

## 5.10 Problems

1. In a series $RLC$ circuit, $R$=100 Ω, $L$=10 $\mu$H, and $C$=3 pF. Plot the magnitude and the phase of the impedance as a function of $\omega$ for $0.6\omega_o < \omega_0 < 1.5\omega_o$.
2. A voltage $10\angle 0$ is applied to the series circuit of Problem 1. Find the voltage across each element for $f$ = 28MHz.
3. A series circuit with $R$=50 Ω, $C$=39 pF, and variable inductor $L$ has an applied voltage V=$10\angle 0$ with a frequency of 16MHz. $L$ is adjusted until the voltage across the resistor is maximum. Find the voltage across each element.
4. A series circuit has $R$= 50 Ω, $L$=1 $\mu$H, and a variable capacitor $C$. Find the value of $C$ for a series resonance at $f$=16MHz.
5. Given a series $RLC$ circuit with $R$=10 Ω, $L$=0.5 $\mu$H, and $C$=220 pF, calculate the resonant, lower and upper half-power frequencies.
6. Show that the resonant frequency $\omega_o$ of an $RLC$ series circuit is the geometric mean of $\omega_1$ and $\omega_2$, the lower and upper half-power frequencies.

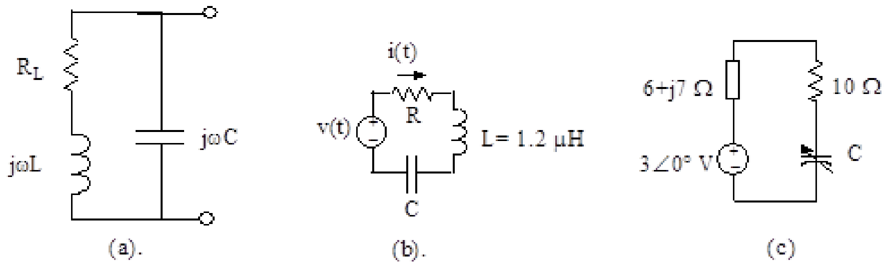



Figure 5.33: Circuits for problems 7 and 10

7. The circuit in Fig. 5.33(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?
8. Show that for a series $RLC$ circuit the quality factor is given by $Q = \omega_o L/R = f_o/BW$.
9. Find $Q$ of the series circuit with $R$=20 Ω, $C$=47 pF and $L$=2 $\mu$H.
10. In the series circuit of Fig. 5.33(b), the instantaneous voltage and current are $v(t) = 1.5\sin(2\pi 10^7 t + 30^o)$ V and $i(t) = 10\sin(2\pi 10^7 t + 30^o)$ mA. Find $R$ and $C$.
11. In the series circuit of Fig. 5.33(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 Ω resistor be a maximum? What is this maximum power deliverable to 10 Ω resistor?

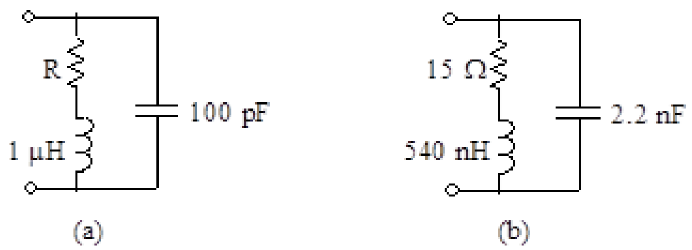



Figure 5.34: Circuits for problems 12 and 13

12. In the parallel circuit of Fig. 5.34(a), determine the resonant frequency if $R$=0 and $R$=1 Ω. Compare them to resonant frequency when $R$=100 Ω.

13. In the parallel circuit in Fig. 5.34(b), find the resonant frequency $f_o$.

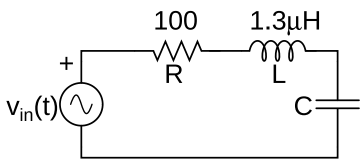



Figure 5.35: Circuit for problems 14 and 15

14. Calculate the voltages across $R$, $L$, and $C$, of the series $RLC$ circuit of Fig. 5.35, if $v_{in}(t) = 5\cos(\omega_o t)$. Choose $C$ such that the circuit resonates at a frequency of $f_o$=28 MHz. Write down the time waveform expressions for these voltages.

15. Calculate the same voltages in problem 14, at the series circut's upper and lower 3 dB frequencies. Write down the time waveform expressions for these voltages.

16. Calculate the DC resistance of a 0.2 mm diameter wire of length 10 cm. Calculate its skin effect cut-off frequency. Calculate the approximate copper loss resistance of an inductor made of this wire at 28 MHz.

17. What must be the length of a 900 nH air core inductor if it has 21 turns and its diameter is 5 mm?

18. Find the number of turns of an air-core inductor with $L$=270 nH and $d = l$ =5 mm.

19. $A_L$ of the T25-10 toroidal core is given as 1.9 nH/T$^2$. Find the number of turns required to make a 615 nH inductor using T25-10.

20. 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 loss resistance of this inductor at 30 MHz.

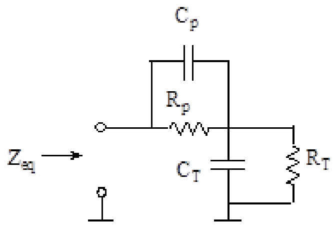



Figure 5.36: An oscilloscope probe with a compensation circuit

21. Consider the all-pass probe compensation circuit given in Fig. 5.36. Show that the equivalent input 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 $\times$10 probe).

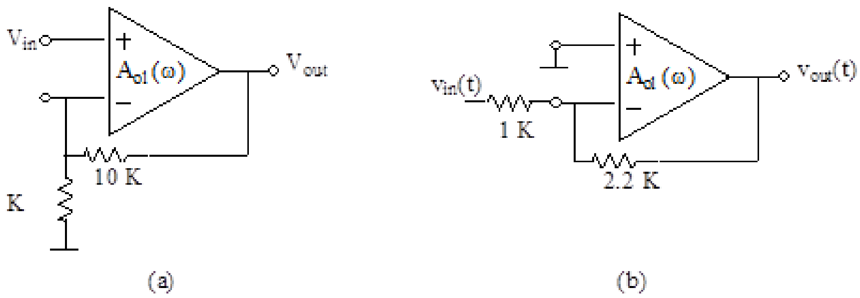



Figure 5.37: OPAMP circuits for (a) problem 23 and (b) problem 24.

22. Consider a tuned amplifier as in Fig. 5.31 with $V_{CC}$ = 9 V, $V_E$ = 1 V, $I_E$ = 2 mA, $\beta = 60$, $V_0$ = 0.7 V, $V_{sat}$ = 0.2 V and $R$ = 220 Ω. Find the values of $R_1$, $R_2$, and $R_3$ for the correct DC bias currents. Set the small-signal gain of the amplifier to $v_{out}/v_{in} = -20$ by choosing the transformer turns-ratio.

23. The RF amplifier in Fig. 5.37(a) is designed using an OPAMP. The open-loop voltage gain of the OPAMP is given as

$$A_{ol}(\omega) = \frac{5 \cdot 10^5}{1 + j\omega/100}$$

from its datasheet. What is the voltage gain of the non-inverting amplifier at 400 kHz? What is its gain at 1 kHz?

24. The same OPAMP is used in an inverting amplifier configuration, as shown in Fig. 5.37(b). What is the output voltage $v_{out}(t)$ when the input is $v_{in}(t) = 2\cos(2\pi 10^6 t + 30^o)$ volts? What is $v_{out}(t)$ when the input is $v_{in}(t) = 2\cos(2\pi 10^3 t + 30^o)$?

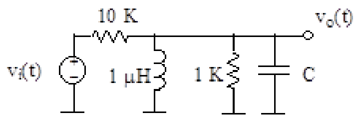



Figure 5.38: Circuit for problem 25.

25. $v_i(t)$ in Fig. 5.38 is given as
$v_i(t) = A_1 \cos(2\pi 16 \cdot 10^6 t) + A_2 \cos(2\pi 8 \cdot 10^6 t)$,
and assume that the output
$v_o(t) = B_1 \cos(2\pi 16 \cdot 10^6 t + \theta_1) + B_2 \cos(2\pi 8 \cdot 10^6 t + \theta_2)$
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 $C$?

# Chapter 6

# FILTERS

Filters are usually used to remove undesired components of a signal. For example, an antenna delivers a complex signal containing many components at a large band of frequencies. It is preferable to remove the unnecessary signal components at frequencies other than the band of interest before the signal is amplified.

Parallel and series tuned circuits can be used to filter such signals. The filtering performance of such circuits is only determined by the quality factor of the circuit. It is often necessary to have filters with improved performance compared to what simple tuned circuits can offer. In what follows, we describe more complex circuits with higher filtering performance.

## 6.1 Motivation

Any electronic filter can be visualized as a block between a source (input) and a load (output). This is depicted in Fig. 6.1. 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. For this purpose, we may need

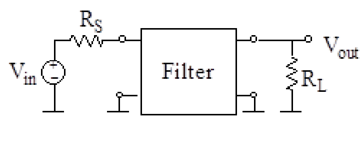



Figure 6.1: An electrical filter

one of the following filter types:

- Low-pass-filter (LPF): Eliminates frequencies higher than a limit.
- High-pass-filter (HPF): Eliminates frequencies lower than a limit.
- Band-pass-filter (BPF): Eliminates frequencies outside two limits.
- Band-stop-filter (BSF): Eliminates frequencies inside two limits.

### 6.1.1 Transducer power gain of a filter

We define the *transducer power gain*, $G_T$, of a filter as

$$\boxed{G_T = \frac{P_L}{P_A}} \tag{6.1}$$

where $P_L$ is the *power delivered* to load, and $P_A$ is the *available power* (see Eq. 5.91 on page 209) from the source. In the absence of a filter and with $R_L = R_S$, we have $P_L = P_A$ or $G_T = 1$. The transducer power gain, $G_T$, is given in terms of voltages and resistances as

$$\boxed{G_T(\omega) = \frac{P_L}{P_A} = \frac{|V_{out}|^2/2R_L}{|V_{in}|^2/8R_S} = \frac{4R_S}{R_L}\left|\frac{V_{out}}{V_{in}}\right|^2} \tag{6.2}$$

where $V_{in}$ and $V_{out}$ are phasors.

$G_T$ is frequently expressed in decibels as

$$G_{T(dB)} = 10\ \log_{10}\frac{P_L}{P_A} = 10\ \log_{10}\frac{4R_S}{R_L} + 20\ \log_{10}\left|\frac{V_{out}}{V_{in}}\right| \tag{6.3}$$

### 6.1.2 First- and second-order low-pass-filters

Consider the first-order LPF given in Fig. 6.2(a). Since

$$\frac{V_{out}}{V_{in}} = \frac{\frac{R_L}{1+j\omega R_L C}}{R_S + \frac{R_L}{1+j\omega R_L C}} = \frac{R_L}{R_L + R_S + j\omega R_L R_S C} = \frac{R_L}{R_L + R_S}\frac{1}{1 + j\omega\frac{R_L R_S}{R_L+R_S}C} \tag{6.4}$$

With the definition

$$\omega_c = \frac{1}{\frac{R_L R_S}{R_L+R_S}C} = \frac{1}{(R_S \parallel R_L)C} \tag{6.5}$$

and combining Eqs. 6.2 and 6.4, we find

$$G_T(\omega) = \frac{4R_L R_S}{(R_L + R_S)^2}\frac{1}{1 + (\omega/\omega_c)^2} \tag{6.6}$$

For the special case of $R_L = R_S$, the transducer power gain simplifies to

$$G_T(\omega) = \frac{1}{1 + (\omega/\omega_c)^2} \tag{6.7}$$

For $\omega \ll \omega_c$ or at $\omega$=0, we have $G_T(0) = 1$, and for $\omega \gg \omega_c$, we have $G_T(\omega) \to (\omega_c/\omega)^2$ showing that it is an LPF.

Since

$$G_T(\omega_c) = \frac{1}{2} \text{ or } G_{T(dB)}(\omega_c) = -3 \text{ dB} \tag{6.8}$$

$\omega_c$ is the 3-dB frequency of the filter.

We can get a similar filtering effect if we use a series inductot instead of a parallel capacitor, as shown in Fig. 6.2(b). In this case, the transducer power gain is the same as Eq. 6.6 with

$$\omega_c = \frac{R_L + R_S}{L} \tag{6.9}$$

We expect a better filtering function if we use both a series inductance and a parallel capacitance in the filter block. The filter in Fig. 6.2(c) has a voltage transfer function

$$\frac{V_{out}}{V_{in}} = \frac{1}{(1 + R_S/R_L) - \omega^2 LC + j\omega(L/R_L + R_S C)} \tag{6.10}$$

This expression looks like a tuned circuit transfer function, and it is not easy to

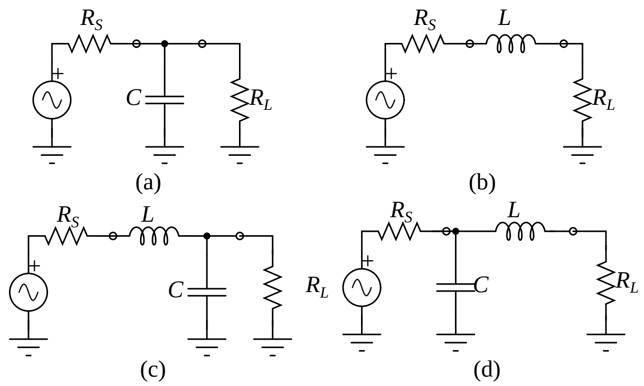



Figure 6.2: First order low-pass-filters using (a) a capacitor and (b) an inductor. Second-order low-pass-filters using (c) an inductor and a capacitor, (d) a capacitor and an inductor.

immediately recognize it as an LPF. However, for the special case of $R_S = R_L$, and with choice of $\omega_c R_S C = \sqrt{2}$ and $\omega_c L/R_S = \sqrt{2}$, we get a *maximally flat* behavior in the pass-band, and the voltage transfer function becomes

$$\frac{V_{out}}{V_{in}} = \frac{1}{2}\frac{1}{1 - (\omega/\omega_c)^2 + j\sqrt{2}\omega/\omega_c} \tag{6.11}$$

where

$$\omega_c = \sqrt{\frac{2}{LC}} \tag{6.12}$$

The transducer power gain of this second-order LPF is given by

$$G_T(\omega) = \frac{4R_S}{R_L}\left|\frac{V_{out}}{V_{in}}\right|^2 = 4\left|\frac{V_{out}}{V_{in}}\right|^2 = \frac{1}{1 + (\omega/\omega_c)^4} \tag{6.13}$$

We note that the transducer power gain of the filter shown in Fig. 6.2(d), where the filter is flipped, is the same as Eq. 6.13.

Transducer power gains of first and second-order filters are plotted in Fig. 6.3 on a semilog plot. The vertical axis is in decibels. Note the asymptotic behavior of the filters above the cutoff frequency. The first-order filter has a slope of −20 dB/dec, while the second-order filter has a −40 dB/dec slope. The second-order filter is superior to the first-order filter 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 attenuated.

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.

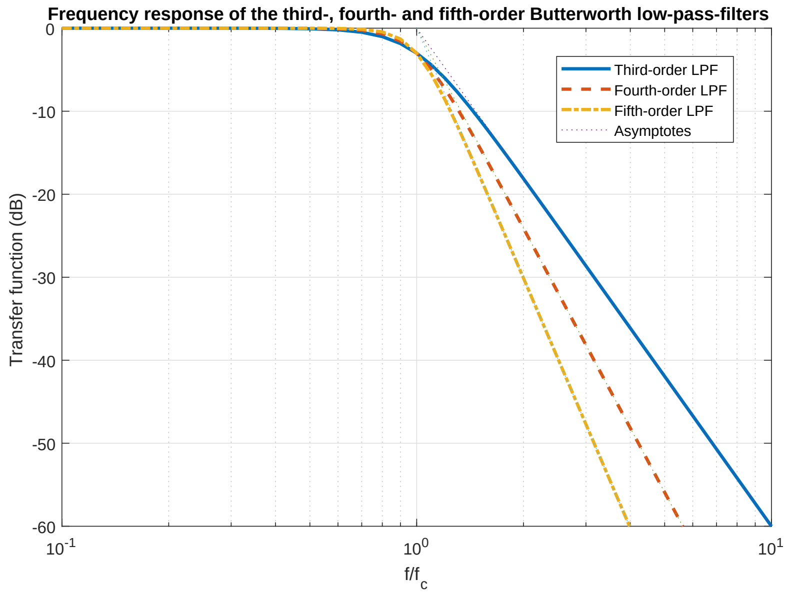



Figure 6.3: The transducer power gain of first and second-order LPFs as a function of normalized frequency.

## 6.2 Polynomial filters

Modern filter theory maps the desired transducer power gain of the filter to the properties of a class of polynomials like Butterworth, Chebyshev polynomials, and elliptic polynomials.

The circuit morphology on which polynomial low-pass-filters are based is *ladder*-type, as shown in Fig. 6.4. The basic building block in polynomial filters is a LPF. HPF, BPF, and BSF are derived from this block. The first element can be either an inductor or a capacitor. The topology containing less number of inductors is generally preferred. The shunt elements in the LPF configurations of Fig. 6.4(a) and (b) are capacitors, and series elements are inductors. At low frequencies, inductors provide a low impedance path from the input signal to the output, while capacitors maintain high impedance to ground, hence the low loss. At high frequencies, the capacitor impedance is low, and therefore there is a loss in the signal at every node. On the other hand, inductors have high impedance, and the division effect at each node on the signal is increased. In

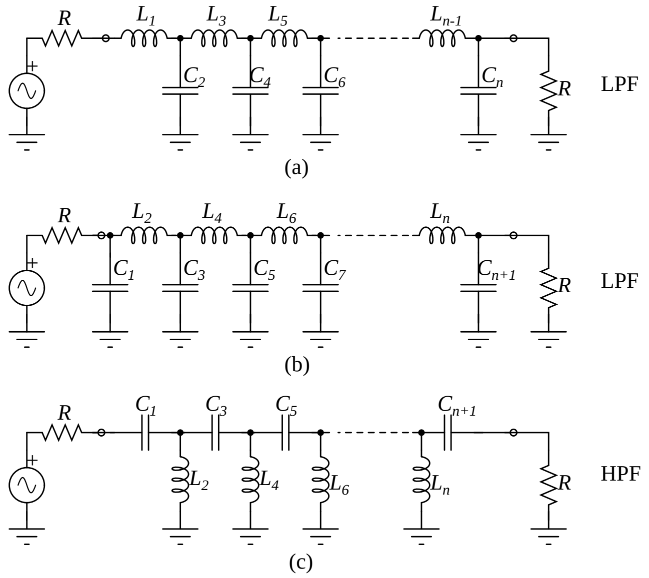



Figure 6.4: Ladder type low-pass-filter prototypes for (a) even, and (b) odd number of elements. (c) High-pass-filter prototype with odd number of elements.

the HPF ladder in Fig. 6.4(c), the series and shunt elements are interchanged compared to LPF, thus yielding exactly the opposite function.

### 6.2.1 Butterworth filters

A Butterworth* low-pass-filter of $n$th order seeks to have a transducer power gain of

$$G_T = \frac{P_L}{P_A} = \frac{1}{1 + (f/f_c)^{2n}} \tag{6.14}$$

where $f_c$ is the 3-dB cutoff frequency. As $f \to \infty$, we find $G_T \to (f_c/f)^{2n}$, defining the asymptotic response.

$G_T = P_L/P_A$ is plotted versus frequency for different number of elements, $n$, in Fig. 6.5. As $n$ increases, the transducer power gain approaches to that of an ideal LPF. Note that for the third-order LPF, the asymptote has a slope of $-60$ dB/decade (one decade is a ten-fold increase), while for the fifth-order filter, the slope is $-30$ dB/octave (one octave is a two-fold increase).

The component values for Butterworth filters, normalized with respect to the termination impedance and cutoff frequency are provided in tables. Normalized coefficients are correct reactance and susceptance values in $\Omega$, for 1 $\Omega$ source and load resistance and a cutoff frequency of $\omega_c = 1$ rad/sec. A table of these coefficients, up to eight elements, is given in Table 6.1. The coefficients for filters with more elements can be obtained from the following relation:

$$b_i = 2\sin\left(\frac{(2i-1)\pi}{2n}\right) \tag{6.15}$$

*Invented by British physicist Stephen Butterworth (1885–1958).

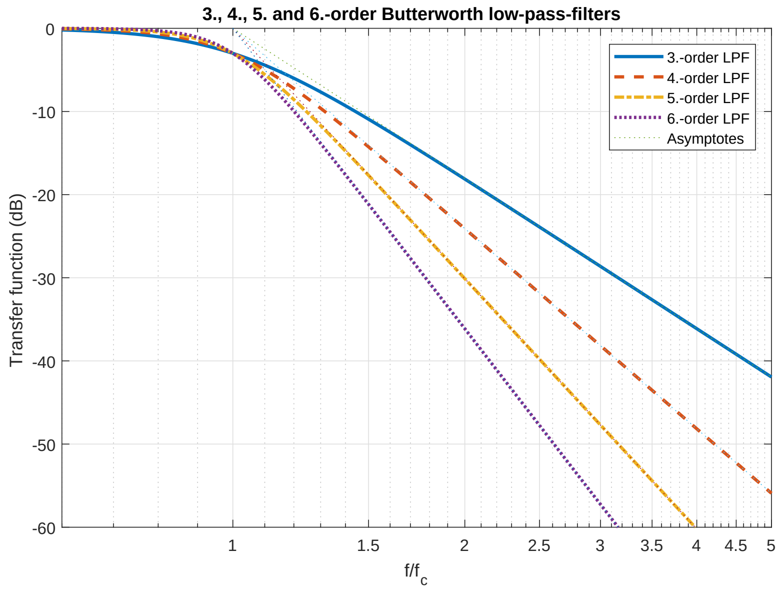



Figure 6.5: Butterworth LPF response for different $n$ values. At $f = f_c$, the filters have 3 dB attenuation.

Using this table for designing a filter is straightforward. Since the coefficients are normalized component values, we must scale them for the given termination resistance and cutoff frequency.

### Low-pass-filter

We use the following procedure to design an $n$th-order Butterworth low-pass-filter of cutoff frequency $f_c$ for load and source impedances of $R$:

1. Use the corresponding Butterworth table value to find the inductor value as

$$L_i = \frac{b_i R}{2\pi f_c} \tag{6.16}$$

| $n$ | $b_1$ | $b_2$ | $b_3$ | $b_4$ | $b_5$ | $b_6$ | $b_7$ | $b_8$ |
|---|---|---|---|---|---|---|---|---|
| 1 | 2.000 | | | | | | | |
| 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 | 2.0000 | 1.8019 | 1.2470 | 0.4450 | |
| 8 | 0.3902 | 1.1111 | 1.6629 | 1.9616 | 1.9616 | 1.6629 | 1.1111 | 0.3902 |

Table 6.1: Table of prototype element values in Butterworth low-pass-filters

2. Use the corresponding Butterworth table value to find the capacitor value as
$$C_i = \frac{b_i}{2\pi R f_c} \tag{6.17}$$

Above the cutoff frequency, the signal is reduced by $20n$ dB/decade or $6n$ dB/oct.

The transducer power gain of an LPF can be drawn easily using the asymptotic lines shown in Fig. 6.3 or 6.5: On a semilog axis, where the frequency axis is logarithmic, first draw a line with zero slope at 0 dB line indicating the asymptotic behavior in the passband. Then, draw a line with slope $-20n$ dB/decade (one decade is a frequency ratio of 1:10) passing through the $(f_c, 0\text{ dB})$ point showing the response in the frequency range above the cut-off frequency. The actual response curve approaches asymptotes at low and high frequencies, passing through $(f_c, -3\text{ dB})$ point. The asymptotes are good approximations for $f < 0.5f_c$ and $f > 2f_c$.

**Example 1**

Let us design a third-order Butterworth low-pass-filter for a cutoff frequency of $f_c$=2 MHz for source and load impedances of $R_S = R_L$=300 Ω. We can use either $L_1 - C_2 - L_3$ topology or $C_1 - L_2 - C_3$ topology. The second one (shown in Fig. 6.6(a)) is preferable since it uses only one inductor. Using the $n = 3$ values in Table 6.1 and $R$=300 Ω, we find

$$C_1 = C_3 = \frac{1.00}{2\pi 300 \cdot 2 \cdot 10^6} = 265\text{ pF and } L_2 = \frac{2.00 \cdot 300}{2\pi 2 \cdot 10^6} = 48.0\mu\text{H}$$

We can estimate the performance of the filter using Fig. 6.5: Since $f_c$ =2 MHz,

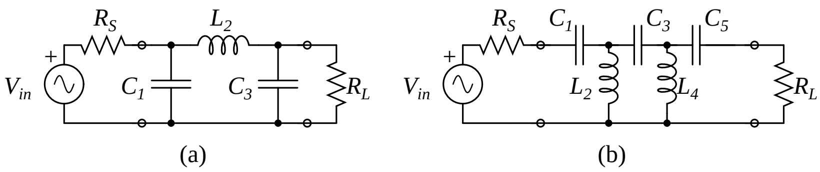



Figure 6.6: (a) Third-order LPF and (b) fifth-order HPF.

at 6 MHz ($f/f_c = 3$) the signal will be attenuated by 28 dB (voltage will be 0.04 times).

---

**Example 2**

We require a low-pass-filter for $R = 50\Omega$, which passes the signals lower than 1 MHz with attenuation less than 1 dB and attenuates signals higher than 3.5 MHz with attenuation of more than 30 dB.

Using the decibel version of the Butterworth filter equation of Eq. 6.14 we write the requirements as

$$10\log_{10}\frac{P_L}{P_A} = 10\log_{10}\frac{1}{1 + (1 \cdot 10^6/f_c)^{2n}} \geq -1\text{dB} \Rightarrow \left(\frac{1 \cdot 10^6}{f_c}\right)^{2n} \leq 0.259$$

$$10\log_{10}\frac{P_L}{P_A} = 10\log_{10}\frac{1}{1+(3.5\cdot 10^6/f_c)^{2n}} \le -30\text{dB} \Rightarrow \left(\frac{3.5\cdot 10^6}{f_c}\right)^{2n} \ge 999$$

Taking the logarithm of both sides and solving two equations under equality case, we write

$$2n\ln\left(\frac{1\cdot 10^6}{f_c}\right) = 27.63n - 2n\ln f_c = \ln 0.259 = -1.35$$

$$2n\ln\left(\frac{3.5\cdot 10^6}{f_c}\right) = 30.13n - 2n\ln f_c = \ln 999 = 6.91$$

Subtracting two equations from each other, we find $n$=3.29. Using one of the equations, we reach $f_c$=1.22 MHz. For filter order, we choose the next greater integer: $n$=4. Substituting $n$=4 in the equations above, we find two different $f_c$ values: 1.18 MHz and 1.47 MHz. Hence, the cutoff frequency can be selected as any $f_c$ in the range 1.18$\le f_c \le$1.47 MHz. Choosing $f_c$=1.3 MHz and the $L_1 - C_2 - L_3 - C_4$ LPF topology, we find

$$L_1 = \frac{0.7654\cdot 50}{2\pi 1.3\cdot 10^6} = 4.68\mu\text{H} \text{ and } C_2 = \frac{1.8478}{2\pi 50\cdot 1.3\cdot 10^6} = 4.52 \text{ nF}$$

$$L_3 = \frac{1.8478\cdot 50}{2\pi 1.3\cdot 10^6} = 11.3\mu\text{H} \text{ and } C_4 = \frac{0.7654}{2\pi 50\cdot 1.3\cdot 10^6} = 1.87 \text{ nF}$$

**Example 3**

Let us consider the OPAMP second-order low-pass-filter of Fig. 3.32(b) in p. 110 and find the condition for making it a Butterworth filter.

We assume that the source resistance, $R_S$, and the load resistance, $R_L$, are $4R_S/R_L$=1. From Eq. 3.65 in p. 112, we find the transducer power gain of Eq. 6.2 as

$$G_T = \left|\frac{V_o}{V_{in}}\right|^2 = \left|\frac{1}{1+j\omega(R_1+R_2)C_1 - \omega^2 R_1R_2C_1C_2}\right|^2 \tag{6.18}$$

or

$$G_T = \frac{1}{1+\omega^2((R_1+R_2)^2C_1^2 - 2R_1R_2C_1C_2) + \omega^4(R_1R_2C_1C_2)^4} = \tag{6.19}$$

This equation is in the form of Butterworth polynomial of Eq. 6.14, if

$$(R_1+R_2)^2C_1 = 2R_1R_2C_2 \tag{6.20}$$

---

**High-pass-filter**

The transfer function of a Butterworth high-pass-filter is written in the following form

$$\frac{P_L}{P_A} = \frac{1}{1+(f_c/f)^{2n}} \tag{6.21}$$

At $f = f_c$, we have $P_L/P_A = 0.5 = -3$ dB defining the cut-off frequency of the high-pass-filter. For $f \to 0$, we have $P_L/P_A \to (f/f_c)^{2n}$ defining the asymptote.

We use the following procedure to design an $n$th-order Butterworth high-pass-filter of cutoff frequency $f_c$ for load and source impedances of $R$:

1. Use the corresponding Butterworth table value to find the inductor value as
$$L_i = \frac{(1/b_i)R}{2\pi f_c} \tag{6.22}$$

2. Use the corresponding Butterworth table value to find the capacitor value as
$$C_i = \frac{1/b_i}{2\pi R f_c} \tag{6.23}$$

Note that we use $1/b_i$ rather than $b_i$ for high-pass-filter component values. Below the cutoff frequency, the filter attenuates the signals with $20n$ dB/decade or $6n$ dB/octave.

As in the case of LPF, the transfer function of an HPF can be drawn easily using the asymptotic lines. This time, the asymptotic line with a slope $+20n$ dB/decade passing through $(f_c, 0 \text{ dB})$ is drawn to indicate the response below the cutoff frequency.

**Example 4**

Let us design a fifth-order Butterworth high-pass-filter for a cut-off frequency of $f_c$=15 MHz for source and load impedances of $R$=50 Ω. We can use either $L_1 - C_2 - L_3 - C_4 - L_5$ topology or $C_1 - L_2 - C_3 - L_4 - C_5$ topology. Again, the second one (shown in Fig. 6.6(b)) is preferable since it uses smaller number of inductors. Using the $n = 5$ values in Table 6.1 and $R$=50 Ω we find

$$C_1 = C_5 = \frac{1/0.618}{2\pi \cdot 50 \cdot 15 \cdot 10^6} = 343 \text{ pF},$$

$$C_3 = \frac{1/2.0}{2\pi \cdot 50 \cdot 15 \cdot 10^6} = 106 \text{ pF}, \quad L_2 = L_4 = \frac{(1/1.618)50}{2\pi \cdot 15 \cdot 10^6} = 0.330\ \mu\text{H}$$

This filter attenuates the signal at 7.5 MHz (one octave lower than $f_c$=15 MHz) by $6\times n$=$6\times 5$=30 dB.

