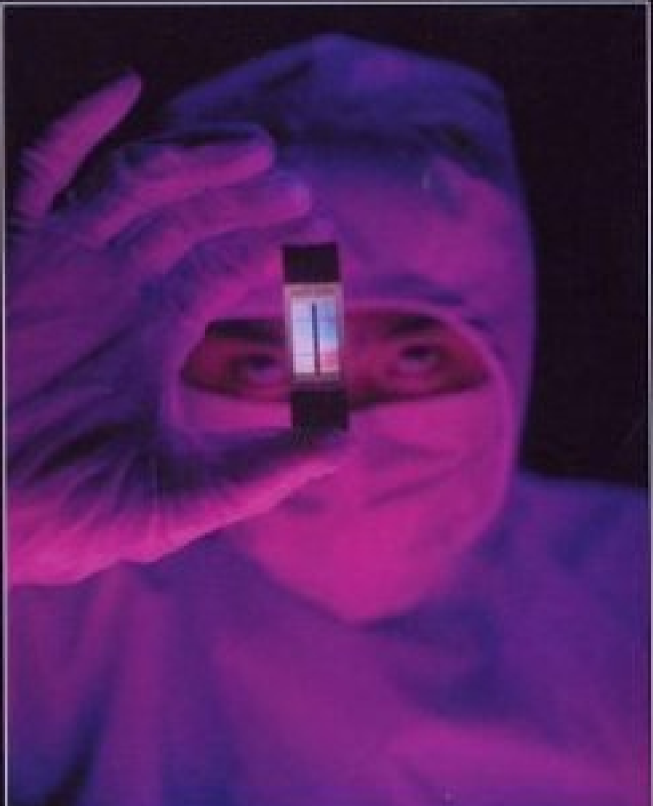
J. DAVID IRWIN

A Brief Introduction to Circuit Analysis



A Brief Introduction to Circuit Analysis

J. David Irwin

AUBURN UNIVERSITY

WILEY

To my loving family:
Edie
Geri, Bruno, Andrew and Ryan
John, Julie, John David and Abi
Laura

Contents

Preface

Overview

Although circuit analysis is fundamental to much of the material in electrical and computer engineering curricula, the circuits sequence in these programs, at universities throughout the world, appears to be occupying a smaller part of the curriculum. The faculty are caught among several constraints: a reduction in the number of hours for graduation, the potential or actual growth of the university's core curriculum, and the need to cover more material in a technology that seems to advance daily. In view of these very real issues, some faculty have made the decision to confine circuits in the curriculum to a single semester course. Thus, it would be advantageous to have a book that covers the set of required topics in such a course, no more and no less. I have designed this book for that purpose.

This book was organized by judiciously selecting chapters from the seventh edition of *Basic Engineering Circuit Analysis*. The pedagogy of that book has been tested over and over. Furthermore, students indicate that it is a book they feel comfortable studying on their own. Thus the proper selection of chapters from *Basic Engineering Circuit Analysis*, 7th Ed., should produce a short book that enhances the benefits of a one-semester course.

Perhaps the primary issue in producing a shorter book is the decision of what to include and what to leave out. There are undoubtedly a variety of opinions on this issue. In a one-semester course, some faculty will cover one set of topics and another faculty will cover a different set—both with excellent reasons for their selection. However, I believe that this book contains the essential topics that will most likely be selected for inclusion in such a one-semester course. The chapter titles for the book are

1. Basic Concepts
2. Resistive Circuits
3. Nodal and Loop Analysis Techniques
4. Additional Analysis Techniques
5. Capacitance and Inductance
6. First- and Second-Order Transient Circuits
7. AC Steady-State Analysis
8. Variable-Frequency Network Performance

This set of chapters covers the resistive circuits, the standard analysis techniques, the capacitor and inductor components, transient analysis, ac steady-state analysis, and the manner in which networks perform as a function of frequency, including such topics as resonance and various kinds of active and passive filters. Although this streamlined approach does not cover such topics as magnetic circuits, electric power circuits, and transform methods, these topics will typically be covered elsewhere in the curriculum, and thus if some topics are to be omitted, these appear to be logical candidates.

This brief book covers all the topics that are basic to an understanding of circuit analysis and, furthermore, many of them are addressed in a just-in-time manner. One important example of this type of coverage is the manner in which the operational amplifier is presented. For example, once it is introduced and the various standard circuits are covered in Chapter 3, these op-amps are used to design circuits in Chapter 4, employed to derive differentiator and integrator circuits in Chapter 5, and finally used in conjunction with the new operational transconductance amplifier (OTA) in Chapter 8 to design active filters.

Because of the intrinsic relationship of this book to the parent text, the features that have made *Basic Engineering Circuit Analysis*, 7th Ed., popular are naturally contained here also. The book's clear and concise explanations, variety of effective learning aids, numerous problems with varying degrees of difficulty, and the number of real-world examples that demonstrate the usefulness of the material have been heralded as components that enhance the use of the text for both students and instructors alike. In addition, although this text is relatively short, it still provides the instructor with great flexibility within the setting for which it is designed. Sections or chapters can be emphasized or skipped in the formation of a coherent presentation. Likewise, the CAD tools can be employed to add a new dimension to the presentation or omitted completely with no loss of continuity.

In the final analysis, the goal of this text is to provide, within the setting of a one-semester course, an effective and efficient mechanism for students to obtain a thorough understanding of the basics of electric circuit analysis and an introduction to design.

Pedagogical Structure Designed to Reinforce Learning

Students don't all learn in the same way. Some are visual learners, while others are more kinesthetic. A "learning styles survey" appears after the preface to help each student determine his or her particular learning style and gives guidance on how to tailor his or her study habits. Pedagogical features are included to fit different learning styles.

- ***Learning Goals***, listed at the beginning of each chapter, provide an overview of the topics within the chapter and the skills and knowledge students should achieve.
- ***Learning Hints*** that appear on many pages of the text help shorten the learning curve. These comments in the margin provide guidance for understanding different facets of the presentation and problems of all types. Coupled with myriad examples, Learning Hints provide readers with a companion tutor. Additionally, they aid the instructor and the student by conveying some of the subtleties that are typically implicit in lecture or in traditional presentation.
- ***Learning Example*** sections, more often than any other component, provide students with the means for acquiring and evaluating new knowledge. The numerous worked-out examples in the text are the hallmark feature.
- An expanded number of real-world examples, labeled ***Learning by Application***, appear in many sections of the text, and at the end of every applicable chapter, answer the question "Why do we study circuit theory?" Applications frequently deal with design issues ranging from very simple matters, such as finding the value of some specific component, to modeling the collapse of the Tacoma Narrows Bridge.
- ***Learning by Doing*** and ***Learning Extensions*** are assessment tools coordinated within the text. The Learning by Doing exercises are quick, simple reinforcements of the principles and provide a check of the reader's understanding of the material. Learning Extensions provide practice for the reader in applying the basic concepts, as well as guidance in understanding the techniques needed to solve the end-of-chapter problems.
- ***Problem-Solving Strategies*** are placed to assist the student in selecting the proper solution technique or combination of techniques applicable in a particular situation. This

assistance not only helps the student understand the subtle differences among various techniques in their application to a particular problem, but also helps eliminate the psychological barrier that sometimes exists in determining a suitable method of attack.

- ***Computer-aided Design (CAD) Tools*** allow students, like all modern engineers, to apply the power of the computer to solve a variety of problems. Special icons are employed within the book to indicate sections where the CAD tools are used. The very latest version of PSPICE by Cadence is used, and this version coupled with the use of both MATLAB and Microsoft EXCEL are integrated within the text and coordinated with the Student Study Guide (discussed later) where Electronics Workbench is also introduced.
- ***Learning by Design*** sections appear at the end of each applicable chapter. This feature provides the reader with an understanding of how to apply what they have learned to the design of circuits. The use of engineering design in a curriculum is a major component of the ABET criteria. The inclusion of this material permits its introduction to the student at an early stage in the curriculum.
- ***Learning Check*** includes both the Summary and Problems, and appears at the end of every chapter. The important topics are reviewed concisely in the Summary as a quick reminder for readers. The problems are segmented by chapter subdivision and graduated in difficulty to permit users to test their understanding of the material and hone their skills in solving different types of problems. *The problem sets also include some problems specifically designed to mimic those that appear on the Fundamentals of Engineering (FE) Exam taken by students in preparation for becoming a Registered Professional Engineer.*

Companion Web Site

Among other items, this site contains *Answers to Selected Problems.*

Supplements

The ***Student Study Guide*** for the seventh edition contains additional detailed examples that track the chapter presentation to aid and check the student's understanding of the problem-solving process. Many of these examples involve computer simulations with PSPICE, MATLAB, Microsoft EXCEL, and Electronics Workbench. A CD bound into the study guide includes circuit simulations and five easy-to-use video segments for demonstrating PSPICE solutions.

EGrade Anonymous Quizzing is also available to students using this text. Students are encouraged to visit our Web site at *www.wiley.com/college/circuitsextra* and register to begin taking practice quizzes on eGrade to increase their circuits problem-solving skills. EGrade questions are organized by topic and are automatically scored to provide immediate feedback, so the student can either drill specifically in problem areas (focusing on topics he or she needs more work in) or just do general practice drills to prepare for a test.

Circuits Extra—Check out the latest offerings for users of Wiley circuits texts.

EGrade On-line Assessment is also available for this text. EGrade is a tool that allows instructors to automate the process of assigning, delivering, grading, and routing all kinds of homework, quizzes, and tests, while providing students with immediate scoring and feedback on their work. Electric circuits test banks in eGrade format are available for instructors who would like to include a Web component in their course in the form of on-line homework and quizzing. Questions are arranged by topic and are in a variety of formats, including fill-in-the-blank, multiple choice, true/false, and more. For more information, and to see a demo of eGrade, visit *www.wiley.com/college/egrade*.

The ***Solutions Manual***, containing solutions to all learning extensions and end-of-chapter problems, and *PowerPoint Slides* for this text are available only to instructors who have adopted the text for classroom use. The solutions manual and PowerPoint slides are available on the Web site at *www.wiley.com/college/irwin*, under the *Instructor's Companion Site*. You must first register for a password on-line and supply your course information for confirmation before you will receive access to these resources.

Circuit Solutions powered by ***JustAsk!*** is a Web site that is essentially a tutor serving the needs of both the student and the instructor. Questions and answers are provided for numerous topics within the chapters. Selected problems are worked in detail and explanations of every facet of the solution are provided. As such, this Web site is a valuable tool in the use of this book. On this site you will gain access to ***The Problem Solving Companion***—a $16.00 value free! This companion supplies you with additional problems and complete solutions that are not found in the text and a listing of important formulas.

Acknowledgments

Because every new circuits book builds upon the editions that have preceded it, I am honored and grateful for the support of so many friends and colleagues who have contributed in a variety of ways to the development of this material. I have been blessed by their kindness and fortunate to have their very insightful counsel. Thus, I want to acknowledge with sincere appreciation the following individuals who have contributed in one way or another to this book. My academic colleagues at other universities:

David Anderson, University of Iowa
Jorge Aravena, LSU
Richard Baker, UCLA
John Choma, USC
James L. Dodd, Mississippi State University
John Durkin, University of Akron
Earl D. Eyman, University of Iowa
Ashok Goel, Michigan Technological University
Arvin Grabel, Northeastern University
Paul Greiling, UCLA
Paul Gray, University of Wisconsin, Platteville
Mohammad Habli, University of New Orleans
John Hadjilogiou, Florida Institute of Technology
Marty Kaliski, Cal Poly, San Luis Obispo
Ralph Kinney, LSU
Robert Krueger, University of Wisconsin
K. S. P. Kumar, University of Minnesota
Jung Young Lee, UCB
James Luster, Snow College
Ian McCausland, University of Toronto
Arthur C. Moeller, Marquette University
Darryl Morrell, Arizona State University
M. Paul Murray, Mississippi State University
Burks Oakley, University of Illinois at Champaign-Urbana
John O'Malley, University of Florida
William R. Parkhurst, Wichita State University
Peyton Peebles, University of Florida
Clifford Pollock, Cornell University
George Prans, Manhattan College

James Rowland, University of Kansas
Robert Sackett, Normandale Community College
Richard Sanford, Clarkson University
Peddapullaiah Sannuti, Rutgers University
Ronald Schultz, Cleveland State University
M. E. Shafeei, Penn State University at Harrisburg
Scott F. Smith, Boise State University
Karen M. St. Germaine, University of Nebraska
Janusz Strazyk, Ohio University
Saad Tabet, Florida State University
Val Tareski, North Dakota State University
Leonard J. Tung, Florida A&M University/Florida State University
Seth Wolpert, University of Maine
Darrell Vines, Texas Tech University

My colleagues at Auburn University:

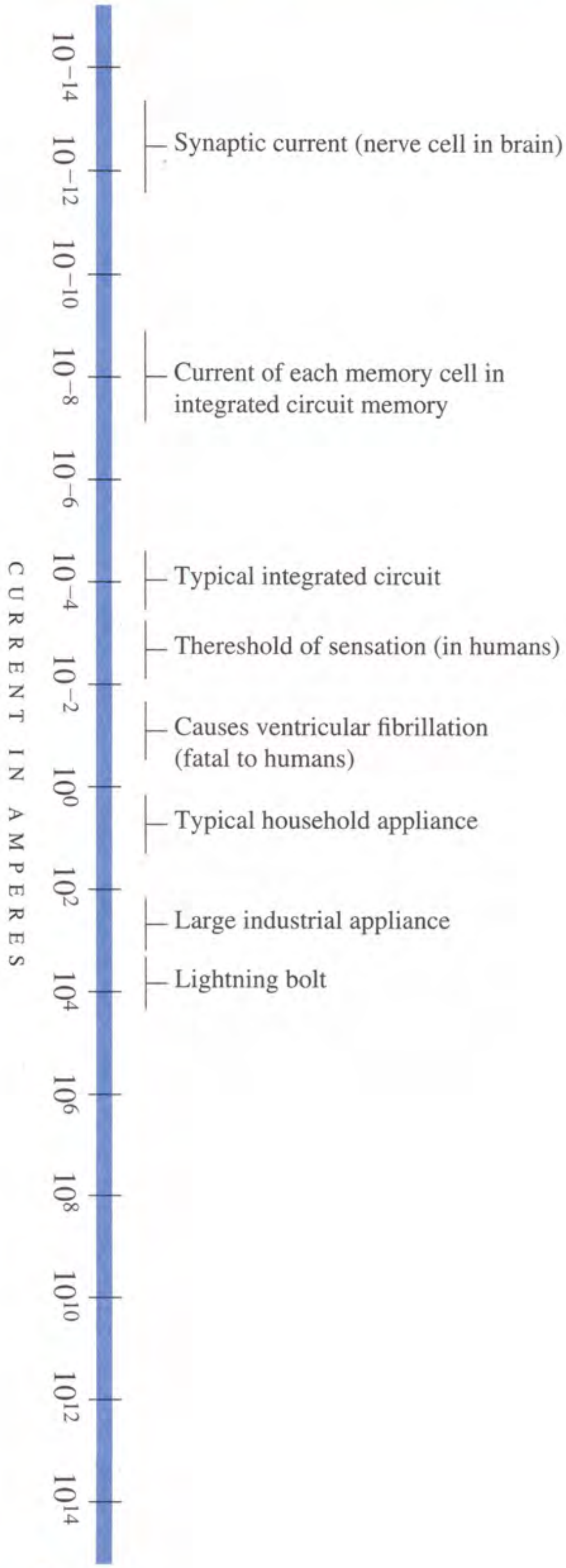
Bill Dillard
R. Mark Nelms
Thomas A. Baginski
Charles A. Gross
James L. Lowry
Zhi Ding
David C. Hill
Henry Cobb
Les Simonton
Betty Kelley
E. R. Graf
L. L. Grigsby
M. A. Honnell
R. C. Jaeger
M. S. Morse
Sung-Won Park
C. L. Rogers
Travis Blalock
Kevin Driscoll
Keith Jones
George Lindsey
David Mack
Paula R. Marino
John Parr
Monty Rickles
James Trivltayakhun
Susan Williamson
Jacinda Woodward

and the team at John Wiley and Sons, Inc.:

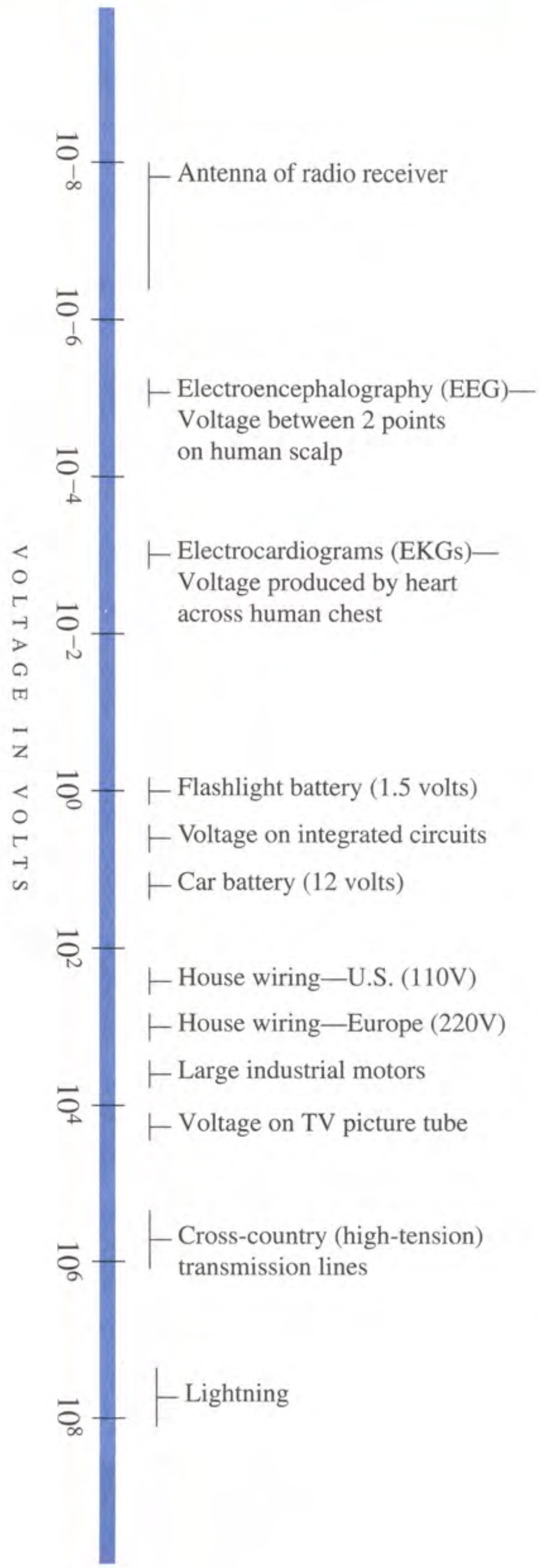
Bill Zobrist
Jovan Yglecias
Katherine Hepburn
Heather Olszyk
Lucille Buonocore
Christine Cervoni
Karin Kincheloe
Gene Aiello

Finally, I wish to express my deep appreciation to my wife, Edie, who cheerfully puts up with a lot of book business.

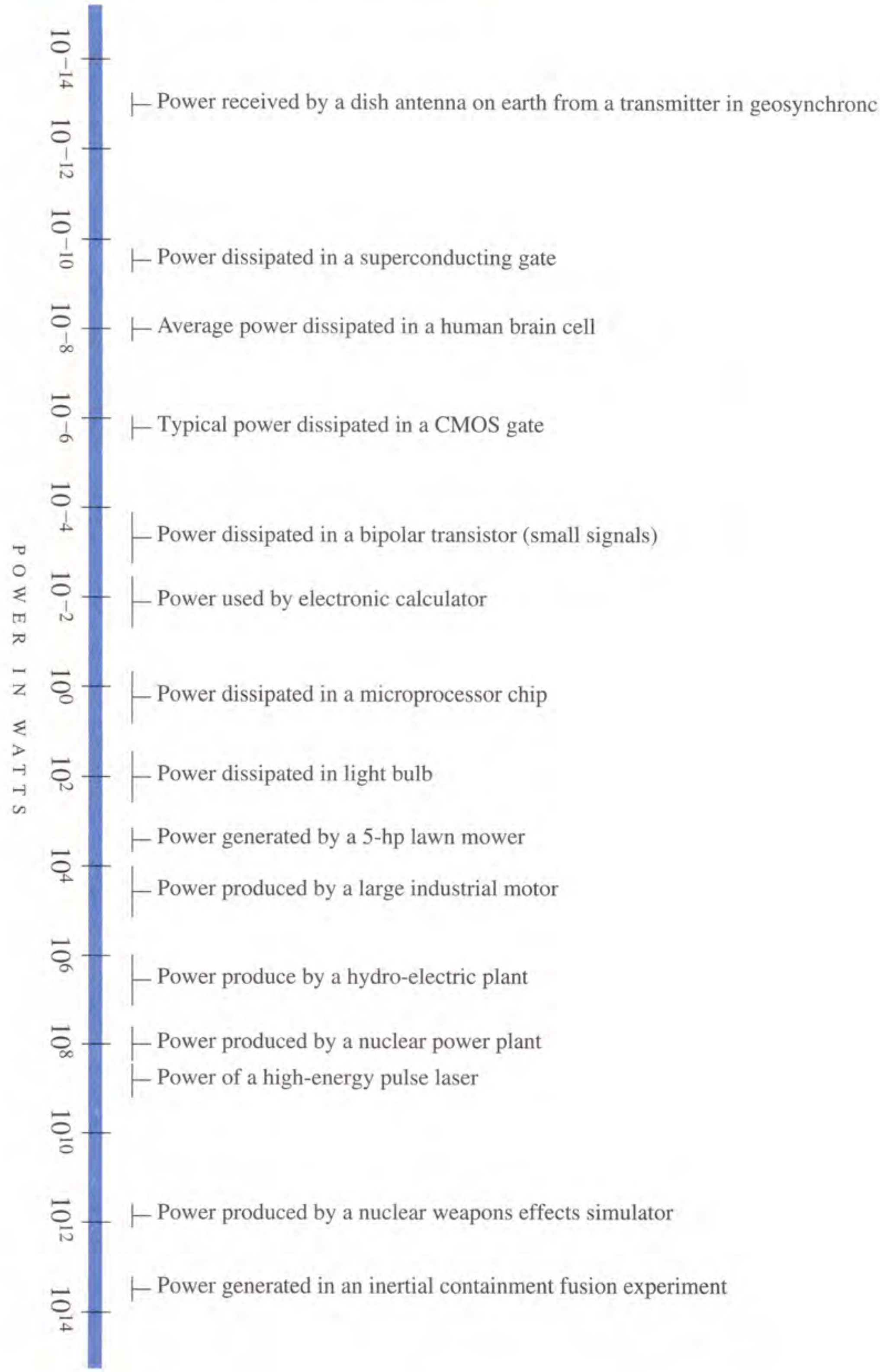
The Range of Current



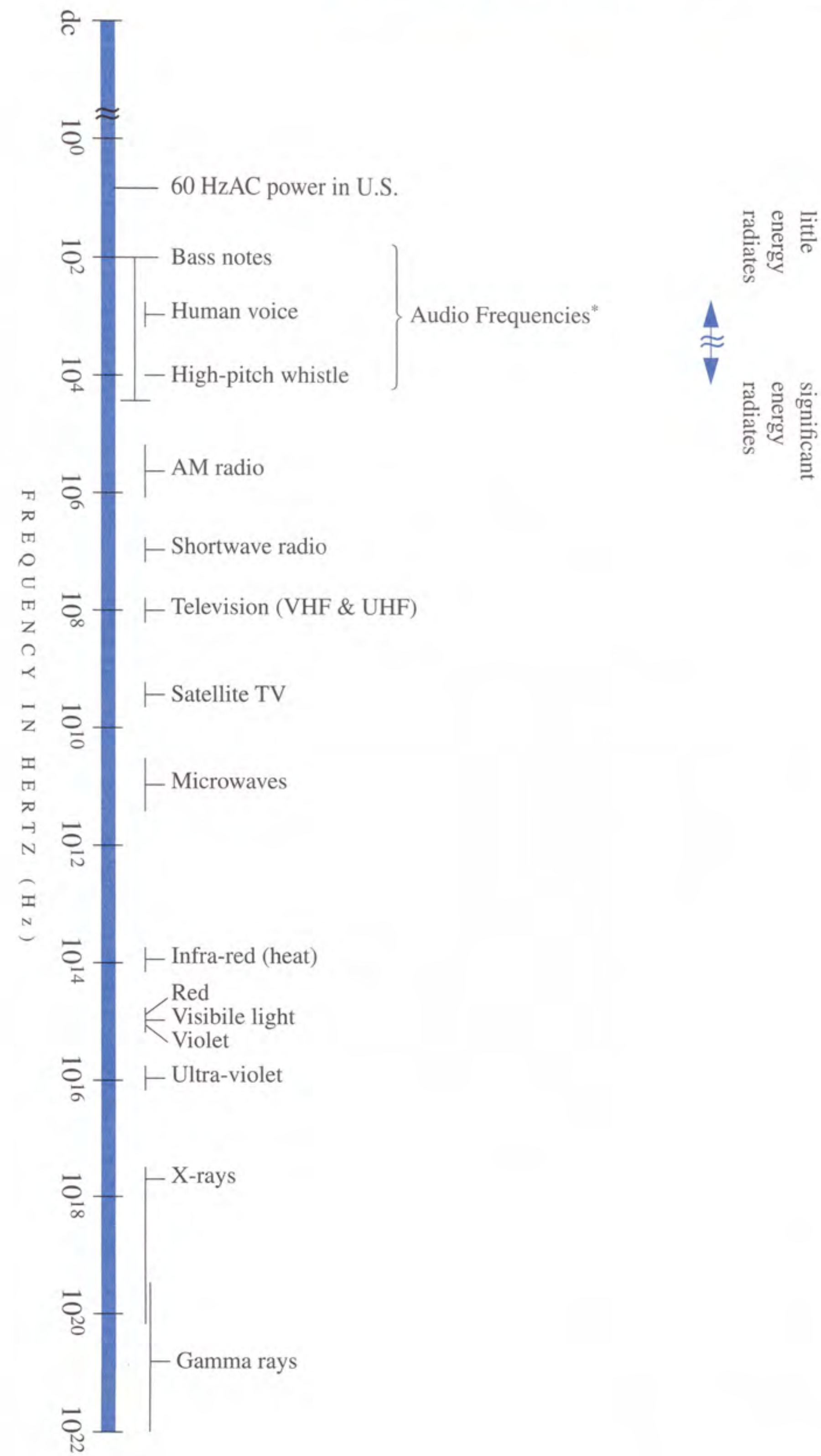
The Range of Voltage



The Range of Power



The Range of Spectrum



Basic Concepts

1

Today we live in a predominantly electrical world. Although this statement may sound strange at first, a moment's reflection will indicate its inherent truth. The two primary areas of electrotechnology that permeate essentially every aspect of our lives are power and information. Without them, life as we know it would undergo stupendous changes. We have learned to generate, convert, transmit, and utilize these technologies for the enhancement of the whole human race.

Electrotechnology is a driving force in the changes that are occurring in every engineering discipline. For example, surveying is now done with lasers and electronic range finders, and automobiles employ electronic dashboards and electronic ignition systems. Industrial processes that range from chemical refineries and metal foundries to wastewater treatment plants use (1) electronic sensors to obtain information about the process, (2) instrumentation systems to gather the information, and (3) computer control systems to process the information and generate electronic commands to actuators, which correct and control the process.

Fundamental to electrotechnology is the area of circuit analysis. A thorough knowledge of this subject provides an understanding of such things as cause and effect, amplification and attenuation, feedback and control, and stability and oscillation. Of critical importance is the fact that the same principles applied to engineering systems can also be applied to economic and social systems. Thus, the ramifications of circuit analysis are immense, and a solid understanding of this subject is well worth the effort expended to obtain it.

In this chapter we will introduce some of the basic quantities that will be used throughout the text. Specifically, we will define electric current, voltage, power and energy, as well as the difference between direct current and alternating current. In addition, we will classify electric elements as either passive or active, the latter of which can be further subdivided into both independent and dependent. This basic introduction will lay the groundwork for our further study of a wide variety of electric circuits.

LEARNING Goals

1.1 System of Units

The system of units we employ is the international system of units, the Système International des Unités, which is normally referred to as the SI standard system. This system, which is composed of the basic units meter (m), kilogram (kg), second (s), ampere (A), degree kelvin (°K), and candela (cd), is defined in all modern physics texts and therefore will not be defined here. However, we will discuss the units in some detail as we encounter them in our subsequent analyses.

The standard prefixes that are employed in SI are shown in Fig. 1.1. Note the decimal relationship between these prefixes. These standard prefixes are employed throughout our study of electric circuits.

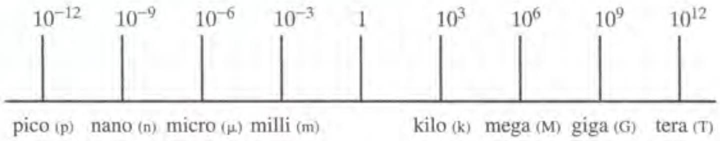


Figure 1.1
Standard SI prefixes.

Circuit technology has changed drastically over the years. For example, in the early 1960s the space on a circuit board occupied by the base of a single vacuum tube was about the size of a quarter (25-cent coin). Today that same space could be occupied by an Intel Pentium integrated circuit chip containing 3.1 million transistors. These chips are the engine for a host of electronic equipment.

1.2 Basic Quantities

Before we begin our analysis of electronic circuits, we must define terms that we will employ. However, in this chapter and throughout the book our definitions and explanations will be as simple as possible to foster an understanding of the use of the material. No attempt will be made to give complete definitions of many of the quantities because such definitions are not only unnecessary at this level but are often confusing. Although most of us have an intuitive concept of what is meant by a circuit, we will simply refer to an *electric circuit* as an interconnection of electrical components, each of which we will describe with a mathematical model.

The most elementary quantity in an analysis of electric circuits is the electric *charge*. Our interest in electric charge is centered around its motion, since charge in motion results in an energy transfer. Of particular interest to us are those situations in which the motion is confined to a definite closed path.

An electric circuit is essentially a pipeline that facilitates the transfer of charge from one point to another. The time rate of change of charge constitutes an electric *current*. Mathematically, the relationship is expressed as

$$i(t) = \frac{dq(t)}{dt} \quad \text{or} \quad q(t) = \int_{-\infty}^{t} i(x)\,dx \qquad \textbf{1.1}$$

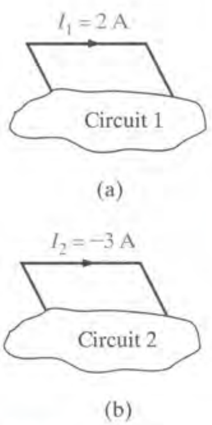


Figure 1.2
Conventional current flow: (a) positive current flow; (b) negative current flow.

where i and q represent current and charge, respectively (lowercase letters represent time dependency and capital letters are reserved for constant quantities). The basic unit of current is the ampere (A) and 1 ampere is 1 coulomb per second.

Although we know that current flow in metallic conductors results from electron motion, the conventional current flow, which is universally adopted, represents the movement of positive charges. It is important that the reader think of current flow as the movement of positive charge regardless of the physical phenomena that take place. The symbolism that will be used to represent current flow is shown in Fig. 1.2. $I_1 = 2$ A in Fig. 1.2a indicates that at any point in the wire shown,

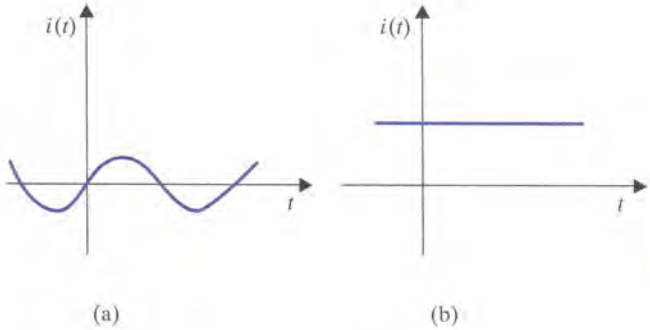


Figure 1.3
Two common types of current: (a) alternating current (ac); (b) direct current (dc).

2 C of charge pass from left to right each second. $I_2 = -3$ A in Fig. 1.2b indicates that at any point in the wire shown, 3 C of charge pass from right to left each second. Therefore, it is important to specify not only the magnitude of the variable representing the current, but also its direction.

There are two types of current that we encounter often in our daily lives, alternating current (ac) and direct current (dc), which are shown as a function of time in Fig. 1.3. *Alternating current* is the common current found in every household and is used to run the refrigerator, stove, washing machine, and so on. Batteries, which are used in automobiles or flashlights, are one source of *direct current*. In addition to these two types of currents, which have a wide variety of uses, we can generate many other types of currents. We will examine some of these other types later in the book. In the meantime, it is interesting to note that the magnitude of currents in elements familiar to us ranges from soup to nuts, as shown in Fig. 1.4.

We have indicated that charges in motion yield an energy transfer. Now we define the *voltage* (also called the *electromotive force* or *potential*) between two points in a circuit as the difference in energy level of a unit charge located at each of the two points. Work or energy, $w(t)$ or W, is measured in joules (J); 1 joule is 1 newton meter (N · m). Hence, voltage [$v(t)$ or V] is measured in volts (V) and 1 volt is 1 joule per coulomb; that is, 1 volt = 1 joule per coulomb = 1 newton meter per coulomb.

If a unit positive charge is moved between two points, the energy required to move it is the difference in energy level between the two points and is the defined voltage. It is extremely important that the variables that are used to represent voltage between two points be defined in such a way that the solution will let us interpret which point is at the higher potential with respect to the other.

LEARNING by Doing

D 1.1 Determine the amount of time required for 100 C of charge to pass through the circuit in Fig. 1.2a.

ANSWER *50 s*

LEARNING Hint

$$V = \frac{W}{q}$$

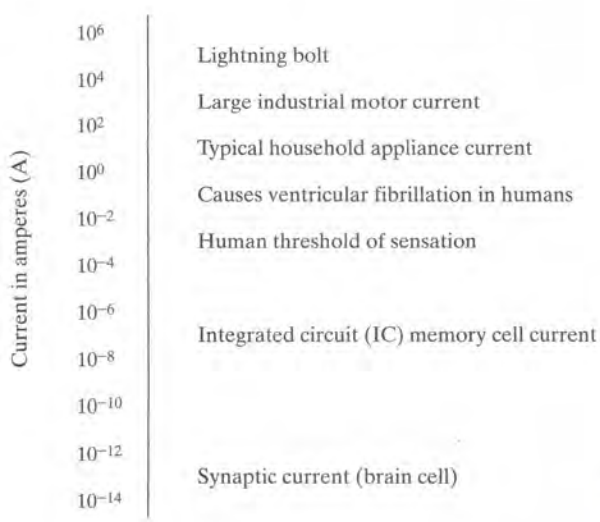


Figure 1.4
Typical current magnitudes.

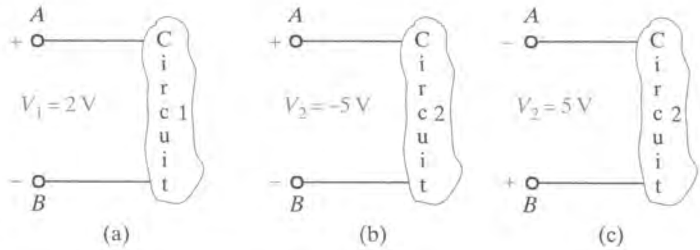


Figure 1.5
Voltage representations.

LEARNING by Doing

D 1.2 Determine the energy required to move 120 C of charge from point *B* to point *A* in the network in Fig. 1.5a.

ANSWER *240 J*

In Fig. 1.5a the variable that represents the voltage between points *A* and *B* has been defined as V_1, and it is assumed that point *A* is at a higher potential than point *B*, as indicated by the + and − signs associated with the variable and defined in the figure. The + and − signs define a reference direction for V_1. If $V_1 = 2$ V, then the difference in potential of points *A* and *B* is 2 V and point *A* is at the higher potential. If a unit positive charge is moved from point *A* through the circuit to point *B*, it will give up energy to the circuit and have 2 J less energy when it reaches point *B*. If a unit positive charge is moved from point *B* to point *A*, extra energy must be added to the charge by the circuit, and hence the charge will end up with 2 J more energy at point *A* than it started with at point *B*.

For the circuit in Fig. 1.5b, $V_2 = -5$ V means that the potential between points *A* and *B* is 5 V and point *B* is at the higher potential. The voltage in Fig. 1.5b can be expressed as shown in Fig. 1.5c. In this equivalent case, the difference in potential between points *A* and *B* is $V_2 = 5$ V, and point *B* is at the higher potential.

Note that it is important to define a variable with a reference direction so that the answer can be interpreted to give the physical condition in the circuit. We will find that it is not possible in many cases to define the variable so that the answer is positive, and we will also find that it is not necessary to do so. A negative number for a given variable gives exactly the same information as a positive number for a new variable that is the same as the old variable, except that it has an opposite reference direction. Hence, when we define either current or voltage, it is absolutely necessary that we specify both magnitude and direction. Therefore, it is incomplete to say that the voltage between two points is 10 V or the current in a line is 2 A, since only the magnitude and not the direction for the variables has been defined.

The range of magnitudes for voltage, equivalent to that for currents in Fig. 1.4, is shown in Fig. 1.6. Once again, note that this range spans many orders of magnitude.

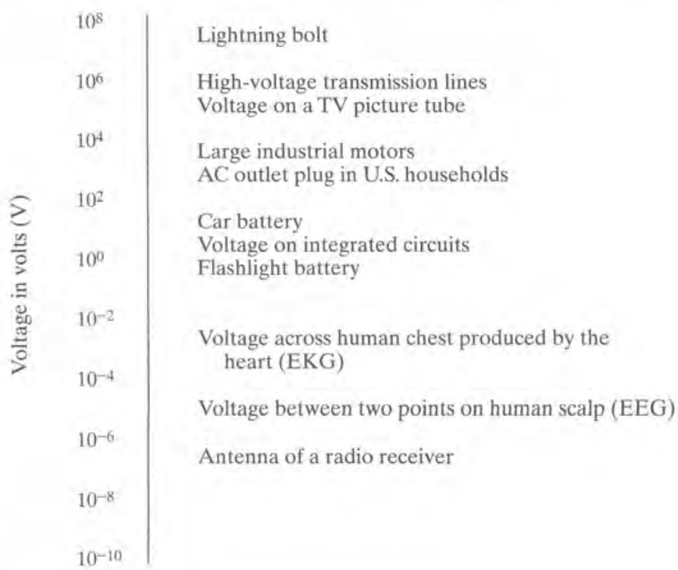


Figure 1.6
Typical voltage magnitudes.

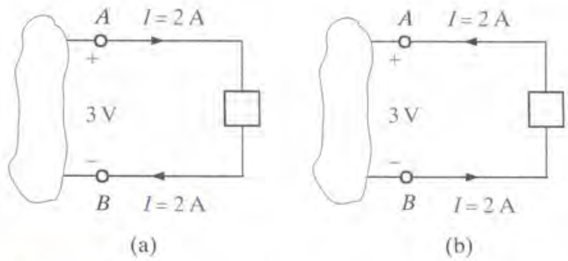
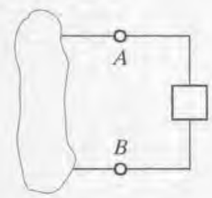


Figure 1.7 *Voltage–current relationships for (a) energy absorbed and (b) energy supplied.*

LEARNING by Doing

D 1.3 Five joules of energy are absorbed by 1 C of charge when moved from point B to point A. Find the voltage between points A and B.



ANSWER *5 V, B is +*

At this point we have presented the conventions that we employ in our discussions of current and voltage. *Energy* is yet another important term of basic significance. Figure 1.7 illustrates the voltage–current relationships for energy transfer. In this figure, the block representing a circuit element has been extracted from a larger circuit for examination. In Fig. 1.7a, energy is being supplied *to* the element by whatever is attached to the terminals. Note that 2 A, that is, 2 C, of charge are moving from point A to point B through the element each second. Each coulomb loses 3 J of energy as it passes through the element from point A to point B. Therefore, the element is absorbing 6 J of energy per second. Note that when the element is *absorbing* energy, a positive current enters the positive terminal. In Fig. 1.7b energy is being supplied *by* the element to whatever is connected to terminals A-B. In this case, note that when the element is *supplying* energy, a positive current enters the negative terminal and leaves via the positive terminal. In this convention a negative current in one direction is equivalent to a positive current in the opposite direction, and vice versa. Similarly, a negative voltage in one direction is equivalent to a positive voltage in the opposite direction.

LEARNING Example 1.1

Suppose that your car will not start. To determine whether the battery is faulty, you turn on the light switch and find that the lights are very dim, indicating a weak battery. You borrow a friend's car and a set of jumper cables. However, how do you connect his car's battery to yours? What do you want his battery to do?

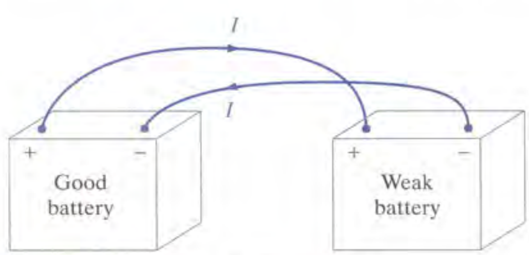


Figure 1.8 *Diagram for Example 1.1.*

SOLUTION Essentially, his car's battery must supply energy to yours, and therefore it should be connected in the manner shown in Fig. 1.8. Note that the positive current leaves the positive terminal of the good battery (supplying energy) and enters the positive terminal of the weak battery (absorbing energy). Note that the same connections are used when charging a battery.

In practical applications there are often considerations other than simply the electrical relations (e.g., safety). Such is the case with jump-starting an automobile. Automobile batteries produce explosive gases that can be ignited accidentally, causing severe physical injury. Be safe—follow the procedure described in your auto owner's manual.

We have defined voltage in joules per coulomb as the energy required to move a positive charge of 1 C through an element. If we assume that we are dealing with a differential amount of charge and energy, then

$$v = \frac{dw}{dq} \tag{1.2}$$

Multiplying this quantity by the current in the element yields

$$vi = \frac{dw}{dq}\left(\frac{dq}{dt}\right) = \frac{dw}{dt} = p \tag{1.3}$$

which is the time rate of change of energy or power measured in joules per second, or watts (W). Since, in general, both v and i are functions of time, p is also a time-varying quantity. Therefore, the change in energy from time t_1 to time t_2 can be found by integrating Eq. (1.3); that is,

$$\Delta w = \int_{t_1}^{t_2} p\,dt = \int_{t_1}^{t_2} vi\,dt \tag{1.4}$$

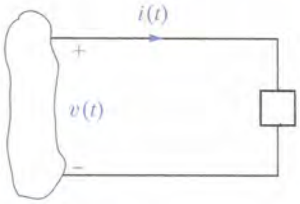


Figure 1.9
Sign convention for power.

LEARNING Hint

The passive sign convention is used to determine whether power is being absorbed or supplied.

At this point, let us summarize our sign convention for power. To determine the sign of any of the quantities involved, the variables for the current and voltage should be arranged as shown in Fig. 1.9. The variable for the voltage $v(t)$ is defined as the voltage across the element with the positive reference at the same terminal that the current variable $i(t)$ is entering. This convention is called the *passive sign convention* and will be so noted in the remainder of this book. The product of v and i, with their attendant signs, will determine the magnitude and sign of the power. If the sign of the power is positive, power is being absorbed by the element; if the sign is negative, power is being supplied by the element.

LEARNING Example 1.2

Given the two diagrams shown in Fig. 1.10, determine whether the element is absorbing or supplying power and how much.

SOLUTION In Fig. 1.10a the power is $P = (2\text{ V})(-4\text{ A}) = -8\text{ W}$. Therefore, the element is supplying power. In Fig. 1.10b, the power is $P = (2\text{ V})(2\text{ A}) = 4\text{W}$. Therefore, the element is absorbing power.

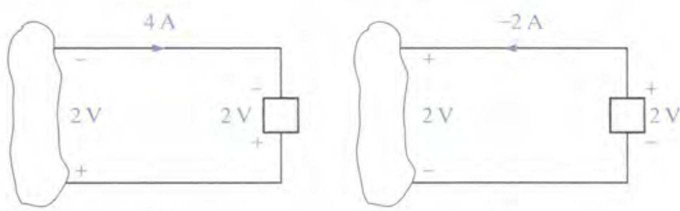


Figure 1.10
Elements for Example 1.2.

(a) (b)

LEARNING EXTENSION

E1.1 Determine the amount of power absorbed or supplied by the elements in Fig. E1.1.

ANSWER (a) $P = -48$ W;
(b) $P = 8$ W.

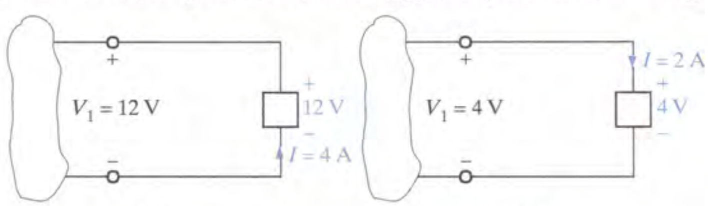


Figure E1.1

(a) (b)

LEARNING Example 1.3

We wish to determine the unknown voltage or current in Fig. 1.11.

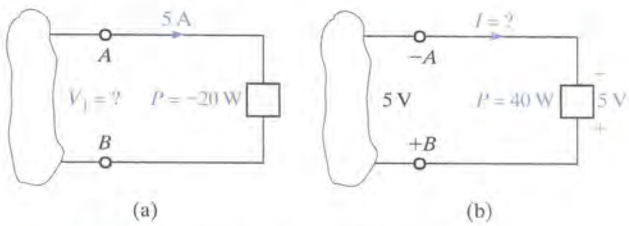


Figure 1.11 *Elements for Example 1.3.*

SOLUTION In Fig. 1.11a, a power of −20 W indicates that the element is delivering power. Therefore, the current enters the negative terminal (terminal *A*), and from Eq. (1.3) the voltage is 4 V. Thus *B* is the positive terminal, *A* is the negative terminal, and the voltage between them is 4 V.

In Fig 1.11b, a power of +40 W indicates that the element is absorbing power and, therefore, the current should enter the positive terminal *B*. The current thus has a value of −8 A, as shown in the figure.

LEARNING EXTENSION

E1.2 Determine the unknown variables in Fig. E1.2.

ANSWER (a) $V_1 = -20$ V; (b) $I = -5$ A.

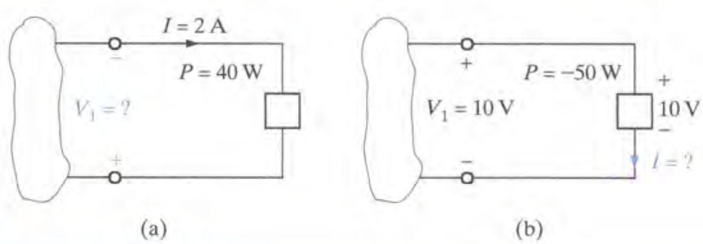


Figure E1.2

Finally, it is important to note that these electrical networks satisfy the principle of conservation of energy. For our present purposes this means that the power supplied in a network is exactly equal to the power absorbed.

1.3 Circuit Elements

Thus far we have defined voltage, current, and power. In the remainder of this chapter we will define both independent and dependent current and voltage sources. Although we will assume ideal elements, we will try to indicate the shortcomings of these assumptions as we proceed with the discussion.

In general, the elements we will define are terminal devices that are completely characterized by the current through the element and/or the voltage across it. These elements, which we will employ in constructing electric circuits, will be broadly classified as being either active or passive. The distinction between these two classifications depends essentially on one thing—whether they supply or absorb energy. As the words themselves imply, an *active* element is capable of generating energy and a *passive* element cannot generate energy.

However, we will show later that some passive elements are capable of storing energy. Typical active elements are batteries, generators, and transistor models. The three common passive elements are resistors, capacitors, and inductors.

In Chapter 2 we will launch an examination of passive elements by discussing the resistor in detail. However, before proceeding with that element, we first present some very important active elements.

1. Independent voltage source
2. Independent current source
3. Two dependent voltage sources
4. Two dependent current sources

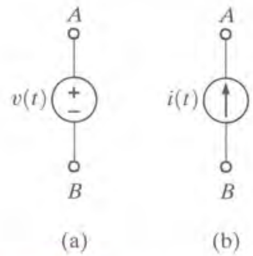


Figure 1.12
Symbols for (a) independent voltage source, (b) independent current source.

INDEPENDENT SOURCES An *independent voltage source* is a two-terminal element that maintains a specified voltage between its terminals *regardless of the current through it*. The general symbol for an independent source, a circle, is shown in Fig. 1.12a. As the figure indicates, terminal A is $v(t)$ volts positive with respect to terminal B.

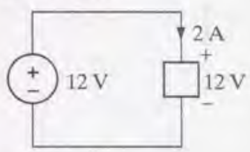
In contrast to the independent voltage source, the *independent current source* is a two-terminal element that maintains a specified current *regardless of the voltage across its terminals*. The general symbol for an independent current source is shown in Fig. 1.12b, where $i(t)$ is the specified current and the arrow indicates the positive direction of current flow.

In their normal mode of operation, independent sources supply power to the remainder of the circuit. However, they may also be connected into a circuit in such a way that they absorb power. A simple example of this latter case is a battery-charging circuit such as that shown in Example 1.1.

It is important that we pause here to interject a comment concerning a shortcoming of the models. In general, mathematical models approximate actual physical systems only under a certain range of conditions. Rarely does a model accurately represent a physical system under every set of conditions. To illustrate this point, consider the model for the voltage source in Fig. 1.12a. We assume that the voltage source delivers v volts regardless of what is connected to its terminals. Theoretically, we could adjust the external circuit so that an infinite amount of current would flow, and therefore the voltage source would deliver an infinite amount of power. This is, of course, physically impossible. A similar argument could be made for the independent current source. Hence, the reader is cautioned to keep in mind that models have limitations and thus are valid representations of physical systems only under certain conditions.

LEARNING by Doing

D 1.4 Determine the power supplied by the 12-V source.



ANSWER *24 W*

LEARNING Example 1.4

Determine the power absorbed or supplied by the elements in the network in Fig. 1.13.

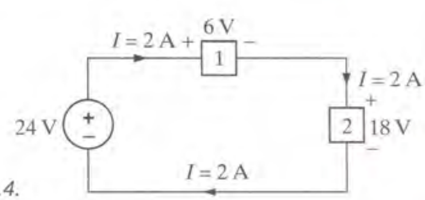


Figure 1.13
Network for Example 1.4.

SOLUTION The current flow is out of the positive terminal of the 24-V source, and therefore this element is supplying $(2)(24) = 48$ W of power. The current is into the positive terminals of elements 1 and 2, and therefore elements 1 and 2 are absorbing $(2)(6) = 12$ W and $(2)(18) = 36$ W, respectively. Note that the power supplied is equal to the power absorbed.

LEARNING Hint

Elements that are connected in series have the same current.

LEARNING EXTENSION

E1.3 Find the power that is absorbed or supplied by the elements in Fig. E1.3.

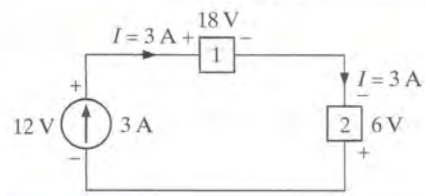


Figure E1.3

ANSWER Current source supplies 36 W, element 1 absorbs 54 W, and element 2 supplies 18 W.

DEPENDENT SOURCES In contrast to the independent sources, which produce a particular voltage or current completely unaffected by what is happening in the remainder of the circuit, dependent sources generate a voltage or current that is determined by a voltage or current at a specified location in the circuit. These sources are very important because they are an integral part of the mathematical models used to describe the behavior of many electronic circuit elements.

For example, metal-oxide-semiconductor field-effect transistors (MOSFETs) and bipolar transistors, both of which are commonly found in a host of electronic equipment, are modeled with dependent sources, and therefore the analysis of electronic circuits involves the use of these controlled elements.

In contrast to the circle used to represent independent sources, a diamond is used to represent a dependent or controlled source. Figure 1.14 illustrates the four types of dependent sources. The input terminals on the left represent the voltage or current that controls the dependent source, and the output terminals on the right represent the output current or voltage of the controlled source. Note that in Figs. 1.14a and d the quantities μ and β are dimensionless constants because we are transforming voltage to voltage and current to current. This is not the case in Figs. 1.14b and c; hence when we employ these elements a short time later, we must describe the units of the factors r and g.

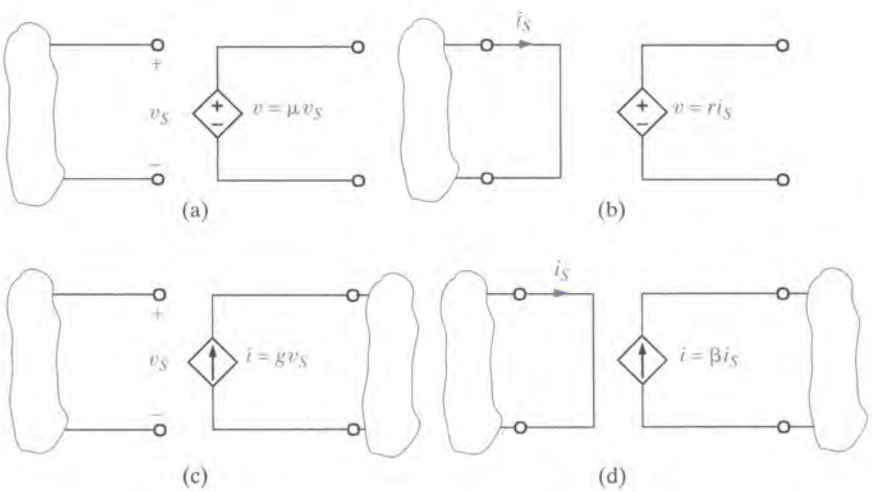


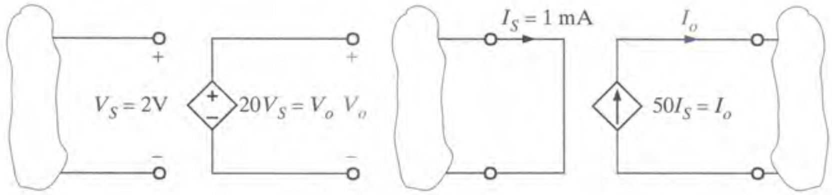
Figure 1.14 Four different types of dependent sources.

LEARNING Example 1.5

Given the two networks shown in Fig. 1.15, we wish to determine the outputs.

SOLUTION In Fig. 1.15a the output voltage is $V_o = \mu V_S$ or $V_o = 20\ V_S = (20)(2\ \text{V}) = 40$ V. Note that the output voltage has been amplified from 2 V at the input terminals to 40 V at the output terminals; that is, the circuit is a voltage amplifier with an amplification factor of 20.

In Fig. 1.15b, the output current is $I_o = \beta I_S = (50)(1\ \text{mA}) = 50$ mA; that is, the circuit has a current gain of 50, meaning that the output current is 50 times greater than the input current.



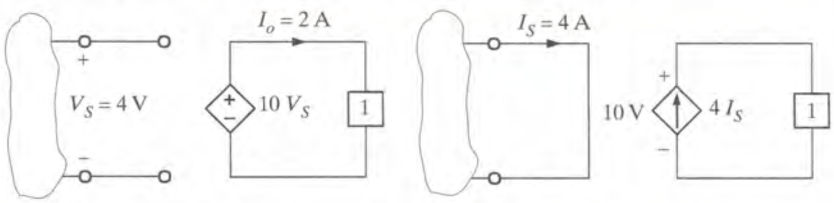
(a) (b)

Figure 1.15
Circuits for Example 1.5.

LEARNING EXTENSION

E1.4 Determine the power supplied by the dependent sources in Fig. E1.4.

ANSWER (a) Power supplied = 80 W; (b) power supplied = 160 W.



(a) (b)

Figure E1.4

LEARNING Example 1.6

Let us find the current I_o in the network in Fig. 1.16.

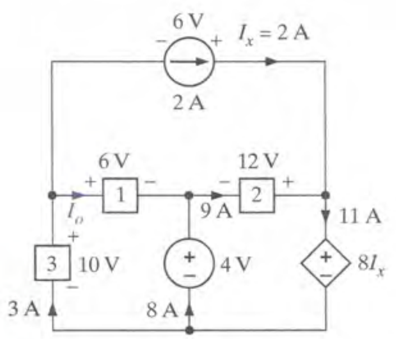


Figure 1.16
Circuit used in Example 1.6.

SOLUTION First, we must determine the power absorbed or supplied by each element in the network. Using the sign convention for power, we find

$$P_{2\text{A}} = (6)(-2) = -12\ \text{W}$$
$$P_1 = (6)(I_o) = 6I_o\ \text{W}$$
$$P_2 = (12)(-9) = -108\ \text{W}$$
$$P_3 = (10)(-3) = -30\ \text{W}$$
$$P_{4\text{V}} = (4)(-8) = -32\ \text{W}$$
$$P_{DS} = (8I_x)(11) = (16)(11) = 176\ \text{W}$$

Since energy must be conserved,

$$-12 + 6I_o - 108 - 30 - 32 + 176 = 0$$

or

$$6I_o + 176 = 12 + 108 + 30 + 32$$

Hence,

$$I_o = 1\text{A}$$

LEARNING EXTENSION

E1.5 Find the power that is absorbed or supplied by the circuit elements in the network in Fig. E1.5.

ANSWER $P_{24\text{ V}} = 96$ W supplied, $P_1 = 32$ W absorbed, $P_{4I_x} = 64$ W absorbed.

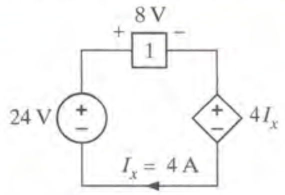


Figure E1.5

LEARNING Check

Summary

- ***The standard prefixes employed***

p = 10^{-12}	k = 10^3
n = 10^{-9}	M = 10^6
μ = 10^{-6}	G = 10^9
m = 10^{-3}	T = 10^{12}

- ***The relationships between current and charge***

$$i(t) = \frac{dq(t)}{dt} \quad \text{or} \quad q(t) = \int_{-\infty}^{t} i(x)\,dx$$

- ***The relationships among power, energy, current, and voltage***

$$p = \frac{dw}{dt} = vi$$

$$\Delta w = \int_{t_1}^{t_2} p\,dt = \int_{t_1}^{t_2} vi\,dt$$

- ***The passive sign convention*** The passive sign convention states that if the voltage and current associated with an element are as shown in Fig. 1.9, the product of v and i, with their attendant signs, determines the magnitude and sign of the power. If the sign is positive, power is being absorbed by the element, and if the sign is negative, the element is supplying power.

- ***Independent and dependent sources*** An ideal independent voltage (current) source is a two-terminal element that maintains a specified voltage (current) between its terminals regardless of the current (voltage) through (across) the element. Dependent or controlled sources generate a voltage or current that is determined by a voltage or current at a specified location in the circuit.

- ***Conservation of energy*** The electric circuits under investigation satisfy the conservation of energy.

Problems

For solutions and additional help on problems marked with ► go to www.wiley.com/college/irwin

SECTION 1.2

1.1 ► If 60 C of charge pass through an electric conductor in 30 seconds, determine the current in the conductor.

1.2 If the current in an electric conductor is 2.4 A, how many coulombs of charge pass any point in a 30-second interval?

1.3 Determine the time interval required for a 12-A battery charger to deliver 4800 C.

1.4 A lightning bolt carrying 30,000 A lasts for 50 microseconds. If the lightning strikes an airplane flying at 20,000 feet, what is the charge deposited on the plane?

1.5 ► Determine the energy required to move 240 C through 6 V.

1.6 Determine the amount of power absorbed or supplied by the element in Fig. P1.6 if (a) $V_1 = 4$ V, $I = 2$ A and (b) $V_1 = -4$ V, $I = -2$ A.

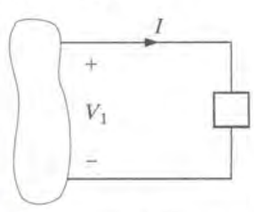


Figure P1.6

1.7 Repeat Problem 1.6 if (a) $V_1 = -6$ V, $I = 3$ A and (b) $V_1 = 6$ V, $I = -3$ A.

1.8 Determine the missing quantity in the circuits in Fig. P1.8.

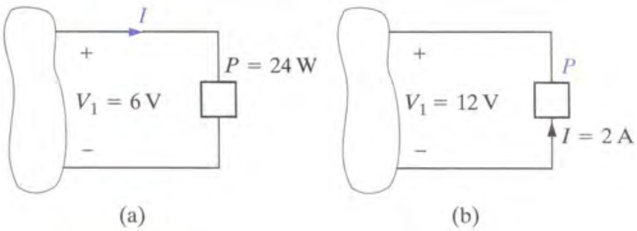


Figure P1.8

1.9 Repeat Problem 1.8 for the circuits in Fig. P1.9.

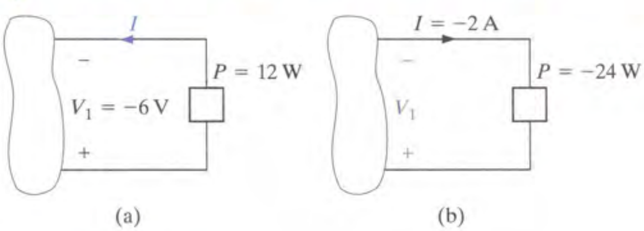


Figure P1.9

1.10 Repeat Problem 1.8 for the circuits in Fig. P1.10.

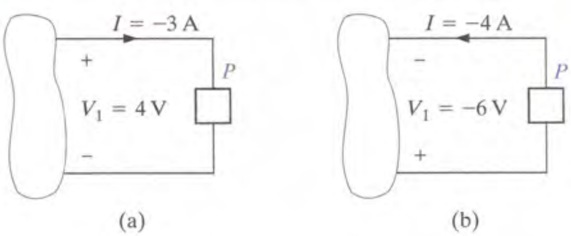


Figure P1.10

1.11 Determine the power supplied to the elements in Fig. P1.11.

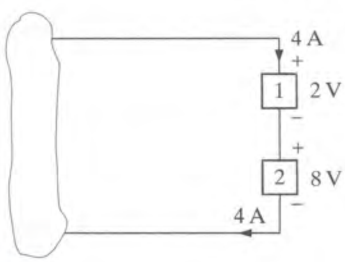


Figure P1.11

1.12 Determine the power supplied to the elements in Fig. P1.12.

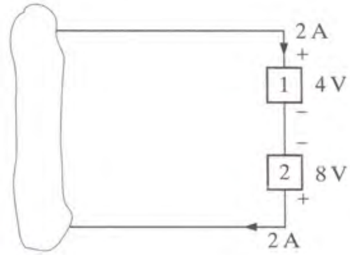


Figure P1.12

1.13 Two elements are connected in series, as shown in Fig. P1.13. Element 1 supplies 24 W of power. Is element 2 absorbing or supplying power, and how much?

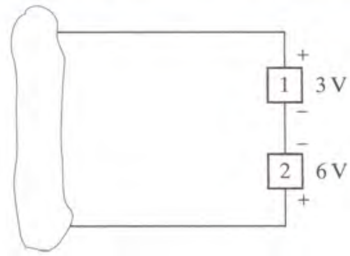


Figure P1.13

1.14 Two elements are connected in series, as shown in Fig. P1.14. Element 1 absorbs 36 W of power. Is element 2 absorbing or supplying power, and how much?

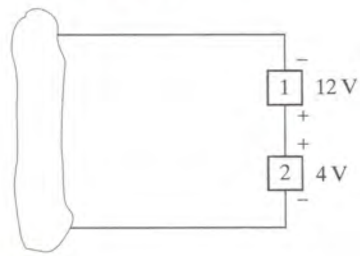
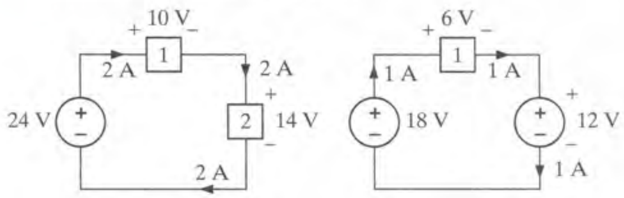


Figure P1.14

SECTION 1.3

Important note: The values used in the problems of Section 1.3 are not arbitrary. They have been selected to satisfy the basic laws of circuit analysis that will be studied in the following chapters.

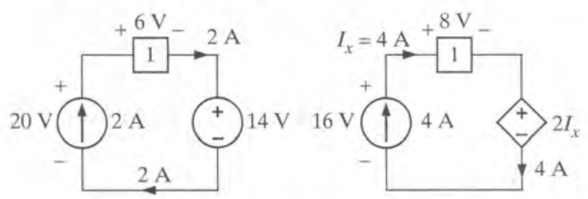
1.15 Determine the power that is absorbed or supplied by the circuit elements in Fig. P1.15.



(a) (b)

Figure P1.15

1.16 Find the power that is absorbed or supplied by the circuit elements in Fig. P1.16.



(a) (b)

Figure P1.16

1.17 Compute the power that is absorbed or supplied by the elements in the network in Fig. P1.17.

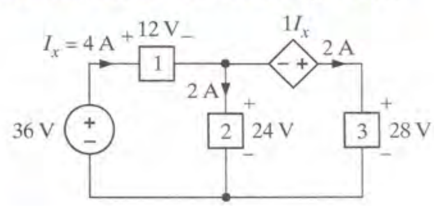


Figure P1.17

1.18 Find I_x in the network in Fig. P1.18.

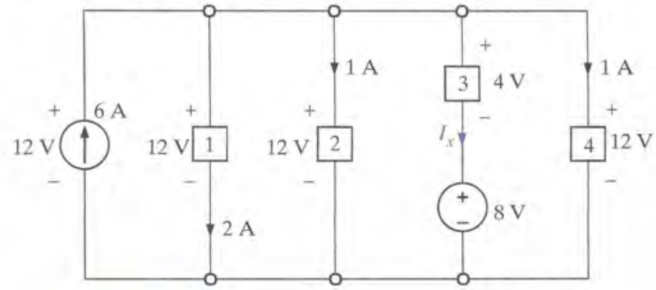


Figure P1.18

1.19 Is the source V_S in the network in Fig. P1.19 absorbing or supplying power, and how much?

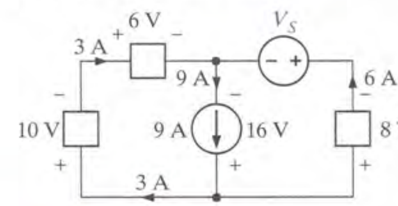


Figure P1.19

1.20 Find V_x in the network in Fig. P1.20.

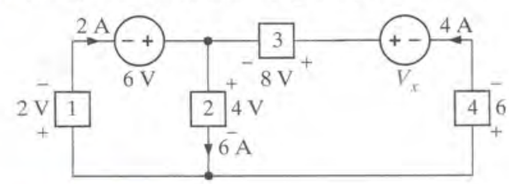


Figure P1.20

1.21 Find I_o in the network in Fig. P1.21.

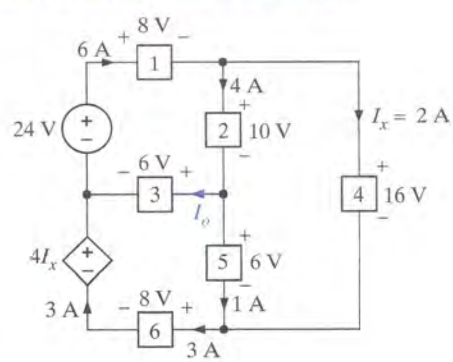


Figure P1.21

2 Resistive Circuits

LEARNING Goals

In this chapter we introduce the basic concepts and laws that are fundamental to circuit analysis. These laws are Ohm's law, Kirchhoff's current law (KCL), and Kirchhoff's voltage law (KVL). We cannot overemphasize the importance of these three laws because they will be used extensively throughout our entire study of circuit analysis. The reader who masters their use quickly will not only find the material in this text easy to learn, but will be well positioned to grasp subsequent topics in the field of electrical engineering.

As a general rule, most of our activities will be confined to analysis; that is, to the determination of a specific voltage, current, or power somewhere in a network. The techniques we introduce have wide application in circuit analysis, even though we will discuss them within the framework of simple networks.

Our approach here is to begin with the simplest passive element, the resistor, and the mathematical relationship that exists between the voltage across it and the current through it, as specified by Ohm's law. As we build our confidence and proficiency by successfully analyzing some elementary circuits, we will introduce other techniques, such as voltage division and current division, that will accelerate our work.

In this chapter we introduce circuits containing dependent sources, which are used to model active devices such as transistors. Thus, our study of circuit analysis provides a natural introduction to many topics in the area of electronics.

Finally, we present a real-world application to indicate the usefulness of circuit analysis, and then we briefly introduce the topic of circuit design in an elementary fashion. In future chapters these topics will be revisited often to present some fascinating examples that describe problems we encounter in our everyday lives.

2.1 Ohm's Law

Ohm's law is named for the German physicist Georg Simon Ohm, who is credited with establishing the voltage–current relationship for resistance. As a result of his pioneering work, the unit of resistance bears his name.

Ohm's law states that the voltage across a resistance is directly proportional to the current flowing through it. The resistance, measured in ohms, is the constant of proportionality between the voltage and current.

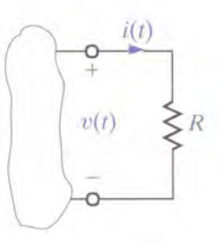
LEARNING Hint

The passive sign convention will be employed in conjunction with Ohm's law.