---

**Band-pass-filter**

Band-pass-filters have a passband with a center frequency of $f_o$ and 3-dB cut-off frequencies of $f_1$ and $f_2$. The bandwidth of the filter is $\Delta f = f_2 - f_1$. It is composed of series and parallel $LC$ branches resonating at $f_o$. They have a transfer function of the following form:

$$\frac{P_L}{P_A} = \frac{1}{1 + (f_o/\Delta f)^{2n}(f/f_o - f_o/f)^{2n}} \tag{6.24}$$

For $f \to 0$, we have $P_L/P_A \to (f\Delta f/f_o^2)^{2n}$, and for $f \to \infty$, we have $P_L/P_A \to (\Delta f/f)^{2n}$ defining the asymptotes. $P_L/P_A = 0.5 = -3$ dB when the denominator is equal to 2. This occurs at frequencies

$$\left(\frac{f}{f_o} - \frac{f_o}{f}\right) = \pm\frac{\Delta f}{f_o} \quad \text{or} \quad f_{1,2} = \sqrt{f_o^2 + \frac{\Delta f^2}{4}} \pm \frac{\Delta f}{2} \tag{6.25}$$

Note that we have $f_1 f_2 = f_0^2$.

We use the following procedure to design an $n$th-order Butterworth band-pass-filter of center frequency $f_o$ and a 3-dB bandwidth of $\Delta f$ for load and source impedances of $R$:

1. Design an $n$th-order low-pass-filter using $\Delta f$ as the cutoff frequency.
2. For every shunt capacitor, $C_i$, add a parallel inductor of value

$$L_i = \frac{1}{(2\pi f_o)^2 C_i} \tag{6.26}$$

3. For every series inductor, $L_i$, add a series capacitor of value

$$C_i = \frac{1}{(2\pi f_o)^2 L_i} \tag{6.27}$$

The transfer function of first-, second-, and third-order band-pass-filters with a normalized bandwidth of $\Delta f/f_o = 0.5$ are plotted in Fig. 6.7 along with the asymptotes at low and high frequencies.

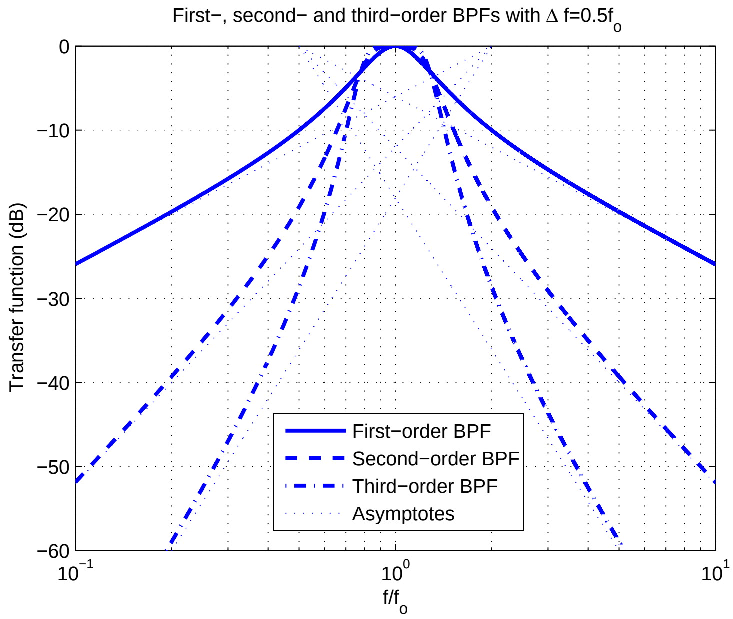



Figure 6.7: Transducer power gain of band-pass-filters with $\Delta f/f_o = 0.5$ as a function of normalized frequency.

An approximate transfer function of a BPF can be drawn using the following steps:

1. Plot the points $(f_1, -3\text{ dB})$, $(f_o, 0\text{ dB})$ and $(f_2, -3\text{ dB})$
2. Draw the high-frequency asymptote with a slope of $-20n$ dB/decade passing through $(\Delta f, 0\text{ dB})$ point.

3. Draw the low-frequency asymptote with a slope of $+20n$ dB/decade passing through $(f_0^2/\Delta f, 0\text{ dB})$ point.

4. Join the points and asymptotes to get an approximate transfer function.

Note that for narrow-band filters, the asymptotes can be used to find the approximate transfer function values instead of the full expression of Eq. 6.24, for $f/f_o > 10$ or forg $f/f_o < 0.1$.

**Example 5**

Let us design a third-order band-pass-filter centered at $f_0$=28 MHz with a bandwidth of $\Delta f$=5 MHz for $R_S = R_L$=50 Ω. We first design a low-pass-filter with $n$=3 using $C_1 - L_2 - C_3$ topology:

$$C_1 = C_3 = \frac{1.0}{2\pi \cdot 50 \cdot 5 \cdot 10^6} = 637 \text{ pF} \quad \text{and} \quad L_2 = \frac{2.0 \cdot 50}{2\pi \cdot 5 \cdot 10^6} = 3.20\mu\text{H}$$

Now we add parallel inductors $L_1$ and $L_3$ and a series capacitor $C_2$ with values found from the resonance condition at 28 MHz, Eq. 5.4 (page 184);

$$L_1 = L_3 = \frac{25330}{28^2 \cdot 637} = 0.0510\mu\text{H} = 51 \text{ nH}, \quad C_2 = \frac{25330}{28^2 \cdot 3.2} = 10.0 \text{ pF}$$

A schematic of this filter is given in Fig. 6.8. We find −3 dB frequencies from

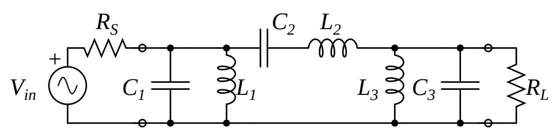



Figure 6.8: 3rd order BPF.

Eq. 6.25, $f_1$=25.61 MHz and $f_2$=30.61 MHz. The filter attenuates input signal by $20 \times 3 = 60$ dB at $f$=50 MHz (one decade higher than $\Delta f$=5 MHz), and at $f$=15.68 MHz (one decade lower than $f_o^2/\Delta f = 28^2/5$=156.8 MHz.

---

### 6.2.2 Chebyshev filters

An $n$th order Chebyshev filter is another polynomial filter seeking to have a transducer power gain of

$$G_T = \frac{P_L}{P_A} = \frac{1}{1 + \varepsilon^2 T_n^2(f/f_c)} \tag{6.28}$$

where $T_n$ is the Chebyshev polynomial† of $n$th order, $\varepsilon$ specifies the passband ripple, and $f_c$ is the cutoff frequency at the passband ripple. The first few

†Invented by Russian Mathematician Pafnuty Lvovich Chebyshev (1821–1894).

| $n$ | $b_1$ | $b_2$ | $b_3$ | $b_4$ | $b_5$ | $b_6$ | $b_7$ | $b_8$ |
|---|---|---|---|---|---|---|---|---|
| 3 | 0.6292 | 0.9703 | 0.6292 | | | | | |
| 4 | 0.7129 | 1.200 | 1.321 | 0.6476 | | | | |
| 5 | 0.7563 | 1.305 | 1.577 | 1.305 | 0.7563 | | | |
| 6 | 0.7814 | 1.360 | 1.690 | 1.535 | 1.497 | 0.7098 | | |
| 7 | 0.7969 | 1.392 | 1.748 | 1.633 | 1.748 | 1.392 | 0.7969 | |
| 8 | 0.8073 | 1.413 | 1.782 | 1.683 | 1.853 | 1.619 | 1.555 | 0.7334 |

Table 6.2: Table of prototype element values in Chebyshev low-pass-filters with passband ripple of 0.01 dB ($\varepsilon = 0.0480$).

| $n$ | $b_1$ | $b_2$ | $b_3$ | $b_4$ | $b_5$ | $b_6$ | $b_7$ | $b_8$ |
|---|---|---|---|---|---|---|---|---|
| 3 | 1.228 | 1.153 | 1.228 | | | | | |
| 4 | 1.303 | 1.284 | 1.976 | 0.8468 | | | | |
| 5 | 1.339 | 1.337 | 2.166 | 1.337 | 1.339 | | | |
| 6 | 1.360 | 1.363 | 2.239 | 1.456 | 2.097 | 0.8838 | | |
| 7 | 1.372 | 1.378 | 2.276 | 1.500 | 2.276 | 1.378 | 1.372 | |
| 8 | 1.380 | 1.388 | 2.296 | 1.522 | 2.341 | 1.493 | 2.135 | 0.8972 |

Table 6.3: Table of prototype element values in Chebyshev low-pass-filters with passband ripple of 0.2 dB ($\varepsilon = 0.2171$).

Chebyshev polynomials can be written as

$$\begin{aligned} T_0(x) &= 1 \\ T_1(x) &= x \\ T_2(x) &= 2x^2 - 1 \\ T_3(x) &= 4x^3 - 3x \\ T_4(x) &= 8x^4 - 8x^2 + 1 \\ T_5(x) &= 16x^5 - 20x^3 + 5x \\ T_6(x) &= 32x^6 - 48x^4 + 18x^2 - 1 \end{aligned}$$

This filter gives a sharper cutoff (a better rejection outside the passband) than the Butterworth filter, but it has a ripple in the passband. The design method is the same as that of the Butterworth filter using the prototype values given in Tables 6.2 or 6.3. Filter characteristics are shown for several $n$ values in Figs. 6.9 and 6.10.

**Example 6**

Let us design a 5.-order 0.2 dB ripple Chebyshev low-pass-filter for a cutoff frequency of $f_c$=20 MHz for source and load impedances of $R_S = R_L$=300 Ω. We use $C_1 - L_2 - C_3 - L_4 - C_5$ topology. Using the $n = 3$ values in Table 6.3 and $R$=300 Ω we find

$$C_1 = C_5 = \frac{1.339}{2\pi 300 \cdot 20 \cdot 10^6} = 35.5 \text{ pF} \text{ and } L_2 = L_4 = \frac{1.337 \cdot 300}{2\pi 20 \cdot 10^6} = 3.19\mu\text{H}$$

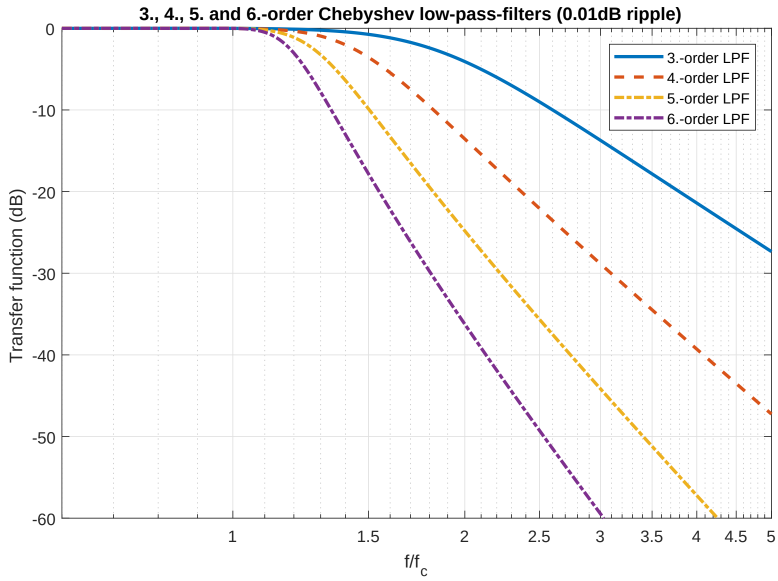



Figure 6.9: Chebyshev LPF response with 0.01 dB ripple for different $n$ values. At $f = f_c$, the filters have 0.01 dB loss.

$$C_3 = \frac{2.166}{2\pi 300 \cdot 20 \cdot 10^6} = 57.5\ pF$$

At $f = f_c$, the filter has an attenuation of 0.2 dB. We can estimate the performance of the filter at 40 MHz using Fig. 6.10: Since $f_c$ =20 MHz, at 40 MHz ($f/f_c = 2$) the signal is attenuated by 38 dB (voltage is 0.013 times).

---

### 6.2.3 Practical aspects of $LC$ filter design

Recall that inductors and capacitors do not behave as they should at frequencies higher than their self-resonance-frequency. Caution must be exercised while choosing the components. Their high-frequency response should be carefully evaluated.

Some types of capacitors work better than others do at higher frequencies. For example, ceramic NPO capacitors work well at RF frequencies with a high $Q$ factor. ESR (equivalent series resistor) of capacitors (especially those with large values) may also limit their performance in filtering applications.

Inductors are generally more expensive and less ideal than capacitors. Hence, filter topologies using fewer number inductors are usually preferred. Careful evaluation of the inductor [9–11] in the frequency range of interest must be done before using it in a filter. Magnetic coupling between the inductors in a filter can be minimized by placing the inductors perpendicular to each other.

While higher-order filters provide more attenuation than lower-order filters, it is challenging to get attenuation levels more than 50 dB on the same printed-circuit-board due to electromagnetic coupling effects between the components.

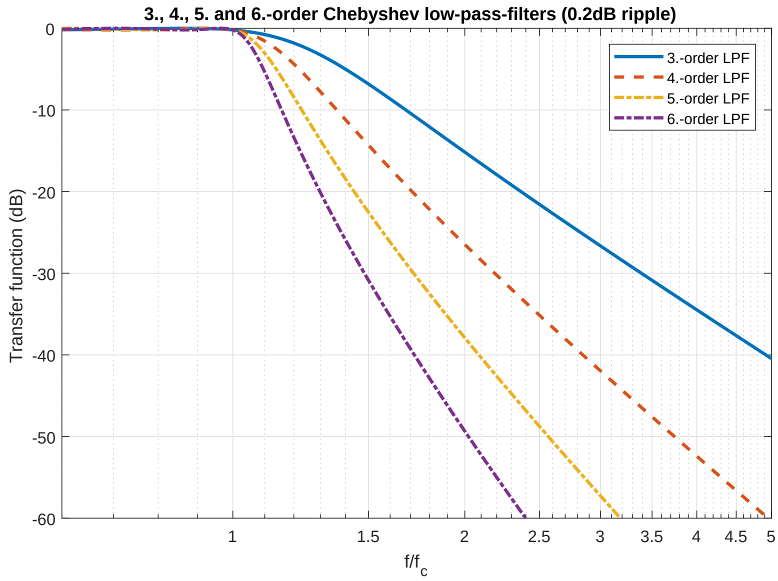



Figure 6.10: Chebyshev LPF response with 0.2 dB ripple for different $n$ values. At $f = f_c$, the filters have 0.2 dB loss.

If higher attenuation levels are desired, electromagnetic *shielding* of components is necessary. Shielding is done by conductive or magnetic materials, surrounding the component to block the electromagnetic field. Copper or sheet iron is commonly used as shielding materials. A conductive enclosure is also known as a *Faraday cage*.

In band-pass-filters, spurious capacitors limit the performance of floating series $LC$ circuits. Shunt $LC$ branches should be preferred, wherever possible.

## 6.3 Impedance matching

We discussed in Section 5.8 that the maximum transfer occurs when the load impedance is equal to the conjugate of the source impedance. If the condition is not satisfied, we need to convert the impedance to the required level using a lossless network.

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.

### 6.3.1 Matching by transformers, narrow-band

We discussed the transformer's impedance transformation property in Section 5.5. By choosing the turns ratio correctly, it is possible to transform the impedances for maximum power transfer.

There are two approaches to designing 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 narrowband or resonant transformers. The idea is simple: if we tune out the magnetizing inductance $L_p$ (see Fig. 5.21) by a parallel capacitor across the primary terminals as depicted in Fig. 6.11(a) (or equivalently the secondary terminals), we end up with only an ideal transformer left at the frequency of interest. So we should choose the capacitor value from the resonance formula as

$$C = \frac{1}{\omega_o^2 L_p} \tag{6.29}$$

With the magnetizing inductance, $L_p$, tuned out, the resistance seen from the input, $R$, is given by

$$R = \left(\frac{n_1}{n_2}\right)^2 R_L \tag{6.30}$$

This is suitable only over a frequency range limited by the $Q$ of the parallel tuned circuit. $Q$ is given by

$$Q = \frac{R}{\omega_o L_p} = \left(\frac{n_1}{n_2}\right)^2 \frac{R_L}{\omega_o L_p} \text{ with a bandwidth } BW = \frac{w_o}{Q} \tag{6.31}$$

In this case, the value of $L_p$ need not be very large, but is chosen to provide the necessary bandwidth.

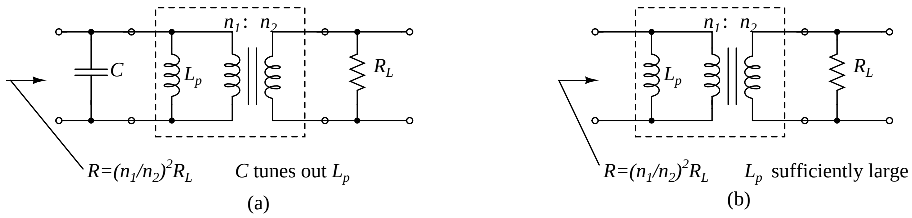



Figure 6.11: (a) Narrow-band transformer, (b) wide-band transformer.

**Example 7**

Design a transformer to transform 20 Ω into 50 Ω at $f_o$=10 MHz. We have a core with $A_L$=3 nH/turns$^2$.

Referring to Eq. 6.30, we should choose the turns ratio as

$$\frac{n_1}{n_2} = \sqrt{\frac{R}{R_L}} = \sqrt{\frac{50}{20}} = 1.58$$

Let us choose $n_1 : n_2$=11:7, since 11/7=1.57. The inductance of the primary is $L_p = n_1^2 A_L$=363 nH. We choose a capacitance to tune out the inductance at 10 MHz. Using the resonance formula of Eq. 5.4 on page 184:

$$C = \frac{25330}{10^2 \cdot 0.363} = 698 \text{ pF}$$

$Q$ factor is given by Eq. 6.31

$$Q = \frac{R}{\omega L_p} = \frac{50}{2\pi 10 \cdot 10^6 \cdot 363 \cdot 10^{-9}} = 2.2$$

Hence the transformation will be valid in a band of $\Delta f = f_2 - f_1 = f_o/Q = 10/2.2 = 4.5$ MHz. Since $f_1 f_2 = 10^2$, we find $f_1$=8 MHz and $f_2$=12.5 MHz.

---

♦ TRC-11 utilizes the narrow-band transformer matching technique to maximize the gain in both transmitter and receiver.

### 6.3.2 Matching by transformers, wide-band

The second approach is wideband transformer design. In this case, we require $|j\omega L_p|$ to be significantly larger than the effective impedance that appears across it over the frequency range of interest so that $L_p$ is negligible. 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 (Fig. 6.11(b)). If $\omega_1$ is the lowest frequency of interest, we should have

$$\omega_1 L_p \geq 4 \left(\frac{n_1}{n_2}\right)^2 R_L \tag{6.32}$$

The turns ratio of the transformer should be chosen as

$$R = \left(\frac{n_1}{n_2}\right)^2 R_L \tag{6.33}$$

While $L_p$ determines the lowest frequency of the impedance transformation, core loss or interwinding capacitance may limit the upper-frequency limit.

**Example 8**

Design a transformer to transform the 4 Ω speaker impedance of the heavy-metal concert given on page 210 to 8 Ω to recover the lost 111 W. We need a wideband transformation covering the whole audio range (20 Hz to 20 kHz). Hence the lowest frequency of interest is $f_1$=20 Hz. We have a core with $A_L$=650 nH/T$^2$.

Referring to Fig. 6.11(b), we should choose the turns ratio of the transformer from Eq. 6.33 as

$$\frac{n_1}{n_2} = \sqrt{\frac{R}{R_L}} = \sqrt{\frac{8}{4}} = 1.41$$

To have a wide-band transformer, from Eq. 6.32 we should choose

$$L_p \geq 4 \left(\frac{n_1}{n_2}\right)^2 \frac{R_L}{\omega_1} = 4\frac{8}{4}\frac{4}{2\pi 20} = 256 \text{ mH}$$

Hence $n_1 = \sqrt{256000/0.65} = 628$. We find $n_2$=445.

---

**Example 9**

Design a transformer to transform 10 Ω into 50 Ω between 4 MHz to 15 MHz. We have a core with $A_L$=3 nH/turns$^2$.

Referring to Fig. 6.11(b), we should choose the turns ratio as

$$\frac{n_1}{n_2} = \sqrt{\frac{50}{10}} = 2.24$$

To have a wide-band transformer, from Eq. 6.32 we should choose

$$L_p \geq 4\frac{50}{10}\frac{10}{2\pi 4 \cdot 10^6} = 7.96\ \mu\text{H}$$

Let $L_p$=8 $\mu$H>7.96 $\mu$H, hence $n_1$=$\sqrt{8000/3}$=51.6. Choose $n_1$=52 turns. We find $n_2$=23.2. Since $n_2$ is not close to an integer, let us choose the better pair of 56:25 (56/25=2.24).

---

### 6.3.3 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 the amplification property of tuned circuits in Section 5.3. The same property provides a means of matching. Fig. 5.8 is repeated here as Fig. 6.12, for convenience.

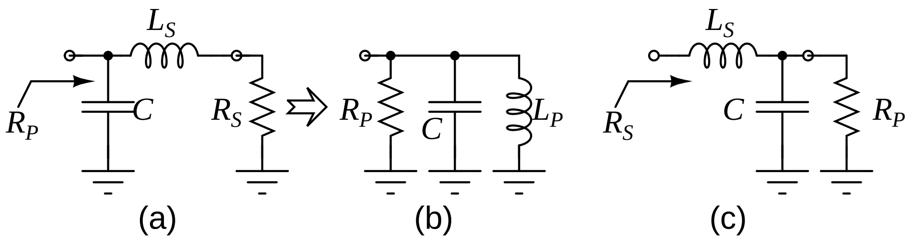



Figure 6.12: (a) $L$-section to transform a smaller resistor $R_S$ to a larger resistance $R_P$, (b) equivalent circuit, (c) $L$-section to transform a larger resistance $R_P$ to a smaller resistance $R_S$.

Referring to Fig. 6.12(a), a *smaller* resistance $R_S$ in series with the inductor is transformed into a *larger* resistance $R_P$

$$R_P = (Q^2+1)R_S \text{ with } \omega_o = \frac{1}{\sqrt{L_S C(1+1/Q^2)}} \text{ and } Q = \frac{\omega_o L_S}{R_S} \qquad (6.34)$$

There is a transformation ratio of $Q^2+1$. The equivalent circuit is shown in Fig. 6.12(b), where

$$L_P = L_S\left(1+\frac{1}{Q^2}\right) \qquad (6.35)$$

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 used backward (see Fig. 6.12(c)) transforms a *larger* resistance $R_P$ into a *smaller* resistance $R_S$:

$$R_S = \frac{R_P}{Q^2+1} \text{ with } \omega_o = \frac{1}{\sqrt{L_P C}} \text{ and } Q = \frac{R_P}{\omega_o L_P} \tag{6.36}$$

**Example 10**

Design a resonant matching circuit to convert 10 Ω to 50 Ω at $f_o$=10 MHz.

Referring to Fig. 6.12(a), $R_S$=10 and $R_P$=50. Hence

$$Q^2 + 1 = \frac{R_P}{R_S} = \frac{50}{10} = 5$$

Hence $Q$=2. Since $Q = \omega_o L_S/R_S$, $L_S = 2 \cdot 10/(2\pi 10^7)$=318 nH and $L_P =$ 397 nH. Using the resonance formula, we find the value of capacitance:

$$C = \frac{25330}{10^2 \cdot 0.397} = 637 \text{ pF}$$

This transformation is valid in the range of $\Delta f = f_2 - f_1 = f_o/Q$=10/2=5 MHz. Since $f_1 f_2 = 10^2$, we find $f_1$=7.8 MHz and $f_2$=12.8 MHz.

---

### 6.3.4 Impedance inverters

Impedance inverters are narrow-band impedance transformers. An impedance inverter circuit is depicted in Fig. 6.13(a). The circuit has a "T" form with two equal series reactances and a parallel reactance of the same magnitude but opposite sign. Another inverter form is shown in Fig. 6.13(b), where the circuit is in "$\pi$" form. $+jX$ can be implemented by inductors, while $-jX$ can be realized by capacitors. In addition to the inverter types shown in Fig. 6.13, it is possible to change the sign of $X$ to interchange the positions of inductors and capacitors, generating two more inverter types.

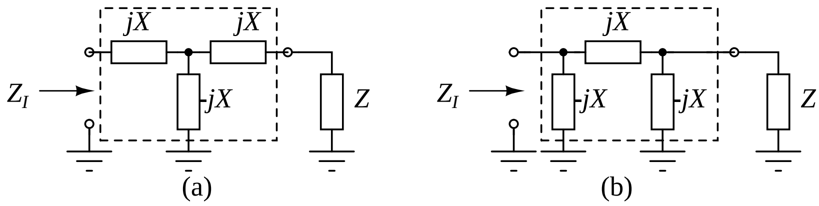



Figure 6.13: Impedance inverters.

When an impedance $Z$ is connected to one end of the first inverter, the impedance seen at the other end, $Z_I$, becomes

$$Z_I = jX + \frac{1}{1/(-jX) + 1/(jX+Z)} = \frac{X^2}{Z} \tag{6.37}$$

This functional circuit is used for many matching and filtering purposes. For example, it can convert a series resonant circuit into a parallel resonant circuit. Assume that $Z$ is the impedance of a series resonant $RLC$ circuit:

$$Z = j\omega L + R + \frac{1}{j\omega C} \tag{6.38}$$

$Z_I$ becomes

$$Z_I = \frac{X^2}{j\omega L + R + \frac{1}{j\omega C}} \tag{6.39}$$

The corresponding admittance, $Y_I$,

$$Y_I = \frac{1}{Z_I} = \frac{j\omega L + R + \frac{1}{j\omega C}}{X^2} = j\omega\frac{L}{X^2} + \frac{R}{X^2} + \frac{1}{j\omega C X^2} \tag{6.40}$$

This is the admittance of a parallel $RLC$ circuit with a capacitance of value $L/X^2$, a conductance of $R/X^2$, and an inductance of $CX^2$.

**Example 11**

Design an impedance inverter to convert $Z$=5 Ω to $Z_I$=50 Ω at 28 MHz.

From Eq. 6.37, we have $X^2$=$Z_I Z$=50·5=250 or $X$=15.8 Ω. We can use either of the inverters shown in Fig. 6.13. To generate $+jX$ we use an inductor of value $15.8/(2\pi 28 \cdot 10^6)$=89.8 nH. To generate $-jX$, we use a capacitor of value $1/(15.8 \cdot 2\pi 28 \cdot 10^6)$=359 pF.

---

**Example 12**

Let us transform a resistance of $R_L$=50 Ω to $R$=100 Ω at a frequency of 16 MHz using different techniques.

Using a transformer we need a turns ratio of $n_1/n_2 = \sqrt{100/50} = \sqrt{2}$. If we can make the primary magnetizing inductance large enough compared to $R = 100\Omega$, it is a wide-band transformation (as in Fig. 6.11(b)). Setting $\omega L_p = 4R = 400$, we find $L_p = 3.97\mu$H. Using a toroidal core with $A_L$=4.5 nH/turns$^2$, we need $n_1$=30 turns, which is a reasonable number. Hence $n_2$=21 turns.

Alternatively, we can make a resonant transformer with less number of turns: Choose $n_1$=14 turns and $n_2$=10 turns. In this case, $L_p = A_L n_1^2 = 4.5 \cdot 14^2$=882 nH. To tune out the primary inductance, we need a capacitor of value $C = 25330/(16^2 \cdot 0.882)$=112 pF in parallel with it (as in Fig. 6.11(a)).

As a third method, we can use an $L$-section as in Fig. 6.12. We choose $R_S = 50\Omega$, and we need to have $R_P = 100\Omega$. So, we have $Q^2 + 1 = R_P/R_S = 2$, and we find $Q = 1$. Since $Q = \omega L_S/R_S = 1$, we determine $L_S$=497 nH and $L_P$=994 nH. The capacitor of the $L$-section is found from the resonance condition: $C = 25330/(16^2 \cdot 0.994)$=99 pF.

As a fourth method, we use one of the impedance inverters of Fig. 6.13. We choose $X = \sqrt{R_L R} = \sqrt{50 \cdot 100}$=70.7. Hence, the series inductor to generate $+jX$ is $L = 70.7/\omega$=703 nH and the shunt capacitor to generate $-jX$ is $C$=140 pF.

---

### 6.3.5 Band-pass-filter design using inverters

To avoid the performance limitation of series $LC$ branches (see Section 6.2.3) in filters, an inverter can be utilized to convert them to parallel $LC$ branches if the filter has a relatively small bandwidth. Consider the second-order band-pass-filter with center frequency $\omega_o$ and bandwidth $\Delta\omega$ shown in Fig. 6.14(a). The component values for the Butterworth filter are

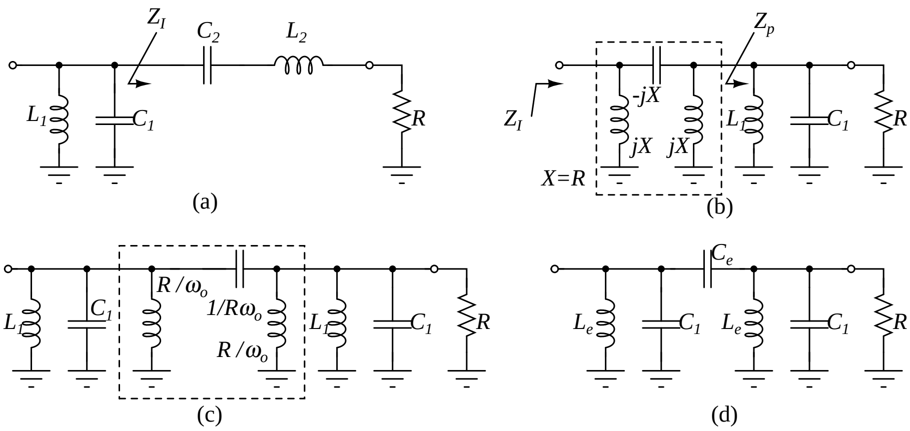



Figure 6.14: (a) Second-order band-pass-filter, (b) inverter with a shunt $LC$ branch, (c) equivalent second-order band-pass-filter, (d) simplified second-order band-pass-filter.

$$C_1 = \frac{1.4142}{\Delta\omega R}, \ \ L_2 = \frac{1.4142 \cdot R}{\Delta\omega}, \ \ L_1 = \frac{1}{\omega_o^2 C_1}, \ \ C_2 = \frac{1}{\omega_o^2 L_2}$$

Now, consider the impedance seen at the input of inverter of Fig. 6.13(b) with $X = R$ shown in Fig. 6.14(b). We can write

$$Z_I = \frac{X^2}{Z_p} = \frac{R^2}{Z_p} = R^2\left(\frac{1}{j\omega L_1} + j\omega C_1 + \frac{1}{R}\right) = \frac{1}{j\omega C_2} + j\omega L_2 + R$$

since $C_2 = L_1/R^2$ and $L_2 = C_1 R^2$. Therefore, the inverter and shunt $L_1C_1$ combination in Fig. 6.14(b) is equivalent to the series $L_2C_2$ branch in Fig. 6.14(a). We show the equivalent filter in Fig. 6.14(c). Since there are two inductors in parallel, we can combine them into one inductor. We find

$$L_e = L_1 \parallel \frac{R}{\omega_o} = \frac{RL_1}{R + \omega_o L_1} \ \text{ and } \ C_e = \frac{1}{R\omega_o} \tag{6.41}$$

The final filter shown in Fig. 6.14(d) is nearly the same as the original filter near the passband where the inverter approximation holds. It has a better performance at lower frequencies, but its performance degrades at higher frequencies. A comparison of the transfer functions of the original second-order BPF with $\Delta f = 0.2 f_o$, and the approximate BPF is given in Fig. 6.15.

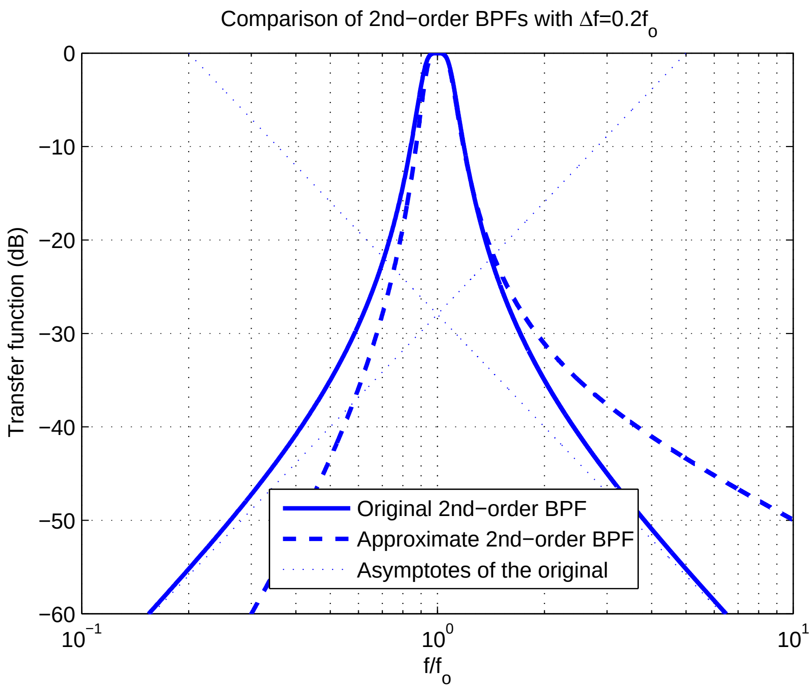



Figure 6.15: Comparison of transfer functions of a second-order Butterworth band-pass-filter with $\Delta f = 0.2 f_o$ and approximate BPF using the inverter.

**Example 13**

Let us design a second-order band-pass-filter (with no series branches) centered at $f_o$=16 MHz with a bandwidth of $\Delta f$=1 MHz for $R_S = R_L$=50 Ω. We first design a low-pass-filter with $n$=2 and bandwidth of 1 MHz using $C_1 - L_2$ topology:

$$C_1 = \frac{1.41}{2\pi \cdot 50 \cdot 1 \cdot 10^6} = 4.5 \text{ nF}$$

Now we add a parallel inductor $L_1$ to $C_1$ from the resonance condition at 16 MHz, using the resonance formula of Eq. 5.4 (page 184):

$$L_1 = \frac{25330}{16^2 \cdot 4500} = 0.22\mu\text{H} = 220 \text{ nH}$$

To get rid of the series branch, we use an inverter with $X$=50 Ω. We need an inductor of $L = R/\omega_o = 50/(2\pi 16 \cdot 10^6)$ =497 nH, and a capacitance of $C_e$=199 pF. Since $L_e$ is the parallel combination of two inductors

$$L_e = \frac{RL_1}{R + \omega_o L_1} = \frac{220 \cdot 497}{220 + 497} = 152 \text{ nH} \tag{6.42}$$

Hence we determined all the values in Fig. 6.14(d).

## 6.4 Crystal filters

Timing circuits in watches, computers, or fixed frequency oscillators in communication equipment require very sharp filter characteristics at a precise resonant frequency. Tuned circuits made of inductors and capacitors cannot meet very tight frequency selectivity requirements. $Q$ of electrical circuits is limited to about 100. So it is challenging to build a filter with a very small bandwidth using just inductors and capacitors. Moreover, the values of inductors may change by the presence of conductors or magnetic materials nearby. For example, the value of an inductor may change as your hands get closer to the inductor. So the stability of the narrow-band filter or a resonator built from an inductor is not acceptable.

On the other hand, mechanical systems can have much higher $Q$ values, and they do not get affected by the presence of a conductor or a magnetic material nearby. For frequency-selective application requiring high $Q$ and stability, mechanical devices made of PZT ceramics and quartz crystals are used [12]. PZT ceramics are *electrostrictive*, and quartz crystals are *piezoelectric* materials. They can convert the electric field applied to 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. The device converts this field into mechanical vibration at the same frequency as the applied voltage. The plate dimensions define the *mechanical resonance* frequencies. When the frequency of the applied field matches one of these resonance frequencies, the impedance that appears across the electrodes makes a dip. The resonance frequency is very stable, and the losses are very small, limited to frictional activity during particle motion.

### Quartz crystals

Quartz crystals have a very high mechanical $Q$, in the order of 100,000, due to their orderly single-crystal structure. A poly-crystalline material does not have such a high mechanical $Q$. Since quartz is also piezoelectric, the mechanical resonance directly influences the electrical properties, and hence quartz displays a similarly high electrical $Q$.

The schematic symbol of a crystal is given in Fig. 6.16(a). There are many

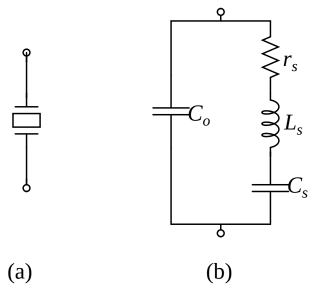



Figure 6.16: Quartz crystal (a) symbol and (b) equivalent circuit.

modes of resonance in a quartz crystal. Each mode has a fundamental resonance frequency and its overtones. We are interested in the fundamental resonance frequency of only one mode of vibration. The equivalent circuit of a quartz crystal in the vicinity of this frequency is also given in Fig. 6.16(b). These equivalent circuit models the impedance at the (electrical) terminals of the quartz crystal. The mechanical properties of quartz and the dimensions of the plate, 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 (inverse of stiffness) of quartz crystal. Resistance $r_s$ models the friction losses during vibration. The only inherently electrical component is $C_o$, which is the capacitance between the electrodes of quartz crystal.

The model in Fig. 6.16(b) has two resonance frequencies:

$$f_s = \frac{1}{2\pi\sqrt{L_s C_s}} \tag{6.43}$$

and

$$f_p = \frac{1}{2\pi\sqrt{L_s C_s C_o/(C_s + C_o)}}. \tag{6.44}$$

$f_s$ 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 $C_o$). $f_p$, on the other hand, is the parallel resonance frequency, where the inductance $L_s$ resonates with the series combination of $C_s$ and $C_o$, i.e., $C_s C_o/(C_s + C_o)$. We note that $f_s < f_p$.

$L_s$ ranges from a few mH to over 10 mH for quartz crystals at 15 MHz. $C_s$ is in the order of fF (femto Farad, $10^{-15}$F) and $r_s$ is in the range of a few ohms to a few tens of ohms. $C_o$ is an electrical component, the clamp capacitance, usually a few pF. We are interested in the frequencies in the vicinity of series resonance.

### 6.4.1 Band-pass-filter using quartz crystals

Suppose we design a second-order band-pass-filter with a center frequency of $\omega_o$ and bandwidth $\Delta\omega$. We use two identical quartz crystals resonant at $\omega_o$ to make a Butterworth band-pass-filter. The method described below can be extended to higher-order filters using more quartz crystals.

Consider the circuit in Fig. 6.17(a). Each quartz crystal resonant at $\omega_o$ provides a series resonance circuit. To employ techniques we use in designing Butterworth filters, we need one series and one parallel resonant circuit. We employ an impedance inverter to invert one of the series resonant circuits into a parallel resonant circuit, as in Section 6.3.4. To satisfy the bandwidth requirement, we choose the external circuit parameters $X$, $R_1$, and $R_2$ appropriately. The inverter inverts the series circuit provided by $XT_2$ and $R_2$ into a parallel tuned circuit. The equivalent circuit of the resulting filter is shown in Fig 6.17(b).

We must set all circuit parameters such that the filter becomes a second-order Butterworth BPF. As we discussed in Section 6.2.1, we first design a low-pass-filter with $\Delta\omega$ as the cutoff frequency, given in Fig. 6.17(c). The termination resistors, $R_o$, of the low-pass-filter prototype in Fig. 6.17(c) are equal. From the inverter formula of Eq. 6.37, we have

$$R_o = R_1 + r_s = \frac{X^2}{R_2 + r_s} \tag{6.45}$$

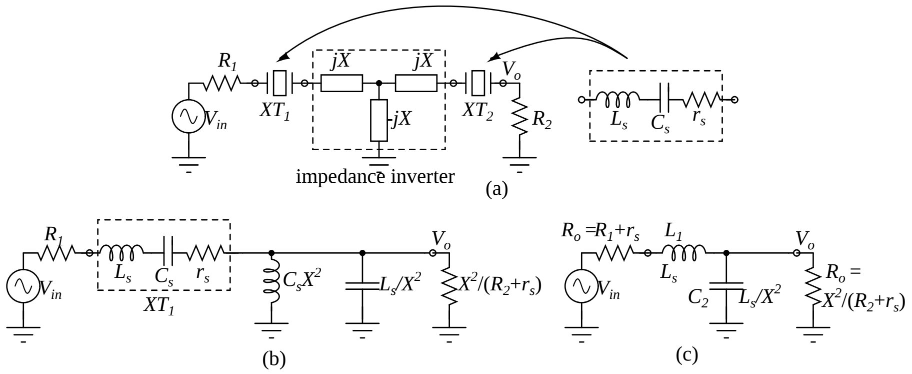



Figure 6.17: (a) Two-crystal ladder filter, (b) the equivalent circuit after inversion operation, (c) the low-pass-filter prototype.

From the low-pass-filter relations of Eq. 6.16 and 6.17, we get

$$L_1 = \frac{1.4142 R_o}{2\pi\Delta f} = L_s \quad \text{and} \quad C_2 = \frac{1.4142}{2\pi\Delta f R_o} = \frac{L_s}{X^2} \tag{6.46}$$

where the factor 1.4142 is taken from the Butterworth table for $n = 2$. Combining the equations in Eq. 6.46, we find

$$C_2 = \frac{L_s}{X^2} = \frac{L_1}{X^2} = \frac{1.4142 R_0}{2\pi\Delta f X^2} = \frac{1.4142}{2\pi\Delta f R_o} \quad \text{or} \quad X^2 = R_o^2 \tag{6.47}$$

Hence we determine the value of $X$ and $R_o$ as follows:

$$X = R_o = \frac{2\pi\Delta f L_s}{1.4142} \tag{6.48}$$

Since the crystals are already resonant at the center frequency, the band-pass-filter design is complete. The termination resistors, $R_1$ and $R_2$ should be chosen as

$$R_1 = R_2 = R_o - r_s = \frac{2\pi\Delta f L_s}{1.4142} - r_s \tag{6.49}$$

Now let us examine how we realize the inverter. The inverter should have a shunt capacitor ($C_i$) and two series inductors ($L_i$), to provide $-jX$ and $jX$ at $f_s$, respectively. The circuit is depicted in Fig. 6.18(a). Here the inverter circuit components are related as

$$X = 2\pi f_s L_i = \frac{1}{2\pi f_s C_i} \tag{6.50}$$

We can ignore and get rid of the series inductors since they are in series with very large inductors of the crystals. Ignoring the series inductors *pull* the center frequency of the band-pass-filter very slightly. The resulting circuit is given in Fig. 6.18(b).

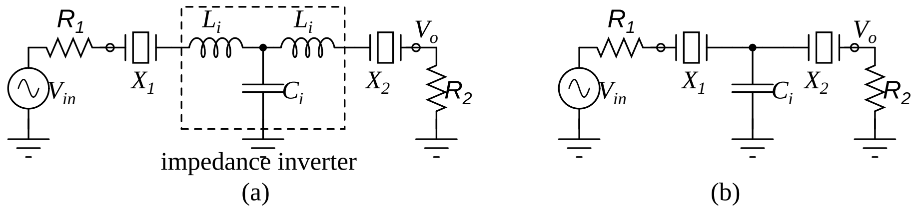



Figure 6.18: Impedance inverter for two-crystal BPF, (a) Proper inverter, (b) Actual circuit with the inductors ignored.

**Example 14**

Suppose we would like to design a second-order BPF centered at $f_o$=16 MHz with a bandwidth of $\Delta f$=10 kHz. If we design this filter using inductors and capacitors, we end up with a very large inductor and a tiny capacitor in the series branch, and a tiny inductor and a very large capacitor in the shunt branch. Let us use two series 16 MHz crystals with an inverter in between. The crystal has $L_s$=15 mH and $r_s = 15\Omega$. From Eq. 6.48, we find

$$X = R_o = R_1 + r_s = R_2 + r_s = \frac{2\pi \cdot 10^4 \cdot 15 \cdot 10^{-3}}{1.4142} = 666\Omega \text{ or } R_1 = R_2 = 651\Omega$$

The inverter inductance of value $L_i$=6.6 $\mu$H is ignored in comparison to $L_s$=15 mH. The inverter capacitor is given by

$$C_i = \frac{1}{2\pi f_o X} = \frac{1}{2\pi \cdot 16 \cdot 10^6 \cdot 666} = 14.9 \text{ pF}$$

We can reduce the bandwidth $\Delta f$ of the filter by a factor of two if we choose $R_1 = R_2 = 318\Omega$ and $C_i$ =29.8 pF.

---

**Example 15**

Design a third-order BPF centered at $f_o$=8.00MHz with a bandwidth of $\Delta f$=20 kHz. We have quartz crystals with $f_s$=8 MHz, $r_s$=10 $\Omega$ and $Q$=100,000.

For the third-order BPF, we need three crystals and two inverters, as shown in Fig. 6.19. From quartz crystal data, we find $L_s = Qr_s/\omega_s$ = 19.9 mH. For

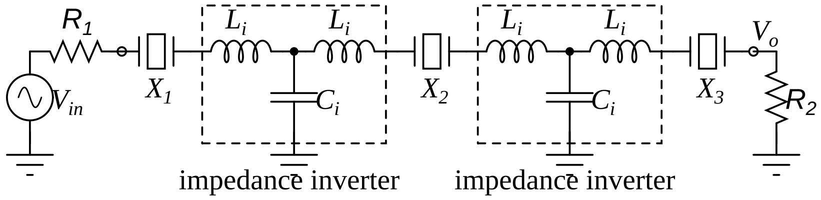



Figure 6.19: Impedance inverter for three-crystal BPF.

$n$=3, the prototype filter values are 1, 2, 1. To match the inductance of the crystal $X_1$ to the first inductance, $L_1$, of the LPF we should have

$$L_s = 19.9 \text{ mH} = L_1 = \frac{R_o}{2\pi\Delta f} = \frac{R_o}{2\pi 20 \cdot 10^3} \tag{6.51}$$

Hence $R_o$=2.5 kΩ. To match the inductance of the crystal $X_2$ to the capacitance, $C_2$, of the bandpass filter, we have

$$C_2 = \frac{2}{2\pi\Delta f R_o} = \frac{2}{2\pi 20 \cdot 10^3 \cdot 2500} = 6.37 \text{ nF} = \frac{L_s}{X^2} \tag{6.52}$$

Hence $X = R_o/\sqrt{2}$=1770 Ω. Hence $L_i = 35$ $\mu$H, and $C_i$=11.2 pF. Note that the third inductance of the BPF, $L_3 = L_1$, is matched to $L_s$ automatically by going through two inverters. We can ignore $L_i$, since it is small compared to $L_s$. The termination resistors should be selected as $R_1 = R_2 = R_o - r_s = 2490$ Ω.

---

♦ TRC-11 has a two-crystal band-pass-filter operating at 15 MHz.

## 6.5 SAW filters

Surface-acoustic-wave (SAW) filters are the most commonly used band-pass-filters in the frequency range 20 MHz to 1000 MHz. They have interdigital surface-acoustic-wave transducers consisting of metal fingers deposited on a piezoelectric crystal like lithium niobate. The input signal is converted to a surface-acoustic-wave travelling on the surface of the crystal. The acoustic signal is converted back to an electrical signal at the output. SAW filters can act like band-pass-filters of very high order ($n > 12$). They can be produced at a low cost with an accurately defined center frequency. All modern TV sets and mobile phones have SAW filters as their band-pass-filters.

## 6.6 Examples

### Example 16

We have a source with $R_S = 50\ \Omega$ and a load impedance as given in Fig. 6.20(a). Design a matching circuit to maximize the power transfer to the 2 Ω load resistor for $\omega = 2\pi 28 \cdot 10^6$. We have a core with $A_L = 2.3$ nH/T$^2$.

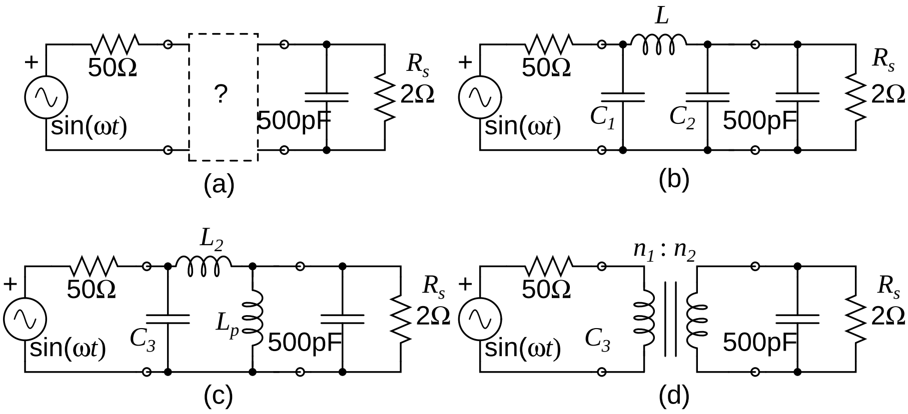



Figure 6.20: (a) Source and load impedances for Example 16, (b) an impedance inverter used as a matching circuit, (c) an $L$-section used as a matching circuit, (d) a transformer used as a matching circuit.

#### Solution

Many possibilities for the matching circuit exist. As a first method, let us use an impedance inverter. We prefer to use a "$\pi$" type impedance inverter as depicted in Fig. 6.20(b), since there is already a shunt capacitor as part of the load impedance. We find $X^2 = 50 \cdot 2 = 100$, or $X = 10$. Hence $C_1 = 1/(10 \cdot 2\pi 28 \cdot 10^6) = 568$ pF and $L = 10/(2\pi 28 \cdot 10^6) = 56.8$ nH obtained with 5 turns on the core. We also have $C_2 + 500 = 568$, hence $C_2 = 68$ pF.

As a second method, let us utilize an $L$-section. First, we tune out the capacitor with a parallel inductor of value

$$L_p = \frac{25330}{28^2 \cdot 500} = 65 \text{ nH}$$

This can be obtained with (squeezed) 5 turns. We need an $L$-section with $Q^2 + 1 = 50/2 = 25$ or $Q = 4.9$. Since $Q = \omega L_2 / R_s$, we get $L_2 = 55.7$ nH (5 turns) and $C_3 = 560$ pF.

As a third method, we use a transformer whose secondary inductance tunes out the 500 pF capacitor. The turns ratio of the transformer must be

$$\frac{n_1}{n_2} = \sqrt{\frac{50}{2}} = 5$$

Hence the primary has 25 turns, and the secondary has 5 turns.

## Example 17

Design a band-pass-filter between the source and load resistor of 500 Ω. 3-dB frequencies are 1 kHz to 5 kHz. We would like to reject the 500 Hz and 9.5 kHz by at least 20 dB.

### Solution

Since $\Delta f = 5000 - 1000 = 4000$, using Eq. 6.25 we write

$$f_1 = 1000 = \sqrt{f_o^2 + \frac{\Delta f^2}{4}} - \frac{\Delta f}{2} = \sqrt{f_o^2 + \frac{4000^2}{4}} - \frac{4000}{2}$$

and find $f_o = 2236$ Hz. Since $-20$ dB means a power ratio of 0.01, using Eq. 6.24 for $f = 500$ Hz, we get

$$0.01 > \frac{P_L}{P_A} = \frac{1}{1 + (2236/4000)^{2n}(500/2236 - 2236/500)^{2n}}$$

To find the order of the filter, we try different $n$ values: For $n = 2$, $P_L/P_A = 0.03$ and for $n = 3$, $P_L/P_A = 0.006 < 0.01$. For $f = 9.5$ kHz, we get

$$0.01 > \frac{P_L}{P_A} = \frac{1}{1 + (2236/4000)^{2n}(9500/2236 - 2236/9500)^{2n}}$$

For $n = 3$, $P_L/P_A = 0.008 < 0.01$.

Therefore, we need a BPF with $n = 3$ as in Fig. 6.8 of page 230. We first design a LPF with $f_c = \Delta f = 4000$ Hz: $C_1 = C_3 = 79.6$ nF, and $L_2 = 40$ mH. We find the inductors and capacitor to resonate at 2236 Hz as $L_1 = L_3 = 63.6$ mH and $C_2 = 127$ nF.

---

## Example 18

The OPAMP circuit shown in Fig. 6.21 is a first-order band-pass-filter. Find the center frequency and the bandwidth in terms of the given component values.

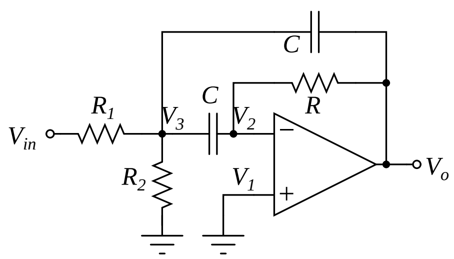



Figure 6.21: OPAMP band-pass-filter.

**Solution**

Assuming that the OPAMP is not saturated, we have $V_1 = V_2 = 0$. We can write the node equation for $V_3$ as

$$\frac{V_3 - V_{in}}{R_1} + \frac{V_3}{R_2} + \frac{V_3 - V_o}{1/j\omega C} + \frac{V_3}{1/j\omega C} = 0$$

Writing KCL at $V_2$

$$\frac{-V_3}{1/j\omega C} + \frac{-V_o}{R} = 0 \quad \text{or} \quad V_3 = -\frac{V_o}{j\omega RC}$$

We combine the equations above to eliminate $V_3$:

$$H(\omega) = \frac{V_o}{V_{in}} = \frac{R}{2R_1} \frac{j\omega 2R_1R_2C}{(R_1 + R_2 - \omega^2 R_1R_2RC^2) + j\omega^2 R_1R_2C}$$

The real part of the denominator becomes zero when

$$\omega_o = \frac{1}{C}\sqrt{\frac{R_1 + R_2}{R_1R_2R}}$$

where the highest gain is obtained

$$H(\omega_o) = \frac{R}{2R_1}$$

The 3-dB frequencies can be found easily if we assume a high-$Q$ and equate the magnitudes of the real and imaginary parts of the denominator:

$$R_1 + R_2 - \omega^2 R_1R_2RC^2 = \pm\omega^2 R_1R_2C$$

Solving the quadratic equation, we find the 3-dB frequencies as

$$\omega_{1,2} = \sqrt{\omega_o^2 + \left(\frac{1}{RC}\right)^2} \mp \frac{1}{RC}$$

Hence the bandwidth of the filter is

$$\Delta\omega = \frac{2}{RC}$$

For example, with $C = 10$ nF and $R = 2$ k, $\Delta f = 16$ kHz. With $R_1 = 1$ k and $R_2 = 10\ \Omega$, we get $f_o = 113$ kHz and a unity gain at the center frequency.

---

## Example 19

The OPAMP circuit shown in Fig. 6.22 is a band-stop-filter or a *notch* filter. It can be used to eliminate unwanted frequencies in a signal. Find the notch frequency and the 3-dB bandwidth of the filter.

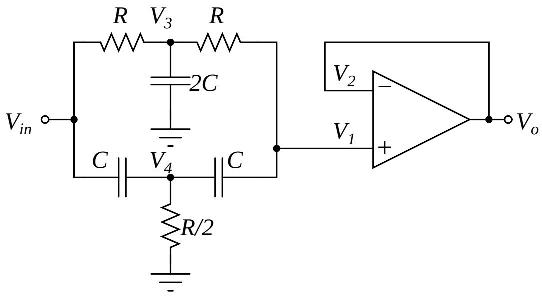



Figure 6.22: Notch filter.

**Solution**

We write the node equations for $V_3$, $V_4$ and $V_1$ as

$$\frac{V_3 - V_{in}}{R} + \frac{V_3}{1/j\omega 2C} + \frac{V_3 - V_1}{R} = 0 \tag{6.53}$$

$$\frac{V_4 - V_{in}}{1/j\omega C} + \frac{V_4}{R/2} + \frac{V_4 - V_1}{1/j\omega C} = 0 \tag{6.54}$$

$$\frac{V_1 - V_3}{R} + \frac{V_1 - V_4}{1/j\omega C} = 0 \qquad \text{or} \qquad V_4 = \frac{1+X}{X}V_1 - \frac{1}{X}V_3 \tag{6.55}$$

where we used $X = j\omega RC$ to simplify the notation. From Eq. 6.53, we get

$$V_3 = \frac{X}{2+2X}(V_{in} + V_1) \tag{6.56}$$

Combining Eqs. 6.54 and 6.55, we find

$$V_3 = -\frac{X^2}{2+2X}V_{in} + \frac{2+4X+X^2}{2+2X}V_1 \tag{6.57}$$

Equating Eqs. 6.56 and 6.57, we get

$$X(V_{in} + V_1) = -X^2 V_{in} + (2 + 4X + X^2)V_1 \tag{6.58}$$

After rearrangement, we reach at

$$\frac{V_o}{V_{in}} = \frac{V_1}{V_{in}} = \frac{1+X^2}{1+4X+X^2} = \frac{1-\omega^2R^2C^2}{(1-\omega^2R^2C^2) + j\omega 4RC} \tag{6.59}$$

since the OPAMP is configured as a unity gain buffer and $V_o = V_1$. The transfer function approaches unity when $\omega \to 0$ and when $\omega \to \infty$. The notch occurs when the numerator is zero:

$$\omega_o = \frac{1}{RC} \tag{6.60}$$

To find the 3-dB frequencies, we set the magnitude of the transfer function to $1/\sqrt{2}$:

$$\left|\frac{V_o}{V_{in}}\right|^2 = \frac{(1-\omega^2R^2C^2)^2}{(1-\omega^2R^2C^2)^2 + (\omega 4RC)^2} = \frac{1}{2} \tag{6.61}$$

The solution of the quadratic equation gives

$$\frac{\omega_{1,2}}{\omega_o} = \sqrt{9 \mp \sqrt{80}} \qquad \text{or} \qquad \frac{\omega_1}{\omega_o} = 0.236 \quad \text{and} \quad \frac{\omega_2}{\omega_o} = 4.23 \tag{6.62}$$

---

## Example 20

Consider the circuit of Fig. 6.23(a) where the reactances are specified at $f =$ 20 MHz. Find the input impedance $Z_{in}$ at the same frequency, $f$, using the impedance inverter formula, rather than series/parallel combination of impedances.

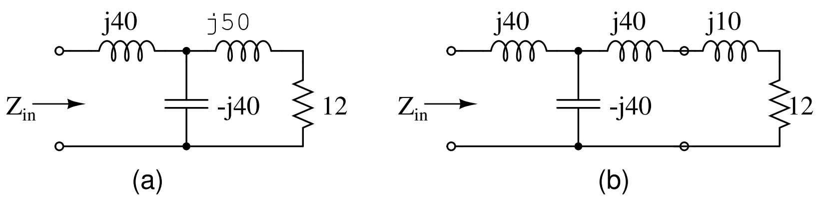



Figure 6.23: (a) Circuit for Example 20, (b) Modified circuit for simple analysis.

**Solution**

To make the circuit look like an impedance inverter we separate the rightmost inductor into two inductors or two reactances $j50 = j40 + j10$, as shown in Fig. 6.23(b). Hence the effective load impedance of the inverter becomes $Z_{out} =$ $12 + j10$. We can find the input impedance, $Z_{in}$, using the impedance inverter formula with $X = 40$:

$$Z_{in} = \frac{X^2}{Z_{out}} = \frac{40^2}{12 + j10} = 78.7 - 65.5$$

---

## Example 21

Find $Z_{in}$ for the circuit of Fig. 6.23(a) at 22 MHz.

**Solution**

The circuit at 22 MHz becomes as shown in Fig. 6.24(a) since the inductive reactances increase by 10% and capacitive reactances decrease by 10%. We modify the circuit so that a portion of the circuit becomes an inverter as in Fig. 6.24(b). We can find the input impedance at 22 MHz using an impedance inverter with $X$=36.4 and a series inductive reactance.

$$Z_{in} = j7.6 + \frac{36.4^2}{12 + j18.6} = 32.4 - j42.6$$

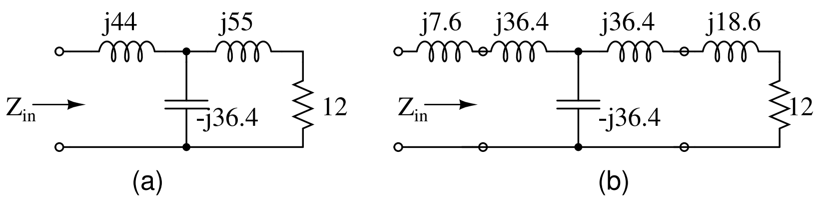



Figure 6.24: (a) Circuit for Example 21, (b) Modified circuit for simple analysis.

## 6.7 Problems

1. Find the amplitude and phase of the voltage at the output of the circuit given in Fig. 6.25 when the input voltage is $2\cos(2\pi 16 \cdot 10^6 t)$ volts. Find the amplitude of the current flowing through the capacitor.

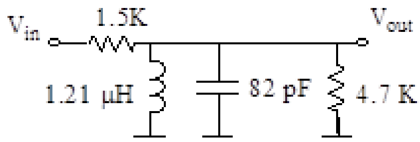



Figure 6.25: Circuit for Problem 1

2. Design a 3rd order Butterworth low-pass-filter with a minimum number of inductors, whose cutoff frequency is 3 kHz, and termination impedances are 1 K.