A circuit element whose electrical characteristic is primarily resistive is called a resistor and is represented by the symbol shown in Fig. 2.1a. A resistor is a physical device that can be purchased in certain standard values in an electronic parts store. These resistors, which find use in a variety of electrical applications, are normally carbon composition or wirewound. In addition, resistors can be fabricated using thick oxide or thin metal films for use in hybrid circuits, or they can be diffused in semiconductor integrated circuits. Some typical discrete resistors are shown in Fig. 2.1b.

The mathematical relationship of Ohm's law is illustrated by the equation

$$v(t) = R \times i(t), \text{ where } R \geqq 0 \qquad 2.1$$



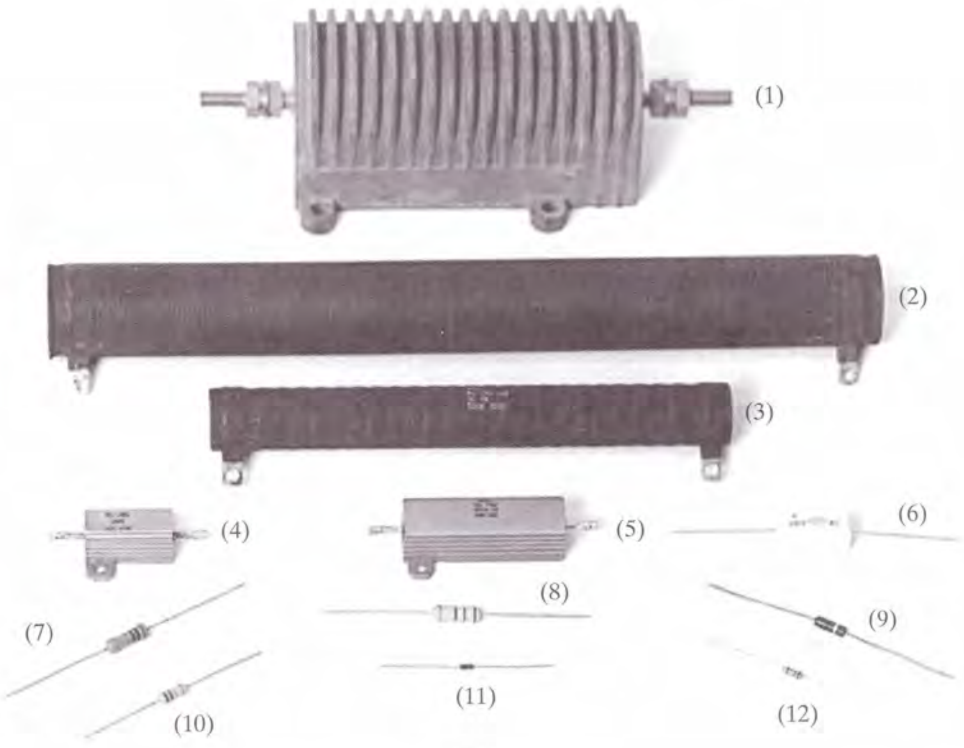


Figure 2.1 (a) Symbol for a resistor; (b) some practical devices. (1), (2), and (3) are high-power resistors. (4) and (5) are high-wattage fixed resistors. (6) is a high-precision resistor. (7)–(12) are fixed resistors with different power ratings.

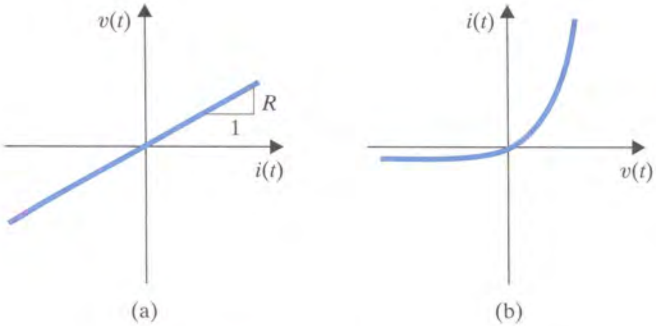
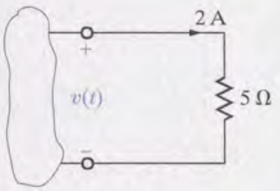


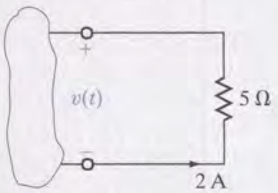
Figure 2.2
Graphical representation of the voltage–current relationship for (a) a linear resistor and (b) a diode.

LEARNING by Doing

D 2.1 Determine $v(t)$ in the circuits.



(a)



(b)

ANSWER *(a)* 10 V
(b) −10 V

or equivalently, by the voltage–current characteristic shown in Fig. 2.2a. Note carefully the relationship between the polarity of the voltage and the direction of the current. In addition, note that we have tacitly assumed that the resistor has a constant value and therefore that the voltage–current characteristic is linear.

The symbol Ω is used to represent ohms, and therefore,

$$1\ \Omega = 1\ \text{V/A}$$

Although in our analysis we will always assume that the resistors are *linear* and are thus described by a straight-line characteristic that passes through the origin, it is important that readers realize that some very useful and practical elements do exist that exhibit a *nonlinear* resistance characteristic; that is, the voltage–current relationship is not a straight line. Diodes and transistors are examples of elements that exhibit nonlinear characteristics, and are used extensively in electric circuits. A typical characteristic for a diode is shown in Fig. 2.2b.

Since a resistor is a passive element, the proper current–voltage relationship is illustrated in Fig. 2.1a. The power supplied to the terminals is absorbed by the resistor. Note that the charge moves from the higher to the lower potential as it passes through the resistor and the energy absorbed is dissipated by the resistor in the form of heat. As indicated in Chapter 1, the rate of energy dissipation is the instantaneous power, and therefore

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

which, using Eq. (2.1), can be written as

$$p(t) = Ri^2(t) = \frac{v^2(t)}{R} \tag{2.3}$$

This equation illustrates that the power is a nonlinear function of either current or voltage and that it is always a positive quantity.

Conductance, represented by the symbol G, is another quantity with wide application in circuit analysis. By definition, conductance is the reciprocal of resistance; that is,

$$G = \frac{1}{R} \tag{2.4}$$

The unit of conductance is the siemens, and the relationship between units is

$$1\ \text{S} = 1\ \text{A/V}$$

Using Eq. (2.4), we can write two additional expressions,

$$i(t) = Gv(t) \tag{2.5}$$

and

$$p(t) = \frac{i^2(t)}{G} = Gv^2(t) \qquad 2.6$$

Equation (2.5) is another expression of Ohm's law.

Two specific values of resistance, and therefore conductance, are very important: $R = 0$ and $R = \infty$.

In examining the two cases, consider the network in Fig. 2.3a. The variable resistance symbol is used to describe a resistor such as the volume control on a radio or television set. As the resistance is decreased and becomes smaller and smaller, we finally reach a point where the resistance is zero and the circuit is reduced to that shown in Fig. 2.3b; that is, the resistance can be replaced by a short circuit. On the other hand, if the resistance is increased and becomes larger and larger, we finally reach a point where it is essentially infinite and the resistance can be replaced by an open circuit, as shown in Fig. 2.3c. Note that in the case of a short circuit where $R = 0$,

$$\begin{aligned} v(t) &= Ri(t) \\ &= 0 \end{aligned}$$

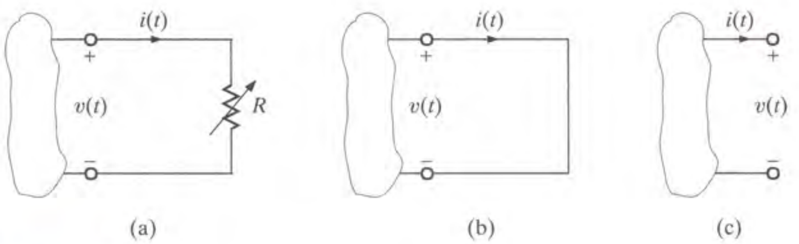


Figure 2.3
Short-circuit and open-circuit descriptions.

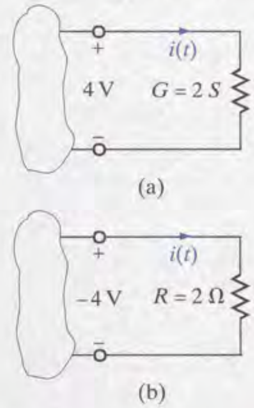
Therefore, $v(t) = 0$, although the current could theoretically be any value. In the open-circuit case where $R = \infty$,

$$\begin{aligned} i(t) &= v(t)/R \\ &= 0 \end{aligned}$$

Therefore, the current is zero regardless of the value of the voltage across the open terminals.

LEARNING by Doing

D 2.2 Determine $i(t)$ in the following two circuits:



ANSWER *(a) 8 A*
(b) −2 A

LEARNING Example 2.1

In the circuit in Fig. 2.4a, determine the current and the power absorbed by the resistor.

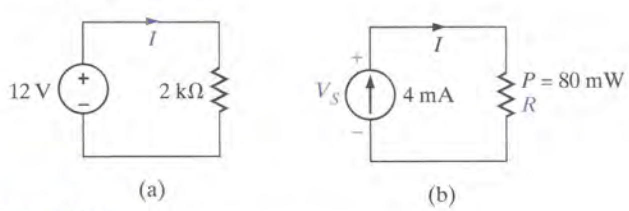


Figure 2.4
Circuits for Examples 2.1 and 2.2.

SOLUTION Using Eq. (2.1), we find the current to be

$$I = V/R = 12/2\text{k} = 6 \text{ mA}$$

Note that because many of the resistors employed in our analysis are in kΩ, we will use k in the equations in place of 1000. The power absorbed by the resistor is given by Eq. (2.2) or (2.3) as

$$\begin{aligned} P &= VI = (12)(6 \times 10^{-3}) = 0.072 \text{ W} \\ &= I^2R = (6 \times 10^{-3})^2(2\text{k}) = 0.072 \text{ W} \\ &= V^2/R = (12)^2/2\text{k} = 0.072 \text{ W} \end{aligned}$$

LEARNING Example 2.2

Given the network in Fig. 2.4b, we wish to find R and V_S.

SOLUTION Using the power relationship, we find that

$$R = P/I^2 = (80 \times 10^{-3})/(4 \times 10^{-3})^2 = 5\text{ k}\Omega$$

The voltage can now be derived using Ohm's law as

$$V_S = IR = (4 \times 10^{-3})(5\text{k}) = 20\text{ V}$$

The voltage could also be obtained from the remaining power relationships in Eqs. (2.2) and (2.3).

Before leaving this initial discussion of circuits containing sources and a single resistor, it is important to note a phenomenon that we will find to be true in circuits containing many sources and resistors. The presence of a voltage source between a pair of terminals tells us precisely what the voltage is between the two terminals regardless of what is happening in the balance of the network. What we do not know is the current in the voltage source. We must apply circuit analysis to the entire network to determine this current. Likewise, the presence of a current source connected between two terminals specifies the exact value of the current through the source between the terminals. What we do not know is the value of the voltage across the current source. This value must be calculated by applying circuit analysis to the entire network. Furthermore, it is worth emphasizing that when applying Ohm's law, the relationship $V = IR$ specifies a relationship between the voltage *directly across* a resistor R and the current that is *present* in this resistor. Ohm's law does not apply when the voltage is present in one part of the network and the current exists in another. This is a common mistake made by students who try to apply $V = IR$ to a resistor R in the middle of the network while using a V at some other location in the network.

LEARNING EXTENSIONS

E2.1 Given the circuits in Fig. E2.1, find (a) the current I and the power absorbed by the resistor in Fig. E2.1a, and (b) the voltage across the current source and the power supplied by the source in Fig. E2.1b.

ANSWER (a) $I = 0.3$ mA, $P = 3.6$ mW, (b) $V_S = 3.6$ V, $P = 2.16$ mW.

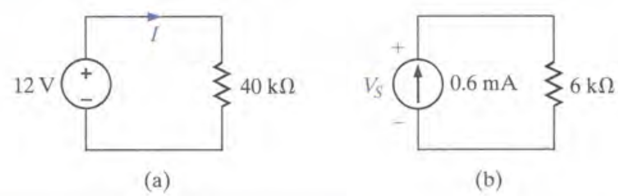


Figure E2.1

E2.2 Given the circuits in Fig. E2.2, find (a) R and V_S in the circuit in Fig. E2.2a, and (b) find I and R in the circuit in Fig. E2.2b.

ANSWER (a) $R = 10$ kΩ, $V_S = 4$ V, (b) $I = 20.8$ mA, $R = 576$ Ω.

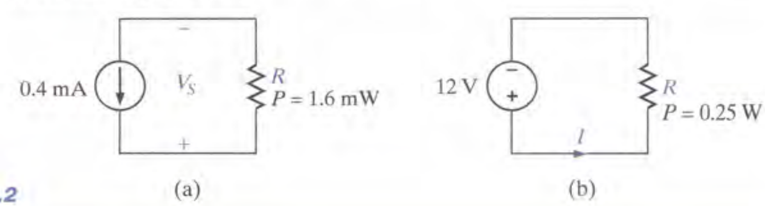


Figure E2.2

2.2 Kirchhoff's Laws

The previous circuits that we have considered have all contained a single resistor and were analyzed using Ohm's law. At this point we begin to expand our capabilities to handle more complicated networks that result from an interconnection of two or more of these simple elements. We will assume that the interconnection is performed by electrical conductors (wires) that have zero resistance; that is, perfect conductors. Because the wires have zero resistance, the energy in the circuit is in essence lumped in each element, and we employ the term *lumped-parameter circuit* to describe the network.

To aid us in our discussion, we will define a number of terms that will be employed throughout our analysis. As will be our approach throughout this text, we will use examples to illustrate the concepts and define the appropriate terms. For example, the circuit shown in Fig. 2.5 will be used to describe the terms *node, loop*, and *branch*. A *node* is simply a point of connection of two or more circuit elements. The reader is cautioned to note that although one node can be spread out with perfect conductors, it is still only one node. For example, node 5 consists of the entire bottom connector of the circuit. In other words, if we start at some point in the circuit and move along perfect conductors in any direction until we encounter a circuit element, the total path we cover represents a single node. Therefore, we can assume that a node is one end of a circuit element together with all the perfect conductors that are attached to it. Examining the circuit, we note that there are numerous paths through it. A *loop* is simply any *closed path* through the circuit in which no node is encountered more than once. For example, starting from node 1, one loop would contain the elements R_1, v_2, R_4, and i_1; another loop would contain R_2, v_1, v_2, R_4, and i_1; and so on. However, the path R_1, v_1, R_5, v_2, R_3, and i_1 is not a loop because we have encountered node 3 twice. Finally, a *branch* is a portion of a circuit containing only a single element and the nodes at each end of the element. The circuit in Fig. 2.5 contains eight branches.

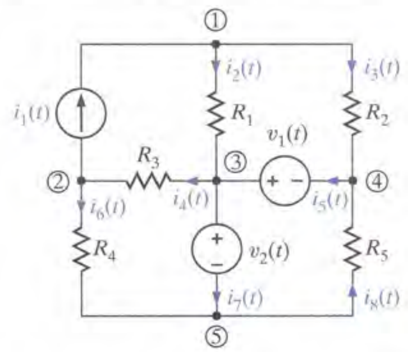


Figure 2.5
Circuit used to illustrate KCL.

Given the previous definitions, we are now in a position to consider Kirchhoff's laws, named after German scientist Gustav Robert Kirchhoff. These two laws are quite simple but extremely important. We will not attempt to prove them because the proofs are beyond our current level of understanding. However, we will demonstrate their usefulness and attempt to make the reader proficient in their use. The first law is *Kirchhoff's current law* (KCL), which states that *the algebraic sum of the currents entering any node is zero*. In mathematical form the law appears as

LEARNING Hint

KCL is an extremely important and useful law.

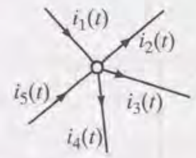
$$\sum_{j=1}^{N} i_j(t) = 0 \qquad 2.7$$

where $i_j(t)$ is the jth current entering the node through branch j and N is the number of branches connected to the node. To understand the use of this law, consider node 3 shown in Fig. 2.5. Applying Kirchhoff's current law to this node yields

$$i_2(t) - i_4(t) + i_5(t) - i_7(t) = 0$$

LEARNING by Doing

D 2.3 Write the KCL equation for the following node:



ANSWER

$i_1(t) - i_2(t) - i_3(t) - i_4(t) + i_5(t) = 0$

We have assumed that the algebraic signs of the currents entering the node are positive and, therefore, that the signs of the currents leaving the node are negative.

If we multiply the foregoing equation by −1, we obtain the expression

$$-i_2(t) + i_4(t) - i_5(t) + i_7(t) = 0$$

which simply states that *the algebraic sum of the currents leaving a node is zero*. Alternatively, we can write the equation as

$$i_2(t) + i_5(t) = i_4(t) + i_7(t)$$

which states that *the sum of the currents entering a node is equal to the sum of the currents leaving the node*. Both of these italicized expressions are alternative forms of Kirchhoff's current law.

Once again it must be emphasized that the latter statement means that the sum of the *variables* that have been defined entering the node is equal to the sum of the *variables* that have been defined leaving the node, not the actual currents. For example, $i_j(t)$ may be defined entering the node, but if its actual value is negative, there will be positive charge leaving the node.

Note carefully that Kirchhoff's current law states that the *algebraic* sum of the currents either entering or leaving a node must be zero. We now begin to see why we stated in Chapter 1 that it is critically important to specify both the magnitude and the direction of a current.

LEARNING Example 2.3

Let us write KCL for every node in the network in Fig. 2.5 assuming that the currents leaving the node are positive.

SOLUTION The KCL equations for nodes 1 through 5 are

$$-i_1(t) + i_2(t) + i_3(t) = 0$$
$$i_1(t) - i_4(t) + i_6(t) = 0$$
$$-i_2(t) + i_4(t) - i_5(t) + i_7(t) = 0$$
$$-i_3(t) + i_5(t) - i_8(t) = 0$$
$$-i_6(t) - i_7(t) + i_8(t) = 0$$

Note carefully that if we add the first four equations we obtain the fifth equation. What does this tell us? Recall that this means that this set of equations is not linearly independent. We can show that the first four equations are, however, linearly independent. Store this idea in memory because it will become very important when we learn how to write the equations necessary to solve for all the currents and voltages in a network in the following chapter.

LEARNING Example 2.4

The network in Fig. 2.5 is represented by the topological diagram shown in Fig. 2.6. We wish to find the unknown currents in the network.

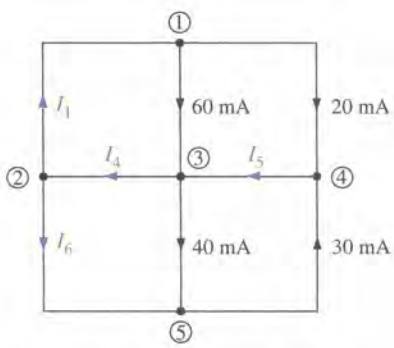


Figure 2.6
Topological diagram for the circuit in Fig. 2.5.

SOLUTION Assuming the currents leaving the node are positive, the KCL equations for nodes 1 through 4 are

$$-I_1 + 0.06 + 0.02 = 0$$
$$I_1 - I_4 + I_6 = 0$$
$$-0.06 + I_4 - I_5 + 0.04 = 0$$
$$-0.02 + I_5 - 0.03 = 0$$

The first equation yields I_1 and the last equation yields I_5. Knowing I_5 we can immediately obtain I_4 from the third equation. Then the values of I_1 and I_4 yield the value of I_6 from the second equation. The results are $I_1 = 80$ mA, $I_4 = 70$ mA, $I_5 = 50$ mA, and $I_6 = -10$ mA.

As indicated earlier, dependent or controlled sources are very important because we encounter them when analyzing circuits containing active elements such as transistors. The following example presents a circuit containing a current-controlled current source.

LEARNING Example 2.5

Let us write the KCL equations for the circuit shown in Fig. 2.7.

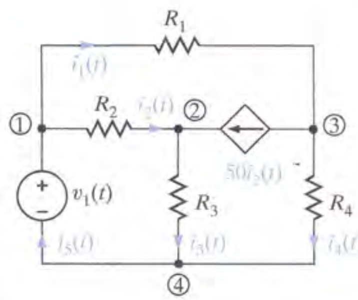


Figure 2.7
Circuit containing a dependent current source.

SOLUTION The KCL equations for nodes 1 through 4 follow.

$$i_1(t) + i_2(t) - i_5(t) = 0$$
$$-i_2(t) + i_3(t) - 50i_2(t) = 0$$
$$-i_1(t) + 50i_2(t) + i_4(t) = 0$$
$$i_5(t) - i_3(t) - i_4(t) = 0$$

If we added the first three equations, we would obtain the negative of the fourth. What does this tell us about the set of equations?

Finally, it is possible to generalize Kirchhoff's current law to include a closed surface. By a closed surface we mean some set of elements completely contained within the surface that are interconnected. Since the current entering each element within the surface is equal to that leaving the element (i.e., the element stores no net charge), it follows that the current entering an interconnection of elements is equal to that leaving the interconnection. Therefore, Kirchhoff's current law can also be stated as follows: *The algebraic sum of the currents entering any closed surface is zero.*

LEARNING Example 2.6

Let us find I_4 and I_1 in the network represented by the topological diagram in Fig. 2.6.

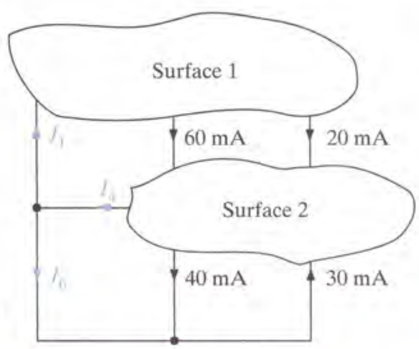


Figure 2.8
Diagram used to demonstrate KCL for a surface.

SOLUTION This diagram is redrawn in Fig. 2.8; node 1 is enclosed in surface 1 and nodes 3 and 4 are enclosed in surface 2. A quick review of the previous example indicates that we derived a value for I_4 from the value of I_5. However, I_5 is now completely enclosed in surface 2. If we apply KCL to surface 2, assuming the currents out of the surface are positive, we obtain

$$I_4 - 0.06 - 0.02 - 0.03 + 0.04 = 0$$

or

$$I_4 = 70 \text{ mA}$$

which we obtained without any knowledge of I_5. Likewise for surface 1, what goes in must come out and, therefore, $I_1 = 80$ mA. The reader is encouraged to cut the network in Fig. 2.6 into two pieces in any fashion and show that KCL is always satisfied at the boundaries.

LEARNING EXTENSIONS

E2.3 Given the networks in Fig. E2.3, find (a) I_1 in Fig. E2.3a and (b) I_T in Fig. E2.3b.

ANSWER (a) $I_1 = -50$ mA, (b) $I_T = 70$ mA.

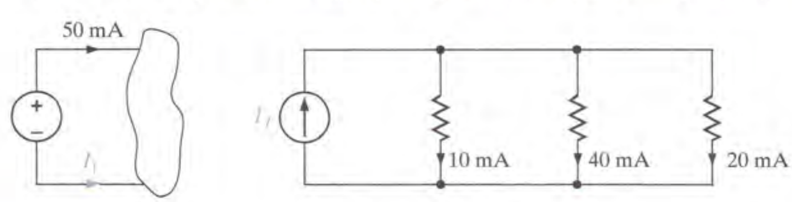


Figure E2.3 (a) (b)

E2.4 Find (a) I_1 in the network in Fig. E2.4a and (b) I_1 and I_2 in the circuit in Fig. E2.4b.

ANSWER (a) $I_1 = 6$ mA, (b) $I_1 = 8$ mA and $I_2 = 5$ mA.

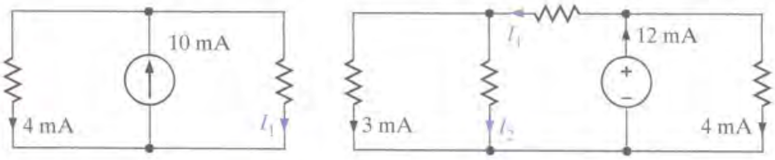


Figure E2.4 (a) (b)

E2.5 Find the current i_x in the circuits in Fig. E2.5.

ANSWER (a) i_x = 4 mA, (b) i_x = 12 mA.

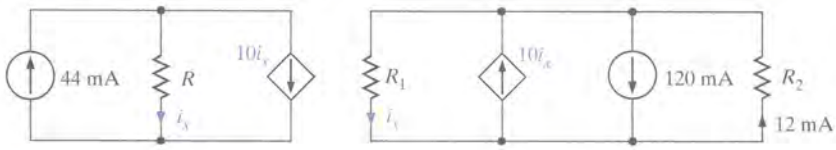
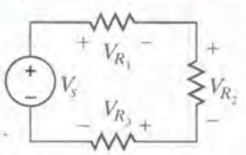


Figure E2.5 (a) (b)

LEARNING by Doing

D 2.4 Write the KVL equation for the following loop, traveling clockwise:



ANSWER
$-V_S + V_{R_1} + V_{R_2} + V_{R_3} = 0$

Kirchhoff's second law, called *Kirchhoff's voltage law* (KVL), states that the *algebraic sum of the voltages around any loop is zero*. As was the case with Kirchhoff's current law, we will defer the proof of this law and concentrate on understanding how to apply it. Once again the reader is cautioned to remember that we are dealing only with lumped-parameter circuits. These circuits are conservative, meaning that the work required to move a unit charge around any loop is zero.

Recall that in Kirchhoff's current law, the algebraic sign was required to keep track of whether the currents were entering or leaving a node. In Kirchhoff's voltage law the algebraic sign is used to keep track of the voltage polarity. In other words, as we traverse the circuit, it is necessary to sum to zero the increases and decreases in energy level. Therefore, it is important we keep track of whether the energy level is increasing or decreasing as we go through each element.

Finally, we employ the convention V_{ab} to indicate the voltage of point a with respect to point b: that is, the variable for the voltage between point a and point b, with point a considered positive relative to point b. Since the potential is measured between two points, it is convenient to use an arrow

LEARNING Example 2.7

Consider the circuit shown in Fig. 2.9. If V_{R_1} and V_{R_2} are known quantities, let us find V_{R_3}.

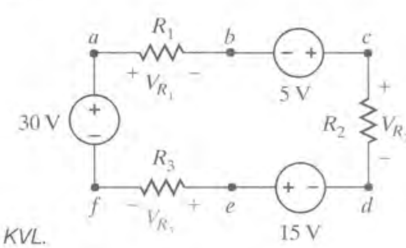


Figure 2.9
Circuit used to illustrate KVL.

SOLUTION In applying KVL, we must traverse the circuit and sum to zero the increases and decreases in energy level. Since the network is a single loop, we have only one closed path. We will adopt a policy of considering an increase in energy level as negative and a decrease in energy level as positive. Using this policy and starting at point a in the network and traversing it in a clockwise direction, we obtain the equation

$$+V_{R_1} - 5 + V_{R_2} - 15 + V_{R_3} - 30 = 0$$

which can be written as

$$+V_{R_1} + V_{R_2} + V_{R_3} = 5 + 15 + 30$$
$$= 50$$

Now suppose that V_{R_1} and V_{R_2} are known to be 18 V and 12 V, respectively. Then $V_{R_3} = 20$ V.

LEARNING Example 2.8

Consider the network in Fig. 2.10.

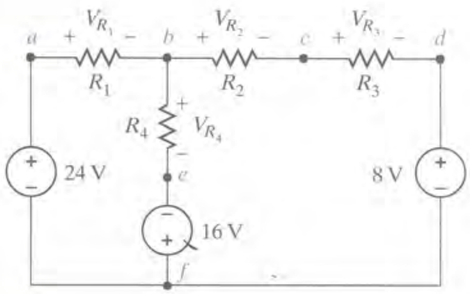


Figure 2.10
Circuit used to explain KVL.

Let us demonstrate that only two of the three possible loop equations are linearly independent.

SOLUTION Note that this network has three closed paths: the left loop, right loop, and outer loop. Applying our policy for writing KVL equations and traversing the left loop starting at point a, we obtain

$$V_{R_1} + V_{R_4} - 16 - 24 = 0$$

The corresponding equation for the right loop starting at point b is

$$V_{R_2} + V_{R_3} + 8 + 16 - V_{R_4} = 0$$

The equation for the outer loop starting at point a is

$$V_{R_1} + V_{R_2} + V_{R_3} + 8 - 24 = 0$$

Note that if we add the first two equations, we obtain the third equation. Therefore, as we indicated in Example 2.3, the three equations are not linearly independent. Once again, we will address this issue in the next chapter and demonstrate that we need only the first two equations to solve for the voltages in the circuit.

between the two points with the head of the arrow located at the positive node. Note that the double-subscript notation, the + and − notation, and the single-headed arrow notation are all the same if the head of the arrow is pointing toward the positive terminal and the first subscript in the double-subscript notation. All of these equivalent forms for labeling voltages are shown in Fig. 2.11. The usefulness of the arrow notation stems from the fact that we may want to label the voltage between two points that are far apart in a network. In this case, the other notations are often confusing.

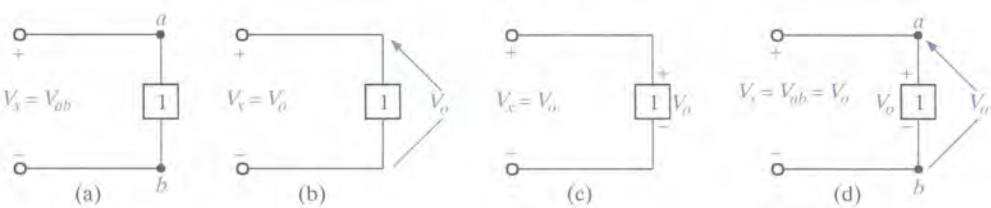


Figure 2.11
Equivalent forms for labeling voltage.

LEARNING Example 2.9

Consider the network in Fig. 2.12a. Let us apply KVL to determine the voltage between two points. Specifically, in terms of the double-subscript notation, let us find V_{ae} and V_{ec}.

SOLUTION The circuit is redrawn in Fig. 2.12b. Since points a and e as well as e and c are not physically close, the arrow notation is very useful. Our approach to determining the unknown voltage is

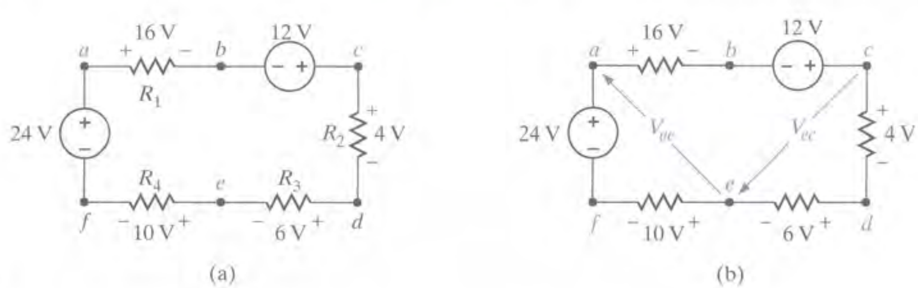


Figure 2.12
Network used in Example 2.9.

to apply KVL with the unknown voltage in the closed path. Therefore, to determine V_{ae} we can use the path *aefa* or *abcdea*. The equations for the two paths in which V_{ae} is the only unknown are

$$V_{ae} + 10 - 24 = 0$$

and

$$16 - 12 + 4 + 6 - V_{ae} = 0$$

Note that both equations yield $V_{ae} = 14$ V. Even before calculating V_{ae}, we could calculate V_{ec} using the path *cdec* or *cefabc*. However, since V_{ae} is now known, we can also use the path *ceabc*. KVL for each of these paths is

$$4 + 6 + V_{ec} = 0$$

$$-V_{ec} + 10 - 24 + 16 - 12 = 0$$

and

$$-V_{ec} - V_{ae} + 16 - 12 = 0$$

Each of these equations yields $V_{ec} = -10$ V.

LEARNING Hint

KVL is an extremely important and useful law.

In general, the mathematical representation of Kirchhoff's voltage law is

$$\sum_{j=1}^{N} v_j(t) = 0 \qquad \textbf{2.8}$$

where $v_j(t)$ is the voltage across the jth branch (with the proper reference direction) in a loop containing N voltages. This expression is analogous to Eq. (2.7) for Kirchhoff's current law.

LEARNING Example 2.10

Given the network in Fig. 2.13 containing a dependent source, let us write the KVL equations for the two closed paths *abda* and *bcdb*.

SOLUTION The two KVL equations are

$$V_{R_1} + V_{R_2} - V_S = 0$$

$$20V_{R_1} + V_{R_3} - V_{R_2} = 0$$

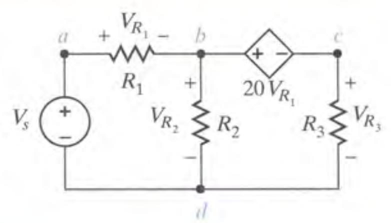


Figure 2.13 Network containing a dependent source.

LEARNING EXTENSIONS

E2.6 Find V_{ad} and V_{eb} in the network in Fig. E2.6.

ANSWER $V_{ad} = 26$ V, $V_{eb} = 10$ V.

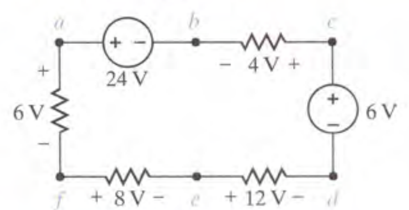


Figure E2.6

E2.7 Find V_{bd} in the circuit in Fig. E2.7.

ANSWER $V_{bd} = 11$ V.

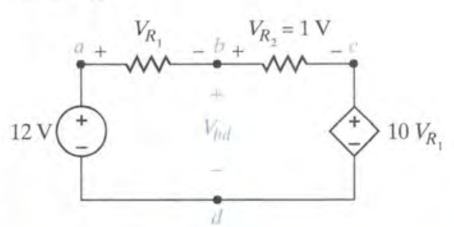


Figure E2.7

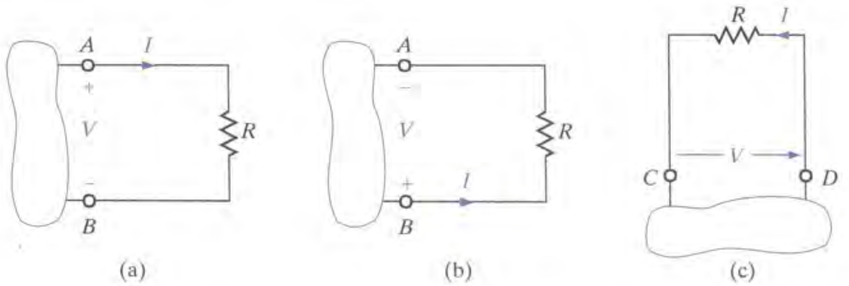


Figure 2.14
Circuits used to explain Ohm's law.

Before proceeding with the analysis of simple circuits, it is extremely important that we emphasize a subtle but very critical point. Ohm's law as defined by the equation $V = IR$ refers to the relationship between the voltage and current as defined in Fig. 2.14a. If the direction of either the current or the voltage, but not both, is reversed, the relationship between the current and the voltage would be $V = -IR$. In a similar manner, given the circuit in Fig. 2.14b, if the polarity of the voltage between the terminals A and B is specified as shown, then the direction of the current I is from point B through R to point A. Likewise, in Fig. 2.14c, if the direction of the current is specified as shown, then the polarity of the voltage must be such that point D is at a higher potential than point C and, therefore, the arrow representing the voltage V is from point C to point D.

LEARNING Hint

The subtleties associated with Ohm's law, as described here, are important and must be adhered to in order to ensure that the variables have the proper sign.

2.3 Single-Loop Circuits

VOLTAGE DIVISION At this point we can begin to apply the laws we have presented earlier to the analysis of simple circuits. To begin, we examine what is perhaps the simplest circuit—a single closed path, or loop, of elements. The elements of a single loop carry the same current and, therefore, are said to be in series. However, we will apply Kirchhoff's voltage law and Ohm's law to the circuit to determine various quantities in the circuit.

Our approach will be to begin with a simple circuit and then generalize the analysis to more complicated ones. The circuit shown in Fig. 2.15 will serve as a basis for discussion. This circuit consists of an independent voltage source that is in series with two resistors. We have assumed that the current flows in a clockwise direction. If this assumption is correct, the solution of the equations that yields the current will produce a positive value. If the current is actually flowing in the opposite direction, the value of the current variable will simply be negative, indicating that the current is flowing in a direction opposite to that assumed. We have also

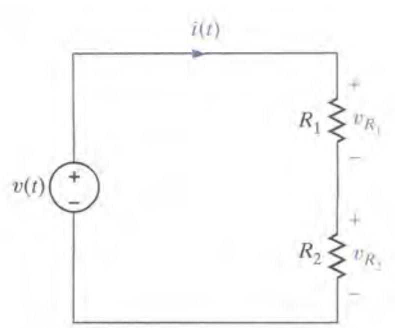


Figure 2.15
Single-loop circuit.

made voltage polarity assignments for v_{R_1} and v_{R_2}. These assignments have been made using the convention employed in our discussion of Ohm's law and our choice for the direction of $i(t)$; that is, the convention shown in Fig. 2.14a.

Applying Kirchhoff's voltage law to this circuit yields

$$-v(t) + v_{R_1} + v_{R_2} = 0$$

or

$$v(t) = v_{R_1} + v_{R_2}$$

However, from Ohm's law we know that

$$v_{R_1} = R_1 i(t)$$
$$v_{R_2} = R_2 i(t)$$

Therefore,

$$v(t) = R_1 i(t) + R_2 i(t)$$

Solving the equation for $i(t)$ yields

$$i(t) = \frac{v(t)}{R_1 + R_2} \qquad \textbf{2.9}$$

Knowing the current, we can now apply Ohm's law to determine the voltage across each resistor:

$$\begin{aligned} v_{R_1} &= R_1 i(t) \\ &= R_1\left[\frac{v(t)}{R_1 + R_2}\right] \\ &= \frac{R_1}{R_1 + R_2} v(t) \end{aligned} \qquad \textbf{2.10}$$

LEARNING Hint

The manner in which voltage divides between two series resistors

Similarly,

$$v_{R_2} = \frac{R_2}{R_1 + R_2} v(t) \qquad \textbf{2.11}$$

Although simple, Eqs. (2.10) and (2.11) are very important because they describe the operation of what is called a *voltage divider*. In other words, the source voltage $v(t)$ *is divided between the resistors* R_1 *and* R_2 *in direct proportion to their resistances.*

In essence, if we are interested in the voltage across the resistor R_1, we bypass the calculation of the current $i(t)$ and simply multiply the input voltage $v(t)$ by the ratio

$$\frac{R_1}{R_1 + R_2}$$

As illustrated in Eq. (2.10), we are using the current in the calculation, but not explicitly.

Note that the equations satisfy Kirchhoff's voltage law, since

$$-v(t) + \frac{R_1}{R_1 + R_2} v(t) + \frac{R_2}{R_1 + R_2} v(t) = 0$$

LEARNING Example 2.11

Consider the circuit shown in Fig. 2.16. The circuit is identical to Fig. 2.15 except that R_1 is a variable resistor such as the volume control for a radio or television set. Suppose that $V_S = 9$ V, $R_1 = 90$ kΩ, and $R_2 = 30$ kΩ.

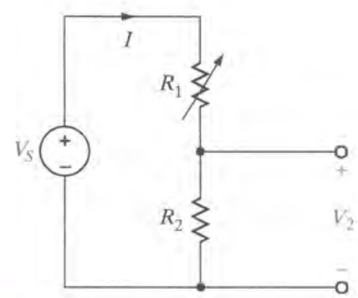


Figure 2.16
Voltage-divider circuit.

Let us examine the change in both the voltage across R_2 and the power absorbed in this resistor as R_1 is changed from 90 kΩ to 15 kΩ.

SOLUTION Since this is a voltage-divider circuit, the voltage V_2 can be obtained directly as

$$V_2 = \left[\frac{R_2}{R_1 + R_2}\right]V_S$$
$$= \left[\frac{30k}{90k + 30k}\right](9)$$
$$= 2.25 \text{ V}$$

Now suppose that the variable resistor is changed from 90 kΩ to 15 kΩ. Then

$$V_2 = \left[\frac{30k}{30k + 15k}\right]9$$
$$= 6 \text{ V}$$

The direct voltage-divider calculation is equivalent to determining the current I and then using Ohm's law to find V_2. Note that the larger voltage is across the larger resistance. This voltage-divider concept and the simple circuit we have employed to describe it are very useful because, as will be shown later, more complicated circuits can be reduced to this form.

Finally, let us determine the instantaneous power absorbed by the resistor R_2 under the two conditions $R_1 = 90$ kΩ and $R_1 = 15$ kΩ. For the case $R_1 = 90$ kΩ, the power absorbed by R_2 is

$$P_2 = I^2R_2 = \left(\frac{9}{120k}\right)^2(30k)$$
$$= 0.169 \text{ mW}$$

In the second case

$$P_2 = \left(\frac{9}{45k}\right)^2(30k)$$
$$= 1.2 \text{ mW}$$

The current in the first case is 75 μA, and in the second case it is 200 μA. Since the power absorbed is a function of the square of the current, the power absorbed in the two cases is quite different.

Let us now demonstrate the practical utility of this simple voltage-divider network.

LEARNING Example 2.12

Consider the circuit in Fig. 2.17a, which is an approximation of a high-voltage dc transmission facility. We have assumed that the bottom portion of the transmission line is a perfect conductor and will justify this assumption in the next chapter. The load can be represented by a resistor of value 183.5 Ω. Therefore, the equivalent circuit of this network is shown in Fig. 2.17b.

Let us determine both the power delivered to the load and the power losses in the line.

SOLUTION Using voltage division, the load voltage is

$$V_{load} = \left[\frac{183.5}{183.5 + 16.5}\right]400k$$
$$= 367 \text{ kV}$$

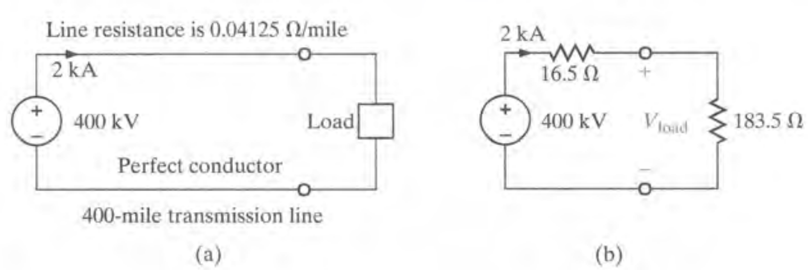


Figure 2.17
A high-voltage dc transmission facility.

(continued)

The input power is 800 MW and the power transmitted to the load is

$$P_{\text{load}} = I^2 R_{\text{load}} = 734 \text{ MW}$$

Therefore, the power loss in the transmission line is

$$P_{\text{line}} = P_{\text{in}} - P_{\text{load}} = I^2 R_{\text{line}} = 66 \text{ MW}$$

Since $P = VI$, suppose now that the utility company supplied power at 200 kV and 4 kA. What effect would this have on our transmission network? Without making a single calculation, we know that because power is proportional to the square of the current, there would be a large increase in the power loss in the line and, therefore, the efficiency of the facility would decrease substantially. That is why, in general, we transmit power at high voltage and low current.

MULTIPLE SOURCE/RESISTOR NETWORKS At this point we wish to extend our analysis to include a multiplicity of voltage sources and resistors. For example, consider the circuit shown in Fig. 2.18a. Here we have assumed that the current flows in a clockwise direction, and we have defined the variable $i(t)$ accordingly. This may or may not be the case, depending on the value of the various voltage sources. Kirchhoff's voltage law for this circuit is

$$+v_{R_1} + v_2(t) - v_3(t) + v_{R_2} + v_4(t) + v_5(t) - v_1(t) = 0$$

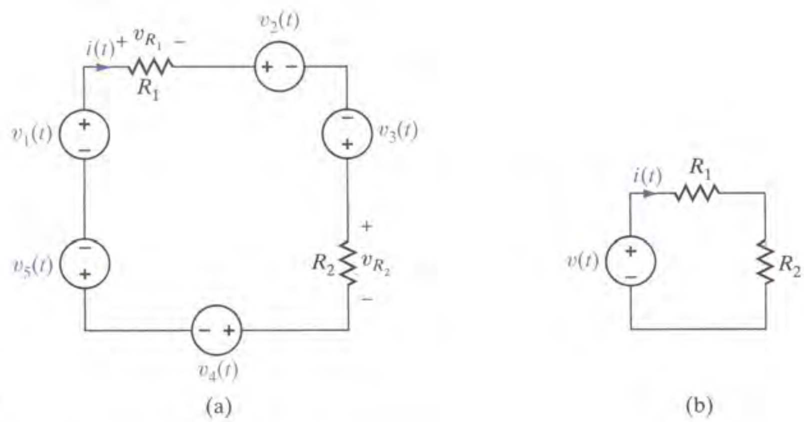


Figure 2.18 Equivalent circuits with multiple sources.

or, using Ohm's law,

$$(R_1 + R_2)i(t) = v_1(t) - v_2(t) + v_3(t) - v_4(t) - v_5(t)$$

which can be written as

$$(R_1 + R_2)i(t) = v(t)$$

where

$$v(t) = v_1(t) + v_3(t) - [v_2(t) + v_4(t) + v_5(t)]$$

so that under the preceding definitions, Fig. 2.18a is equivalent to Fig. 2.18b. In other words, the sum of several voltage sources in series can be replaced by one source whose value is the algebraic sum of the individual sources. This analysis can, of course, be generalized to a circuit with N series sources.

Now consider the circuit with N resistors in series, as shown in Fig. 2.19a. Applying Kirchhoff's voltage law to this circuit yields

$$v(t) = v_{R_1} + v_{R_2} + \cdots + v_{R_N} = R_1 i(t) + R_2 i(t) + \cdots + R_N i(t)$$

and therefore,

$$v(t) = R_S i(t) \tag{2.12}$$

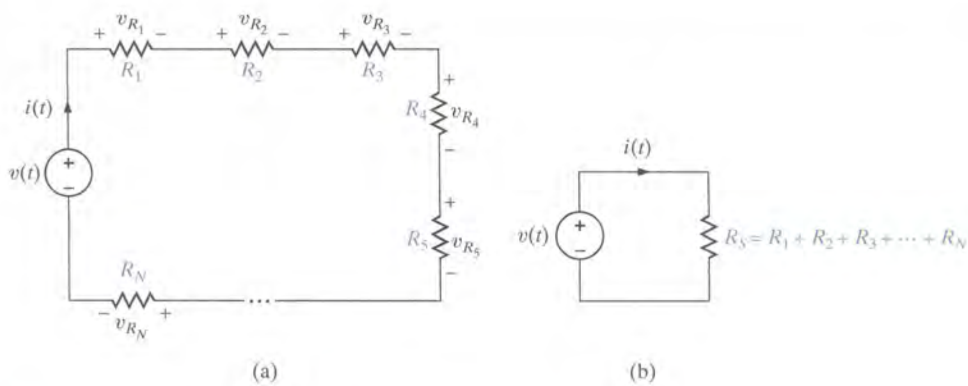


Figure 2.19 *Equivalent circuits.*

where

$$R_S = R_1 + R_2 + \cdots + R_N \quad \textbf{2.13}$$

and hence,

$$i(t) = \frac{v(t)}{R_S} \quad \textbf{2.14}$$

Note also that for any resistor R_i in the circuit, the voltage across R_i is given by the expression

$$v_{R_i} = \frac{R_i}{R_S} v(t) \quad \textbf{2.15}$$

which is the voltage-division property for multiple resistors in series.

Equation (2.13) illustrates that *the equivalent resistance of N resistors in series is simply the sum of the individual resistances.* Thus, using Eq. (2.13), we can draw the circuit in Fig. 2.19b as an equivalent circuit for the one in Fig. 2.19a.

LEARNING Example 2.13

Given the circuit in Fig. 2.20a, let us find I, V_{bd}, and the power absorbed by the 30-kΩ resistor. Finally, let us use voltage division to find V_{bc}.

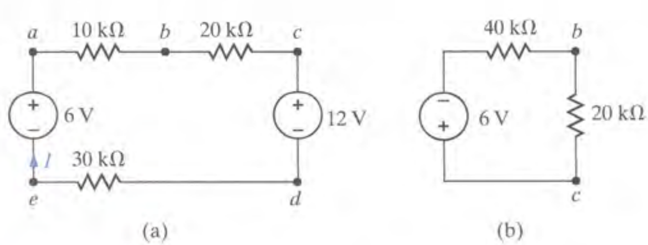


Figure 2.20 *Circuit used in Example 2.13.*

SOLUTION KVL for the network yields the equation

$$10kI + 20kI + 12 + 30kI - 6 = 0$$
$$60kI = -6$$
$$I = -0.1 \text{ mA}$$

Therefore, the magnitude of the current is 0.1 mA, but its direction is opposite to that assumed.

The voltage V_{bd} can be calculated using either of the closed paths *abdea* or *bcdb*. The equations for both cases are

$$10kI + V_{bd} + 30kI - 6 = 0$$

and

$$20kI + 12 - V_{bd} = 0$$

Using $I = -0.1$ mA in either equation yields $V_{bd} = 10$ V. Finally, the power absorbed by the 30-kΩ resistor is

$$P = I^2R = 0.3 \text{ mW}$$

Now from the standpoint of determining the voltage V_{bc}, we can simply add the sources since they are in series, add the remaining resistors since they are in series, and reduce the network to that shown in Fig. 2.20b. Then

$$V_{bc} = \frac{20k}{20k + 40k}(-6)$$
$$= -2 \text{ V}$$

LEARNING EXTENSIONS

E2.8 Find I and V_{bd} in the circuit in Fig. E2.8.

ANSWER $I = -0.05$ mA and $V_{bd} = 10$ V

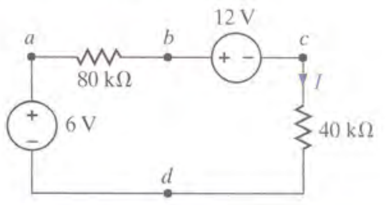


Figure E2.8

E2.9 In the network in Fig. E2.9, if V_{ad} is 3 V, find V_S.

ANSWER $V_S = 9$ V.

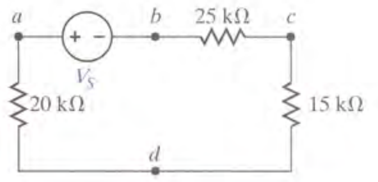


Figure E2.9

2.4 Single-Node-Pair Circuits

CURRENT DIVISION An important circuit is the single-node-pair circuit. In this case the elements have the same voltage across them and, therefore, are in *parallel*. We will, however, apply Kirchhoff's current law and Ohm's law to determine various unknown quantities in the circuit.

Following our approach with the single-loop circuit, we will begin with the simplest case and then generalize our analysis. Consider the circuit shown in Fig. 2.21. Here we have an independent current source in parallel with two resistors.

Since all of the circuit elements are in parallel, the voltage $v(t)$ appears across each of them. Furthermore, an examination of the circuit indicates that the current $i(t)$ is into the upper node of the circuit and the currents $i_1(t)$ and $i_2(t)$ are out of the node. Since KCL essentially states that what goes in must come out, the question we must answer is how $i_1(t)$ and $i_2(t)$ divide the input current $i(t)$.

Applying Kirchhoff's current law to the upper node, we obtain

$$i(t) = i_1(t) + i_2(t)$$

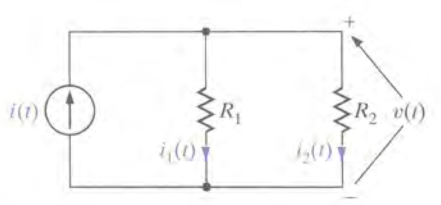


Figure 2.21
Simple parallel circuit.

and, employing Ohm's law, we have

$$\begin{aligned} i(t) &= \frac{v(t)}{R_1} + \frac{v(t)}{R_2} \\ &= \left(\frac{1}{R_1} + \frac{1}{R_2}\right)v(t) \\ &= \frac{v(t)}{R_p} \end{aligned}$$

where

$$\frac{1}{R_p} = \frac{1}{R_1} + \frac{1}{R_2} \qquad 2.16$$

LEARNING Hint

The parallel resistance equation

$$R_p = \frac{R_1 R_2}{R_1 + R_2} \qquad 2.17$$

Therefore, the equivalent resistance of two resistors connected in parallel is equal to the product of their resistances divided by their sum. Note also that this equivalent resistance R_p is always less than either R_1 or R_2. Hence, by connecting resistors in parallel we reduce the overall resistance. In the special case when $R_1 = R_2$, the equivalent resistance is equal to half of the value of the individual resistors.

The manner in which the current $i(t)$ from the source divides between the two branches is called *current division* and can be found from the preceding expressions. For example,

$$\begin{aligned} v(t) &= R_p i(t) \\ &= \frac{R_1 R_2}{R_1 + R_2} i(t) \end{aligned} \qquad 2.18$$

and

LEARNING Hint

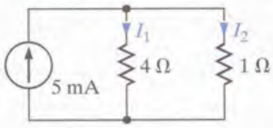
The manner in which current divides between two parallel resistors

$$i_1(t) = \frac{v(t)}{R_1}$$

$$i_1(t) = \frac{R_2}{R_1 + R_2} i(t) \qquad 2.19$$

LEARNING by Doing

D 2.5 Find I_1 and I_2 in the following circuit:



ANSWER $I_1 = 1$ mA
$I_2 = 4$ mA

and

$$\begin{aligned} i_2(t) &= \frac{v(t)}{R_2} \\ &= \frac{R_1}{R_1 + R_2} i(t) \end{aligned} \qquad 2.20$$

Equations (2.19) and (2.20) are mathematical statements of the current-division rule.

LEARNING Example 2.14

Given the network in Fig. 2.22a, let us find I_1, I_2, and V_o.

SOLUTION First, it is important to recognize that the current source feeds two parallel paths. To emphasize this point, the circuit is redrawn as shown in Fig. 2.22b. Applying current division, we obtain

$$I_1 = \left[\frac{40\text{k} + 80\text{k}}{60\text{k} + (40\text{k} + 80\text{k})}\right](0.9 \times 10^{-3})$$
$$= 0.6 \text{ mA}$$

and

$$I_2 = \left[\frac{60\text{k}}{60\text{k} + (40\text{k} + 80\text{k})}\right](0.9 \times 10^{-3})$$
$$= 0.3 \text{ mA}$$

Note that the larger current flows through the smaller resistor, and vice versa. In addition, note that if the resistances of the two paths are equal, the current will divide equally between them. KCL is satisfied since $I_1 + I_2 = 0.9$ mA.

The voltage V_o can be derived using Ohm's law as

$$V_o = 80\text{k}I_2$$
$$= 24 \text{ V}$$

The problem can also be approached in the following manner. The total resistance seen by the current source is 40 kΩ, i.e., 60 kΩ in parallel with the series combination of 40 kΩ and 80 kΩ as shown in Fig. 2.23c. The voltage across the current source is then

$$V_1 = (0.9 \times 10^{-3})40\text{k}$$
$$= 36 \text{ V}$$

Now that V_1 is known, we can apply voltage division to find V_o.

$$V_o = \left(\frac{80\text{k}}{80\text{k} + 40\text{k}}\right)V_1$$
$$= \left(\frac{80\text{k}}{120\text{k}}\right)36$$
$$= 24 \text{ V}$$

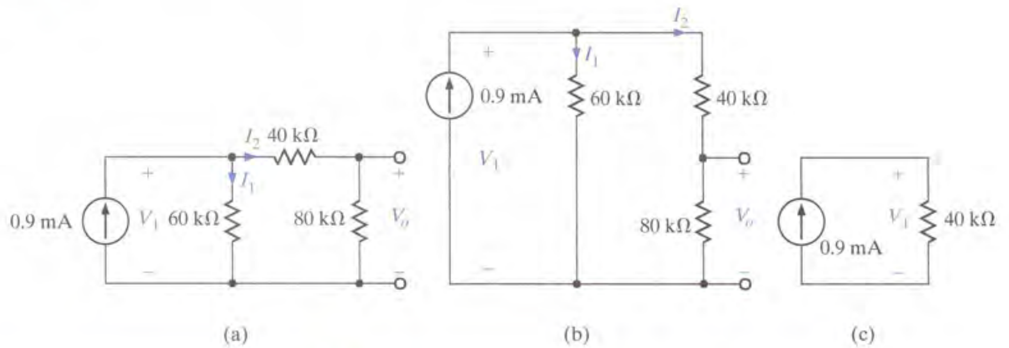


Figure 2.22
Circuit used in Example 2.14.

LEARNING Example 2.15

A typical car stereo consists of a 2-W audio amplifier and two speakers represented by the diagram shown in Fig. 2.23a. The output circuit of the audio amplifier is in essence a 430-mA current source and the speakers each have a resistance of 4 Ω. Let us determine the power absorbed by the speakers.

SOLUTION The audio system can be modeled as shown in Fig. 2.23b. Since the speakers are both 4-Ω devices, the current will split evenly between them and the power absorbed by each speaker is

$$P = I^2R$$
$$= (215 \times 10^{-3})^2(4)$$
$$= 184.9 \text{ mW}$$

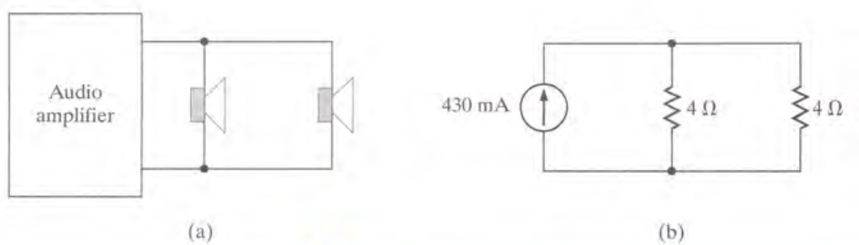


Figure 2.23
Circuits used in Example 2.15.

LEARNING EXTENSION

E2.10 Find the currents I_1 and I_2 and the power absorbed by the 40-kΩ resistor in the network in Fig. E2.10.

ANSWER $I_1 = 12$ mA, $I_2 = -4$ mA, and $P_{40\,k\Omega} = 5.76$ W.

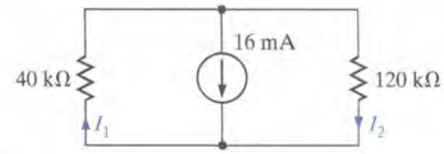


Figure E2.10

MULTIPLE SOURCE/RESISTOR NETWORKS Let us now extend our analysis to include a multiplicity of current sources and resistors in parallel. For example, consider the circuit shown in Fig. 2.24a. We have assumed that the upper node is $v(t)$ volts positive with respect to the lower node. Applying Kirchhoff's current law to the upper node yields

$$i_1(t) - i_2(t) - i_3(t) + i_4(t) - i_5(t) - i_6(t) = 0$$

or

$$i_1(t) - i_3(t) + i_4(t) - i_6(t) = i_2(t) + i_5(t)$$

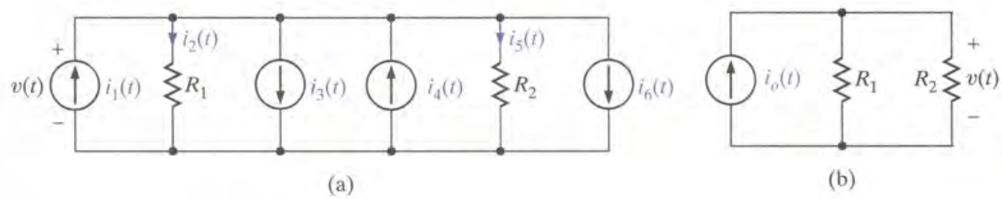


Figure 2.24 Equivalent circuits.

The terms on the left side of the equation all represent sources that can be combined algebraically into a single source; that is,

$$i_o(t) = i_1(t) - i_3(t) + i_4(t) - i_6(t)$$

which effectively reduces the circuit in Fig. 2.24a to that in Fig. 2.24b. We could, of course, generalize this analysis to a circuit with N current sources. Using Ohm's law, the currents on the right side of the equation can be expressed in terms of the voltage and individual resistances so that the KCL equation reduces to

$$i_o(t) = \left(\frac{1}{R_1} + \frac{1}{R_2}\right)v(t)$$

Now consider the circuit with N resistors in parallel, as shown in Fig. 2.25a. Applying Kirchhoff's current law to the upper node yields

$$\begin{aligned} i_o(t) &= i_1(t) + i_2(t) + \cdots + i_N(t) \\ &= \left(\frac{1}{R_1} + \frac{1}{R_2} + \cdots + \frac{1}{R_N}\right)v(t) \end{aligned} \qquad \textbf{2.21}$$

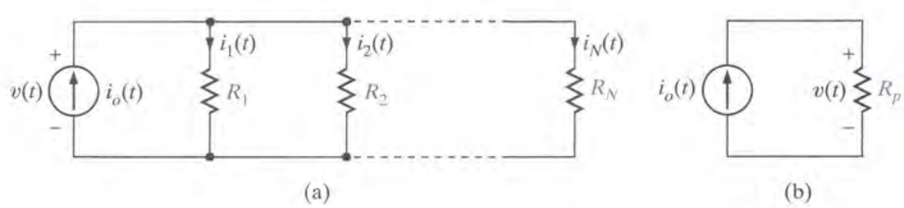


Figure 2.25 Equivalent circuits.

or

$$i_o(t) = \frac{v(t)}{R_p} \qquad \textbf{2.22}$$

where

$$\frac{1}{R_p} = \sum_{i=1}^{N} \frac{1}{R_i} \qquad \textbf{2.23}$$

so that as far as the source is concerned, Fig. 2.25a can be reduced to an equivalent circuit, as shown in Fig. 2.25b.

The current division for any branch can be calculated using Ohm's law and the preceding equations. For example, for the *j*th branch in the network of Fig. 2.25a,

$$i_j(t) = \frac{v(t)}{R_j}$$

Using Eq. (2.22), we obtain

$$i_j(t) = \frac{R_p}{R_j} i_o(t) \qquad \textbf{2.24}$$

which defines the current-division rule for the general case.

LEARNING Example 2.16

Given the circuit in Fig. 2.26a, we wish to find the current in the 12-kΩ load resistor.

SOLUTION To simplify the network in Fig. 2.26a, we add the current sources algebraically and combine the parallel resistors in the following manner:

$$\frac{1}{R_p} = \frac{1}{18\text{k}} + \frac{1}{9\text{k}} + \frac{1}{12\text{k}}$$

$$R_p = 4\ \text{k}\Omega$$

Using these values we can reduce the circuit in Fig. 2.26a to that in Fig. 2.26b. Now, applying current division, we obtain

$$I_L = -\left[\frac{4\text{k}}{4\text{k} + 12\text{k}}\right](1 \times 10^{-3})$$

$$= -0.25\ \text{mA}$$

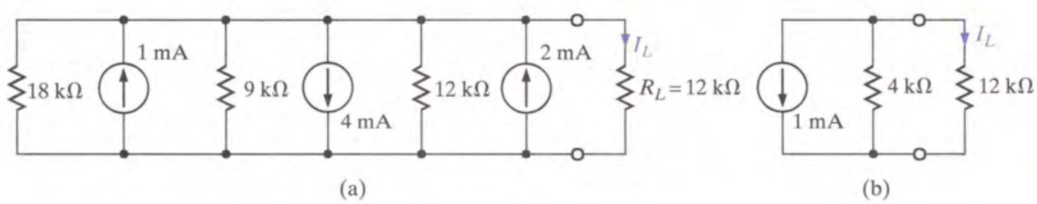


Figure 2.26 Circuits used in Example 2.16.

LEARNING EXTENSION

E2.11 Find the power absorbed by the 6-kΩ resistor in the network in Fig. E2.11.

ANSWER $P = 2.67$ mW.

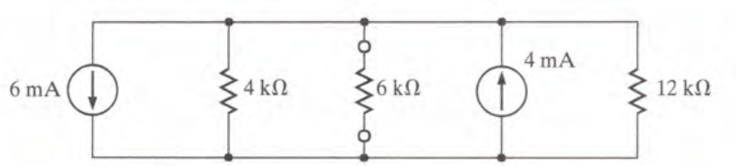


Figure E2.11

2.5 Series and Parallel Resistor Combinations

We have shown in our earlier developments that the equivalent resistance of N resistors in series is

$$R_S = R_1 + R_2 + \cdots + R_N \qquad \textbf{2.25}$$

and the equivalent resistance of N resistors in parallel is found from

$$\frac{1}{R_p} = \frac{1}{R_1} + \frac{1}{R_2} + \cdots + \frac{1}{R_N} \qquad \textbf{2.26}$$

Let us now examine some combinations of these two cases.

LEARNING Example 2.17

We wish to determine the resistance at terminals A-B in the network in Fig. 2.27a.

SOLUTION Starting at the opposite end of the network from the terminals and combining resistors as shown in the sequence of circuits in Fig. 2.27, we find that the equivalent resistance at the terminals is 5 kΩ.

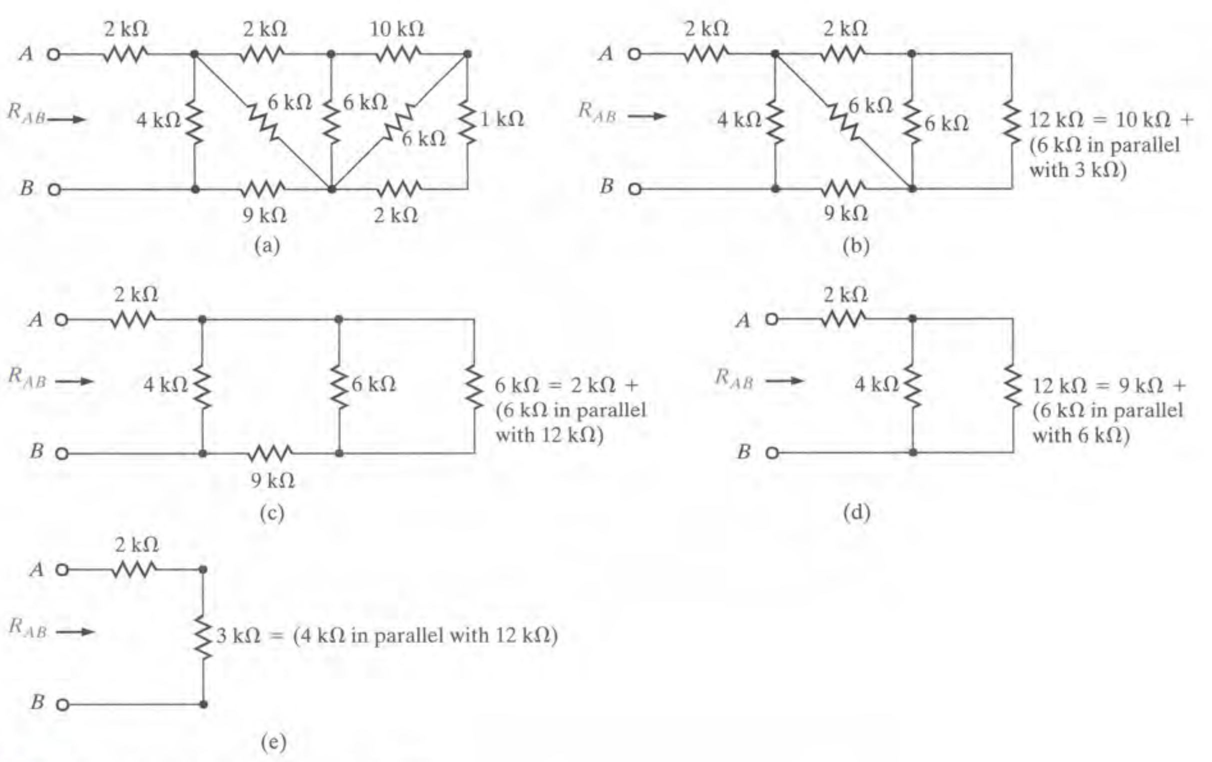


Figure 2.27 Simplification of a resistance network.

LEARNING EXTENSIONS

E2.12 Find the equivalent resistance at the terminals *A-B* in the network in Fig. E2.12.

ANSWER $R_{AB} = 22\ \text{k}\Omega$.

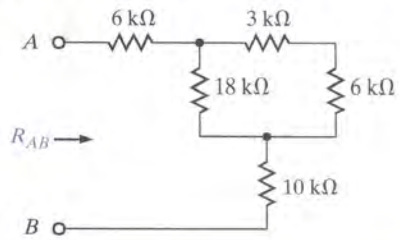


Figure E2.12

E2.13 Find the equivalent resistance at the terminals *A-B* in the circuit in Fig. E2.13.

ANSWER $R_{AB} = 3\ \text{k}\Omega$.

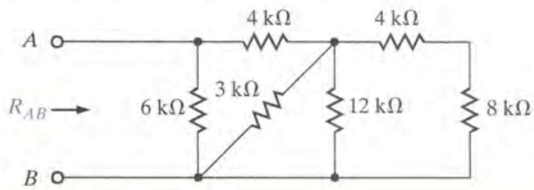


Figure E2.13

Problem-Solving Strategy Simplifying Resistor Combinations

When trying to determine the equivalent resistance at a pair of terminals of a network composed of an interconnection of numerous resistors, it is recommended that the analysis begin at the end of the network opposite the terminals. Two or more resistors are combined to form a single resistor, thus simplifying the network by reducing the number of components as the analysis continues in a steady progression toward the terminals. The simplification involves the following:

1. *Resistors in series.* Resistors R_1 and R_2 are in series if they are connected in tandem and carry exactly the same current. They can then be combined into a single resistor R_S, where $R_S = R_1 + R_2$.
2. *Resistors in parallel.* Resistors R_1 and R_2 are in parallel if they are connected to the same two nodes and have exactly the same voltage across their terminals. They can then be combined into a single resistor R_p, where $R_p = R_1R_2/(R_1 + R_2)$.

These two combinations are used repeatedly, as needed, to reduce the network to a single resistor at the pair of terminals.