3. Find the amplitude and phase of the voltage at the output of the circuit given in Fig. 6.26 when the input current is $10\cos(1.82 \cdot 10^8 t)$ mA. Find the amplitude of the current flowing through the inductor.

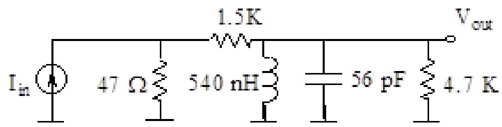



Figure 6.26: Circuit of Problem 3

4. Find the 3-dB cutoff frequency of the low-pass-filter given in Fig. 6.27.

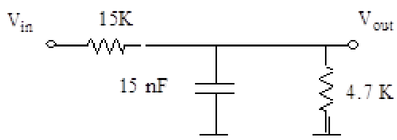



Figure 6.27: Circuit for Problem 4

5. Design a 3rd order Butterworth high-pass-filter with a minimum number of inductors, whose cutoff frequency is 3 MHz, and termination impedances are 75 Ω.

6. Design a Butterworth LP filter that has an attenuation of at most 1 dB at 16 MHz and an attenuation of at least 20 dB at 32 MHz (first, find the minimum number of components). Find reactive element values for a filter with a minimum number of inductors. Finally, check the attenuation at specified frequencies for your filter.

7. Design a Butterworth band-pass-filter with a 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.

8. Assume we have an ideal transformer with a primary/secondary winding ratio of 1:2. What impedance appears across the primary if a 300 Ω is connected across the secondary?

9. Assume we wind up an excellent real transformer with no loss and with a 1:2 primary/secondary winding ratio. The primary inductance is 1 $\mu$H. Find the impedance across the primary at 10 MHz when 300 Ω is connected across the secondary. What is the phase shift between the primary voltage and primary current?

10. Calculate the transformer current and the magnetizing current for the transformer and load given in Problem 9 when the transformer is driven by a voltage of $2\angle 0^o$volts at 10 MHz.

11. A real transformer has a primary inductance of 1 $\mu$H, and a turns ratio of 12:24. Assuming that the transformer has no loss and no leakage, calculate the impedance across the primary at 28 MHz when a 560 Ω load is connected across the secondary. What is the phase shift between the primary voltage and primary current?

12. Find the value of parallel capacitance required across the primary to tune out the primary inductance at 28 MHz for the transformer and load given in Problem 11. Find the impedance across the primary at 28 MHz. Calculate the phase shift between the primary voltage and primary current when this capacitance is connected across the primary.

13. Match the 200 Ω load resistor to the source side for maximum power transfer at 16 MHz, using T38-8/90 ($A_L$=20 nH/turns$^2$) in the circuit shown in Fig. 6.28. Determine $H(\omega) = V_{out}(\omega)/V_s(\omega)$ at 16 MHz.

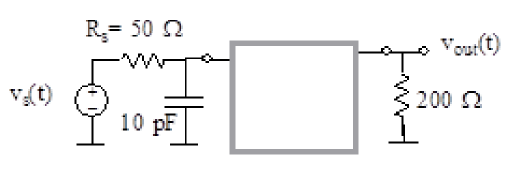



Figure 6.28: Circuit for Problem 13

14. Match a 200 Ω resistor to 50 Ω source resistance using an $L$-section at 5 MHz. Find the frequency range of the match.

15. Design a band-pass-filter centered at 10 MHz with a bandwidth of 20 kHz using two quartz crystals resonant at 10 MHz. The quartz crystals have $Q = 80,000$ and $r_s$=10 Ω.

16. Find the input impedance, $Z_{in}$, of the circuit in Fig. 6.29 using the impedance inverter formula.

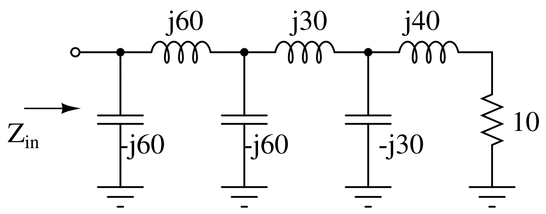



Figure 6.29: Circuit for Problem 16

# Chapter 7

# DIODES IN TELECOMMUNICATIONS

We used diodes to rectify the AC voltage and convert it to a DC 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, the diode becomes almost a short circuit. Otherwise, it remains open. This useful property of diodes is exploited in many applications in telecommunications electronics [13].

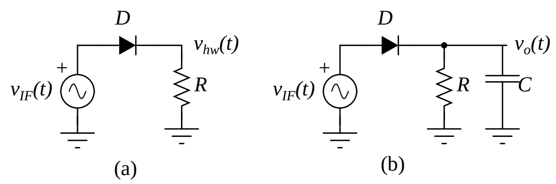



Figure 7.1: (a) Half-wave rectifier, (b) envelope detector.

## 7.1 Envelope detector

With a sinusoidal modulating signal $v_m(t) = V_m \cos(2\pi f_m t)$ and a carrier signal of $V_c \cos(2\pi f_{IF} t)$, the AM (amplitude modulated) signal at the output of the second IF amplifier is

$$v_{IF}(t) = V_c \left(1 + \frac{V_m}{V_c} \cos(2\pi f_m t)\right) \cos(2\pi f_{IF} t) \tag{7.1}$$

The waveform of this signal is plotted in Fig. 7.2(a) for $V_c = 1$, $V_m = 0.7$, and $f_{IF}$=20 kHz, $f_m = 1$ kHz. The modulation index (see Eq. 1.6 at page 7) of this signal is $m = V_m/V_c$=0.7.

AM *demodulation* is the act of separating the information signal $v_m(t)$ from its carrier $V_c \cos(2\pi f_{IF} t)$. The simplest and oldest method of doing this is called *envelope detection*. In an envelope detector, the signal is half-wave rectified and then low pass filtered. We employ the rectification property of diodes in envelope detection. A half-wave rectifier is shown in Fig. 7.1(a). It is composed of an

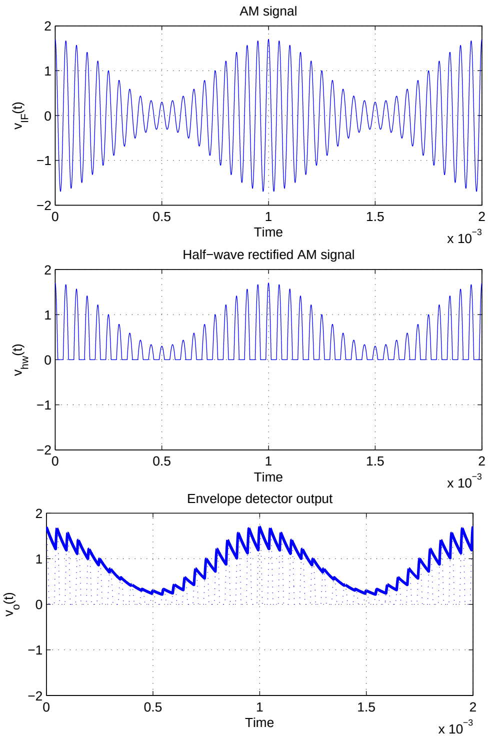



Figure 7.2: AM Waveform, $v_{IF}(t)$ (upper) for $f_{IF}$=20 kHz, $f_m = 1$ kHz, half-wave rectified AM waveform, $v_{hw}(t)$ (middle) and envelope detector output, $v_o(t)$ (lower).

ideal diode and a resistor.

The half-wave rectified AM signal, $v_{hw}(t)$, using this rectifier is shown in Fig. 7.2(b). The diode rectifies the AM signal, and only positive half cycles appear across the resistor. The mechanism is similar to the power rectification problem in Chapter 2, except in this case, the amplitude varies with respect to time.

Fig. 7.1(b) shows the schematic of an envelope detector where a capacitor is added in parallel with the resistor. The resulting signal output, $v_o(t)$ is depicted in Fig. 7.2(c) where the envelope detector output looks like the original sine wave of $v_m(t)$ with a DC shift. $v_o(t)$ follows the envelope with some ripple as in the case of power rectification.

Here, we cannot increase the capacitance, $C$, hence the time constant $RC$, indefinitely to reduce the ripple. The time constant, $RC$, must be chosen such that the capacitor can discharge fast enough and its voltage can follow the maximum negative slope of the envelope, $-V_m 2\pi f_m$. Since the slope of decaying exponential $V_c \exp(-t/RC)$ at $t = 0$ is given by $-V_c/RC$, we should have $V_c/RC > V_m 2\pi f_m$, or

$$RC < \frac{V_c}{2\pi f_m V_m} \tag{7.2}$$

Otherwise, the detected envelope signal suffers from what is known as *failure to follow* distortion or *diagonal* distortion. This upper limit on $RC$ causes some ripple on the detected waveform, particularly during the up-sloping phases of the envelope, as seen in Fig. 7.2(c).

We used a very low carrier frequency ($f_{IF}$ = 20 kHz) to demonstrate the function of the envelope detector and to exaggerate the ripple. In a real case, the carrier frequency is much higher. Fig. 7.3 shows $v_{hw}(t)$ and $v_o(t)$ for $f_{IF}$ = 100 kHz. In this case, the ripple is smaller, and the envelope detector output is approximately the same as the original sine wave of $v_m(t)$ except for the DC shift. The ripple is negligible with a carrier frequency in the MHz range.

The highest ripple occurs when the AM signal amplitude is maximum ($V_c + V_m$). If $f_m \ll f_{IF}$ and $1/RC \ll f_{IF}$, the peak-to-peak ripple in the output can be estimated from the peak-to-peak ripple formula of Eq. 4.7 for the half-wave rectifier:

$$V_r = (V_c + V_m)\frac{1}{RCf_{IF}} \tag{7.3}$$

where the voltage drop across the envelope detector diode is ignored.

Note that the DC shift amount (or the average value) at the envelope detector output is the same as the amplitude of the carrier, $V_c$. The DC shift voltage can be used to determine the carrier amplitude of the AM signal.

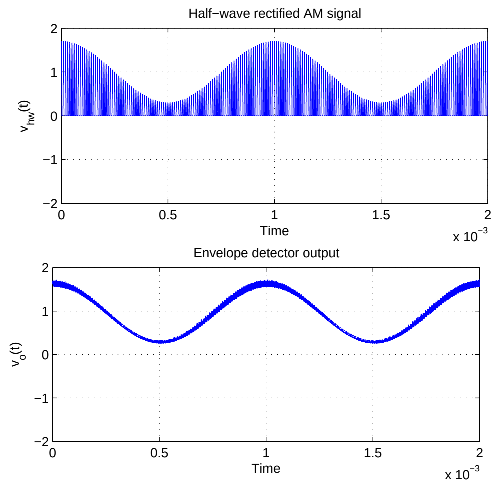



Figure 7.3: Half-wave rectified AM waveform, $v_{hw}(t)$, (upper) for $f_{IF}$=100 kHz, $f_m$ = 1 kHz and envelope detector output, $v_o(t)$ (lower).

### 7.1.1 Real diodes in envelope detectors

Real diodes have a threshold voltage $V_o$, which has an adverse effect on the detected signal. Assume that we replace the ideal diode in Fig. 7.1 with a real diode. Let us model the diode by a piecewise linear model of Fig. 4.3 on page 136. The operation of the half-wave rectifier of Fig. 7.1 with a real diode is shown in Fig. 7.4(a).

The diode conducts only after the voltage across its terminals exceeds the threshold voltage $V_o$. $V_o$ can be taken approximately as 0.6 V, in which case the detected envelope of a $v_{IF}(t)$ of, for example, 2 $V_{pp}$ amplitude is severely distorted, as illustrated in Fig. 7.4(b).

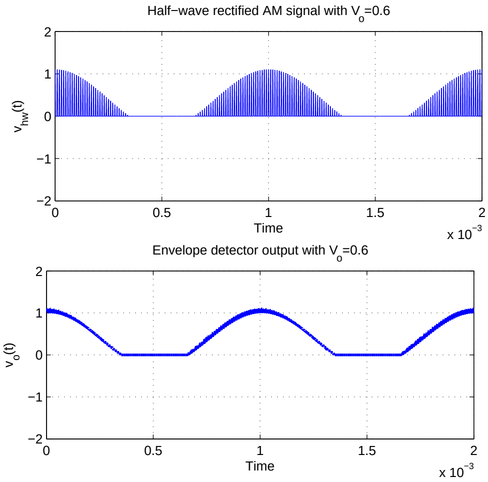



Figure 7.4: Half-wave rectified AM waveform, $v_{hw}(t)$, (upper) for $f_{IF}$=100 kHz, $f_m$ = 1 kHz for a real diode with $V_o$=0.6 V and the corresponding envelope detector output, $v_o(t)$ (lower).

We overcome this problem by passing a small current through the diode at all times. Consider the circuit in Fig. 7.5(a) (where the diode is detecting the positive envelope of AM signal). $C_c$ and $L$ form a resonant circuit at the carrier frequency. $C_c$ also prevents DC current from flowing through the input AC source. The DC current source $I_{dc}$ sets the average diode current to $I_{dc}$ at all times, assuring that the diode is on even when the carrier amplitude approaches zero. Hence, the output voltage $v_o(t)$ can follow the positive envelope for all input voltage levels.

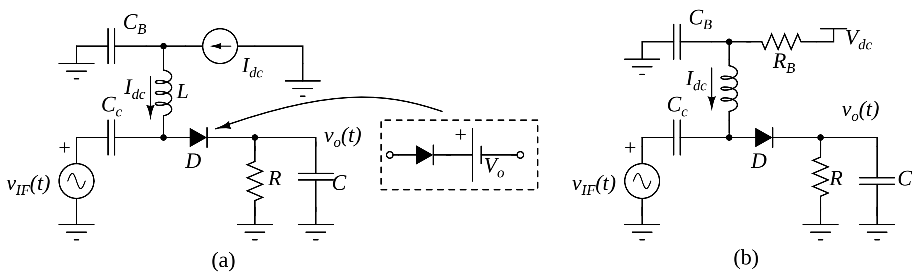



Figure 7.5: Envelope detector using a real diode with biasing.

This preconditioning of a diode by forcing a DC current to flow through it is called *biasing*. We implement this solution using the circuit given in Fig. 7.5(b). The DC current source is implemented by a large resistor $R_B$ connected to a positive voltage supply, $V_{dc}$. The value of $I_{dc}$ is $(V_{dc} - V_o)/(R_B + R)$.

We employ a resonant circuit at the carrier frequency composed of $C_c$ and $L$. $C_B$ is a very large capacitor providing an AC ground. The resonant circuit has a band-pass-filter action eliminating frequencies other than the carrier frequency. It also boosts the signal level by the $Q$ factor of the resonant circuit.

The output time constant $RC$ is chosen sufficiently small in such a way to follow the envelope variation at the highest AM modulation frequency.

## 7.2 Automatic gain control

$v_{IF}(t)$ is a scaled version of the amplitude of the RF signal delivered by the antenna. The received signal amplitude may vary over a large range depending on how far the transmitter is. The change in amplitude can be compensated by changing the receiver's gain. If the received signal is small, the gain must be set high. On the other hand, if the transmitter is very close by, the received signal can be very high. In this case, there may even be saturation in the receiver amplifiers due to the very large input signal. A deep saturation may cause the envelope information in the AM signal to disappear and the envelope detector output to become zero. Therefore, it is desirable to reduce the gain of the receiver in such a way that the second IF amplifier is not saturated.

Wireless communication channels have peculiar properties, particularly at HF band. One such effect is called *fading*. The received signal amplitude occasionally changes in time rather slowly due to propagation mechanisms in the HF band. Two signals may arrive at the receiver antenna using different paths. The path difference of these two signals may create either constructive or destructive interference in the received signal. This variation usually has a period of more than a few tens of seconds. When listening to the receiver output under fading, one feels that the voice slowly fades out and then comes back. It is, of course, possible to compensate for this effect by changing the gain manually. However, this can be disturbing.

Therefore, it is desirable to have an *automatic gain control* where the receiver detects the input signal's level and automatically adjusts its gain to supply a stable output signal amplitude.

Recall that the envelope detector output is the same as the instantaneous amplitude of the carrier's envelope. The average carrier amplitude can be found using a low-pass-filter to filter the envelope detector output. The cutoff frequency of the low-pass-filter should be low enough to eliminate the modulation signal altogether. For example, a corner frequency of less than 1 Hz may be suitable. This low-pass-filtered and slowly varying signal is proportional to the incoming signal amplitude. This signal can be used to vary the gain of the receiver. If the average carrier amplitude is large, we should decrease the gain. On the other hand, if it is small, the gain must be increased. For this purpose, we need a circuit element whose resistance changes in relation to a voltage.

### 7.2.1 PIN diode

A PIN diode is a special semiconductor diode. PIN refers to the semiconductor structure of the diode: These diodes have three layers of semiconductor material, p-type, intrinsic (undoped), and n-type, as opposed to commonly used two-layers, p-type and n-type, in p-n junction diodes. Because of its structure, it is a very slow acting diode. Once the diode is turned on, it does not turn off quickly. Conversely, it does not turn on easily if the diode is off. Although it is a very slow diode, it has a very small junction capacitance making it suitable for high frequencies. (p-n junction rectifier diodes like 1N4007 are also very slow, but they have a very large junction capacitance.)

PIN diode acts like a regular diode for low-frequency signals (for signals lower than about 100 kHz) with highly nonlinear characteristics. At high frequencies (higher than about 5 MHz), it acts like a linear resistor even for large signal amplitudes. This high-frequency resistance is inversely proportional to the DC or low-frequency current passing through the diode. If the DC current is not present, the PIN diode acts as an open-circuit at high frequencies even if the instantaneous high-frequency voltage across the diode is a large positive voltage. A short burst of high-frequency positive voltage is not enough to turn on the diode.

On the other hand, the PIN diode acts as a resistor with a small value (a few ohms), when there is a positive DC current through it. This is true even if the instantaneous high-frequency voltage may be several volts negative.

With these interesting properties, a PIN diode can be used as the element we need for the automatic gain control mechanism.

### 7.2.2 Automatic gain control using a PIN diode

Consider the automatic gain control (AGC) circuit shown in Fig. 7.6. The envelope detector output $v_o(t)$ goes to a low-pass-filter (LPF) composed of $R_{73}$ and $C_{71}$. The cutoff frequency of this LPF is smaller than the lowest AM modulation frequency. This means that the LPF's output is proportional to the average carrier amplitude.

LPF's output is fed to the positive input of an OPAMP for comparison with a set point voltage at the negative input of the OPAMP. A large resistance is placed in the feedback path of the OPAMP, causing it to act like a high-gain amplifier. The OPAMP output determines the current for a shunt PIN diode placed between the two IF amplifier stages. Note that a zener diode ($D_{72}$) is placed in series with a resistor ($R_{75}$) between the OPAMP output and the PIN diode. The OPAMP output voltage should be larger than the zener voltage to generate any current in the PIN diode.

If the average carrier amplitude is higher than the set point, the OPAMP output voltage increases, causing an increase in the PIN diode current. With an increased current, the PIN diode's RF resistance becomes smaller, reducing the gain of the IF amplifier. This action continues until the two input pins of the OPAMP have the same voltage, i.e., the average carrier amplitude equals the set point voltage.

Conversely, if the average carrier amplitude is lower than the set point, the OPAMP output becomes zero, providing no current to the PIN diode. In this case, the IF amplifiers have the highest possible gain.

In summary, the AGC circuit provides the highest IF gain if the average carrier amplitude is less than the set point. For higher carrier amplitudes, the IF gain is adjusted to give a constant IF amplifier output voltage.

This AGC network is an example of a negative feedback system. The feedback system adjusts the gain of the IF amplifier in such a way to keep the

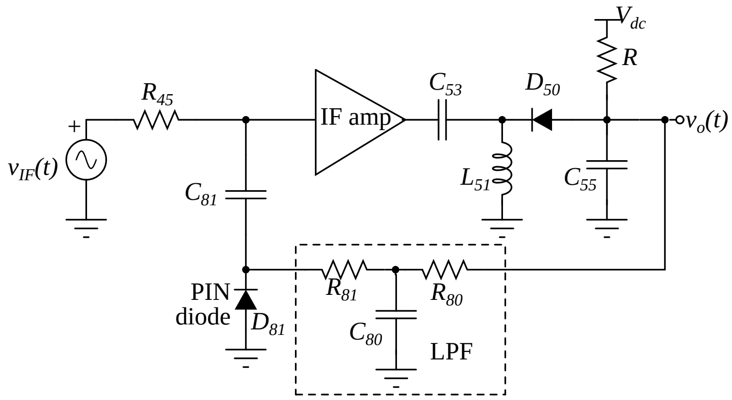



Figure 7.6: Automatic gain control circuit.

average IF amplifier output voltage nearly at the same level. As a consequence, the saturation of the IF amplifier is also avoided.

## 7.3 Signal presence indicator

In a receiver circuit, it is desirable to have an indicator showing the presence of a signal at the input frequency. The existence of the PIN diode current is a good sign for this purpose. When the OPAMP output is larger than the zener voltage, PIN diode current is present. So the same OPAMP output can be used to turn on an LED. A resistor, $R_{76}$, in series with the LED limits the current. So, the LED turnd on as soon as the AGC circuit begins to limit the IF amplifier gain.

- ⧫ TRC-11 has one signal diode in the amplitude demodulator, one PIN diode to change the gain of the IF amplifier, and LED as the signal presence indicator, and one LED as power indicator.

## 7.4 Problems

1. The AM signal $2[1 + 0.7\cos(2\pi 10^3 t)]\cos(2\pi 28 \cdot 10^6)$ is fed to an envelope detector as in Fig. 7.1(b) with $R$=1 k$\Omega$. Find the maximum value of capacitor, to avoid failure to follow distortion.

2. If $RC = 1/(2\pi f_m)$ in the circuit of Fig. 7.1(b), estimate the maximum ripple on the detected envelope, if the modulation index is $m = 1$. Assume that $f_m \ll f_{IF}$. What is the minimum ripple? Is there any failure-to-follow distortion?

3. Determine and sketch the waveform at the output for the circuits given in Fig. 7.7. Assume that the diode is ideal.

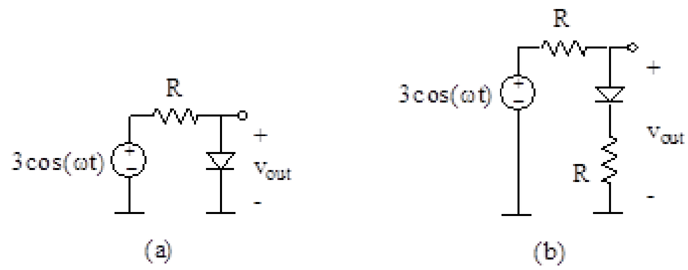



Figure 7.7: Circuits for Problem 3

4. In Fig. 7.8, $v(t)$ is $0.5\cos(\omega t)$ for both circuits. Assume that the diode can be modelled by the approximate model of Fig. 4.3(c) and (d) on page 136, with $V_0$=0.7 V. What is $v_{out}(t)$ for both circuits, if $I = 0$? Find the minimum value of $I$ for which there is an undistorted replica of $v(t)$ at the output. For both circuits, 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 Thévenin equivalent circuit, comprising both sources, across the detector circuit)

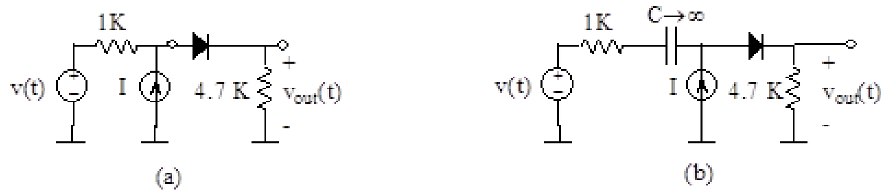



Figure 7.8: Circuits for Problem 4

5. Consider the 28 MHz amplifier shown in Fig. 7.9 built using an OPAMP. RF resistance of the PIN diode can be varied between 10 Ω to 1000 Ω, while the DC current source $I_{DC}$ is varied between 10 mA to 0.1 mA.

Find the gain, $|V_{out}/V_{in}|$, of the amplifier at 28 MHz, for $I_{DC}$=10 mA and 0.1 mA. Assume that the reactance of 1 nF capacitor is negligible at 28 MHz.

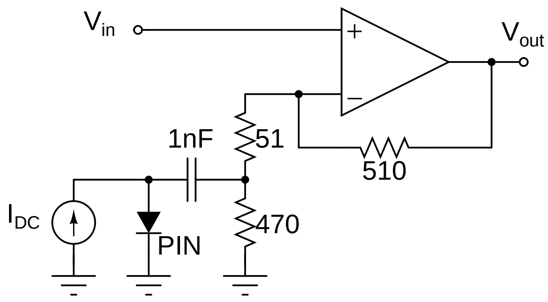



Figure 7.9: RF amplifier circuit for problem 5

6. Find the value of the resistor in the LED drive circuit of Fig. 4.17(c) for a voltage source of $V_s$=5 V, when a white-light power LED with a current of 1.5 A is to be connected. With 1.5 A current, the voltage drop across the LED is 3.1 V. If the light conversion efficiency of the LED itself is 100 lumens/W, what is the light conversion efficiency of the LED with the series resistor?

# Chapter 8

# FREQUENCY CONVERSION

## 8.1 Mixers

As discussed in Chapter 1, mixers are nonlinear devices with three ports to perform the multiplication process. They can be built from diodes, transistors or other nonlinear devices. The concept of mixing is easily understood with a switch mixer.

### 8.1.1 Switch mixer

Consider the mixer shown in Fig. 8.1, where we have sinusoidal input signals for both RF and LO inputs of the mixer.

$$v_{RF}(t) = A\sin(\omega_{RF}t) \quad \text{and} \quad v_{LO}(t) = \sin(\omega_{IF}t) \tag{8.1}$$

The switch is opened and closed at the frequency of $v_{LO}$. The switch is closed if $v_{LO}(t) \geq 0$, and it is open if $v_{LO}(t) < 0$. Therefore, we can write the

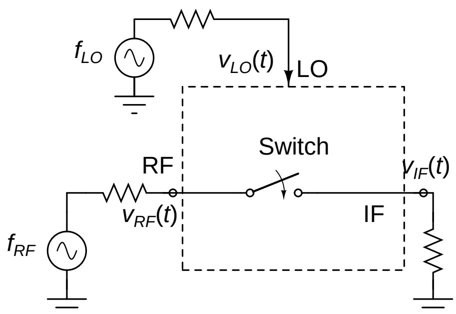



Figure 8.1: A switch mixer.

output signal, $v_{IF}(t)$, in terms of the input signal, $v_{RF}(t)$, and the switching function, $s(t)$, as

$$v_{IF}(t) = v_{RF}(t)s(t) \tag{8.2}$$

where the switching function $s(t)$ is a unity-amplitude square-wave given by

$$s(t) = \begin{cases} 1 & \text{if} \;\; v_{LO}(t) \geq 0 \\ 0 & \text{if} \;\; v_{LO}(t) < 0 \end{cases} \tag{8.3}$$

In Chapter 1, we showed that a square wave can be written as the sum of sinusoids. From Eq. 1.4 in page 5, we write

$$s(t) = \frac{1}{2} + \frac{2}{\pi}\sin(\omega_{LO}t) + \frac{2}{3\pi}\sin(3\omega_{LO}t) + \frac{2}{5\pi}\sin(5\omega_{LO}t) + \ldots \tag{8.4}$$

Hence Eq. 8.2 can be written as

$$v_{IF}(t) = A\sin(\omega_{RF}t)\left(\frac{1}{2} + \frac{2}{\pi}\sin(\omega_{LO}t) + \frac{2}{3\pi}\sin(3\omega_{LO}t) + \ldots\right) \tag{8.5}$$

Using trigonometric identities, we can arrive at

$$\begin{aligned} v_{IF}(t) = {} & \frac{A}{2}\sin(\omega_{RF}t) + \frac{A}{\pi}\left[\cos(\omega_{LO} - \omega_{RF})t - \cos(\omega_{LO} + \omega_{RF})t\right] \\ & + \frac{A}{3\pi}\left[\cos(3\omega_{LO} - \omega_{RF})t - \cos(3\omega_{LO} + \omega_{RF})t\right] + \ldots \end{aligned} \tag{8.6}$$

The IF output of the mixer contains sinusoids at many different frequencies, $\omega_{RF}$, $\omega_{LO} - \omega_{RF}$, $\omega_{LO} + \omega_{RF}$, $3\omega_{LO} - \omega_{RF}$, $3\omega_{LO} + \omega_{RF}$, etc. albeit at reduced amplitude.

### 8.1.2 Conversion gain of a mixer

An important parameter in the evaluation of mixers is the *conversion gain.* The conversion gain is defined as

$$G_c = \frac{P_o}{P_i} \tag{8.7}$$

where $P_o$ is the total power delivered to a matched load at IF output and $P_i$ is the total available input power at the RF input. This expression is similar to the gain of an amplifier, except here the input and output frequencies are different.

While passive mixers built from diodes may have a conversion gain less than unity, many mixers made in the form of integrated circuits have a conversion gain larger than one.

- ♦ TRC-11 utilizes one mixer in the integrated-circuit form, with a conversion gain larger than one.

## 8.2 Amplitude modulator

Suppose that a tuned RF amplifier is used to amplify an RF signal. Since the amplifier is tuned, no distortion in the output occurs even when the input signal level is too high. With a large input signal, the peak value of the output RF signal is determined by the supply voltage of the tuned amplifier rather than the input signal amplitude. If the supply voltage of that RF amplifier is varied,

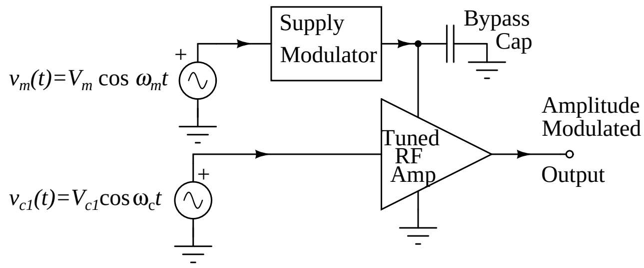



Figure 8.2: Amplitude modulator circuit. The input signal of the tuned RF amplifier should be so large that the output magnitude is determined by the supply voltage.

the amplified signal follows the variation (see Fig. 8.2. This principle is used in TRC-11 to obtain an amplitude modulator.

The supply voltage modulation is achieved using a PNP BJT circuit, the input of which is controlled by the modulation signal at audio frequency. The collector of the BJT is connected to the supply voltage of an RF amplifier through an RF bypass capacitor. The bypass capacitor selected such that it acts like a short circuit at RF frequency but it is an open circuit at the audio frequencies. Hence the supply voltage of the tuned RF amplifier varies exactly like the modulation signal.

## 8.3 Oscillators

An oscillator is a sinusoidal signal generator. Frequency is determined by a resonant circuit inside the oscillator. The resonator can be built, for example, from $LC$ circuits or quartz crystals.

Every resonator has some loss. If this loss can be compensated by an active circuit, a continuous sinusoidal signal can be obtained. Depending on the oscillator structure, some oscillators provide a square-wave output, rather than the sinusoidal signal.

It is possible to obtain an oscillator by using a positive feedback between the output and input of an amplifier using a resonator. The resonator determines the frequency of oscillation while the amplifier provides the continuity of the oscillation.

The quality of an oscillator is determined by the purity and stability of the output signal. Oscillators using high-$Q$ resonators usually have high quality output signals. Oscillators using $LC$ circuits as their resonators typically produce low quality signals, since $Q$ of electrical circuits are limited. Oscillators using quartz crystals are called crystal oscillators and they provide a well-defined and stable frequency output signal. Many laboratory signal generators use crystal oscillators as their reference frequency. If higher stability is desired, crystals may be placed in temperature-controlled ovens to get frequencies that are more precise.

### 8.3.1 Oscillator concept

We discussed feedback in Section 3.9.1 and noted that we always use negative feedback for amplification. In oscillators, we need *positive feedback*.

Consider the parallel $RLC$ circuit given in Fig. 8.3(a), with a current source connected in parallel. We assume that the current source $i(t)$ contains many

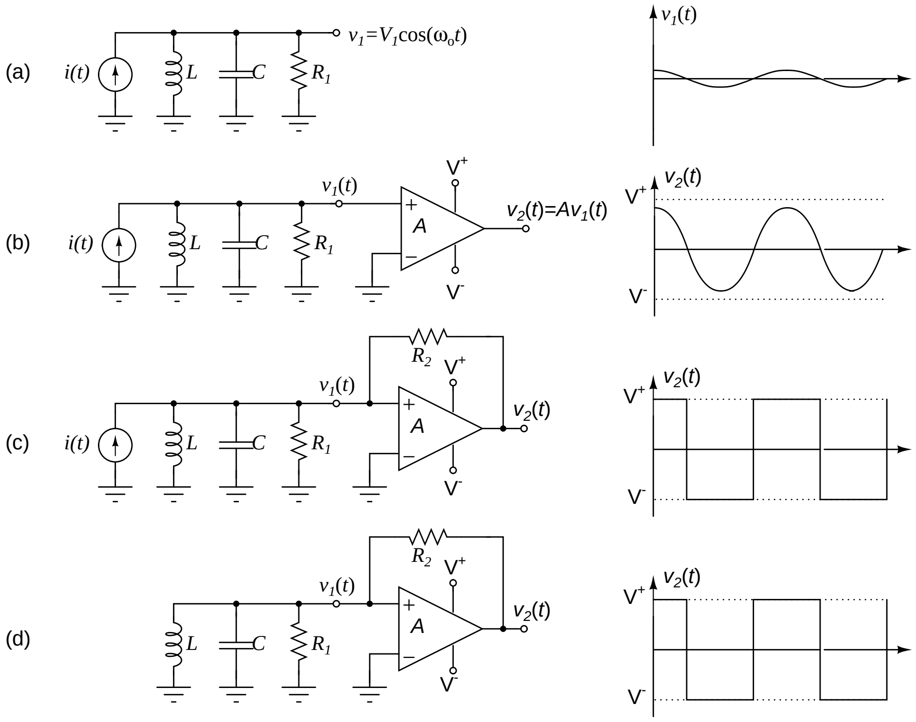



Figure 8.3: Oscillator concept: (a) $RLC$ circuit driven by a wide-band current source, (b) amplification, (c) positive feedback, (d) current source removed.

sinusoidal components at all frequencies and its amplitude is very small. The *noise* in electronics is such a signal. Only the current component in the vicinity of $\omega_o = 1/\sqrt{LC}$ 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

$$v_1(t) = V_1 \cos(\omega_o t) \tag{8.8}$$

where $V_1$ is very small. If an OPAMP with a gain of $A$ is connected to this node, $v_1(t)$ is amplified by the factor $A$ as shown in Fig. 8.3(b).

The supply voltage levels limit the output voltage swing of OPAMPs. When the amplified signal amplitude $AV_1$ approaches to positive supply voltage, $V^+$, or to the negative supply voltage, $V^- = -V^+$, the output waveform gets distorted. We obtain a clipped waveform. The OPAMP is said to be saturated (recall Eq. 3.48 on page 105). If the 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 Fig. 8.3(c), $v_1(t)$ is modified. Initially, an additive sample from output increases $v_1(t)$ to

$$v_1(t) = V_1 \cos(\omega_o t) + \frac{R_1}{R_1 + R_2} A V_1 \cos(\omega_o t) \tag{8.9}$$

The additive component has an amplitude of $R_1/(R_1 + R_2)(AV_1)$ which is much larger than the signal directly created by $i(t)$. This component alone drives the amplifier deep into saturation, since $AR_1/(R_1 + R_2)(AV_1) \gg V^+$. We immediately have a square wave at the output as $v_2(t)$. The peak-to-peak amplitude of this wave is $2V^+$. This waveform is also shown in Fig. 8.3(c).

When we have a square wave at the output, the feedback signal is also a square wave. However, the tank circuit picks the fundamental component of this square wave, producing a $v_1(t)$ as

$$v_1(t) = V_1 \cos(\omega_o t) + \frac{R_1}{R_1 + R_2} b_1 V^+ \cos(\omega_o t) \tag{8.10}$$

From Eq. 1.4 of page 5 we know that $b_1 = 4/\pi$ 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 Fig. 8.3(d).

Practically, we never include an explicit current source unit to start the oscillation, because it is not necessary. There is always *noise* in electronic circuits, creating the current needed for the start of the oscillation. The current source is always there.

It is possible to deduce from the above discussion that if

$$\frac{R_1}{R_1 + R_2} A = 1 \tag{8.11}$$

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 called the *Barkhausen oscillation criterion.*

In the circuit of Fig. 8.3, we have the amplitude limiting mechanism of saturating amplifier. Since the fundamental component of the saturated output is $b_1 V^+$, the peak-to-peak amplitude, $V_{1pp}$, of sinusoidal signal $v_1(t)$ is always

$$V_{1pp} = 2 \frac{R_1}{R_1 + R_2} b_1 V^+ = \frac{8}{\pi} \frac{R_1}{R_1 + R_2} V^+ \tag{8.12}$$

As long as the gain is large enough to keep the amplifier in saturation with this input, the oscillation is sustained. The larger gain of the amplifier helps oscillations to start easily. To make sure that the oscillation starts, the feedback ratio $R_1/(R_1 + R_2)$ and the gain $A$ must be such that

$$\frac{R_1}{R_1 + R_2} A > 1 \tag{8.13}$$

We note that it is possible to replace $LC$ circuit of the oscillator with a quartz crystal operating at its parallel resonance frequency (see Eq. 6.44 in page 242). In this case, we get a very stable oscillator since the resonance frequency is determined by the quartz crystal.

**Example 1**

Suppose we would like to design an oscillator as in Fig. 8.3(d) at $f_o$=12 MHz using LM7171, an OPAMP with a gain-bandwidth product of $GBW$=200 MHz. We have supply voltages of $V^+$=15 V and $V^- = -15$ V.

The gain $A$ of the OPAMP at 12 MHz is $GBW/f_o$=16.6. Using Eq. 8.13, we write

$$\frac{R_1}{R_1+R_2} > \frac{1}{16.6} = 0.06$$

LM7171 data sheet recommends a feedback resistor $R_2$=510 Ω. Therefore, we need to choose $R_1 > 32.5$ Ω. Let us choose $R_1$=330 Ω (comfortably larger than the limit). To get a $Q$=10, we choose

$$Q = \frac{R_1 \parallel R_2}{\omega_o L} = 10 \text{ or } L = \frac{R_1 \parallel R_2}{2\pi f_o 10} = \frac{200}{2\pi \cdot 12 \cdot 10^6 \cdot 10} = 265 \text{ nH}$$

From the resonance condition at 12 MHz, we find the capacitor value as

$$C = \frac{25330}{12^2 \cdot 0.265} = 664 \text{ pF}$$

From Eq. 8.12, the peak-to-peak signal amplitude, $V_{1pp}$ is given by

$$V_{1pp} = \frac{8}{\pi}\frac{R_1}{R_1+R_2}V^+ = \frac{8}{\pi}\frac{330}{330+510}15 = 15$$

In the real circuit, $V_{1pp}$ will be less than 15 V$_{pp}$, since the saturation voltages of the LM7171 are about ±13 V rather than ±15 V.

---

### 8.3.2 Negative resistance

Negative resistance concept is another way of analyzing oscillator circuits. Consider the OPAMP circuit shown in Fig. 8.4(a). Let us find the input impedance $Z_{in}$ assuming that the OPAMP is not saturated. Hence we have $V_1 = V_2 = V_{in}$. From the voltage divider relation and from the node equation at $V_1$, we write

$$V_2 = \frac{R_1}{R_1+R_2}V_o \quad \text{and} \quad I_{in} = \frac{V_1 - V_o}{R_3} \tag{8.14}$$

Rearranging and combining the equations, we find

$$V_o = \left(1 + \frac{R_2}{R_1}\right)V_2 \quad \text{and} \quad I_{in} = -V_{in}\frac{R_2}{R_1 R_3} \tag{8.15}$$

Therefore, we find the input impedance $Z_{in}$ as

$$Z_{in} = \frac{V_{in}}{I_{in}} = -\frac{R_1 R_3}{R_2} \tag{8.16}$$

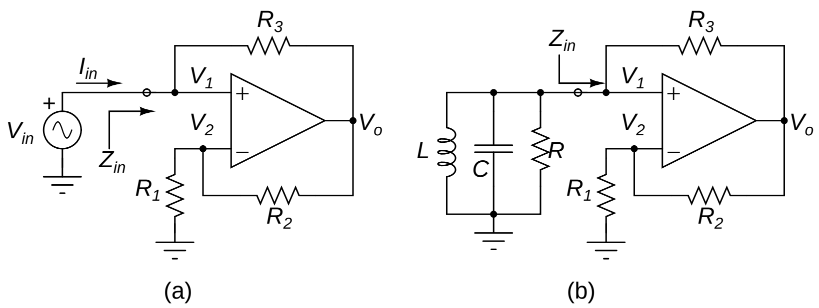



Figure 8.4: (a) OPAMP circuit with a negative input resistance, (b) an oscillator built with the negative resistance.

a negative quantity. This negative resistance can be used to compensate the loss of a resonator to form an oscillator. Fig. 8.4(b) shows an $RLC$ circuit resonator connected to the negative resistance input of the OPAMP. If the negative resistance is chosen to be equal to the parallel resistance $R$ of $RLC$ circuit,

$$R = -Z_{in} = \frac{R_1 R_3}{R_2} \tag{8.17}$$

the parallel combination of $R$ and $-R$ will yield an open-circuit:

$$R \parallel Z_{in} = R \parallel (-R) = \frac{-R^2}{R - R} \Rightarrow \infty \tag{8.18}$$

In this case, an oscillation at the resonance frequency of $LC$ can be maintained indefinitely. In practice, it is difficult to satisfy the precise equality of a negative resistance to a positive resistance. To guarantee an oscillation, we choose the values such that

$$\frac{R_1 R_3}{R_2} < R \tag{8.19}$$

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.

### 8.3.3 Colpitts Oscillator

Consider the circuit shown in Fig. 8.5(a) where a *transconductance amplifier* is shown in the dashed box. As shown in page 104, a transconductance amplifier has a voltage input and current source output. The amount of the current in the current source is determined by the input voltage. The multiplier factor, $g_m$, is known as the *transconductance*. Two impedances, $Z_1$ and $Z_2$ are connected as shown. Let us find the input impedance $Z_{in}$:

$$Z_{in} = \frac{V_b}{I_b} \tag{8.20}$$

Using KVL, we write

$$V_b = I_b Z_1 + V_e \tag{8.21}$$

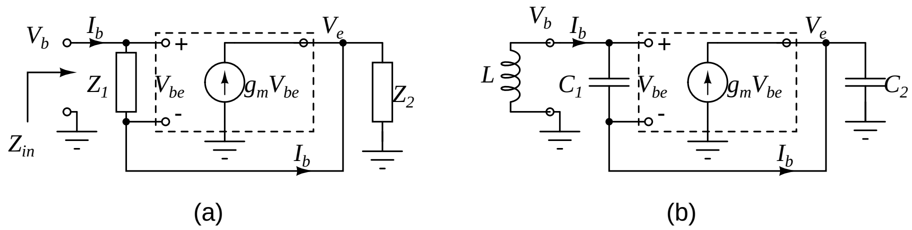



Figure 8.5: (a) A transimpedance amplifier with the feedback impedance $Z_1$ and the load impedance $Z_2$. , (b) Colpitts oscillator.

The voltage $V_e$ can be found using KCL as

$$V_e = (g_m V_{be} + I_b)Z_2 = (g_m(V_b - V_e) + I_b)Z_2 \tag{8.22}$$

Solving for $V_e$, we find

$$V_e = \frac{g_m V_b + I_b}{1 + g_m Z_2} Z_2 \tag{8.23}$$

Substituting this value in Eq. 8.21 and solving for $V_b$, we find

$$V_b = (g_m Z_1 Z_2 + Z_1 + Z_2)\, I_b \tag{8.24}$$

Hence the input impedance is found as

$$Z_{in} = \frac{V_b}{I_b} = g_m Z_1 Z_2 + Z_1 + Z_2 \tag{8.25}$$

Now, suppose that the feedback element $Z_1$ and the load element $Z_2$ are capacitors, $C_1$ and $C_2$. For this particular case, $Z_1 = 1/(j\omega C_1)$ and $Z_2 = 1/(j\omega C_2)$. For $Z_{in}$, we get

$$Z_{in} = \frac{V_b}{I_b} = -\frac{g_m}{\omega^2 C_1 C_2} + \frac{1}{j\omega}\frac{C_1 + C_2}{C_1 C_2} \tag{8.26}$$

Note that the real part is negative and the imaginary part is due to the series combination of the capacitors $C_1$ and $C_2$. The negative resistance can be utilized to obtain an oscillator.

When the input of the circuit is connected to an inductance as shown in Fig. 8.5(b), we get a *Colpitts oscillator* invented by American engineer Edwin Colpitts (1872–1949) in 1918. The negative resistance compensates the loss of the inductor and the circuit produces a sustained oscillation. The oscillation frequency is given by

$$f_o = \frac{1}{2\pi}\sqrt{\frac{C_1 + C_2}{L C_1 C_2}} \tag{8.27}$$

If a quartz crystal is connected instead of the inductor, the resulting circuit is known as *Colpitts crystal oscillator*. In that case, the oscillation frequency is determined essentially by the series resonance frequency of the crystal.

**Example 2**

Consider a Colpitts oscillator (as in Fig. 8.5(b)) to operate at 28 MHz. If $C_1$=33 pF, $C_2$=33 pF, and $g_m$=0.005 S, find the value and minimum $Q$ of the inductor.

From Eq. 8.26, we find $Z_{in} = -148 - j344$. Hence we must have $L = 344/(2\pi 28 \cdot 10^6) = 1.96\ \mu$H. If the series resistance of $L$ satisfies $r \leq 148\ \Omega$, the oscillation will be maintained. Hence we should have $Q \geq \omega L/r = 2.32$.

---

### 8.3.4 Pierce Oscillator

A commonly used oscillator circuit depicted in Fig. 8.6(a) is known as Pierce oscillator. It was invented by George W. Pierce (1872–1956) in 1923. A digital inverter is used as the amplifying element of the oscillator. The feedback resistance $R_f$ assures that the inverter acts like a high gain inverting amplifier. The capacitors $C_1$ and $C_2$ and the quartz crystal form the feedback circuit, providing the extra phase shift necessary for oscillation [14]. The total gain of the oscillation loop should be equal to unity, with zero (or $-2\pi$) phase shift. Therefore, the oscillation condition can be written as

$$\frac{v_i}{v_o}(-A) = 1 \tag{8.28}$$

Fig. 8.6(b) shows the equivalent circuits of the quartz crystal and the amplifier

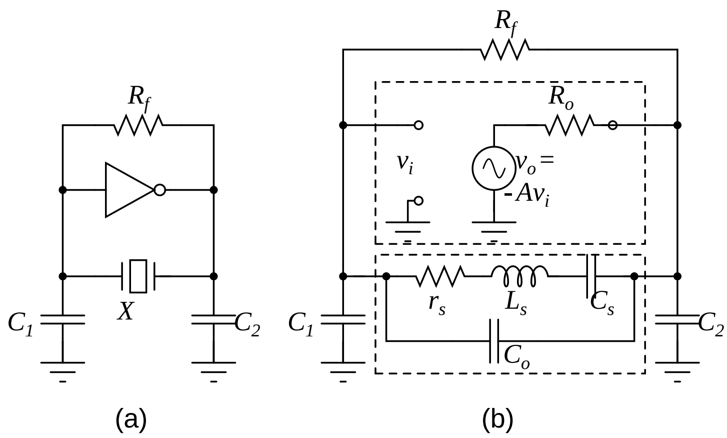



Figure 8.6: (a) Pierce oscillator using a digital inverter with crystal feedback, (b) its equivalent circuit.

with an output resistor of $R_o$. The amplifier introduces a $-\pi$ phase shift. To find the oscillation frequency, we need the find the frequency at which the feedback circuit provides a phase shift of $-\pi$. For this purpose, we should find the transfer function, $v_i/v_o$ of the feedback circuit from the output to input using phasors.

**Example 3**

Consider the Pierce oscillator built using a digital CMOS inverter integrated circuit, 74HC04, and a 16 MHz quartz crystal. The 14-pin integrated circuit has six inverters inside. The oscillator uses only one of the inverters, which has a propagation delay of 6 ns with a supply voltage of 6 V. We have $C_1$=$C_2$=12 pF, $R_f$=1 MΩ and $R_o$=100 Ω. The 16 MHz quartz crystal has parameters $L_s$=15 mH, $r_s$ = 15 Ω, $C_s$=6.6 fF, $C_o$=5 pF.

The amplitude and phase of the transfer function, $v_i/v_o$, of the feedback circuit is plotted in Fig. 8.7. The series and parallel resonant frequencies are about 15.999 MHz and 16.006 MHz, respectively. 6 ns delay of the inverter corresponds to a phase shift of (6 ns)(16 MHz)(360$^o$)=34.6$^o$. Hence a phase shift of $-180^o - 34.6^o = -214.6^o$ is already provided by the inverter. We need to find the frequency where the feedback network has $-360^o - (-214.6^o) = -145.4^o = -2.54$ rad phase shift. We see that this feedback circuit provides a phase shift of $-2.54$ rad between $f_s$ and $f_p$ near 15.9992 MHz, where the amplitude is 12.2 dB. Since the gain of the inverting amplifier is definitely larger than $-12.2$ dB, the oscillation condition is easily satisfied at this frequency. We note that it is possible to shift the oscillation frequency slightly by changing the values of $C_1$ and $C_2$.

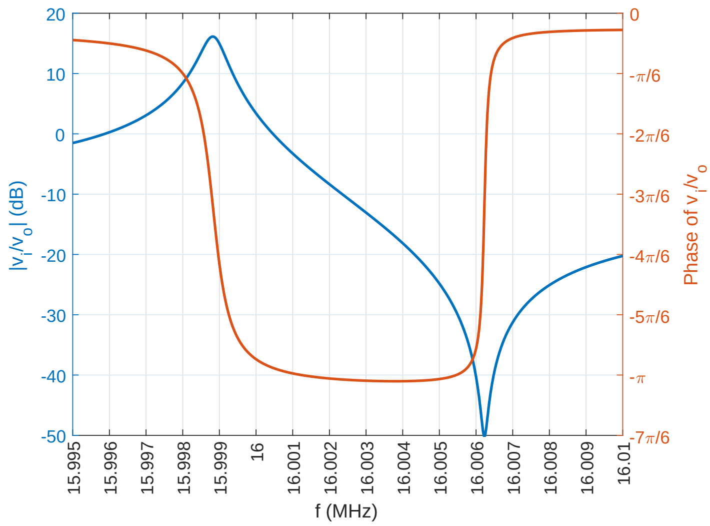



Figure 8.7: The amplitude and phase of the transfer function, $v_i/v_o$, of the feedback circuit of Example 3 Pierce oscillator.

---

♦ There are two oscillators in TRC-11. The first one is the local oscillator that generates a signal at 12.00 MHz, which is a Colpitts crystal oscillator as shown in Fig. 8.8. Here, SA602A is an integrated circuit acting as a transconductance amplifier necessary for oscillation. We have the capacitors of Colpitts oscillator as $C_1 = 39$ pF and $C_2 = 18$ pF.

The second oscillator is also a Colpitts crystal oscillator at 27 MHz generating the transmitter frequency of transmitter with $C_1$=47 pF and $C_2$=10 p.

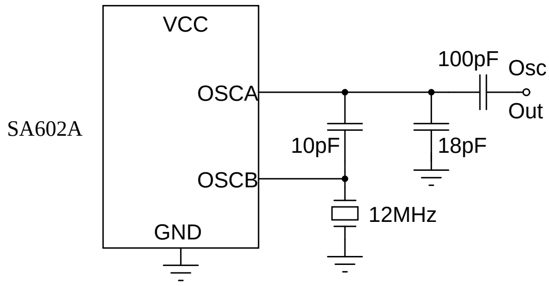



Figure 8.8: The Colpitts oscillator of TRC-11.

### 8.3.5 Frequency control in oscillators

In the oscillator circuit of Fig. 8.3(d), the frequency determining parameters are $C$ and $L$. 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 or *tuning* diode). The symbol of the varactor diode is shown in Fig. 8.9(a). Varactor diodes are used with a reverse bias voltage. The capacitance across the cathode and an-

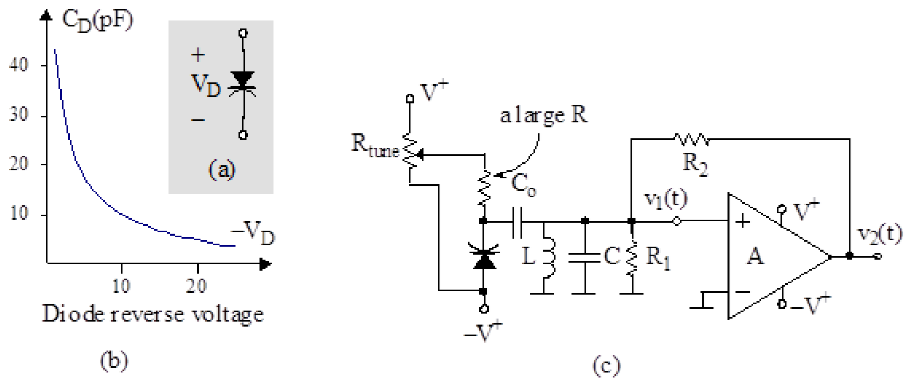



Figure 8.9: (a) The symbol of a varactor diode, (b) the capacitance variation of a varactor diode as a function of reverse voltage, (c) a voltage controlled oscillator built by a varactor.

ode depends on the level of reverse bias voltage. The variation of the diode capacitance with respect to reverse diode voltage is given in Fig. 8.9(b).

The circuit in Fig. 8.9(c) delineates the way a varactor diode is used in a variable frequency oscillator circuit. The potentiometer $R_{tune}$, connected between two supply voltages, control the reverse voltage bias on the diode. The capacitance $C_o$ is a DC block capacitor, preventing a short-circuit of the DC bias voltage by the inductance $L$. The large series resistor only serves to isolate the tuned circuit elements from $R_{tune}$, 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

$$C + \frac{C_D C_o}{C_D + C_o} \tag{8.29}$$

Adjusting the potentiometer can now vary the resonance frequency of the tank circuit. Since the oscillation frequency is determined by a voltage, this circuit is called a *voltage controlled oscillator* (VCO).

## 8.4 Superheterodyne receiver

A good receiver has good *selectivity.* Selectivity means the ability to select a signal among many neighboring signals in the frequency spectrum. It requires a good band-pass-filter. We learned how to build a good band-pass-filter using quartz crystals. However such filters operate only at a fixed center frequency. It is very difficult to build band-pass-filters with variable center frequency. Superheterodyne concept explained below gets around this problem.

US engineer Edwin Armstrong (1890–1954) invented the *superheterodyne* (often shortened to superhet) concept in 1918 during World War I as illustrated in Fig. 8.10. This technique is commonly used in today's receivers: Radio receivers, TV receivers, satellite receivers, mobile phones. It uses a frequency conversion technique using mixers. A mixer converts the incoming RF signal to a fixed *intermediate frequency* (IF) where the fixed frequency band-pass-filter is centered. The mixer achieves the frequency conversion by a multiplication operation. Sup-

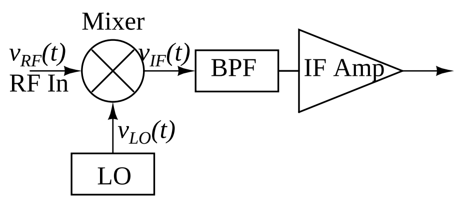



Figure 8.10: Superheterodyne concept invented by Edwin Armstrong.

pose the input RF signal is a sinusoidal signal $v_{RF}(t) = V_{RF}\cos(\omega_{RF}t)$. A variable frequency signal source is the *local oscillator* (LO). It generates a sinusoidal signal $v_{LO} = V_{LO}\cos(\omega_{LO}t)$. The function of the mixer is to multiply these two signals. The output of the mixer, $v_{IF}(t)$ is

$$v_{IF}(t) = v_{RF}(t) \times v_{LO}(t) = V_{RF}V_{LO}\cos(\omega_{RF}t)\cos(\omega_{LO}t) \tag{8.30}$$

Using trigonometric identities we can write it as the sum of two sinusoids at

| System | IF Freq. | LO Freq. | RF Freq. | Image Freq. |
|---|---|---|---|---|
| AM Radio | 455 kHz | 990–2155 kHz | 535–1700 kHz | 1445–2610 kHz |
| FM Radio | 10.7 MHz | 77.3–97.3 MHz | 88–108 MHz | 66.6–86.6 MHz |

Table 8.1: IF, LO, RF and image frequencies of some commonly used systems

sum and difference frequencies:

$$v_{IF}(t) = v_{RF}(t) \cdot v_{LO}(t) = \frac{1}{2} V_{RF} V_{LO} [\cos((\omega_{RF} + \omega_{LO})t) + \cos((\omega_{RF} - \omega_{LO})t)] \tag{8.31}$$

One sinusoid is at the sum frequency of $\omega_{RF} + \omega_{LO}$, the other is at the difference frequency of $\omega_{RF} - \omega_{LO}$.

Suppose the BPF is centered at $\omega_{IF}$. If we choose the local oscillator frequency such that $\omega_{RF} - \omega_{LO} = \omega_{IF}$, then one of the sinusoids at the output of the mixer will be able to pass through the BPF and get amplified by the IF amplifier. Hence, the RF signal at the frequency of

$$\omega_{RF} = \omega_{LO} + \omega_{IF} \text{ or } f_{RF} = f_{LO} + f_{IF} \tag{8.32}$$

will be selected and be amplified, while all other signals will be rejected by the BPF. As an example, let us choose $f_{LO}$=12 MHz and $f_{IF}$=15 MHz. In this case, $f_{RF}$=27 MHz is the selected frequency.

We note that there is one more RF frequency that can be selected:

$$f_{RFi} = f_{IF} - f_{LO} \tag{8.33}$$

since $f_{RFi} + f_{LO} = f_{IF}$ is the frequency of the first sinusoid of Eq. 8.31. This frequency is called the *image frequency*. In the example above, the image frequency is at $f_{RFi}$=3 MHz. To prevent two distinct frequencies to be amplified, we need to get rid of the image frequency at the RF input before it gets into the mixer. For this purpose, we use a band-pass-filter to reject the image frequency. Since the image frequency, $f_{RFi}$, is far away from the desired $f_{RF}$, the rejection can be easily achieved using a simple BPF and it does not have to be a variable frequency. In the example above, we need to reject 3 MHz while passing 27 MHz signal.

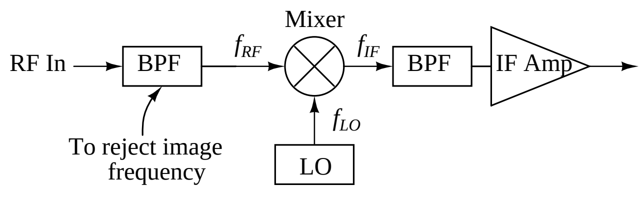



Figure 8.11: Superheterodyne receiver with image rejection.

Table 8.4 lists the IF, LO, RF and image frequencies for some commonly used systems.

# 8.5 Examples

## Example 4

We have a superheterodyne receiver with image rejection as in Fig. 8.11. $f_{RF} =$ 28 MHz, $f_{LO} = 27.5$ MHz and $f_{IF} = 500$ kHz. The BPF to reject the image frequency is depicted in Fig. 8.12(a). The transformer is wound on a core with $A_L = 2.1$ nH/T$^2$. The antenna is represented with purely resistive source impedance of 70 Ω. Find the value of $C_s$ to receive at 28 MHz. What is the image frequency? What is the image rejection in dB?

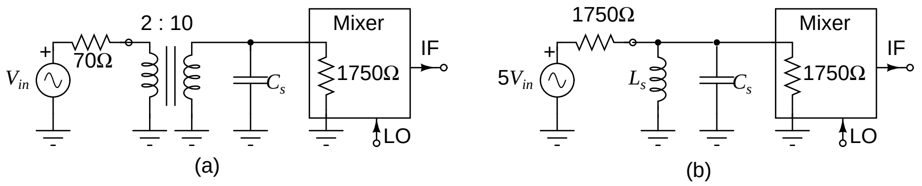



Figure 8.12: (a) Image reject filter of a superheterodyne receiver, (b) Components transferred to the secondary side.

### Solution

The source resistor of 70 Ω and the input voltage source can be transferred to the secondary side as $70(10/2)^2 = 1750$ Ω and $(10/2)V_{in}$ as depicted in Fig. 8.12(b). Hence a maximum power transfer is achieved since the source resistor is equal to the load resistor. The inductance of the secondary is $L_s = 2.1 \cdot 10^2 = 210$ nH. We can find $C_s$ to resonate with $L_s$ at 28 MHz using

$$C_s = \frac{25330}{28^2 \cdot 0.21} = 154 \text{ pF}$$

The resulting circuit is a band-pass-filter with $n = 1$. Comparing with the LPF prototype, we find from Eq. 6.17 in page 226

$$\Delta f = \frac{b_1}{2\pi R C_s} = \frac{2}{2\pi 1750 \cdot 154 \cdot 10^{-12}} = 1.18 \text{ MHz}$$

The image frequency is at $f_{RFi} = f_{LO} - f_{IF} = 27.5 - 0.5 = 27$ MHz. From Eq. 6.24 of page 228 with $n = 1$, $f_o = 28$ MHz and $f = 27$ MHz, we find

$$\frac{P_L}{P_A} = \frac{1}{1 + (f_o/\Delta f)^2 (f/f_o - f_o/f)^2} = \frac{1}{1 + (28/1.18)^2 (27/28 - 28/27)^2} = 0.25$$

Hence the rejection of the image frequency is $10 \log_{10}(0.25) = -6.0$ dB. Note that it is not good idea to use the asymptotic approximations here, since the frequency $f$ is very close to the center frequency, $f_o$ and $f < 0.1 f_o$ is not satisfied.