RESISTOR SPECIFICATIONS Some important parameters that are used to specify resistors are the resistor's value, tolerance, and power rating. The tolerance specifications for resistors are typically 5% and 10%. A listing of standard resistor values with their specified tolerances is shown in Table 2.1.

Table 2.1 Standard resistor values for 5% and 10% tolerances (10% values shown in boldface)

1.0	10	100	1.0k	10k	100k	1.0M	10M
1.1	11	110	1.1k	11k	110k	**1.1M**	11M
1.2	**12**	**120**	**1.2k**	**12k**	**120k**	**1.2M**	**12M**
1.3	13	130	1.3k	13k	130k	1.3M	13M
1.5	**15**	**150**	**1.5k**	**15k**	**150k**	**1.5M**	**15M**
1.6	16	160	1.6k	16k	160k	1.6M	16M
1.8	**18**	**180**	**1.8k**	**18k**	**180k**	**1.8M**	**18M**
2.0	20	200	2.0k	20k	200k	2.0M	20M
2.2	**22**	**220**	**2.2k**	**22k**	**220k**	**2.2M**	**22M**
2.4	24	240	2.4k	24k	240k	2.4M	
2.7	**27**	**270**	**2.7k**	**27k**	**270k**	**2.7M**	
3.0	30	300	3.0k	30k	300k	3.0M	
3.3	**33**	**330**	**3.3k**	**33k**	**330k**	**3.3M**	
3.6	36	360	3.6k	36k	360k	3.6M	
3.9	**39**	**390**	**3.9k**	**39k**	**390k**	**3.9M**	
4.3	43	430	4.3k	43k	430k	4.3M	
4.7	**47**	**470**	**4.7k**	**47k**	**470k**	**4.7M**	
5.1	51	510	5.1k	51k	510k	5.1M	
5.6	**56**	**560**	**5.6k**	**56k**	**560k**	**5.6M**	
6.2	62	620	6.2k	62k	620k	6.2M	
6.8	**68**	**680**	**6.8k**	**68k**	**680k**	**6.8M**	
7.5	75	750	7.5k	75k	750k	7.5M	
8.2	**82**	**820**	**8.2k**	**82k**	**820k**	**8.2M**	
9.1	91	910	9.1k	91k	910k	9.1M	

The power rating for a resistor specifies the maximum power that can be dissipated by the resistor. Some typical power ratings for resistors are $\frac{1}{4}$ W, $\frac{1}{2}$ W, 1 W, 2 W, and so forth, up to very high values for high-power applications. Thus in selecting a resistor for some particular application, one important selection criterion is the expected power dissipation.

LEARNING Example 2.18

Given the network in Fig. 2.28, we wish to find the range for both the current and power dissipation in the resistor if R is a 2.7-kΩ resistor with a tolerance of 10%.

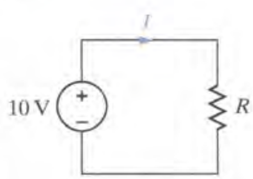


Figure 2.28
Circuit used in Example 2.18.

SOLUTION Using the equations $I = V/R = 10/R$ and $P = V^2/R = 100/R$, the minimum and maximum values for the resistor, current, and power are outlined next.

Minimum resistor value $= R(1 - 0.1) = 0.9\,R = 2.43\text{ k}\Omega$

Maximum resistor value $= R(1 + 0.1) = 1.1\,R = 2.97\text{ k}\Omega$

Minimum current value $= 10/2970 = 3.37$ mA

Maximum current value $= 10/2430 = 4.12$ mA

Minimum power value $= 100/2970 = 33.7$ mW

Maximum power value $= 100/2430 = 41.2$ mW

Thus the range for the current and power are 3.37 mA to 4.12 mA and 33.7 mW to 41.2 mW, respectively.

LEARNING Example 2.19

Given the network shown in Fig. 2.29: (a) find the required value for the resistor R; (b) use Table 2.1 to select a standard 10% tolerance resistor for R; (c) using the resistor selected in (b), determine the voltage across the 3.9-kΩ resistor; (d) calculate the percent error in the voltage V_1, if the standard resistor selected in (b) is used; and (e) determine the power rating for this standard component.

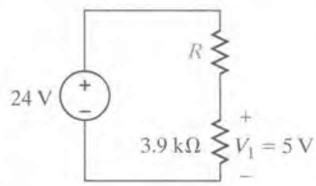


Figure 2.29
Circuit used in Example 2.19.

SOLUTION

(a) Using KVL, the voltage across R is 19 V. Then using Ohm's law, the current in the loop is

$$I = 5/3.9\text{k} = 1.282 \text{ mA}$$

The required value of R is then

$$R = 19/0.001282 = 14.82 \text{ k}\Omega$$

(b) As shown in Table 2.1, the nearest standard 10% tolerance resistor is 15 kΩ.

(c) Using the standard 15-kΩ resistor, the actual current in the circuit is

$$I = 24/18.9\text{k} = 1.2698 \text{ mA}$$

and the voltage across the 3.9-kΩ resistor is

$$V = IR = (0.0012698)(3.9\text{k}) = 4.952 \text{ V}$$

(d) The percent error involved in using the standard resistor is

$$\% \text{ Error} = (4.952 - 5)/5 \times 100 = -0.96\%$$

(e) The power absorbed by the resistor R is then

$$P = IR = (0.0012698)^2(15\text{k}) = 24.2 \text{ mW}$$

Therefore, even a quarter-watt resistor is adequate in this application.

LEARNING by Doing

D2.6 Find the possible range of resistance for the following resistors:

(a) A 27-Ω resistor with a tolerance of 5%

(b) A 1.5-kΩ resistor with a tolerance of 10%

ANSWER *(a) 25.65 Ω to 28.35 Ω, (b) 1.35 kΩ to 1.65 kΩ.*

2.6 Circuits with Series-Parallel Combinations of Resistors

At this point we have learned many techniques that are fundamental to circuit analysis. Now we wish to apply them and show how they can be used in concert to analyze circuits. We will illustrate their application through a number of examples that will be treated in some detail.

LEARNING Example 2.20

We wish to find all the currents and voltages labeled in the ladder network shown in Fig. 2.30a.

SOLUTION To begin our analysis of the network, we start at the right end of the circuit and combine the resistors to determine the total resistance seen by the 12-V source. This will allow us to calculate the current I_1. Then employing KVL, KCL, Ohm's law, and/or voltage and current division, we will be able to calculate all currents and voltages in the network.

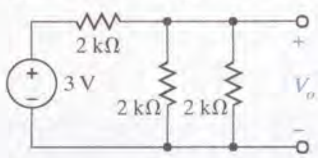
At the right end of the circuit, the 9-kΩ and 3-kΩ resistors are in series and, thus, can be combined into one equivalent 12-kΩ resistor. This resistor is in parallel with the 4-kΩ resistor, and their combination yields an equivalent 3-kΩ resistor, shown at the right edge of the circuit in Fig. 2.30b. In Fig. 2.30b the two 3-kΩ resistors are in series and their combination is in parallel with the 6-kΩ resistor. Combining all three resistances yields the circuit shown in Fig. 2.30c.

Applying Kirchhoff's voltage law to the circuit in Fig. 2.30c yields

$$I_1(9\text{k} + 3\text{k}) = 12$$
$$I_1 = 1 \text{ mA}$$

LEARNING by Doing

D 2.7 Find V_o in the following network:



ANSWER $V_o = 1$ V

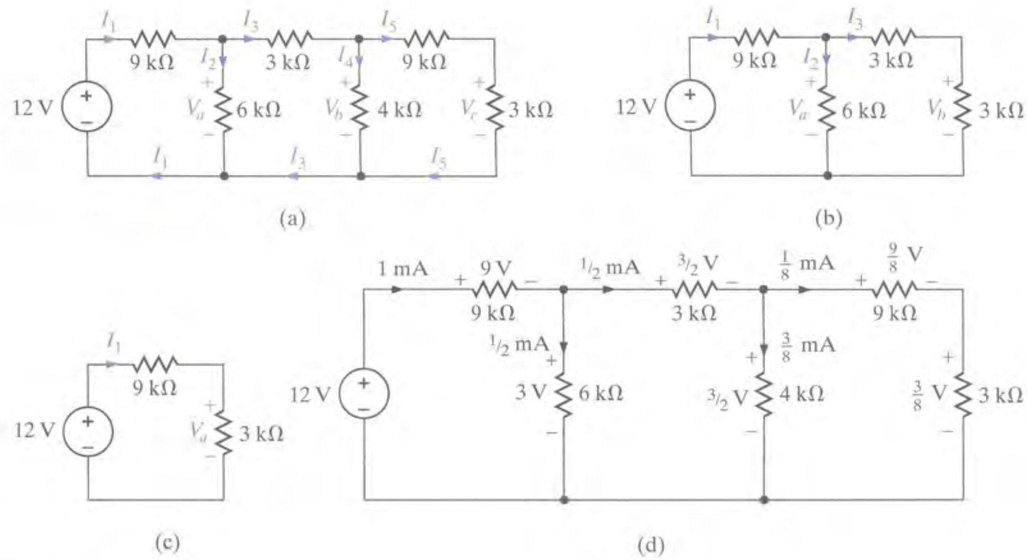


Figure 2.30
Analysis of a ladder network.

V_a can be calculated from Ohm's law as

$$\begin{aligned} V_a &= I_1(3\text{k}) \\ &= 3\text{ V} \end{aligned}$$

or, using Kirchhoff's voltage law,

$$\begin{aligned} V_a &= 12 - 9\text{k}I_1 \\ &= 12 - 9 \\ &= 3\text{ V} \end{aligned}$$

Knowing I_1 and V_a, we can now determine all currents and voltages in Fig. 2.30b. Since $V_a = 3$ V, the current I_2 can be found using Ohm's law as

$$\begin{aligned} I_2 &= \frac{3}{6\text{k}} \\ &= \frac{1}{2}\text{ mA} \end{aligned}$$

Then, using Kirchhoff's current law, we have

$$\begin{aligned} I_1 &= I_2 + I_3 \\ 1 \times 10^{-3} &= \frac{1}{2} \times 10^{-3} + I_3 \\ I_3 &= \frac{1}{2}\text{ mA} \end{aligned}$$

Note that the I_3 could also be calculated using Ohm's law:

$$\begin{aligned} V_a &= (3\text{k} + 3\text{k})I_3 \\ I_3 &= \frac{3}{6\text{k}} \\ &= \frac{1}{2}\text{ mA} \end{aligned}$$

Applying Kirchhoff's voltage law to the right-hand loop in Fig. 2.30b yields

$$\begin{aligned} V_a - V_b &= 3\text{k}I_3 \\ 3 - V_b &= \frac{3}{2} \\ V_b &= \frac{3}{2}\text{ V} \end{aligned}$$

or, since V_b is equal to the voltage drop across the 3-kΩ resistor, we could use Ohm's law as

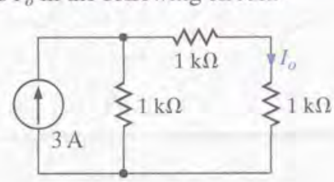
$$\begin{aligned} V_b &= 3\text{k}I_3 \\ &= \frac{3}{2}\text{ V} \end{aligned}$$

We are now in a position to calculate the final unknown currents and voltages in Fig. 2.30a. Knowing V_b, we can calculate I_4 using Ohm's law as

$$\begin{aligned} V_b &= 4\text{k}I_4 \\ I_4 &= \frac{\frac{3}{2}}{4\text{k}} \\ &= \frac{3}{8}\text{ mA} \end{aligned}$$

LEARNING by Doing

D 2.8 Find I_o in the following circuit:



ANSWER $I_o = 1$ A

(continued)

Then, from Kirchhoff's current law, we have

$$I_3 = I_4 + I_5$$
$$\frac{1}{2} \times 10^{-3} = \frac{3}{8} \times 10^{-3} + I_5$$
$$I_5 = \frac{1}{8}\text{ mA}$$

We could also have calculated I_5 using the current-division rule. For example,

$$I_5 = \frac{4\text{k}}{4\text{k} + (9\text{k} + 3\text{k})} I_3 = \frac{1}{8}\text{ mA}$$

Finally, V_c can be computed as

$$V_c = I_5(3\text{k}) = \frac{3}{8}\text{ V}$$

V_c can also be found using voltage division (i.e., the voltage V_b will be divided between the 9-kΩ and 3-kΩ resistors). Therefore,

$$V_c = \left[\frac{3\text{k}}{3\text{k} + 9\text{k}}\right] V_b = \frac{3}{8}\text{ V}$$

Note that Kirchhoff's current law is satisfied at every node and Kirchhoff's voltage law is satisfied around every loop, as shown in Fig. 2.30d.

The following example is, in essence, the reverse of the previous example in that we are given the current in some branch in the network and are asked to find the value of the input source.

LEARNING Example 2.21

Given the circuit in Fig. 2.31 and $I_4 = \frac{1}{2}$ mA, let us find the source voltage V_o.

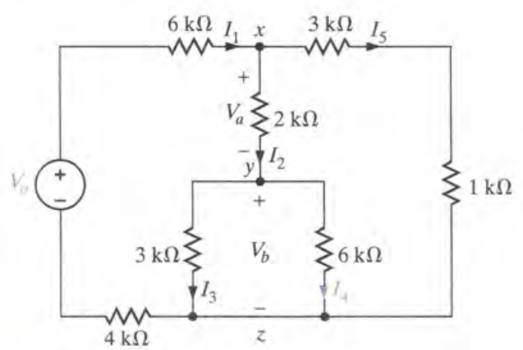


Figure 2.31 Example circuit for analysis.

SOLUTION If $I_4 = \frac{1}{2}$ mA, then from Ohm's law, $V_b = 3$ V. V_b can now be used to calculate $I_3 = 1$ mA. Kirchhoff's current law applied at node y yields

$$I_2 = I_3 + I_4 = 1.5\text{ mA}$$

Then, from Ohm's law, we have

$$V_a = (1.5 \times 10^{-3})(2\text{k}) = 3\text{ V}$$

Since $V_a + V_b$ is now known, I_5 can be obtained:

$$I_5 = \frac{V_a + V_b}{3\text{k} + 1\text{k}} = 1.5\text{ mA}$$

Applying Kirchhoff's current law at node x yields

$$I_1 = I_2 + I_5 = 3\text{ mA}$$

Now KVL applied to any closed path containing V_o will yield the value of this input source. For example, if the path is the outer loop, KVL yields

$$-V_o + 6\text{k}I_1 + 3\text{k}I_5 + 1\text{k}I_5 + 4\text{k}I_1 = 0$$

Since $I_1 = 3$ mA and $I_5 = 1.5$ mA,

$$V_o = 36\text{ V}$$

If we had selected the path containing the source and the points x, y, and z, we would obtain

$$-V_o + 6\text{k}I_1 + V_a + V_b + 4\text{k}I_1 = 0$$

Once again, this equation yields

$$V_o = 36\text{ V}.$$

LEARNING EXTENSIONS

E2.14 Find V_o in the network in Fig. E2.14.

ANSWER $V_o = 2$ V.

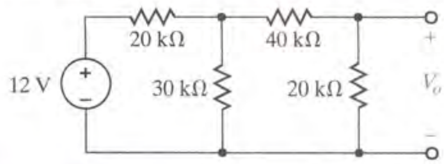


Figure E2.14

E2.15 Find V_S in the ci[illegible]

ANSWER $V_S = 9$ V.

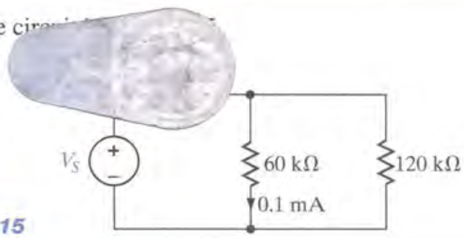


Figure E2.15

E2.16 Find I_S in the circuit in Fig. E2.16.

ANSWER $I_S = 0.3$ mA.

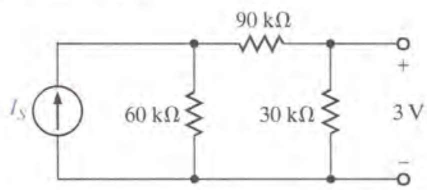


Figure E2.16

Problem-Solving Strategy
Analyzing Circuits Containing a Single Source and a Series-Parallel Interconnection of Resistors

Step 1. Systematically reduce the resistive network so that the resistance seen by the source is represented by a single resistor.

Step 2. Determine the source current for a voltage source or the source voltage if a current source is present.

Step 3. Expand the network, retracing the simplification steps, and apply Ohm's law, KVL, KCL, voltage division, and current division to determine all currents and voltages in the network.

2.7 Wye ⇌ Delta Transformations

To provide motivation for this topic, consider the circuit in Fig. 2.32. Note that this network has essentially the same number of elements as contained in our recent examples. However, when we attempt to reduce the circuit to an equivalent network containing the source V_1 and an equivalent resistor R, we find that nowhere is a resistor in series or parallel with another. Therefore, we cannot attack the problem directly using the techniques that we have learned thus far. We can, however, replace one portion of the network with an equivalent circuit, and this conversion will permit us, with ease, to reduce the combination of resistors to a single equivalent resistance. This conversion is called the wye-to-delta or delta-to-wye transformation.

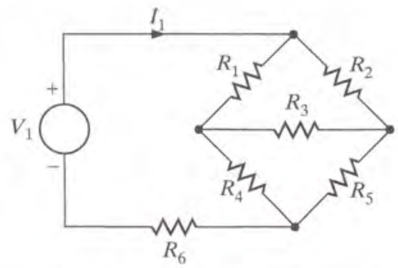


Figure 2.32
Network used to illustrate the need for the wye ⇌ delta transformation.

Consider the networks shown in Fig. 2.33. Note that the resistors in Fig. 2.33a form a Δ (delta) and the resistors in Fig. 2.33b form a Y (wye). If both of these configurations are connected at only three terminals *a*, *b*, and *c*, it would be very advantageous if an equivalence could be established between them. It is, in fact, possible to relate the resistances of one network to those of the other such that their terminal characteristics are the same. This relationship between the two network configurations is called the Y-Δ transformation.

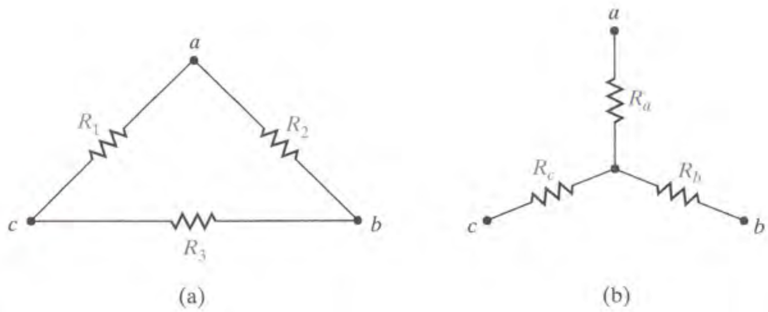


Figure 2.33
Delta and wye resistance networks.

The transformation that relates the resistances R_1, R_2, and R_3 to the resistances R_a, R_b, and R_c is derived as follows. For the two networks to be equivalent at each corresponding pair of terminals, it is necessary that the resistance at the corresponding terminals be equal (e.g., the resistance at terminals *a* and *b* with *c* open-circuited must be the same for both networks). Therefore, if we equate the resistances for each corresponding set of terminals, we obtain the following equations:

$$\begin{aligned} R_{ab} &= R_a + R_b = \frac{R_2(R_1 + R_3)}{R_2 + R_1 + R_3} \\ R_{bc} &= R_b + R_c = \frac{R_3(R_1 + R_2)}{R_3 + R_1 + R_2} \\ R_{ca} &= R_c + R_a = \frac{R_1(R_2 + R_3)}{R_1 + R_2 + R_3} \end{aligned} \tag{2.27}$$

Solving this set of equations for R_a, R_b, and R_c yields

$$\begin{aligned} R_a &= \frac{R_1 R_2}{R_1 + R_2 + R_3} \\ R_b &= \frac{R_2 R_3}{R_1 + R_2 + R_3} \\ R_c &= \frac{R_1 R_3}{R_1 + R_2 + R_3} \end{aligned} \tag{2.28}$$

Similarly, if we solve Eq. (2.27) for R_1, R_2, and R_3, we obtain

$$R_1 = \frac{R_aR_b + R_bR_c + R_aR_c}{R_b}$$
$$R_2 = \frac{R_aR_b + R_bR_c + R_aR_c}{R_c} \qquad \textbf{2.29}$$
$$R_3 = \frac{R_aR_b + R_bR_c + R_aR_c}{R_a}$$

Equations (2.28) and (2.29) are general relationships and apply to any set of resistances connected in a Y or Δ. For the balanced case where $R_a = R_b = R_c$ and $R_1 = R_2 = R_3$, the equations above reduce to

$$R_Y = \tfrac{1}{3}R_\Delta \qquad \textbf{2.30}$$

and

$$R_\Delta = 3R_Y \qquad \textbf{2.31}$$

It is important to note that it is not necessary to memorize the formulas in Eqs. (2.28) and (2.29). Close inspection of these equations and Fig. 2.33 illustrates a definite pattern to the relationships between the two configurations. For example, the resistance connected to point *a* in the wye (i.e., R_a) is equal to the product of the two resistors in the Δ that are connected to point *a* divided by the sum of all the resistances in the delta. R_b and R_c are determined in a similar manner. Similarly, there are geometrical patterns associated with the equations for calculating the resistors in the delta as a function of those in the wye.

Let us now examine the use of the delta ⇌ wye transformation in the solution of a network problem.

LEARNING Example 2.22

Given the network in Fig. 2.34a, let us find the source current I_S.

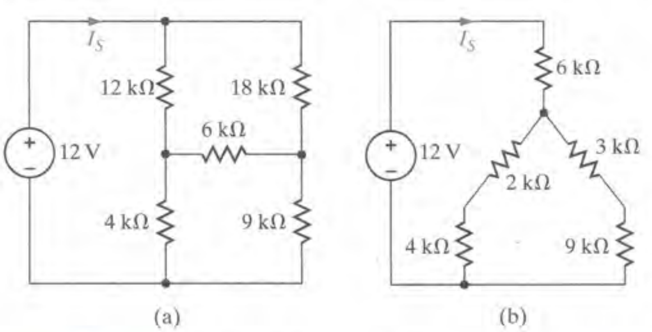


Figure 2.34 *Circuits used in Example 2.22.*

SOLUTION Note that none of the resistors in the circuit are in series or parallel. However, careful examination of the network indicates that the 12k-, 6k-, and 18k-ohm resistors, as well as the 4k-, 6k-, and 9k-ohm resistors each form a delta that can be converted to a wye. Furthermore, the 12k-, 6k-, and 4k-ohm resistors, as well as the 18k-, 6k-, and 9k-ohm resistors each form a wye that can be converted to a delta. Any one of these conversions will lead to a solution. We will perform a delta-to-wye transformation on the 12k-, 6k-, and 18k-ohm resistors, which leads to the circuit in Fig. 2.34b. The 2k- and 4k-ohm resistors, like the 3k- and 9k-ohm resistors, are in series and their parallel combination yields a 4k-ohm resistor. Thus, the source current is

$$I_S = 12/(6\text{k} + 4\text{k})$$
$$= 1.2 \text{ mA}.$$

2.8 Circuits with Dependent Sources

In Chapter 1 we outlined the different kinds of dependent sources. These controlled sources are extremely important because they are used to model physical devices such as *npn* and *pnp* bipolar junction transistors (BJTs) and field-effect transistors (FETs) that are either metal-oxide-semiconductor field-effect transistors (MOSFETs) or insulated-gate field-effect transistors (IGFETs). These basic structures are, in turn, used to make analog and digital devices. A typical analog device is an operational amplifier (op-amp). Typical digital devices are random access memories (RAMs), read-only memories (ROMs), and microprocessors. We will now show how to solve simple one-loop and one-node circuits that contain these dependent sources. Although the following examples are fairly simple, they will serve to illustrate the basic concepts.

Problem-Solving Strategy Circuits with Dependent Sources

Step 1. When writing the KVL and/or KCL equations for the network, treat the dependent source as though it were an independent source.

Step 2. Write the equation that specifies the relationship of the dependent source to the controlling parameter.

Step 3. Solve the equations for the unknowns. Be sure that the number of linearly independent equations matches the number of unknowns.

The following four examples will each illustrate one of the four types of dependent sources: current-controlled voltage source, current-controlled current source, voltage-controlled voltage source, and voltage-controlled current source.

LEARNING Example 2.23

Let us determine the voltage V_o in the circuit in Fig. 2.35.

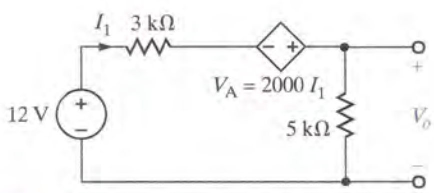


Figure 2.35 Circuit used in Example 2.23.

SOLUTION Applying KVL, we obtain

$$-12 + 3kI_1 - V_A + 5kI_1 = 0$$

where

$$V_A = 2000I_1$$

and the units of the multiplier, 2000, are ohms. Solving these equations yields

$$I_1 = 2\text{ mA}$$

Then

$$V_o = (5\text{ k})I_1$$
$$= 10\text{ V}$$

LEARNING Example 2.24

Given the circuit in Fig. 2.36 containing a current-controlled current source, let us find the voltage V_o.

SOLUTION Applying KCL at the top node, we obtain

$$10 \times 10^{-3} + \frac{V_S}{2k + 4k} + \frac{V_S}{3k} - 4I_o = 0$$

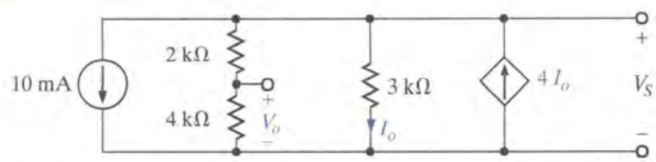


Figure 2.36 Circuit used in Example 2.24.

where

$$I_o = \frac{V_S}{3k}$$

Substituting this expression for the controlled source into the KCL equation yields

$$10^{-2} + \frac{V_S}{6k} + \frac{V_S}{3k} - \frac{4V_S}{3k} = 0$$

Solving this equation for V_S, we obtain

$$V_S = 12 \text{ V}$$

The voltage V_o can now be obtained using a simple voltage divider; that is,

$$V_o = \left[\frac{4k}{2k + 4k}\right] V_S = 8 \text{ V}$$

LEARNING Example 2.25

The network in Fig. 2.37 contains a voltage-controlled voltage source. We wish to find V_o in this circuit.

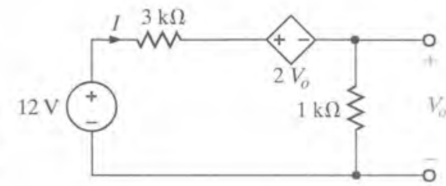


Figure 2.37 Circuit used in Example 2.25.

SOLUTION Applying KVL to this network yields

$$-12 + 3kI + 2V_o + 1kI = 0$$

where

$$V_o = 1kI$$

Hence, the KVL equation can be written as

$$-12 + 3kI + 2kI + 1kI = 0$$

or

$$I = 2 \text{ mA}$$

Therefore,

$$V_o = 1kI = 2 \text{ V}$$

LEARNING Example 2.26

An equivalent circuit for a FET common-source amplifier or BJT common-emitter amplifier can be modeled by the circuit shown in Fig. 2.38a. We wish to determine an expression for the gain of the amplifier, which is the ratio of the output voltage to the input voltage.

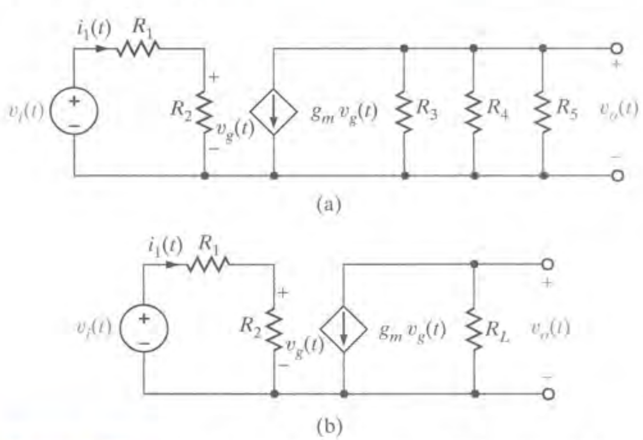


Figure 2.38 Example circuit containing a voltage-controlled current source.

SOLUTION Note that although this circuit, which contains a voltage-controlled current source, appears to be somewhat complicated, we are actually in a position now to solve it with techniques we have studied up to this point. The loop on the left, or input to the amplifier, is essentially detached from the output portion of the amplifier on the right. The voltage across R_2 is $v_g(t)$, which controls the dependent current source.

To simplify the analysis, let us replace the resistors R_3, R_4, and R_5 with R_L such that

$$\frac{1}{R_L} = \frac{1}{R_3} + \frac{1}{R_4} + \frac{1}{R_5}$$

Then the circuit reduces to that shown in Fig. 2.38b. Applying Kirchhoff's voltage law to the input portion of the amplifier yields

$$v_i(t) = i_1(t)(R_1 + R_2)$$

and

$$v_g(t) = i_1(t)R_2$$

Solving these equations for $v_g(t)$ yields

$$v_g(t) = \frac{R_2}{R_1 + R_2} v_i(t)$$

(continued)

From the output circuit, note that the voltage $v_o(t)$ is given by the expression

$$v_o(t) = -g_m v_g(t) R_L$$

Combining this equation with the preceding one yields

$$v_o(t) = \frac{-g_m R_L R_2}{R_1 + R_2} v_i(t)$$

Therefore, the amplifier gain, which is the ratio of the output voltage to the input voltage, is given by

$$\frac{v_o(t)}{v_i(t)} = -\frac{g_m R_L R_2}{R_1 + R_2}$$

Reasonable values for the circuit parameters in Fig. 2.38a are $R_1 = 100\ \Omega$, $R_2 = 1\ \text{k}\Omega$, $g_m = 0.04$ S, $R_3 = 50\ \text{k}\Omega$, and $R_4 = R_5 = 10\ \text{k}\Omega$. Hence, the gain of the amplifier under these conditions is

$$\frac{v_o(t)}{v_i(t)} = \frac{-(0.04)(4.545)(10^3)(1)(10^3)}{(1.1)(10^3)}$$

$$= -165.29$$

Thus, the magnitude of the gain is 165.29.

At this point it is perhaps helpful to point out again that when analyzing circuits with dependent sources, we first treat the dependent source as though it were an independent source when we write a Kirchhoff's current or voltage law equation. Once the equation is written, we then write the controlling equation that specifies the relationship of the dependent source to the unknown variable. For instance, the first equation in Example 2.24 treats the dependent source like an independent source. The second equation in the example specifies the relationship of the dependent source to the voltage, which is the unknown in the first equation.

LEARNING EXTENSIONS

E2.17 Find V_o in the circuit in Fig. E2.17.

ANSWER $V_o = 12$ V.

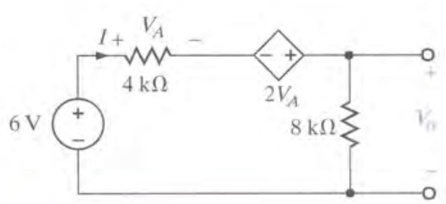


Figure E2.17

E2.18 Find V_o in the network in Fig. E2.18.

ANSWER $V_o = 8$ V.

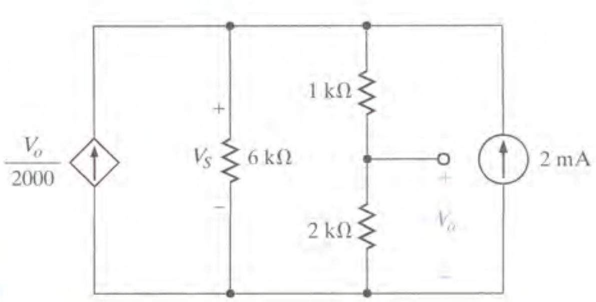


Figure E2.18

Learning by Application

Throughout this book we endeavor to present a wide variety of examples that demonstrate the usefulness of the material under discussion in a practical environment. To enhance our presentation of the practical aspects of circuit analysis and design, we have dedicated sections, such as this one, in most chapters for the specific purpose of presenting additional application-oriented examples.

LEARNING Example 2.27

An equivalent circuit for a transistor amplifier used in a portable tape player is shown in Fig. 2.39. The ideal independent source, V_S, and R_S represent the magnetic head playback circuitry. The dependent source, R_o and R_{in} model the transistor. Finally, the resistor R_L models the load, which in this case is another transistor circuit. Let us find the voltage gain of the network.

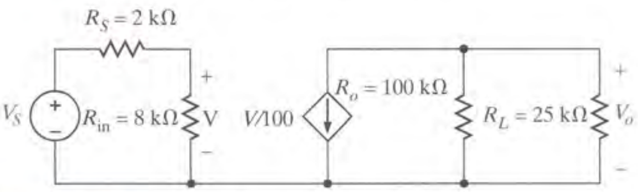


Figure 2.39 *Transistor amplifier circuit model.*

SOLUTION The gain of the transistor amplifier can be derived as follows. The output voltage can be expressed as

$$V_o = -\frac{V}{100}(R_o//R_L)$$

where $(R_o//R_L)$ represents R_o in parallel with R_L.

From the input voltage divider, we can express V as a function of V_S.

$$V = V_S\left(\frac{R_{in}}{R_{in} + R_S}\right)$$

Therefore,

$$V_o = -\frac{V_S}{100}\left(\frac{R_{in}}{R_{in} + R_S}\right)(R_o//R_L)$$

Given the component values in Fig. 2.39, the voltage gain is

$$A_V = \frac{V_o}{V_S} = -160$$

LEARNING Example 2.28

A Wheatstone Bridge circuit is an accurate device for measuring resistance. The circuit, shown in Fig. 2.40, is used to measure the unknown resistor R_x. The center leg of the circuit contains a galvanometer, which is a very sensitive device that can be used to measure current in the microamp range. When the unknown resistor is connected to the bridge, R_3 is adjusted until the current in the galvanometer is zero, at which point the bridge is balanced. In this balanced condition

$$\frac{R_1}{R_3} = \frac{R_2}{R_x}$$

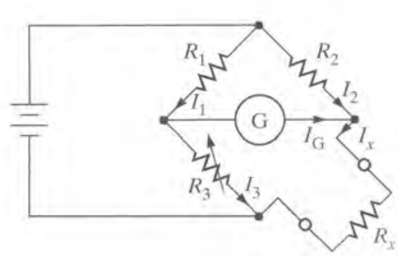


Figure 2.40 *The Wheatstone bridge circuit.*

so that

$$R_x = \left(\frac{R_2}{R_1}\right)R_3$$

Engineers also use this bridge circuit to measure strain in solid material. For example, a system used to determine the weight of a truck is shown in Fig. 2.41a. The platform is supported by cylinders on which strain gauges are mounted. The strain gauges, which measure strain when the cylinder deflects under load, are connected to a Wheatstone bridge as shown in Fig. 2.41b. The strain gauge has a resistance of 120 Ω under no-load conditions and changes value under load. The variable resistor in the bridge is a calibrated precision device.

Weight is determined in the following manner. The ΔR_3 required to balance the bridge represents the Δ strain, which when multiplied by the modulus of elasticity yields the Δ stress. The Δ stress multiplied by the cross-sectional area of the cylinder produces the Δ load, which is used to determine weight.

Let us determine the value of R_3 under no load when the bridge is balanced and its value when the resistance of the strain gauge changes to 120.24 Ω under load.

(continued)

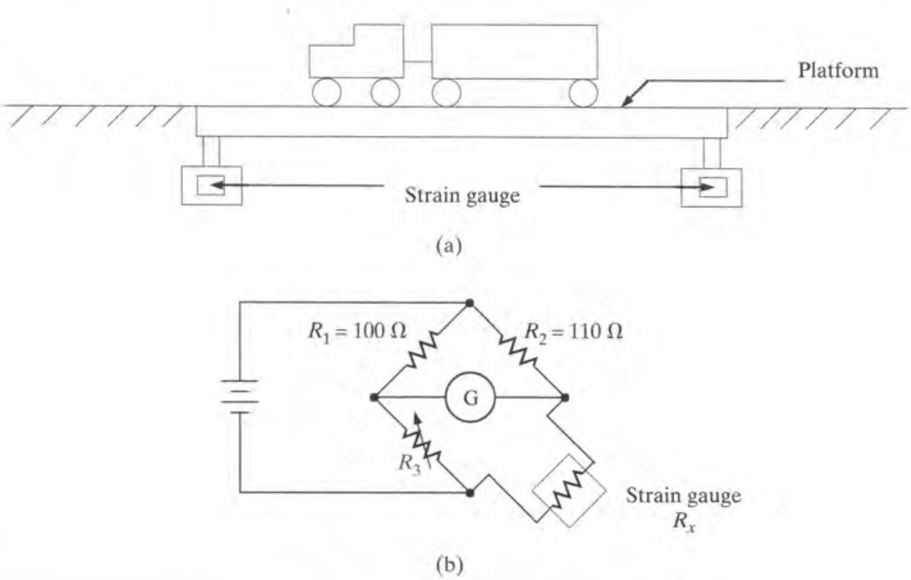


Figure 2.41
Diagrams used in Example 2.28.

SOLUTION Using the balance equation for the bridge, the value of R_3 at no load is

$$\begin{aligned} R_3 &= \left(\frac{R_1}{R_2}\right)R_x \\ &= \left(\frac{100}{110}\right)(120) \\ &= 109.0909\ \Omega \end{aligned}$$

Under load, the value of R_3 is

$$\begin{aligned} R_3 &= \left(\frac{100}{110}\right)(120.24) \\ &= 109.3091\ \Omega \end{aligned}$$

Therefore, the ΔR_3 is

$$\begin{aligned} \Delta R_3 &= 109.3091 - 109.0909 \\ &= 0.2182\ \Omega \end{aligned}$$

Learning by Design

Most of this text is concerned with circuit analysis; that is, given a circuit in which all the components are specified, analysis involves finding such things as the voltage across some element or the current through another. Furthermore, the solution of an analysis problem is generally unique. In contrast, design involves determining the circuit configuration that will meet certain specifications. In addition, the solution is generally not unique in that there may be many ways to satisfy the circuit/performance specifications. It is also possible that there is no solution that will meet the design criteria.

In addition to meeting certain technical specifications, designs normally must also meet other criteria, such as economic, environmental, and safety constraints. For example, if a circuit design that meets the technical specifications is either too expensive or unsafe, it is not viable regardless of its technical merit.

At this point, the number of elements that we can employ in circuit design is limited primarily to the linear resistor and the active elements we have presented. However, as we progress through the text we will introduce a number of other elements (for example, the op-amp, capacitor, and inductor), which will significantly enhance our design capability.

We begin our discussion of circuit design by considering a couple of simple examples that demonstrate the selection of specific components to meet certain circuit specifications.

LEARNING Example 2.29

A circuit board within a stereo amplifier requires the use of three different voltages at the circuit nodes. The voltages needed are 4 V, 16 V, and 24 V. Criteria such as weight, cost, and size dictate that these voltages be supplied with a single voltage source. Furthermore, heat constraints dictate that the supply must dissipate less than 5 W. Assuming no loading effect from the balance of the network, design a voltage string (multiple-resistor voltage divider) that will satisfy the requirements.

SOLUTION A 24-V source can directly provide one of the voltages, and the voltage divider in Fig. 2.42 can be used to generate the other two. The power requirement limits our choices to current less than 208.33 mA. The exact value we choose is somewhat arbitrary and, therefore, we select $I = 200$ mA. The total resistance is then

$$R_1 + R_2 + R_3 = \frac{24}{0.2} = 120\ \Omega$$

Using voltage division, we find that

$$\frac{R_3}{R_1 + R_2 + R_3} = \frac{R_3}{120} = \frac{4}{24}$$

or

$$R_3 = 20\ \Omega$$

Similarly,

$$\frac{R_2}{120} = \frac{16 - 4}{24}$$

or

$$R_2 = 60\ \Omega$$

Finally, since the total resistance is $120\ \Omega$, $R_1 = 40\ \Omega$.

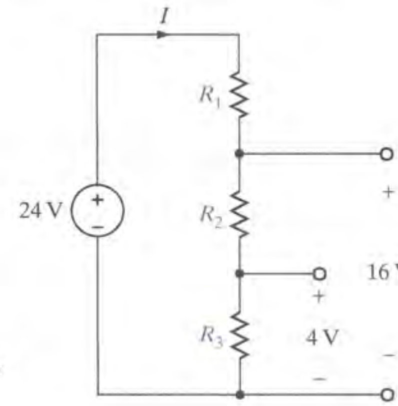


Figure 2.42 Voltage string for generating intermediate voltages.

LEARNING Example 2.30

The network in Fig. 2.43 is an equivalent circuit for a transistor amplifier used in a stereo preamplifier. The input circuitry, consisting of a 2-mV source in series with a 500-Ω resistor, models the output of a compact disk player. The dependent source, R_{in}, and R_o model the transistor, which amplifies the signal and then sends it to the power amplifier. The 10-kΩ load resistor models the input to the power amplifier that actually drives the speakers. We must design a transistor amplifier as shown in Fig. 2.43 that will provide an overall gain of −200. In practice we do not actually vary the device parameters to achieve the desired gain; rather we select a transistor from the manufacturer's data books that will satisfy the required specification. The model parameters for three different transistors are listed as follows:

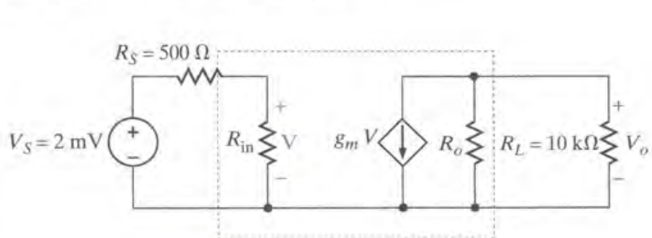


Figure 2.43 Transistor amplifier circuit model.

Manufacturer's transistor parameter values

Part Number	R_{in} (kΩ)	R_o (kΩ)	g_m (mA/V)
1	1.0	50	50
2	2.0	75	30
3	8.0	80	20

Design the amplifier by choosing the transistor that produces the most accurate gain. What is the percent error of your choice?

SOLUTION The output voltage can be written

$$V_o = -g_m V(R_o//R_L)$$

Using voltage division at the input to find V,

$$V = V_S\left(\frac{R_{in}}{R_{in} + R_S}\right)$$

Combining these two expressions, we can solve for the gain:

$$A_V = \frac{V_o}{V_S} = -g_m\left(\frac{R_{in}}{R_{in} + R_S}\right)(R_o//R_L)$$

Using the parameter values for the three transistors, we find that Obviously, the best alternative is transistor number 2, which has a gain error of

$$\text{Percent error} = \left(\frac{211.8 - 200}{200}\right) \times 100\% = 5.9\%$$

LEARNING Check

Summary

- ***Ohm's law*** $V = IR$
- ***The passive sign convention with Ohm's law*** The current enters the resistor terminal with the positive voltage reference.
- ***Kirchhoff's current law (KCL)*** The algebraic sum of the currents leaving (entering) a node is zero.
- ***Kirchhoff's voltage law (KVL)*** The algebraic sum of the voltages around any closed path is zero.
- ***Solving a single-loop circuit*** Determine the loop current by applying KVL and Ohm's law.
- ***Solving a single-node-pair circuit*** Determine the voltage between the pair of nodes by applying KCL and Ohm's law.
- ***The voltage-division rule*** The voltage is divided between two series resistors in direct proportion to their resistance.
- ***The current-division rule*** The current is divided between two parallel resistors in reverse proportion to their resistance.
- ***The equivalent resistance of a network of resistors*** Combine resistors in series by adding their resistances. Combine resistors in parallel by adding their conductances. The wye-to-delta and delta-to-wye transformations are also an aid in reducing the complexity of a network.
- ***Short circuit*** Zero resistance, zero voltage; the current in the short is determined by the rest of the circuit.
- ***Open circuit*** Zero conductance, zero current; the voltage across the open terminals is determined by the rest of the circuit.

Problems

For solutions and additional help on problems marked with ► go to www.wiley.com/college/irwin

SECTION 2.1

2.1 ► **Find the current I and the power supplied by the source in the network in Fig. P2.1.**

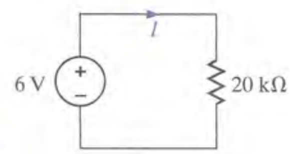


Figure P2.1

2.2 In the circuit in Fig. P2.2, find the voltage across the current source and the power absorbed by the resistor.

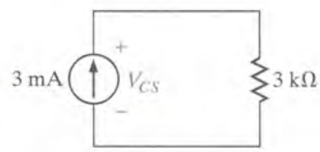


Figure P2.2

2.3 If the 10-kΩ resistor in the network in Fig. P2.3 absorbs 2.5 mW, find V_S.

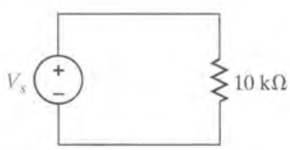


Figure P2.3

2.4 In the network in Fig. P2.4, the power absorbed by R_x is 5 mW. Find R_x.

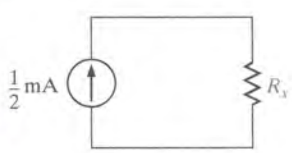


Figure P2.4

2.5 ► **In the network in Fig. P2.5, the power absorbed by R_x is 20 mW. Find R_x.**

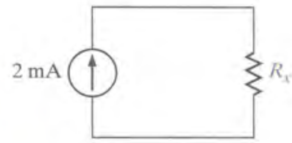


Figure P2.5

2.6 A model for a standard two D-cell flashlight is shown in Fig. P2.6. Find the power dissipated in the lamp.

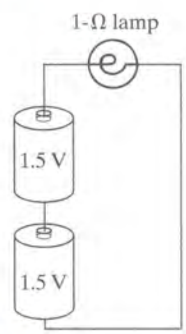


Figure P2.6

2.7 An automobile uses two halogen headlights connected as shown in Fig. P2.7. Determine the power supplied by the battery if each headlight draws 3 A of current.

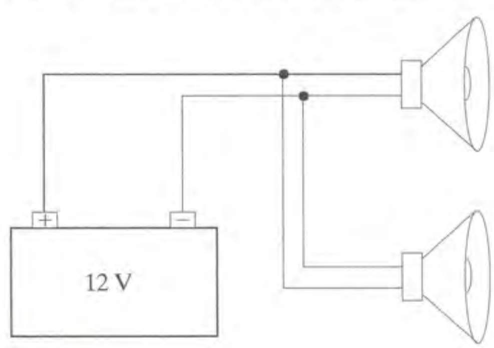


Figure P2.7

2.8 Many years ago a string of Christmas tree lights was manufactured in the form shown in Fig. P2.8a. Today the lights are manufactured as shown in Fig. P2.8b. Is there a good reason for this change?

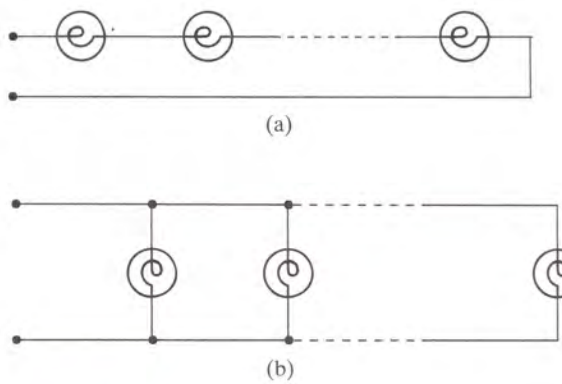


Figure P2.8

SECTION 2.2

2.9 Find I_1 in the network in Fig. P2.9.

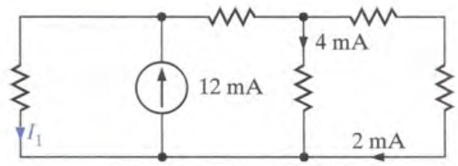


Figure P2.9

2.10 Find I_1 and I_2 in the circuit in Fig. P2.10.

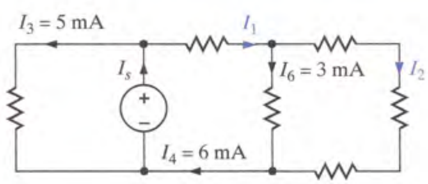


Figure P2.10

2.11 Find I_x and I_y in the network in Fig. P2.11.

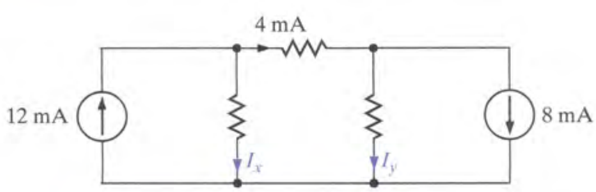


Figure P2.11

2.12 Find I_x, I_y, and I_z in the circuit in Fig. P2.12.

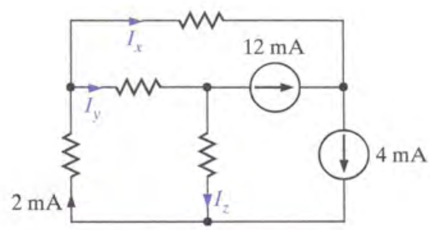


Figure P2.12

2.13 Find I_x in the circuit in Fig. P2.13.

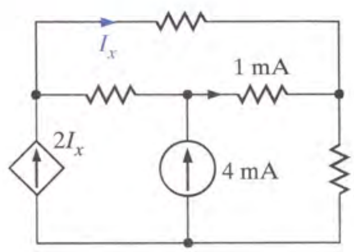


Figure P2.13

2.14 Find I_x, I_y, and I_z in the network in Fig. P2.14.

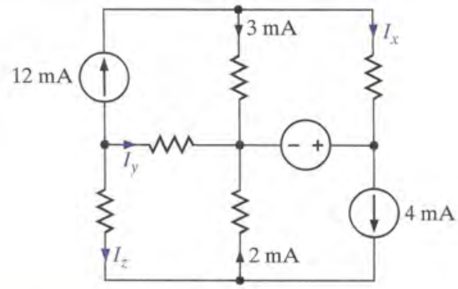


Figure P2.14

2.15 Find I_x in the circuit in Fig. P2.15.

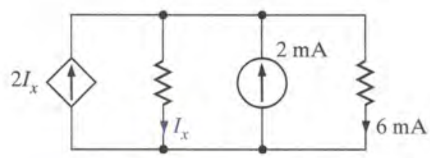


Figure P2.15

2.16 Find I_x in the network in Fig. P2.16.

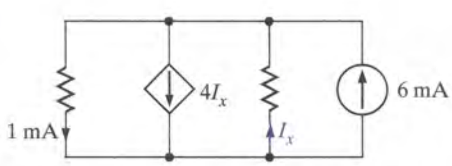


Figure P2.16

2.17 Find V_x in the circuit in Fig. P2.17.

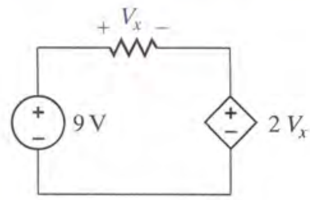


Figure P2.17

2.18 Find V_{bd} in the circuit in Fig. P2.18.

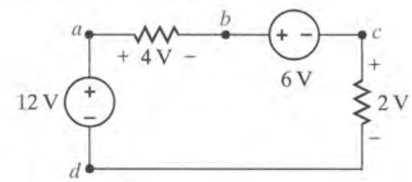


Figure P2.18

2.19 Find V_{ad} in the network in Fig. P2.19.

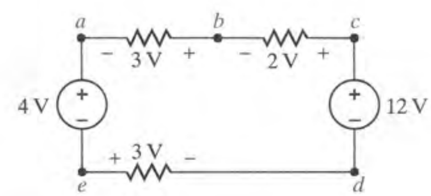


Figure P2.19

2.20 Find V_{af} and V_{ec} in the circuit in Fig. P2.20.

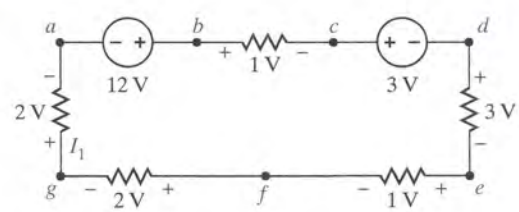


Figure P2.20

2.21 Find V_{ac} in the circuit in Fig. P2.21.
►

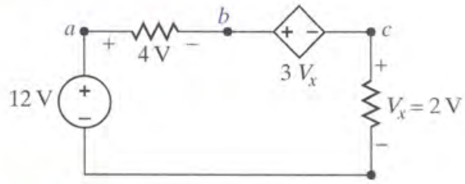


Figure P2.21

2.22 Find V_{ad} and V_{ce} in the circuit in Fig. P2.22.

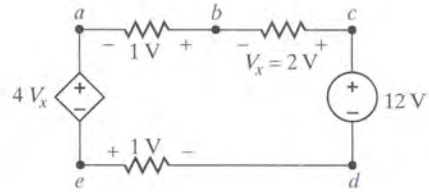


Figure P2.22

SECTION 2.3

2.23 Find V_{ab} in the network in Fig. P2.23.

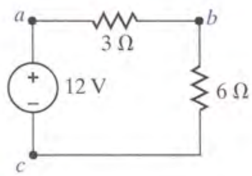


Figure P2.23

2.24 Find V_{bd} in the network in Fig. P2.24.

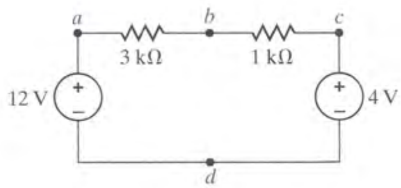


Figure P2.24

2.25 Find V_x in the circuit in Fig. P2.25.
►

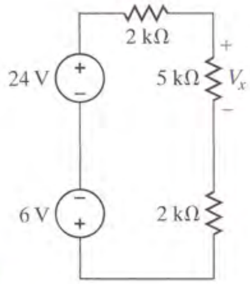


Figure P2.25

2.26 Find V_x in the circuit in Fig. P2.26.

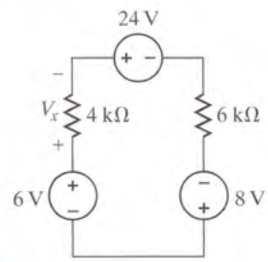


Figure P2.26

2.27 Find V_x in the network in Fig. P2.27.

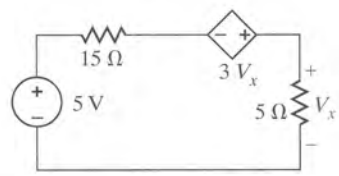


Figure P2.27

2.28 Find V_1 in the network in Fig. P2.28.

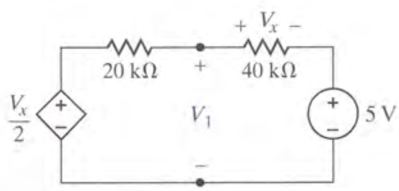


Figure P2.28

2.29 **Find the power absorbed by the 30-kΩ resistor in the circuit in Fig. P2.29.**

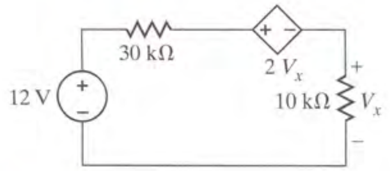


Figure P2.29

SECTION 2.4

2.30 Find I_o in the network in Fig. P2.30.

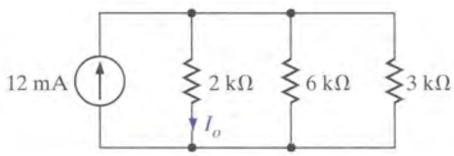


Figure P2.30

2.31 Find I_o in the circuit in Fig. P2.31.

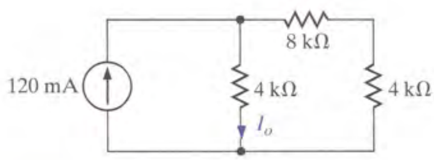


Figure P2.31

2.32 Find I_o in the network in Fig. P2.32.

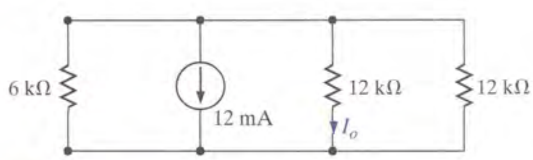


Figure P2.32

2.33 **Find V_o in the circuit in Fig. P2.33.**

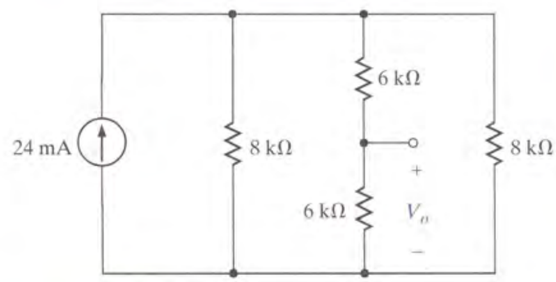


Figure P2.33

2.34 Find I_o in the network in Fig. P2.34.

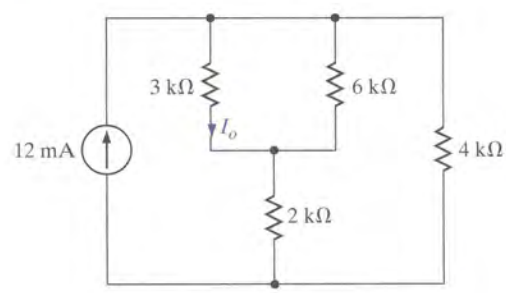


Figure P2.34

2.35 Determine I_L in the circuit in Fig. P2.35.

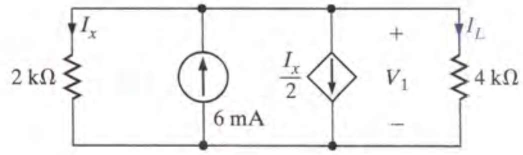


Figure P2.35

2.36 Determine I_L in the circuit in Fig. P2.36.

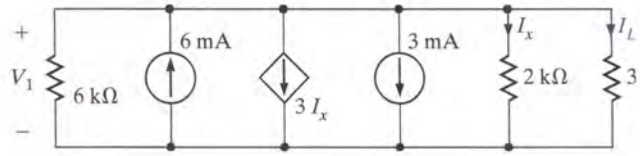


Figure P2.36

SECTION 2.5

2.37 Find R_{AB} in the circuit in Fig. P2.37.

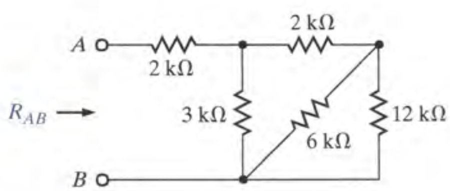


Figure P2.37

2.38 Find R_{AB} in the circuit in Fig. P2.38.

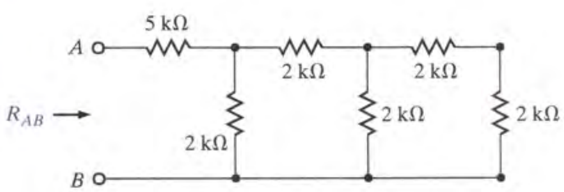


Figure P2.38

2.39 Find R_{AB} in the network in Fig. P2.39.

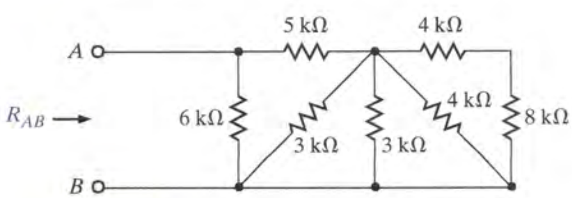


Figure P2.39

2.40 Find R_{AB} in the circuit in Fig. P2.40.

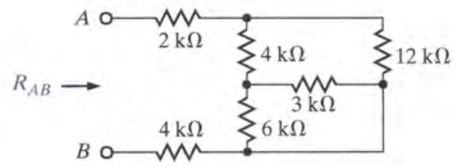


Figure P2.40

2.41 Find R_{AB} in the network in Fig. P2.41.

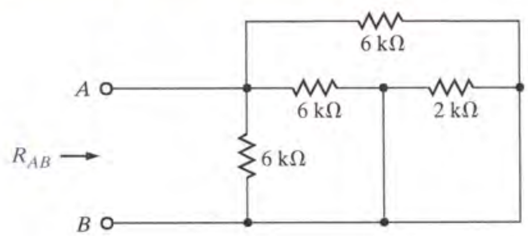


Figure P2.41

2.42 Find R_{AB} in the circuit in Fig. P2.42.

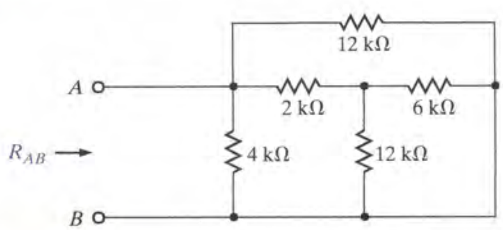


Figure P2.42

2.43 Find R_{AB} in the circuit in Fig. P2.43.

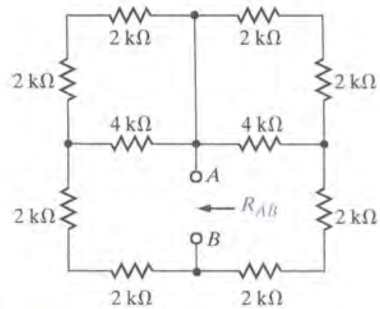


Figure P2.43

2.44 Find the range of resistance for the following resistors.

(a) 1 kΩ with a tolerance of 5%

(b) 470 Ω with a tolerance of 2%

(c) 22 kΩ with a tolerance of 10%

2.45 **Given the network in Fig. P2.45, find the possible range of values for the current and power dissipated by the following resistors.**

(a) 390 Ω with a tolerance of 1%

(b) 560 Ω with a tolerance of 2%

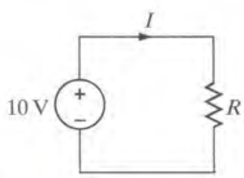


Figure P2.45

2.46 Given the circuit in Fig. P2.46.

(a) find the required value of R.

(b) use Table 2.1 to select a standard 10% tolerance resistor for R.

(c) calculate the actual value of I.

(d) determine the percent error between the actual value of I and that shown in the circuit.

(e) determine the power rating for the resistor R.

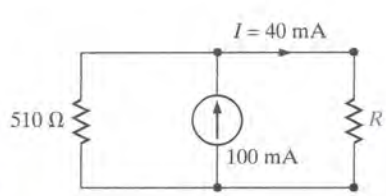


Figure P2.46

2.47 The resistors R_1 and R_2 shown in the circuit in Fig. P2.47 are 1 Ω with a tolerance of 5% and 2 Ω with a tolerance of 10%, respectively.

(a) What is the nominal value of the equivalent resistance?

(b) Determine the positive and negative tolerance for the equivalent resistance.

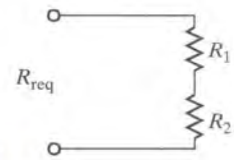


Figure P2.47

2.48 Find I_1 and V_o in the circuit in Fig. P2.48.

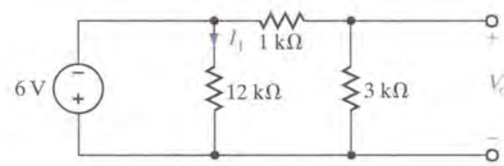


Figure P2.48

2.49 **Find I_1 and V_o in the circuit in Fig. P2.49.**

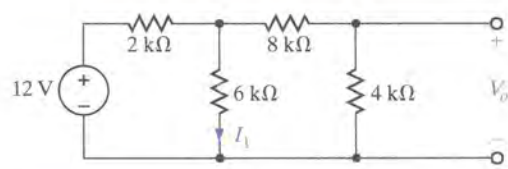


Figure P2.49

2.50 Find I_1 in the circuit in the Fig. P2.50.

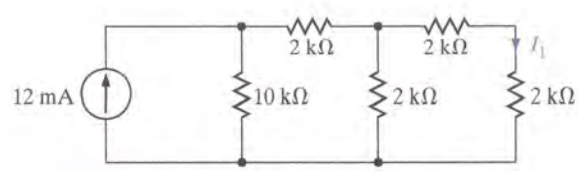


Figure P2.50

2.51 Determine V_o in the network in Fig. P2.51.

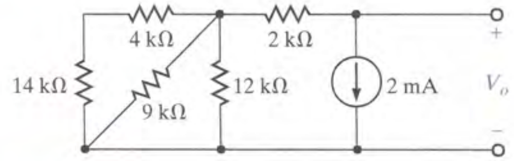


Figure P2.51

2.52 Find V_o in the network in Fig. P2.52.

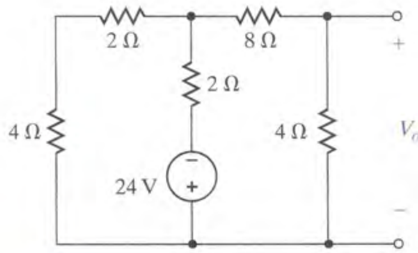


Figure P2.52

2.53 Find I_o in the circuit in Fig. P2.53.

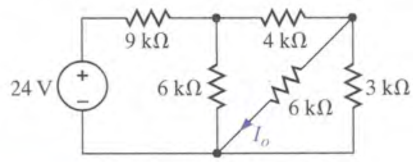


Figure P2.53

2.54 Find V_o in the circuit in Fig. P2.54.

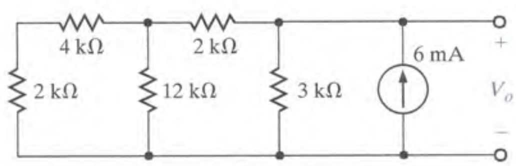


Figure P2.54

2.55 Find I_o in the network in Fig. P2.55.

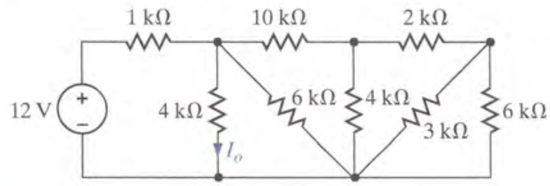


Figure P2.55

2.56 Find V_o in the network in Fig. P2.56.

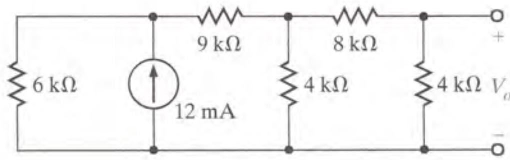


Figure P2.56

2.57 Find I_o in the circuit in Fig. P2.57.

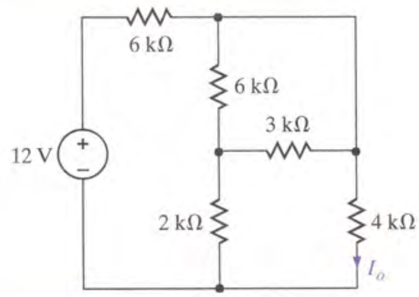


Figure P2.57

2.58 If V_o = 4 V in the network in Fig. P2.58, find V_S.

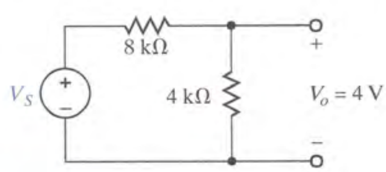


Figure P2.58

2.59 If the power absorbed by the 4-kΩ resistor in the network in Fig. P2.59 is 36 mW, find I_o.

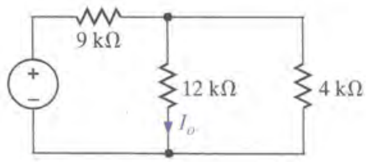


Figure P2.59

2.60 If the power absorbed by the 4-kΩ resistor in the circuit in Fig. P2.60 is 36 mW, find V_S.

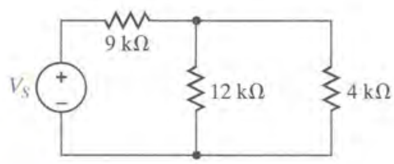


Figure P2.60

2.61 ► In the network in Fig. P2.61, the power absorbed by the 4-Ω resistor is 100 W. Find V_S.

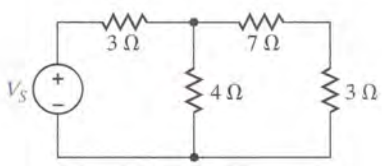


Figure P2.61

2.62 In the network in Fig. P2.62, V_o = 6 V. Find I_S.

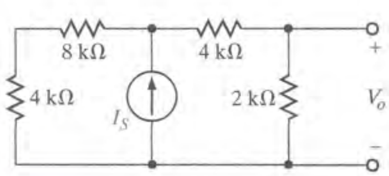


Figure P2.62

2.63 In the circuit in Fig. P2.63, I = 4 mA. Find V_S.

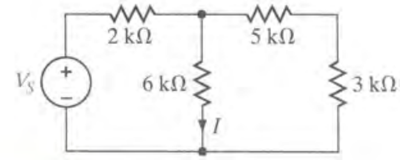


Figure P2.63

2.64 In the circuit in Fig. P2.64, I_o = 2 mA. Find I_S.

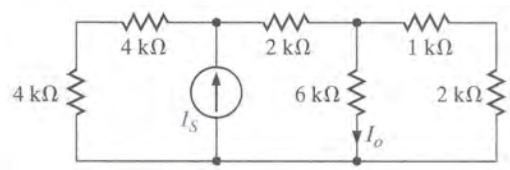


Figure P2.64

2.65 ► In the network in Fig. P2.65, V_1 = 12 V. Find V_S.