---

## Example 5

Repeat the problem above when $f_{LO} = 25$ MHz and $f_{IF} = 3$ MHz.

**Solution**

In this case, the image frequency is at $f_{RFi} = f_{LO} - f_{IF} = 25 - 3 = 22$ MHz. We find the image rejection as

$$\frac{P_L}{P_A} = \frac{1}{1 + (f_o/\Delta f)^2 (f/f_o - f_o/f)^2} = \frac{1}{1 + (28/1.18)^2 (22/28 - 28/22)^2} = 0.0074$$

The rejection of the image frequency is $10 \log_{10}(0.0074) = -21.3$ dB, obviously a better rejection value than the value found in the previous example.

---

## Example 6

The OPAMP circuit shown in Fig. 8.13(a) acts like a square-wave oscillator. Unlike the negative resistance oscillator of Fig. 8.4(b), the OPAMP operates in the saturated output region. What is the frequency of the square wave, if $V^+ = V^-$?

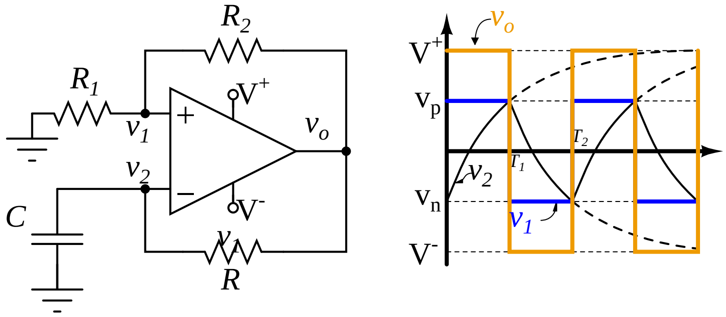



Figure 8.13: (a) Square wave oscillator, (b) voltage waveforms.

**Solution**

Referring to Fig. 8.13(b), at $t = 0^+$, the output voltage is $v_o = V^+$. Therefore, the voltage, $v_1(0^+)$, at the positive input terminal of OPAMP is found from the voltage divider as

$$v_1(0^+) = \frac{R_1}{R_1 + R_2} V^+ = v_p$$

At $t = 0^+$, the voltage, $v_2$, (equal to the capacitor voltage) at the negative input terminal is *assumed* to be equal to an unknown voltage $v_2(0^+) = v_n < 0$ which will be determined later.

The capacitor $C$ charges toward $V^+$ with a time constant $\tau = RC$. During the charging period ($0 < t < T_1$) we have $v_1(t) > v_2(t)$, and hence the output voltage remains at $v_o(t) = V^+$ for $0 < t < T_1$. Using the procedure of page 46, we write

$$v_2(t) = V^+ + (v_n - V^+) e^{-t/\tau} \quad \text{for} \quad t > 0$$

When $v_2(t)$ exceeds the voltage $v_p$ at $t = T_1^+$, we get $v_2(T_1^+) > v_1(T_1^+)$ and the output voltage jumps to the negative saturation voltage $V^-$. Thus the voltage,

$v_1(T_1^+)$, at the positive input terminal of OPAMP becomes

$$v_1(T_1^+) = \frac{R_1}{R_1 + R_2} V^- = v_n$$

Now, the capacitor discharges toward $V^-$ with the same time constant $\tau = RC$. Since we have $v_1(t) < v_2(t)$, the output voltage remains at $v_o(t) = V^-$ for $T_1 < t < T_2$. Using the same procedure we arrive at

$$v_2(t) = V^- + (v_p - V^-)e^{-(t-T_1)/\tau} \text{ for } t > T_1$$

At $t = T_2^+$, $v_2$ goes below $v_n$ and as a result the output voltage jumps back to $V^+$, completing the full cycle. We can find $T_1$ using the exponential equations above:

$$v_2(T_1) = v_p = V^+ + (v_n - V^+)e^{-T_1/\tau} \text{ or } T_1 = \tau \ln\left(\frac{V^+ - v_n}{V^+ - v_p}\right)$$

In a similar manner, we find $T_2$:

$$v_2(T_2) = v_n = V^- + (v_p - V^-)e^{-(T_2-T_1)/\tau} \text{ or } T_2 - T_1 = \tau \ln\left(\frac{V^- - v_p}{V^- - v_n}\right)$$

If $V^- = -V^+$, substituting the values of $v_p$ and $v_n$ we get

$$T_1 = \tau \ln\left(\frac{V^+ - v_n}{V^+ - v_p}\right) = \tau \ln\left(\frac{2R_1}{R_2} + 1\right) \text{ and } T_2 - T_1 = \tau \ln\left(\frac{2R_1}{R_2} + 1\right)$$

Hence the frequency of the square wave is

$$f = \frac{1}{T_2} \text{ with } T_2 = 2RC \ln\left(\frac{2R_1}{R_2} + 1\right)$$

We note that this type of OPAMP oscillator is not suitable for frequencies above 100 kHz.

---

## Example 7

The OPAMP circuit depicted in Fig. 8.14 is known as *phase shift* oscillator. Find the frequency of oscillation and the conditions for oscillation.

### Solution

Assuming that the OPAMP is not saturated, we have $V_1 = V_2 = 0$. We write the node equations for the nodes $V_3$, $V_4$ and $V_5$ as

$$\frac{V_3 - V_o}{1/j\omega C} + \frac{V_3}{R} + \frac{V_3 - V_4}{1/j\omega C} = 0 \tag{8.34}$$

$$\frac{V_4 - V_3}{1/j\omega C} + \frac{V_4}{R} + \frac{V_4 - V_5}{1/j\omega C} = 0 \tag{8.35}$$

and

$$\frac{V_5 - V_4}{1/j\omega C} + \frac{V_5}{R} = 0 \qquad \text{or} \qquad V_4 = \frac{1 + j\omega RC}{j\omega RC} V_5 \tag{8.36}$$

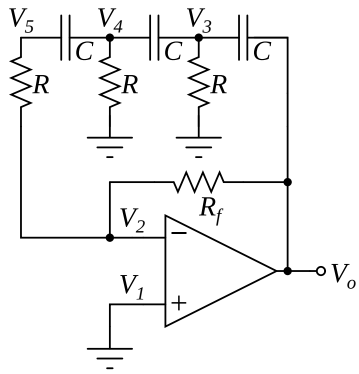



Figure 8.14: Phase shift oscillator.

To simplify the notation, we set $X = j\omega RC$. Combining Eqs. 8.34 and 8.36, we get

$$V_3 = \frac{X}{1+2X} V_o + \frac{1+X}{1+2X} V_5 \tag{8.37}$$

Combining Eqs. 8.35 and 8.36, we reach at

$$V_3 = \frac{1+2X}{X} \frac{1+X}{X} V_o - V_o \tag{8.38}$$

Equating the right hand sides of Eqs. 8.37 and 8.38

$$\frac{X}{1+2X} V_o + \frac{1+X}{1+2X} V_5 = \frac{(1+2X)(1+X) - X^2}{X^2} V_5 \tag{8.39}$$

After simplification we get

$$\frac{V_5}{V_o} = \frac{X^3}{1+5X+6X^2+X^3} = \frac{-j\omega^3 R^3 C^3}{(1-6\omega^2 R^2 C^2) + j(5\omega RC - \omega^3 R^3 C^3)} \tag{8.40}$$

From the OPAMP inverting amplifier configuration, we also have

$$V_o = -\frac{R_f}{R} V_5 \tag{8.41}$$

To get a sustained oscillation, we set the loop gain to unity using Eqs. 8.40 and 8.41:

$$-\frac{R_f}{R} \frac{-j\omega^3 R^3 C^3}{(1-6\omega^2 R^2 C^2) + j(5\omega RC - \omega^3 R^3 C^3)} = 1 \tag{8.42}$$

Note that if

$$1 - 6\omega^2 R^2 C^2 = 0 \qquad \text{or} \qquad \omega = \frac{1}{\sqrt{6}RC} \tag{8.43}$$

the second term of Eq. 8.42 becomes negative real and we obtain

$$\frac{R_f}{R} \frac{\omega^2 R^2 C^2}{5 - \omega^2 R^2 C^2} = 1 \tag{8.44}$$

So, the oscillation occurs at

$$\omega_o = \frac{1}{\sqrt{6}\,RC} \qquad \text{if} \qquad R_f \geq 29R \tag{8.45}$$

Since the oscillation requires a relatively high gain of the inverting amplifier and OPAMPs have a gain-bandwidth product limitation (see page 208), this oscillator is suitable only at low frequencies.

## 8.6 Problems

1. Consider a superheterodyne receiver with $f_{IF}$=1 MHz and $f_{RF}$=28 MHz. (a) Find the image frequency. (b) Design an input band-pass-filter (as shown in Fig. 8.11) that passes 28 MHz with an attenuation no more than 1 dB and rejects the image signal by at least 40 dB. (Assume that source and load impedance of the filter is 50 Ω.)

2. Consider the circuit in Fig. 8.15(a). Find the transfer function $H(\omega) = V_2(\omega)/V_1(\omega)$. Determine the frequency, $\omega_o$, at which $\angle H(\omega_o) = 0$, when $R_3 = R_4$ and $C_3 = C_4$. What is $|H(\omega_o)|$ at that frequency?

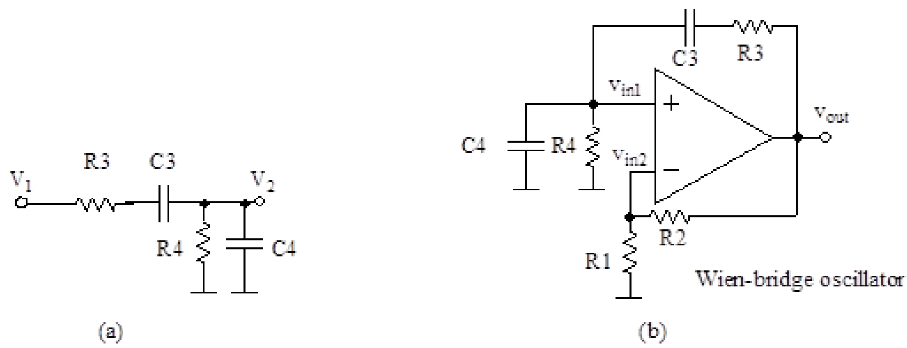



Figure 8.15: Circuits for problems 2 and 3

3. The circuit shown in Fig. 8.15(b) is called the *Wien-bridge oscillator*. Max Wien (1866–1938) invented the Wien bridge. William Hewlett (1913–2001) of Hewlett-Packard company was the first to build an Wien-bridge oscillator as the first product of the company. The network shown in Fig. 8.15(a) provides the positive feedback. The circuit oscillates at the frequency $\omega_o$ where $\angle H(\omega_o)$=0 (as given in problem 2), if the condition $(1 + R_2/R_1)|H(\omega_o)| > 1$ is satisfied. If $R_3 = R_4$ and $C_3 = C_4$, what is the minimum value of $R_2/R_1$ so that the circuit can oscillate?

4. Consider a parallel $LC$ circuit with $L$=320 nH and a resonance frequency of 28 MHz. A varactor diode is placed in parallel with it. When the reverse voltage across the varactor diode is changed between 1 to 10 V, the capacitance of the diode varies between 4 pF to 1.5 pF. Find the new resonance frequency with the varactor reverse bias at 1 V and at 10 V.

5. A transconductance amplifier with $G_f$=0.02 S is connected as a Colpitts oscillator as shown in Fig. 8.16(a). Find the value of the inductance if an oscillation at 12 MHz is desired, and if $C_1$=10 pF and $C_2$=15 pF. Find the maximum value of the resistance, $r$, that still allows an oscillation.

6. Repeat the problem 5, if there is a stray capacitance of $C_s$=7 pF exists as shown in Fig. 8.16(b).

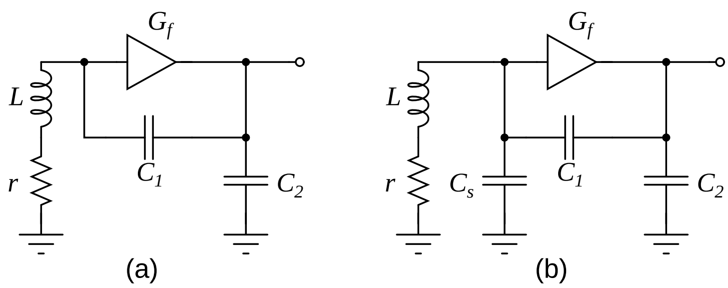



Figure 8.16: (a) Circuit for problem 5, (b) circuit for problem 6.

# Chapter 9

# ON THE AIR

Danish physicist Hans Christian Øersted (1777–1851) showed in 1820 that a current carrying wire creates a magnetic field. The same year, André Marie Ampére found that two parallel current-carrying wires can attract or repel each other, depending on the relative direction of the currents. In 1831, Michael Faraday observed that a moving magnet through a loop of wire can create a current. Scottish physicist James Clerk Maxwell (1831–1879) was the first to formulate the relations between electric and magnetic fields in a unified theory. His formulation of 1865 shows that a changing magnetic field creates an electric field, a changing electric field creates a magnetic field and predicts that the combination of these fields, called *electromagnetic waves*, propagate at the speed of light. Four elegant vector equations* describing this behavior is known as Maxwell's equations, which underpins much of the modern technological world. In 1888, Heinrich Rudolf Hertz confirmed experimentally the existence of electromagnetic waves predicted by Maxwell.

Antennas convert electrical signals into electromagnetic waves for transmission, and they also work in the other direction to convert electromagnetic waves to electrical signals for reception. Antennas are combinations of pieces of conductors of specific lengths and shapes. There are many different types of antennas for numerous applications. More detailed information on antennas can be found in other books [15–17].

What follows in this chapter is a descriptive theory of electromagnetics and antennas.

## 9.1 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

$$\lambda(\mathrm{m}) = \frac{300}{f(\mathrm{MHz})} \tag{9.1}$$

Wavelength can also be interpreted as the distance an electromagnetic wave travels in one full cycle.

*Maxwell's original formulation of electromagnetic theory contained 20 equations. Oliver Heaviside reduced the equation count to four.

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 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.

### Radiation from a dipole antenna

A dipole antenna is made up of two pieces of conductor wires or poles aligned on a straight line with a small isolating gap in between. A dipole is shown in Fig. 9.1. The generation of electromagnetic waves by the dipole is also shown in the same figure. When a voltage source $V_s$ is connected to these two conductors

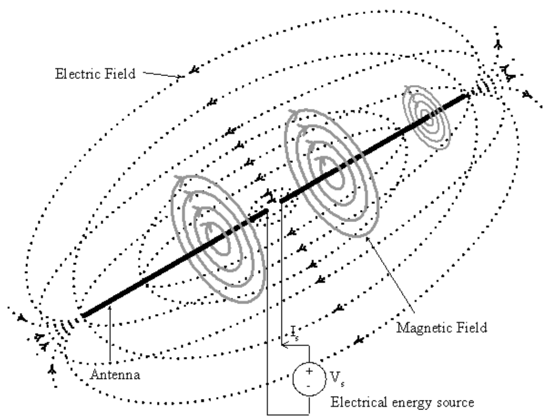



Figure 9.1: Dipole antenna and radio wave generation concept.

across the gap, as shown in Fig. 9.1, an electric field between the entire surfaces of two conductors is produced due to the potential difference between them. This field is time varying at the frequency of the source. The electric field is denoted by letter $E$ 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 Fig. 9.2(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 Fig. 9.2(b). The current along the antenna produces

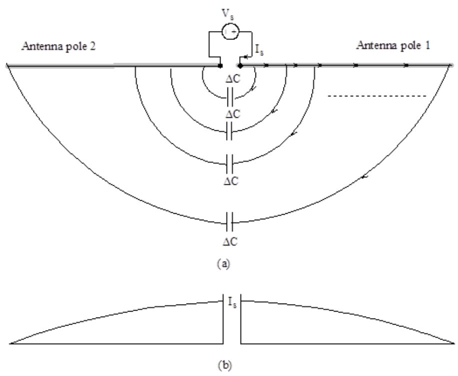



Figure 9.2: (a) Distributed capacitance on antenna, and (b) current amplitude distribution along antenna.

a magnetic field shown in Fig. 9.1. Again, the frequency of the magnetic field is the same as that of the voltage source. Letter $H$ denotes the magnetic field and it has units of A/m. The electric and magnetic fields have magnitude and direction and hence they can be modelled as vectors. Fig. 9.3 depicts color-coded E and H fields in the vicinity of a dipole antenna as determined from a finite-element simulation[†].

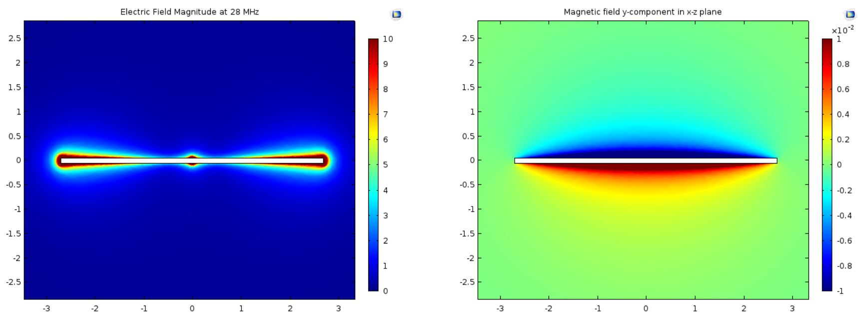



Figure 9.3: Magnitude of E-field (left) and y-component of H-field (right) shown in z-x plane for a dipole antenna placed along z-axis (along horizontal direction).

During propagation in space, some components of $E$ and $H$ 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. Fig. 9.4 demonstrates the change in the E-field at progressively higher distances from the dipole antenna.

When we observe the electromagnetic wave emanating from a dipole at a far away distance, the equal phase surfaces, called wavefronts[‡], appear like con-

[†] www.comsol.com, Comsol Inc.

[‡] Wavefronts are the surfaces defined by electric or magnetic waves which have the same phase. The fields which have the same phase must have left the antenna at the same time instant.

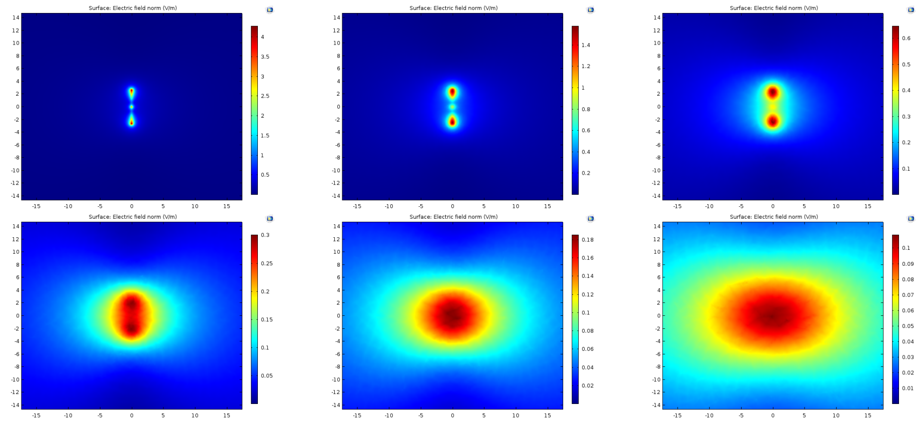


Figure 9.4: Color coded E-field plots of a 28 MHz dipole antenna (5.35 m long) placed along z-axis (vertical direction). E-fields are shown at planes parallel to z-axis at distances 0.25, 0.5, 1.0, 2.0, 4.0 and 8.0 m away.

centric 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 Fig. 9.5. The direction of propagation in this figure is outward from the center.

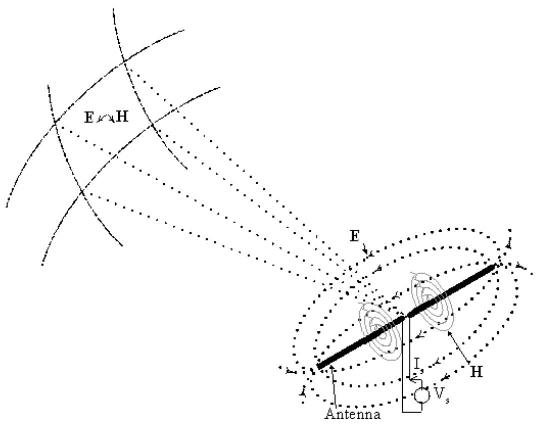



Figure 9.5: Radio wave far away from the source dipole.

The electromagnetic wave generated at some instant gets away from the dipole in all directions, at a speed of $3{\cdot}10^8$m/sec.

Now let us take a closer look at the field shown on the patch over the spherical surface, in Fig. 9.5. 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 Fig. 9.6. 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

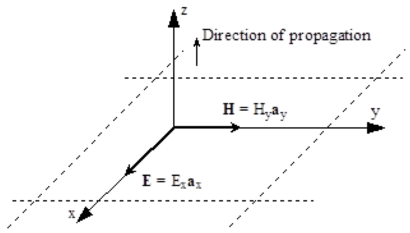



Figure 9.6: Orthogonal electric and magnetic fields and the direction of propagation.

in x and y directions. We can write the two field vectors **E** and **H** as

$$\mathbf{E} = E_x \mathbf{a}_x \text{ and } \mathbf{H} = H_y \mathbf{a}_y \tag{9.2}$$

$E_x$ and $H_y$ in an electromagnetic wave propagating in space are related to each other as

$$\frac{E_x}{H_y} = \eta_o = \sqrt{\frac{\mu_o}{\epsilon_o}} \tag{9.3}$$

Here $\mu_o$ and $\epsilon_o$ are the permeability and the permittivity of the free space (and air), respectively. Their values are

$$\mu_o = 4\pi \cdot 10^{-7}\text{H/m} \text{ and } \epsilon_o = 8.85 \cdot 10^{-12}\text{F/m} \tag{9.4}$$

$\eta_o$ is called the *free-space wave impedance* with a unit of $\Omega$ and it can be calculated as

$$\eta_o = 120\pi\,\Omega \approx 377\Omega \tag{9.5}$$

The speed of light, $c$, is also related to permeability and permittivity of free space as

$$c = \frac{1}{\sqrt{\mu_o \epsilon_o}}. \tag{9.6}$$

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, $P$, in the $z$ direction of the wave depicted in Fig. 9.6, is given in terms of the product of the electric field phasor and the complex conjugate of magnetic field phasor as

$$P = \frac{1}{2}\mathcal{Re}\{E_x H_y^*\} = \frac{|E_x|^2}{2\eta_o} \tag{9.7}$$

The unit of power density is $\text{W/m}^2$. The total power over an area can be calculated by integrating $|E_x|^2/2\eta_o$ over that area.

An antenna, which radiates in all directions with equal preference, is called an *omnidirectional* antenna. For example, a dipole whose length is very small compared to the wavelength can be approximated as an omnidirectional antenna. In such antennas, the electric field is uniformly distributed over the spherical equal phase surface. At a distance $r$ from the antenna, the power density, $P(r)$, is uniform over the sphere and is given as

$$P(r) = \frac{P_o}{4\pi r^2} \tag{9.8}$$

where $P_o$ is the power delivered to the antenna and $4\pi r^2$ is the area of the sphere. From Eq. 9.7 and 9.8 the electric field at that distance becomes

$$|E_x(r)| = \sqrt{2\eta_o P(r)} = \sqrt{\frac{2\eta_o P_o}{4\pi r^2}} = \frac{\sqrt{60 P_o}}{r} \tag{9.9}$$

where $P_o$ is in watts and $r$ is in meters. For example, a transmitter delivering 10 mW to an omnidirectional (no directivity) antenna generates an electric field strength of 0.8 mV/m at 1 km distance.

### 9.1.1 Receiving dipole 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 dipole antenna correctly aligned in an electromagnetic wave parallel to electric field, **E**, is shown in Fig. 9.7. The open circuit voltage between the two conductors

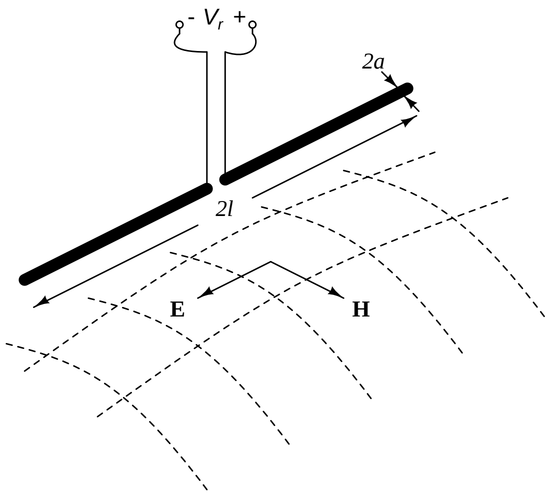



Figure 9.7: A dipole of total length $2l$ correctly aligned to receive the incoming electromagnetic field.

is the potential difference between them. The potential of each conductor is the potential at its mid-point. Suppose the dipole has a total length of $2l$ meters (where $l$ is very small compared to wavelength). The received voltage is approximately given as

$$V_r = E_x\, l \tag{9.10}$$

For example, the open-circuit voltage developed across a 1.2 m long dipole is 0.48 mV when the incident electric field magnitude is 0.8 mV/m.

## 9.2 Dipole antenna impedance

The energy radiated from an dipole antenna appears as lost energy to the circuit, which drives it. As far as the circuit is concerned, the dipole 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*.

The radiation resistance of a short dipole of total length $2l$, far away from other conductors, is given approximately in $\Omega$ as

$$R_{rd} \approx 80\pi^2 \left(\frac{l}{\lambda}\right)^2 \quad \text{for} \quad \frac{l}{\lambda} < 0.1 \tag{9.11}$$

For example, the radiation resistance of a 2 m long dipole at 27 MHz has a radiation resistance of 6.4 $\Omega$. For longer dipoles, we can use the following polynomial approximation

$$R_{rd} \approx -0.4787 + 46\frac{l}{\lambda} + 15.64\left(\frac{l}{\lambda}\right)^2 + 3873\left(\frac{l}{\lambda}\right)^3 \quad \text{for} \quad 0.1 < \frac{l}{\lambda} < 0.25 \tag{9.12}$$

The effective capacitance between the two conductors depends on the diameter of the conductor, as well as its length. This capacitance, $C_d$, can be calculated for a cylindrical conductor of radius $a$ as

$$C_d \approx \pi\epsilon_o \frac{l}{\ln(l/a) - 1} \quad \text{for} \quad \frac{l}{\lambda} < 0.1 \tag{9.13}$$

where both $l$ and $a$ are in meters and $C_d$ is in Farads. For example, the capacitance of a short dipole of length 20 cm with a diameter of 1 mm is 0.647 pF. A short dipole can be approximated by the radiation resistance $R_{rd}$ in series with the capacitance $C_d$.

If the dipole is not so short, the inductances of the lines are no longer negligible. In this case, the series reactance of the dipole antenna is given by

$$X_d = -120\cot\left(2\pi\frac{l}{\lambda}\right)[\ln(l/\lambda) - 1] + X_L(l/\lambda) \quad \text{for} \quad \frac{l}{\lambda} < 0.25 \tag{9.14}$$

where $X_L(l/\lambda)$ can be calculated by the following polynomial approximation

$$X_L(l/\lambda) \approx -0.4456 + 106.86\frac{l}{\lambda} - 342.64\left(\frac{l}{\lambda}\right)^2 + 2382\left(\frac{l}{\lambda}\right)^3 \tag{9.15}$$

### 9.2.1 Monopole antenna

Monopole or whip antennas used to be the most commonly encountered antennas [18]. Old car radio antennas and old 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 Fig. 9.8. Conducting surfaces behave like mirrors to electromagnetic waves. As a direct result of this physical property, a pole of length $l$ placed on a conducting surface behaves like a dipole of length $2l$. The electrical connections are between the pole and the ground plane. The radiation resistance of a short monopole of length $l$ with an infinite conducting ground plane is

$$R_{rm} \approx 40\pi^2 \left(\frac{l}{\lambda}\right)^2 \quad \text{for} \quad \frac{l}{\lambda} < 0.1 \tag{9.16}$$

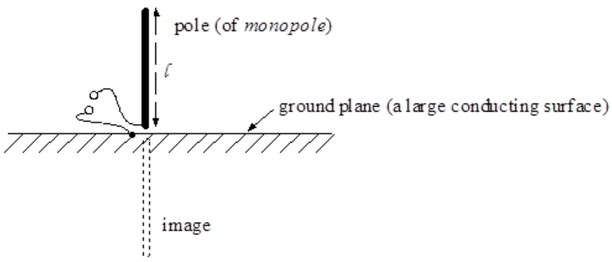



Figure 9.8: A monopole antenna.

which is half as much as the resistance of a dipole of length $2l$.

For longer monopoles, one can use half the resistance of the dipole given in Eq. 9.12.

$$R_{rm} \approx -0.2393+23.0\frac{l}{\lambda}+7.82\left(\frac{l}{\lambda}\right)^2+1936\left(\frac{l}{\lambda}\right)^3 \quad \text{for} \quad 0.1<\frac{l}{\lambda}<0.25 \quad (9.17)$$

The ground plane diameter should be at least $10\lambda$ and there should be no other conductors nearby for these equations to be accurate.

The radiated power in an antenna is

$$P_o = R_r\frac{I^2}{2} \quad (9.18)$$

For the same input current $I$, a dipole (of length $2l$ ) radiates a total power $P_r$ to the entire space, while a monopole (of length $l$) radiates only to half space, hence a total power of $P_r/2$. Therefore, the radiation resistance of monopole is half as much as the radiation resistance of an equivalent dipole. For example, a monopole of length 10 cm at 28 MHz has a radiation resistance of 0.034 Ω. If the length is 50 cm, the radiation resistance becomes 0.86 Ω.

Similarly a monopole antenna of length $\lambda/4$ placed on a large conducting surface behaves like a $\lambda/2$ long dipole.

The effective capacitance between the conductor (of length $l$ and radius $a$) and the infinitely large ground plane in a monopole antenna is given by

$$C_m \approx 2\pi\epsilon_o\frac{l}{\ln(l/a)-1} \quad \text{for} \quad \frac{l}{\lambda}<0.1 \quad (9.19)$$

where $C_m$ is in Farads and both $l$ and $a$ are in meters. Therefore, the capacitance of a monopole of length $l$ is the twice as much as the capacitance of a dipole of length $2l$. For example, the capacitance of a monopole of length 10 cm with a diameter of 1 mm is 1.29 pF.

When the monopole is not short, half of the reactance given in Eq. 9.14 can be used in the equivalent circuit of the monopole:

$$X_m = \frac{X_d}{2} = -60\cot\left(2\pi\frac{l}{\lambda}\right)\left[\ln\left(l/\lambda\right)-1\right]+\frac{X_L\left(l/\lambda\right)}{2} \quad \text{for} \quad \frac{l}{\lambda}<0.25 \quad (9.20)$$

Fig. 9.9 shows the series resistance, $R_m$, and series reactance, $X_m$, of a monopole antenna using Eqs. 9.17 and 9.20 for various $l/a$ ratios. For example, a monopole

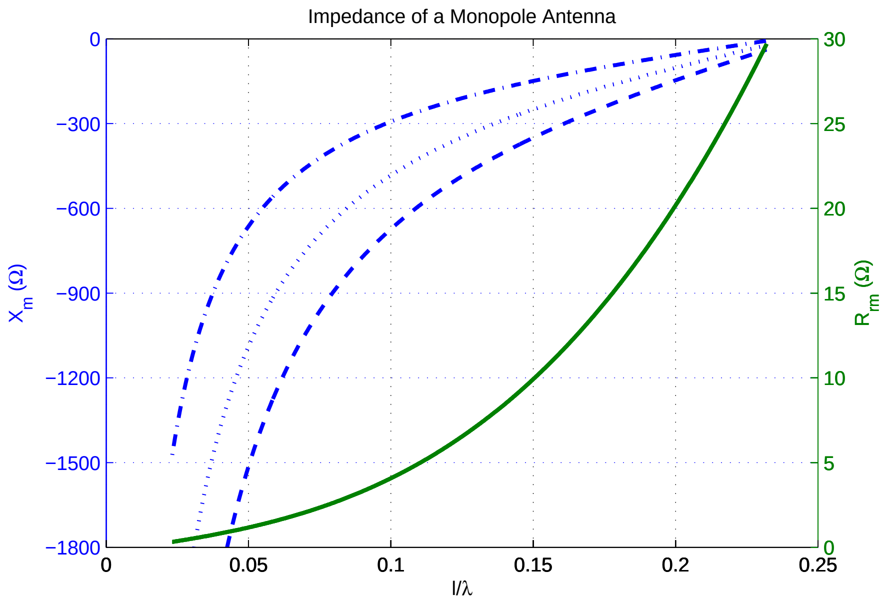



Figure 9.9: Series resistance (solid) and series reactance with $l/a$=100 (dash-dot), $l/a$=1000 (dotted), $l/a$=10000 (dashed) of a monopole antenna placed on a very large planar conducting ground plane.

of length 120 cm with a diameter of 1 mm has a series resistance of 5.15 Ω and a series reactance of −474 Ω.

The reactive component becomes zero when the monopole length is $l$=0.25$\lambda$ for an infinitely thin conductor. Under this condition, when the antenna impedance becomes purely resistive, the antenna is known as quarter-wave monopole antenna and it becomes a very good radiator. You can refer to Table 1.2 on page 3 to determine the lengths of quarter-wave monopole antennas. For example, a quarter-wave GSM-900 antenna for a cellular phone should be about 8 cm long, while a quarter-wave Wi-Fi antenna is about 3 cm.

When the conductor radius, $a$, increases the resonance occurs at a lower $l/\lambda$ value. We note that presence of a conductor within $\lambda$ distance of the monopole antenna will change these curves significantly. For example, if there are nearby conductors, the capacitance increases and the resonance occurs at a lower frequency. If the ground conductor is not a perfect conductor, there will be an additional loss term due to ground loss, increasing the series resistance.

The equivalent circuits of a transmitting and receiving monopole antennas are shown in Fig. 9.10. In the transmitting monopole, the power dissipated in $R_{rm}$ is the power transmitted, $P_o$. To maximize the power transmitted, one may try to tune out the negative series reactance of the monopole antenna by a series inductance. If the series reactance is very large, the inductance should be also large. The loss of such an inductance may be significant, reducing the power transmitted. If possible, it is much better to increase the size of the antenna to increase the power transmitted.

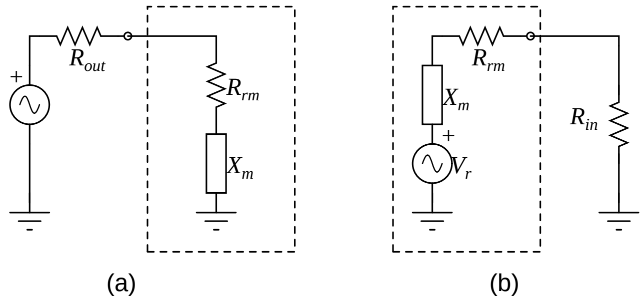



Figure 9.10: Equivalent circuits of (a) a transmitting monopole antenna fed by a source of impedance $R_{out}$ and (b) a receiving monopole antenna feeding a receiver of input impedance $R_{in}$.

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 handset in case of mobile phones.

⧫ TRC-11 uses one short monopole antenna functioning both as transmitting and receiving antenna.

## 9.3 Atmospheric propagation

As indicated by Eq. 9.9, in free space, **E** field of an electromagnetic wave obeys the inverse law: The electric field of the wave is proportional to the inverse of the distance from the transmitter. Since the power density is proportional to the square of the electric field, the power density is proportional to the inverse of the square of the distance. Doubling the distance will reduce the signal level by 6 dB. Electromagnetic waves normally travel in straight lines (called *line-of-sight*) unless they are reflected by a conductor.

The earth is surrounded by *ionosphere*, a conducting layer of gas ionized by the radiation of sun. The exact distribution of ionization in the atmosphere depends on the time of the day, on the season of the year and on the year of the 11-year sunspot cycle. It typically extends between 60 km to 500 km above the earth surface. The ionosphere sometimes acts like an absorber of electromagnetic waves and sometimes acts like a reflector (see Fig. 9.11).

At frequencies between 30 kHz and 3 MHz (wavelengths between 10 km to 100 m), the electromagnetic waves are guided between the conducting surface of earth and the lower (60 to 90 km high) portion of the ionosphere (AM radio band falls into this category). These waves are called *ground waves*, and they follow the curvature of the earth. But they are rapidly attenuated due to loss in the imperfect conductors of earth and ionosphere. These frequencies travel

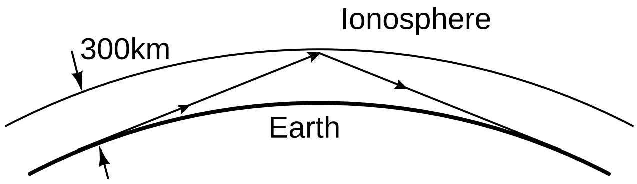



Figure 9.11: Electromagnetic waves reflecting from the conducting layers of ionosphere.

longer at night when the lower layers of ionosphere disappear. This is why AM radios receive higher number of stations at night.

Electromagnetic waves at frequencies between 3 MHz and 30 MHz (wavelengths between 100 m to 10 m), are reflected by the higher layers of ionosphere (up to 500 km high) if their frequencies are below a *critical frequency*, $f_c$. The critical frequency is determined by the ionosphere conditions and hence it depends on the time of the day, the season of the year and the year of 11-year cycle. $f_c$ is typically less than 10 MHz, but it may go as high as 30 MHz. Higher frequencies are attenuated less at the reflections of the imperfect conductors of the ionosphere and the earth surface. Therefore, the best propagation occurs at frequencies just below the critical frequency. For example, if $f_c > 28$ MHz, a communication at 28 MHz may be possible with a small power (1 W) transmitter at distances as high as 4000 km.

Above the critical frequency, $f_c$, no reflection at the ionosphere occurs and the electromagnetic waves penetrate through it. Hence such electromagnetic waves (for example, FM radio band) can be received only in line-of-sight mode. However, during the hot days of summer, especially in Mediterranean Sea and the Persian Gulf regions, it may be possible to receive FM signals from distances 1000 to 4000 km due to a type of radio propagation known as *tropospheric ducting*. In this mode of propagation, the waves do not travel in straight lines but rather in curves, due to an abnormal distribution of temperature in the atmosphere.

### 9.3.1 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". Table 9.1 lists some of the US amateur radio bands.

As a hobby, amateurs [19] communicate surprisingly long distances using very low power. For example, an amateur (call sign TA2BG) operating from Ankara, Turkey using a transmitting power of only 3 W at 40 m band was heard from Indonesia (more than 9000 km away) using WSJT-X computer program [20] and JT65 protocol [21].[§]

The operators using transmitters are required to have amateur licenses. If you intend to use your TRC-11 at higher power for purposes other than the

[§]You can refer to the interesting web page https://pskreporter.info and the map inside for real time information about atmospheric propagation in different amateur bands.

requirements of this course, you must contact your national regulatory agency and obtain an amateur license.

| Band | Frequency (MHz) | Properties |
|---|---|---|
| 160 m | 1.8–2.0 | Long distance at night, noisy in summer |
| 80 m | 3.5–4.0 | Best at night, works best in winter |
| 40 m | 7.0–7.3 | Most reliable all season band |
| 20 m | 14.00–14.35 | Most popular during daytime (*) |
| 10 m | 28.0–29.7 | Best long distance (*) |
| 6 m | 50–54 | Line-of-sight (LOS) propagation, sporadic E |
| 2 m | 144–148 | LOS propagation, tropospheric refraction |
| 1.25 m | 219–225 | LOS propagation |
| 70 cm | 420–450 | LOS propagation |
| 33 cm | 902–908 | LOS propagation |
| 23 cm | 1240–1300 | LOS propagation |

Table 9.1: Some US amateur radio bands. (*) Communications on HF bands with wavelengths 20 m or shorter depend critically on the state of the ionosphere, which in turn depends on the 11-year sunspot cycle.

## 9.4 Problems

1. A short monopole of length 20 cm is used as the antenna of a mobile set operating at 150 MHz. What is the open circuit voltage generated at a receiving antenna, when another set transmits 2W at 3km distance?

2. Assume we want to use a dipole antenna, which has a radiation resistance of 7 Ω with TRC-11. Calculate the total length of the antenna. Calculate the antenna capacitance $C_d$ if the diameter of the poles is 1 cm. Calculate the value of series inductor required to tune this capacitor at 27 MHz. What is the $Q$ of this antenna?

3. A monopole of length 30 cm and diameter 1 mm receives a signal at 24 MHz. The transmitter is 10 W and it is at a distance of 10 km. The monopole is connected to a receiver with an input impedance of 120 Ω. Find the voltage amplitude at the input of the receiver.

4. A typical FM car antenna is a dipole antenna integrated in the back windshield of the car. Estimate the total length ($2l$) of the dipole antenna with purely real impedance for your favorite FM station.

# Appendix A

# LTSpice Tutorial

LTSpice is a analog circuit simulator with integrated schematic capture and waveform viewer. It is distributed freely by Linear Technology Corp (now Analog Devices). You can download it from the web site of the company:

`www.analog.com/en/design-center/design-tools-and-calculators.html`

It is a very powerful program and can be used to simulate any linear or nonlinear circuit of any size. It is highly recommended that you learn how to use it effectively.

The schematic entry can use more than 2,000 symbols. You can also draw your own symbols for devices you wish to import into the program.

Transient (Time domain) analysis can be used to simulate circuits containing linear ($R$, $L$ or $C$) or nonlinear (diodes) elements of any order. Note that the analytic analysis methods presented in the text deals only with time-domain analysis of first-order circuits. LTSpice can cope with any order.

AC analysis can be used to find the transfer function of circuits containing linear elements only. LTSpice solves the circuits using the phasor approach.

## A.1 DC analysis of a resistive circuit

- File → New Schematic (or click New Schematic icon) for the purpose of generating the schematic shown in Fig. A.1.
- Press F2 (or click Component icon) → voltage (to enter a voltage source), left-click to enter it on the schematic window.
- Right click on the voltage source → 5 (to set the voltage at 5 V)
- Press r (or click Resistor icon) (to enter a resistor)
- Ctrl-R (to rotate the resistor) before placing on the schematic
- Right click on the resistor → 2 kΩ (to define the value of resistor as 2 kΩ)
- Enter the other resistor in a similar manner.
- Press F2 → current (to enter a current source)

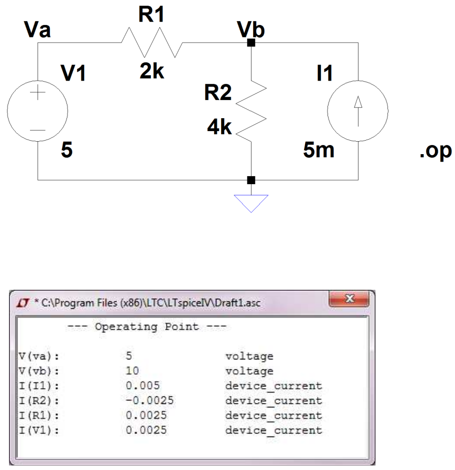


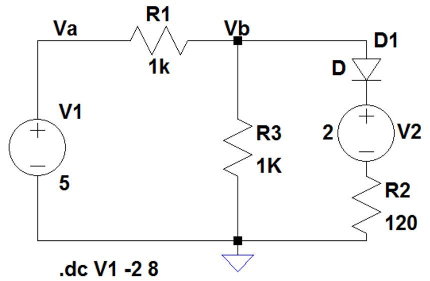

Figure A.1: LTSpice schematic of a resistive circuit and the result window after DC analysis.

- Ctrl-R twice to flip the current source before placing it in the schematic.
- Right click on the current source → 4 m (to set the current at 5 mA)
- Press g (or click Ground icon) → place it on the bottom node (to define the ground node)
- Press F3 (or click Wire icon) (to join the components using Wire tool)
- Press F4 (or click Label Net) → Va and place it above V1 as the node name. Two nodes with the same node name are assumed to be connected (without a wire between them). This is useful to simplify the schematic if it becomes too crowded with wires.
- Add node name Vb in a similar manner.
- Simulate → Edit Simulation Cmd → DC op pnt and place it on the schematic (.op should appear on the schematic).
- Simulate → Run (or click Run icon)
- A result window will appear as in Fig. A.1 showing the voltage and current values.
- The reference direction of currents of resistors are determined by the original placement or rotation of the component. If the reference label of a resistor is on the right-hand-side, the current is defined downwards, otherwise it is upwards. To modify the reference direction click Move icon, choose the resistor by placing a rectangle over it and ctrl-R to rotate it.

## A.2 Input-output relation of a diode circuit

We would like to obtain $V_b$ as a function of $V_a$ for a circuit containing a diode (Fig. A.2).

- File → New Schematic (or click New Schematic icon) for the purpose of generating the schematic shown in Fig. A.2.



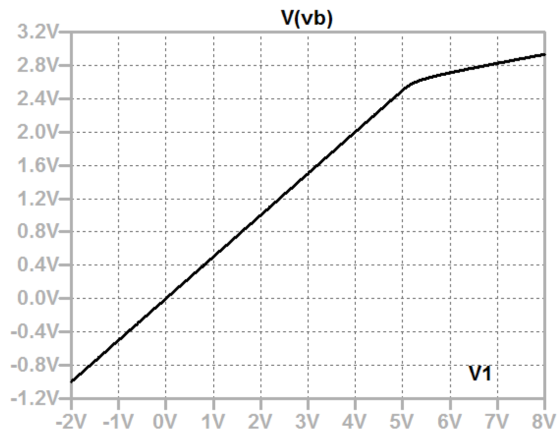




Figure A.2: LTSpice input-output relation analysis of a diode circuit.

- Place the voltage sources and resistors. Set their values.
- Press d (or click Diode icon) and place the diode.
- Place the Ground node and join the components using Wire tool.
- Press F4 (or click Label Net icon) → Va (to label the input node)
- Label the output node as Vb
- Simulate → Edit Simulation Cmd → Dc sweep → Name of 1st Source to Sweep: V1, Start Value: -2, Stop Value: 8. Place the Spice directive (.dc V1 -2 8) on the schematic.
- Simulate: Click Run icon

- An empty graph should appear. You can add a trace by going to schematic window and clicking on the node when red voltage probe appears. Click Vb node to see the relation $V_b$ versus $V_a$.
- Plot Settings → Notes and Annotations → Place Text: Va. Place it near X-axis.
- You can copy the graph using Tools → Copy bitmap to Clipboard (for pasting into another application)

## A.3 Time domain analysis of an $RC$ Circuit

- File → New Schematic (or click New Schematic icon) for the purpose of generating the schematic shown in Fig. A.3

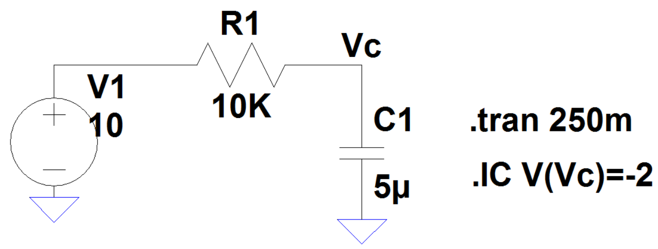



Figure A.3: LTSpice schematic of the $RC$ circuit.

- Place the voltage source and resistor. Set their values.
- Press c (or click Capacitor icon) (to enter a capacitor)
- Right click on the capacitor → 5u (to define 5 $\mu$F)
- Place the Ground node and join the components using the Wire tool.
- Label the capacitor voltage as Vc
- Press s (or click SPICE Directive icon) → .IC V(Vc)=-2 and place it on the schematic (to specify the initial capacitor voltage at −2 V using the Initial Condition directive)
- Simulate → Edit Simulation Cmd → Stop Time: 250 m (to make an analysis from 0 to 250msec), place the command on the schematic.
- At this point you should have a schematic as shown in Fig. A.3.
- Simulate: Click Run icon
- A graph should appear (see Fig. A.4. You can add new traces by clicking on the node with the voltage probe. You can plot currents by bringing the cursor over a component and clicking. You can remove the unwanted traces by right clicking on the trace name and clicking on "Delete this trace" button.

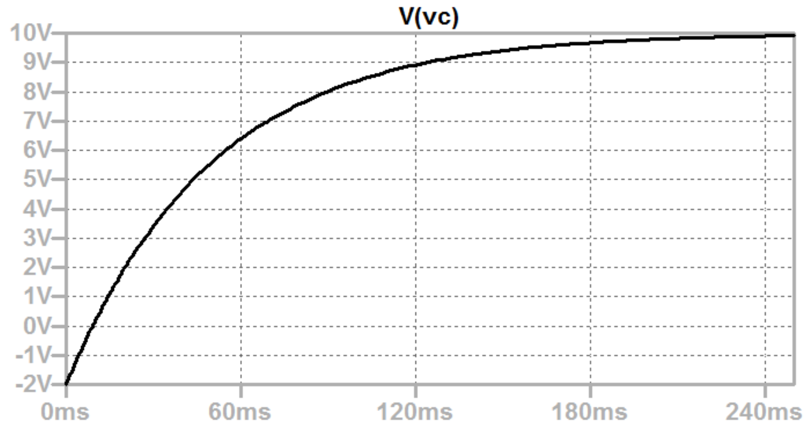



Figure A.4: LTSpice simulation result of capacitor voltage for the $RC$ circuit.

- You can attach cursors to read the values of the graphs. For this purpose, right click on the trace name and click Attached cursor combo box. You can attach one or two cursors. Using two cursors, you can determine the difference between two points on the graph.
- In a time domain simulation, the instantaneous power dissipated on a component can be determined by clicking ALT-Left click on the component. An expression for the instantaneous power (involving voltage and current) will display in the graph window. The average value of this expression can be found by CTRL-Left click on the instantaneous power expression in the graph window. Note that for an accurate average value the simulation time should be an integer multiple of the excitation period. When a lossless component like a capacitor is in a circuit with sinusoidal excitation, the instantaneous power on the capacitor in the steady-state will change between positive and negative values with an average value of zero. (Because of numerical errors, that average value might be a small but a non-zero value.) If the circuit is not excited sinusoidally, the power may be non-zero, indicating the finite stored energy in the capacitor.
- You can change the colors of the graphs using Tools → Color Preferences
- Change the initial condition of the capacitor and the value of the resistor. Find out what happens.
- You can also specify an initial condition on a branch current. For example, IR1(0)=1 mA can be specified by .IC i(R1)=1 m

## A.4 Time domain analysis of a second order $RC$ circuit

- Draw the schematic as shown in Fig. A.5. Enter the values of the components.

- For the voltage source choose Advanced. Choose PULSE. Vinitial:0, Von:5, Tdelay:0, Trise: 1n, Tfall: 1n, Ton: 5 m to define a 5 ms wide 5 V pulse with 1 ns rise and fall times.

- In Edit Simulation Command enter Stop Time: 10 m (to simulate up to 10 ms). Place that command on the schematic.

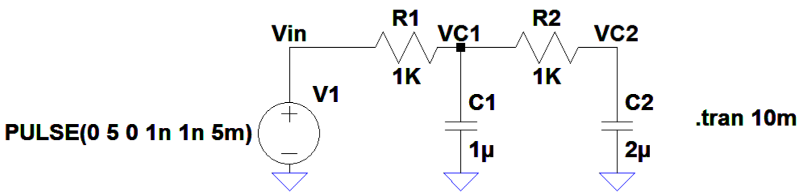



Figure A.5: LTSpice schematic of a second-order *RC* circuit.

- Press Run command. Click on the Vin, VC1 and VC2 nodes to display the voltage waveforms as shown in Fig. A.6. Note that the waveforms are not of the kind that are solved analytically in the text. LTSpice can handle circuits of any order :)

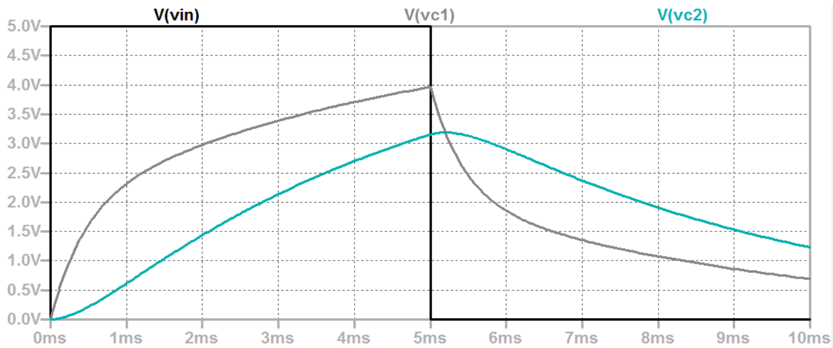



Figure A.6: LTSpice simulation of the time domain response to a pulse for the second-order *RC* circuit.

## A.5 AC Analysis of a series *RLC* circuit

The transfer function of an *RLC* circuit is to be found as follows:

- Draw the schematic as shown in Fig. A.7. Enter the values of the components. R=51, L=8 $\mu$H, C=50 pF.

- For the voltage source choose Advanced.

- Write Small signal AC analysis, AC amplitude: 1 volt.

- In Edit Simulation Command screen click AC analysis tab.

- Use Type of Sweep=Decade, Number of points per decade=100.
- Write Start (1Meg for 1 MHz) and Stop (30Meg for 30 MHz) frequencies.
- Place the .ac dec 100 1Meg 30Meg command at the schematic. At this time, you should have the schematic shown in Fig. A.7.

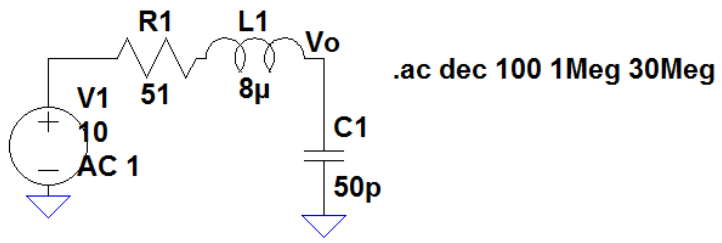



Figure A.7: LTSpice schematic of the *RLC* circuit.

- Press Run command. You should have the graph as shown in Fig. A.8. The left Y-axis shows the magnitude in dB, while the right Y-axis shows

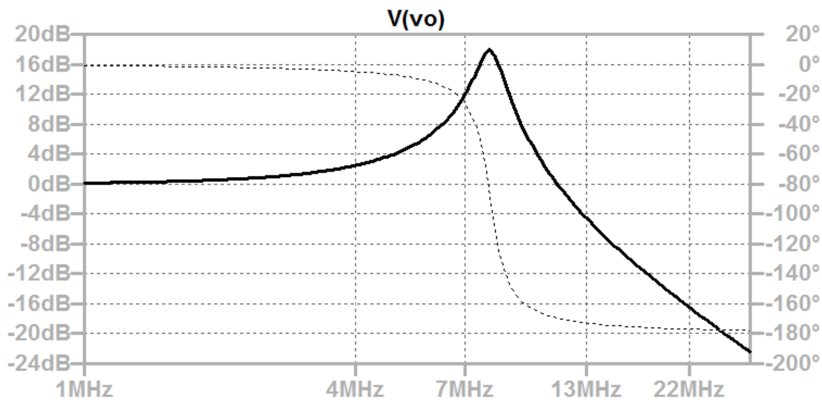



Figure A.8: LTSpice simulation of the transfer function amplitude (solid) and phase (dotted) of the *RLC* circuit.

the phase shift in degrees.

- You can find the 3-dB bandwidth of the circuit by CTRL-Left click on the voltage expression on the graph window. The resulting list shows the peak value and the 3-dB bandwidth with respect to that peak value.
- In Plot settings, click Grid.
- You can change the graph properties in Tools → Control Panel → Waveforms

- Copy the plot using Tools → Copy bitmap to Clipboard to paste it into another application.

## A.6 Analysis of an OPAMP circuit

Consider an inverting amplifier with a gain of 10 built with an OPAMP, LT1413. The OPAMP is powered with ±12 V supplies .

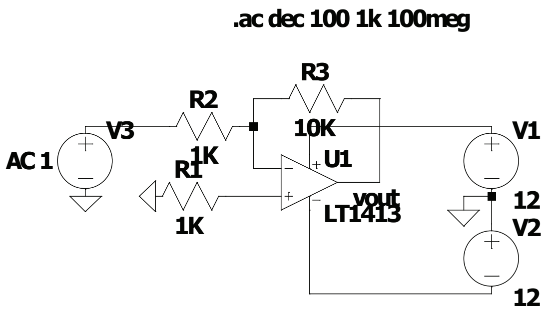



Figure A.9: LTSpice schematic of the OPAMP circuit for AC analysis.

- Draw the schematic shown in Fig. A.9. To choose the OPAMP, click on Component icon, choose [Opamps]. Then click on LT1413. Note that LTSpice contains the models for components produced by Analog Devices only. You can add other models if SPICE models are available.
- To perform a small-signal AC analysis, define Small signal AC analysis, AC amplitude: 1. Here, input is specified as unity, so that the output voltage gives the gain directly. An output voltage of 10 does not mean that we have a peak voltage of 10 V, it means we have a gain of 20. Since we are doing small-signal analysis, the saturation voltages of OPAMP does not come into picture. On the other hand, the supply voltages must be present in order to get a correct answer.
- In Edit Simulation Command screen click AC analysis tab.
- Use Type of Sweep=Decade, Number of points per decade=100.
- Write Start (1K for 1 KHz) and Stop (100Meg for 100 MHz) frequencies.
- Place the .ac dec 100 1Meg 30Meg command at the schematic.
- Press Run command. You should have the graph as shown in Fig. A.10. The gain at low frequencies is 20 dB as expected. But the gain drops above 100 KHz, because of the OPAMP limitations. The phase shift at low frequencies is nearly $180^o$, indicating the inversion.
- To perform a large signal analysis, enter a large signal excitation to the input source by Right-Click: SINE, Amplitude[V]: 1.5, Freq[Hz]: 1K

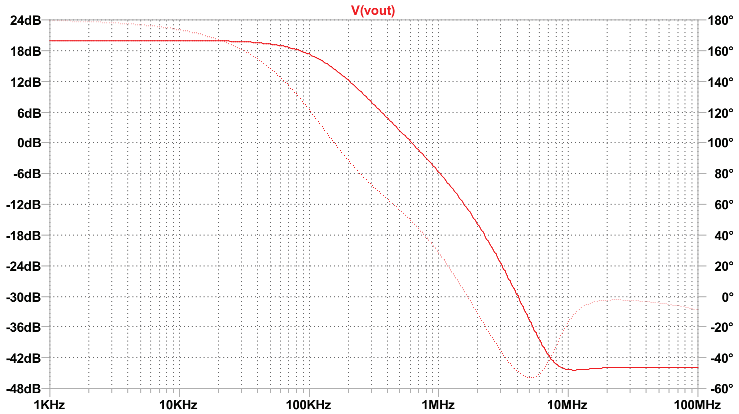



Figure A.10: LTSpice simulation of the transfer function amplitude (solid) and phase (dotted) of the inverting amplifier circuit.

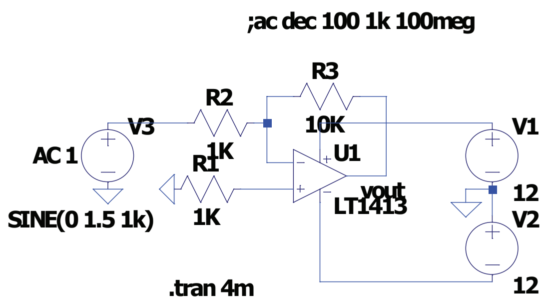



Figure A.11: LTSpice schematic of the OPAMP circuit for transient analysis.

- In Edit Simulation Command, click Transient analysis tab. Set the Stop time to 4m, to see four cycles. The schematic window should look like in Fig. A.11.
- Press Run command. You should now have the graph showing the clipped sine wave as shown in Fig. A.12.

## A.7 Analysis of a BJT circuit

Consider a BJT amplifier built with 2N2222.

- Draw the schematic shown in Fig. A.13. To choose the BJT, click on Component icon, choose npn. Then right-click on BJT. Click Pick New Transistor. Choose 2N2222.
- First perform a transient analysis to make sure that the BJT operates in ACT region. For the input source, choose Sine wave in the Advanced tab. Choose the frequency as 1K (1 KHz) and the peak amplitude as 10m (10 mV). Perform a transient analysis for 10m (10 ms). Observe that the

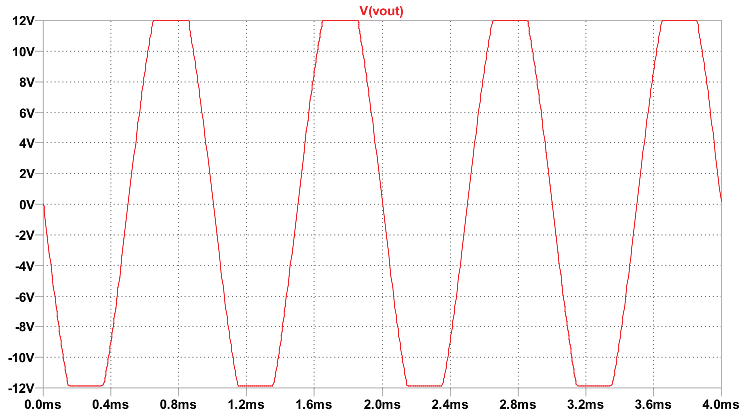



Figure A.12: LTSpice transient simulation of the OPAMP circuit.