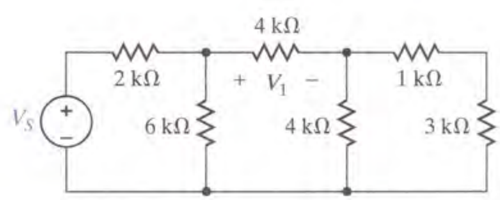


Figure P2.65

2.66 In the circuit in Fig. P2.66, V_o = 2 V. Find I_S.

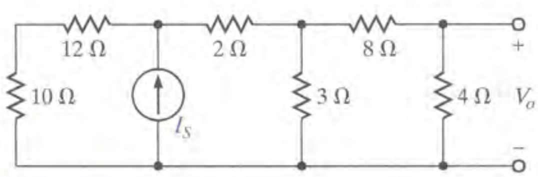


Figure P2.66

2.67 In the network in Fig. P2.67, $V_o = 6$ V. Find I_S.

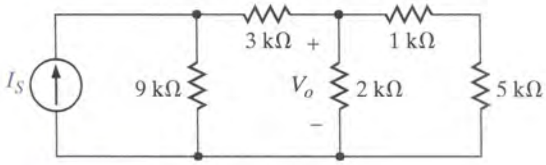


Figure P2.67

2.68 In the circuit in Fig. P2.68, $I_o = 2$ A. Find I_S.

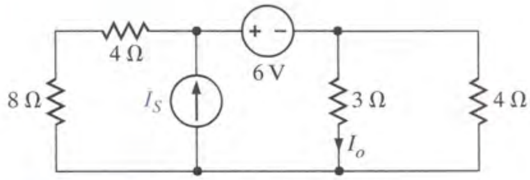


Figure P2.68

2.69 If $I_o = 4$ mA in the circuit in Fig. P2.69, find I_S.
►

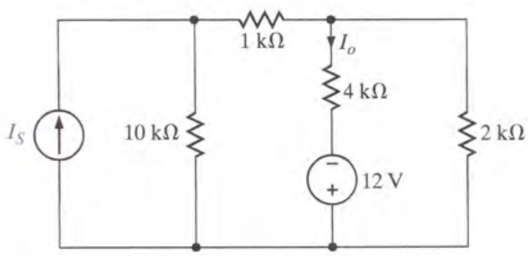


Figure P2.69

2.70 Find I_o in the circuit in Fig. P2.70.

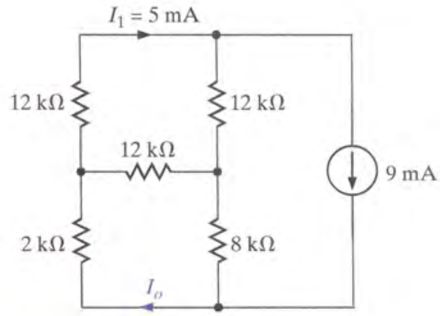


Figure P2.70

2.71 Find V_o in the circuit in Fig. P2.71.

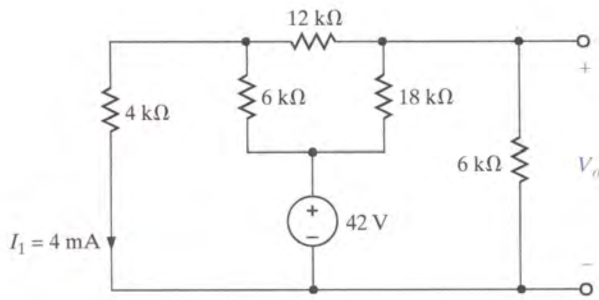


Figure P2.71

2.72 Find the power absorbed by the network in Fig. P2.72.

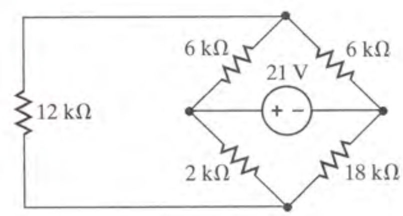


Figure P2.72

2.73 Find I_o in the circuit in Fig. P2.73.
►

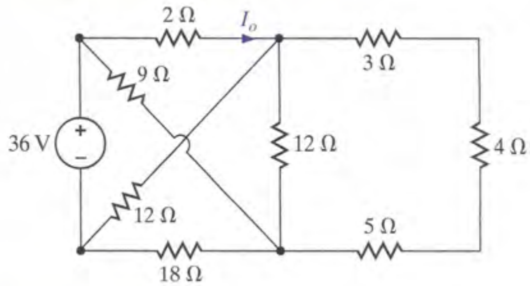


Figure P2.73

2.74 Find I_o in the circuit in Fig. P2.74.

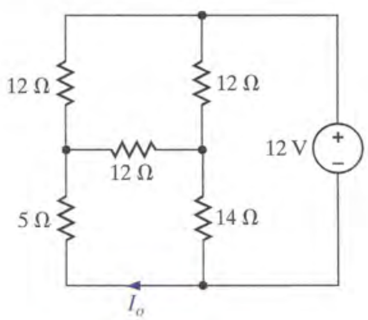


Figure P2.74

2.75 Find I_o in the circuit in Fig. P2.75.

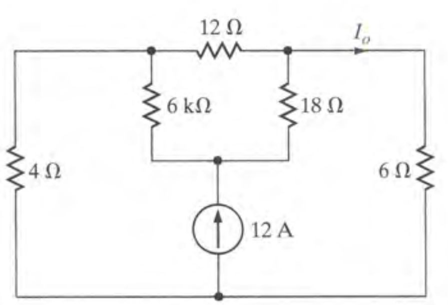


Figure P2.75

2.76 Find I_o in the circuit in Fig. P2.76.

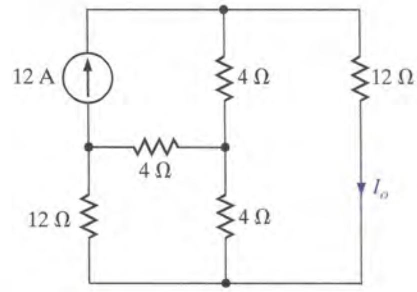


Figure P2.76

2.77 Find I_o in the circuit in Fig. P2.77.
►

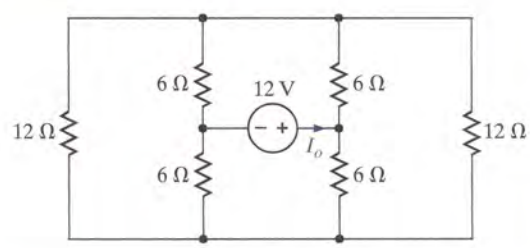


Figure P2.77

2.78 Find V_o in the circuit in Fig. P2.78.

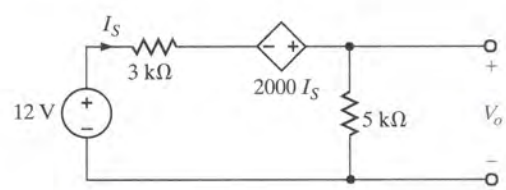


Figure P2.78

2.79 Find V_o in the network in Fig. P2.79.

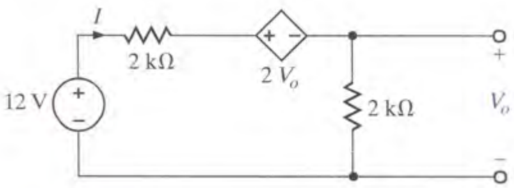


Figure P2.79

2.80 Find V_o in the network in Fig. P2.80.

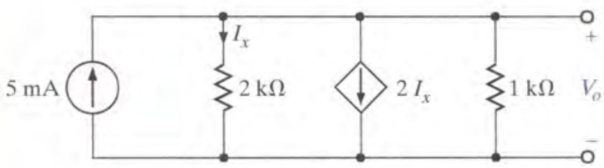


Figure P2.80

2.81 ► **Find I_o in the network in Fig. P2.81.**

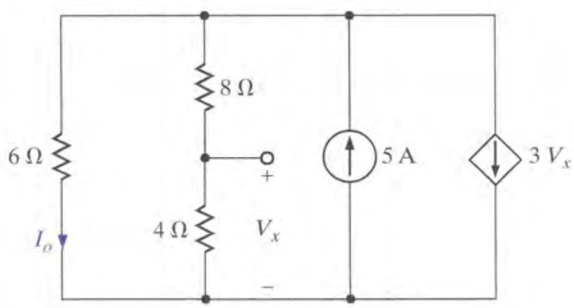


Figure P2.81

2.82 Find V_o in the circuit in Fig. P2.82.

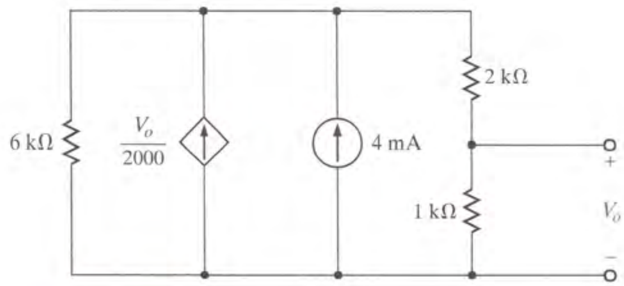


Figure P2.82

2.83 A single-stage transistor amplifier is modeled as shown in Fig. P2.83. Find the current in the load R_L.

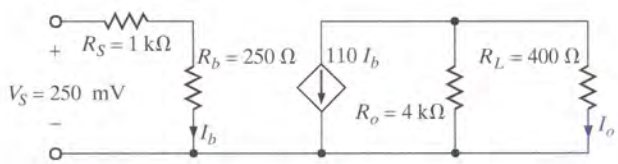


Figure P2.83

2.84 A typical transistor amplifier is shown in Fig. P2.84. Find the amplifier gain G (i.e., the ratio of the output voltage to the input voltage).

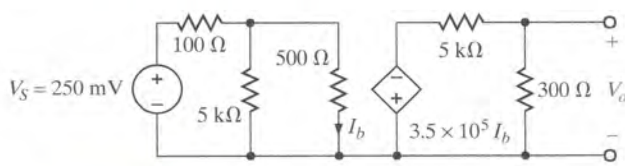


Figure P2.84

2.85 ► **For the network in Fig. P2.85, choose the values of R_{in} and R_o such that V_o is maximized. What is the resulting ratio, V_o/V_S?**

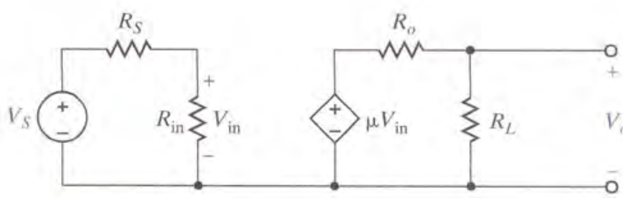


Figure P2.85

2.86 In many amplifier applications we are concerned not only with voltage gain, but also with power gain.

Power gain = A_p = (power delivered to the load)/(power delivered by the input)

Find the power gain for the circuit in Fig. P2.86, where $R_L = 50$ kΩ.

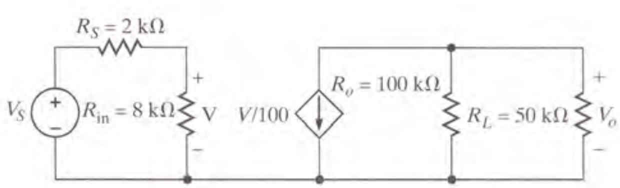


Figure P2.86

2.87 Find the power absorbed by the 10-kΩ resistor in the circuit in Fig. P2.87.

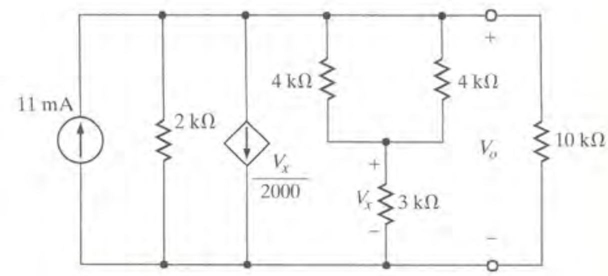


Figure P2.87

2.88 Find the power absorbed by the 12-kΩ resistor in the circuit in Fig. P2.88.

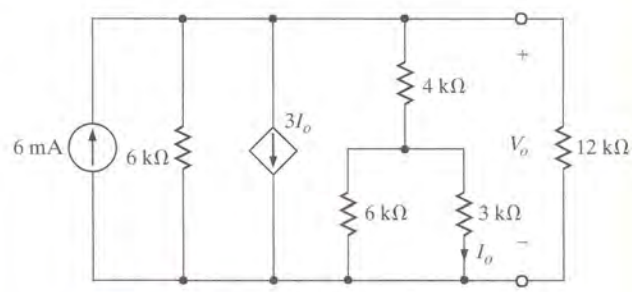


Figure P2.88

Typical Problems Found on the FE Exam

2FE-1 ► **Find the power generated by the source in the network in Fig. 2PFE-1.**

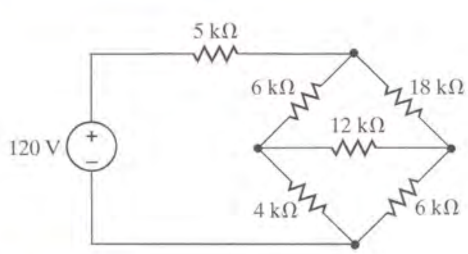


Fig. 2PFE-1

2FE-2 Find the equivalent resistance of the circuit in Fig. 2PFE-2 at the terminals *A*-*B*.

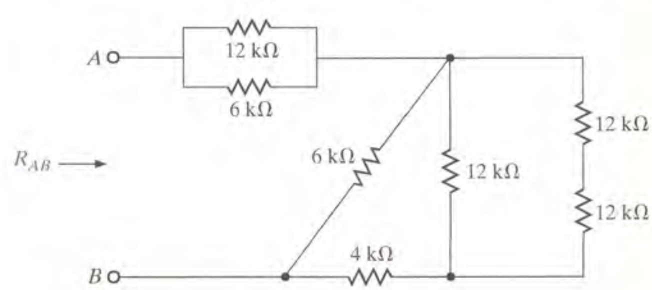


Fig. 2PFE-2

2FE-3 **Find the voltage V_o in the network in Fig. 2PFE-3.** ►

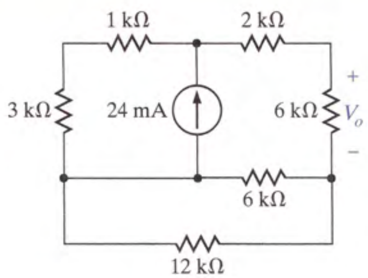


Fig. 2PFE-3

2FE-4 Find the current I_o in the circuit in Fig. 2PFE-4.

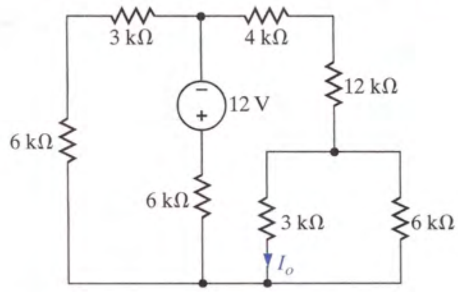


Fig. 2PFE-4

Nodal and Loop Analysis Techniques

3

In Chapter 2 we analyzed the simplest possible circuits, those containing only a single-node pair or a single loop. We found that these circuits can be completely analyzed via a single algebraic equation. In the case of the single-node-pair circuit (i.e., one containing two nodes, one of which is a reference node), once the node voltage is known, we can calculate all the currents. In a single-loop circuit, once the loop current is known, we can calculate all the voltages.

In this chapter we extend our capabilities in a systematic manner so that we can calculate all currents and voltages in circuits that contain multiple nodes and loops. Our analyses are based primarily on two laws with which we are already familiar: Kirchhoff's current law (KCL) and Kirchhoff's voltage law (KVL). In a nodal analysis we employ KCL to determine the node voltages, and in a loop analysis we use KVL to determine the loop currents.

We present and discuss a very important commercially available circuit known as the operational amplifier, or op-amp. Op-amps are used in literally thousands of applications, including such things as compact disk (CD) players, random access memories (RAMs), analog-to-digital (A/D) and digital-to-analog (D/A) converters, headphone amplifiers, and electronic instrumentation of all types. Finally, we discuss the terminal characteristics of this circuit and demonstrate its use in practical applications as well as circuit design.

LEARNING Goals

3.1 Nodal Analysis

In a nodal analysis the variables in the circuit are selected to be the node voltages. The node voltages are defined with respect to a common point in the circuit. One node is selected as the reference node, and all other node voltages are defined with respect to that node. Quite often this node is the one to which the largest number of branches are connected. It is commonly called *ground* because it is said to be at ground-zero potential, and it sometimes represents the chassis or ground line in a practical circuit.

We will select our variables as being positive with respect to the reference node. If one or more of the node voltages are actually negative with respect to the reference node, the analysis will indicate it.

In order to understand the value of knowing all the node voltages in a network, we consider once again the network in Fig. 2.30, which is redrawn in Fig. 3.1. The voltages, V_S, V_a, V_b, and V_c, are all measured with respect to the bottom node, which is selected as the reference and labeled with the ground symbol ⏚. Therefore, the voltage at node 1 is $V_S = 12$ V with respect to the reference node 5; the voltage at node 2 is $V_a = 3$ V with respect to the reference node 5, and so on. Now note carefully that once these node voltages are known, we can immediately calculate any branch current or the power supplied or absorbed by any element, since we know the voltage across every element in the network. For example, the voltage V_1 across the left-most 9-kΩ resistor is the difference in potential between the two ends of the resistor; that is,

$$\begin{aligned} V_1 &= V_S - V_a \\ &= 12 - 3 \\ &= 9 \text{ V} \end{aligned}$$

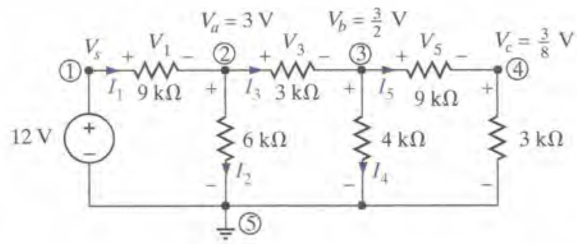


Figure 3.1
Circuit with known node voltages.

This equation is really nothing more than an application of KVL around the left-most loop; that is,

$$-V_S + V_1 + V_a = 0$$

In a similar manner, we find that

$$V_3 = V_a - V_b$$

and

$$V_5 = V_b - V_c$$

Then the currents in the resistors are

$$\begin{aligned} I_1 &= \frac{V_1}{9\text{k}} = \frac{V_S - V_a}{9\text{k}} \\ I_3 &= \frac{V_3}{3\text{k}} = \frac{V_a - V_b}{3\text{k}} \\ I_5 &= \frac{V_5}{9\text{k}} = \frac{V_b - V_c}{9\text{k}} \end{aligned}$$

In addition,

$$I_2 = \frac{V_a - 0}{6\text{k}}$$
$$I_4 = \frac{V_b - 0}{4\text{k}}$$

since the reference node 5 is at zero potential.

Thus, as a general rule, if we know the node voltages in a circuit, we can calculate the current through any resistive element using Ohm's law; that is,

$$i = \frac{v_m - v_N}{R} \quad \text{3.1}$$

as illustrated in Fig. 3.2.

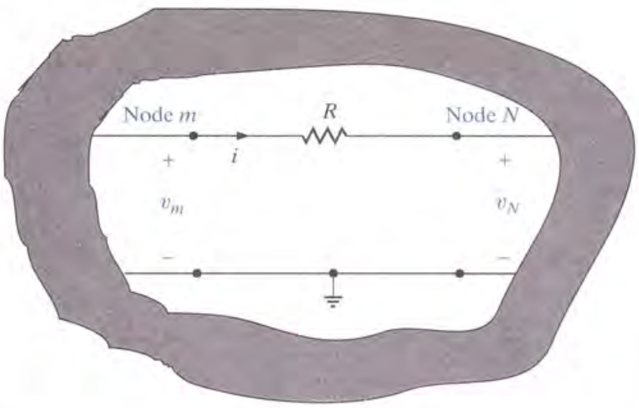


Figure 3.2
Circuit used to illustrate Ohm's law in a multiple-node network.

In Example 2.3 we illustrated that the number of linearly independent KCL equations for an N-node network was $N - 1$. Furthermore, we found that in a two-node circuit, in which one node was the reference node, only one equation was required to solve for the unknown node voltage. What is illustrated in this simple case is true in general; that is, in an N-node circuit one linearly independent KCL equation is written for each of the $N - 1$ nonreference nodes, and this set of $N - 1$ linearly independent simultaneous equations, when solved, will yield the $N - 1$ unknown node voltages.

It is instructive to treat nodal analysis by examining several different types of circuits and illustrating the salient features of each. We begin with the simplest case. However, as a prelude to our discussion of the details of nodal analysis, experience indicates that it is worthwhile to digress for a moment to ensure that the concept of node voltage is clearly understood.

At the outset it is important to specify a reference. For example, to state that the voltage at node A is 12 V means nothing unless we provide the reference point; that is, the voltage at node A is 12 V with respect to what. The circuit in Fig. 3.3 illustrates a portion of a network containing three nodes, one of which is the reference node.

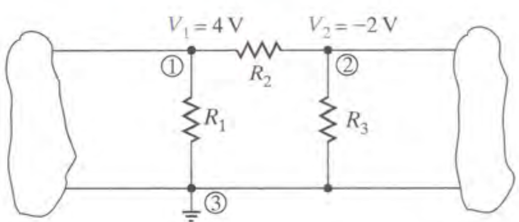


Figure 3.3
An illustration of node voltages.

The voltage $V_1 = 4$ V is the voltage at node 1 with respect to the reference node 3. Similarly, the voltage $V_2 = -2$ V is the voltage at node 2 with respect to node 3. In addition, however, the voltage at node 1 with respect to node 2 is +6 V and the voltage at node 2 with respect to node 1 is −6 V. Furthermore, since the current will flow from the node of higher potential to the node of lower potential, the current in R_1 is from top to bottom, the current in R_2 is from left to right, and the current in R_3 is from bottom to top.

These concepts have important ramifications in our daily lives. If a man were hanging in midair with one hand on one line and one hand on another and the dc line voltage of each line was exactly the same, the voltage across his heart would be zero and he would be safe. If, however, he let go of one line and let his feet touch the ground, the dc line voltage would then exist from his hand to his foot with his heart in the middle. He would probably be dead the instant his foot hit the ground.

In the town where I live, a young man tried to retrieve his parakeet that had escaped its cage and was outside sitting on a power line. He stood on a metal ladder and with a metal pole reached for the parakeet; when the metal pole touched the power line, the man was killed instantly. Electric power is vital to our standard of living, but it is also very dangerous. The material in this book *does not* qualify you to handle it safely. Therefore, always be extremely careful around electric circuits.

Now as we begin our discussion of nodal analysis, our approach will be to begin with simple cases and proceed in a systematic manner to those that are more challenging. Numerous examples will be the vehicle used to demonstrate each facet of this approach. Finally, at the end of this section, we will outline a strategy for attacking any circuit using nodal analysis.

CIRCUITS CONTAINING ONLY INDEPENDENT CURRENT SOURCES Consider the network shown in Fig. 3.4. There are three nodes, and the bottom node is selected as the reference node. The branch currents are assumed to flow in the directions indicated in the figures. If one or more of the branch currents are actually flowing in a direction opposite to that assumed, the analysis will simply produce a branch current that is negative.

Applying KCL at node 1 yields

$$-i_A + i_1 + i_2 = 0$$

LEARNING Hint

Employing the passive sign convention.

Using Ohm's law ($i = Gv$) and noting that the reference node is at zero potential, we obtain

$$-i_A + G_1(v_1 - 0) + G_2(v_1 - v_2) = 0$$

or

$$(G_1 + G_2)v_1 - G_2v_2 = i_A$$

KCL at node 2 yields

$$-i_2 + i_B + i_3 = 0$$

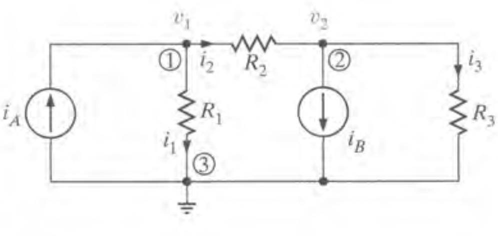


Figure 3.4
A three-node circuit.

or

$$-G_2(v_1 - v_2) + i_B + G_3(v_2 - 0) = 0$$

which can be expressed as

$$-G_2 v_1 + (G_2 + G_3)v_2 = -i_B$$

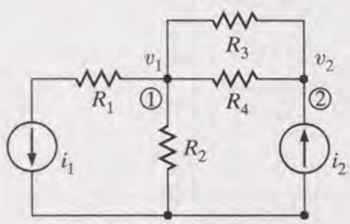
Therefore, the two equations for the two unknown node voltages v_1 and v_2 are

$$(G_1 + G_2)v_1 - G_2 v_2 = i_A$$
$$-G_2 v_1 + (G_2 + G_3)v_2 = -i_B \qquad \textbf{3.2}$$

Note that the analysis has produced two simultaneous equations in the unknowns v_1 and v_2. They can be solved using any convenient technique, and modern calculators and personal computers are very efficient tools for their application.

LEARNING by Doing

D 3.1 For the following network, write the KCL equations for nodes 1 and 2.



ANSWER

$$i_1 + \frac{v_1}{R_2} + \frac{v_1 - v_2}{R_3} + \frac{v_1 - v_2}{R_4} = 0$$

$$i_2 + \frac{v_1 - v_2}{R_3} + \frac{v_1 - v_2}{R_4} = 0$$

In what follows, we will demonstrate three techniques for solving linearly independent simultaneous equations: Gaussian elimination, matrix analysis, and the MATLAB mathematical software package. A brief refresher that illustrates the use of both Gaussian elimination and matrix analysis in the solution of these equations is provided in the Problem-Solving Companion for this text. The use of the MATLAB software is straightforward, and we will demonstrate its use as we encounter the application.

The KCL equations at nodes 1 and 2 produced two linearly independent simultaneous equations:

$$-i_A + i_1 + i_2 = 0$$
$$-i_2 + i_B + i_3 = 0$$

The KCL equation for the third node (reference) is

$$+i_A - i_1 - i_B - i_3 = 0$$

Note that if we add the first two equations, we obtain the third. Furthermore, any two of the equations can be used to derive the remaining equation. Therefore, in this $N = 3$ node circuit, only $N - 1 = 2$ of the equations are linearly independent and required to determine the $N - 1 = 2$ unknown node voltages.

Note that a nodal analysis employs KCL in conjunction with Ohm's law. Once the direction of the branch currents has been *assumed*, then Ohm's law, as illustrated by Fig. 3.2 and expressed by Eq. (3.1), is used to express the branch currents in terms of the unknown node voltages. We can assume the currents to be in any direction. However, once we assume a particular direction, we must be very careful to write the currents correctly in terms of the node voltages using Ohm's law.

LEARNING Example 3.1

Suppose that the network in Fig. 3.4 has the following parameters: $I_A = 1$ mA, $R_1 = 12$ kΩ, $R_2 = 6$ kΩ, $I_B = 4$ mA, and $R_3 = 6$ kΩ. Let us determine all node voltages and branch currents.

SOLUTION For purposes of illustration we will solve this problem using Gaussian elimination, matrix analysis, and MATLAB. Using the parameter values Eq. (3.2) becomes

$$V_1\left[\frac{1}{12k} + \frac{1}{6k}\right] - V_2\left[\frac{1}{6k}\right] = 1 \times 10^{-3}$$

$$-V_1\left[\frac{1}{6k}\right] + V_2\left[\frac{1}{6k} + \frac{1}{6k}\right] = -4 \times 10^{-3}$$

where we employ capital letters because the voltages are constant. The equations can be written as

$$\frac{V_1}{4k} - \frac{V_2}{6k} = 1 \times 10^{-3}$$

$$-\frac{V_1}{6k} + \frac{V_2}{3k} = -4 \times 10^{-3}$$

Using Gaussian elimination, we solve the first equation for V_1 in terms of V_2:

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

This value is then substituted into the second equation to yield

$$\frac{-1}{6k}\left(\frac{2}{3}V_2 + 4\right) + \frac{V_2}{3k} = -4 \times 10^{-3}$$

or

$$V_2 = -15 \text{ V}$$

This value for V_2 is now substituted back into the equation for V_1 in terms of V_2, which yields

$$V_1 = \frac{2}{3}V_2 + 4$$

$$= -6 \text{ V}$$

The circuit equations can also be solved using matrix analysis. The general form of the matrix equation is

$$\mathbf{GV} = \mathbf{I}$$

where in this case

$$\mathbf{G} = \begin{bmatrix} \frac{1}{4k} & -\frac{1}{6k} \\ -\frac{1}{6k} & \frac{1}{3k} \end{bmatrix}, \mathbf{V} = \begin{bmatrix} V_1 \\ V_2 \end{bmatrix}, \text{and } \mathbf{I} = \begin{bmatrix} 1 \times 10^{-3} \\ -4 \times 10^{-3} \end{bmatrix}$$

The solution to the matrix equation is

$$\mathbf{V} = \mathbf{G}^{-1}\mathbf{I}$$

and therefore,

$$\begin{bmatrix} V_1 \\ V_2 \end{bmatrix} = \begin{bmatrix} \frac{1}{4k} & \frac{-1}{6k} \\ \frac{-1}{6k} & \frac{1}{3k} \end{bmatrix}^{-1} \begin{bmatrix} 1 \times 10^{-3} \\ -4 \times 10^{-3} \end{bmatrix}$$

To calculate the inverse of **G**, we need the adjoint and the determinant. The adjoint is

$$\text{Adj } \mathbf{G} = \begin{bmatrix} \frac{1}{3k} & \frac{1}{6k} \\ \frac{1}{6k} & \frac{1}{4k} \end{bmatrix}$$

and the determinant is

$$|\mathbf{G}| = \left(\frac{1}{3k}\right)\left(\frac{1}{4k}\right) - \left(\frac{-1}{6k}\right)\left(\frac{-1}{6k}\right)$$

$$= \frac{1}{18k^2}$$

Therefore,

$$\begin{bmatrix} V_1 \\ V_2 \end{bmatrix} = 18k^2 \begin{bmatrix} \frac{1}{3k} & \frac{1}{6k} \\ \frac{1}{6k} & \frac{1}{4k} \end{bmatrix} \begin{bmatrix} 1 \times 10^{-3} \\ -4 \times 10^{-3} \end{bmatrix}$$

$$= 18k^2 \begin{bmatrix} \frac{1}{3k^2} - \frac{4}{6k^2} \\ \frac{1}{6k^2} - \frac{1}{k^2} \end{bmatrix}$$

$$= \begin{bmatrix} -6 \\ -15 \end{bmatrix}$$

The MATLAB solution begins with the set of equations expressed in matrix form as

```
G*V=I
```

where the symbol * denotes the multiplication of the voltage vector **V** by the coefficient matrix **G**. Then once the MATLAB software is loaded into the PC, the coefficient matrix (**G**) and the vector **V** can be expressed in MATLAB notation by typing in the rows of the matrix or vector at the prompt >>. Use semicolons to separate rows and spaces to separate columns. Brackets are used to denote vectors or matrices. When the matrix **G** and the vector **I** have been defined, then the solution equation

```
V=inv(G)*I
```

which is also typed in at the prompt >>, will yield the unknown vector **V**.

The matrix equation for our circuit expressed in decimal notation is

$$\begin{bmatrix} 0.00025 & -0.00016666 \\ -0.00016666 & 0.0003333 \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \end{bmatrix} = \begin{bmatrix} 0.001 \\ -0.004 \end{bmatrix}$$

If we now input the coefficient matrix **G**, then the vector **I** and finally the equation V = inv(G)*I, the computer screen containing these data and the solution vector **V** appears as follows:

```
>> G = [0.00025 -0.000166666;
-0.000166666 0.00033333]
   G =
          1.0e-003 *
          0.2500        -0.1667
         -0.1667         0.3333
>> I = [0.001 ; -0.004]
   I =
          0.0010
         -0.0040
>> V = inv(G)*I
   V =
        -6.0001
       -15.0002
```

Knowing the node voltages, we can determine all the currents using Ohm's law:

$$I_1 = \frac{V_1}{R_1} = \frac{-6}{12\text{k}} = -\frac{1}{2}\text{ mA}$$

$$I_2 = \frac{V_1 - V_2}{6\text{k}} = \frac{-6-(-15)}{6\text{k}} = \frac{3}{2}\text{ mA}$$

and

$$I_3 = \frac{V_2}{6\text{k}} = \frac{-15}{6\text{k}} = -\frac{5}{2}\text{ mA}$$

Figure 3.5 illustrates the results of all the calculations. Note that KCL is satisfied at every node.

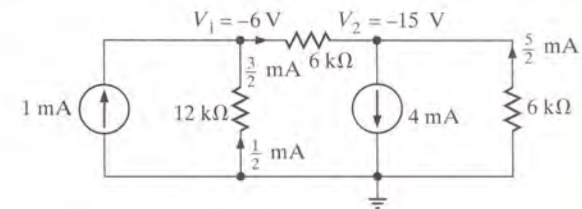


Figure 3.5 *Circuit used in Example 3.1.*

Let us now examine the circuit in Fig. 3.6. The current directions are assumed as shown in the figure.

At node 1, KCL yields

$$i_1 - i_A + i_2 - i_3 = 0$$

or

$$\frac{v_1}{R_1} - i_A + \frac{v_1 - v_2}{R_2} - \frac{v_3 - v_1}{R_3} = 0$$

$$v_1\left(\frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3}\right) - v_2\frac{1}{R_2} - v_3\frac{1}{R_3} = i_A$$

At node 2, KCL yields

$$-i_2 + i_4 - i_5 = 0$$

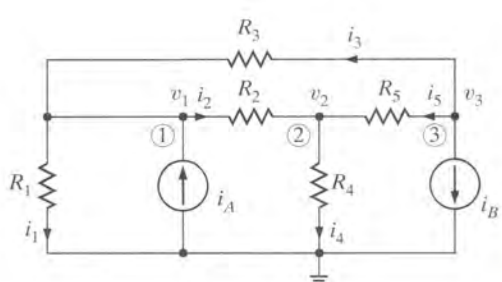


Figure 3.6
A four-node circuit.

or

$$-\frac{v_1 - v_2}{R_2} + \frac{v_2}{R_4} - \frac{v_3 - v_2}{R_5} = 0$$

$$-v_1\frac{1}{R_2} + v_2\left(\frac{1}{R_2} + \frac{1}{R_4} + \frac{1}{R_5}\right) - v_3\frac{1}{R_5} = 0$$

At node 3, the equation is

$$i_3 + i_5 + i_B = 0$$

or

$$\frac{v_3 - v_1}{R_3} + \frac{v_3 - v_2}{R_5} + i_B = 0$$

$$-v_1\frac{1}{R_3} - v_2\frac{1}{R_5} + v_3\left(\frac{1}{R_3} + \frac{1}{R_5}\right) = -i_B$$

Grouping the node equations together, we obtain

$$\begin{aligned} v_1\left(\frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3}\right) - v_2\frac{1}{R_2} - v_3\frac{1}{R_3} &= i_A \\ -v_1\frac{1}{R_2} + v_2\left(\frac{1}{R_2} + \frac{1}{R_4} + \frac{1}{R_5}\right) - v_3\frac{1}{R_5} &= 0 \\ -v_1\frac{1}{R_3} - v_2\frac{1}{R_5} + v_3\left(\frac{1}{R_3} + \frac{1}{R_5}\right) &= -i_B \end{aligned} \qquad \mathbf{3.3}$$

Note that our analysis has produced three simultaneous equations in the three unknown node voltages v_1, v_2, and v_3. The equations can also be written in matrix form as

$$\begin{bmatrix} \frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3} & -\frac{1}{R_2} & -\frac{1}{R_3} \\ -\frac{1}{R_2} & \frac{1}{R_2} + \frac{1}{R_4} + \frac{1}{R_5} & -\frac{1}{R_5} \\ -\frac{1}{R_3} & -\frac{1}{R_5} & \frac{1}{R_3} + \frac{1}{R_5} \end{bmatrix} \begin{bmatrix} v_1 \\ v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} i_A \\ 0 \\ -i_B \end{bmatrix} \qquad \mathbf{3.4}$$

At this point it is important that we note the symmetrical form of the equations that describe the two previous networks. Equations (3.2) and (3.3) exhibit the same type of symmetrical form. The **G** matrix for each network is a symmetrical matrix. This symmetry is not accidental. The node equations for networks containing only resistors and independent current sources can always be written in this symmetrical form. We can take advantage of this fact and learn to write the equations by inspection. Note in the first equation of (3.2) that the coefficient of v_1 is the sum of all the conductances connected to node 1 and the coefficient of v_2 is the negative of the conductances connected between node 1 and node 2. The right-hand side of the equation is the sum of the currents entering node 1 through current sources. This equation is KCL at node 1. In the second equation in (3.2), the coefficient of v_2 is the sum of all the conductances connected to node 2, the coefficient of v_1 is the negative of the conductance connected between node 2 and node 1, and the right-hand side of the equation is the sum of the currents entering node 2 through current sources. This equation is KCL at node 2. Similarly, in the first equation in (3.3) the coefficient of v_1 is the sum of the conductances connected to node 1, the coefficient of v_2 is the negative of the conductance connected between node 1 and node 2, the coefficient of v_3 is the negative of the conductance connected between node 1 and node 3, and the right-hand side of the equation is the sum of the currents entering node 1 through current sources. The other two equations in (3.3) are obtained in a similar manner. In general, if KCL is applied to node j with node voltage v_j, the coefficient of v_j is the sum of all the conductances connected to node j and the coefficients of the other node voltages $(\text{e.g.}, v_{j-1}, v_{j+1})$ are the negative of the sum of the conductances connected directly between these nodes and node j. The right-hand side of the equation is equal to the sum of the currents entering the node via current sources. Therefore, the left-hand side of the equation represents the sum of the currents leaving node j and the right-hand side of the equation represents the currents entering node j.

LEARNING Example 3.2

Let us apply what we have just learned to write the equations for the network in Fig. 3.7 by inspection. Then given the following parameters, we will determine the node voltages using MATLAB: $R_1 = R_2 = 2\ \text{k}\Omega$, $R_3 = R_4 = 4\ \text{k}\Omega$, $R_5 = 1\ \text{k}\Omega$, $i_A = 4$ mA, and $i_B = 2$ mA.

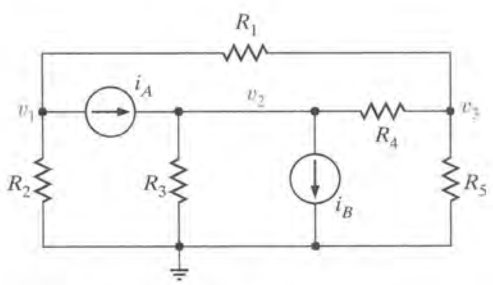


Figure 3.7 Circuit used in Example 3.2.

SOLUTION The equations are

$$v_1\left(\frac{1}{R_1}+\frac{1}{R_2}\right) - v_2(0) - v_3\left(\frac{1}{R_1}\right) = -i_A$$

$$-v_1(0) + v_2\left(\frac{1}{R_3}+\frac{1}{R_4}\right) - v_3\left(\frac{1}{R_4}\right) = i_A - i_B$$

$$-v_1\left(\frac{1}{R_1}\right) - v_2\left(\frac{1}{R_4}\right) + v_3\left(\frac{1}{R_1}+\frac{1}{R_4}+\frac{1}{R_5}\right) = 0$$

which can also be written directly in matrix form as

$$\begin{bmatrix} \frac{1}{R_1}+\frac{1}{R_2} & 0 & -\frac{1}{R_1} \\ 0 & \frac{1}{R_3}+\frac{1}{R_4} & -\frac{1}{R_4} \\ -\frac{1}{R_1} & -\frac{1}{R_4} & \frac{1}{R_1}+\frac{1}{R_4}+\frac{1}{R_5} \end{bmatrix} \begin{bmatrix} v_1 \\ v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} -i_A \\ i_A - i_B \\ 0 \end{bmatrix}$$

Both the equations and the **G** matrix exhibit the symmetry that will always be present in circuits that contain only resistors and current sources.

If the component values are now used, the matrix equation becomes

$$\begin{bmatrix} \frac{1}{2k}+\frac{1}{2k} & 0 & -\frac{1}{2k} \\ 0 & \frac{1}{4k}+\frac{1}{4k} & -\frac{1}{4k} \\ -\frac{1}{2k} & -\frac{1}{4k} & \frac{1}{2k}+\frac{1}{4k}+\frac{1}{1k} \end{bmatrix} \begin{bmatrix} v_1 \\ v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} -0.004 \\ 0.002 \\ 0 \end{bmatrix}$$

or

$$\begin{bmatrix} 0.001 & 0 & -0.0005 \\ 0 & 0.0005 & -0.00025 \\ -0.0005 & -0.00025 & 0.00175 \end{bmatrix} \begin{bmatrix} v_1 \\ v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} -0.004 \\ 0.002 \\ 0 \end{bmatrix}$$

If we now employ these data with the MATLAB software, the computer screen containing the data and the results of the MATLAB analysis is as shown next.

```
>> G = [0.001 0 -0.0005 ; 0 0.0005
-0.00025 ; -0.0005 -0.00025 0.00175]

   G =

        0.0010          0    -0.0005
             0     0.0005    -0.0003
       -0.0005    -0.0003     0.0018

>> I = [-0.004 ; 0.002 ; 0]

   I =

       -0.0040
        0.0020
             0

>> V = inv(G)*I

   V =

       -4.3636
        3.6364
       -0.7273
```

LEARNING EXTENSIONS

E3.1 Write the node equations for the circuit in Fig. E3.1.

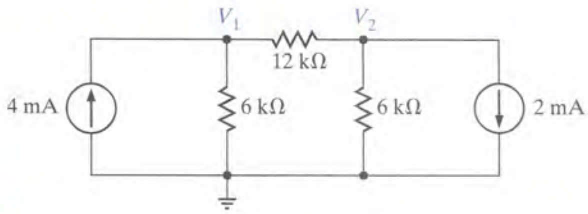


Figure E3.1

ANSWER

$$\frac{1}{4\text{k}}V_1 - \frac{1}{12\text{k}}V_2 = 4 \times 10^{-3},$$
$$\frac{-1}{12\text{k}}V_1 + \frac{1}{4\text{k}}V_2 = -2 \times 10^{-3}.$$

E3.2 Find all the node voltages in the network in Fig. E3.2 using MATLAB.

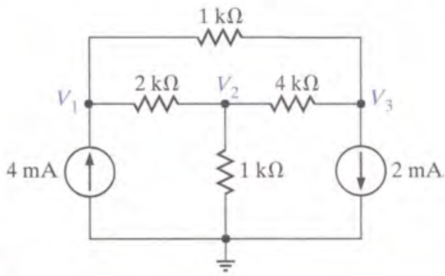


Figure E3.2

ANSWER $V_1 = 5.4286$ V, $V_2 = 2.000$ V, $V_3 = 3.1429$ V.

CIRCUITS CONTAINING DEPENDENT CURRENT SOURCES The presence of a dependent source may destroy the symmetrical form of the nodal equations that define the circuit. Consider the circuit shown in Fig. 3.8, which contains a current-controlled current source. The KCL equations for the nonreference nodes are

$$\beta i_o + \frac{v_1}{R_1} + \frac{v_1 - v_2}{R_2} = 0$$

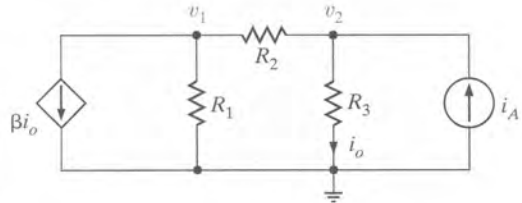


Figure 3.8
Circuit with a dependent source.

and

$$\frac{v_2 - v_1}{R_2} + i_o - i_A = 0$$

where $i_o = v_2/R_3$. Simplifying the equations, we obtain

$$(G_1 + G_2)v_1 - (G_2 - \beta G_3)v_2 = 0$$
$$-G_2v_1 + (G_2 + G_3)v_2 = i_A$$

or in matrix form

$$\begin{bmatrix} (G_1 + G_2) & -(G_2 - \beta G_3) \\ -G_2 & (G_2 + G_3) \end{bmatrix} \begin{bmatrix} v_1 \\ v_2 \end{bmatrix} = \begin{bmatrix} 0 \\ i_A \end{bmatrix}$$

Note that the presence of the dependent source has destroyed the symmetrical nature of the node equation.

LEARNING Example 3.3

Let us determine the node voltages for the network in Fig. 3.8 given the following parameters:

$\beta = 2$ $\quad R_2 = 6\text{ k}\Omega \quad i_A = 2\text{ mA}$

$R_1 = 12\text{ k}\Omega \quad R_3 = 3\text{ k}\Omega$

SOLUTION Using these values with the equations for the network yields

$$\frac{1}{4k}V_1 + \frac{1}{2k}V_2 = 0$$

$$-\frac{1}{6k}V_1 + \frac{1}{2k}V_2 = 2 \times 10^{-3}$$

Solving these equations using any convenient method yields $V_1 = -24/5$ V and $V_2 = 12/5$ V. We can check these answers by determining the branch currents in the network and then using that information to test KCL at the nodes. For example, the current from top to bottom through R_3 is

$$I_o = \frac{V_2}{R_3} = \frac{12/5}{3k} = \frac{4}{5k}\text{ A}$$

Similarly, the current from right to left through R_2 is

$$I_2 = \frac{V_2 - V_1}{R_2} = \frac{12/5 - (-24/5)}{6k} = \frac{6}{5k}\text{ A}$$

All the results are shown in Fig. 3.9. Note that KCL is satisfied at every node.

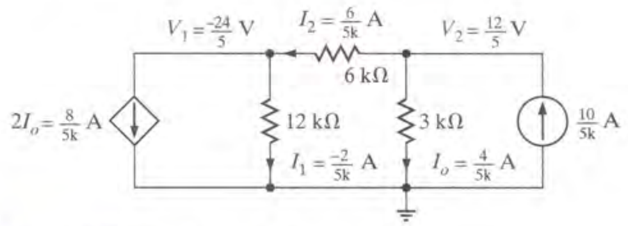


Figure 3.9 Circuit used in Example 3.3.

LEARNING Example 3.4

Let us determine the set of linearly independent equations that when solved will yield the node voltages in the network in Fig. 3.10. Then given the following component values, we will compute the node voltages using MATLAB: $R_1 = 1\text{ k}\Omega$, $R_2 = R_3 = 2\text{ k}\Omega$, $R_4 = 4\text{ k}\Omega$, $i_A = 2$ mA, $i_B = 4$ mA, and $\alpha = 2$.

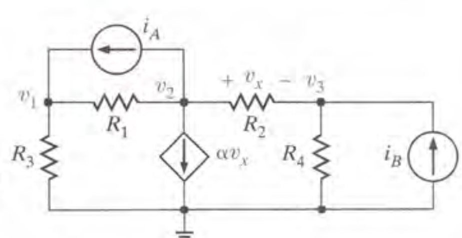


Figure 3.10 Circuit containing a voltage-controlled current source.

SOLUTION Applying KCL at each of the nonreference nodes yields the equations

$$G_3 v_1 + G_1(v_1 - v_2) - i_A = 0$$

$$i_A + G_1(v_2 - v_1) + \alpha v_x + G_2(v_2 - v_3) = 0$$

$$G_2(v_3 - v_2) + G_4 v_3 - i_B = 0$$

where $v_x = v_2 - v_3$. Simplifying these equations, we obtain

$$(G_1 + G_3)v_1 - G_1 v_2 = i_A$$

$$-G_1 v_1 + (G_1 + \alpha + G_2)v_2 - (\alpha + G_2)v_3 = -i_A$$

$$-G_2 v_2 + (G_2 + G_4)v_3 = i_B$$

Given the component values, the equations become

$$\begin{bmatrix} \frac{1}{1k} + \frac{1}{2k} & -\frac{1}{k} & 0 \\ -\frac{1}{k} & \frac{1}{k} + 2 + \frac{1}{2k} & -\left(2 + \frac{1}{2k}\right) \\ 0 & -\frac{1}{2k} & \frac{1}{2k} + \frac{1}{4k} \end{bmatrix} \begin{bmatrix} v_1 \\ v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} 0.002 \\ -0.002 \\ 0.004 \end{bmatrix}$$

or

$$\begin{bmatrix} 0.0015 & -0.001 & 0 \\ -0.001 & 2.0015 & -2.0005 \\ 0 & -0.0005 & 0.00075 \end{bmatrix} \begin{bmatrix} v_1 \\ v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} 0.002 \\ -0.002 \\ 0.004 \end{bmatrix}$$

The MATLAB input and output listings are shown next.

```
>> G = [0.0015 -0.001 0 ; -0.001
2.0015 -2.0005 ; 0 -0.0005 0.00075]

G =

    0.0015   -0.0010         0
   -0.0010    2.0015   -2.0005
         0   -0.0005    0.0008
```

```
>> I = [0.002 ; -0.002 ; 0.004]

   I =

        0.0020
       -0.0020
        0.0040
```

```
>> V = inv(G)*I

   V =

       11.9940
       15.9910
       15.9940
```

LEARNING EXTENSIONS

E3.3 Find the node voltages in the circuit in Fig. E3.3.

ANSWER $V_1 = 16$ V, $V_2 = -8$ V.

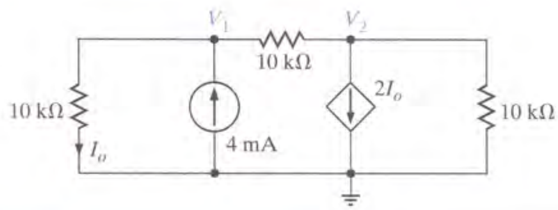


Figure E3.3

E3.4 Find the voltages V_o in the network in Fig. E3.4.

ANSWER $V_o = 4$ V.

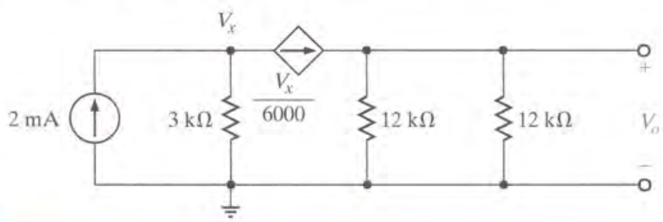


Figure E3.4

CIRCUITS CONTAINING INDEPENDENT VOLTAGE SOURCES As is our practice, in our discussion of this topic we will proceed from the simplest case to those that are more complicated. The simplest case is that in which an independent voltage source is connected to the reference node. The following example illustrates this case.

LEARNING Example 3.5

Consider the circuit shown in Fig. 3.11a. Let us determine all node voltages and branch currents.

SOLUTION This network has three nonreference nodes with labeled node voltages V_1, V_2, and V_3. Based on our previous discussions, we would assume that in order to find all the node voltages we would need to write a KCL equation at each of the nonreference nodes. The resulting three linearly independent simultaneous equations would produce the unknown node voltages. However, note that V_1 and V_3 are known quantities because an independent voltage source is connected directly between the nonreference node and each of these nodes. Therefore, $V_1 = 12$ V and $V_3 = -6$ V. Furthermore, note that the current through the 9-kΩ resistor is $[12 - (-6)]/9\text{k} = 2$ mA from left to right. We do not know V_2 or the current in the remaining resistors. However, since only one node voltage is unknown, a single-node equation will produce it. Applying KCL to this center node yields

$$\frac{V_2 - V_1}{12\text{k}} + \frac{V_2 - 0}{6\text{k}} + \frac{V_2 - V_3}{12\text{k}} = 0$$

or

$$\frac{V_2 - 12}{12\text{k}} + \frac{V_2}{6\text{k}} + \frac{V_2 - (-6)}{12\text{k}} = 0$$

from which we obtain

$$V_2 = \frac{3}{2}\text{ V}$$

Once all the node voltages are known, Ohm's law can be used to find the branch currents shown in Fig. 3.11b. The diagram illustrates that KCL is satisfied at every node.

Note that the presence of the voltage sources in this example has simplified the analysis, since two of the three linear independent equations are $V_1 = 12$ V and $V_3 = -6$ V. We will find that as a general rule, any time voltage sources are present between nodes, the node voltage equations that describe the network will be simpler.

LEARNING Hint

Any time an independent voltage source is connected between the reference node and a nonreference node, the nonreference node voltage is known.

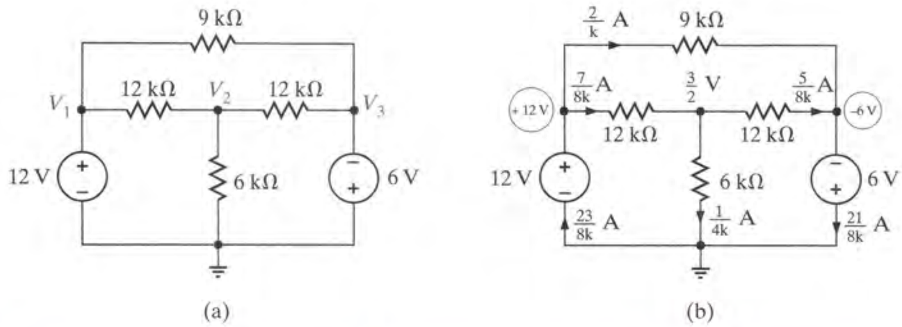


Figure 3.11
Circuit used in Example 3.5.

LEARNING EXTENSION

E3.5 Use nodal analysis to find the current I_o in the network in Fig. E3.5.

ANSWER $I_o = \frac{3}{4}$ mA.

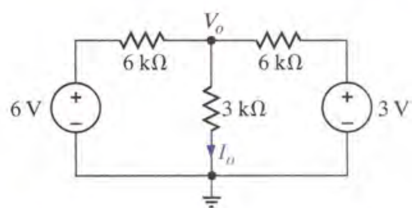


Figure E3.5

Next let us consider the case in which an independent voltage source is connected between two nonreference nodes. Once again, we will use an example to illustrate the approach.

LEARNING Example 3.6

We wish to find the currents in the two resistors in the circuit in Fig. 3.12a.

SOLUTION If we try to attack this problem in a brute force manner, we immediately encounter a problem. Thus far, branch currents were either known source values or could be expressed as the branch voltage divided by the branch resistance. However, the branch current through the 6-V source is certainly not known and cannot be directly expressed using Ohm's law. We can, of course, give this current a name and write the KCL equations

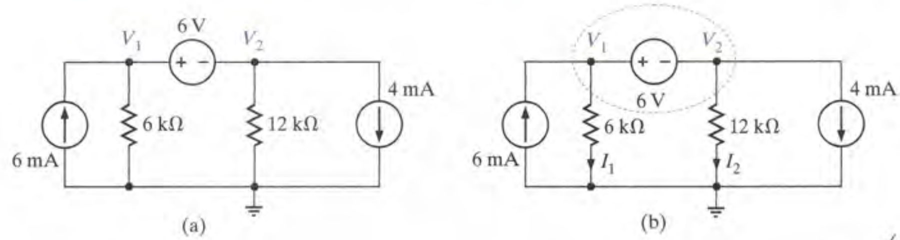


Figure 3.12
Circuits used in Example 3.6.

(continued)

at the two nonreference nodes in terms of this current. However, this approach is no panacea because this technique will result in *two* linearly independent simultaneous equations in terms of *three* unknowns; that is, the two node voltages and the current in the voltage source.

To solve this dilemma, we recall that $N - 1$ linearly independent equations are required to determine the $N - 1$ nonreference node voltages in an N-node circuit. Since our network has three nodes, we need two linearly independent equations. Now note that if somehow one of the node voltages is known, we immediately know the other; that is, if V_1 is known, then $V_2 = V_1 - 6$. If V_2 is known, then $V_1 = V_2 + 6$. Therefore, the difference in potential between the two nodes is *constrained* by the voltage source and, hence,

$$V_1 - V_2 = 6$$

This constraint equation is one of the two linearly independent equations needed to determine the node voltages.

Next consider the network in Fig. 3.12b, in which the 6-V source is completely enclosed within the dashed surface. The constraint equation governs this dashed portion of the network. The remaining equation is obtained by applying KCL to this dashed surface, which is commonly called a *supernode*. Recall that in Chapter 2 we demonstrated that KCL must hold for a surface, and this technique eliminates the problem of dealing with a current through a voltage source. KCL for the supernode is

$$-6 \times 10^{-3} + \frac{V_1}{6\text{k}} + \frac{V_2}{12\text{k}} + 4 \times 10^{-3} = 0$$

Solving these equations yields $V_1 = 10$ V and $V_2 = 4$ V and, hence, $I_1 = 5/3$ mA and $I_2 = 1/3$ mA. A quick check indicates that KCL is satisfied at every node.

LEARNING Hint

The supernode technique

- Use it when a branch between two nonreference nodes contains a voltage source.
- First encircle the voltage source and the two connecting nodes to form the supernode.
- Write the equation that defines the voltage relationship between the two nonreference nodes as a result of the presence of the voltage source.
- Write the KCL equation for the supernode.
- If the voltage source is dependent, then the controlling equation for the dependent source is also needed.

LEARNING Example 3.7

Let us determine the current I_o in the network in Fig. 3.13a.

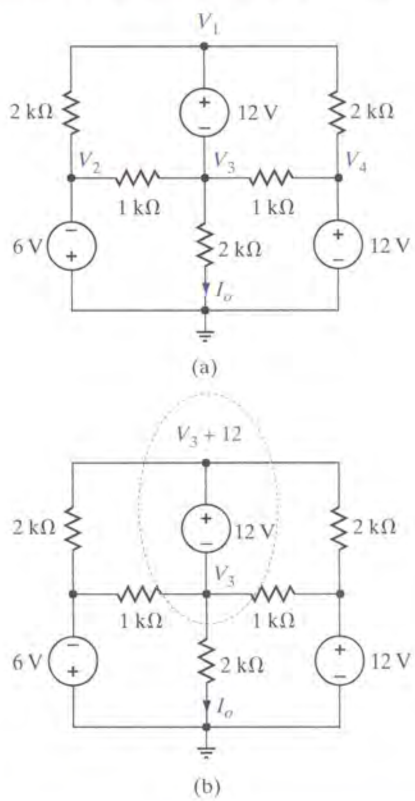


Figure 3.13
Example circuit with supernodes.

SOLUTION Examining the network, we note that node voltages V_2 and V_4 are known and the node voltages V_1 and V_3 are constrained by the equation

$$V_1 - V_3 = 12$$

The network is redrawn in Fig. 3.13b.

Since we want to find the current I_o, V_1 (in the supernode containing V_1 and V_3) is written as $V_3 + 12$. The KCL equation at the supernode is then

$$\frac{V_3 + 12 - (-6)}{2\text{k}} + \frac{V_3 + 12 - 12}{2\text{k}} + \frac{V_3 - (-6)}{1\text{k}} + \frac{V_3 - 12}{1\text{k}} + \frac{V_3}{2\text{k}} = 0$$

Solving the equation for V_3 yields

$$V_3 = -\frac{6}{7}\text{ V}$$

I_o can then be computed immediately as

$$I_o = \frac{-\frac{6}{7}}{2\text{k}} = -\frac{3}{7}\text{ mA}$$

LEARNING EXTENSION

E3.6 Use nodal analysis to find I_o in the network in Fig. E3.6.

ANSWER $I_o = 3.8$ mA.

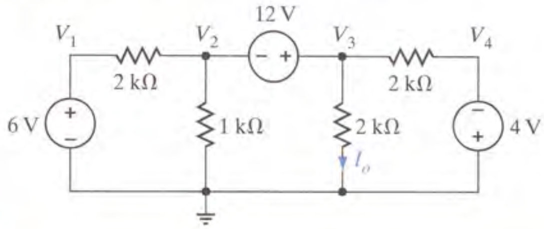


Figure E3.6

CIRCUITS CONTAINING DEPENDENT VOLTAGE SOURCES As the following examples will indicate, networks containing dependent (controlled) sources are treated in the same manner as described earlier.

LEARNING Example 3.8

We wish to find I_o in the network in Fig. 3.14.

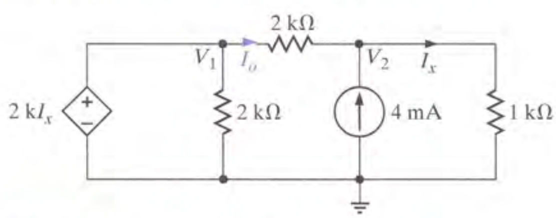


Figure 3.14 *Circuit used in Example 3.8.*

SOLUTION Since the dependent voltage source is connected between the node labeled V_1 and the reference node,

$$V_1 = 2\text{k}I_x$$

KCL at the node labeled V_2 is

$$\frac{V_2 - V_1}{2\text{k}} - \frac{4}{\text{k}} + \frac{V_2}{1\text{k}} = 0$$

where

$$I_x = \frac{V_2}{1\text{k}}$$

Solving these equations yields $V_2 = 8$ V and $V_1 = 16$ V. Therefore

$$I_o = \frac{V_1 - V_2}{2\text{k}}$$

$$= 4 \text{ mA}$$

LEARNING Example 3.9

Let us find the current I_o in the network in Fig. 3.15.

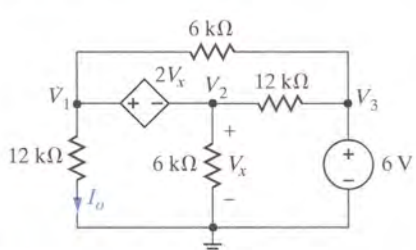


Figure 3.15 *Circuit used in Example 3.9.*

SOLUTION This circuit contains both an independent voltage source and a voltage-controlled voltage source. Note that $V_3 = 6$ V, $V_2 = V_x$, and a supernode exists between the nodes labeled V_1 and V_2.

Applying KCL to the supernode, we obtain

$$\frac{V_1 - V_3}{6\text{k}} + \frac{V_1}{12\text{k}} + \frac{V_2}{6\text{k}} + \frac{V_2 - V_3}{12\text{k}} = 0$$

where the constraint equation for the supernode is

$$V_1 - V_2 = 2V_x$$

The final equation is

$$V_3 = 6$$

Solving these equations, we find that

$$V_1 = \frac{9}{2} \text{ V}$$

and, hence,

$$I_o = \frac{V_1}{12\text{k}} = \frac{3}{8} \text{ mA}$$

LEARNING EXTENSION

E3.7 Use nodal analysis to find I_o in the circuit in Fig. E3.7.

ANSWER $I_o = \frac{4}{3}$ mA.

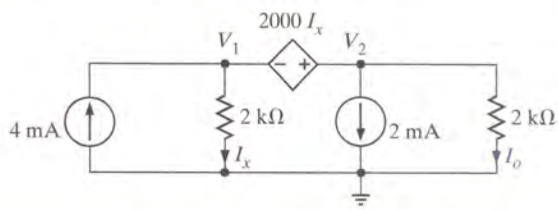


Figure E3.7

Problem-Solving Strategy Nodal Analysis

- Select one node in the N-node circuit as the reference node. Assume that the node voltage is zero and measure all node voltages with respect to this node.
- If only independent current sources are present in the network, write the KCL equations at the $N - 1$ nonreference nodes. If dependent current sources are present, write the KCL equations as is done for networks with only independent current sources; then write the controlling equations for the dependent sources.
- If voltage sources are present in the network, they may be connected (1) between the reference node and a nonreference node or (2) between two nonreference nodes. In the former case, if the voltage source is an independent source, then the voltage at one of the nonreference nodes is known. If the source is dependent, it is treated as an independent source when writing the KCL equation, but an additional constraint equation is necessary, as described previously.

  In the latter case, if the source is independent, the voltage between the two nodes is constrained by the value of the voltage source, and an equation describing this constraint represents one of the $N - 1$ linearly independent equations required to determine the N-node voltages. The surface of the network described by the constraint equation (i.e., the source and two connecting nodes) is called a supernode. One of the remaining $N - 1$ linearly independent equations is obtained by applying KCL at this supernode. If the voltage source is dependent, it is treated as an independent source when writing the KCL equations, but an additional constraint equation is necessary, as described previously.

3.2 Loop Analysis

In a nodal analysis the unknown parameters are the node voltages, and KCL is employed to determine them. In contrast to this approach, a loop analysis uses KVL to determine currents in the circuit. Once the currents are known, Ohm's law can be used to calculate the voltages. Recall that, in Chapter 2, we found that a single equation was sufficient to determine the current in a circuit containing a single loop. If the circuit contains N independent loops, we will show that N independent simultaneous equations will be required to describe the network. For now we will assume that the circuits are planar, which simply means that we can draw the circuit on a sheet of paper in a way such that no conductor crosses another conductor.

Our approach to loop analysis will mirror that used in nodal analysis (i.e., we will begin with simple cases and systematically proceed to those that are more difficult). Then at the end of this section we will outline a general strategy for employing loop analysis.

CIRCUITS CONTAINING ONLY INDEPENDENT VOLTAGE SOURCES To begin our analysis, consider the circuit shown in Fig. 3.16. Let us also identify two loops, *A-B-E-F-A* and *B-C-D-E-B*. We now define a new set of current variables called *loop currents*, which can be used to find the physical currents in the circuit. Let us assume that current i_1 flows in the first loop and that current i_2 flows in the second loop. Then the branch current flowing from *B* to *E* through R_3 is $i_1 - i_2$. The directions of the currents have been assumed. As was the case in the nodal analysis, if the actual currents are not in the direction indicated, the values calculated will be negative.

Applying KVL to the first loop yields

$$+v_1 + v_3 + v_2 - v_{S1} = 0$$

KVL applied to loop 2 yields

$$+v_{S2} + v_4 + v_5 - v_3 = 0$$

where $v_1 = i_1R_1$, $v_2 = i_1R_2$, $v_3 = (i_1 - i_2)R_3$, $v_4 = i_2R_4$, and $v_5 = i_2R_5$.

Substituting these values into the two KVL equations produces the two simultaneous equations required to determine the two loop currents; that is,

$$i_1(R_1 + R_2 + R_3) - i_2(R_3) = v_{S1}$$
$$-i_1(R_3) + i_2(R_3 + R_4 + R_5) = -v_{S2}$$

or in matrix form

$$\begin{bmatrix} R_1 + R_2 + R_3 & -R_3 \\ -R_3 & R_3 + R_4 + R_5 \end{bmatrix} \begin{bmatrix} i_1 \\ i_2 \end{bmatrix} = \begin{bmatrix} v_{S1} \\ -v_{S2} \end{bmatrix}$$

At this point it is important to define what is called a *mesh*. A mesh is a special kind of loop that does not contain any loops within it. Therefore, as we traverse the path of a mesh, we do not encircle any circuit elements. For example, the network in Fig. 3.16 contains two meshes defined by the paths *A-B-E-F-A* and *B-C-D-E-B*. The path *A-B-C-D-E-F-A* is a loop, but it is not a mesh. Since the majority of our analysis in this section will involve writing KVL equations for meshes, we will refer to the currents as mesh currents and the analysis as a *mesh analysis*.

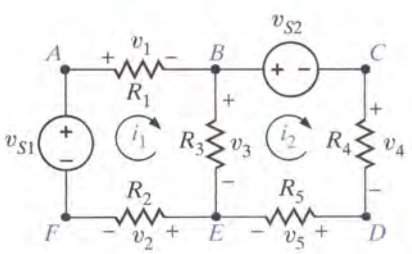


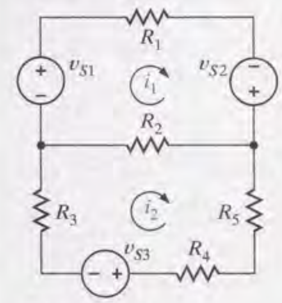
Figure 3.16 A two-loop circuit.

LEARNING Hint

The equations employ the passive sign convention.

LEARNING by Doing

D 3.2 Write the mesh equations for the following circuit.



ANSWER

$-v_{S1} + i_1R_1 - v_{S2} + (i_1 - i_2)R_2 = 0$

$i_2R_3 + (i_2 - i_1)R_2 + i_2R_S + i_2R_4 + v_{S3} = 0$

LEARNING Example 3.10

Consider the network in Fig. 3.17a. We wish to find the current I_o.

SOLUTION We will begin the analysis by writing mesh equations. Note that there are no + and − signs on the resistors. However, they are not needed, since we will apply Ohm's law to each resistive element as we write the KVL equations. The equation for the first mesh is

$$-12 + 6kI_1 + 6k(I_1 - I_2) = 0$$

The KVL equation for the second mesh is

$$6k(I_2 - I_1) + 3kI_2 + 3 = 0$$

where $I_o = I_1 - I_2$.

Solving the two simultaneous equations yields $I_1 = 5/4$ mA and $I_2 = 1/2$ mA. Therefore, $I_o = 3/4$ mA. All the voltages and currents in the network are shown in Fig. 3.17b. Recall from nodal analysis that once the node voltages were determined, we could

(continued)

check our analysis using KCL at the nodes. In this case we know the branch currents and can use KVL around any closed path to check our results. For example, applying KVL to the outer loop yields

$$-12 + \frac{15}{2} + \frac{3}{2} + 3 = 0$$
$$0 = 0$$

Since we want to calculate the current I_o, we could use loop analysis, as shown in Fig. 3.17c. Note that the loop current I_1 passes through the center leg of the network and, therefore, $I_1 = I_o$. The two loop equations in this case are

$$-12 + 6k(I_1 + I_2) + 6kI_1 = 0$$

and

$$-12 + 6k(I_1 + I_2) + 3kI_2 + 3 = 0$$

Solving these equations yields $I_1 = 3/4$ mA and $I_2 = 1/2$ mA. Since the current in the 12-V source is $I_1 + I_2 = 5/4$ mA, these results agree with the mesh analysis.

Finally, for purposes of comparison, let us find I_o using nodal analysis. The presence of the two voltage sources would indicate that this is a viable approach. Applying KCL at the top center node, we obtain

$$\frac{V_o - 12}{6k} + \frac{V_o}{6k} + \frac{V_o - 3}{3k} = 0$$

and hence,

$$V_o = \frac{9}{2}\text{ V}$$

and then

$$I_o = \frac{V_o}{6k} = \frac{3}{4}\text{ mA}$$

Note that in this case we had to solve only one equation instead of two.

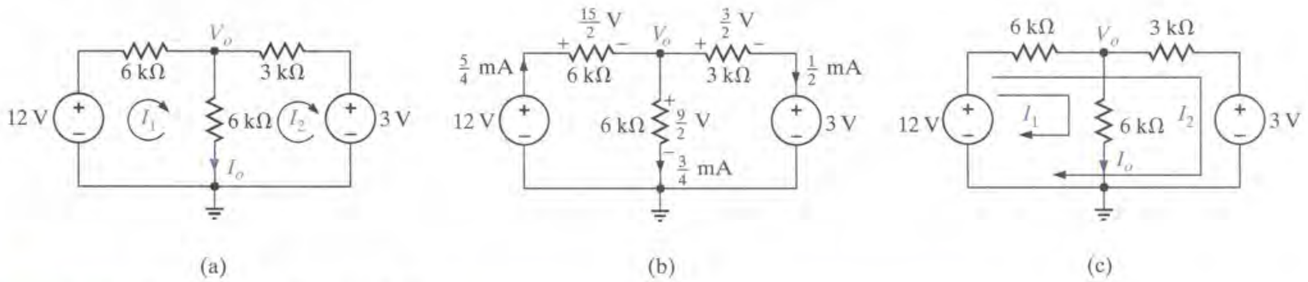


Figure 3.17 Circuits used in Example 3.10.

Once again we are compelled to note the symmetrical form of the mesh equations that describe the circuit in Fig. 3.16. Note that the coefficient matrix for this circuit is symmetrical.

Since this symmetry is generally exhibited by networks containing resistors and independent voltage sources, we can learn to write the mesh equations by inspection. In the first equation, the coefficient of i_1 is the sum of the resistances through which mesh current 1 flows, and the coefficient of i_2 is the negative of the sum of the resistances common to mesh current 1 and mesh current 2. The right-hand side of the equation is the algebraic sum of the voltage sources in mesh 1. The sign of the voltage source is positive if it aids the assumed direction of the current flow and negative if it opposes the assumed flow. The first equation is KVL for mesh 1. In the second equation, the coefficient of i_2 is the sum of all the resistances in mesh 2, the coefficient of i_1 is the negative of the sum of the resistances common to mesh 1 and mesh 2, and the right-hand side of the equation is the algebraic sum of the voltage sources in mesh 2. In general, if we assume all of the mesh currents to be in the same direction (clockwise or counterclockwise), then if KVL is applied to mesh j with mesh current i_j, the coefficient of i_j is the sum of the resistances in mesh j and the coefficients of the other mesh currents (e.g., i_{j-1}, i_{j+1}) are the negatives of the resistances common to these meshes and mesh j. The right-hand side of the equation is equal to the algebraic sum of the voltage sources in mesh j. These voltage sources have a positive sign if they aid the current flow i_j and a negative sign if they oppose it.

LEARNING Example 3.11

Let us write the mesh equations by inspection for the network in Fig. 3.18. Then we will use MATLAB to solve for the mesh currents.

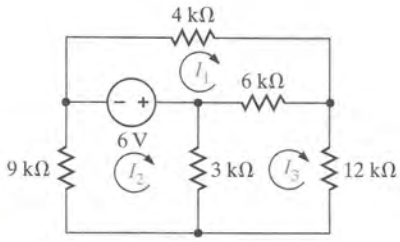


Figure 3.18 Circuit used in Example 3.11.

SOLUTION The three linearly independent simultaneous equations are

$$(4k + 6k)I_1 - (0)I_2 - (6k)I_3 = -6$$
$$-(0)I_1 + (9k + 3k)I_2 - (3k)I_3 = 6$$
$$-(6k)I_1 - (3k)I_2 + (3k + 6k + 12k)I_3 = 0$$

or in matrix form

$$\begin{bmatrix} 10k & 0 & -6k \\ 0 & 12k & -3k \\ -6k & -3k & 21k \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \\ I_3 \end{bmatrix} = \begin{bmatrix} -6 \\ 6 \\ 0 \end{bmatrix}$$

Note the symmetrical form of the equations. The general form of the matrix equation is

$$\mathbf{RI} = \mathbf{V}$$

and the solution of this matrix equation is

$$\mathbf{I} = \mathbf{R}^{-1}\mathbf{V}$$

The input/output data for a MATLAB solution are as follows:

```
>> R = [10e3 0 -6e3; 0 12e3 -3e3;
-6e3 -3e3 21e3]
   R =
             10000        0    -6000
                 0    12000    -3000
             -6000    -3000    21000
>> V = [ -6 ; 6 ; 0]
   V =
         -6
          6
          0
>> I = inv(R)*V
   I =
      1.0e-003 *
        -0.6757
         0.4685
        -0.1261
```

CIRCUITS CONTAINING INDEPENDENT CURRENT SOURCES Just as the presence of a voltage source in a network simplified the nodal analysis, the presence of a current source simplifies a loop analysis. The following examples illustrate the point.

LEARNING EXTENSION

E3.8 Use mesh equations to find V_o in the circuit in Fig. E3.8.

ANSWER $V_o = \frac{33}{5}$ V.

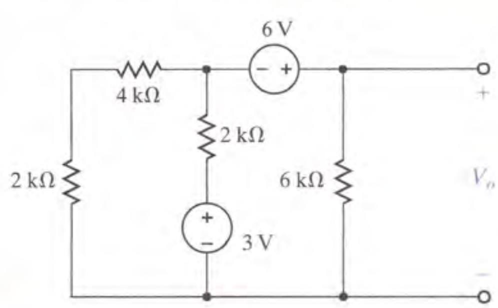


Figure E3.8

LEARNING Example 3.12

We wish to find V_o in the network in Fig. 3.19.

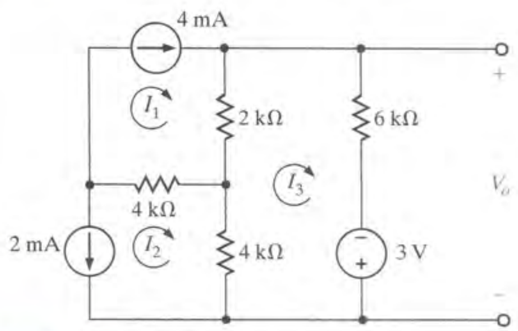


Figure 3.19 Circuit used in Example 3.12.

SOLUTION Since the currents I_1 and I_2 pass directly through a current source, two of the three required equations are

$$I_1 = 4 \times 10^{-3}$$
$$I_2 = -2 \times 10^{-3}$$

The third equation is KVL for the mesh containing the voltage source; that is,

$$4k(I_3 - I_2) + 2k(I_3 - I_1) + 6kI_3 - 3 = 0$$

These equations yield

$$I_3 = \frac{1}{4} \text{ mA}$$

and hence,

$$V_o = 6kI_3 - 3 = \frac{-3}{2} \text{ V}$$

What we have demonstrated in the previous example is the general approach for dealing with independent current sources when writing KVL equations; that is, use one loop through each current source. The number of "window panes" in the network tells us how many equations we need. Additional KVL equations are written to cover the remaining circuit elements in the network. The following example illustrates this approach.

LEARNING Example 3.13

Let us find I_o in the network in Fig. 3.20a.

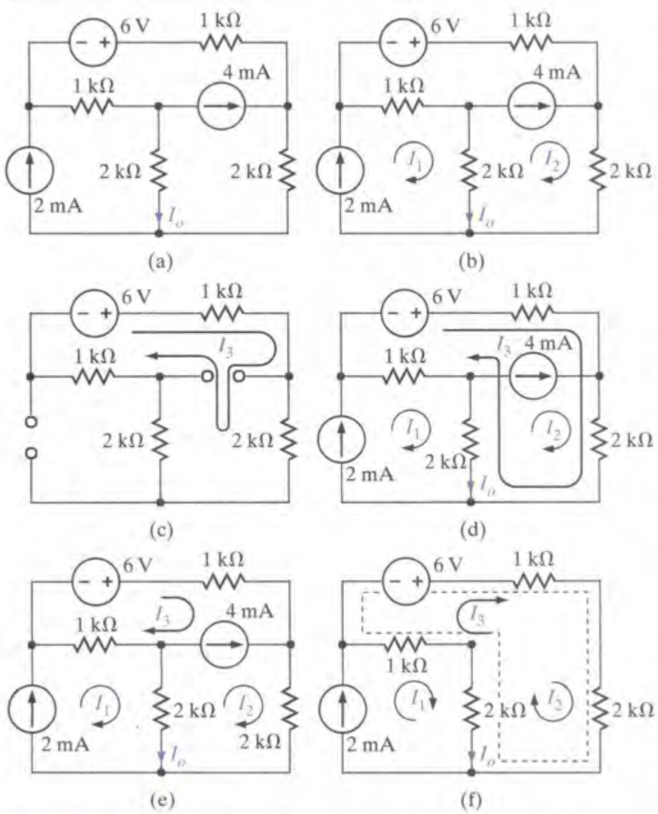


Figure 3.20 Circuits used in Example 3.13.

LEARNING Hint

In this case the 4-mA current source is located on the boundary between two meshes. Thus we will demonstrate two techniques for dealing with this type of situation. One is a special loop technique and the other is known as the supermesh approach.

SOLUTION First, we select two loop currents I_1 and I_2 such that I_1 passes directly through the 2-mA source, and I_2 passes directly through the 4-mA source, as shown in Fig. 3.20b. Therefore, two of our three linearly independent equations are

$$I_1 = 2 \times 10^{-3}$$
$$I_2 = 4 \times 10^{-3}$$

The remaining loop current I_3 must pass through the circuit elements not covered by the two previous equations and cannot, of course, pass through the current sources. The path for this remaining loop current can be obtained by open-circuiting the current sources, as shown in Fig. 3.20c. When all currents are labeled on the original circuit, the KVL equation for this last loop, as shown in Fig. 3.20d, is

$$-6 + 1kI_3 + 2k(I_2 + I_3) + 2k(I_3 + I_2 - I_1) + 1k(I_3 - I_1) = 0$$

Solving the equations yields

$$I_3 = \frac{-2}{3} \text{ mA}$$

and therefore,

$$I_o = I_1 - I_2 - I_3 = \frac{-4}{3} \text{ mA}$$

Next consider the supermesh technique. In this case the three mesh currents are specified as shown in Fig. 3.20e, and since the voltage across the 4-mA current source is unknown, it is assumed to be V_x. The mesh currents constrained by the current sources are

$$I_1 = 2 \times 10^{-3}$$
$$I_2 - I_3 = 4 \times 10^{-3}$$

The KVL equations for meshes 2 and 3, respectively, are

$$2kI_2 + 2k(I_2 - I_1) - V_x = 0$$
$$-6 + 1kI_3 + V_x + 1k(I_3 - I_1) = 0$$

Adding the last two equations yields

$$-6 + 1kI_3 + 2kI_2 + 2k(I_2 - I_1) + 1k(I_3 - I_1) = 0$$

Note that the unknown voltage V_x has been eliminated. The two constraint equations, together with this latter equation, yield the desired result.

The purpose of the supermesh approach is to avoid introducing the unknown voltage V_x. The supermesh is created by mentally removing the 4-mA current source, as shown in Fig. 3.20f. Then writing the KVL equation around the dotted path, which defines the supermesh, using the original mesh currents as shown in Fig. 3.20e, yields

$$-6 + 1kI_3 + 2kI_2 + 2k(I_2 - I_1) + 1k(I_3 - I_1) = 0$$

Note that this supermesh equation is the same as that obtained earlier by introducing the voltage V_x.

LEARNING EXTENSIONS

E3.9 Find V_o in the network in Fig. E3.9.

ANSWER $V_o = \frac{33}{5}$ V.

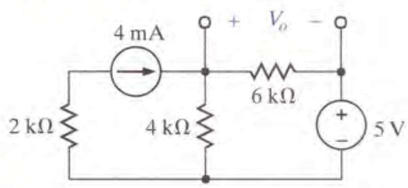


Figure E3.9

3.10 Find V_o in the network in Fig. E3.10.

ANSWER $V_o = \frac{32}{5}$ V.

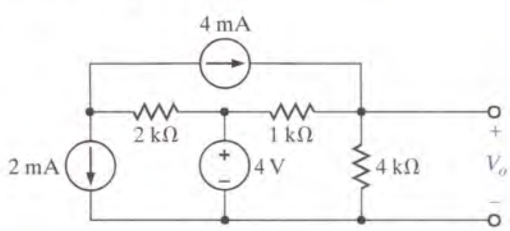


Figure E3.10

CIRCUITS CONTAINING DEPENDENT SOURCES We deal with circuits containing dependent sources just as we have in the past. First, we treat the dependent source as though it were an independent source when writing the KVL equations. Then we write the controlling equation for the dependent source. The following examples illustrate the point.

LEARNING Example 3.14

The network in Fig. 3.21 contains both a current-controlled voltage source and a voltage-controlled current source. Let us use MATLAB to determine the loop currents.

SOLUTION The equations for the loop currents shown in the figure are

$$I_1 = \frac{4}{k}$$
$$I_2 = \frac{V_x}{2k}$$
$$-1kI_x + 2k(I_3 - I_1) + 1k(I_3 - I_4) = 0$$
$$1k(I_4 - I_3) + 1k(I_4 - I_2) + 12 = 0$$

(continued)

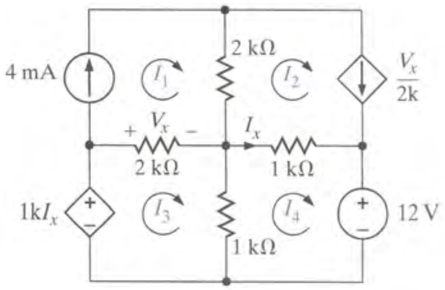


Figure 3.21 *Circuit used in Example 3.14.*

where

$$V_x = 2k(I_3 - I_1)$$
$$I_x = I_4 - I_2$$

Combining these equations yields

$$I_1 = \frac{4}{k}$$
$$I_1 + I_2 - I_3 = 0$$
$$1kI_2 + 3kI_3 - 2kI_4 = 8$$
$$1kI_2 + 1kI_3 - 2kI_4 = 12$$

In matrix form the equations are

$$\begin{bmatrix} 1 & 0 & 0 & 0 \\ 1 & 1 & -1 & 0 \\ 0 & 1k & 3k & -2k \\ 0 & 1k & 1k & -2k \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \\ I_3 \\ I_4 \end{bmatrix} = \begin{bmatrix} \frac{4}{k} \\ 0 \\ 8 \\ 12 \end{bmatrix}$$

The input and output data for the MATLAB solution are as follows:

```
>> R = [ 1 0 0 0 ; 1 1 -1 0;
0 1000 3000 -2000;
0 1000 1000 -2000]

R =

     1       0       0       0
     1       1      -1       0
     0    1000    3000   -2000
     0    1000    1000   -2000

>> V = [ 0.004; 0; 8; 12]

V =

   0.0040
        0
   8.0000
  12.0000

>> I = inv(R)*V

I =

   0.0040
  -0.0060
  -0.0020
  -0.0100
```

As a final point, it is very important to examine the circuit carefully before selecting an analysis approach. One method could be much simpler than another, and a little time invested up front may save a lot of time in the long run.

LEARNING EXTENSION

E3.11 Use mesh analysis to find V_o in the circuit in Fig. E3.11.

ANSWER $V_o = 12$ V.

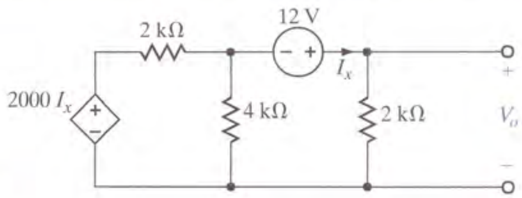


Figure E3.11

Problem-Solving Strategy
Loop Analysis

- One loop current is assigned to each independent loop in a circuit that contains N independent loops.
- If only independent voltage sources are present in the network, write the N linearly independent KVL equations, one for each loop. If dependent voltage sources are present, write the KVL equation as is done for circuits with only independent voltage sources; then write the controlling equations for the dependent sources.
- If current sources are present in the network, either of two techniques can be used. In the first case, one loop current is selected to pass through one of the current sources. This is done for each current source in the network. The remaining loop currents (N − the number of current sources) are determined by open-circuiting the current sources in the network and using this modified network to select them. Once all these currents are defined in the original network, the N loop equations can be written. The second approach is similar to the first with the exception that if two mesh currents pass through a particular current source, a supermesh is formed around this source. The two required equations for the meshes containing this source are the constraint equations for the two mesh currents that pass through the source and the supermesh equation. As indicated earlier, if dependent current sources are present, the controlling equations for these sources are also necessary.

LEARNING EXTENSIONS

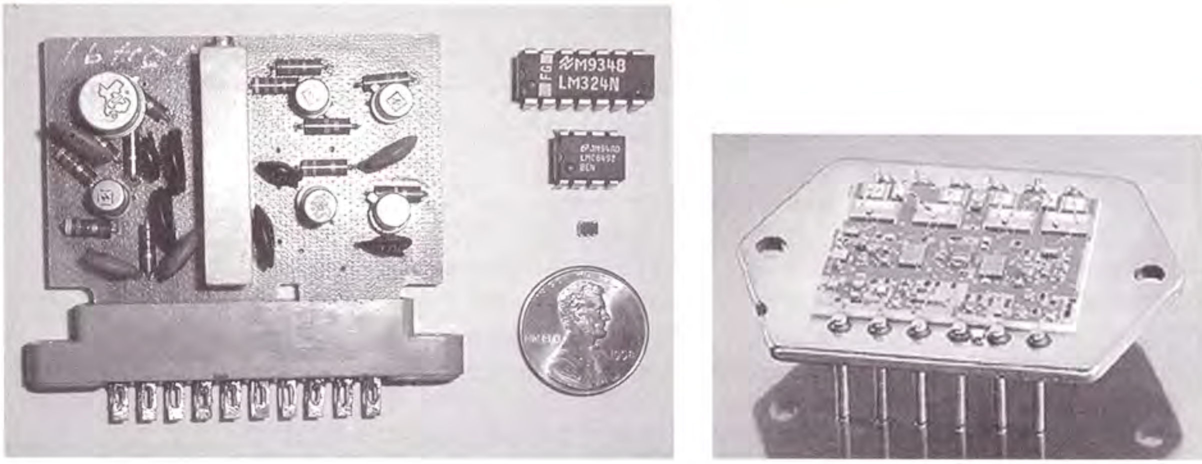
E3.12 Use loop analysis to solve the network in Example 3.5 and compare the time and effort involved in the two solution techniques.

E3.13 Use nodal analysis to solve the circuit in Example 3.12 and compare the time and effort involved in the two solution strategies.

3.3 Circuits with Operational Amplifiers

It can be argued that the operational amplifier, or op-amp as it is commonly known, is the single most important integrated circuit for analog circuit design. It is a versatile interconnection of transistors and resistors that vastly expands our capabilities in circuit design, from engine control systems to cellular phones. Early op-amps were built with vacuum tubes, making them bulky and power hungry. The invention of the transistor at Bell Labs in 1947 allowed engineers to create op-amps that were much smaller and more efficient. Still, the op-amp itself consisted of individual transistors and resistors interconnected on a printed circuit board (PCB). When the manufacturing process for integrated circuits (ICs) was developed around 1970, engineers could finally put all of the op-amps transistors and resistors onto a single IC chip. Today, it is common to find as many as four high quality op-amps on a single IC for as little as \$0.40. A sample of commercial op-amps is shown in Fig. 3.22.

Let us first examine the origin of the term operational amplifier. Originally, the op-amp was designed to perform mathematical operations such as addition, subtraction, differentiation, and integration. By adding simple networks to the op-amp, we can create these "building blocks" as well as other functions such as voltage scaling, current-to-voltage conversion, and a myriad of more complex applications.

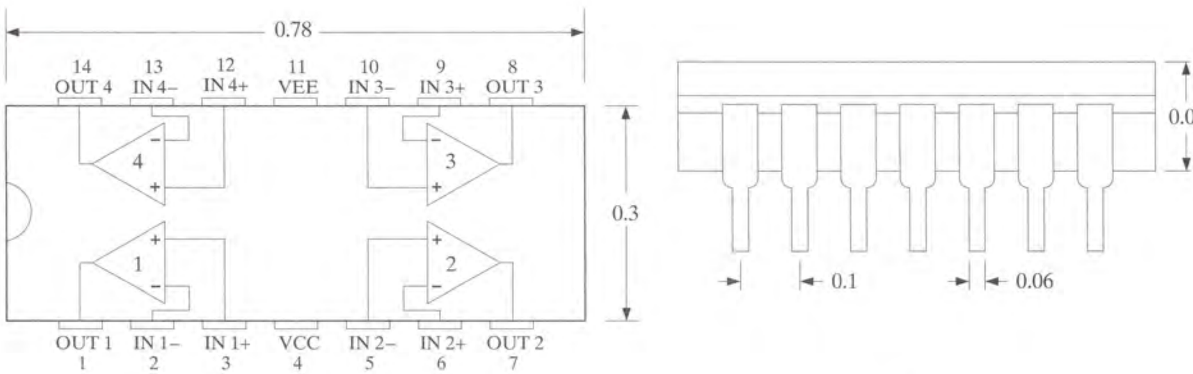


(a) (b)

Figure 3.22 *A selection of op-amps. On the left in (a) is a discrete op-amp assembled on a printed circuit board (PCB). On the right from top to bottom, a LM324 DIP (dual in-line pack), LMC6294 DIP, and MAX4240 in a SO-5 package (small outline/5 pins). A penny is shown for purposes of comparison. In (b) is the APEX PA03 with its lid removed showing individual transistors and resistors.*

How can we, understanding only sources and resistors, hope to comprehend the performance of the op-amp? The answer is modeling. When all the bells and whistles are removed, an op-amp is simply a very good voltage amplifier. In other words, the output voltage is a scaled replica of the input voltage. Modern op-amps are such good amplifiers that it is easy to create an accurate, first-order model. As mentioned earlier, the op-amp is very popular and is used extensively in circuit design at all levels. We should not be surprised to find that op-amps are available for every application—low voltage, high voltage, micro-power, high speed, high current, and so forth. Fortunately, the topology of our model is independent of these issues.

We begin our discussion with the general purpose LM324 quad (four in a pack) op-amp from National Semiconductor. The pinout for the LM324 is shown in Fig. 3.23 for a DIP (dual in-



(a) (b)

Figure 3.23 *(a) The pinout and (b) the dimensional diagram of one side of the LM324 quad op-amp. Note the pin pitch (distance pin-to-pin) is 0.1 inches, a standard for DIP packages.*

line pack) style package with the dimensions specified in inches. Recognizing there are four identical op-amps in the package, we will focus on amplifier 1. Pins 3 and 2 are the input pins, IN_+ and IN_-, and are called the noninverting and inverting inputs, respectively. The output is at pin 1. The relationship that exists between the output and input voltages is

$$V_o = A_o(IN_+ - IN_-) \tag{3.5}$$

where all voltages are measured with respect to ground and A_o is the gain of the op-amp. From Eq. (3.5), we see that when IN_+, increases, so will V_o. However, if IN_- increases, then V_o will decrease—hence the names noninverting and inverting inputs. We mentioned earlier that op-amps are very good voltage amplifiers. How good? Typical values for A_o are between 10,000 and 1,000,000!

To provide amplification, we need power. This power is obtained from dc voltage sources connected to pins 4 and 11, called V_{CC} and V_{EE}, respectively. Actual values for these power supplies can vary widely depending on the application, from as little as one volt up to several hundred volts. Traditionally, V_{CC} is a positive dc voltage with respect to ground and V_{EE} is either a negative voltage or ground itself.

We can model the input/output relationship of the op-amp, as specified in Eq. (3.5), using a dependent voltage source. The currents into and out of the op-amp terminals (pins 3, 2, and 1) are fairly proportional to the pin voltages; that is, the relationship is essentially that specified by Ohm's law. Thus, we model the I-V performance with two resistors, one at the input terminals (R_i) and another at the output (R_o). The resultant circuit is shown in Fig. 3.24.

Let us now examine the values for A_o, R_i, and R_o. Consider the network in Fig. 3.25, where we have modeled the driving circuit with V_S and a resistance R_{Th1}, and the output load with a resistor R_L.

Since the op-amp is designed to be an excellent voltage amplifier, let us write an equation for the overall gain of the circuit $V_{\text{out}}/V_{\text{in}}$. Using voltage division at the input and again at the output, we quickly produce the expression

$$\frac{V_{\text{out}}}{V_{\text{in}}} = \left[\frac{R_i}{R_i + R_{\text{Th1}}}\right] A_o \left[\frac{R_L}{R_o + R_L}\right]$$

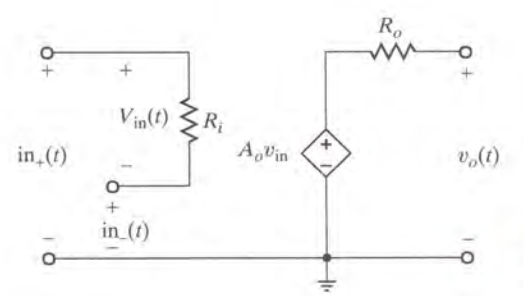


Figure 3.24
A simple model of the gain characteristics of an op-amp.

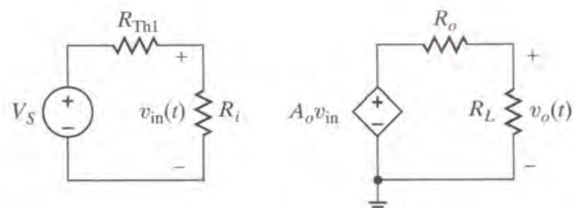


Figure 3.25
A network that depicts an op-amp circuit. V_S and R_{Th1} model the driving circuit, while the load is modeled by R_L.

Table 3.1 A list of commercial op-amps and their model values

Manufacturer	Part No.	A_o (V/V)	R_i (MV)	R_o (Ω)	Comments
National	LM324	100,000	1.0	20	General purpose, up to ±16 V supplies, very inexpensive
National	LMC6492	50,000	10^7	150	Low voltage, rail-to-rail inputs and outputs
Maxim	MAX4240	20,000	45	160	Micro-power (1.8 V supply @ 10 μA), rail-to-rail in inputs and outputs
Apex	PA03	125,000	10^5	2	High voltage, ±75 V, and high output current capability, 30 A. That's 2 kW!

To maximize the gain, regardless of the input resistance and load values, we make A_o very large and the voltage division ratios as close to unity as possible. The ideal scenario requires A_o be infinite, R_i to be infinite, and R_o to be zero, yielding a large overall gain of A_o. Table 3.1 shows the actual values of A_o, R_i, and R_o for a sampling of commercial op-amps intended for very different applications. Although A_o, R_i, and R_o are not ideal, they do approximate the ideal conditions.

The power supplies affect performance in two ways. First, each op-amp has minimum and maximum supply ranges, sometimes called rail-to-rail (a trademark of Motorola Corporation) over which the op-amp is guaranteed to function. Second, for proper operation, the input and output voltages are limited to no more than the supply voltages (Op-amps are available that have input and/or output voltage ranges beyond the supply rails: however, these devices constitute a very small percentage of the op-amp market and will not be discussed here). If the inputs and output can reach within a few dozen millivolts of the supplies, then the inputs and output are called rail-to-rail. Otherwise, the inputs/output limits are more severe—usually a volt or so away from the supply values. Combining the model in Fig. 3.25, the values in Table 3.1, and these I/O limitations, we can produce the graphs in Fig. 3.26, which show the output–input relationship for each op-amp outlined in Table 3.1. From the graph we see that the LM6492 and MAX 4240 have rail-to-rail outputs and the LM324 and PA03 do not.

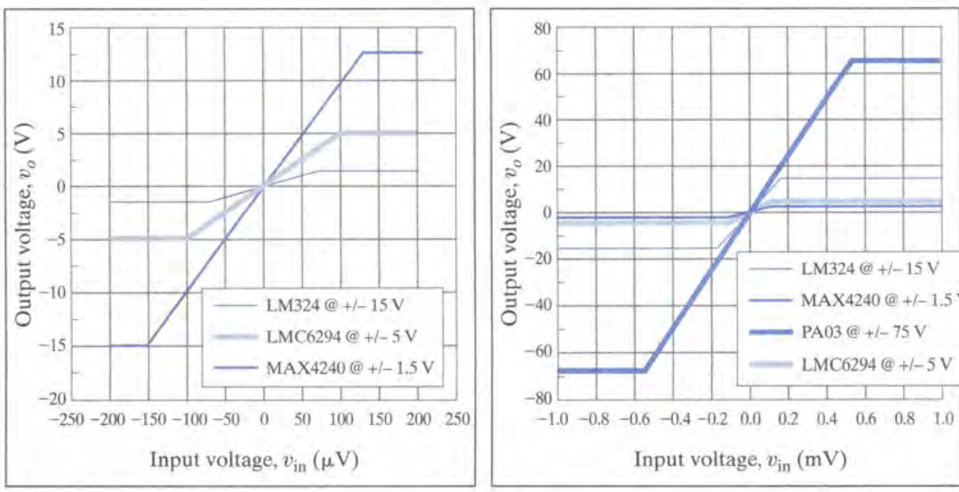


Figure 3.26
Transfer plots for the op-amps listed in Table 3.1. The supply voltages are listed in the plot legends. Note that the LMC6492 and MAX4240 have rail-to-rail output voltages, and the LM324 and PA03 do not.

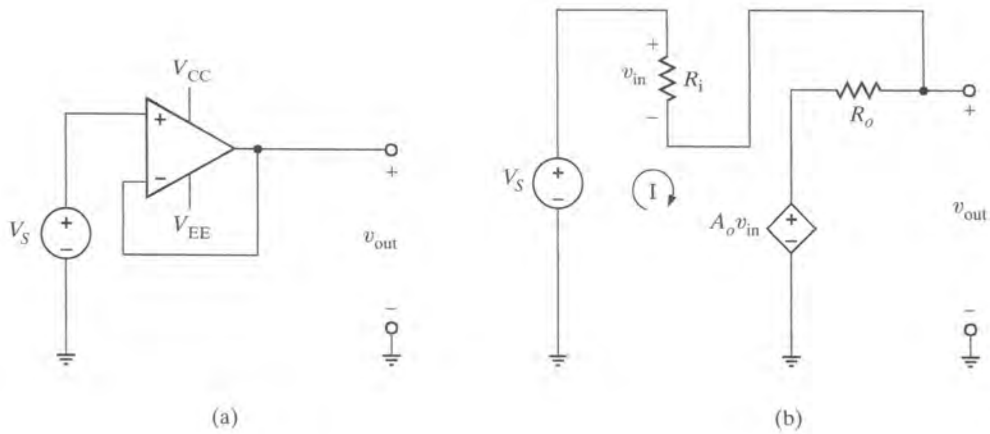


Figure 3.27
Circuit (a) and model (b) for the unity gain buffer.

In order to examine the performance of the op-amp in a practical circuit, consider the network shown in Fig. 3.27a called a unity-gain buffer. Note that the op-amp schematic symbol includes the power supplies. Employing the model in Fig. 3.25 yields the circuit in Fig. 3.27b, containing just resistors and dependent sources, which we can easily analyze. The loop equations for the network are

$$V_S = IR_i + IR_o + A_o V_{in}$$
$$V_{out} = IR_o + A_o V_{in}$$
$$V_{in} = IR_i$$

Solving for the gain, V_{out}/V_S, we find

$$\frac{V_{out}}{V_S} = \frac{1}{1 + \dfrac{R_i}{R_o + A_o R_i}}$$

For $R_o \ll R_i$, we have

$$\frac{V_{out}}{V_S} \approx \frac{1}{1 + \dfrac{1}{A_o}}$$

And, if A_o is indeed $\gg 1$,

$$\frac{V_{out}}{V_S} \approx 1$$

Thus, the origin of the name, unity gain buffer, should be apparent. Table 3.2 shows the actual gain values for $V_S = 1$ V using the op-amps listed in Table 3.1. Note how close the gain is to unity and how small the input voltage and current are. These results lead us to simplify

Table 3.2 Unity gain buffer performance for the op-amps listed in Table 3.1

Op-Amp	Buffer Gain	V_{in} (μV)	I (pA)
LM324	0.999990	9.9999	9.9998
LMC6492	0.999980	19.999	1.9999×10^{-6}
MAX4240	0.999950	49.998	1.1111
PA05	0.999992	7.9999	7.9999×10^{-5}

Table 3.3 Consequences of the ideal op-amp model on input terminal I/V values

Model Assumption	**Terminal Result**
$A_o \to \infty$	Input voltage $\to$ 0 V
$R_i \to \infty$	Input current $\to$ 0 A

the op-amp in Fig. 3.24 significantly. Hence, we introduce the *ideal op-amp model*, where A_o and R_i are infinite and R_o is zero. This selection of parameter values produces two important results for analyzing op-amp circuits, which are listed in Table 3.3.

From Table 3.3 we find that the ideal model for the op-amp is reduced to that shown in Fig. 3.28. The important characteristics of the model are as follows: (1) Since R_i is extremely large, the input currents to the op-amp are approximately zero (i.e., $i_+ \approx i_- \approx 0$); and (2) if the output voltage is to remain bounded, then as the gain becomes very large and approaches infinity, the voltage across the input terminals must simultaneously become infinitesimally small so that as $A \to \infty$, $v_+ - v_- \to 0$ (i.e., $v_+ - v_- = 0$ or $v_+ = v_-$). The difference between these input voltages is often called the *error signal* for the op-amp $\left(\text{i.e., } v_+ - v_- = v_e\right)$.

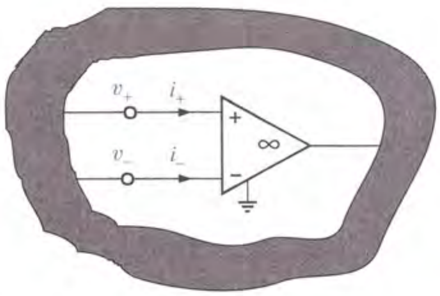


Figure 3.28
Ideal model for an operational amplifier. Model parameters: $i_+ = i_- = 0$, $v_+ = v_-$.

The ground terminal ⏚ shown on the op-amp is necessary for signal current return, and it guarantees that Kirchhoff's current law is satisfied at both the op-amp and the ground node in the circuit.

In summary, then, our ideal model for the op-amp is simply stated by the following conditions:

$$\begin{aligned} i_+ &= i_- = 0 \\ v_+ &= v_- \end{aligned} \qquad \mathbf{3.6}$$

These simple conditions are extremely important because they form the basis of our analysis of op-amp circuits.

Let us now use the ideal model to re-examine the unity gain buffer, redrawn again in Fig. 3.29, where the input voltage and currents are shown as zero. Given that $v_{in} = v_+ - v_-$ is zero, the voltage at both op-amp inputs is V_S. Since the inverting input is physically connected to the output, V_{out} is also V_S—therefore, unity gain!

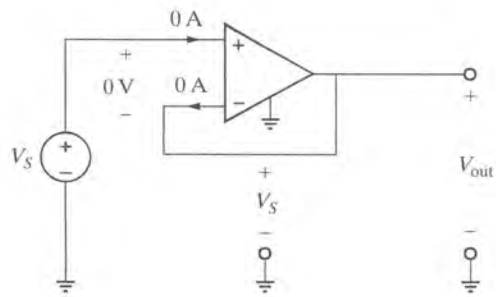


Figure 3.29
As ideal op-amp configured as a unity gain buffer.

An obvious question at this point is this: If $v_o = v_S$, why not just connect v_S to v_o via two parallel connection wires; why do we need to place an op-amp between them? The answer to this question is fundamental and provides us with some insight that will aid us in circuit analysis and design.

Consider the circuit shown in Fig. 3.30a. In this case v_o is not equal to v_S because of the voltage drop across R_S:

$$v_o = v_S - iR_S$$

However, in Fig. 3.30b, the input current to the op-amp is zero and, therefore, v_S appears at the op-amp input. Since the gain of the op-amp configuration is 1, $v_o = v_S$. In Fig. 3.30a the resistive network's interaction with the source caused the voltage v_o to be less than v_S. In other words, the resistive network loads the source voltage. However, in Fig. 3.30b the op-amp isolates the source from the resistive network, and therefore the voltage follower is referred to as a *buffer amplifier* because it can be used to isolate one circuit from another. The energy supplied to the resistive network in the first case must come from the source v_S, whereas in the second case it comes from the power supplies that supply the amplifier, and little or no energy is drawn from v_S.

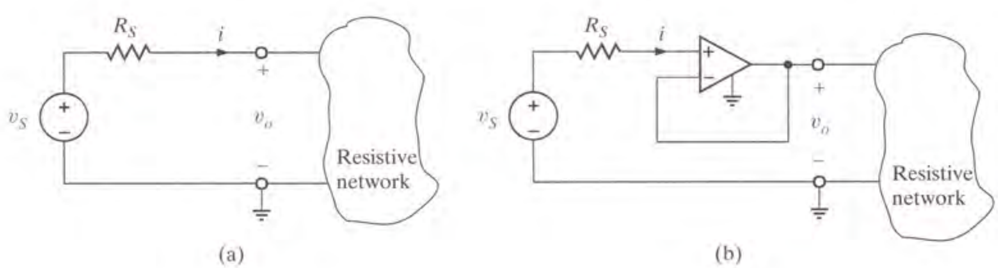


Figure 3.30
Illustration of the isolation capability of a voltage follower.

As a general rule, when analyzing op-amp circuits we write nodal equations at the op-amp input terminals, using the ideal op-amp model conditions. The following example demonstrates the simplicity of this approach.

LEARNING Example 3.15

Let us determine the gain of the basic inverting op-amp configuration shown in Fig. 3.31.

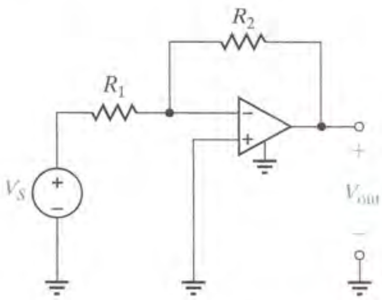


Figure 3.31 *The basic inverting gain stage.*

SOLUTION Using the ideal op-amp model conditions, we see that $v_+ = 0$ and, therefore $v_- = 0$. If we now write a node equation at the negative terminal of the op-amp, we obtain

$$\frac{v_S - 0}{R_1} + \frac{v_o - 0}{R_2} = 0$$

or

$$\frac{v_o}{v_S} = -\frac{R_2}{R_1}$$

Note that the gain is a simple resistor ratio. This fact makes the amplifier very versatile in that we can control the gain accurately and alter its value by changing only one resistor. Also, the gain is essentially independent of op-amp parameters. Since the precise values of A, R_i, and R_o are sensitive to such factors as temperature, radiation, and age, their elimination results in a gain that is stable regardless of the immediate environment. Since it is much easier to employ the ideal op-amp model rather than the nonideal model, unless otherwise stated we will use the ideal op-amp assumptions to analyze circuits that contain operational amplifiers.

LEARNING Example 3.16

Consider the op-amp circuit shown in Fig. 3.32. Let us determine an expression for the output voltage.

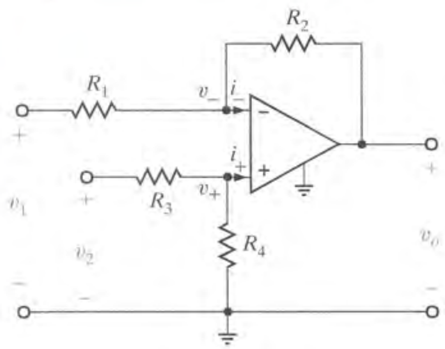


Figure 3.32 *Differential amplifier operational amplifier circuit.*

SOLUTION The node equation at the inverting terminal is

$$\frac{v_1 - v_-}{R_1} + \frac{v_o - v_-}{R_2} = i_-$$

At the noninverting terminal KCL yields

$$\frac{v_2 - v_+}{R_3} = \frac{v_+}{R_4} + i_+$$

However, $i_+ = i_- = 0$ and $v_+ = v_-$. Substituting these values into the two preceding equations yields

$$\frac{v_1 - v_-}{R_1} + \frac{v_o - v_-}{R_2} = 0$$

and

$$\frac{v_2 - v_-}{R_3} = \frac{v_-}{R_4}$$

Solving these two equations for v_o results in the expression

$$v_o = \frac{R_2}{R_1}\left(1 + \frac{R_1}{R_2}\right)\frac{R_4}{R_3 + R_4}v_2 - \frac{R_2}{R_1}v_1$$

Note that if $R_4 = R_2$ and $R_3 = R_1$, the expression reduces to

$$v_o = \frac{R_2}{R_1}(v_2 - v_1)$$

Therefore, this op-amp can be employed to subtract two input voltages.

LEARNING Example 3.17

The circuit shown in Fig. 3.33a is a precision differential voltage-gain device. It is used to provide a single-ended input for an analog-to-digital converter. We wish to derive an expression for the output of the circuit in terms of the two inputs.

SOLUTION To accomplish this, we draw the equivalent circuit shown in Fig. 3.33b. Recall that the voltage across the input terminals of the op-amp is approximately zero and the currents into the op-amp input terminals are approximately zero. Note that we can write node equations for node voltages v_1 and v_2 in terms of v_o and v_a. Since we are interested in an expression for v_o in terms of the voltages v_1 and v_2, we simply eliminate the v_a terms from the two node equations. The node equations are

$$\frac{v_1 - v_o}{R_2} + \frac{v_1 - v_a}{R_1} + \frac{v_1 - v_2}{R_G} = 0$$

$$\frac{v_2 - v_a}{R_1} + \frac{v_2 - v_1}{R_G} + \frac{v_2}{R_2} = 0$$

Combining the two equations to eliminate v_a, and then writing v_o in terms of v_1 and v_2, yields

$$v_o = (v_1 - v_2)\left(1 + \frac{R_2}{R_1} + \frac{2R_2}{R_G}\right)$$

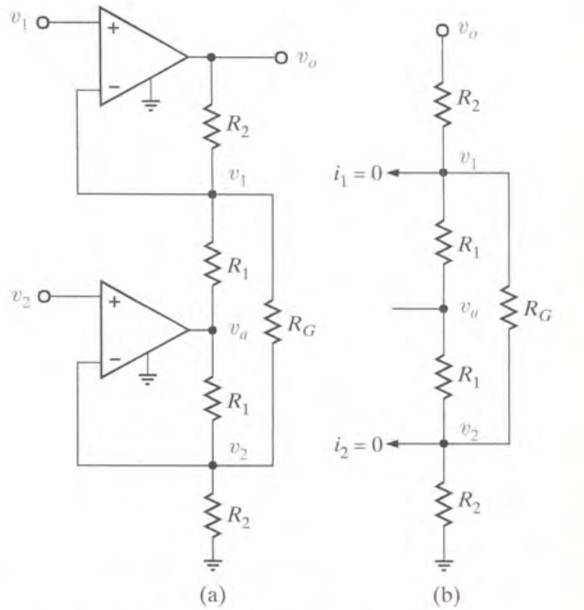


Figure 3.33 *Instrumentation amplifier circuit.*

LEARNING EXTENSIONS

E3.14 Find I_o in the network in Fig. E3.14.

ANSWER $I_o = 8.4$ mA.

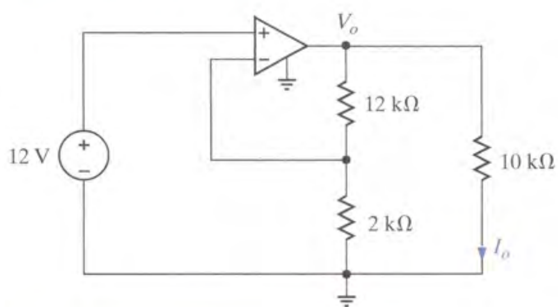


Figure E3.14

E3.15 Determine the gain of the op-amp circuit in Fig. E3.15.

ANSWER $\frac{V_o}{V_S} = 1 + \frac{R_2}{R_1}$.

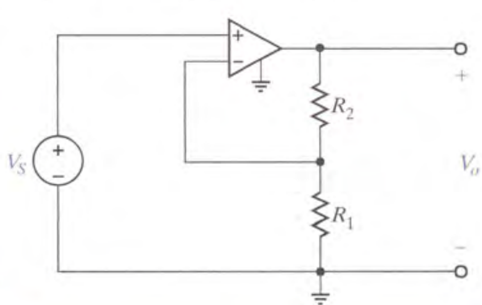


Figure E3.15

LEARNING EXTENSION

E3.16 Determine both the gain and the output voltage of the op-amp configuration shown in Fig. E3.16.

ANSWER $V_o = 0.101$ V, gain = 101.

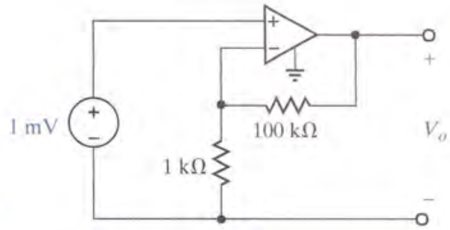


Figure E3.16

COMPARATORS A comparator, a variant of the op-amp, is designed to compare the noninverting and inverting input voltages. As shown in Fig. 3.34, when the noninverting input voltage is greater, the output goes as high as possible, at or near V_{CC}. On the other hand, if the inverting input voltage is greater, the output goes as low as possible, at or near V_{EE}. Of course, an ideal op-amp can do the same thing, that is, swing the output voltage as far as possible. However, op-amps are not designed to operate with the outputs saturated; whereas comparators are. As a result, comparators are faster and less expensive than op-amps.

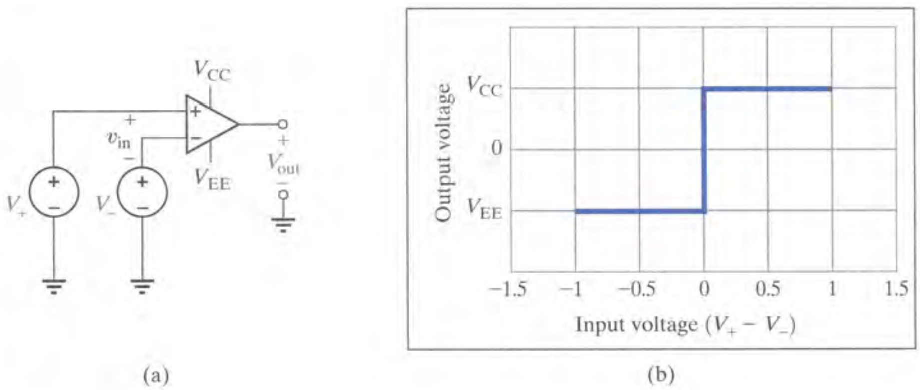


Figure 3.34
(a) An ideal comparator and (b) its transfer curve.

We will present two very different quad comparators in this text, National Semiconductor's LM339 and Maxim's MAX917. Note that the LM339 requires a resistor, called a pull-up resistor, connected between the output pin and V_{CC}. The salient features of these products are listed in Table 3.4. From Table 3.4, it is easy to surmise that the LM339 is a general purpose comparator whereas the MAX917 is intended for low-power applications such as hand-held products.

A common comparator application is the zero-crossing detector, shown in Fig. 3.35a using a LM339 with ±5 V supplies. As seen in Fig. 3.35b, when V_S is positive, V_{out} should be near +5 V and when V_S is negative, V_{out} should be near −5 V. The output changes value on every zero crossing!

Table 3.4 A listing of some of the features of the LM339 and MAX917 comparators

Product	Min. Supply	Max. Supply	Supply Current	Max. Output Current	Typical $R_{PULL-UP}$
LM339	2 V	36 V	3 mA	50 mA	3 kΩ
MAX919	1.8 V	5.5 V	0.8 μA	8 mA	NA

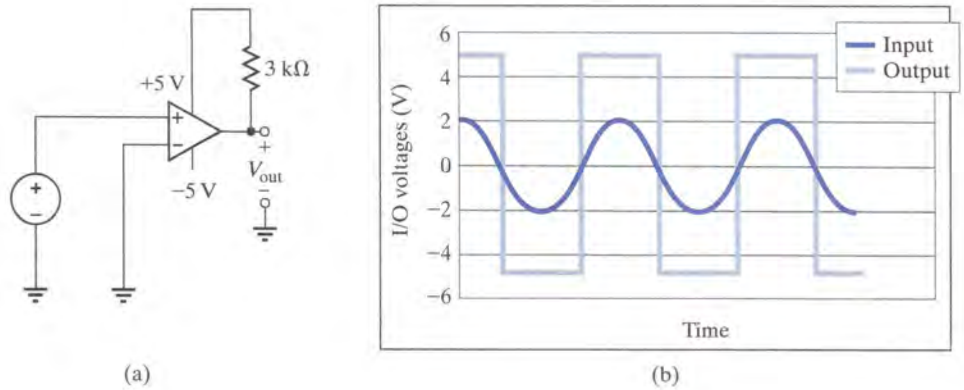


Figure 3.35
(a) A zero-crossing detector and (b) the corresponding input/output waveforms.

Learning by Application

At this point, we have a new element, the op-amp, which we can effectively employ in both applications and circuit design. This device is an extremely useful element that vastly expands our capability in these areas. Because of its ubiquitous nature, the addition of the op-amp to our repertoire of circuit elements permits us to deal with a wide spectrum of practical circuits. Thus, we will employ it here, and also use it throughout this text.

LEARNING Example 3.18

The circuit in Fig. 3.36 is an electronic ammeter. It operates as follows: The unknown current, I through R_I produces a voltage, V_I. V_I is amplified by the op-amp to produce a voltage, V_o, which is proportional to I. The output voltage is measured with a simple voltmeter. We want to find the value of R_2 such that 10 V appears at V_o for each milliamp of unknown current.

SOLUTION Since the current into the op-amp + terminal is zero, the relationship between V_I and I is

$$V_I = IR_I$$

The relationship between the input and output voltages is

$$V_o = V_I\left(1 + \frac{R_2}{R_1}\right)$$

or, solving the equation for V_o/I, we obtain

$$\frac{V_o}{I} = R_I\left(1 + \frac{R_2}{R_1}\right)$$

Using the required ratio V_o/I of 10^4 and resistor values from Fig. 3.36, we can find that

$$R_2 = 9\ \text{k}\Omega$$

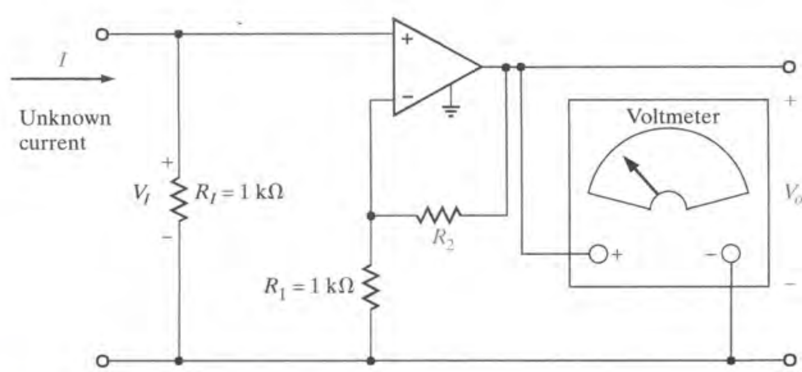


Figure 3.36
Electronic ammeter.

Learning by Design

LEARNING Example 3.19

A typical stereo system is shown in block form in Fig. 3.37a. The phonograph output signal is only about 2 μV. The standard input voltage for stereo power amplifiers is about 2 mV. Therefore, the phonograph signal must be amplified by a factor of 1000 before it reaches the poweramp. To accomplish this, a special-purpose amp, called the phono preamp, is used. Stereo manufacturers place them within the preamplifier cabinet, as shown in Fig. 3.37a.

Let us design a phono preamp using an ideal op-amp that has an input resistance of at least 1 MΩ and a voltage gain of 1000. All resistors must be less than 10 MΩ to limit noise.

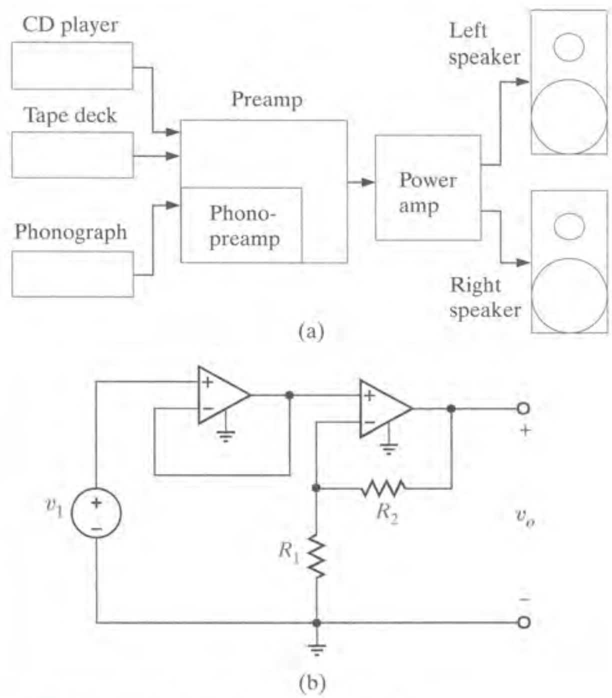


Figure 3.37 Multistage phonograph amplifier.

SOLUTION One possible network is shown in Fig. 3.37b. The input resistance requirement can be easily met with a voltage follower as the first stage of the amplifier. The second stage, or gain stage, can be a noninverting op-amp configuration. We will show in later chapters that the overall voltage gain is the product of the gains of the two stages,

$$A_V = A_{V1}A_{V2} = (1)(1 + (R_2/R_1))$$

To achieve a gain of 1000, we select $R_1 = 1\text{ k}\Omega$ and $R_2 = 999\text{ k}\Omega$.

LEARNING Example 3.20

Let us design a temperature sensor that operates from a 3-V supply, and has a visual display consisting of five LEDs—that is, light-emitting diodes. Only one LED should be on at any time, indicating one of the following temperature ranges: less than 65, 65 to 70, 70 to 75, 75 to 80, and greater than 80°F.

SOLUTION In our proposed sensor, shown in Fig. 3.38, resistor R_X and a thermistor, R_T (a temperature sensitive resistor), form a voltage divider to produce the voltage V_T.

$$V_T = 3\left[\frac{R_X}{R_T + R_X}\right] \quad \textbf{3.6}$$

In the temperature range of interest, a curve fit to a particular commercial thermistor's *R-T* data yields

$$R_T = 57.45e^{-0.0227T} \quad \textbf{3.7}$$

with R_T in kΩ and T in degrees Fahrenheit. From Eqs. (3.6) and (3.7), we see that increasing temperature causes R_T to decrease and V_T to increase. The voltage V_T appears at the unity gain buffer output, where it is divided between R_1, R_2, R_3, and R_4, yielding intermediate voltages V_2, V_3, and V_4. All four comparators and the dc voltage reference, V_{ref}, are contained in the MAX919 package listed in Table 3.4. In this package, $V_{ref} = 1.245$ V.

Based on fundamental comparator operation, when $V_T < V_{ref}$, the output voltage of comparator C_1 will be low, near zero volts. Since $V_4 < V_3 < V_2 < V_T$, all the other comparator output voltages will be low as well. Thus, there is no voltage difference across LED_2, LED_3, LED_4, or LED_5, and these LEDs are off. However, the voltage across LED_1 is not zero. Current will flow from the 3-V supply, through R_{LED} and LED_1, into the output terminal of C_1, through its on-chip circuitry, out of the MAX 919 ground pin and back to the 3-V source, turning on LED_1. This is the desired display for $T < 65$°F.

Then, at exactly 65°F, LED_1 should turn off and LED_2 should turn on. This requires the output of C_1 to go high, near 3 V, while all other comparator outputs remain low. Now, only LED_2 has a nonzero voltage across it. This can be seen in the plots in Fig. 3.39.

Both display scenarios (<65° and 65°-70°) will occur properly if $V_T = V_{\text{ref}}$ precisely at $T = 65°\text{F}$. Thus, we must find R_X such that $V_T = V_{\text{ref}}$ at 65°F. From Eqs. (3.6) and (3.7),

$$1.245 = 3\left[\frac{R_X}{57.45e^{-0.0227(65)} + R_X}\right]$$

which yields $R_X = 9.32\text{ k}\Omega$. This is the case until T reaches 70°F, where the output of C_2 must go high, turning off LED_2 and turning on LED_3. Thus, we require $V_2 = V_{\text{ref}}$ at exactly $T = 70°\text{F}$. Now, $V_4 < V_3 < V_2 = V_{\text{ref}}$ and $V_T > V_{\text{ref}}$. Repeating this idea at $T = 75$ and 80°F yields the voltage division equations

$$V_2|_{T=70} = 1.245 = \left[\frac{R_2 + R_3 + R_4}{R_\Sigma}\right]V_T|_{T=70}$$

$$V_3|_{T=75} = 1.245 = \left[\frac{R_3 + R_4}{R_\Sigma}\right]V_T|_{T=75}$$

$$V_4|_{T=80} = 1.245 = \left[\frac{R_4}{R_\Sigma}\right]V_T|_{T=80}$$

where $R_\Sigma = R_1 + R_2 + R_3 + R_4$. Arbitrarily selecting R_Σ to be 100 kΩ and using Eqs. (3.6) and (3.7), the required resistor values are $R_4 = 83.11\text{ k}\Omega$, $R_3 = 5.01\text{ k}\Omega$, $R_2 = 5.60\text{ k}\Omega$, and $R_1 = 6.28\text{ k}\Omega$.

The LEDs employed in this design have a voltage of 2 V when on and a desired current of 1 mA. Consider again the case where LED_1 is on; that is, the output of comparator C_1 is zero. By KVL, we have

$$3 = I_{\text{LED}}R_{\text{LED}} + V_{\text{LED}} = (0.001)R_{\text{LED}} + 2.0$$

yielding $R_{\text{LED}} = 1\text{ k}\Omega$. Since only one LED is on at any time, there is exactly one resistor, R_{LED}, for every possible display scenario.

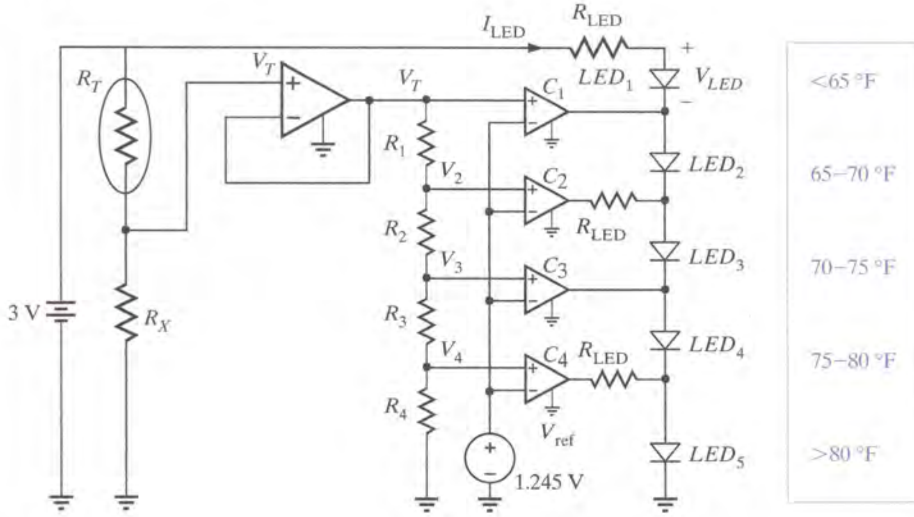


Figure 3.38
The temperature sensor schematic diagram.

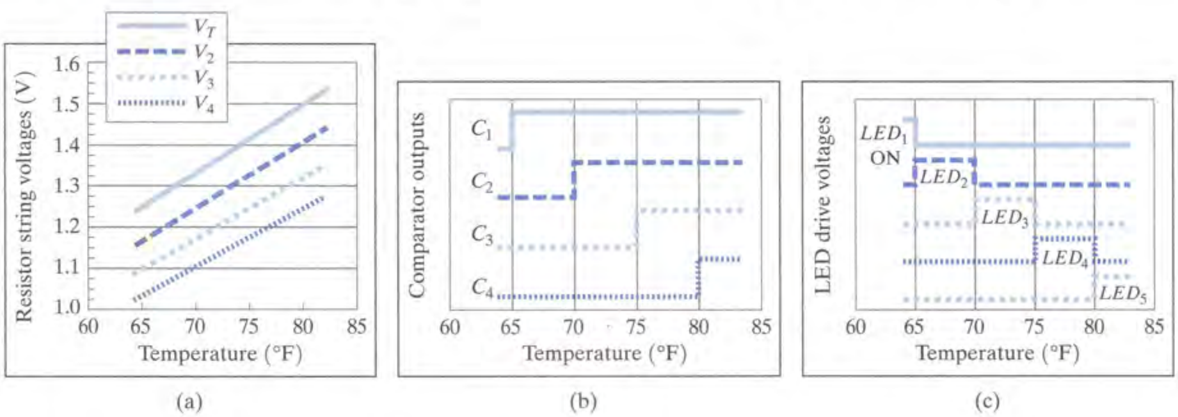


Figure 3.39
Critical voltages in the temperature sensor of Fig. 3.38. Resistor string voltages (a) are compared to V_{ref} to produce the comparator output voltages in (b). Resistors R_X and R_1–R_4 have been selected such that the comparator outputs change exactly at the desired temperature boundaries. The LEDs drive voltages (c) are such that only one LED is on at any time.

LEARNING Check

Summary

- ***Nodal analysis for an N-node circuit***
- Select one node in the N-node circuit as the reference node. Assume that the node voltage is zero and measure all node voltages with respect to this node.
- If only independent current sources are present in the network, write the KCL equations at the $N - 1$ nonreference nodes. If dependent current sources are present, write the KCL equations as is done for networks with only independent current sources; then write the controlling equations for the dependent sources.
- If voltage sources are present in the network, they may be connected (1) between the reference node and a nonreference node or (2) between two nonreference nodes. In the former case, if the voltage source is an independent source, then the voltage at one of the nonreference nodes is known. If the source is dependent, it is treated as an independent source when writing the KCL equations, but an additional constraint equation is necessary.

  In the latter case, if the source is independent, the voltage between the two nodes is constrained by the value of the voltage source and an equation describing this constraint represents one of the $N - 1$ linearly independent equations required to determine the N-node voltages. The surface of the network described by the constraint equation (i.e., the source and two connecting nodes) is called a supernode. One of the remaining $N - 1$ linearly independent equations is obtained by applying KCL at this supernode. If the voltage source is dependent, it is treated as an independent source when writing the KCL equations, but an additional constraint equation is necessary.

- ***Loop analysis for an N-loop circuit***
- One loop current is assigned to each independent loop in a circuit that contains N independent loops.
- If only independent voltage sources are present in the network, write the N linearly independent KVL equations, one for each loop. If dependent voltage sources are present, write the KVL equations as is done for circuits with only independent voltage sources; then write the controlling equations for the dependent sources.
- If current sources are present in the network, either of two techniques can be used. In the first case, one loop current is selected to pass through one of the current sources. This is done for each current source in the network. The remaining loop currents ($N -$ the number of current sources) are determined by open-circuiting the current sources in the network and using this modified network to select them. Once all these currents are defined in the original network, the N-loop equations can be written. The second approach is similar to the first with the exception that if two mesh currents pass through a particular current source, a supermesh is formed around this source. The two required equations for the meshes containing this source are the constraint equations for the two mesh currents that pass through the source and the supermesh equation. If dependent current sources are present, the controlling equations for these sources are also necessary.
- ***Ideal op-amp model*** For an ideal op-amp, $i_+ = i_- = 0$ and $v_+ = v_-$. Both nodal and loop analysis are useful in solving circuits containing operational amplifiers.

Problems

For solutions and additional help on problems marked with ► go to www.wiley.com/college/irwin

SECTION 3.1

3.1 Find I_o in the circuit in Fig. P3.1 using nodal analysis. ►

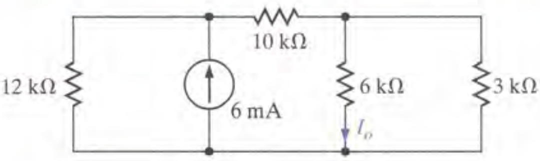


Figure P3.1

3.2 Find I_o in the circuit in Fig. P3.2 using nodal analysis.

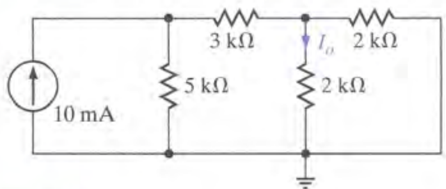


Figure P3.2

3.3 Find V_2 in the circuit in Fig. P3.3 using nodal analysis.

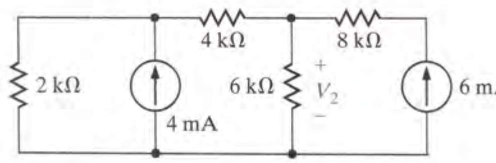


Figure P3.3

3.4 Use nodal analysis to find V_o in the circuit in Fig. P3.4.

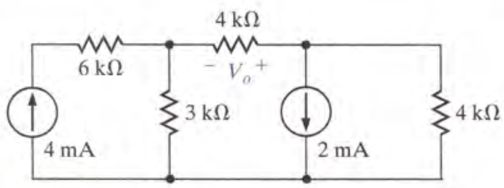


Figure P3.4

3.5 Find I_o in the circuit in Fig. P3.5 using nodal analysis.

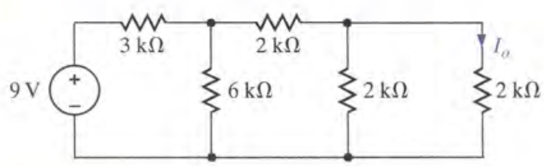


Figure P3.5

3.6 Use nodal analysis to find both V_1 and V_o in the circuit in Fig. P3.6.

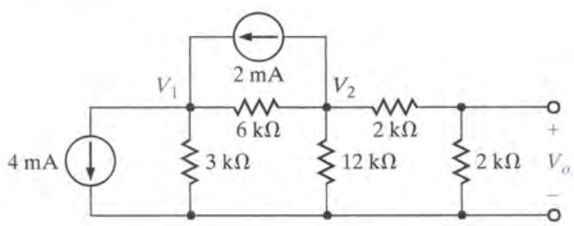


Figure P3.6

3.7 Find V_1 and V_2 in the circuit in Fig. P3.7 using nodal analysis. Then solve the problem using MATLAB and compare your answers.

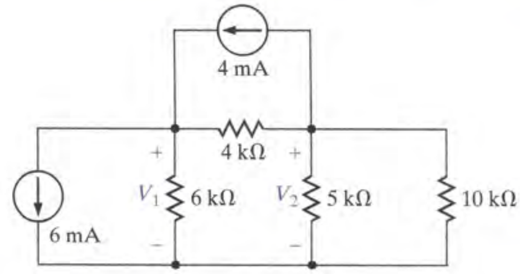


Figure P3.7

3.8 Find I_o in the network in Fig. P3.8 using nodal analysis.

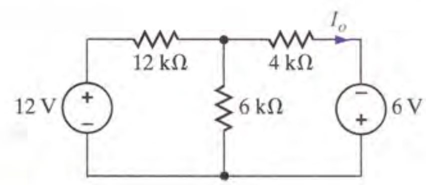


Figure P3.8

3.9 Find V_o in the network in Fig. P3.9 using nodal analysis.

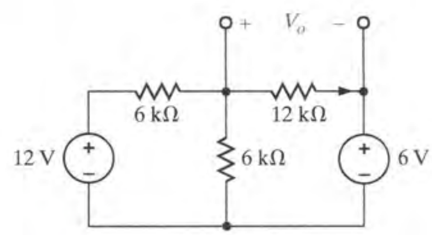


Figure P3.9

3.10 Use nodal analysis to find I_o and I_1 in the network in Fig. P3.10.

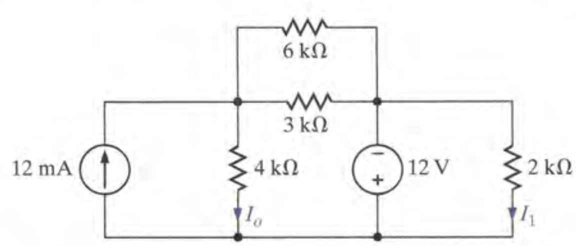


Figure P3.10

3.11 Find I_o in the circuit in Fig. P3.11 using nodal analysis.

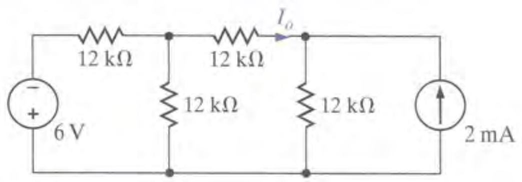


Figure P3.11

3.12 Use nodal analysis to find V_o in the network in Fig. P3.12. Then solve the problem using MATLAB and compare your answers.