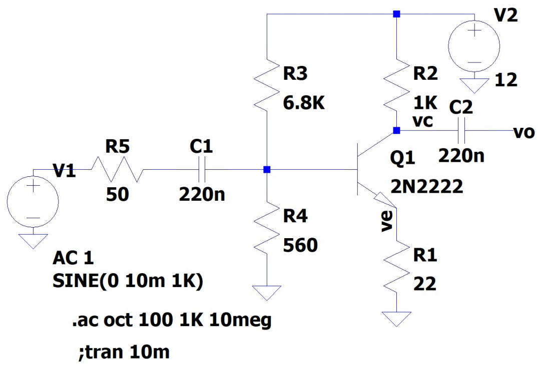



Figure A.13: LTSpice schematic of the BJT amplifier circuit for transient and AC analysis.

collector-to-emitter voltage (V(vc)-V(ve)) is greater than the saturation voltage (about 0.2 V) as shown in Fig. A.14.

- To see the saturation, enter a larger signal excitation to the input source by Right-Click: SINE, Amplitude[V]: 1, Freq[Hz]: 1K. Click Run to get the waveform shown in Fig. A.15 as the collector voltage.
- To perform a small-signal AC analysis, set the input voltage source in the Advanced tab as Small signal AC analysis, AC amplitude: 1.
- In Edit Simulation Command screen click AC analysis tab.
- Use Type of Sweep=Octave, Number of points per decade=100.
- Write Start (1K for 1 KHz) and Stop (10Meg for 10 MHz) frequencies.
- Place the .ac oct 100 1K 10Meg command at the schematic.
- Press Run command. You should have the graph as shown in Fig. A.16. There is a 3-dB corner frequency of gain at low frequencies around 1.4 KHz.

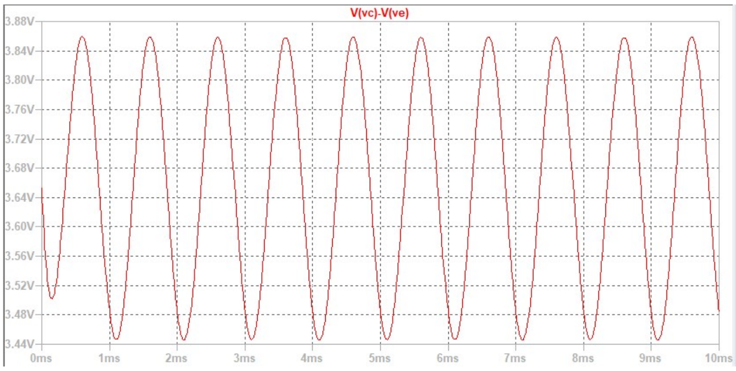



Figure A.14: LTSpice simulation of the transient response of the BJT amplifier.

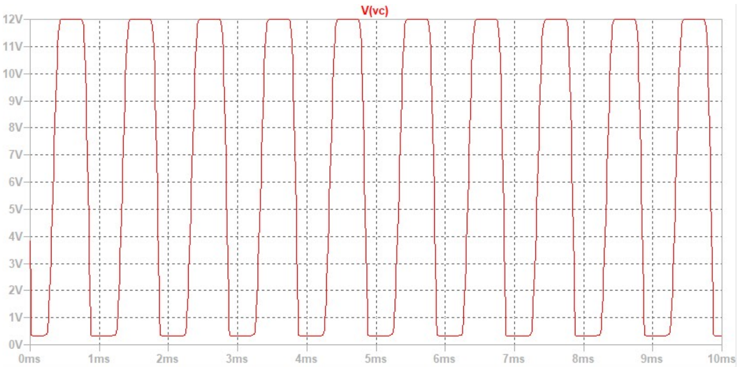



Figure A.15: LTSpice simulation of the BJT amplifier with a larger input signal.

This is due to high-pass response of C1, the DC-block capacitor. In the mid-band, the gain is about 31 dB. The high frequency 3-dB corner frequency is about 10 MHz because of the internal capacitors of BJT. The phase shift at mid frequencies is nearly $180^o$, indicating the inversion of the gain.

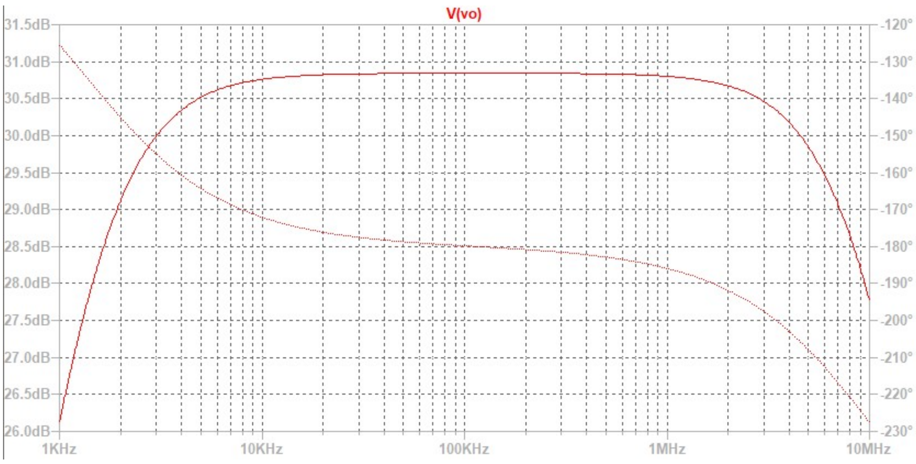



Figure A.16: LTSpice simulation of the gain amplitude (solid) and phase (dotted) of the BJT amplifier circuit.

# Appendix B

# Significant figures

Significant figures of a number indicate its precision. In Electrical Engineering two or three significant figures are typically sufficient in most cases. Rules for identifying significant figures:

1. All non-zero digits or zeros appearing between two non-zero digits are significant. For example, 53.4 V and 3.01 mA have 3 s.f. (three significant figures).

2. Leading zeros are not significant. For example, 0.000507 V has 3 s.f.

3. Trailing zeros in a decimal point number after the decimal point are significant. For example, 1.00 $\mu$A, 2.30 k$\Omega$ or 0.00540 mS all have 3 s.f.

4. There may be an ambiguity in the trailing zeros of a number not containing a decimal point. For example, it is not clear whether 7800 $\Omega$ has two, three or four significant figures. To resolve this problem

   - A decimal point may be added after the number. For example, 7800. $\Omega$ has 4 s.f.
   - An appropriate unit may be used to add a decimal point. For example, use 7.80 k$\Omega$ or 7.8 k$\Omega$ instead of 7800 $\Omega$ to signify 3 s.f. or 2 s.f., respectively.
   - Scientific notation can be used. For example, $7.8 \cdot 10^3$ $\Omega$ has 2 s.f.

The rules for determining the number of significant figures in a given problem can be stated as

1. For addition and subtraction of numbers with known significant figures, the result should have as many *decimal* places as the number with the smallest number of *decimal* places. For example, $145.2 + 1.245 = 146.4$ or $4.78 \cdot 10^{-2} - 2.678 \cdot 10^{-4} = 4.75 \cdot 10^{-2}$

2. For multiplication and division of numbers with known significant figures, the results should have as many significant figures as the number with the smallest number of significant figures. For example, $20.11 \times 1.2 = 24$ or $1.854 \cdot 10^{-8}/5.35 \cdot 10^4 = 3.47 \cdot 10^{-13}$. Hence, writing $10.1\sqrt{2}10.1 = 14.284$ is poor engineering. One should write $10.1\sqrt{2} = 14.3$.

## Unit prefixes

| Symbol | Name | Magnitude | Symbol | Name | Magnitude |
|---|---|---|---|---|---|
| y | yocto | $10^{-24}$ | da | deca | $10^{1}$ |
| z | zepto | $10^{-21}$ | h | hecto | $10^{2}$ |
| a | atto | $10^{-18}$ | k | kilo | $10^{3}$ |
| f | femto | $10^{-15}$ | M | mega | $10^{6}$ |
| p | pico | $10^{-12}$ | G | giga | $10^{9}$ |
| n | nano | $10^{-9}$ | T | tera | $10^{12}$ |
| $\mu$ | micro | $10^{-6}$ | P | peta | $10^{15}$ |
| m | milli | $10^{-3}$ | E | exa | $10^{18}$ |
| c | centi | $10^{-2}$ | Z | zetta | $10^{21}$ |
| d | deci | $10^{-1}$ | Y | yotta | $10^{24}$ |

# Appendix C

# Answers to selected problems

## Chapter 1

1. $\lambda$=17 m and $\lambda$=1.7 cm
3. 34 dB.
4. $2.0\cos(\omega_o + \omega_s)t + 2\cos(\omega_o - \omega_s)t$
5. Mixer output:

$$\frac{1}{2}(\cos(\omega_o - \omega_s)t + \cos(\omega_o + \omega_s)t) + \cos(\omega_o - 2\omega_s)t + \cos(\omega_o + 2\omega_s)t$$
$$+\frac{3}{2}(\cos(\omega_o - 3\omega_s)t + \cos(\omega_o + 3\omega_s)t) + 2(\cos(\omega_o - 4\omega_s)t + \cos(\omega_o + 4\omega_s)t)$$

## Chapter 2

1. (a) 5.6 k±10%, (b) 390 k±10%, (c) 750 Ω±5%
2. $P_R$=0.27 W, $P_V$=0.27 W, $P_I$=0.
3. $5/\sqrt{3}$.
4. (a) 46 Ω (b) 42 Ω (c) 8.4 k (d) 2.0 k (e) 49 Ω
5. (a) 7.1 (b) 0.99 (c) 20.
6. (a) 159 Hz (b) 50.0 Hz (d) $\omega/2\pi$
7. 58 hrs, 8.7 kilojoules
8. $V_1$=4.53 V, $V_2$=6.11 V
9. $I_1$=0.25 $\mu$A

10. (a) $I_1$=10.9 mA, $I_2$=1.07 mA, $V_1$=54.6 mV, (b) $I_1$=2.96 mA, $I_2$=0.926 mA, $V_1$=2.04 V, (c) $I_1$=96.5 mA, $I_2$=1.66 mA, $V_1$=12.6 V, (d) $I_1$=-1.29 mA, $V_1$=-4.27 V, (e) $I_1$=-0.467 mA, $I_2$=-7.85 mA, $V_1$=-1.54 V

11. 
$$i_L(t)(\text{mA}) = \begin{cases} -89.2 + 119.2e^{-t/21.3} & \text{for } 0 < t < 40\ \mu\text{s} \\ 0.8 - 71.8e^{-(t-40)/21.3} & \text{for } t \geq 40\ \mu\text{s} \end{cases}$$
$$v_R(t) = \begin{cases} -12 - 111.7e^{-t/21.3} & \text{for } 0 < t < 40\ \mu\text{s} \\ -12 + 1077e^{-(t-40)/1.33} & \text{for } t \geq 40\ \mu\text{s} \end{cases}$$

12. 
$$i_L(t)(\text{mA}) = \begin{cases} 0.1 - 0.095e^{-t/416} & \text{for } 0 < t < 100\ \mu\text{s} \\ -0.1 + 0.1253e^{-(t-100)/416} & \text{for } t \geq 100\ \mu\text{s} \end{cases}$$

13. (a) $v_C(t) = I_S R - I_S R e^{-t/RC}$ for $t \geq 0$. $i_C(t) = I_S e^{-t/RC}$ for $t \geq 0$. (b) $i_L(t) = I_S - I_S e^{-t/(L/R)}$ for $t \geq 0$. $v_L(t) = I_S R e^{-t/(L/R)}$ for $t \geq 0$.

14. 1070 Ω

15. Mass of the three lines: 1.57 t. Power loss over 200 km: 84.4 kW. With 34.5 kV, the mass should be 190 t!

16. $R_1/R_2$=7/8=0.875. $R_1$=3.3 k, $R_2$=3.9 k. Using 5% tolerances: Max $V_{out}$=8.50 V, Min $V_{out}$=7.75 V, Error= +6% −3%.

17. $(C_1 + C_2)/C_1$=12/5=2.4, $C_2/C_1$=1.4, $C_2$=47 nF, $C_1$=33 nF. Max $V_{out}$=5.54 V, Min $V_{out}$=4.38 V, Error= +11% −12%.

18. 137 pF

19. (a) $i_L(0^-) = -2.0$ mA, $v_L(0^-)$ =0.0, (b) $i_L(0^+) = -2.0$ mA, $v_L(0^+)$ =3.3 V, (c) $v_L(t) = 0.0$ for $t < 0$, and $v_L(t) = 3.3e^{-t/4.5\mu}$ for $t \geq 0$.

20. Voltage gain=20. dB. When $R_{in} = R_L$, power gain=100=20. dB. When $R_{in} = 10R_L$, power gain=1000=30. dB.

21. Using Eq. 2.58
$$i_L(t) = \begin{cases} 10^5 t & \text{for } 0 < t \leq 5\ \mu\text{s} \\ 0.5\ A & \text{for } t \geq 5\ \mu\text{s} \end{cases}$$

22. Using Eq. 2.43
$$v_C(t) = \begin{cases} -2 + 500t & \text{for } 0 < t \leq 12\text{ms} \\ 4\ V & \text{for } t \geq 12\text{ ms} \end{cases}$$

# Chapter 3

1. (a) -59.1-j9.39=59.9$\angle(-171^o)$, (b) -2.43+j3.11=3.94$\angle 128^o$,
   (c) 0.630+j3.92=3.97$\angle 80.9^o$,
   (d) $2.54{\cdot}10^{-3} - j4.58 \cdot 10^{-4} = 2.58 \cdot 10^{-3}\angle(-10.2^o)$

8. (a) $140\angle -0.79$, (b) $33\angle -0.79$, (c) $f = 0$: 94; $f = 100$ kHz: $93\angle -0.069$; $f = 500$ kHz: $82\angle -0.27$; $f = 1$ MHz: $67\angle -0.34$; $f = 1.44$ MHz: $59\angle -0.32$; $f = 5$ MHz: $48\angle -0.14$; (d) $140\angle 0.78$, (e) $f = 1$ MHz: $150\angle 0.020$; $f = 19$ MHz: $180\angle 0.16$; (f) $f = 1$ MHz: $12\angle 1.6$; $f = 36.5$ MHz: $390\angle 1.4$; $f = 73$ MHz: $720\angle 1.5$.

9. For $V$=5$\angle\pi/6$ V: (a) $I$=0.036$\angle 1.3 \Rightarrow i(t) = 0.036\cos(2\pi 7.2 \cdot 10^6 t + 1.3)$. (b) $I$=0.15$\angle 1.3 \Rightarrow i(t) = 0.15\cos(2\pi 720 \cdot 10^3 t + 1.3)$. (d) $I$=0.036$\angle -0.26$ $\Rightarrow i(t) = 0.036\cos(2\pi 7.2 \cdot 10^6 t - 0.26)$. (e) $f = 1$ MHz: $I$=0.0333$\angle 28.9^o$ $\Rightarrow i(t) = 0.0333\cos(2\pi 10^6 t + 28.9^o)$.

   For $I$=2$\angle 90^o$ mA: (a) $V$=0.286$\angle 135 \Rightarrow v(t) = 0.286\cos(2\pi 7.2 \cdot 10^6 t + 135^o)$. (b) $V$=66.5$\angle 45^o$ mV $\Rightarrow v(t) = 66.5\cos(2\pi 720 \cdot 10^3 t + 45^o)$. (d) $V = 0.282\angle 135^o \Rightarrow v(t) = 0.282\cos(2\pi 7.2 \cdot 10^6 t + 135^o)$.

   For $V = 3.77\angle 138.5^o$: (a) $I$=0.027$\angle -176^o \Rightarrow i(t) = 0.027\cos(2\pi 7.2 \cdot 10^6 t - 176^o)$. (b) $I$=0.114$\angle -176^o \Rightarrow i(t) = 0.114\cos(2\pi 720 \cdot 10^3 t - 176^o)$.

10. 8(e) with 9(a): 1 MHz: $I_L = 33\angle 26.5^o$ mA; 19 MHz: $I_L = 19\angle 24^o$ mA
    8(e) with 9(b): 1 MHz: $I_L = 2\angle 87^o$ mA; 19 MHz: $I_L = 1.5\angle 45^o$ mA
    8(e) with 9(c): 1 MHz: $I_L = 25\angle 135^o$ mA; 19 MHz: $I_L = 15\angle 86^o$ mA

11. (a) 2.5 k in series with 0.69 H. (b) 541 Ω in series with 1.7 $\mu$F (c) 975 Ω in series with 27 nF (d) 707 Ω in series with 1.4 nF (e) 43.6 Ω in series with 320 pF.

12. 25 MHz: $58.3\cos(2\pi 25 \cdot 10^6 t - 45.5^o)$ mA.
    50 MHz: $36.8\cos(2\pi 50 \cdot 10^6 t - 63.8^o)$ mA.

13. $f$=1.33 MHz

14. R = 587 kΩ

15. $f = 77.3$ kHz

16. (a) 2.5 kΩ in parallel with 23 H. (d) 1.4 kΩ in parallel with 707 pF.

17. (b) $V_{TH}$ = 2.12 V, $Z_{eq}$ = 396$\angle{-\pi/4}$ Ω. (c) $V_{TH}$ = 0.446$\angle 63.1^o$ V, $Z_{eq}$ = 892$\angle{-26.5^o}$ Ω. (d) $V_{TH}$=1.91 V, $R_{eq}$=1.81 kΩ. (f) $V_{TH}$ = 0.79 V, $R_{eq}$ = 808 Ω. (k) $V_{TH}$ = 0.26 V, $R_{eq}$ = 1.3 kΩ

18. (b) $I_N$ = 5.4$\angle\pi/4$ mA, $Z_{eq}$ = 396$\angle{-\pi/4}$ Ω. (c) $I_N$ = 0.5$\angle 89.6^o$ mA, $Z_{eq}$ = 892$\angle{-26.5^o}$ Ω. (d) $I_N$=2.07 mA, $R_{eq}$=1.81 kΩ. (f) $I_N$ = 0.98 mA, $R_{eq}$ = 808 Ω. (k) $I_N$ = 0.2 mA, $R_{eq}$ = 1.3 kΩ

19. (a) $I_N = I(\omega)/2$, $Z_{eq}$ = 97 Ω. (b) $I_N = \dfrac{V(\omega)}{2R}$,
    $$Z_{eq} = \frac{2R\omega L}{\sqrt{4R^2+\omega^2 L^2}}\angle\left(\tan^{-1}\frac{2R}{\omega L}\right)$$

20. $V_{eq} = 0.042 \cdot V(\omega)$, $Z_{eq} = 49\ \Omega$

21. (c) At DC: $V_{TH} = V_{in}$, $Z_{eq} = 50\ \Omega$.
    At 20 MHz: $V_{TH} = 0.57\angle{-0.96} \cdot V_{in}$, $Z_{eq} = 50.3\angle 1.24\ \Omega$.
    At 40 MHz: $V_{TH} = 0.34\angle{-1.2} \cdot V_{in}$, $Z_{eq} = 125\angle 1.5\ \Omega$.

22. (a) $V_{out} = 1$ V, (b) $V_{out} = 0.17$ V, (c) $V_{out} = 1$ V

25. (b) $V_o/V_{in}$=3.14 (c) $V_o/V_{in}$=-14.4 (d) $V_o = -2.55\ V_1 - 3.73V_2$
    (e) $V_o = 4.73V_1 - 3.73V_2$ (f) $V_o = 1.67V_1 - 3.73V_2$ (g) $V_o = -102V_{in}$
    (h) $V_o = -10400V_{in}$.

26. (b) $\frac{V_o}{V_{in}} = -3.13 + j\frac{6.44\cdot 10^5}{\omega}$ (e) $\frac{V_o}{V_{in}} = \frac{4.13}{1+j\omega 4.95\cdot 10^{-7}}$
    (g) $\frac{V_o}{V_{in}} = -(2 + j\omega 1.6 \cdot 10^{-6})$

27. (e) For $f \to 0$, $\frac{V_o}{V_{in}} \to 4.13$; for $f \to \infty$, $\frac{V_o}{V_{in}} \to -j\frac{8.34\cdot 10^6}{\omega}$

28. $f$=876 kHz.

29.
$$V(t) = V_{dc}\frac{1 + \Delta(t)/d_o}{1 + (C_c/(C_o + C_c))(\Delta(t)/d_o)} \approx V_{dc}\left(1 + \frac{C_o}{C_o + C_c}\frac{\Delta(t)}{d_o}\right)$$

# Chapter 4

1.
$$v_C(t) = \begin{cases} 2.3 - 2.3e^{-t/0.002} & \text{for } 0 < t \le 0.0020 \\ 1.45 & \text{for } t \ge 0.0020 \end{cases}$$

2. Using Eq. 2.58
$$i_C = L(t) = \begin{cases} 7 \cdot 10^4 t & \text{for } 0 < t \le 5\ \mu\text{s} \\ 0.35 - 5.7 \cdot 10^4(t - 5 \cdot 10^{-6})) & \text{for } 5\mu\text{s} \le t \le 11.14\ \mu\text{s} \\ 0 & \text{for } t > 11.14\ \mu\text{s} \end{cases}$$

3. From the voltage doubler circuit of Fig. 4.40 in p. 172, we have $V_{C1} = 9.4$ V and $V_{C2} = 18.8$ V. Hence $v_1 = 10\sin(\omega t) - 9.4$. $C_3$ and $D_3$ forms a voltage clamper for this input ⇒ $V_{C3}$=18.8 V. $D_4$ and $C_4$ forms a half-wave rectifier ⇒ $V_{C4} = 18.8$ V. The output voltage is $v_o = V_{C2} + V_{C4}$=37.6 V. This circuit is a *voltage quadrupler*.

4. (a) ACT, 6.92 V, (b) ACT, 6.72 V, (c) ACT, 6.61 V.

5. (a) ACT, 6.17 V, (b) ACT, 5.80 V, (c) ACT, 5.59 V.

6. (a) ACT, 2.34 V, (b) ACT, 2.68 V, (c) ACT, 2.82 V.

7. ACT, 8.75 V.

8. SAT, −4.28 V.

9. ACT, 6.96 V.

10. ACT, $A_V$=125.
11. ACT, $A_V$=281.
13. $A_V$=4.43.
14. $A_V$=0.99.

# Chapter 5

2. $V_R = 5.94\angle 53.5^o$, $V_L = 104\angle 143.5^o$, $V_C = 112.5\angle -36.5$.
3. $V_R = 10\angle 0$, $V_L = 51\angle 90^o$, $V_C = 51\angle -90^o$.
4. $C$=98.9 pF
5. $\omega_o = 95.35 \cdot 10^6$ rps $\omega_1 = 85.87 \cdot 10^6$ rps $\omega_2 = 105.87 \cdot 10^6$ rps
7. $$\omega_o = \left[\frac{1}{LC} - \left(\frac{R_L}{L}\right)^2\right]^{1/2}$$
   If $R_S > \sqrt{L/C}$, no resonance occurs.
9. $Q$=10.3
10. $R$=150 Ω, $C$=211 pF.
11. $C$=2.27 nF. 176 mW.
12. For $R$=0 $f_o$=15.915 MHz. For $R$=1 $f_o$=15.914 MHz. For $R$=100 no resonance.
13. $f_o$=1.33 MHz.
14. $C$=24.8 pF. $v_R(t) = 5\cos\omega_o t$, $v_L(t) = 11.4\cos(\omega_o t + 90^o)$, $v_C(t) = 11.4\cos(\omega_o t - 90^o)$.
15. Lower 3-dB: $f_1$=22.6 MHz, $v_R(t) = 3.5\cos(\omega_1 t - 45^o)$, $v_L(t) = 6.5\cos(\omega_1 t + 45^o)$, $v_C(t) = 10.0\cos(\omega_1 t - 135^o)$.
    Upper 3-dB: $f_1$=34.8 MHz, $v_R(t) = 3.5\cos(\omega_1 t + 45^o)$, $v_L(t) = 10.0\cos(\omega_1 t + 135^o)$, $v_C(t) = 6.5\cos(\omega_1 t - 45^o)$.
16. $R_{DC}$=54 mΩ. $f_{sc}$=2 MHz. $R$=200 mΩ at 28 MHz.
17. $l$=9.8 mm.
18. $N$=9.
19. 18 turns
20. $A_L$=2.65 nH/T$^2$. $R$=0.24 Ω.
23. At $f$=400 kHz, $V_{out}/V_{in}$=$9.6\angle -29^o$.
    At $f$=1 kHz, $V_{out}/V_{in}$=$11\angle -0.08^o$.
24. At $f$=1 MHz, $V_{out}$=$4.08\angle -172^o$.
    At $f$=1 kHz, $V_{out}$=$4.4\angle -150^o$.
25. $C$=99 pF $|B_1/A_1| = 0.09$ $|B_2/A_2| = 0.007$

# Chapter 6

1. $V_{out}$=1.52$\angle-0.1^o$, $I_C = 12\angle 89.9^o$
2. $C_1 = C_3 = 53$ nF $L_2 = 0.1$ H
3. $V_{out}$=0.353$\angle 0^o$, $I_L = 3.59\angle-90^o$
4. 2.96 kHz.
5. $C_1 = C_3$=707 pF, $L_2$=2 $\mu$H
6. $C_1 = C_5$=150 pF, $L_2 = L_4$=688 nF $C_3$=340 pF
7. $C_1$=2.1 nF $L_1$=13.3 nH
8. 75 $\Omega$.
9. $Z$=48.2$\angle 50^o$ $50^o$
10. $I_t = 26.7\angle 0$ $I_m = 31.8\angle-90^o$
11. $Z = 109\angle 38.5^o$, $38.5^o$.
12. $C = 32.3$ pF, $Z = 140$, $0^o$.
13. A transformer with 22 turns of primary and 44 turns of secondary. $H(\omega) = 1$ at 16 MHz.
14. $L_S$=2.76 $\mu$H, $C$=276 pF. $f_1$=3.77 MHz, $f_2$=6.65 MHz.
15. $R_1 = R_2 = 1120$ $\Omega$, $C_i = 14$ pF.
16. $Z_{in} = 40 + j40$.

# Chapter 7

1. $C < 0.23$ $\mu$F.

3(a). For $\cos(\omega t) > 0 \Rightarrow V_{out}$=0
For $\cos(\omega t) < 0 \Rightarrow V_{out}$=$3\cos(\omega t)$

3(b). For $\cos(\omega t) > 0 \Rightarrow V_{out}$=$1.5\cos(\omega t)$
For $\cos(\omega t) < 0 \Rightarrow V_{out}$=$3\cos(\omega t)$

4(a). For $I = I_{min}$=1.2 mA $\Rightarrow V_{out}$=$0.41(\cos\omega t + 1)$
For $I$=0.7 mA $\Rightarrow V_{out}$ is half-wave rectified

4(b). For $I = I_{min}$=88 $\mu$A $\Rightarrow V_{out}$=$0.41(\cos\omega t + 1)$
For $I$=0 mA $\Rightarrow V_{out}$ is half-wave rectified

5. $|V_{out}/V_{in}|$=2.38 when $I_{DC}$=0.1 mA, $|V_{out}/V_{in}|$=9.64 when $I_{DC}$=10 mA.
6. $R$=1.27 $\Omega$. Efficiency=62 lumens/W

# Chapter 8

2. $H(\omega) = j\omega R_3C_3/(1 - \omega^2 R_3^2 C_3^2 + 3j\omega R_3C_3)$, $\omega_o = 1/R_3C_3$, $|H(\omega_o)| = 1/3$
3. $R_2/R_3 > 2$.
4. With 1 V bias: 27.46 MHz, with 10 V bias: 27.80 MHz.
5. $L$=29.8 $\mu$H, $r$=23.4 k$\Omega$.
6. $L$=24.8 $\mu$H, $r$=148 $\Omega$.

# Chapter 9

1. 0.73 mV
2. $2l$=209 cm, $C_d$=6.66 pF. $L$=5.22 $\mu$H. $Q$=126.

# Bibliography


[1] D. Rutledge, *The Electronics of Radio.* Cambridge University Press, 1999.

[2] P. Nahin, *The Science of Radio: With MATLAB and Electronics Workbench demonstrations, 2nd Edition.* Springer-Verlag, 2001.

[3] ARRL, *The ARRL Handbook for Radio Communications 2017*, 2017.

[4] R. White and R. Doering, *Electrical Engineering Uncovered*, 2nd ed. Pearson, 2001.

[5] N. Balabanian, *Electric Circuits.* McGraw-Hill, 1995.

[6] S. Sze and M. Lee, *Semiconductor Devices: Physics and Technology*, 3rd ed. John Wiley & Sons, Inc., 2012.

[7] R. Gayakwad, *Op-Amps and linear integrated circuits*, 4th ed. Prentice-Hall, 1999.

[8] K. Blair, *Audio Engineering Handbook.* McGraw-Hill, 1988.

[9] "Ferroxcube home page," http://www.ferroxcube.com, accessed: 2017-10-31.

[10] "Micrometals home page," http://www.micrometals.com, accessed: 2017-10-31.

[11] "MMG Canada home page," http://www.mmgaca.com, accessed: 2017-10-31.

[12] "PTI home page," http://www.piezotech.com, accessed: 2017-10-31.

[13] K. Clarke and D. Hess, *Communication Circuits: Analysis and Design*, 2nd ed. Krieger Pub Co., 1994.

[14] R. Lourens, "Practical PICmicro oscillator analysis and design," http://ww1.microchip.com/downloads/en/AppNotes/00943A.pdf, accessed: 2017-10-31.

[15] R. Johnson, *Antenna Engineering Handbook.* McGraw-Hill, 1993.

[16] ARRL, *Antenna Engineering Handbook.*

[17] C. Balanis, *Antenna Theory: Analysis and Design.* Wiley, 2016.

[18] "Vertically polarized dipoles and monopoles, directivity, effective height, and antenna factor," *IEEE Trans. on Broadcasting*, vol. 56, pp. 379–409.

[19] "What is ham radio?" http://www.arrl.org/what-is-ham-radio, accessed: 2018-01-08.

[20] J. Taylor, "WSJT-X user guide," http://physics.princeton.edu/pulsar/k1jt/wsjtx-doc/wsjtx-main-1.7.0.html, accessed: 2018-01-07.

[21] ——, "The JT65 communications protocol," http://physics.princeton.edu/pulsar/K1JT/JT65.pdf, accessed: 2018-01-07.

# Index