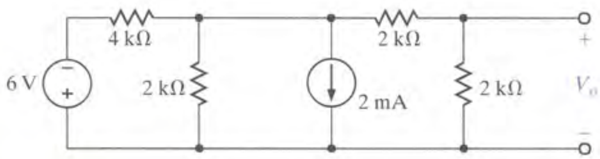


Figure P3.12

3.13 Find I_o in the network in Fig. P3.13.

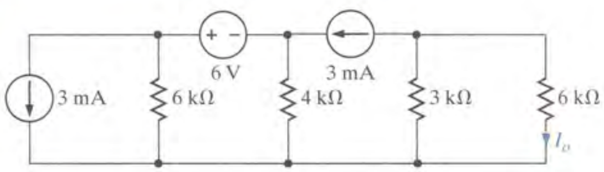


Figure P3.13

3.14 Find V_o in the network in Fig. P3.14.

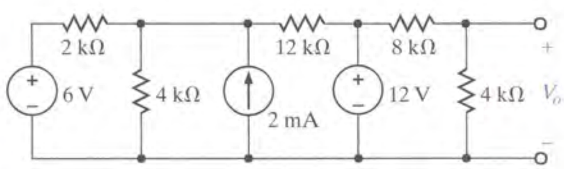


Figure P3.14

3.15 Use nodal analysis to find V_o in the circuit in Fig. P3.15.

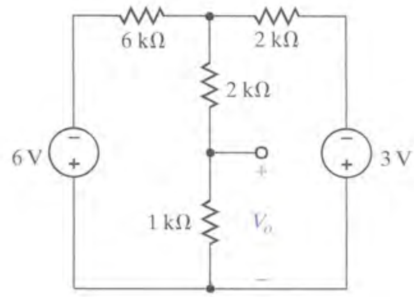


Figure P3.15

3.16 Write the node equations for the circuit in Fig. P3.16 in matrix form, and find all the node voltages using MATLAB.

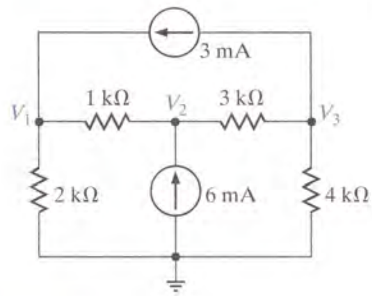


Figure P3.16

3.17 Find I_1 in the network in Fig. P3.17.

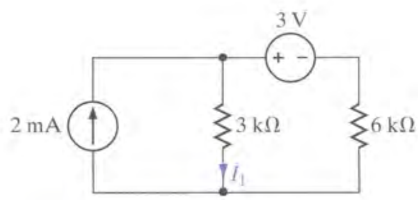


Figure P3.17

3.18 Find I_o in the network in Fig. P3.18.

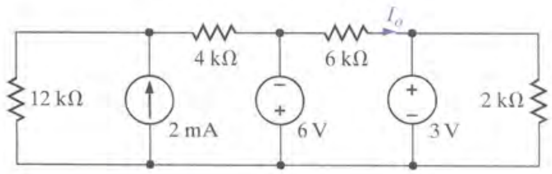


Figure P3.18

3.19 Find I_o in the circuit in Fig. P3.19.

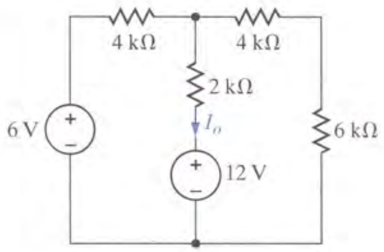


Figure P3.19

3.20 Find I_o in the network in Fig. P3.20 using nodal analysis.

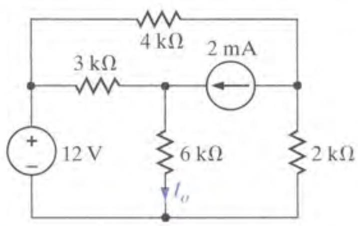


Figure P3.20

3.21 **Use nodal analysis to find V_o in the network in Fig. P3.21. Then solve this problem using MATLAB and compare your answers.**

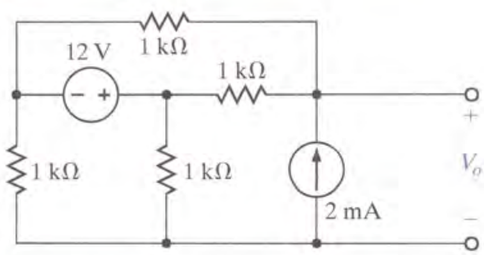


Figure P3.21

3.22 Find V_o in the circuit in Fig. P3.22 using nodal analysis.

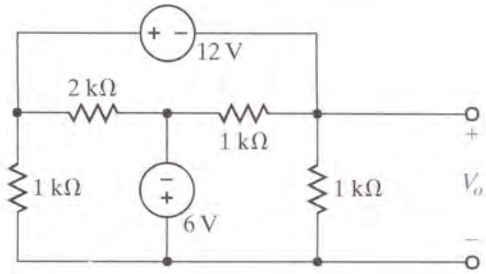


Figure P3.22

3.23 Use nodal analysis to find I_o in the network in Fig. P3.23.

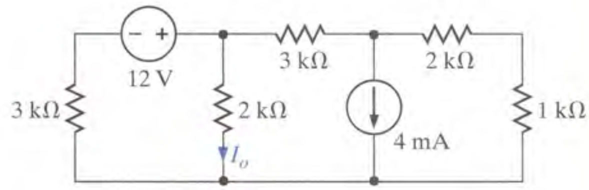


Figure P3.23

3.24 Find I_o in the network in Fig. P3.24 using nodal analysis.

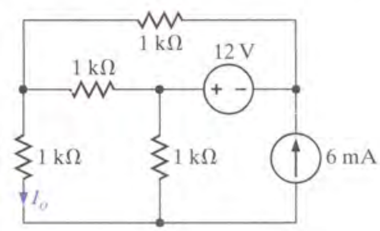


Figure P3.24

3.25 Use nodal analysis to find V_o in the circuit in Fig. P3.25. ►

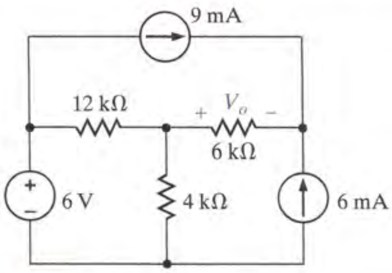


Figure P3.25

3.26 Find V_o in the circuit in Fig. P3.26 using nodal analysis.

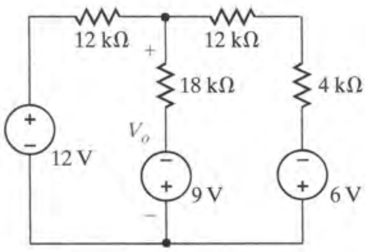


Figure P3.26

3.27 Find V_o in the network in Fig. P3.27 using nodal analysis.

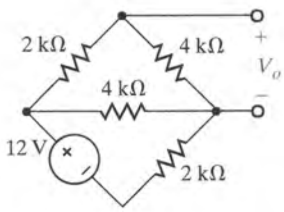


Figure P3.27

3.28 Find V_o in the network in Fig. P3.28 using nodal analysis.

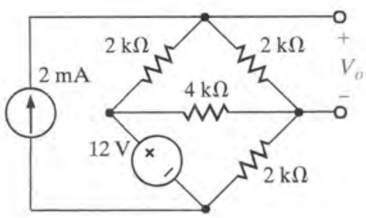


Figure P3.28

3.29 Find V_o in the network in Fig. P3.29 using nodal analysis. ►

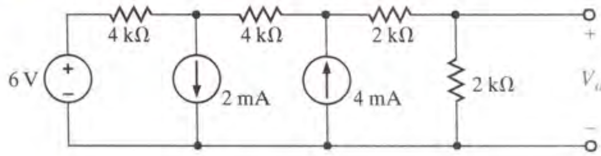


Figure P3.29

3.30 Find I_o in the circuit in Fig. P3.30 using nodal analysis.

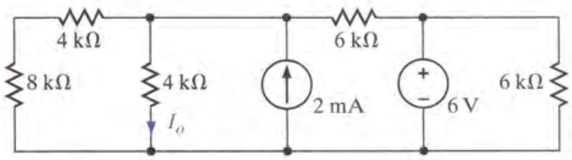


Figure P3.30

3.31 Find V_o in the circuit in Fig. P3.31 using nodal analysis.

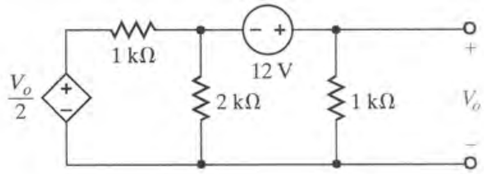


Figure P3.31

3.32 Use nodal analysis to find V_o in the network in Fig. P3.32.

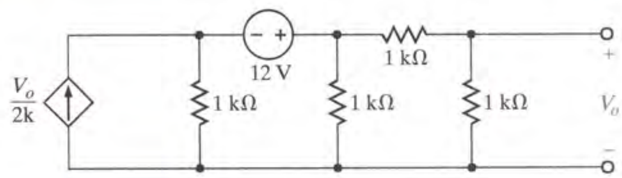


Figure P3.32

3.33 ► **Find V_o in the circuit in Fig. P3.33 using nodal analysis. Then solve the problem using MATLAB and compare your answers.**

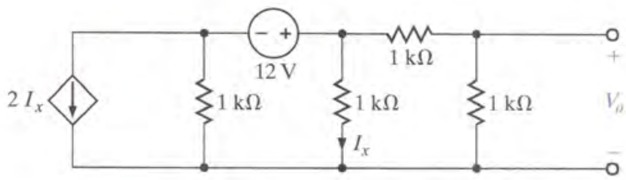


Figure P3.33

3.34 Find I_o in the network in Fig. P3.34.

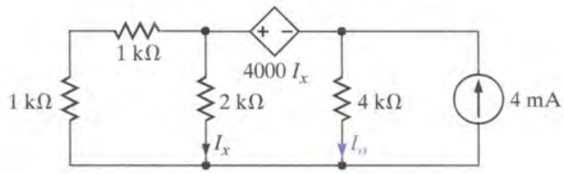


Figure P3.34

3.35 Find V_o in the circuit in Fig. P3.35 using nodal analysis.

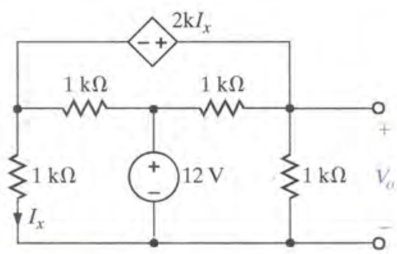


Figure P3.35

3.36 Find I_o in the circuit in Fig. P3.36 using nodal analysis.

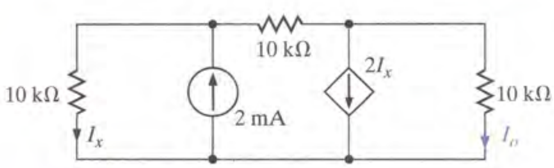


Figure P3.36

3.37 ► **Find V_o in the network in Fig. P3.37 using nodal analysis.**

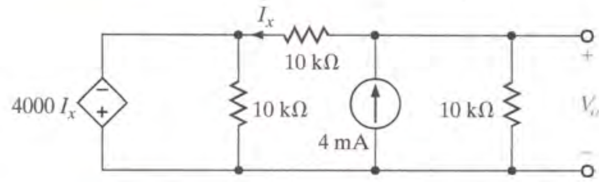


Figure P3.37

3.38 Use nodal analysis to find V_o in the network in Fig. P3.38. In addition, determine all branch currents and check KCL at every node.

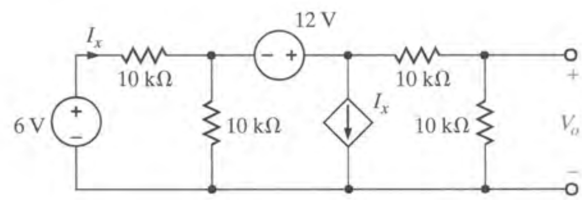


Figure P3.38

3.39 Use nodal analysis to find V_o in the circuit in Fig. P3.39.

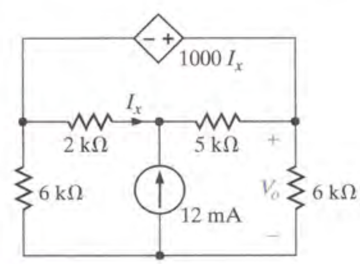


Figure P3.39

3.40 Use nodal analysis to find V_o in the circuit in Fig. P3.40.

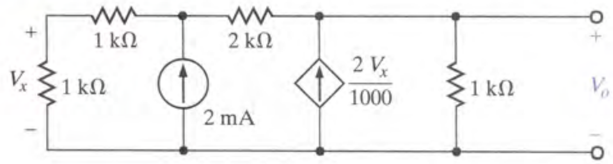


Figure. P3.40

3.41 ► **Use MATLAB to find the node voltages in the network in Fig. P3.41.**

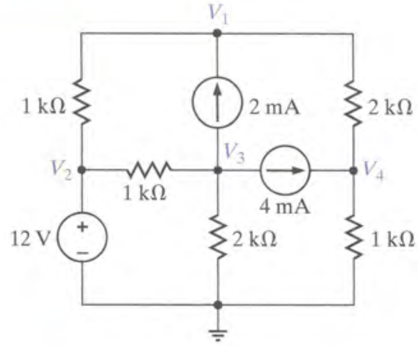


Figure P3.41

SECTION 3.2

3.42 Use mesh equations to find V_o in the circuit in Fig. P3.42.

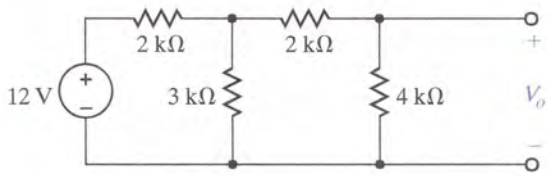


Figure P3.42

3.43 Find V_o in the network in Fig. P3.43 using mesh equations.

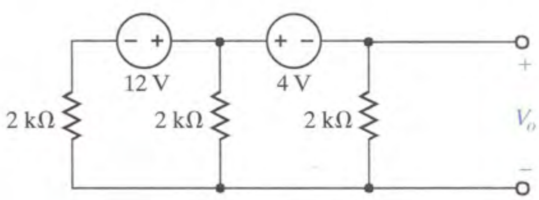


Figure P3.43

3.44 Use mesh analysis to find V_o in the circuit in Fig. P3.44.

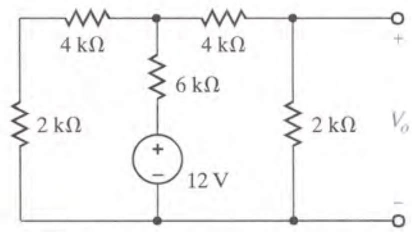


Figure P3.44

3.45 ► **Use mesh analysis to find V_o in the network in Fig. P3.45.**

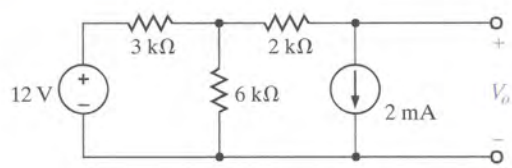


Figure P3.45

3.46 Use loop analysis to find V_o in the circuit in Fig. P3.46.

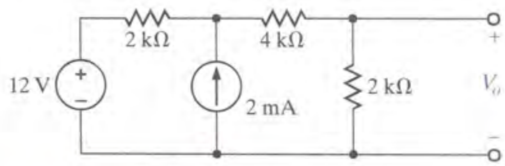


Figure P3.46

3.47 Use loop analysis to find I_o in the circuit in Fig. P3.47.

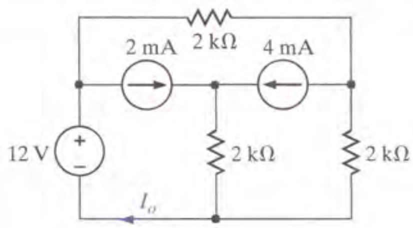


Figure P3.47

3.48 Use both nodal analysis and mesh analysis to find I_o in the circuit in Fig. P3.48.

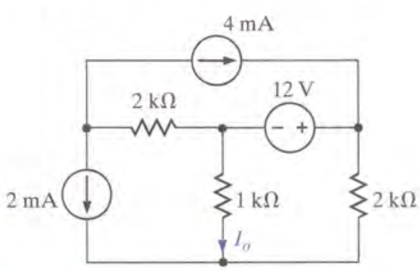


Figure P3.48

3.49 Find I_o in the network in Fig. P3.49 using mesh analysis.

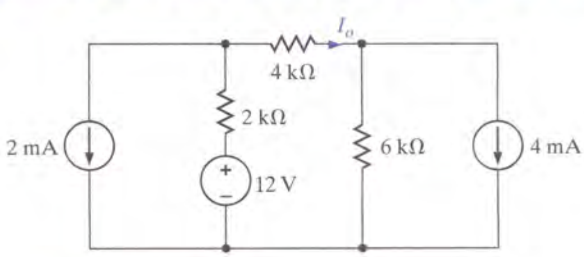


Figure P3.49

3.50 Find I_o in the network in Fig. P3.50 using mesh analysis.

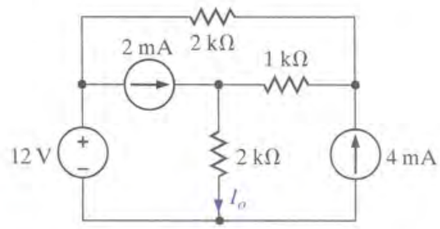


Figure P3.50

3.51 Find V_o in the circuit in Fig. P3.51 using mesh analysis.

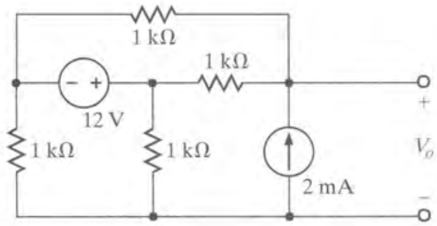


Figure P3.51

3.52 Use loop analysis to find V_o in the network in Fig. P3.52.

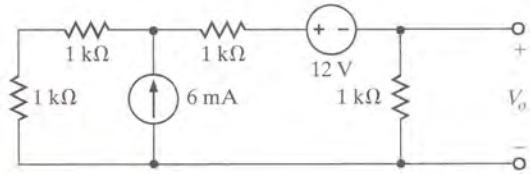


Figure P3.52

3.53 Find I_o in the network in Fig. P3.53 using loop analysis. Then solve the problem using MATLAB and compare your answers.

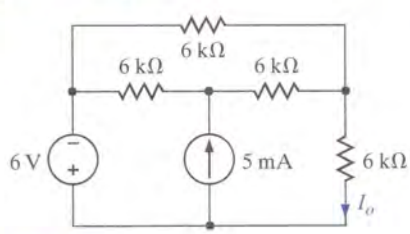


Figure P3.53

3.54 Use loop analysis to find I_o in the circuit in Fig. P3.54.

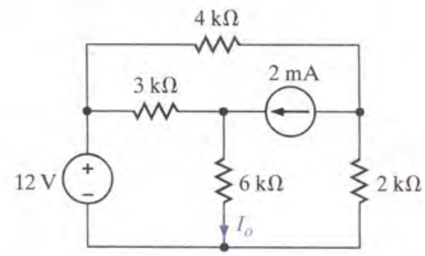


Figure P3.54

3.55 Find V_o in the network in Fig. P3.55 using both mesh and nodal analyses.

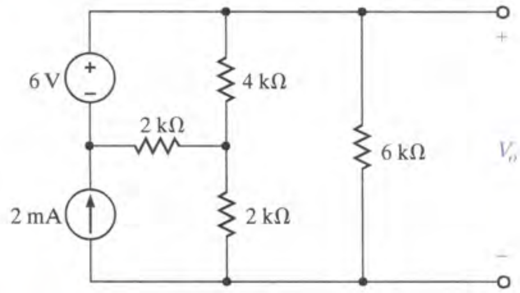


Figure P3.55

3.56 Use loop analysis to find V_o in the circuit in Fig. P3.56.

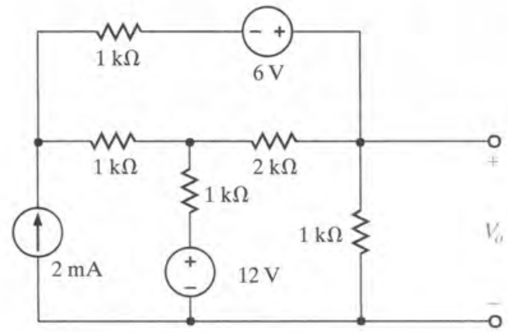


Figure P3.56

3.57 **Use loop analysis to find I_o in the network in Fig. P3.57.** ►

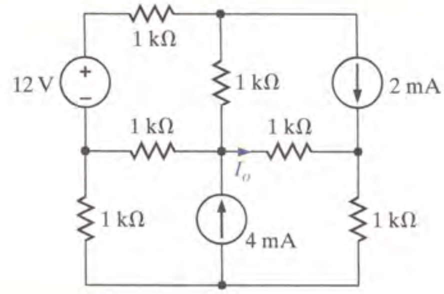


Figure P3.57

3.58 Use loop analysis to find V_o in the network in Fig. P3.58.

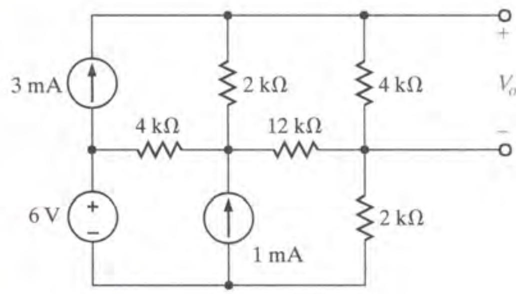


Figure P3.58

3.59 Find V_o in the network in Fig. P3.59.

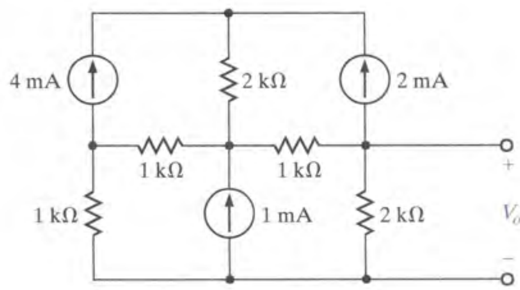


Figure P3.59

3.60 Find V_o in the circuit in Fig. P3.60.

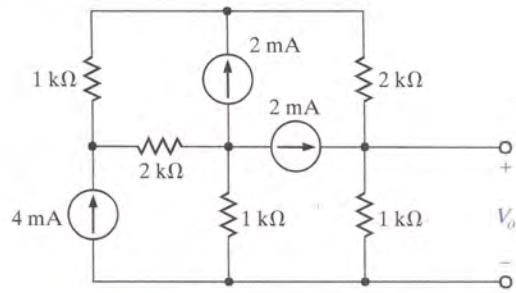


Figure P3.60

3.61 Find I_o in the circuit in Fig. P3.61.

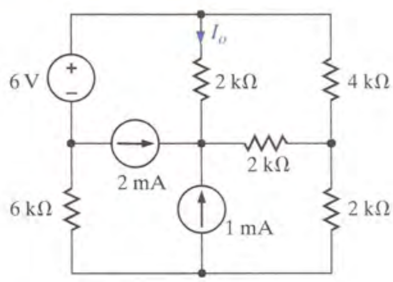


Figure P3.61

3.62 Use loop analysis to find V_o in the network in Fig. P3.62.

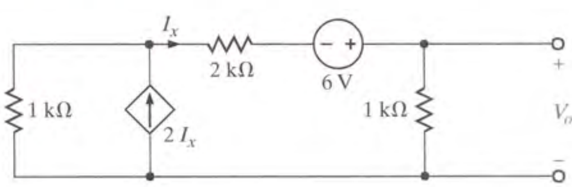


Figure P3.62

3.63 Use mesh analysis to find V_o in the circuit in Fig. P3.63.

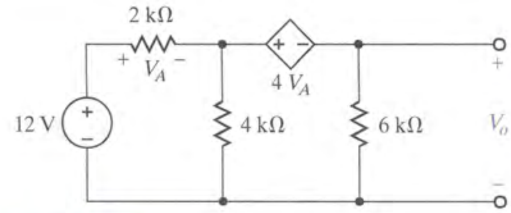


Figure P3.63

3.64 Use loop analysis to find V_o in the circuit in Fig. P3.64.

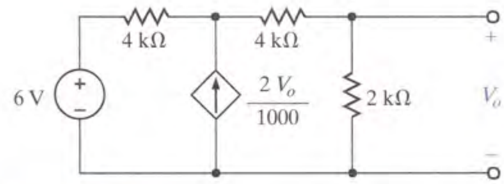


Figure P3.64

3.65 Find V_o in the circuit in Fig. P3.65 using mesh analysis.

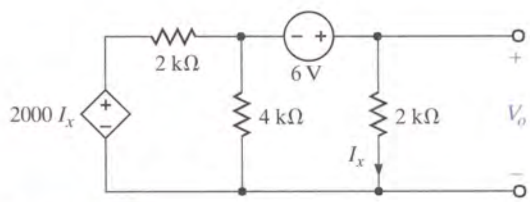


Figure P3.65

3.66 Use both nodal analysis and mesh analysis to find V_o in the circuit in Fig. P3.66.

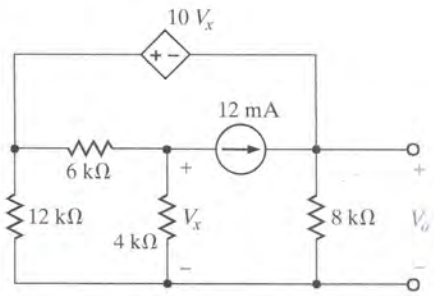


Figure P3.66

3.67 Using mesh analysis, find V_o in the circuit in Fig. P3.67.

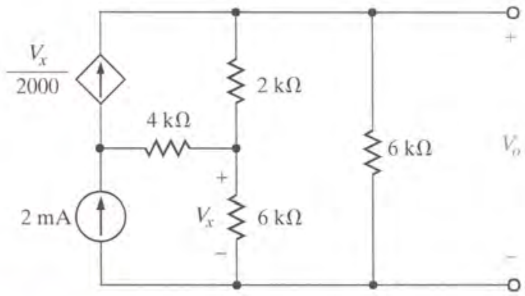


Figure P3.67

3.68 Find V_o in the network in Fig. P3.68.

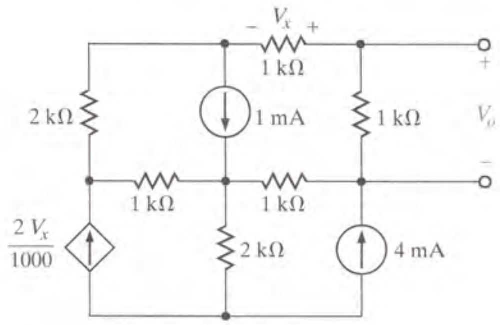


Figure P3.68

3.69 **Use MATLAB to find the mesh currents in the network in Fig. P3.69.**

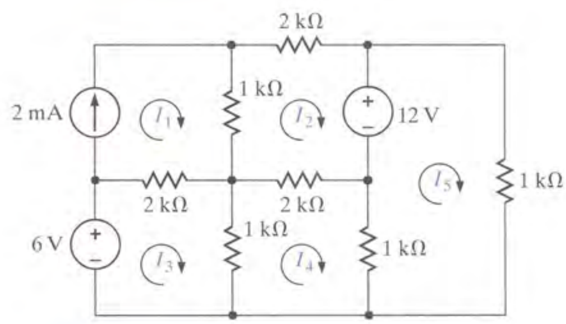


Figure P3.69

SECTION 3.3

Assume that all op-amps in this section are ideal.

3.70 Find V_o in the circuit in Fig. P3.70.

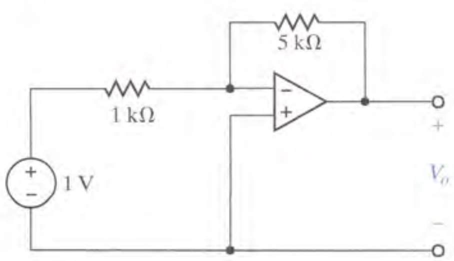


Figure P3.70

3.71 Find V_o in the network in Fig. P3.71 and explain what effect R_1 has on the output.

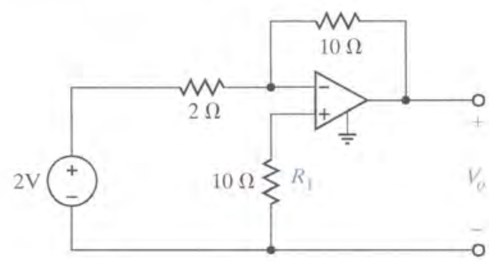


Figure P3.71

3.72 Find V_o in the network in Fig. P3.72.

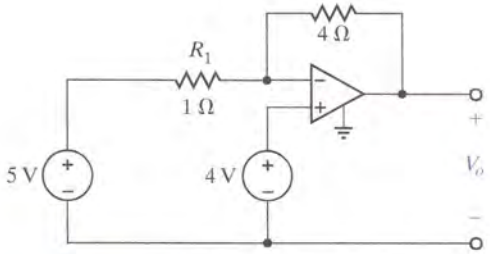


Figure P3.72

3.73 The network in Fig. P3.73 is a current-to-voltage converter or transresistance amplifier. Find v_o/i_S for this network.

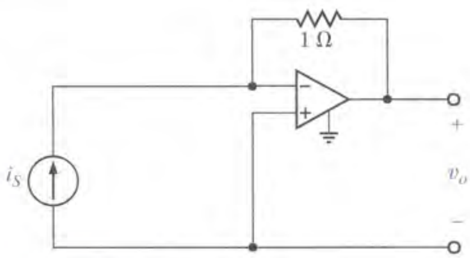


Figure P3.73

3.74 Find V_o in the circuit in Fig. P3.74.

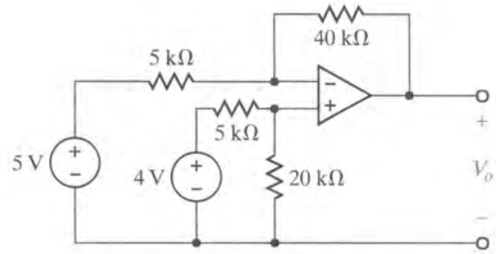


Figure P3.74

3.75 Find V_o in the circuit in Fig. P3.75.

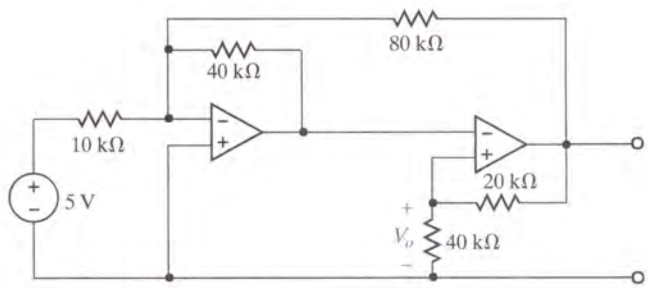


Figure P3.75

Typical Problems Found on the FE Exam

3FE-1 **Find V_o in the circuit in Fig. 3PFE-1.**

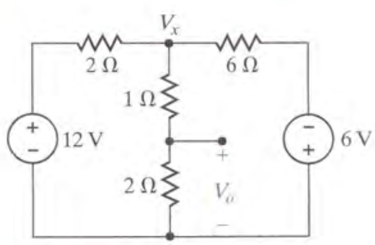


Figure 3PFE-1

3FE-2 Determine the power dissipated in the 6-ohm resistor in the network in Fig. 3PFE-2.

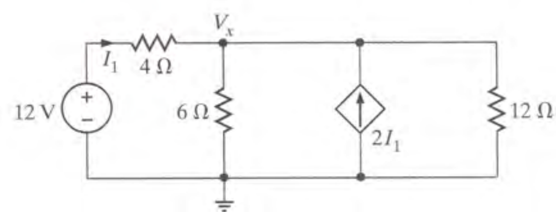


Figure 3PFE-2

3FE-3 ► **Find the current I_x in the 4-ohm resistor in the circuit in Fig. 3PFE-3.**

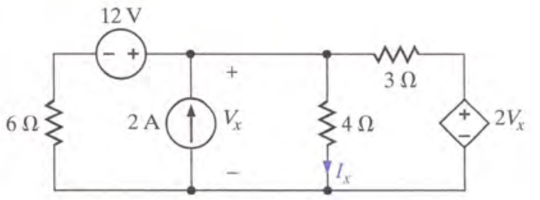


Figure 3PFE-3

3FE-4 Determine the voltage V_o in the circuit in Fig. 3PFE-4.

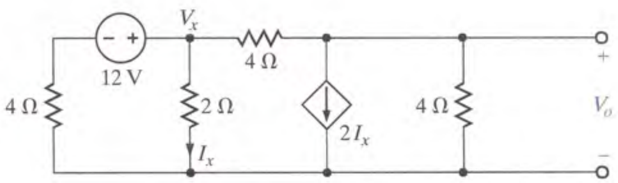


Figure 3PFE-4

3FE-5 Given the summing amplifier shown in Fig. 3PFE-5, select the values of R_2 that will produce an output voltage of −3 V.

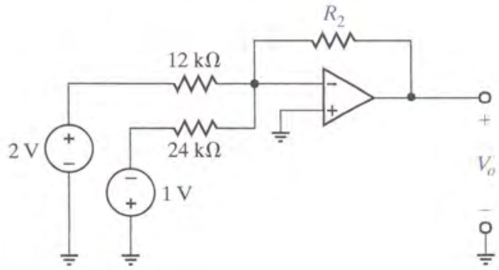


Figure 3PFE-5

3FE-6 Determine the output voltage V_o of the summing op-amp circuit shown in Fig. 3PFE-6.

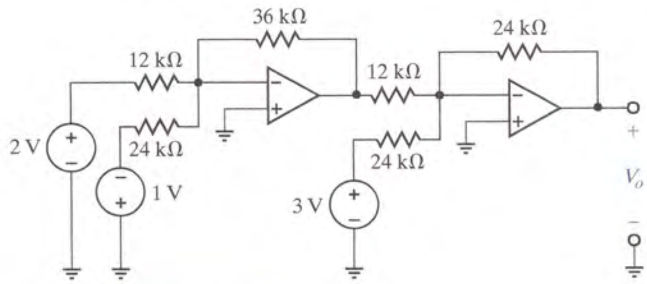


Figure 3PFE-6

Additional Analysis Techniques

4

At this point, we have mastered the ability to solve networks containing both independent and dependent sources using either nodal or loop analysis. In this chapter we introduce several new analysis techniques that bolster our arsenal of circuit analysis tools. We will find that in some situations these techniques lead to a quick solution and in other cases they do not. However, these new techniques in many cases do provide an insight into the circuit's operation that cannot be gained from a nodal or loop analysis.

In many practical situations we are interested in the analysis of some portion of a much larger network. If we can model the remainder of the network with a simple equivalent circuit, then our task will be much simpler. For example, consider the problem of analyzing some simple electronic device that is connected to the ac wall plug in our house. In this case the complete circuit includes not only the electronic device but the utility's power grid, which is connected to the device through the circuit breakers in the home. However, if we can accurately model everything outside the device with a simple equivalent circuit, then our analysis will be tractable. Two of the theorems that we present in this chapter will permit us to do just that.

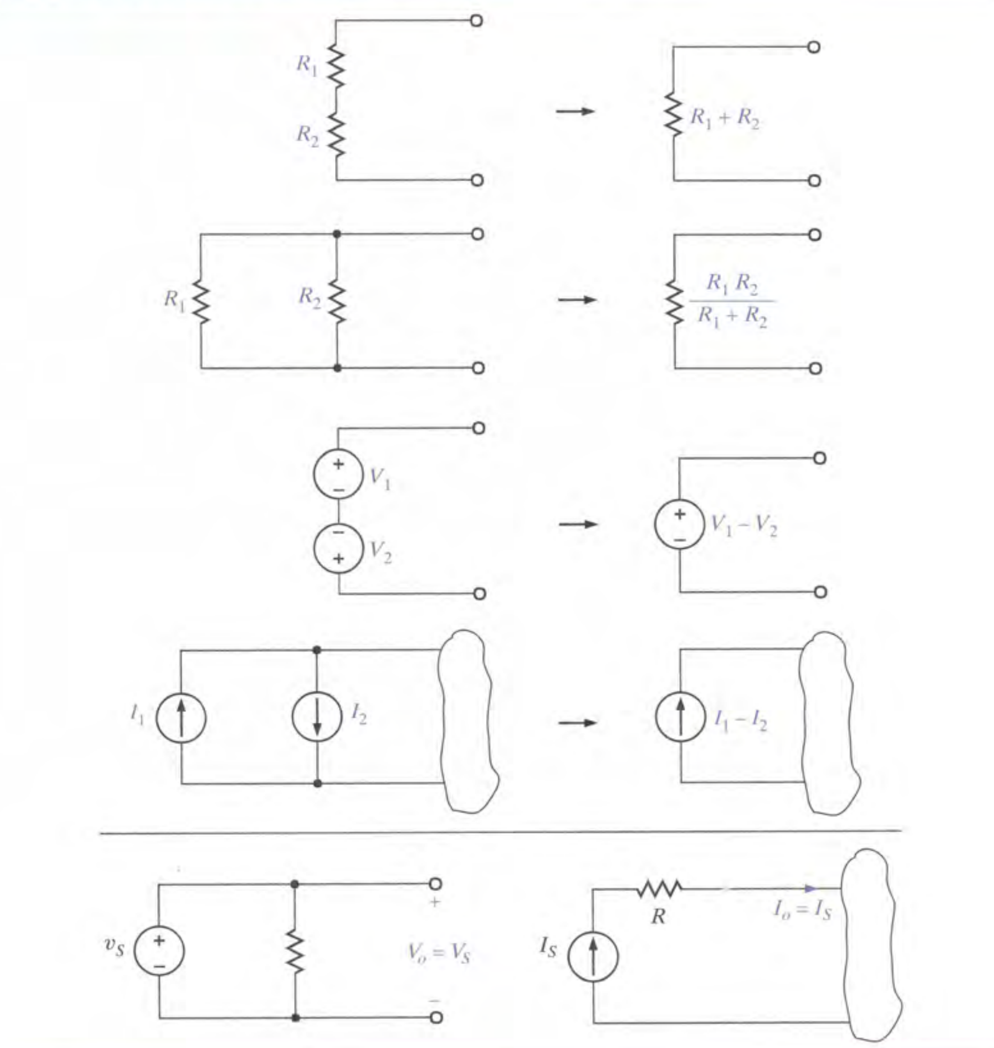
LEARNING Goals

4.1 Introduction

Before introducing additional analysis techniques, let us review some of the topics we have used either explicitly or implicitly in our analyses thus far.

EQUIVALENCE Table 4.1 is a short compendium of some of the equivalent circuits that have been employed in our analyses. This listing serves as a quick review as we begin to look at other techniques that can be used to find a specific voltage or current somewhere in a network and provide additional insight into the network's operation. In addition to the forms listed in the table, it is important to note that a series connection of current sources or a parallel

Table 4.1 Equivalent circuit forms



connection of voltage sources is forbidden unless the sources are pointing in the same direction and have exactly the same values.

LINEARITY All the circuits we have analyzed thus far have been linear circuits. Most of the circuits we will analyze in the remainder of the book will also be linear circuits, and any deviation from this type of network will be specifically identified as such.

Linearity requires both additivity and homogeneity (scaling). It can be shown [see the previous edition of this book] that the circuits that we are examining satisfy this important property. The following example illustrates one way in which this property can be used.

LEARNING Example 4.1

For the circuit shown in Fig. 4.1, we wish to determine the output voltage V_{out}. However, rather than approach the problem in a straightforward manner and calculate I_o, then I_1, then I_2, and so on, we will use linearity and simply assume that the output voltage is $V_{out} = 1$ V. This assumption will yield a value for the source voltage. We will then use the actual value of the source voltage and linearity to compute the actual value of V_{out}.

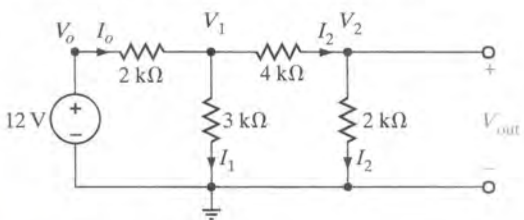


Figure 4.1 Circuit used in Example 4.1.

SOLUTION If we assume that $V_{out} = V_2 = 1$ V, then

$$I_2 = \frac{V_2}{2\text{k}} = 0.5 \text{ mA}$$

V_1 can then be calculated as

$$V_1 = 4\text{k}I_2 + V_2$$
$$= 3 \text{ V}$$

Hence,

$$I_1 = \frac{V_1}{3\text{k}} = 1 \text{ mA}$$

Now, applying KCL,

$$I_o = I_1 + I_2 = 1.5 \text{ mA}$$

Then

$$V_o = 2\text{k}I_o + V_1$$
$$= 6 \text{ V}$$

Therefore, the assumption that $V_{out} = 1$ V produced a source voltage of 6 V. However, since the actual source voltage is 12 V, the actual output voltage is 1 V(12/6) = 2 V.

LEARNING EXTENSION

E4.1 Use linearity and the assumption that $I_o = 1$ mA to compute the correct current I_o in the circuit in Fig. E4.1 if $I = 6$ mA.

ANSWER $I_o = 3$ mA.

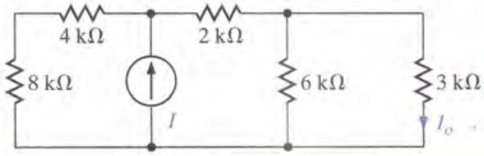


Figure E4.1

4.2 Superposition

To provide motivation for this subject, let us examine a simple circuit in which two sources contribute to the current in the network.

LEARNING Example 4.2

Consider the circuit in Fig. 4.2a in which the actual values of the voltage sources are left unspecified. Let us use this network to examine the concept of superposition.

SOLUTION The mesh equations for this network are

$$6ki_1(t) - 3ki_2(t) = v_1(t)$$
$$-3ki_1(t) + 9ki_2(t) = -v_2(t)$$

Solving these equations for $i_1(t)$ yields

$$i_1(t) = \frac{v_1(t)}{5k} - \frac{v_2(t)}{15k}$$

In other words, the current $i_1(t)$ has a component due to $v_1(t)$ and a component due to $v_2(t)$. In view of the fact that $i_1(t)$ has two components, one due to each independent source, it would be interesting to examine what each source acting alone would contribute to $i_1(t)$. For $v_1(t)$ to act alone, $v_2(t)$ must be zero. As we pointed out in Chapter 2, $v_2(t) = 0$ means that the source $v_2(t)$ is replaced with a short circuit. Therefore, to determine the value of $i_1(t)$ due to $v_1(t)$ only, we employ the circuit in Fig. 4.2b and refer to this value of $i_1(t)$ as $i_1'(t)$.

$$i_1'(t) = \frac{v_1(t)}{3k + \frac{(3k)(6k)}{3k + 6k}} = \frac{v_1(t)}{5k}$$

Let us now determine the value of $i_1(t)$ due to $v_2(t)$ acting alone and refer to this value as $i_1''(t)$. Using the network in Fig. 4.2c,

$$i_2''(t) = -\frac{v_2(t)}{6k + \frac{(3k)(3k)}{3k + 3k}} = \frac{-2v_2(t)}{15k}$$

Then, using current division, we obtain

$$i_1''(t) = \frac{-2v_2(t)}{15k}\left(\frac{3k}{3k + 3k}\right) = \frac{-v_2(t)}{15k}$$

Now, if we add the values of $i_1'(t)$ and $i_1''(t)$, we obtain the value computed directly; that is,

$$i_1(t) = i_1'(t) + i_1''(t) = \frac{v_1(t)}{5k} - \frac{v_2(t)}{15k}$$

Note that we have *superposed* the value of $i_1'(t)$ on $i_1''(t)$, or vice versa, to determine the unknown current.

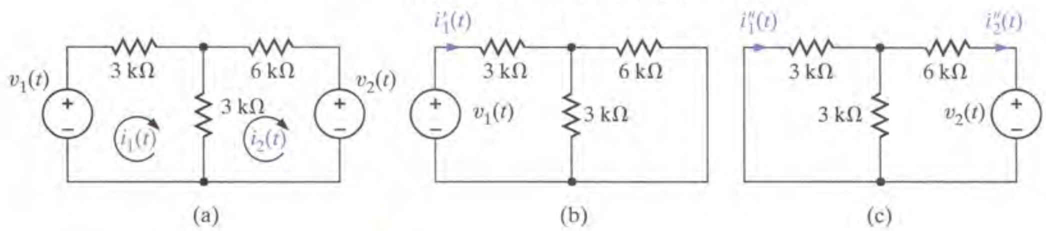


Figure 4.2 *Circuits used to illustrate superposition.*

What we have demonstrated in Example 4.2 is true in general for linear circuits and is a direct result of the property of linearity. *The principle of superposition*, which provides us with this ability to reduce a complicated problem to several easier problems—each containing only a single independent source—states that

> In any linear circuit containing multiple independent sources, the current or voltage at any point in the network may be calculated as the algebraic sum of the individual contributions of each source acting alone.

When determining the contribution due to an independent source, any remaining voltage sources are made zero by replacing them with short circuits, and any remaining current sources are made zero by replacing them with open circuits.

Although superposition can be used in linear networks containing dependent sources, it is not useful in this case since the dependent source is never made zero.

It is interesting to note that, as the previous example indicates, superposition provides some insight in determining the contribution of each source to the variable under investigation.

We will now demonstrate superposition with two examples and then provide a problem-solving strategy for the use of this technique. For purposes of comparison, we will also solve the networks using both node and loop analyses. Furthermore, we will employ these same networks when demonstrating subsequent techniques, if applicable.

LEARNING Example 4.3

Let us use superposition to find V_o in the circuit in Fig. 4.3a.

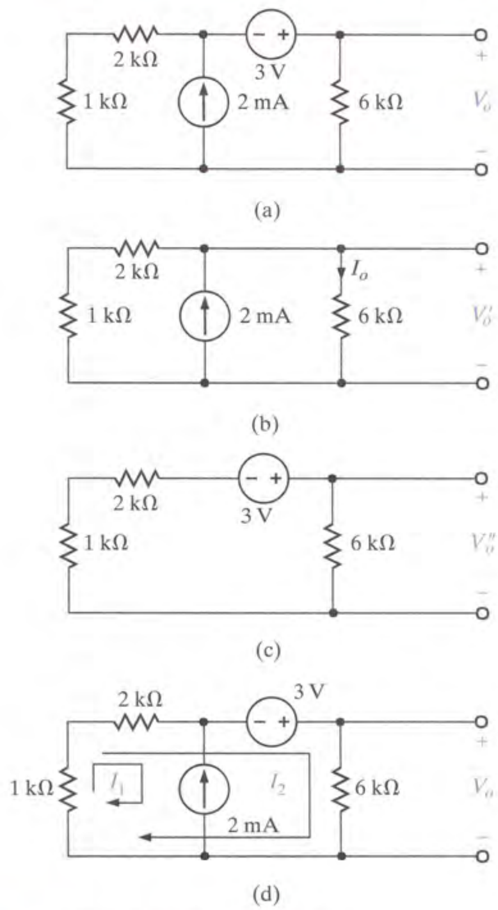


Figure 4.3 Circuits used in Example 4.3.

SOLUTION The contribution of the 2-mA source to the output voltage is found from the network in Fig. 4.3b, using current division

$$I_o = (2 \times 10^{-3})\left(\frac{1\text{k} + 2\text{k}}{1\text{k} + 2\text{k} + 6\text{k}}\right) = \frac{2}{3}\text{ mA}$$

and

$$V_o' = I_o(6\text{k}) = 4\text{ V}$$

The contribution of the 3-V source to the output voltage is found from the circuit in Fig. 4.3c. Using voltage division,

$$V_o'' = 3\left(\frac{6\text{k}}{1\text{k} + 2\text{k} + 6\text{k}}\right)$$

$$= 2\text{ V}$$

Therefore,

$$V_o = V_o' + V_o'' = 6\text{ V}$$

Although we used two separate circuits to solve the problem, both were very simple.

If we use nodal analysis and Fig. 4.3a to find V_o and recognize that the 3-V source and its connecting nodes form a supernode, V_o can be found from the node equation

$$\frac{V_o - 3}{1\text{k} + 2\text{k}} - 2 \times 10^{-3} + \frac{V_o}{6\text{k}} = 0$$

which yields $V_o = 6$ V. In addition, loop analysis applied as shown in Fig. 4.3d produces the equations

$$I_1 = -2 \times 10^{-3}$$

and

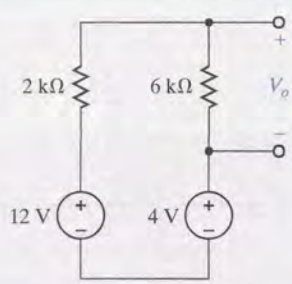
$$3\text{k}(I_1 + I_2) - 3 + 6\text{k}I_2 = 0$$

which yield $I_2 = 1$ mA and hence $V_o = 6$ V.

LEARNING by Doing

D 4.1 Use superposition to find V_o in the following network.

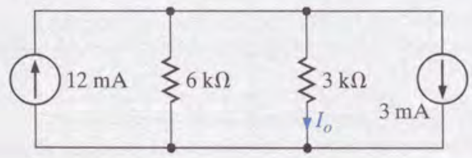
ANSWER $V_o = V_o' + V_o'' = (9 - 3) = 6\text{ V}$



D 4.2 Use superposition to find I_o in the following network.

ANSWER

$I_o = I_o' + I_o'' = (8 - 2) = 6$ mA



LEARNING Example 4.4

Consider now the network in Fig. 4.4a. Let us use superposition to find V_o.

SOLUTION The contribution of the 6-V source to V_o is found from the network in Fig. 4.4b, which is redrawn in Fig. 4.4c.

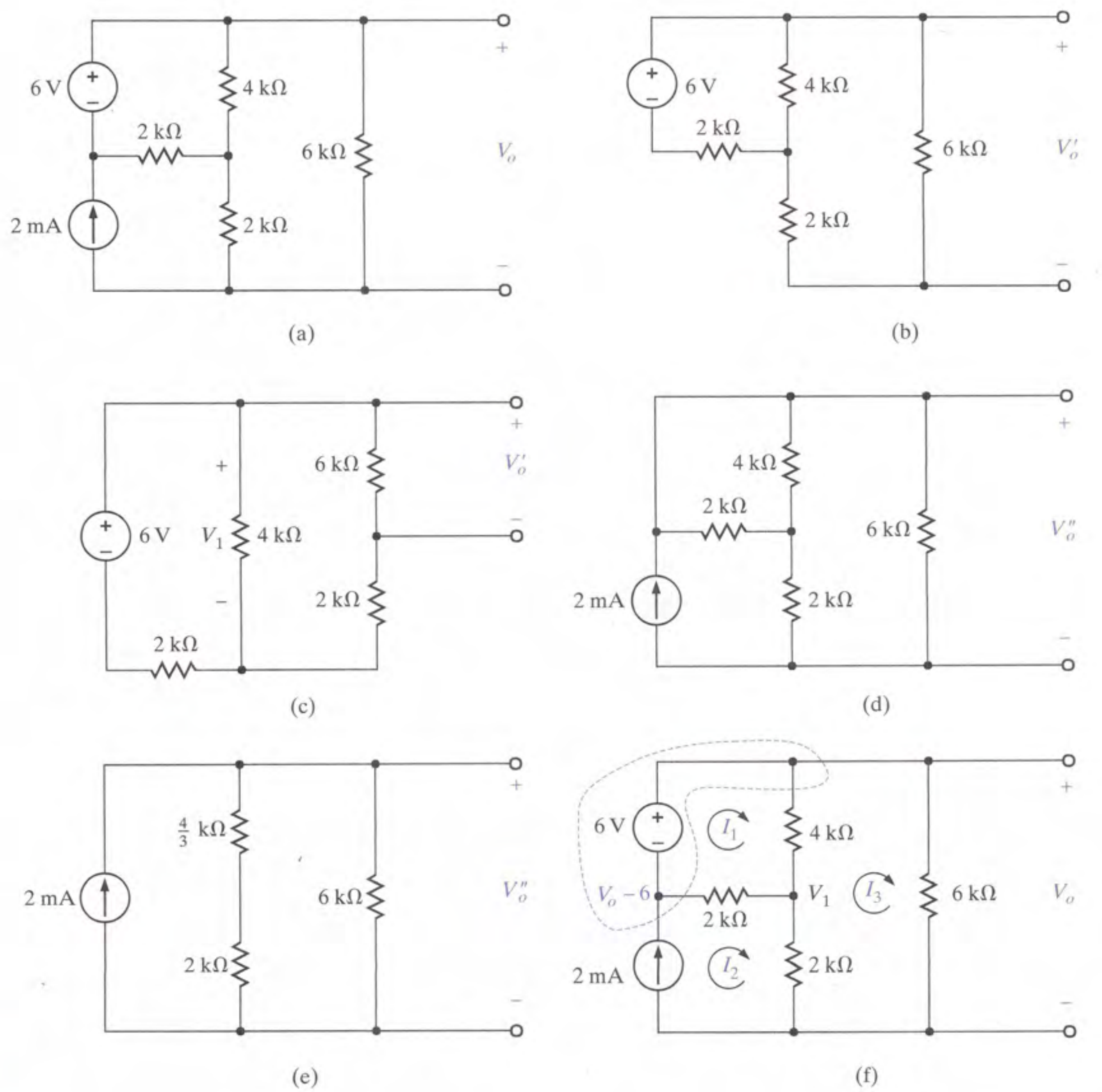


Figure 4.4 Circuits used in Example 4.4.

The $2\text{ k}\Omega + 6\text{ k}\Omega = 8\text{-k}\Omega$ resistor and $4\text{-k}\Omega$ resistor are in parallel, and their combination is an $8/3\text{-k}\Omega$ resistor. Then, using voltage division,

$$V_1 = 6\left(\frac{\frac{8}{3}\text{k}}{\frac{8}{3}\text{k} + 2\text{k}}\right) = \frac{24}{7}\text{ V}$$

Applying voltage division again,

$$V_o' = V_1\left(\frac{6\text{k}}{6\text{k} + 2\text{k}}\right) = \frac{18}{7}\text{ V}$$

The contribution of the 2-mA source is found from Fig. 4.4d, which is redrawn in Fig. 4.4e. V_o'' is simply equal to the product of the current source and the parallel combination of the resistors; that is,

$$V_o'' = (2 \times 10^{-3})\left(\frac{10}{3}\text{k}//6\text{k}\right) = \frac{30}{7}\text{ V}$$

Then

$$V_o = V_o' + V_o'' = \frac{48}{7}\text{ V}$$

A nodal analysis of the network can be performed using Fig. 4.4f. The equation for the supernode is

$$-2 \times 10^{-3} + \frac{(V_o - 6) - V_1}{2\text{k}} + \frac{V_o - V_1}{4\text{k}} + \frac{V_o}{6\text{k}} = 0$$

The equation for the node labeled V_1 is

$$\frac{V_1 - V_o}{4\text{k}} + \frac{V_1 - (V_o - 6)}{2\text{k}} + \frac{V_1}{2\text{k}} = 0$$

Solving these two equations, which already contain the constraint equation for the supernode, yields $V_o = 48/7$ V.

Once again, referring to the network in Fig. 4.4f, the mesh equations for the network are

$$-6 + 4\text{k}(I_1 - I_3) + 2\text{k}(I_1 - I_2) = 0$$
$$I_2 = 2 \times 10^{-3}$$
$$2\text{k}(I_3 - I_2) + 4\text{k}(I_3 - I_1) + 6\text{k}I_3 = 0$$

Solving these equations, we obtain $I_3 = 8/7$ mA and, hence, $V_o = 48/7$ V.

Problem-Solving Strategy
Applying Superposition

- In a network containing multiple independent sources, each source can be applied independently with the remaining sources turned off.
- To turn off a voltage source, replace it with a short circuit, and to turn off a current source, replace it with an open circuit.
- When the individual sources are applied to the circuit, all the circuit laws and techniques we have learned, or will soon learn, can be applied to obtain a solution.
- The results obtained by applying each source independently are then added together algebraically to obtain a solution.

Superposition can be applied to a circuit with any number of dependent and independent sources. In fact, superposition can be applied to such a network in a variety of ways. For example, a circuit with three independent sources can be solved using each source acting alone, as we have just demonstrated, or we could use two at a time and sum the result with that obtained from the third acting alone. In addition, the independent sources do not have to assume their actual value or zero. However, it is mandatory that the sum of the different values chosen add to the total value of the source.

Superposition is a fundamental property of linear equations and, therefore, can be applied to any effect that is linearly related to its cause. In this regard it is important to point out that although superposition applies to the current and voltage in a linear circuit, it cannot be used to determine power because power is a nonlinear function.

LEARNING EXTENSION

E4.2 Compute V_o in the circuit in Fig. E4.2 using superposition.

ANSWER $V_o = \frac{4}{3}$ V.

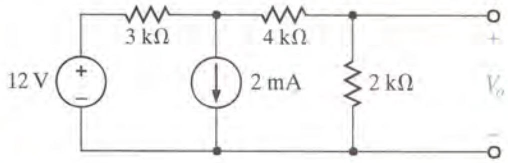


Figure E4.2

4.3 Thévenin's and Norton's Theorems

Thus far we have presented a number of techniques for circuit analysis. At this point we will add two theorems to our collection of tools that will prove to be extremely useful. The theorems are named after their authors, M. L. Thévenin, a French engineer, and E. L. Norton, a scientist formerly with Bell Telephone Laboratories.

Suppose that we are given a circuit and that we wish to find the current, voltage, or power that is delivered to some resistor of the network, which we will call the load. *Thévenin's theorem* tells us that we can replace the entire network, exclusive of the load, by an equivalent circuit that contains only an independent voltage source in series with a resistor in such a way that the current–voltage relationship at the load is unchanged. *Norton's theorem* is identical to the preceding statement except that the equivalent circuit is an independent current source in parallel with a resistor.

Note that this is a very important result. It tells us that if we examine any network from a pair of terminals, we know that with respect to those terminals, the entire network is equivalent to a simple circuit consisting of an independent voltage source in series with a resistor or an independent current source in parallel with a resistor.

In developing the theorems, we will assume that the circuit shown in Fig. 4.5a can be split into two parts, as shown in Fig. 4.5b. In general, circuit *B* is the load and may be linear or nonlinear. Circuit *A* is the balance of the original network exclusive of the load and must be linear. As such, circuit *A* may contain independent sources, dependent sources and resistors, or any other linear element. We require, however, that a dependent source and its control variable appear in the same circuit.

Circuit *A* delivers a current *i* to circuit *B* and produces a voltage v_o across the input terminals of circuit *B*. From the standpoint of the terminal relations of circuit *A*, we can replace circuit *B* by a voltage source of v_o volts (with the proper polarity), as shown in Fig. 4.5c. Since the terminal voltage is unchanged and circuit A is unchanged, the terminal current *i* is unchanged.

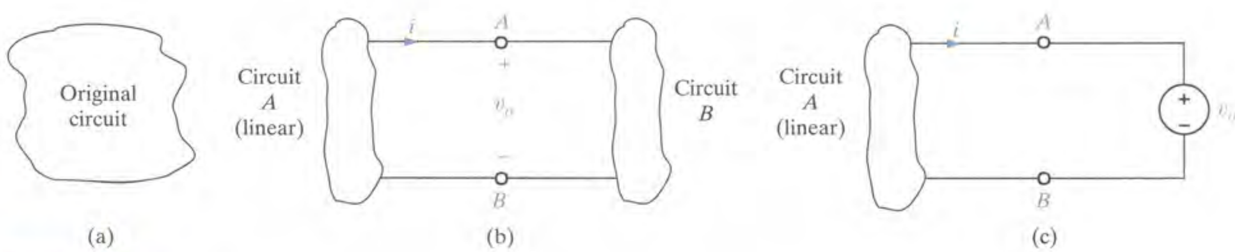


Figure 4.5
Concepts used to develop Thévenin's theorem.

Now, applying the principle of superposition to the network shown in Fig. 4.5c, the total current i shown in the figure is the sum of the currents caused by all the sources in circuit A and the source v_o that we have just added. Therefore, via superposition the current i can be written

$$i = i_o + i_{sc} \tag{4.1}$$

where i_o is the current due to v_o with all independent sources in circuit A made zero (i.e., voltage sources replaced by short circuits and current sources replaced by open circuits), and i_{sc} is the short-circuit current due to all sources in circuit A with v_o replaced by a short circuit.

The terms i_o and v_o are related by the equation

$$i_o = \frac{-v_o}{R_{Th}} \tag{4.2}$$

where R_{Th} is the equivalent resistance looking back into circuit A from terminals A-B with all independent sources in circuit A made zero.

Substituting Eq. (4.2) into Eq. (4.1) yields

$$i = -\frac{v_o}{R_{Th}} + i_{sc} \tag{4.3}$$

This is a general relationship and, therefore, must hold for any specific condition at terminals A-B. As a specific case, suppose that the terminals are open circuited. For this condition, $i = 0$ and v_o is equal to the open-circuit voltage v_{oc}. Thus, Eq. (4.3) becomes

$$i = 0 = \frac{-v_{oc}}{R_{Th}} + i_{sc} \tag{4.4}$$

Hence,

$$v_{oc} = R_{Th} i_{sc} \tag{4.5}$$

This equation states that the open-circuit voltage is equal to the short-circuit current times the equivalent resistance looking back into circuit A with all independent sources made zero. We refer to R_{Th} as the Thévenin equivalent resistance.

Substituting Eq. (4.5) into Eq. (4.3) yields

$$i = \frac{-v_o}{R_{Th}} + \frac{v_{oc}}{R_{Th}}$$

or

$$v_o = v_{oc} - R_{Th} i \tag{4.6}$$

Let us now examine the circuits that are described by these equations. The circuit represented by Eq. (4.6) is shown in Fig. 4.6a. The fact that this circuit is equivalent at terminals A-B to circuit A in Fig. 4.5 is a statement of *Thévenin's theorem*. The circuit represented by Eq. (4.3) is shown in Fig. 4.6b. The fact that this circuit is equivalent at terminals A-B to circuit A in Fig. 4.5 is a statement of *Norton's theorem*.

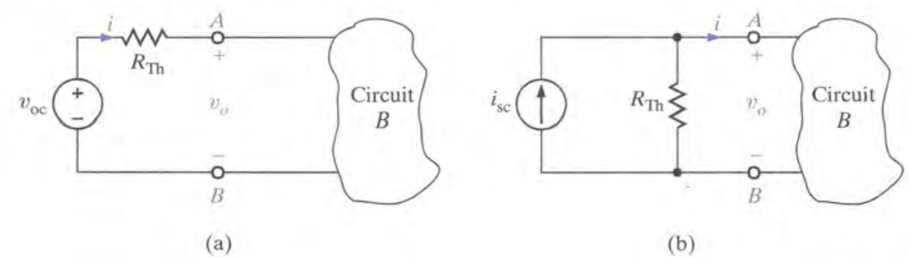


Figure 4.6
Thévenin and Norton equivalent circuits.

The relationships specified in Fig. 4.6 and Eq. (4.5) have added significance because they represent what is called a *source transformation* or *source exchange*. What these relationships tell us is that if we have embedded within a network a current source i in parallel with a resistor R, we can replace this combination with a voltage source of value $v = iR$ in series with the resistor R. The reverse is also true; that is, a voltage source v in series with a resistor R can be replaced with a current source of value $i = v/R$ in parallel with the resistor R. Parameters within the circuit (e.g., an output voltage) are unchanged under these transformations.

We must emphasize that the two equivalent circuits in Fig. 4.6 *are equivalent only at the two external nodes*. For example, if we disconnect circuit B from both networks in Fig. 4.6, the equivalent circuit in Fig. 4.6b dissipates power, but the one in Fig. 4.6a does not.

LEARNING Example 4.5

We will now demonstrate how to find V_o in the circuit in Fig. 4.7a using the repeated application of source transformation.

SOLUTION If we begin at the left end of the network in Fig. 4.7a, the series combination of the 12-V source and 3-kΩ resistor is converted to a 4-mA current source in parallel with the 3-kΩ resistor. If we combine this 3-kΩ resistor with the 6-kΩ resistor, we obtain the circuit in Fig. 4.7b. Note that at this point we have eliminated one circuit element. Continuing the reduction, we convert the 4-mA source and 2-kΩ resistor into an 8-V source in series with this same 2-kΩ resistor. The two 2-kΩ resistors that are in series are now combined to produce the network in Fig. 4.7c. If we now convert the combination of the 8-V source and 4-kΩ resistor into a 2-mA source in parallel with the 4-kΩ resistor and combine the resulting current source with the other 2-mA source, we arrive at the circuit shown in Fig. 4.7d. At this point, we can simply apply current division to the two parallel resistance paths and obtain

$$I_o = (4 \times 10^{-3})\left(\frac{4\text{k}}{4\text{k} + 4\text{k} + 8\text{k}}\right) = 1 \text{ mA}$$

and hence,

$$V_o = (1 \times 10^{-3})(8\text{k}) = 8 \text{ V}$$

The reader is encouraged to consider the ramifications of working this problem using any of the other techniques we have presented.

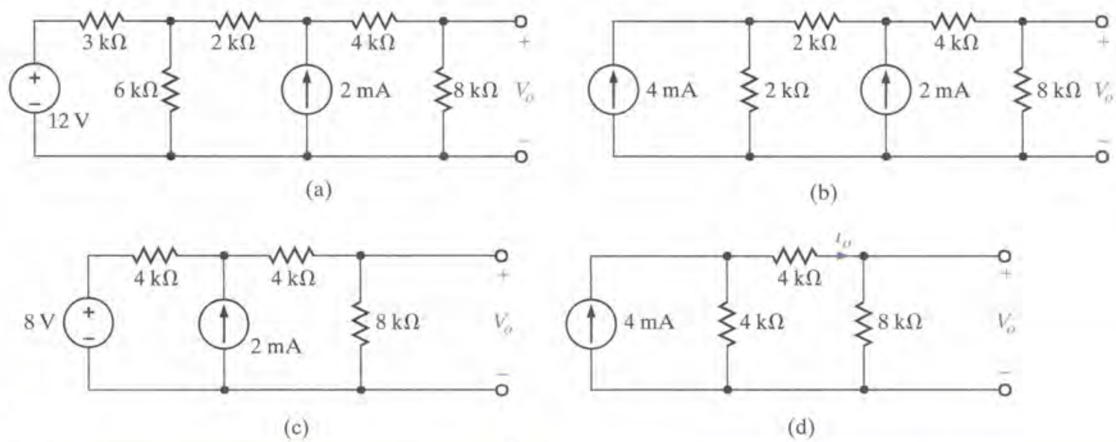


Figure 4.7 Circuits used in Example 4.5.

Note that this systematic, sometimes tedious, transformation allows us to reduce the network methodically to a simpler equivalent form with respect to some other circuit element. However, we should also realize that this technique is worthless for circuits of the form shown in Fig. 4.4. Furthermore, although applicable to networks containing dependent sources, it is not as useful as other techniques, and care must be taken not to transform the part of the circuit that contains the control variable.

Having demonstrated that there is an inherent relationship between the Thévenin equivalent circuit and the Norton equivalent circuit, we now proceed to apply these two important and useful theorems. The manner in which these theorems are applied depends on the structure of the original network under investigation. For example, if only independent sources are present, we can calculate the open-circuit voltage or short-circuit current and the Thévenin equivalent resistance. However, if dependent sources are also present, the Thévenin equivalent will be determined by calculating v_{oc} and i_{sc}, since this is normally the best approach for determining R_{Th} in a network containing dependent sources. Finally, if circuit A contains no *independent* sources, then both v_{oc} and i_{sc} will necessarily be zero. (Why?) Thus, we cannot determine R_{Th} by v_{oc}/i_{sc}, since the ratio is indeterminate. We must look for another approach. Notice that if $v_{oc} = 0$, then the equivalent circuit is merely the unknown resistance R_{Th}. If we apply an external source to circuit A—a test source v_t—and determine the current, i_t, which flows into circuit A from v_t, then R_{Th} can be determined from $R_{Th} = v_t/i_t$. Although the numerical value of v_t need not be specified, we could let $v_t = 1$ V and then $R_{Th} = 1/i_t$. Alternatively, we could use a current source as a test source and let $i_t = 1$ A; then $v_t = (1)R_{Th}$.

Before we begin our analysis of several examples that will demonstrate the utility of these theorems, remember that these theorems, in addition to an alternate method of attack, often permit us to solve several small problems rather than one large one. They allow us to replace a network, no matter how large, *at a pair of terminals* with a Thévenin or Norton equivalent circuit. In fact, we could represent the entire U.S. power grid at a pair of terminals with one of the equivalent circuits. Once this is done we can quickly analyze the effect of different loads on a network. Thus, these theorems provide us with additional insight into the operation of a specific network.

CIRCUITS CONTAINING ONLY INDEPENDENT SOURCES

LEARNING Example 4.6

Let us use Thévenin's and Norton's theorems to find V_o in the network in Example 4.3.

SOLUTION The circuit is redrawn in Fig. 4.8a. To determine the Thévenin equivalent, we break the network at the 6-kΩ load as shown in Fig. 4.8b. KVL indicates that the open-circuit voltage, V_{oc}, is equal to 3 V plus the voltage V_1, which is the voltage across the current source. The 2 mA from the current source flows through the two resistors (where else could it possibly go!) and, therefore, $V_1 = (2 \times 10^{-3})(1\text{k} + 2\text{k}) = 6$ V. Therefore, $V_{oc} = 9$ V. By making both sources zero, we can find the Thévenin equivalent resistance, R_{Th}, using the circuit in Fig. 4.8c. Obviously, $R_{Th} = 3$ kΩ.

Now our Thévenin equivalent circuit, consisting of V_{oc} and R_{Th}, is connected back to the original terminals of the load, as shown in Fig. 4.8d. Using a simple voltage divider, we find that $V_o = 6$ V.

To determine the Norton equivalent circuit at the terminals of the load, we must find the short-circuit current as shown in Fig. 4.8e. Note that the short circuit causes the 3-V source to be directly across (i.e., in parallel with) the resistors and the current source. Therefore, $I_1 = 3/(1\text{k} + 2\text{k}) = 1$ mA. Then, using KCL, $I_{sc} = 3$ mA. We have already determined R_{Th} and, therefore, connecting the Norton equivalent to the load results in the circuit in Fig. 4.8f. Hence, V_o is equal to the source current multiplied by the parallel resistor combination, which is 6 V.

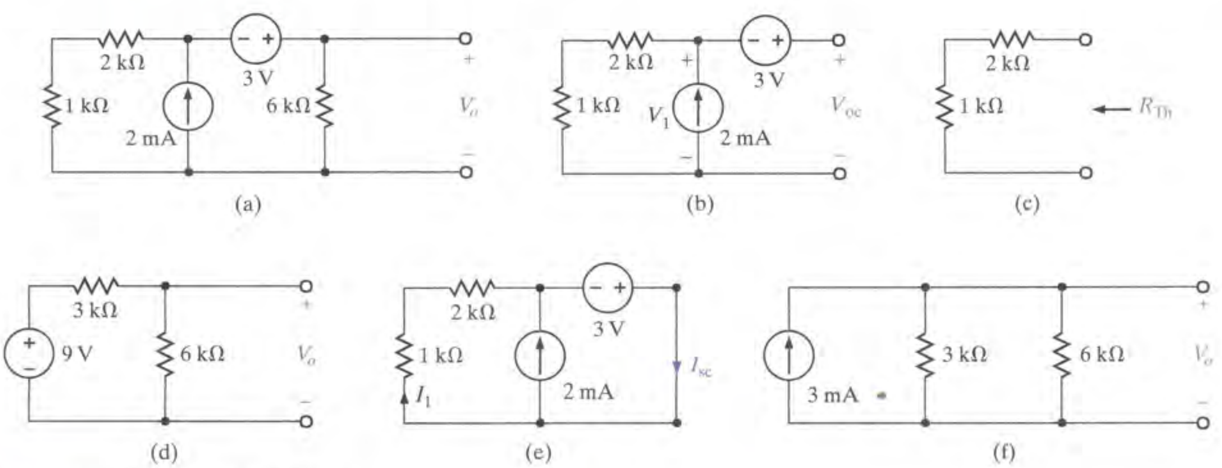


Figure 4.8 *Circuits used in Example 4.6.*

Consider for a moment some salient features of this example. Note that in applying the theorems there is no point in breaking the network to the left of the 3-V source, since the resistors in parallel with the current source are already a Norton equivalent, which can be immediately changed to a Thévenin equivalent using source transformation! Furthermore, once the network has been simplified using a Thévenin or Norton equivalent, we simply have a new network with which we can apply the theorems again. The following example illustrates this approach.

LEARNING Example 4.7

Let us use Thévenin's theorem to find V_o in the network in Fig. 4.7a, which is redrawn in Fig. 4.9a.

SOLUTION If we break the network to the left of the current source, the open-circuit voltage V_{oc_1} is as shown in Fig. 4.9b. Since there is no current in the 2-kΩ resistor and therefore no voltage across it, V_{oc_1} is equal to the voltage across the 6-kΩ resistor, which can be determined by voltage division as

$$V_{oc_1} = 12\left(\frac{6k}{6k + 3k}\right) = 8 \text{ V}$$

The Thévenin equivalent resistance, R_{Th_1}, is found from Fig. 4.9c as

$$R_{Th_1} = 2k + \frac{(3k)(6k)}{3k + 6k} = 4 \text{ k}\Omega$$

Connecting this Thévenin equivalent back to the original network produces the circuit shown in Fig. 4.9d. We can now apply Thévenin's theorem again, and this time we break the network to the right of the current source as shown in Fig. 4.9e. In this case V_{oc_2} is

$$V_{oc_2} = (2 \times 10^{-3})(4k) + 8 = 16 \text{ V}$$

and R_{Th_2} obtained from Fig. 4.9f is 4 kΩ. Connecting this Thévenin equivalent to the remainder of the network produces the circuit shown in Fig. 4.9g. Simple voltage division applied to this final network yields $V_o = 8$ V. Norton's theorem can be applied in a similar manner to solve this network; however, we save that solution as an exercise.

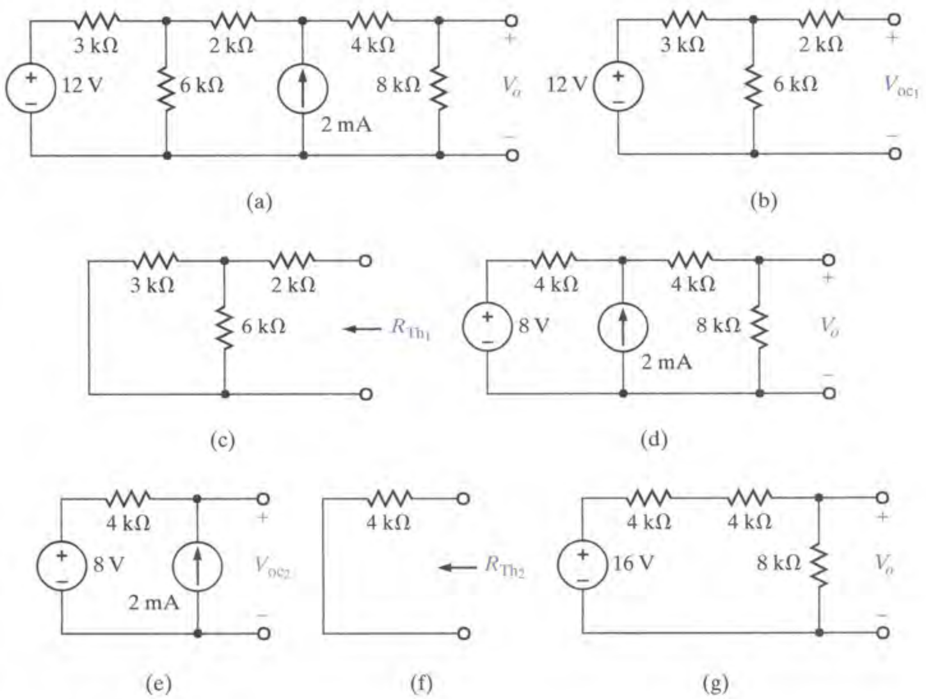
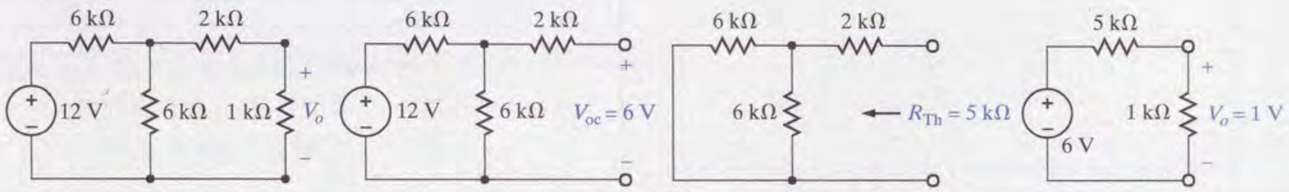


Figure 4.9 Circuits used in Example 4.7.

LEARNING by Doing

D 4.3 Find V_o in the following network using Thévenin's theorem.

ANSWER



LEARNING Example 4.8

It is instructive to examine the use of Thévenin's and Norton's theorems in the solution of the network in Fig. 4.4a, which is redrawn in Fig. 4.10a.

SOLUTION If we break the network at the 6-kΩ load, the open-circuit voltage is found from Fig. 4.10b. The equations for the mesh currents are

$$-6 + 4kI_1 + 2k(I_1 - I_2) = 0$$

and

$$I_2 = 2 \times 10^{-3}$$

from which we easily obtain $I_1 = 5/3$ mA. Then, using KVL, V_{oc} is

$$V_{oc} = 4kI_1 + 2kI_2$$
$$= 4k\left(\frac{5}{3} \times 10^{-3}\right) + 2k(2 \times 10^{-3})$$
$$= \frac{32}{3}\text{ V}$$

R_{Th} is derived from Fig. 4.10c and is

$$R_{Th} = (2k//4k) + 2k = \frac{10}{3}\text{ k}\Omega$$

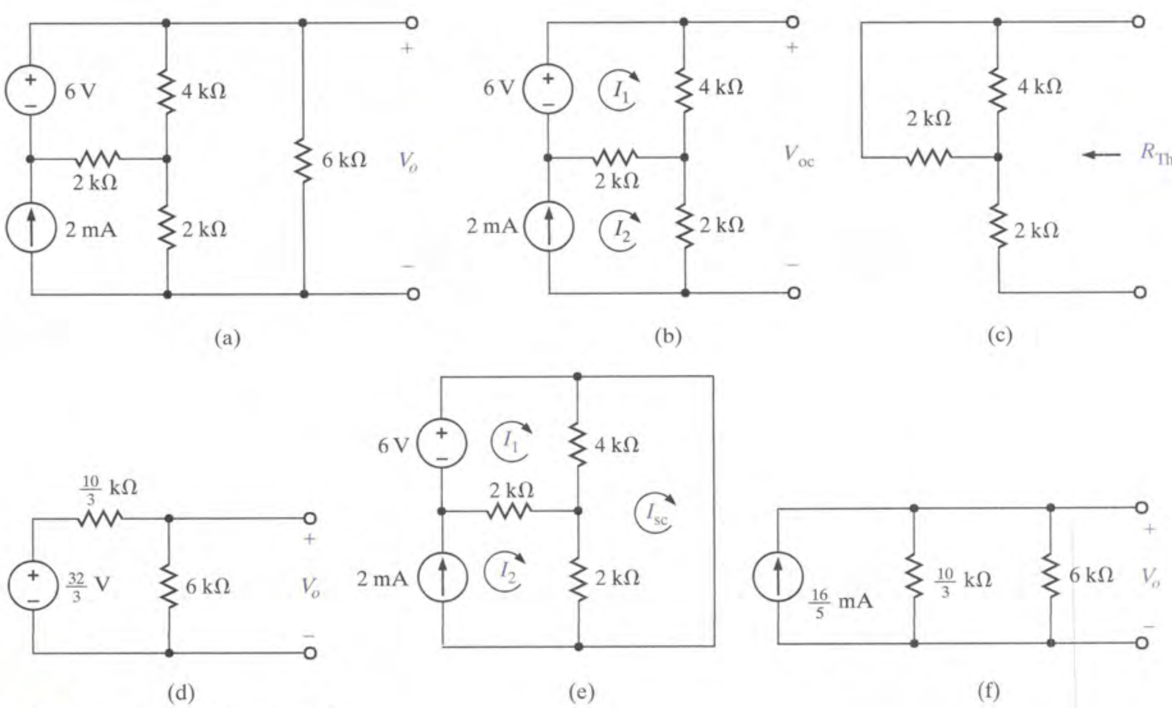


Figure 4.10 Circuits used in Example 4.8.

(continued)

Attaching the Thévenin equivalent to the load produces the network in Fig. 4.10d. Then using voltage division, we obtain

$$V_o = \frac{32}{3}\left(\frac{6\text{k}}{6\text{k} + \frac{10}{3}\text{k}}\right)$$
$$= \frac{48}{7}\text{ V}$$

In applying Norton's theorem to this problem, we must find the short-circuit current shown in Fig. 4.10e. At this point the quick-thinking reader stops immediately! Three mesh equations applied to the original circuit will immediately lead to the solution, but the three mesh equations in the circuit in Fig. 4.10e will provide only part of the answer, specifically the short-circuit current. Sometimes the use of the theorems is more complicated than a straightforward attack using node or loop analysis. This would appear to be one of those situations. Interestingly, it is not. We can find I_{sc} from the network in Fig. 4.10e without using the mesh equations. The technique is simple, but a little tricky, and so we ignore it at this time. Having said all these things, let us now finish what we have started. The mesh equations for the network in Fig. 4.10e are

$$-6 + 4\text{k}(I_1 - I_{sc}) + 2\text{k}(I_1 - 2 \times 10^{-3}) = 0$$
$$2\text{k}(I_{sc} - 2 \times 10^{-3}) + 4\text{k}(I_{sc} - I_1) = 0$$

where we have incorporated the fact that $I_2 = 2 \times 10^{-3}$ A. Solving these equations yields $I_{sc} = 16/5$ mA. R_{Th} has already been determined in the Thévenin analysis. Connecting the Norton equivalent to the load results in the circuit in Fig. 4.10f. Applying Ohm's law to this circuit yields $V_o = 48/7$ V.

LEARNING EXTENSIONS

E4.3 Use Thévenin's theorem to find V_o in the network in Fig. E4.3.

ANSWER $V_o = -3$V.

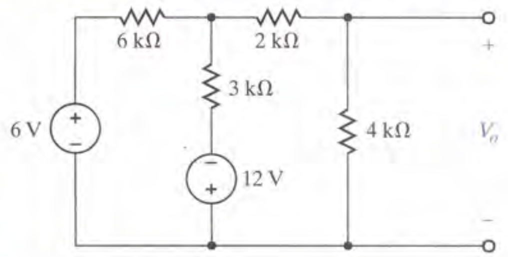


Figure E4.3

E4.4 Find V_o in the circuit in Fig. E4.2 using (a) source exchange and (b) both Thévenin's and Norton's theorems. When deriving the Norton equivalent circuit, break the network to the left of the 4-kΩ resistor. Why?

ANSWER $V_o = \frac{4}{3}$ V.

CIRCUITS CONTAINING ONLY DEPENDENT SOURCES As we have stated earlier, the Thévenin or Norton equivalent of a network containing only dependent sources is R_{Th}. The following examples will serve to illustrate how to determine this Thévenin equivalent resistance.

LEARNING Example 4.9

We wish to determine the Thévenin equivalent of the network in Fig. 4.11a at the terminals A-B.

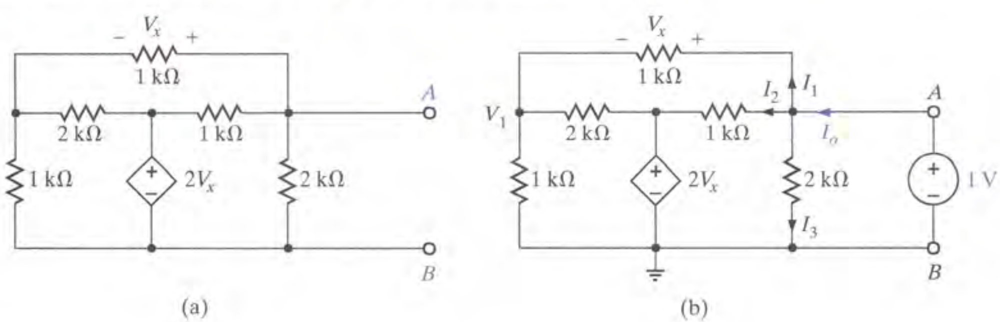


Figure 4.11 Networks employed in Example 4.9.

SOLUTION Our approach to this problem will be to apply a 1-V source at the terminals as shown in Fig. 4.11b and then compute the current I_o and $R_{Th} = 1/I_o$.

The equations for the network in Fig. 4.11b are as follows. KVL around the outer loop specifies that

$$V_1 + V_x = 1$$

The KCL equation at the node labeled V_1 is

$$\frac{V_1}{1k} + \frac{V_1 - 2V_x}{2k} + \frac{V_1 - 1}{1k} = 0$$

Solving the equations for V_x yields $V_x = 3/7$ V. Knowing V_x, we can compute the currents I_1, I_2, and I_3. Their values are

$$I_1 = \frac{V_x}{1k} = \frac{3}{7}\text{ mA}$$

$$I_2 = \frac{1 - 2V_x}{1k} = \frac{1}{7}\text{ mA}$$

$$I_3 = \frac{1}{2k} = \frac{1}{2}\text{ mA}$$

Therefore,

$$I_o = I_1 + I_2 + I_3 = \frac{15}{14}\text{ mA}$$

and

$$R_{Th} = \frac{1}{I_o} = \frac{14}{15}\text{ k}\Omega$$

LEARNING Example 4.10

Let us determine R_{Th} at the terminals A-B for the network in Fig. 4.12a.

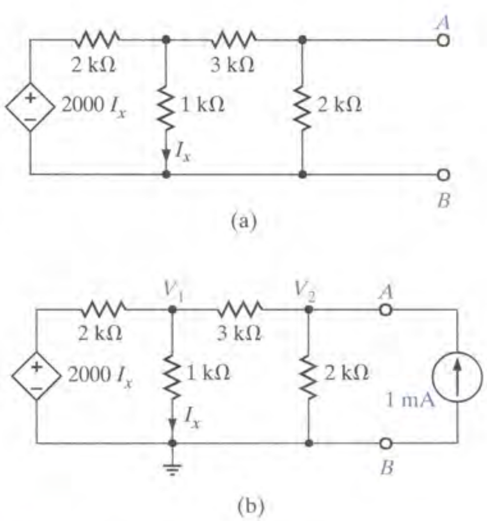


Figure 4.12 Networks used in Example 4.10.

SOLUTION Our approach to this problem will be to apply a 1-mA current source at the terminals A-B and compute the terminal voltage V_2 as shown in Fig. 4.12b. Then $R_{Th} = V_2/0.001$.

The node equations for the network are

$$\frac{V_1 - 2000I_x}{2k} + \frac{V_1}{1k} + \frac{V_1 - V_2}{3k} = 0$$

$$\frac{V_2 - V_1}{3k} + \frac{V_2}{2k} = 1 \times 10^{-3}$$

and

$$I_x = \frac{V_1}{1k}$$

Solving these equations yields

$$V_2 = \frac{10}{7}\text{ V}$$

and hence,

$$R_{Th} = \frac{V_2}{1 \times 10^{-3}} = \frac{10}{7}\text{ k}\Omega$$

CIRCUITS CONTAINING BOTH INDEPENDENT AND DEPENDENT SOURCES In these types of circuits we must calculate both the open-circuit voltage and short-circuit current to calculate the Thévenin equivalent resistance. Furthermore, we must remember that we cannot split the dependent source and its controlling variable when we break the network to find the Thévenin or Norton equivalent.

We now illustrate this technique with a circuit containing a current-controlled voltage source.

LEARNING Example 4.11

Let us use Thévenin's theorem to find V_o in the network in Fig. 4.13a.

SOLUTION To begin, we break the network at points A-B. Could we break it just to the right of the 12-V source? No! Why? The open-circuit voltage is calculated from the network in Fig. 4.13b. Note that we now use the source $2000I'_x$ because this circuit is different from that in Fig. 4.13a. KCL for the supernode around the 12-V source is

$$\frac{(V_{oc} + 12) - (-2000I'_x)}{1\text{k}} + \frac{V_{oc} + 12}{2\text{k}} + \frac{V_{oc}}{2\text{k}} = 0$$

where

$$I'_x = \frac{V_{oc}}{2\text{k}}$$

yielding $V_{oc} = -6\ V$.

I_{sc} can be calculated from the circuit in Fig. 4.13c. Note that the presence of the short circuit forces I''_x to zero and, therefore, the network is reduced to that shown in Fig. 4.13d.

Therefore,

$$I_{sc} = \frac{-12}{\frac{2}{3}\text{k}} = -18\text{ mA}$$

Then

$$R_{Th} = \frac{V_{oc}}{I_{sc}} = \frac{1}{3}\text{ k}\Omega$$

Connecting the Thévenin equivalent circuit to the remainder of the network at terminals A-B produces the circuit in Fig. 4.13e. At this point, simple voltage division yields

$$V_o = (-6)\left(\frac{1\text{k}}{1\text{k} + 1\text{k} + \frac{1}{3}\text{k}}\right) = \frac{-18}{7}\text{ V}$$

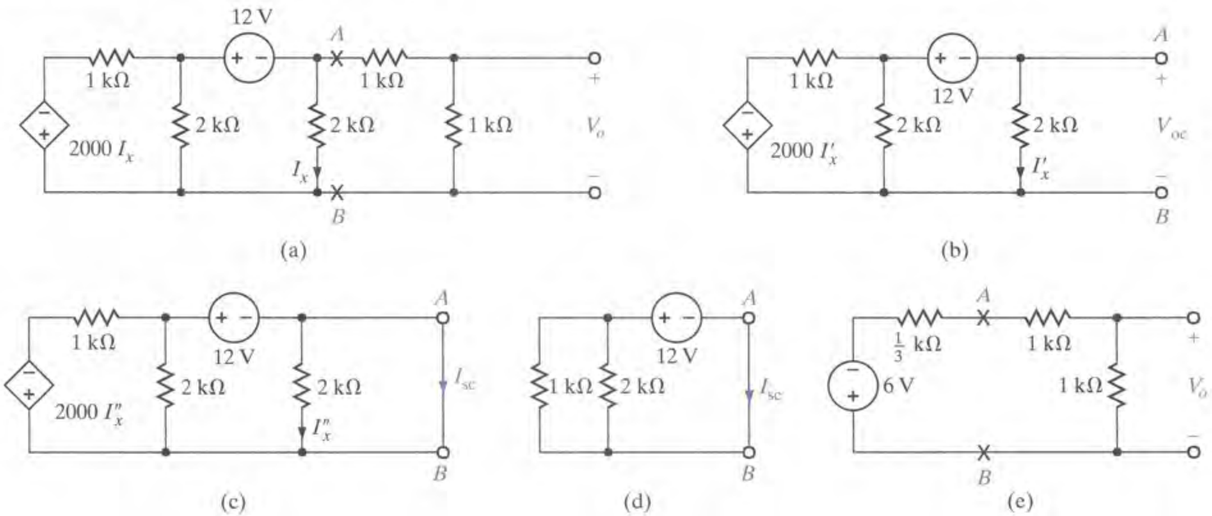


Figure 4.13 Circuits used in Example 4.11.

Problem-Solving Strategy Applying Thévenin's Theorem

- Remove the load and find the voltage across the open-circuit terminals, V_{oc}. All the circuit analysis techniques presented here can be used to compute this voltage.
- Determine the Thévenin equivalent resistance of the network at the open terminals with the load removed. Three different types of circuits may be encountered in determining the resistance, R_{Th}.
  - If the circuit contains only independent sources, they are made zero by replacing the voltage sources with short circuits and the current sources with open circuits. R_{Th} is then found by computing the resistance of the purely resistive network at the open terminals.

- If the circuit contains only dependent sources, an independent voltage or current source is applied at the open terminals and the corresponding current or voltage at these terminals is measured. The voltage/current ratio at the terminals is the Thévenin equivalent resistance. Since there is no energy source, the open-circuit voltage is zero in this case.
- If the circuit contains both independent and dependent sources, the open-circuit terminals are shorted and the short-circuit current between these terminals is determined. The ratio of the open-circuit voltage to the short-circuit current is the resistance R_{Th}.

- If the load is now connected to the Thévenin equivalent circuit, consisting of V_{oc} in series with R_{Th}, the desired solution can be obtained.

The problem-solving strategy for Norton's theorem is essentially the same as that for Thévenin's theorem with the exception that we are dealing with the short-circuit current instead of the open-circuit voltage.

LEARNING EXTENSION

E4.5 Find V_o in the circuit in Fig. E4.5 using Thévenin's theorem.

ANSWER $V_o = \frac{36}{13}$ V.

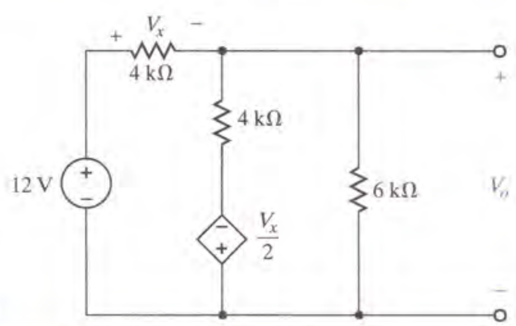


Figure E4.5

At this point it is worthwhile to pause for a moment and reflect about what we have learned; that is, let us compare the use of node or loop analysis with that of the theorems discussed in this chapter. When we examine a network for analysis, one of the first things we should do is count the number of nodes and loops. Next we consider the number of sources. For example, are there a number of voltage sources or current sources present in the network? All these data, together with the information that we expect to glean from the network, give a basis for selecting the simplest approach. With the current level of computational power available to us, we can solve the node or loop equations that define the network in a flash.

With regard to the theorems, we have found that in some cases the theorems do not necessarily simplify the problem and a straightforward attack using node or loop analysis is as good an approach as any. This is a valid point provided that we are simply looking for some particular voltage or current. However, the real value of the theorems is the insight and understanding that they provide about the physical nature of the network. For example, superposition tells us what each source contributes to the quantity under investigation. However, a computer solution of the node or loop equations does not tell us the effect of changing certain parameter values in the circuit. It does not help us understand the concept of loading a network or the ramifications of interconnecting networks or the idea of matching a network for maximum power transfer. The theorems help us to understand the effect of using a transducer at the input of an amplifier with a given input resistance. They help us explain the effect of a load, such as a speaker, at the output of an amplifier. We derive none of this information from a node or loop analysis. In fact, as a simple example, suppose that a network at a specific pair of terminals has a Thévenin equivalent circuit consisting of a voltage source in series with a 2-kΩ resistor. If we connect a 2-Ω resistor to the network at these terminals, the voltage across the 2-Ω resistor will be essentially nothing. This result is fairly obvious using the Thévenin theorem approach; however, a node or loop analysis gives us no clue as to why we have obtained this result.

We have studied networks containing only dependent sources. This is a very important topic because all electronic devices, such as transistors, are modeled in this fashion. Motors in power systems are also modeled in this way. We use these amplification devices for many different purposes, such as speed control for automobiles.

In addition, it is interesting to note that when we employ source transformation as we did in Example 4.5, we are simply converting back and forth between a Thévenin equivalent circuit and a Norton equivalent circuit.

Finally, we have a powerful tool at our disposal that can be used to provide additional insight and understanding for both circuit analysis and design. That tool is Microsoft EXCEL, and it permits us to study the effects, on a network, of varying specific parameters. The following example will illustrate the simplicity of this approach.

LEARNING Example 4.12

We wish to use Microsoft EXCEL to plot the Thévenin equivalent parameters V_{oc} and R_{Th} for the circuit in Fig. 4.14 over the R_x range 0 to 10 kΩ.

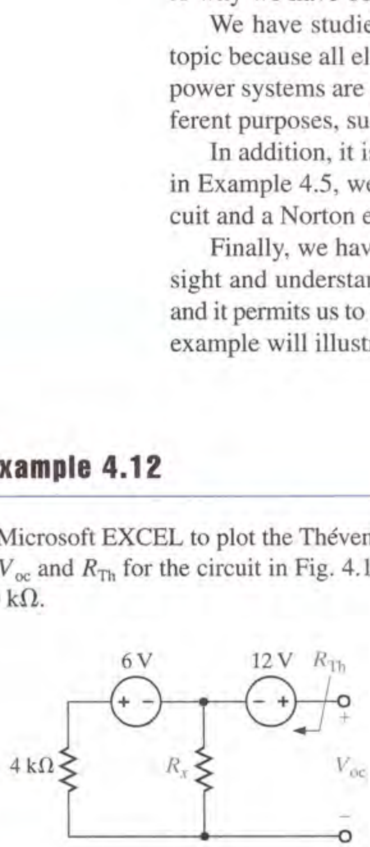


Figure 4.14 Circuit used in Example 4.12.

SOLUTION The Thévenin resistance is easily found by replacing the voltage sources with short circuits. The result is

$$R_{Th} = 4//R_x = \frac{4R_x}{4 + R_x} \quad \textbf{4.7}$$

where R_x and R_{Th} are in kΩ. Superposition can be used effectively to find V_{oc}. If the 12-V source is replaced by a short circuit

$$V_{oc_1} = -6\left[\frac{R_x}{R_x + 4}\right]$$

Applying this same procedure for the 6-V source yields

$$V_{oc_2} = 12$$

and the total open-circuit voltage is

$$V_{oc} = 12 - 6\left[\frac{R_x}{R_x + 4}\right] \quad \textbf{4.8}$$

In EXCEL we wish to (1) vary R_x between 0 and 10 kΩ, (2) calculate R_{Th} and V_{oc} at each R_x value, and (3) plot V_{oc} and R_{Th} versus R_x. We begin by opening EXCEL and entering column headings as shown in Fig. 4.15a. Next, we enter a zero in the first cell of the R_x column at column-row location A4. To automatically fill the column with values, go to the Edit menu and select Fill/Series to open the window shown in Fig. 4.15b, which has already been edited appropriately for 101 data points. The result is a series of R_x values from 0 to 10 kΩ in 100 Ω steps. To enter Eq. (4.8), go to location B4 (right under the V_{oc} heading). Enter the following text and do not forget the equal sign:

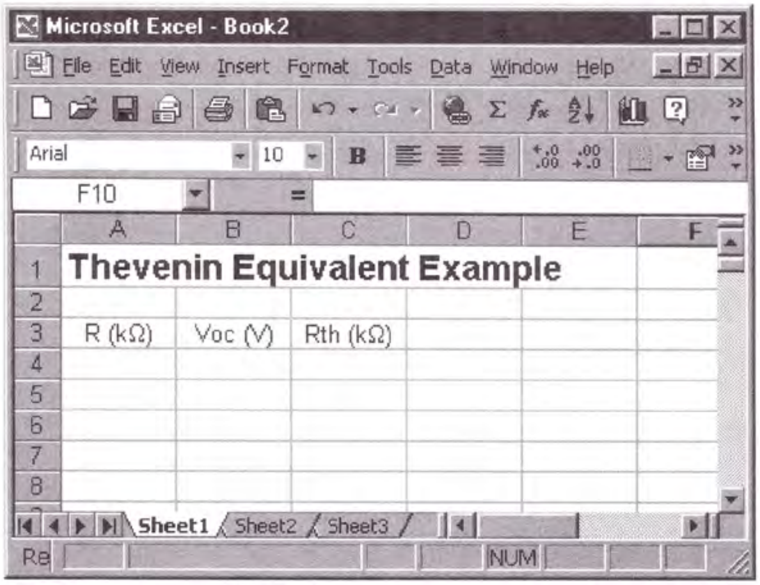
```
=12-6*A4/(A4+4)
```

This is Eq. (4.8) with R_x replaced by the first value for R_x, which is at column-row location A4. Similarly for R_{Th}, enter the following expression at C4.

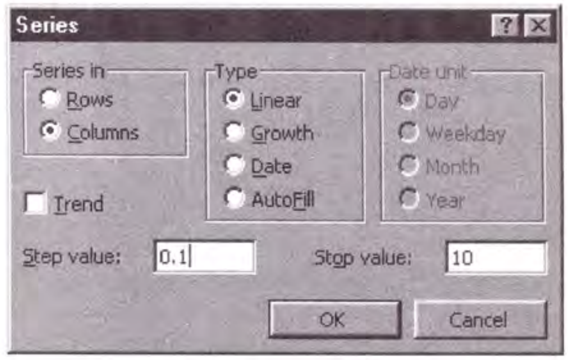
```
=4*A4/(A4+4)
```

To replicate the expression in cell B4 for all R_x values, select cell B4, grab the lower right corner of the cell, hold and drag down to cell B104, and release. Repeat for R_{Th} by replicating cell C4.

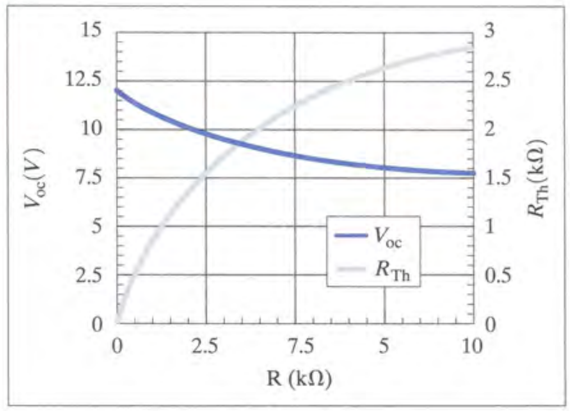
To plot the data, first drag the cursor across all cells between A4 and C104. Next, from the Insert menu, select Chart. We recommend strongly that you choose the XY (Scatter) chart type. EXCEL will take you step by step through the basic formatting of your chart, which, after some manipulations, might look similar to the chart in Fig. 4.15c.



(a)



(b)



(c)

Figure 4.15
(a) The EXCEL spreadsheet for Example 4.12 showing the desired column headings. (b) The Fill/Series window edited for varying R_x and (c) the final plot of V_{oc} and R_{Th}.

4.4 Maximum Power Transfer

In circuit analysis we are sometimes interested in determining the maximum power that can be delivered to a load. By employing Thévenin's theorem, we can determine the maximum power that a circuit can supply and the manner in which to adjust the load to effect maximum power transfer.

Suppose that we are given the circuit shown in Fig. 4.16. The power that is delivered to the load is given by the expression

$$P_{\text{load}} = i^2 R_L = \left(\frac{v}{R + R_L}\right)^2 R_L$$

We want to determine the value of R_L that maximizes this quantity. Hence, we differentiate this expression with respect to R_L and equate the derivative to zero.

$$\frac{dP_{\text{load}}}{dR_L} = \frac{(R + R_L)^2 v^2 - 2v^2 R_L (R + R_L)}{(R + R_L)^4} = 0$$

which yields

$$R_L = R$$

In other words, maximum power transfer takes place when the load resistance $R_L = R$. Although this is a very important result, we have derived it using the simple network in Fig. 4.16. However, we should recall that v and R in Fig. 4.16 could represent the Thévenin equivalent circuit for any linear network.

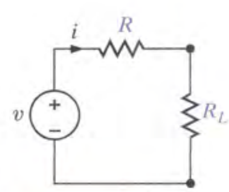


Figure 4.16
Equivalent circuit for examining maximum power transfer.

LEARNING Example 4.13

Let us find the value of R_L for maximum power transfer in the network in Fig. 4.17a and the maximum power that can be transferred to this load.

SOLUTION To begin, we derive the Thévenin equivalent circuit for the network exclusive of the load. V_{oc} can be calculated from the circuit in Fig. 4.17b. The mesh equations for the network are

$$I_1 = 2 \times 10^{-3}$$

$$3k(I_2 - I_1) + 6kI_2 + 3 = 0$$

Solving these equations yields $I_2 = 1/3$ mA and, hence,

$$V_{oc} = 4kI_1 + 6kI_2$$
$$= 10 \text{ V}$$

R_{Th}, shown in Fig. 4.17c, is 6 kΩ; therefore, $R_L = R_{Th} = 6$ kΩ for maximum power transfer. The maximum power transferred to the load is

$$P_L = \left(\frac{10}{12k}\right)^2 (6k) = \frac{25}{6} \text{ mW}$$

LEARNING by Doing

D 4.4 In the network in Fig. 4.16, $v = 12$ V and $R = 2$ kΩ. Determine the R_L for maximum power transfer and the maximum power transferred.

ANSWER $R_L = 2$ kΩ
$P_{max} = 18$ mW

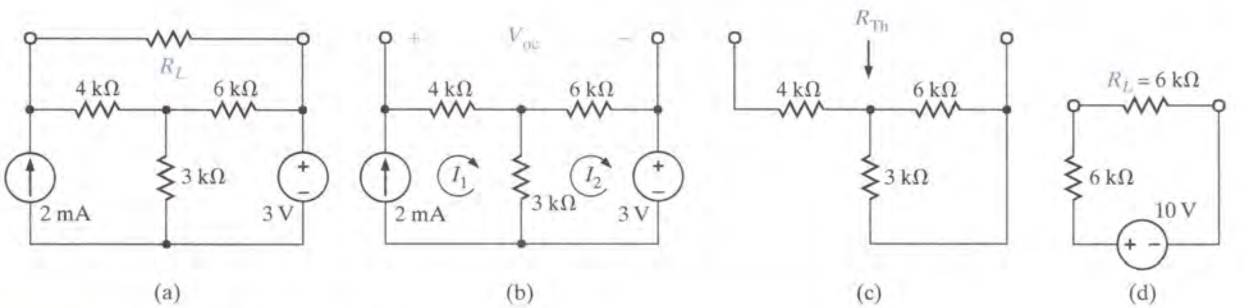


Figure 4.17 Circuits used in Example 4.13.

LEARNING Example 4.14

Let us find R_L for maximum power transfer and the maximum power transferred to this load in the circuit in Fig. 4.18a.

SOLUTION We wish to reduce the network to the form shown in Fig. 4.16. We could form the Thévenin equivalent circuit by breaking the network at the load. However, a close examination of the network indicates that our analysis will be simpler if we break the network to the left of the 4-kΩ resistor. When we do this, however, we must realize that for maximum power transfer $R_L = R_{Th} + 4$ kΩ. V_{oc} can be calculated from the network in Fig. 4.18b. Forming a supernode around the dependent source and its connecting nodes, the KCL equation for this supernode is

$$\frac{V_{oc} - 2000I'_x}{1k + 3k} + (-4 \times 10^{-3}) + \frac{V_{oc}}{2k} = 0$$

where

$$I'_x = \frac{V_{oc}}{2k}$$

These equations yield $V_{oc} = 8$ V. The short-circuit current can be found from the network in Fig. 4.18c. It is here that we find the advantage of breaking the network to the left of the 4-kΩ resistor. The short circuit shorts the 2-kΩ resistor and, therefore, $I''_x = 0$. Hence, the circuit is reduced to that in Fig. 4.18d, where clearly $I_{sc} = 4$ mA. Then

$$R_{Th} = \frac{V_{oc}}{I_{sc}} = 2 \text{ k}\Omega$$

Connecting the Thévenin equivalent to the remainder of the original circuit produces the network in Fig. 4.18e. For maximum power transfer $R_L = R_{Th} + 4$ kΩ $= 6$ kΩ, and the maximum power transferred is

$$P_L = \left(\frac{8}{12k}\right)^2 (6k) = \frac{8}{3} \text{ mW}$$

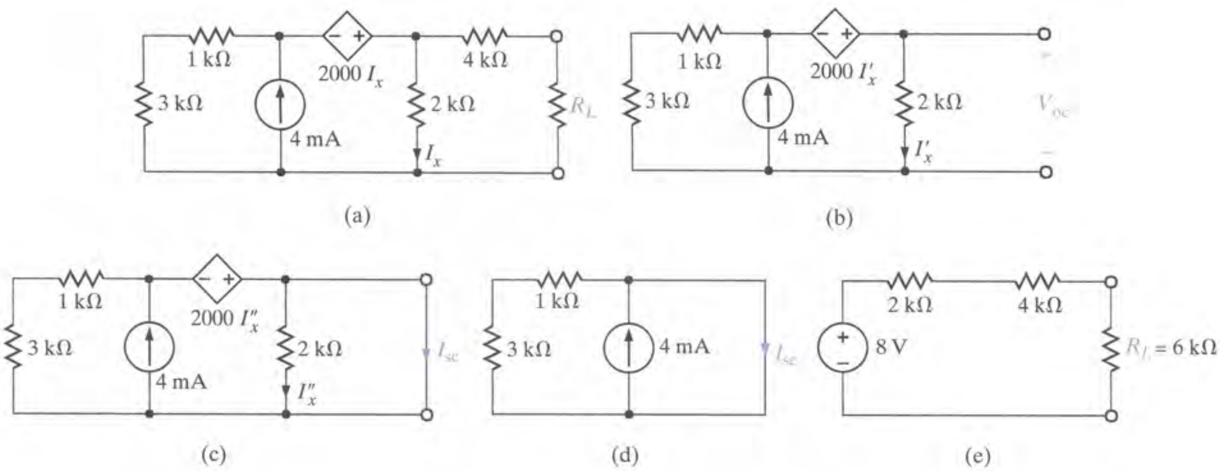


Figure 4.18 *Circuit used in Example 4.14.*

LEARNING EXTENSION

E4.6 Given the circuit in Fig. E4.6, find R_L for maximum power transfer and the maximum power transferred.

ANSWER $R_L = 6\ \text{k}\Omega$, $P_L = 2/3\ \text{mW}$.

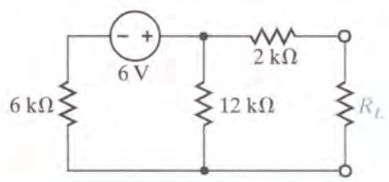


Figure E4.6

4.5 dc Spice Analysis Using Schematic Capture

INTRODUCTION The original version of SPICE (Simulation Program with Integrated Circuit Emphasis) was developed at the University of California at Berkeley. It quickly became an industry standard for simulating integrated circuits. With the advent and development of the PC industry, several companies began selling PC- and Macintosh-compatible versions of SPICE. One company, OrCAD Corporation, a division of Cadence Design Systems, Inc., produces a PC-compatible version called PSPICE, which we will discuss in some detail in this text.

In SPICE, circuit information such as the names and values of resistors and sources as well as how they are interconnected is input using data statements with a specific format. The particulars for every element must be typed in a precise order. This makes debugging difficult since you must know the proper format to recognize formatting errors.

PSPICE, as well as other SPICE-based simulators, now employs a feature known as schematic capture. Using an editor, we bypass the cryptic formatting, and simply draw the circuit diagram, assigning element values via dialog boxes. The editor then converts the circuit diagram to an original SPICE format for actual simulation. In the student version of Release 9.1, the most current software available from ORCAD, there are two editors, CAPTURE and *Schematics*. CAPTURE is intended for printed circuit board (PCB) design, while *Schematics* is more generic. When installing PSPICE, the window in Fig. 4.19 will appear, giving you the option of choosing either or both editors. In this text, we will use *Schematics* exclusively.

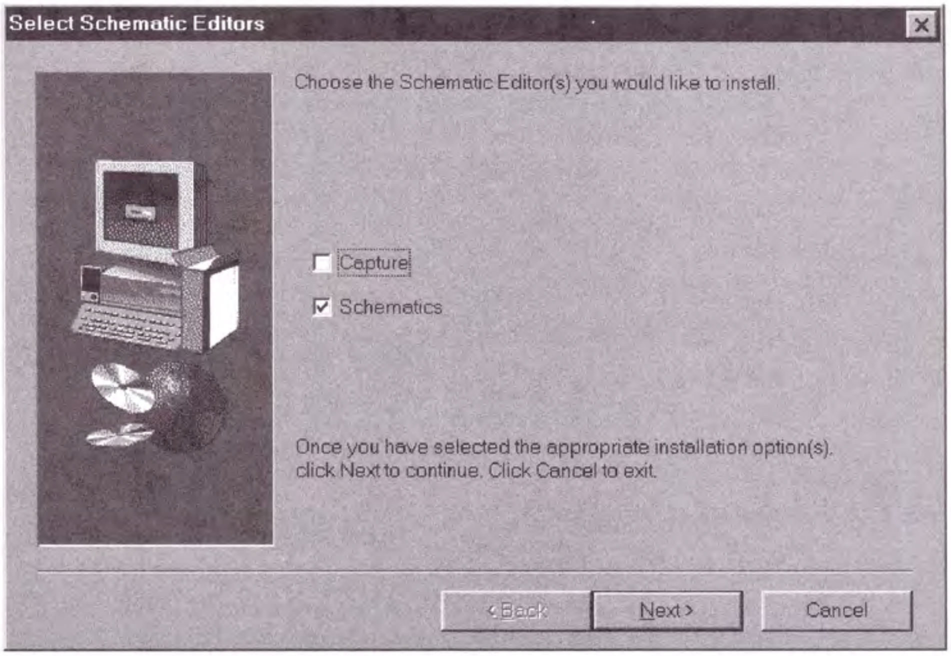


Figure 4.19
In the student version of Release 9.1, you can choose to install either or both schematic capture editors.

When elements such as resistors and voltage sources are given values, it is convenient to use scale factors. PSPICE supports the following list.

T	tera	10^{12}	K	kilo	10^{3}	N	nano	10^{-9}
G	giga	10^{9}	M	milli	10^{-3}	P	pico	10^{-12}
MEG	mega	10^{6}	U	micro	10^{-6}	F	femto	10^{-15}

Component values must be immediately followed by a character—spaces are not allowed. Any text can follow the scale factor. Finally, PSPICE is case insensitive, so there is no difference between 1 Mohm and 1 mohm in PSPICE. Figure 4.20 shows the five-step procedure for simulating circuits using *Schematics* and PSPICE. We will demonstrate this procedure in the following example.

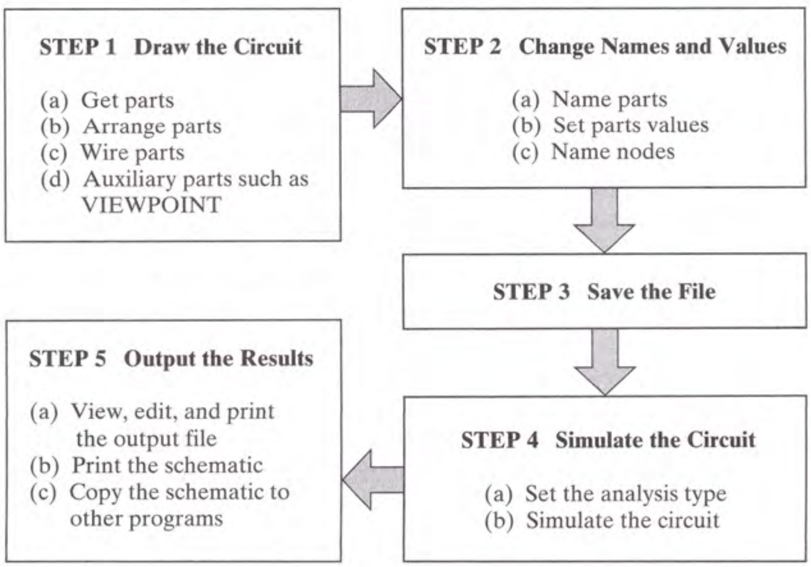


Figure 4.20
The five-step procedure for PSPICE simulations.

A dc SIMULATION EXAMPLE The following font conventions will be used throughout this tutorial. Uppercase text refers to PSPICE programs, menus, dialog boxes, and utilities. All boldface text denotes keyboard or mouse inputs. In each instance, the case in boldface text matches that in PSPICE. Let us simulate the circuit in Fig. 4.21 using PSPICE. Following the flowchart procedure in Fig. 4.20, the first step is to open *Schematics* using the **Start/Programs/Pspice Student/Schematics** sequence of pop-up menus. When *Schematics* opens, our screen will change to the *Schematics* editor window shown in Fig. 4.22.

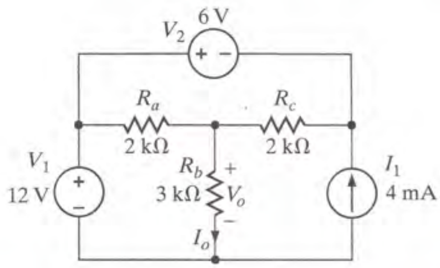


Figure 4.21 *A dc circuit used for simulation.*

Figure 4.22 *The Schematics editor window.*

Step 1: Drawing the Schematic

Next, we obtain and place the required parts: three resistors, two dc voltage sources, and a dc current source. To get the voltage sources, left-click on the **Draw** menu and select **Get New Part** as shown in Fig. 4.23. The **Part Browser** in Fig. 4.24 appears, listing all the parts available in PSPICE. Since we do not know the *Schematics* name for a dc voltage source, select **Libraries**, and the **Library Browser** in Fig. 4.25 appears. This browser lists all the parts libraries available to us. The dc voltage source is called VDC and is located in the SOURCE.slb library, as seen in Fig. 4.25. Thus, we select the part VDC and click OK. The box in Fig. 4.24 reappears.

If we now select **Place & Close** we revert back to the *Schematics* editor in Fig. 4.22 with one difference: the mouse pointer has become a dc voltage source symbol. The source can be positioned within the drawing area by moving the mouse and then placed by left-clicking *once*. To place the second source, *move the mouse some distance*, and left-click again to place V2. To stop adding sources, we right-click once. Moving the mouse between part placements keeps the parts from stacking atop one another in the diagram. In Fig. 4.21, V2 is oriented horizontally. To rotate

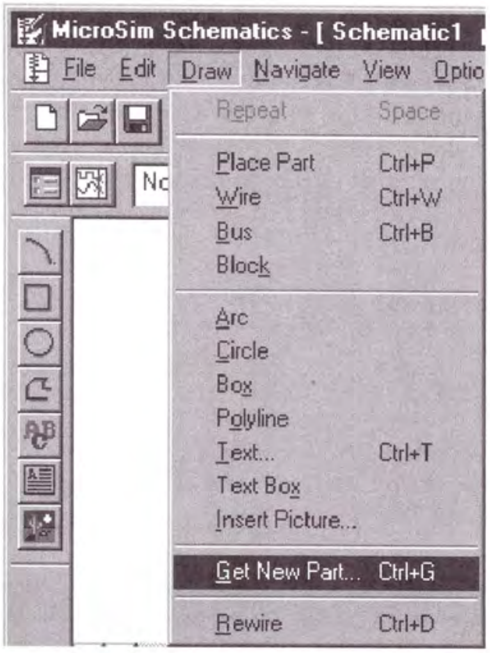


Figure 4.23 *Getting a new part in* Schematics.

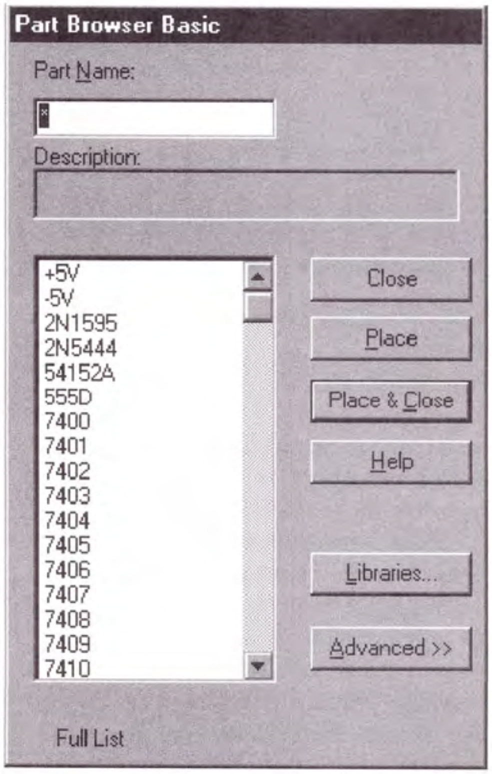


Figure 4.24 *The Part Browser window.*

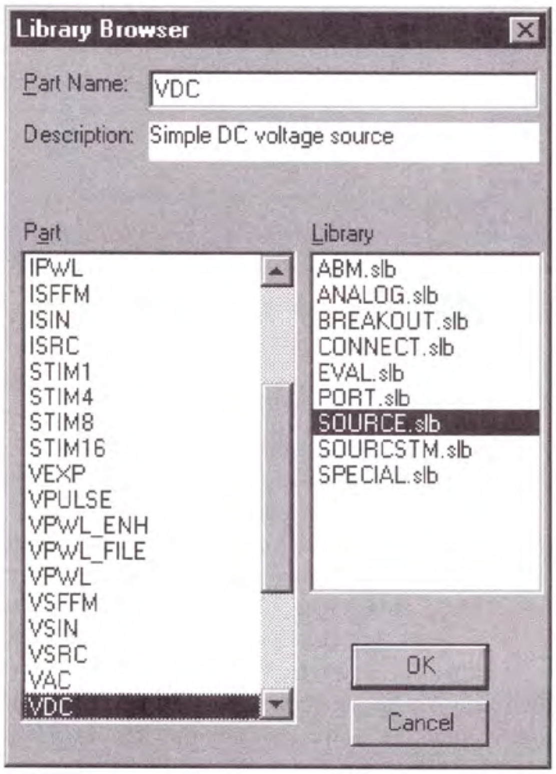


Figure 4.25 *The Library Browser window.*

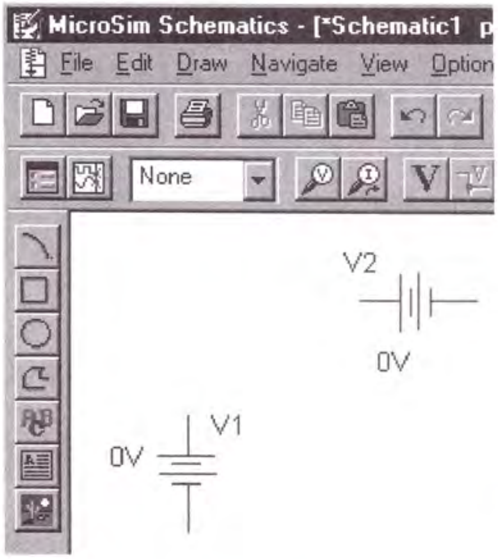


Figure 4.26
The Schematics *editor after placing the dc voltage sources.*

V2 in your schematic, select it by clicking on it once. The part should turn red. In the **Edit** menu, choose **Rotate**. This causes V2 to spin 90° counterclockwise. Next, we click on V2 and drag it to the desired location. A diagram similar to that shown in Fig. 4.26 should result.

Next, we place the resistors. Repeat the process of **Draw/Get New Part**, except this time, when the **Part Browser Basic** dialog box appears, we type in **R** and select **Place & Close**. The mouse pointer then becomes a resistor. Place each resistor and right-click when done. Note that the resistors are automatically assigned default values of 1 kΩ. Current sources are in the SOURCE.slb library and are called IDC. Get one, place it, and rotate it twice. The resulting schematic is shown in Fig. 4.27.

The parts can now be interconnected. Go to the **Draw** menu and select **Wire**. The mouse pointer will turn into a symbolic pencil. To connect the top of V1 to R1, point the pencil at the end of the wire stub protruding from V1, click once and release. Next, we move the mouse up and over to the left end of R1. A line is drawn up and over a 90° angle, appearing dashed as

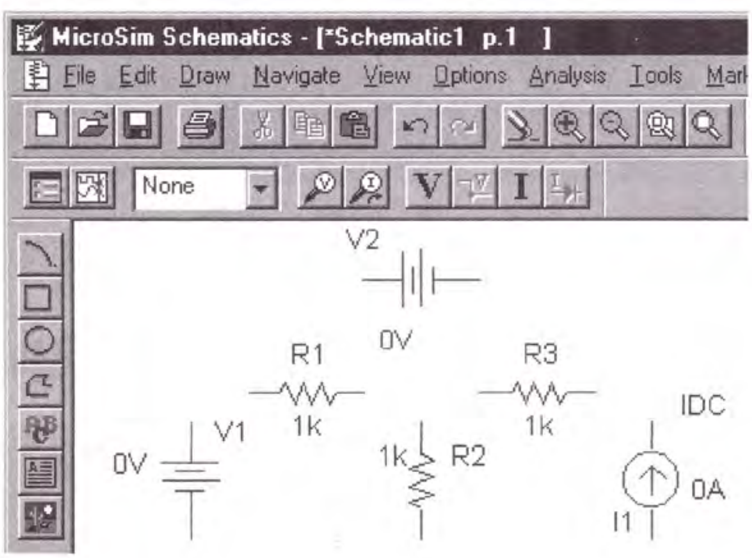


Figure 4.27
The schematic after part placement and ready for wiring.

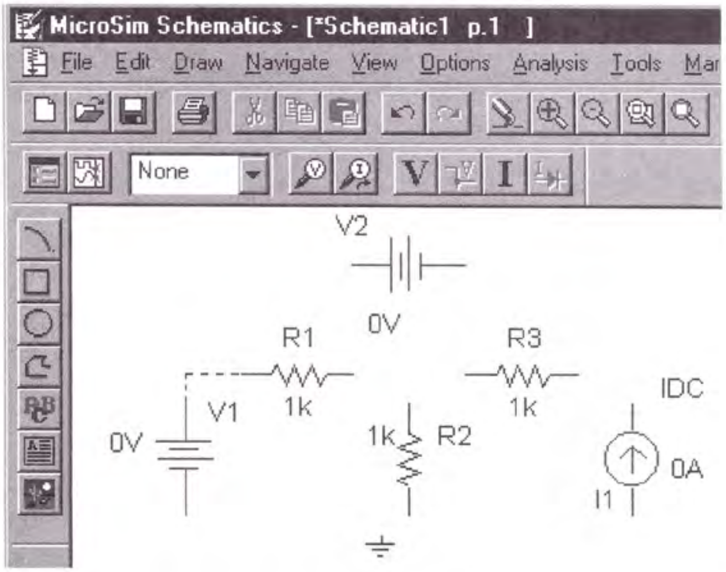


Figure 4.28
Caught in the act of connecting V1 to R1.

shown in Fig. 4.28. *Dashed lines are not yet wires!* We must left-click again to complete and "cut" the connection. The dashed lines become solid and the wiring connection is made. Excess wire fragments (extended dashed lines) can be removed by selecting **Redraw** from the **View** menu. The remaining wires can be drawn using a *Schematics* shortcut. To reactivate the wiring pencil, double right-click. This shortcut reactivates the most recent mouse use. We simply repeat the steps listed above to complete the wiring.

PSPICE requires that all schematics have a ground or reference terminal. The ground node voltage will be zero and all other node voltages are referenced to it. We can use either the analog ground (AGND) or the earth ground (EGND) part from the PORT.slb library shown in Fig. 4.25. Get this part and place it at the bottom of the schematic as shown in Fig. 4.29. Make sure that the part touches the bottom wire in the diagram so that the node dot appears in your schematic. The wiring is now completed.

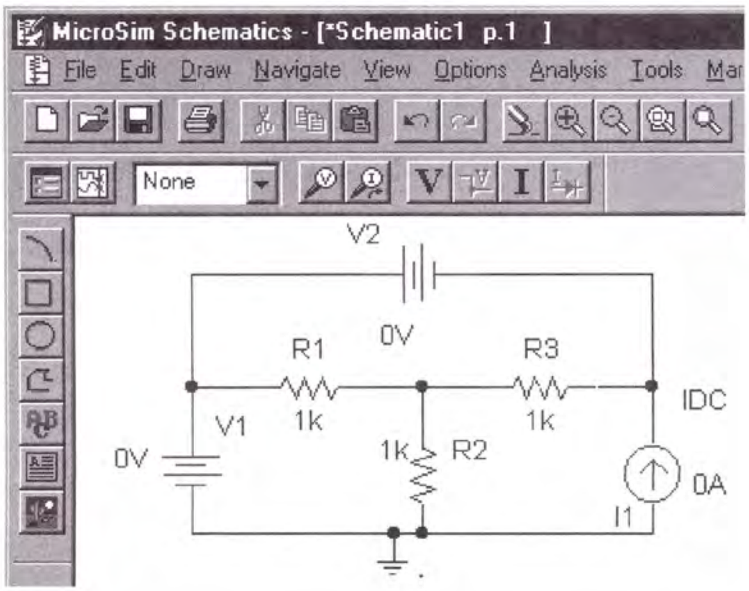


Figure 4.29
Schematic with all parts and wiring completed.

Step 2: Changing Component Names and Values

To change the name of R1 to Ra, double-click on the text "R1." The **Edit Reference Designator** dialog box appears as shown in Fig. 4.30. Simply type in the new name for the resistor, **Ra**, and select **OK**. Next we will change the resistor's value by double-clicking on the value, "1k." Now the **Attributes** dialog box in Fig. 4.31 appears. Type in **2k** and select **OK**. In a similar manner, edit the names and values of the other parts. The circuit shown in Fig. 4.32 is now ready to be saved.

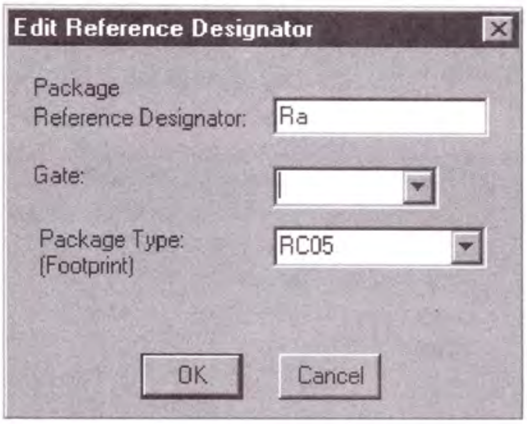


Figure 4.30
Changing the name of R1 to Ra.

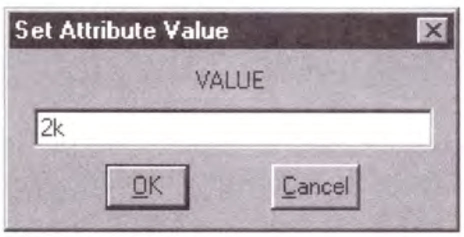


Figure 4.31
Changing the value of Ra from 1k to 2k.

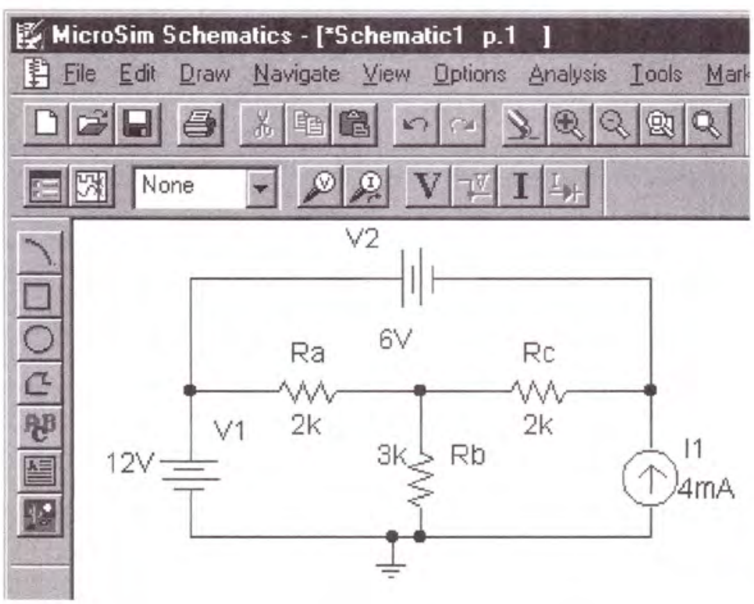
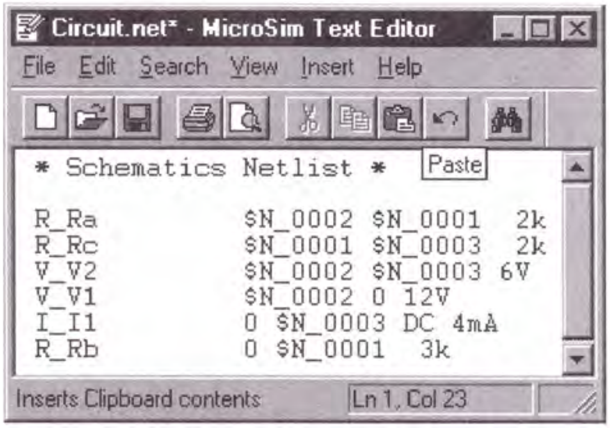


Figure 4.32
The finished schematic, ready for simulation.

Step 3: Saving the Schematic

To save the schematic, simply go to the **File** menu and select **Save**. All *Schematics* files are automatically given the extension .sch.

The Netlist The netlist is the old-fashioned SPICE code listing for circuit diagrams drawn in *Schematics*. A netlist can be created directly or as part of the simulation process. To create the netlist directly, go to the **Analysis** menu in Fig. 4.22 and select **Create Netlist**. You should receive either the message *Netlist Created* or a dialog box informing you of netlist errors. To view the netlist, return to the **Analysis** menu and select **Examine Netlist**, which opens the file shown in Fig. 4.33. All six elements appear along with their proper values. The text $N_001 $N_002 and so on are the node numbers that *Schematics* created when it converted the diagram to a netlist.




```
* Schematics Netlist *

R_Ra        $N_0002 $N_0001  2k
R_Rc        $N_0001 $N_0003  2k
V_V2        $N_0002 $N_0003 6V
V_V1        $N_0002 0 12V
I_I1        0 $N_0003 DC 4mA
R_Rb        0 $N_0001  3k
```

Figure 4.33
The netlist for our circuit.

By tracing through the node numbers we can be assured that our circuit is properly connected. The source V1 is 12 volts positive at node 2 with respect to node 0. EGND is always node number zero. Ra connects node 2 to node 1 and so forth.

How did PSPICE generate these node numbers in this particular order? In PSPICE, the terminals of a part symbol are called pins and are numbered. For example, the pin numbers for resistor, dc voltage, and current source symbols are shown in Fig. 4.34. None of the parts in Fig. 4.34 have been rotated. When the **Get New Part** sequence generates a part with a horizontal orientation, like the resistor, pin 1 is on the left. All vertically oriented parts (the sources) have pin 1 at the top. In the netlist, the order of the node numbers is always pin 1 then pin 2 for each component. Furthermore, when a part is rotated, the pin numbers also rotate. The most critical consequence of the pin numbers is current direction. PSPICE simulations always report the current flowing into pin 1 and out of pin 2.

To change the node number order of a part in the netlist, we must rotate that part 180°. Note that the order of node numbers for Rb in Fig. 4.33 is node 0 (EGND) then node 1. Accordingly, simulation results for the current in Rb will be the negative of Io as defined in Fig. 4.21. To force the netlist to agree with Fig. 4.21, we must rotate Rb twice. To do this,

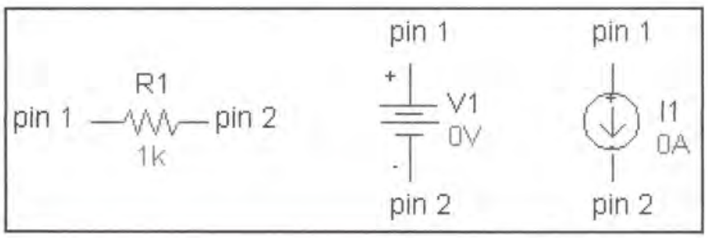


Figure 4.34
A selection of parts showing the pin-numbering format used in Schematics.

remove the wiring, click on Rb and select **Rotate** in the **Edit** menu twice, rewire and resave the file.

Step 4: Simulating the Circuit

The node voltages in the circuit are typically of interest to us, and *Schematics* permits us to identify each node with a unique name. For example, if we wish to call the node at Rb, Vo, we simply double-click on the wiring at the output node and the dialog box in Fig. 4.35 will appear. Then type **Vo** in the space shown and select **OK**.

Figure 4.35
Creating a custom node name.

Simulation results for dc node voltages and branch currents can be displayed directly on the schematic. In the **Analysis/Display Results on Schematic** menu, choose **Enable Voltage Display** and **Enable Current Display**. This will display all voltages and currents. Unwanted voltage and current displays can be deleted by selecting the data and pressing the DELETE key.

Individual node voltages can also be displayed using the VIEWPOINT (dc voltmeter) from the SPECIAL library. VIEWPOINT parts are placed at nodes and display dc node voltages with respect to ground. We will use a VIEWPOINT part to display Vo and **Analysis/Display/Enable Current Display** for the current, Io. The completed diagram will appear as shown in Fig. 4.36.

Simulation begins by choosing the type of analysis we wish to perform. This is done by selecting **Setup** from the **Analysis** menu. The SETUP dialog box is shown in Fig. 4.37. A *Schematics* dc analysis is requested by selecting **Bias Point Detail** then **Close**. The simulation results will include all node voltages, the currents through all voltage sources, and the total power dissipation. These data will be found in the output text file, accessible at **Analysis/Examine Output**. All VIEWPOINT voltages and currents will also be displayed on the schematic page.

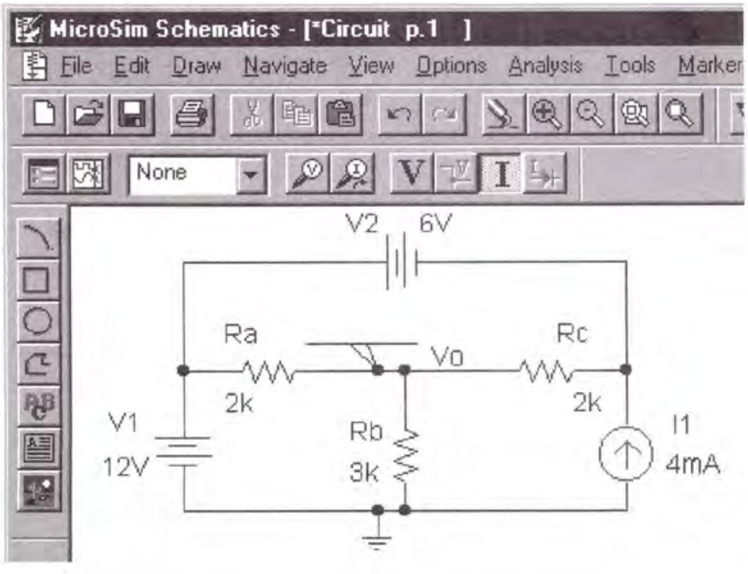


Figure 4.36
The finalized circuit, ready for simulation.

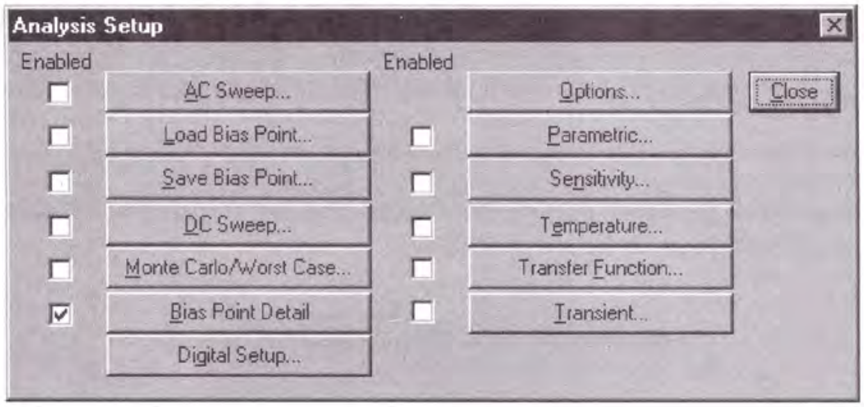


Figure 4.37
The ANALYSIS SETUP window showing the kinds of simulation we can request.

Clearly, a number of different analyses could be requested—for example, a **DC Sweep**. In this case, *Schematics* will ask for the dc source's value you wish to sweep, the start and stop values, and the increment. The simulation results will contain all dc node voltages and branch currents as a function of the varying source value. Two additional analyses, **Transient** and **AC Sweep**, will be discussed in subsequent chapters.

After exiting the SETUP dialog box, select **Simulate** from the **Analysis** menu. When the simulation is finished, your schematic should look much like that shown in Fig. 4.38, where Vo and Io are displayed automatically.

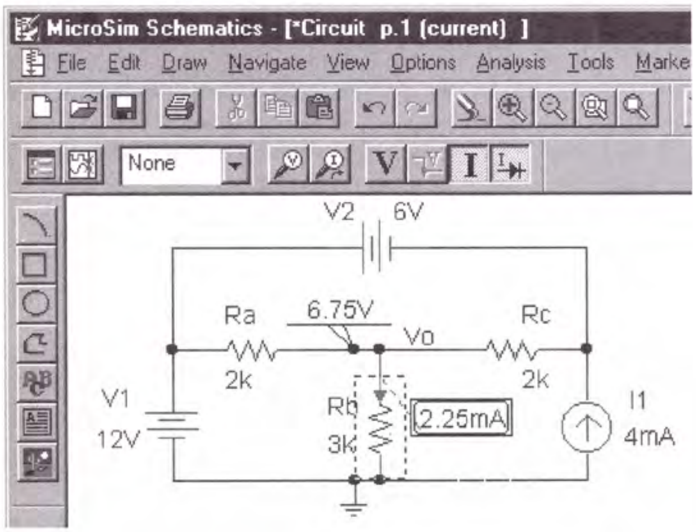


Figure 4.38
The simulation results: Vo = 6.75 V and Io = 2.25 mA.

Step 5: Viewing and Printing the Results

All node voltages, voltage source currents, and total power dissipation are contained in the output file. To view these data, select **Examine Output** from the **Analysis** menu. Within the output file is a section containing the node voltages, voltage source currents, and total power dissipation as shown in Fig. 4.39. Indeed, Vo is 6.75 V and the total power dissipation is 5.25 mW. The current bears closer inspection. PSPICE says the currents through V1 and V2 are 1.75 mA and −4.375 mA, respectively. Recall from the netlist discussion that in PSPICE, current flows from a part's pin number 1 to pin 2. As seen in Fig. 4.34, pin 1 is at the positive end of the voltage source. PSPICE is telling us that the current flowing top-down through V1 in Fig. 4.21 is 1.75 mA, while 4.375 mA flows through V2 left-to-right. Based on the passive sign convention, V1 is consuming power while V2 generates power! Since the output file is simply a text editor, the file can be edited, saved, printed, or copied and pasted to other programs.

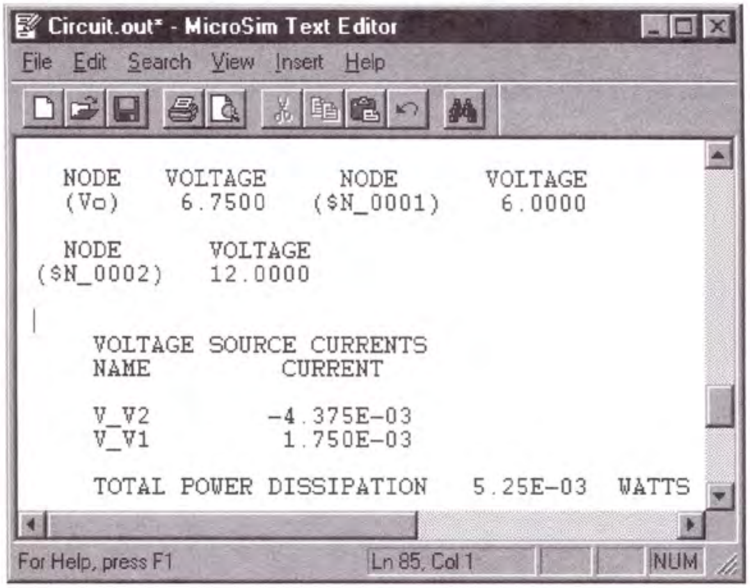


Figure 4.39
The output file simulation results.

To print the circuit diagram, point the mouse pointer above and left of the upper left hand corner of the diagram, click left, hold, and drag the mouse beyond the lower right edge of the drawing. A box will grow as you drag, eventually surrounding the circuit. Go to the **File** menu and select **Print**. The dialog box in Fig. 4.40 will appear. Select the options **Only Print Selected Area** and **User Definable Zoom Factor**. For most of the small schematics you will create, a scale of 125% to 200% will do fine. Other options are self-explanatory. Finally, select **OK** to print.

If you prefer that the grid dots not appear in the printout, go to the **Options/Display Options** menu and de-select the GRID ON option. To change the grid color, go to the **Options/Display Preferences** menu, select **Grid** and select a color from the drop-down edit box.

To incorporate your schematic into other applications such as text processors, draw a box around the diagram as described above, then under the **Edit** menu, select **Copy to Clipboard**. You can now paste the circuit into other programs.

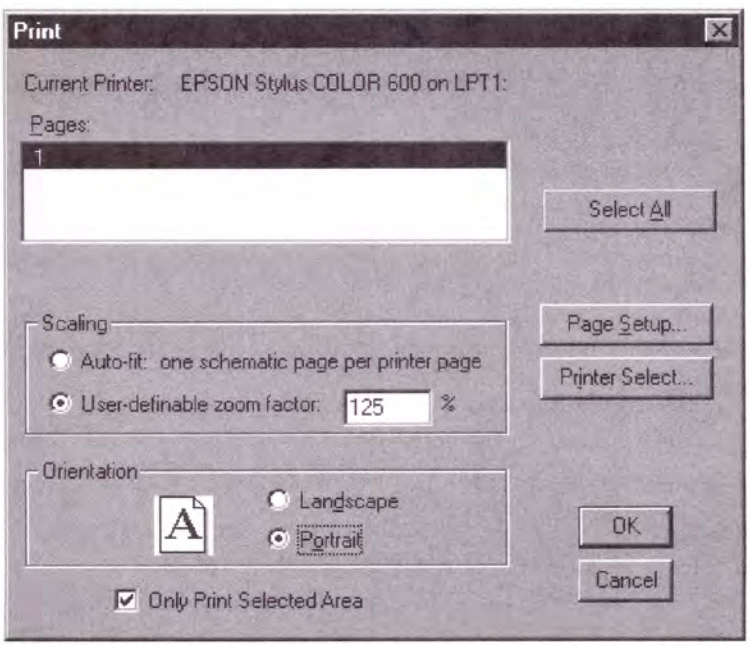


Figure 4.40
The Schematics Print window.

LEARNING Example 4.15

Let us use PSPICE to find the voltage V_o and the current I_x in the circuit in Fig. 4.41.

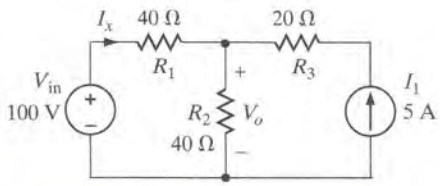


Figure 4.41 *Circuit used in Example 4.15.*

SOLUTION The PSPICE *Schematics* diagram is shown in Fig. 4.42. From the diagram, we find that $V_o = 150$ V and $I_x = -1.25$ A.

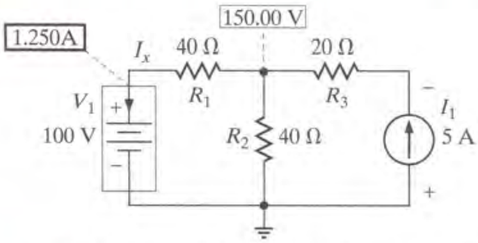


Figure 4.42 *The Schematics diagram for the network in Figure 4.41.*

Learning by Application

LEARNING Example 4.16

On Monday afternoon, Connie suddenly remembers that she has a term paper due Tuesday morning. When she sits at her computer to start typing, she discovers that the computer mouse doesn't work. After disassembly and some inspection, she finds that the mouse contains a printed circuit board that is powered by a 5-V supply contained inside the computer case. Furthermore, the board is found to contain several resistors, some op-amps, and one unidentifiable device, which is connected directly to the computer's 5-V supply as shown in Fig. 4.43a. Using a voltmeter to measure the node voltages, Connie confirms that all resistors and op-amps are functioning properly and the power supply voltage reaches the mouse board. However, without knowing the mystery device's function within the circuit, she cannot determine its condition. A phone call to the manufacturer reveals that the device is indeed linear but is also proprietary. With some persuasion, the manufacturer's representative agrees that if Connie can find the Thévenin equivalent circuit for the element at nodes *A-B* with the computer on, he will tell her if it is functioning properly. Armed with a single 1-kΩ resistor and a voltmeter, Connie attacks the problem.

SOLUTION To find the Thévenin equivalent for the unknown device, together with the 5-V source, Connie first isolates nodes *A* and *B* from the rest of the devices on the board to measure the open-circuit voltage. The resulting voltmeter reading is $V_{AB} = 2.4$ V. Thus, the Thévenin equivalent voltage is 2.4 V. Then she connects the 1-kΩ resistor at nodes *A-B* as shown in Fig. 4.43b. The voltmeter reading is now $V_{AB} = 0.8$ V. Using voltage division to express V_{AB} in terms of V_{Th}, R_{Th}, and R_{test} in Fig. 4.43b yields the expression

$$0.8 = V_{Th}\left(\frac{1k}{1k + R_{Th}}\right)$$

Solving the equations for R_{Th}, we obtain

$$R_{Th} = 2.0 \text{ k}\Omega$$

Therefore, the unknown device and the 5-V source can be represented at the terminals *A-B* by the Thévenin equivalent circuit shown in Fig. 4.43c. When Connie phones the manufacturer with the data, the representative informs her that the device has indeed failed.

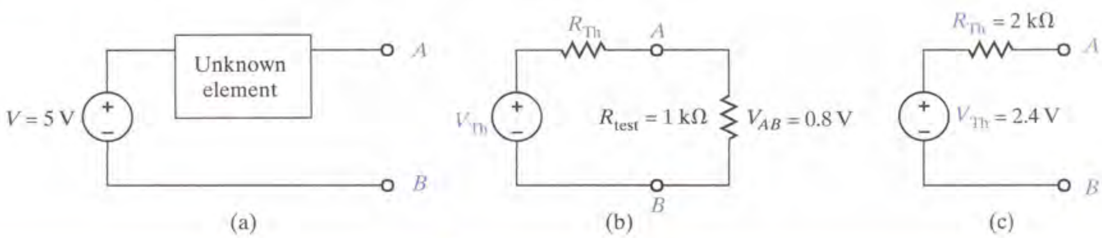


Figure 4.43 *Networks used in Example 4.16.*

Learning by Design

LEARNING Example 4.17

If the battery current in a laptop computer becomes too large, damage may occur to the system components. We wish to design an overcurrent sensor to protect a laptop from this condition.

SOLUTION We propose the circuit, shown in Fig. 4.44, to monitor the battery current and output a control signal, OVER, that can be used to disconnect the battery. The major components employed are a differential amplifier, a comparator, and a resistor used to sense the battery current that supplies the laptop components. Under normal operating conditions, the control signal voltage is low, typically zero, but if the battery current exceeds a predetermined trip point, this voltage becomes high, near V_{batt}. The comparator circuit will compare the sensed voltage to a dc reference, V_{trip}, to generate the overcurrent signal. Since the power lost in R_{sense} is not available to the computer components, V_{sense} must be small. However, to provide a reasonably sized voltage to the comparator, we will amplify V_{sense} before comparing it to V_{trip}. A differential amplifier is employed because V_{sense} is not referenced to ground. When this amplifier output, V_A, exceeds V_{trip}, the comparator output will be approximately equal to the dc supply voltage, V_{batt}.

Using a typical laptop battery voltage of 12 V and a maximum power consumption of about 54 W yields a corresponding battery current of 4.5 A. If we select the current value for the trip point to be 9 A, and further select the reference voltage to be halfway between the battery voltage and ground, that is, $V_{trip} = 6$ V, then the resistors R_A and R_B form a voltage divider and each is chosen to be 10 kΩ. Next, if we choose R_{sense} such that the power it consumes during normal operation is 0.5% of the total power, then

$$R_{sense} = \frac{(0.005)P}{I_{batt}^2} = \frac{(0.005)(54)}{4.5^2} = 13.3 \text{ m}\Omega$$

Ohm's law relates V_{sense} at the trip point to the trip point current, and thus

$$V_{sense} = (I_{trip})(R_{sense}) = (9)(0.0133) = 0.12 \text{ V}$$

Since, at the trip point, $V_A = V_{trip} = 6$ V, V_{sense} must be amplified by $6/0.12 = 50$. As indicated earlier, the gain of a differential amplifier is R_2/R_1. If we select $R_1 = 1$ kΩ, then $R_2 = 50$ kΩ will produce the proper gain. A PSPICE plot of simulated overvoltage versus battery current is shown in Fig. 4.45. Note that OVER is low until $I_{batt} = 9$ A, at which point OVER goes high. Clearly, other circuitry is needed to interpret the OVER output signal, but the design of the sensing circuitry has been completed.

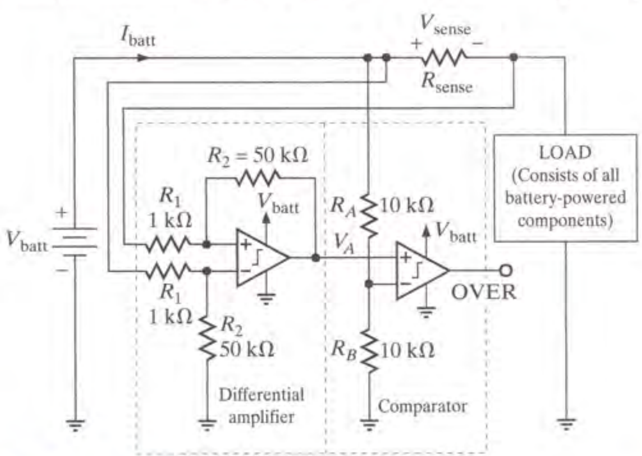


Figure 4.44 A simple overcurrent sensor circuit.

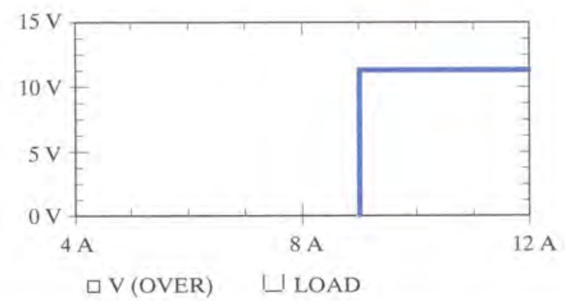


Figure 4.45 Simulation results of the overcurrent sensor circuit in Figure 4.44.

LEARNING Example 4.18

A chemical sensor outputs a voltage proportional to carbon monoxide concentration. The defining equation is

$$V_S = C/250$$

where C is carbon monoxide in ppm. In a particular application, the concentration range of interest is 125 to 250 ppm, which corresponds to a sensor voltage of 0.5 to 1 V. We want to expand this range to produce a new voltage, V_o, between 0 and 5 V. Let us design a circuit that performs this conversion.

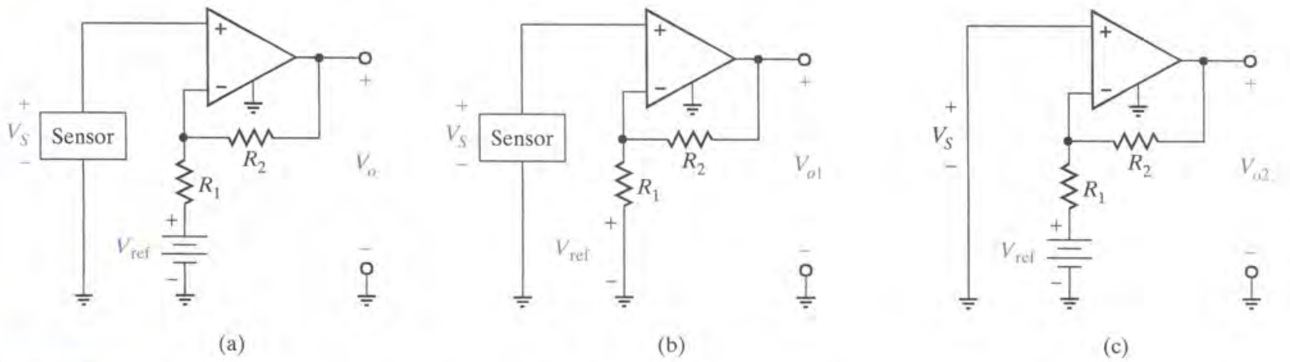


Figure 4.46
(a) The general level shifter. Superposition subcircuits for contributions of (b) V_S and (c) V_{ref}.

SOLUTION The linear relationship between V_o and V_S is of the form

$$V_o = mV_S + b$$

the parameters are set such that a concentration of 125 ppm produces $V_S = 0.5$ V and $V_o = 0$ V, while 250 ppm yields $V_S = 1$ V and $V_o = 5$ V, then

$$0 = m(0.5) + b \qquad \text{and} \qquad 5 = m(1.0) + b$$

Solving for m and b, we find $m = 10$ and $b = -5$. Therefore,

$$V_o = 10V_S - 5 \tag{4.9}$$

Now consider the op-amp circuit in Fig. 4.46a, called a level shifter. Using superposition, the output voltage can be easily determined. First, we find the contribution of V_S to V_o by setting V_{ref} to zero as shown in Fig. 4.46b. This is the classic noninverting gain stage, where

$$V_{o1} = \left[1 + \frac{R_2}{R_1}\right]V_S$$

Next, we find the affect of V_{ref} by setting V_S to zero. This yields the classic inverting gain stage, shown in Fig. 4.46c, where

$$V_{o2} = -\left[\frac{R_2}{R_1}\right]V_{ref}$$

The output voltage is the sum of these contributions

$$V_o = \left[1 + \frac{R_2}{R_1}\right]V_S - \left[\frac{R_2}{R_1}\right]V_{ref} \tag{4.10}$$

Comparing Eqs. (4.9) and (4.10), we see that

$$1 + \frac{R_2}{R_1} = 10 \qquad \frac{R_2}{R_1} = 9 \qquad V_{ref} = \frac{5}{9}\text{ V}$$

Arbitrarily choosing $R_1 = 10$ kΩ yields $R_2 = 90$ kΩ. Voltage division can produce V_{ref} from a 5-V supply voltage. The final circuit is shown in Fig. 4.47, where the unity gain buffer isolates the voltage division resistor string from the level shifter.

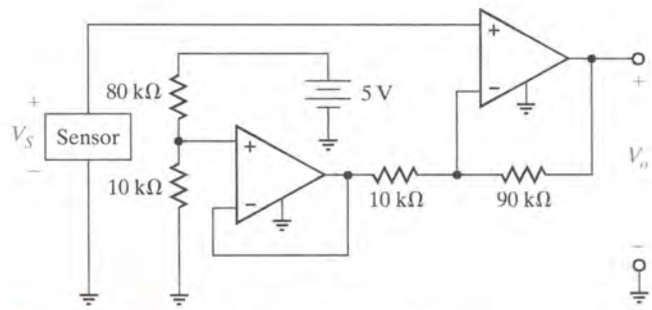


Figure 4.47 *The final level shifter circuit.*

LEARNING Check

Summary

- Linearity: This property requires both additivity and homogeneity. Using this property, the voltage or current somewhere in a network can be determined by assuming a specific value for the variable and then determining what source value is required to produce it. The ratio of the specified source value to that computed from the assumed value of the variable, together with the assumed value of the variable, can be used to obtain a solution.

- In a linear network containing multiple independent sources, the principle of superposition allows us to compute any current or voltage in the network as the algebraic sum of the individual contributions of each source acting alone.
- Superposition is a linear property and does not apply to nonlinear functions such as power.
- Using Thévenin's theorem, we can replace some portion of a network at a pair of terminals with a voltage source V_{oc} in series with a resistor R_{Th}. V_{oc} is the open-circuit voltage at the terminals, and R_{Th} is the Thévenin equivalent resistance obtained by looking into the terminals with all independent sources made zero.
- Using Norton's theorem, we can replace some portion of a network at a pair of terminals with a current source I_{sc} in parallel with a resistor R_{Th}. I_{sc} is the short-circuit current at the terminals and R_{Th} is the Thévenin equivalent resistance.
- Source transformation permits us to replace a voltage source V in series with a resistance R by a current source $I = V/R$ in parallel with the resistance R. The reverse is also true.

  This is an interchange relationship between Thévenin and Norton equivalent circuits.
- Maximum power transfer can be achieved by selecting the load R_L to be equal to R_{Th} found by looking into the network from the load terminals.
- dc PSPICE with Schematic Capture is an effective tool in analyzing dc circuits.

Problems

For solutions and additional help on problems marked with ► go to www.wiley.com/college/irwin

SECTION 4.1

4.1 ► **Find I_o in the circuit in Fig. P4.1 using linearity and the assumption that $I_o = 1$ mA.**

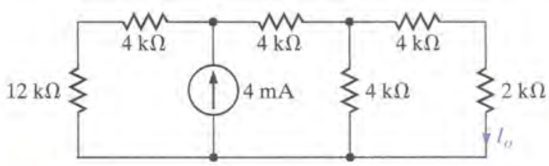


Figure P4.1

4.2 Find I_o in the network in Fig. P4.2 using linearity and the assumption that $I_o = 1$ mA.

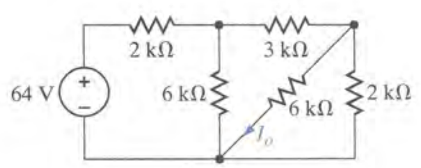


Figure P4.2

4.3 Find V_o in the network in Fig. P4.3 using linearity and the assumption that $V_o = 1$ mV.

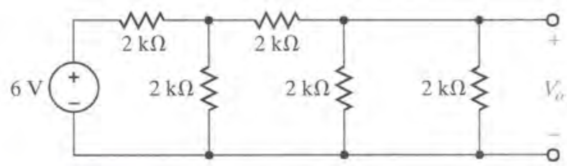


Figure P4.3

4.4 Find V_o in the circuit in Fig. P4.4 using linearity and the assumption that $V_o = 1$ V.

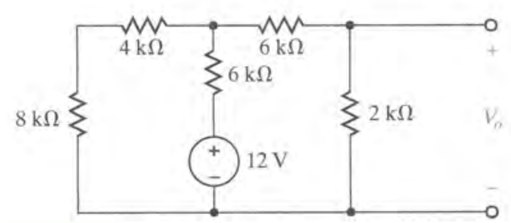


Figure P4.4

SECTION 4.2

4.5 In the network in Fig. P4.5, find I_o using superposition.

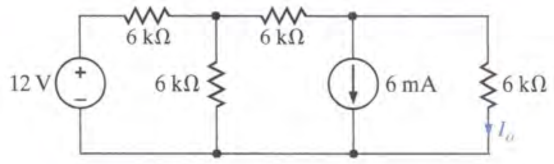


Figure P4.5

4.6 Find I_o in the circuit in Fig. P4.6 using superposition.

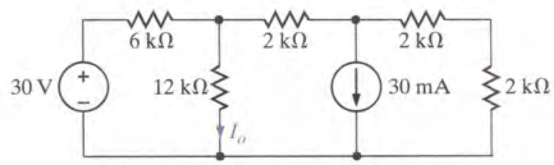


Figure P4.6

4.7 In the network in Fig. P4.7, find V_o using superposition.

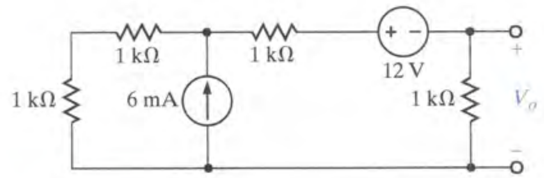


Figure P4.7

4.8 Find V_o in the network in Fig. P4.8 using superposition.

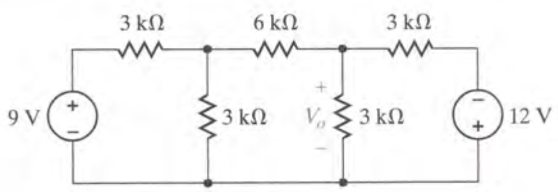


Figure P4.8

4.9 Find I_o in the network in Fig. P4.9 using superposition.

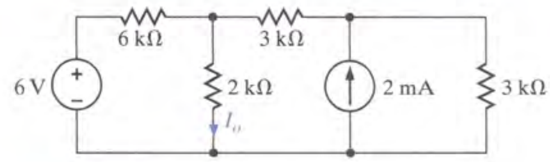


Figure P4.9

4.10 Find I_o in the network in Fig. P4.10 using superposition.

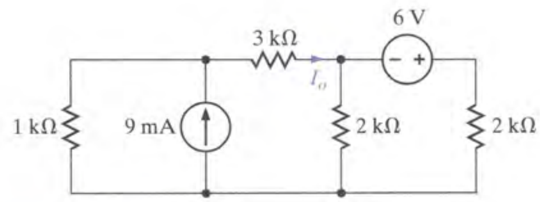


Figure P4.10

4.11 Find I_o in the network in Fig. P4.11 using superposition.

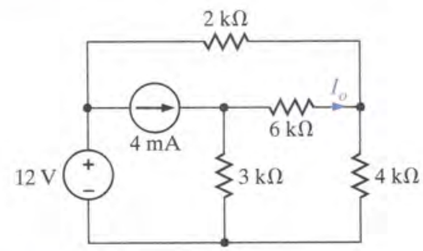


Figure P4.11

4.12 Find I_o in the network in Fig. P4.12 using superposition.

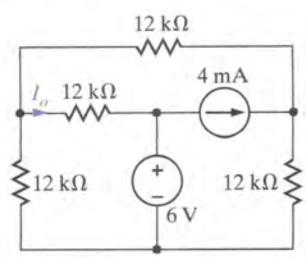


Figure P4.12

4.13 **Find I_o in the circuit in Fig. P4.13 using superposition.**

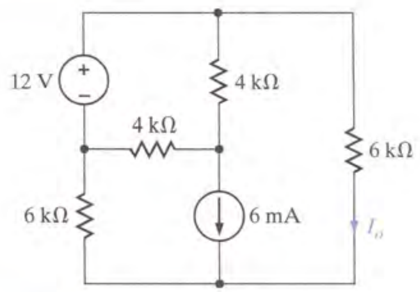


Figure P4.13

4.14 Use superposition to find I_o in the circuit in Fig. P4.14.

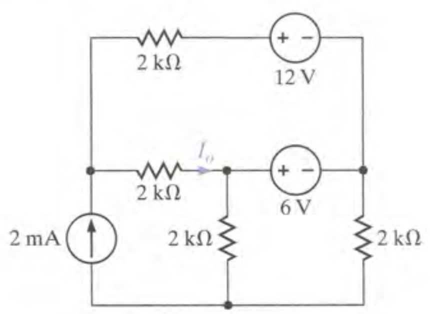


Figure P4.14

4.15 Find I_o in the network in Fig. P4.15 using superposition.

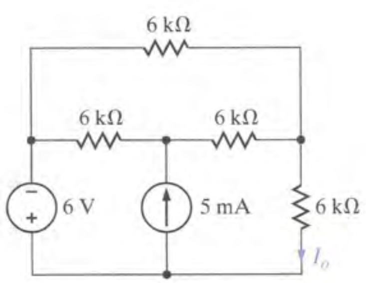


Figure P4.15

4.16 Find I_o in the network in Fig. P4.16 using superposition.

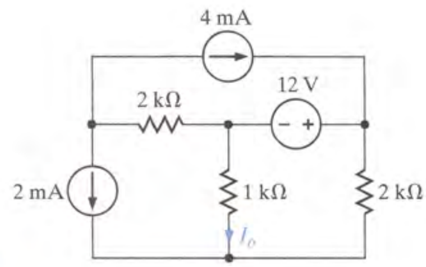


Figure P4.16

4.17 **Find I_o in the network in Fig. P4.17 using superposition.**

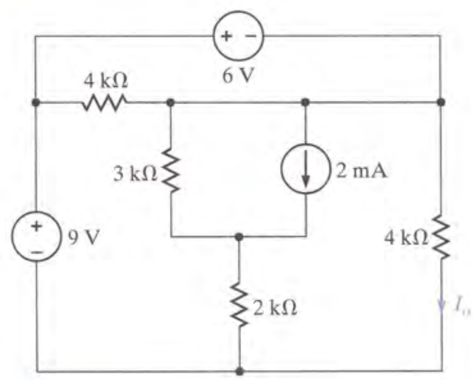


Figure P4.17

4.18 Find V_o in the network in Fig. P4.18 using superposition.

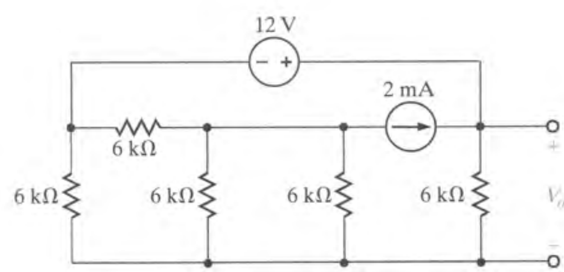


Figure P4.18

SECTION 4.3

4.19 Use source transformation to find I_o in the circuit in Fig. P4.19.

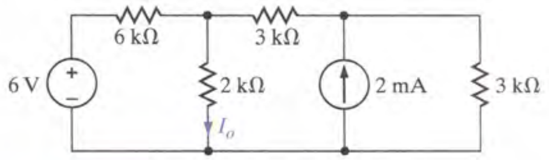


Figure P4.19

4.20 Find V_o in the network in Fig. P4.20 using source transformation.

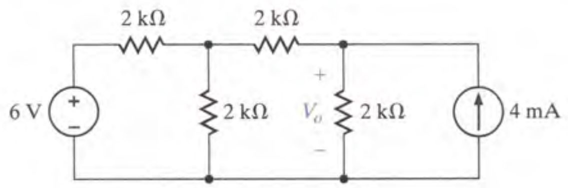


Figure P4.20

4.21 ► **Use source transformation to find V_o in the network in Fig. P4.21.**

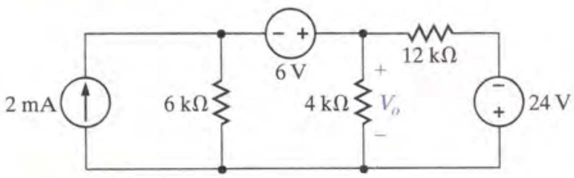


Figure P4.21

4.22 Find V_o in the network in Fig. P4.22 using source transformation.

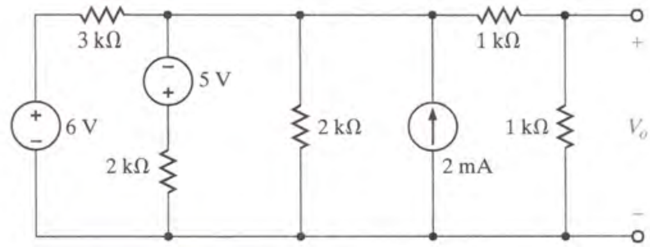


Figure P4.22

4.23 Find I_o in the circuit in Fig. P4.23 using source transformation.

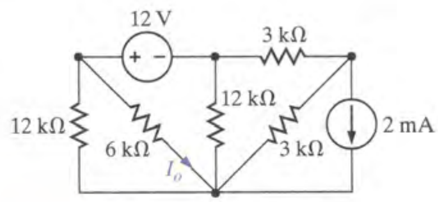


Figure P4.23

4.24 Find I_o in the network in Fig. P4.24 using source transformation.

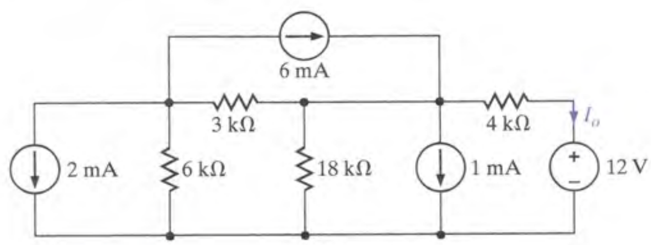


Figure P4.24

4.25 ► **Use source transformation to find I_o in the circuit in Fig. P4.25.**

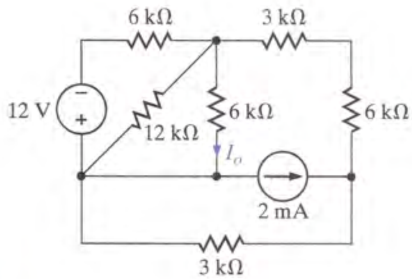


Figure P4.25

4.26 Find V_o in the network in Fig. P4.26 using source transformation.

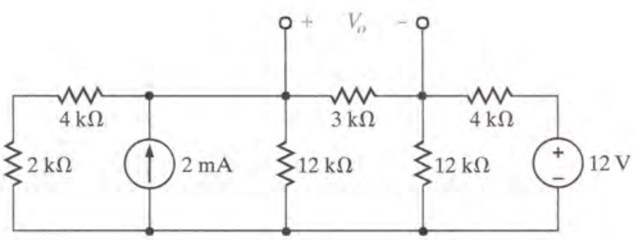


Figure P4.26

4.27 Find I_o in the circuit in Fig. P4.27 using source transformation.

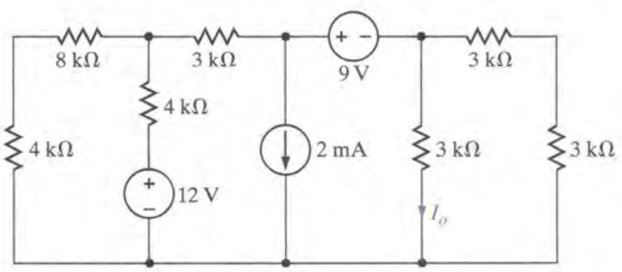


Figure P4.27

4.28 Find I_o in the network in Fig. P4.28 using source transformation.

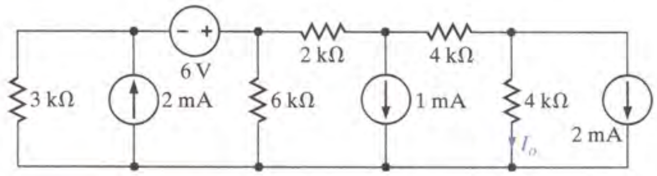


Figure P4.28

4.29 ► **Find I_o in the network in Fig. P4.29 using source transformation.**

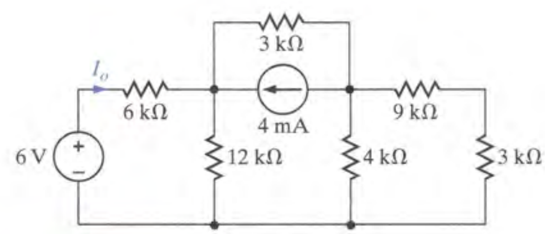


Figure P4.29

4.30 Find I_o in the network in Fig. P4.30 using source transformation.

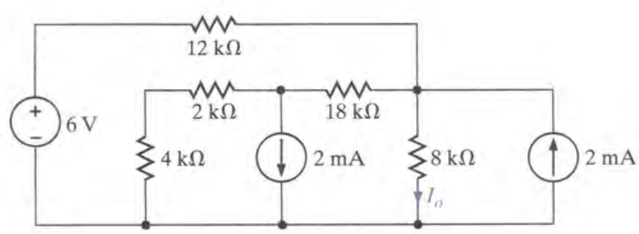


Figure P4.30

4.31 Find I_o in the network in Fig. P4.31 using source transformation.

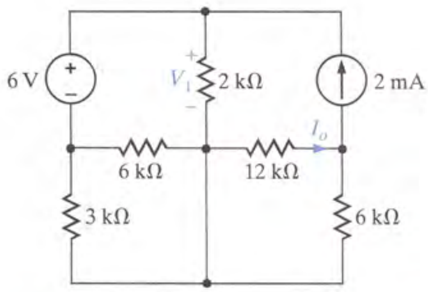


Figure P4.31

4.32 Use Thévenin's theorem to find V_o in the network in Fig. P4.32.

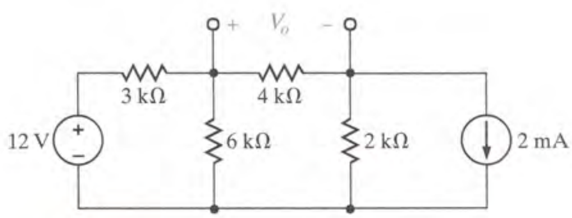


Figure P4.32

4.33 **Use Thévenin's theorem to find V_o in the network in Fig. P4.33.**

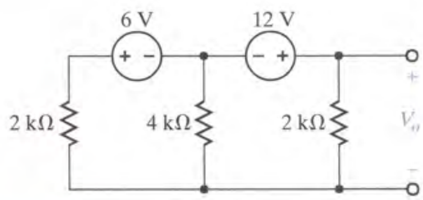


Figure P4.33

4.34 Use Thévenin's theorem to find I_o in the network in Fig. P4.34.

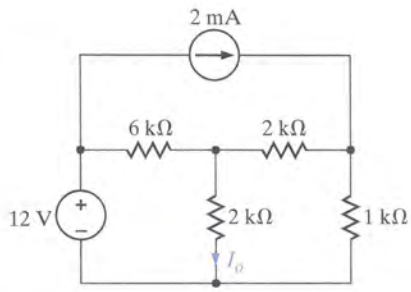


Figure P4.34

4.35 Find I_o in the network in Fig. P4.35 using Thévenin's theorem.

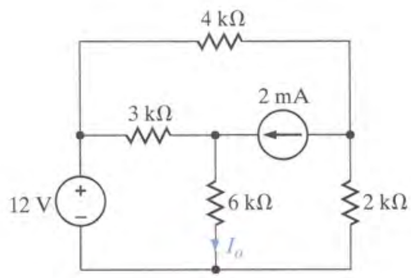


Figure P4.35

4.36 Find V_o in the circuit in Fig. P4.36 using Thévenin's theorem.

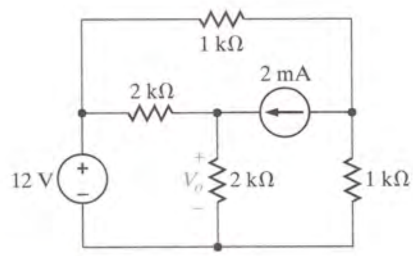


Figure P4.36

4.37 ► **Find I_o in the circuit in Fig. P4.37 using Thévenin's theorem.**

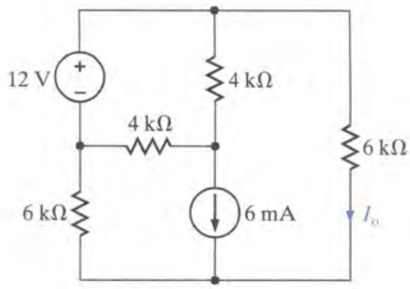


Figure P4.37

4.38 Find I_o in the network in Fig. P4.38 using Thévenin's theorem.

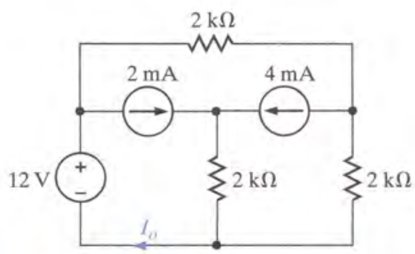


Figure P4.38

4.39 Find V_o in the circuit in Fig. P4.39 using Thévenin's theorem.

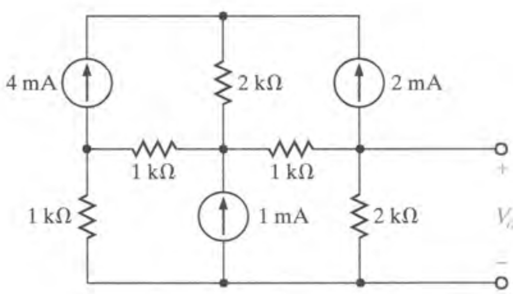


Figure P4.39

4.40 Find V_o in the circuit in Fig. P4.40 using Thévenin's theorem.

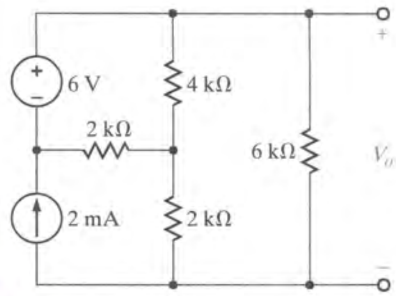


Figure P4.40

4.41 ► **Find I_o in the network in Fig. P4.41 using Thévenin's theorem.**

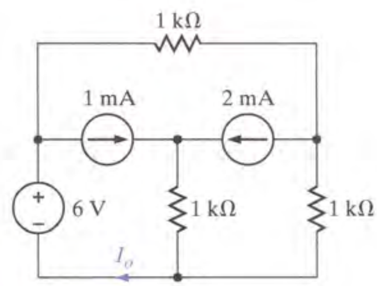


Figure P4.41

4.42 Find I_o in the network in Fig. P4.42 using Thévenin's theorem.

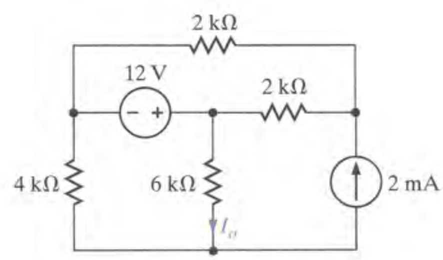


Figure P4.42

4.43 Find I_o in the network in Fig. P4.43 using Thévenin's theorem.

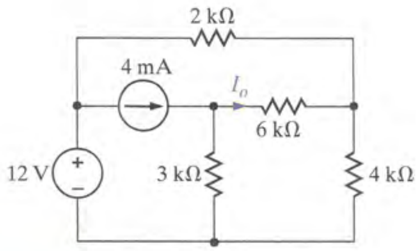


Figure P4.43

4.44 Find I_o in the network in Fig. P4.44 using Thévenin's theorem.

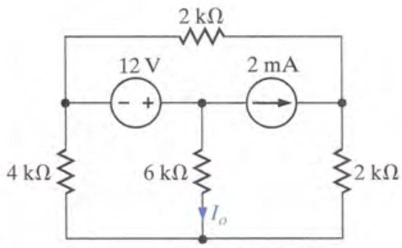


Figure P4.44

4.45 **Find V_o in the network in Fig. P4.45 using Thévenin's theorem.**

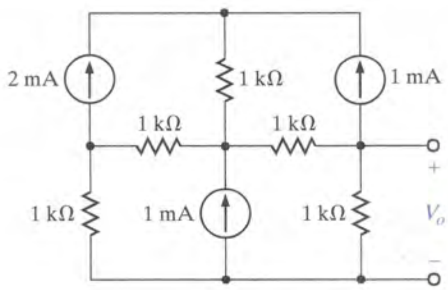


Figure P4.45

4.46 Find V_o in the network in Fig. P4.46 using Thévenin's theorem.

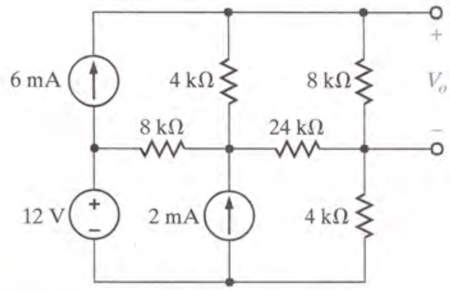


Figure P4.46

4.47 Use a combination of Thévenin's theorem and superposition to find V_o in the circuit in Fig. P4.47.

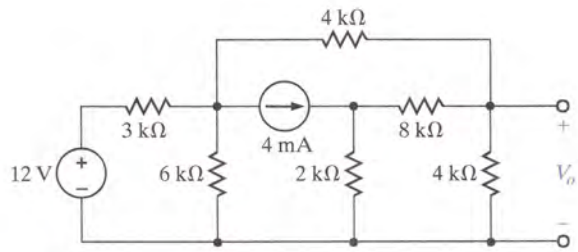


Figure P4.47

4.48 In the network in Fig. P4.48, find V_o using Thévenin's theorem.

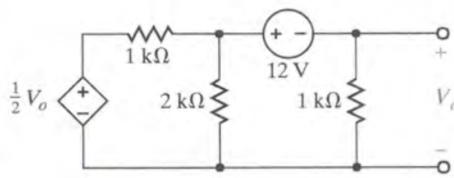


Figure P4.48

4.49 Find the Thévenin equivalent of the network in Fig. P4.49 at the terminals *A-B*.

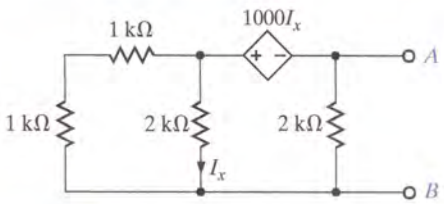


Figure P4.49

4.50 Find the Thévenin equivalent of the network in Fig. P4.50 at the terminals *A-B*.

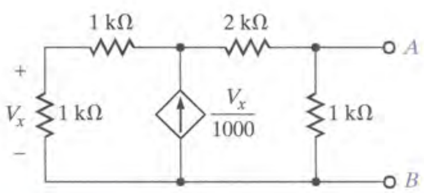


Figure P4.50

4.51 Find V_o in the network in Fig. P4.51 using Thévenin's theorem.

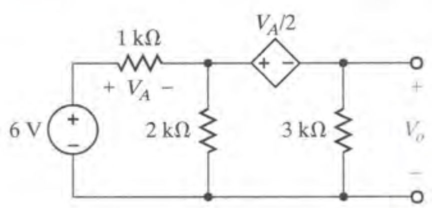


Figure P4.51

4.52 Find V_o in the network in Fig. P4.52 using Thévenin's theorem.

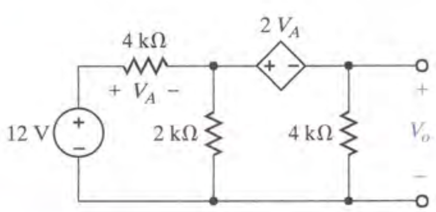


Figure P4.52

4.53 Find V_o in the circuit in Fig. P4.53 using Thévenin's theorem.

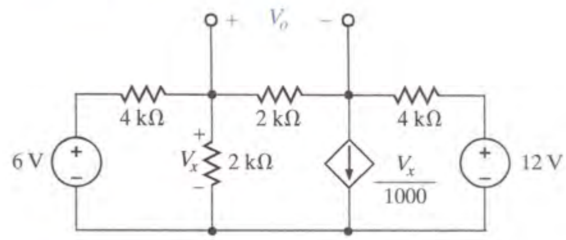


Figure P4.53

4.54 Use Thévenin's theorem to find V_o in the circuit in Fig. P4.54.

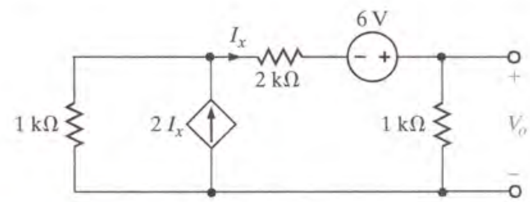


Figure P4.54

4.55 Use Thévenin's theorem to find I_o in the circuit in Fig. P4.55.

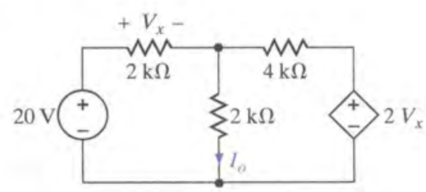


Figure P4.55

4.56 Find V_o in the network in Fig. P4.56 using Thévenin's theorem.

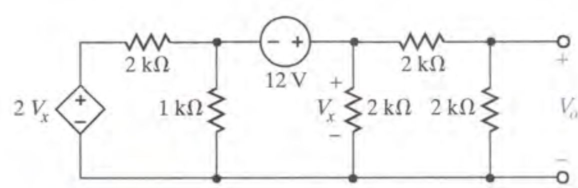


Figure P4.56

4.57 ► **Use Thévenin's theorem to find V_o in the network in Fig. P4.57.**

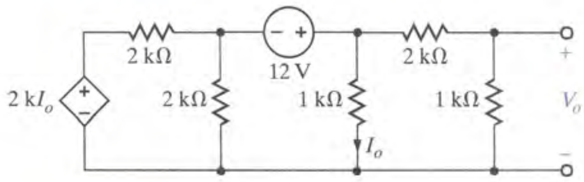


Figure P4.57

4.58 Find V_o in the network in Fig. P4.58 using Thévenin's theorem.

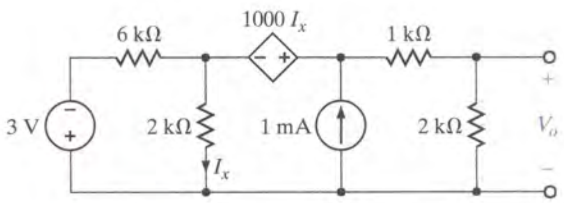


Figure P4.58

4.59 Use Thévenin's theorem to find V_o in the circuit in Fig. P4.59.

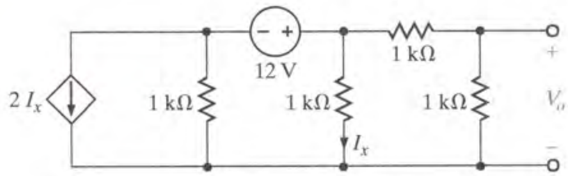


Figure P4.59

4.60 Find V_o in the network in Fig. P4.60 using Thévenin's theorem.

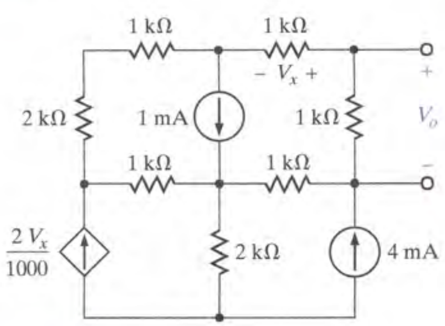


Figure P4.60

4.61 ► **Use Norton's theorem to find V_o in the network in Fig. P4.61.**

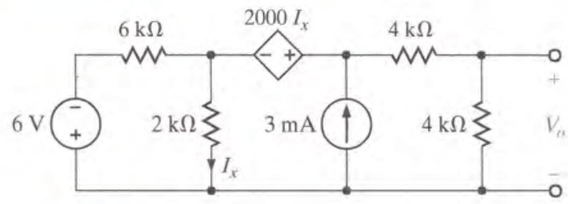


Figure P4.61

4.62 Find I_o in the network in Fig. P4.62 using Norton's theorem.

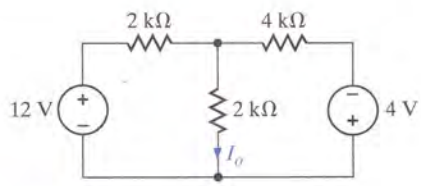


Figure P4.62

4.63 Use Norton's theorem to find I_o in the circuit in Fig. P4.63.

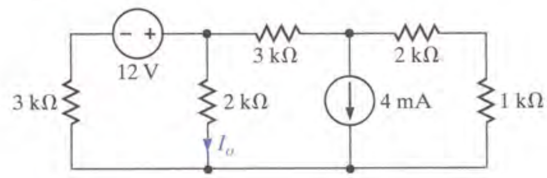


Figure P4.63

4.64 Find I_o in the network in Fig. P4.64 using Norton's theorem.

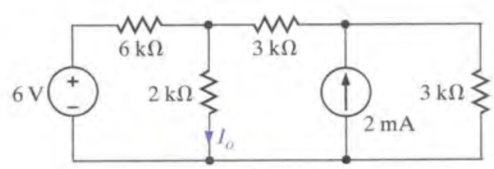


Figure P4.64

4.65 **Find I_o in the network in Fig. P4.65 using Norton's theorem.**

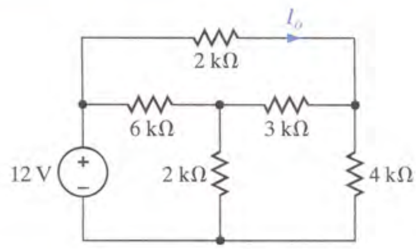


Figure P4.65

4.66 Use Norton's theorem to find V_o in the network in Fig. P4.66.

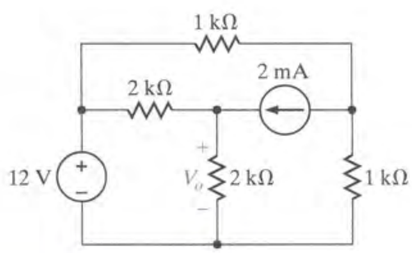


Figure P4.66

4.67 Find V_o in the network in Fig. P4.67 using Norton's theorem.

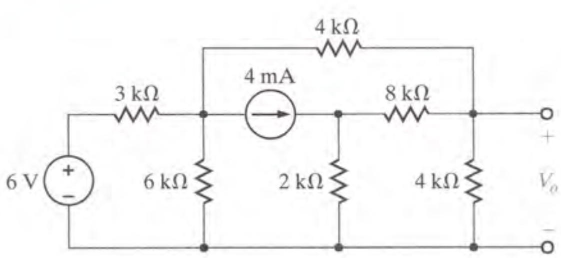


Figure P4.67

4.68 Given the linear circuit in Fig. P4.68, it is known that when a 2-kΩ load is connected to the terminals A-B, the load current is 10 mA. If a 10-kΩ load is connected to the terminals, the load current is 6 mA. Find the current in a 20-kΩ load.

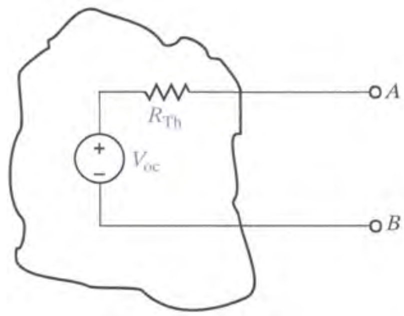


Figure P4.68

4.69 **If an 8-kΩ load is connected to the terminals of the network in Fig. P4.69, $V_{AB} = 16$ V. If a 2-kΩ load is connected to the terminals, $V_{AB} = 8$ V. Find V_{AB} if a 20-kΩ load is connected to the terminals.**

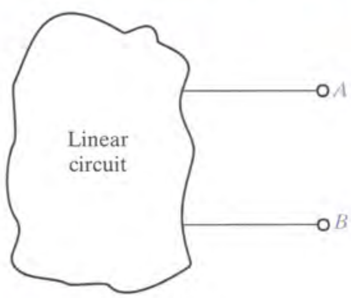


Figure P4.69

SECTION 4.4

4.70 Find R_L for maximum power transfer and the maximum power that can be transferred in the network in Fig. P4.70.

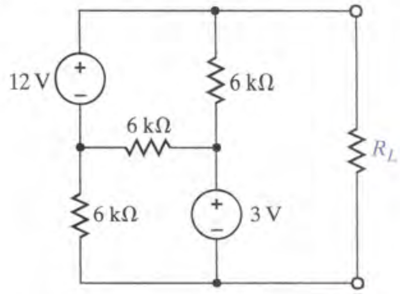


Figure P4.70

4.71 Find R_L for maximum power transfer and the maximum power that can be transferred in the network in Fig. P4.71.

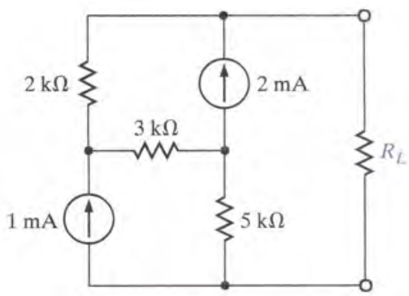


Figure P4.71

4.72 In the network in Fig. P4.72, find R_L for maximum power transfer and the maximum power that can be transferred to this load.

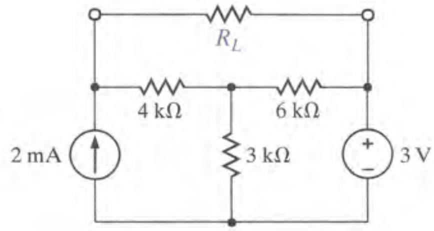


Figure P4.72

4.73 Find R_L for maximum power transfer and the maximum power that can be transferred in the network in Fig. P4.73.

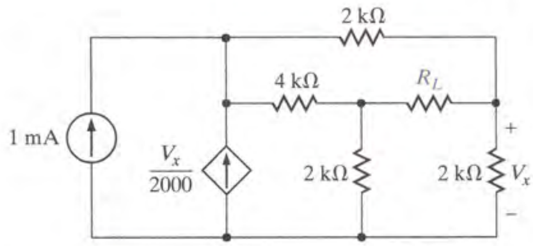


Figure P4.73

SECTION 4.5

4.74 Solve problem 4.5 using PSPICE.

4.75 Solve problem 4.21 using PSPICE.

4.76 Solve problem 4.32 using PSPICE.

4.77 Solve problem 4.36 using PSPICE.

Typical Problems Found on the FE Exam

4FE-1 Determine the maximum power that can be delivered to the load R_L in the network in Fig. 4PFE-1.

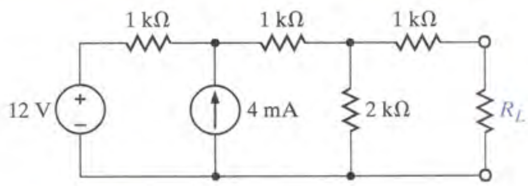


Fig. 4PFE-1

4FE-2 Find the value of the load R_L in the network in Fig. 4PFE-2 that will achieve maximum power transfer, and determine the value of the maximum power.

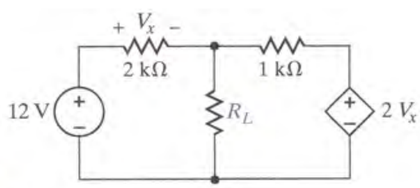


Fig. 4PFE-2

4FE-3 Find the value of R_L in the network in Fig. 4PFE-3 for maximum power transfer to this load.

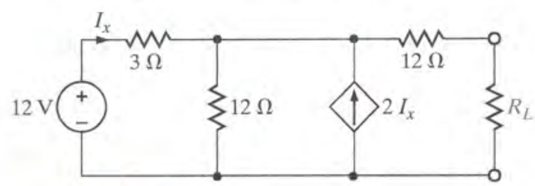


Fig. 4PFE-3

Capacitance and Inductance

5

Have you ever wondered how a tiny camera battery is able to produce a blinding flash or how a hand-held "stun gun" can deliver 50,000 V? The answer is energy storage, and in this chapter we introduce two elements that possess this property: the capacitor and the inductor. Both capacitors and inductors are linear elements; however, unlike the resistor, their terminal characteristics are described by linear differential equations. Another distinctive feature of these elements is their ability to absorb energy from the circuit, store it temporarily, and later return it. Elements that possess this energy storage capability are referred to simply as *storage elements*.

Capacitors are capable of storing energy when a voltage is present across the element. The energy is actually stored in an electric field not unlike that produced by sliding across a car seat on a dry winter day. Conversely, inductors are capable of storing energy when a current is passing through them, causing a magnetic field to form. This phenomenon can be demonstrated by placing a needle compass in the vicinity of a current. The current causes a magnetic field whose energy deflects the compass needle.

A very important circuit, which employs a capacitor in a vital role, is also introduced. This circuit, known as an op-amp integrator, produces an output voltage that is proportional to the integral of the input voltage. The significance of this circuit is that any system (for example, electrical, mechanical, hydraulic, biological, social, economic, and so on) that can be described by a set of linear differential equations with constant coefficients can be modeled by a network consisting of op-amp integrators. Thus, very complex and costly systems can be tested safely and inexpensively prior to construction and implementation.

Finally, we examine some practical circuits where capacitors and inductors are normally found or can be effectively used in circuit design.

LEARNING Goals

5.1 Capacitors

LEARNING Hint

Note the use of the passive sign convention.

A *capacitor* is a circuit element that consists of two conducting surfaces separated by a nonconducting, or *dielectric*, material. A simplified capacitor and its electrical symbol are shown in Fig. 5.1.

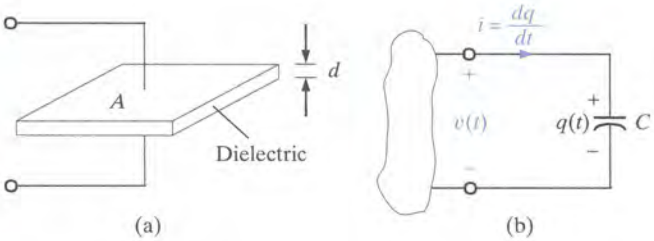


Figure 5.1 Capacitor and its electrical symbol.

There are many different kinds of capacitors, and they are categorized by the type of dielectric material that is used between the conducting plates. Although any good insulator can serve as a dielectric, each type has characteristics that make it more suitable for particular applications.

For general applications in electronic circuits (e.g., coupling between stages of amplification) the dielectric material may be paper impregnated with oil or wax, mylar, polystyrene, mica, glass, or ceramic.

Ceramic dielectric capacitors constructed of barium titanates have a large capacitance-to-volume ratio because of their high dielectric constant. Mica, glass, and ceramic dielectric capacitors will operate satisfactorily at high frequencies.

Aluminum electrolytic capacitors, which consist of a pair of aluminum plates separated by a moistened borax paste electrolyte, can provide high values of capacitance in small volumes. They are typically used for filtering, bypassing, and coupling, and in power supplies and motor-starting applications. Tantalum electrolytic capacitors have lower losses and more stable characteristics than those of aluminum electrolytic capacitors. Figure 5.2 shows a variety of typical discrete capacitors.

Figure 5.2 Some typical capacitors (courtesy of Cornell Dubilier).

In addition to these capacitors, which we deliberately insert in a network for specific applications, stray capacitance is present any time there is a difference in potential between two conducting materials separated by a dielectric. Because this stray capacitance can cause unwanted coupling between circuits, extreme care must be exercised in the layout of electronic systems on printed circuit boards.

Capacitance is measured in coulombs per volt or farads. The unit *farad* (F) is named after Michael Faraday, a famous English physicist. Capacitors may be fixed or variable and typically range from thousands of microfarads (μF) to a few picofarads (pF).

Figure 5.3
A 55-F, 500-J capacitor and a 10-in ruler for comparison.

However, capacitor technology, initially driven by the modern interest in electric vehicles, is rapidly changing. For example, the capacitor in the photograph in Fig. 5.3 was designed and built at the Space Power Institute at Auburn University to achieve high energy and high power density capacitors for application in electric vehicles and space. This capacitor, which is specially constructed, is rated at 55 F, 500 joules (J). It is interesting to calculate the dimensions of a simple equivalent capacitor consisting of two parallel plates each of area A, separated by a distance d as shown in Fig. 5.1. We learned in basic physics that the capacitance of two parallel plates of area A, separated by distance d, is

$$C = \frac{\varepsilon_o A}{d}$$

where ε_o, the permitivity of free space, is 8.85×10^{-12} F/m. If we assume the plates are separated by a distance in air of the thickness of one sheet of oil-impregnated paper, which is about 1.016×10^{-4} m, then

$$55\text{ F} = \frac{(8.85 \times 10^{-12})A}{1.016 \times 10^{-4}}$$

$$A = 6.3141 \times 10^{8}\text{ m}^2$$

and since 1 square mile is equal to 2.59×10^{6} square meters, the area is

$$A \approx 244 \text{ square miles}$$

which is the area of a medium-sized city! It would now seem that the capacitor in the photograph is much more impressive than it originally appeared. This capacitor is actually constructed using multiple layers of a combination of pressed carbon and metal fibers, which are heat-treated to adhere to metal foil. The metal foils are connected to form a stack of 18 cells, within this geometry. There are literally millions of pieces of carbon and metal fibers employed to obtain the required surface area.

Suppose now that a source is connected to the capacitor shown in Fig. 5.1; then positive charges will be transferred to one plate and negative charges to the other. The charge on the capacitor is proportional to the voltage across it such that

$$q = Cv \qquad \textbf{5.1}$$

where C is the proportionality factor known as the capacitance of the element in farads.

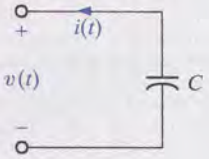
The charge differential between the plates creates an electric field that stores energy. Because of the presence of the dielectric, the conduction current that flows in the wires that connect the capacitor to the remainder of the circuit cannot flow internally between the plates. However, via electromagnetic field theory it can be shown that this conduction current is equal to the displacement current that flows between the plates of the capacitor and is present any time that an electric field or voltage varies with time.

Our primary interest is in the current–voltage terminal characteristics of the capacitor. Since the current is

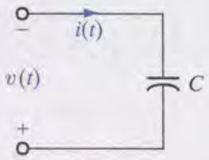
$$i = \frac{dq}{dt}$$

LEARNING by Doing

D 5.1 Write the i-v relationship for the following capacitors.



(a)



(b)

ANSWER

(a) $i(t) = -C\dfrac{dv(t)}{dt}$

(b) $i(t) = -C\dfrac{dv(t)}{dt}$

then for a capacitor

$$i = \frac{d}{dt}Cv$$

which for constant capacitance is

$$i = C\frac{dv}{dt} \tag{5.2}$$

Equation (5.2) can be rewritten as

$$dv = \frac{1}{C}i\,dt$$

Now integrating this expression from $t = -\infty$ to some time t and assuming $v(-\infty) = 0$ yields

$$v(t) = \frac{1}{C}\int_{-\infty}^{t} i(x)\,dx \tag{5.3}$$

where $v(t)$ indicates the time dependence of the voltage. Equation (5.3) can be expressed as two integrals, so that

$$\begin{aligned} v(t) &= \frac{1}{C}\int_{-\infty}^{t_0} i(x)\,dx + \frac{1}{C}\int_{t_0}^{t} i(x)\,dx \\ &= v(t_0) + \frac{1}{C}\int_{t_0}^{t} i(x)\,dx \end{aligned} \tag{5.4}$$

where $v(t_0)$ is the voltage due to the charge that accumulates on the capacitor from time $t = -\infty$ to time $t = t_0$.

The energy stored in the capacitor can be derived from the power that is delivered to the element. This power is given by the expression

$$p(t) = v(t)i(t) = Cv(t)\frac{dv(t)}{dt} \tag{5.5}$$

and hence the energy stored in the electric field is

$$\begin{aligned} w_C(t) &= \int_{-\infty}^{t} Cv(x)\frac{dv(x)}{dx}\,dx = C\int_{-\infty}^{t} v(x)\frac{dv(x)}{dx}\,dx \\ &= C\int_{v(-\infty)}^{v(t)} v(x)\,dv(x) = \tfrac{1}{2}Cv^2(x)\Big|_{v(-\infty)}^{v(t)} \\ &= \tfrac{1}{2}Cv^2(t)\text{ J} \end{aligned} \tag{5.6}$$

since $v(t = -\infty) = 0$. The expression for the energy can also be written using Eq. (5.1) as

$$w_C(t) = \frac{1}{2}\frac{q^2(t)}{C} \tag{5.7}$$

Equations (5.6) and (5.7) represent the energy stored by the capacitor, which, in turn, is equal to the work done by the source to charge the capacitor.

The polarity of the voltage across a capacitor being charged is shown in Fig. 5.1b. In the ideal case the capacitor will hold the charge for an indefinite period of time if the source is removed. If at some later time an energy-absorbing device (e.g., a flashbulb) is connected across the capacitor, a discharge current will flow from the capacitor and, therefore, the capacitor will supply its stored energy to the device.

LEARNING Example 5.1

If the charge accumulated on two parallel conductors charged to 12 V is 600 pC, what is the capacitance of the parallel conductors?

SOLUTION Using Eq. (5.1), we find that

$$C = \frac{Q}{V} = \frac{(600)(10^{-12})}{12} = 50 \text{ pF}$$

LEARNING Example 5.2

The voltage across a 5-μF capacitor has the waveform shown in Fig. 5.4a. Determine the current waveform.

SOLUTION Note that

$$\begin{aligned} v(t) &= \frac{24}{6 \times 10^{-3}} t & 0 \le t \le 6 \text{ ms} \\ &= \frac{-24}{2 \times 10^{-3}} t + 96 & 6 \le t < 8 \text{ ms} \\ &= 0 & 8 \text{ ms} \le t \end{aligned}$$

Using Eq. (5.2), we find that

$$\begin{aligned} i(t) &= C \frac{dv(t)}{dt} & \\ &= 5 \times 10^{-6}(4 \times 10^3) & 0 \le t \le 6 \text{ ms} \\ &= 20 \text{ mA} & 0 \le t \le 6 \text{ ms} \\ i(t) &= 5 \times 10^{-6}(-12 \times 10^3) & 6 \le t \le 8 \text{ ms} \\ &= -60 \text{ mA} & 6 \le t < 8 \text{ ms} \end{aligned}$$

and

$$i(t) = 0 \qquad 8 \text{ ms} \le t$$

Therefore, the current waveform is as shown in Fig. 5.4b and $i(t) = 0$ for $t > 8$ ms.

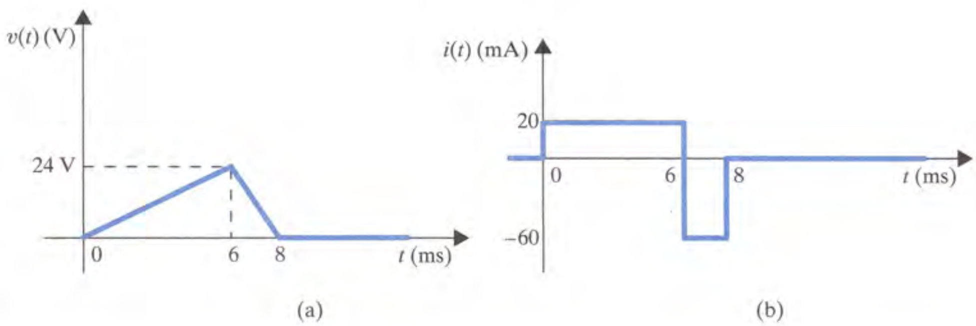


Figure 5.4
Voltage and current waveforms for a 5-μF capacitor.

LEARNING Example 5.3

Determine the energy stored in the electric field of the capacitor in Example 5.2 at $t = 6$ ms.

SOLUTION Using Eq. (5.6), we have

$$w(t) = \tfrac{1}{2} C v^2(t)$$

At $t = 6$ ms,

$$\begin{aligned} w(6 \text{ ms}) &= \tfrac{1}{2}(5 \times 10^{-6})(24)^2 \\ &= 1440 \ \mu\text{J} \end{aligned}$$

LEARNING EXTENSION

E5.1 A 10-μF capacitor has an accumulated charge of 500 nC. Determine the voltage across the capacitor.

ANSWER 0.05 V.

LEARNING Example 5.4

The current in an initially uncharged 4-μF capacitor is shown in Fig. 5.5a. Let us derive the waveforms for the voltage, power, and energy and compute the energy stored in the electric field of the capacitor at $t = 2$ ms.

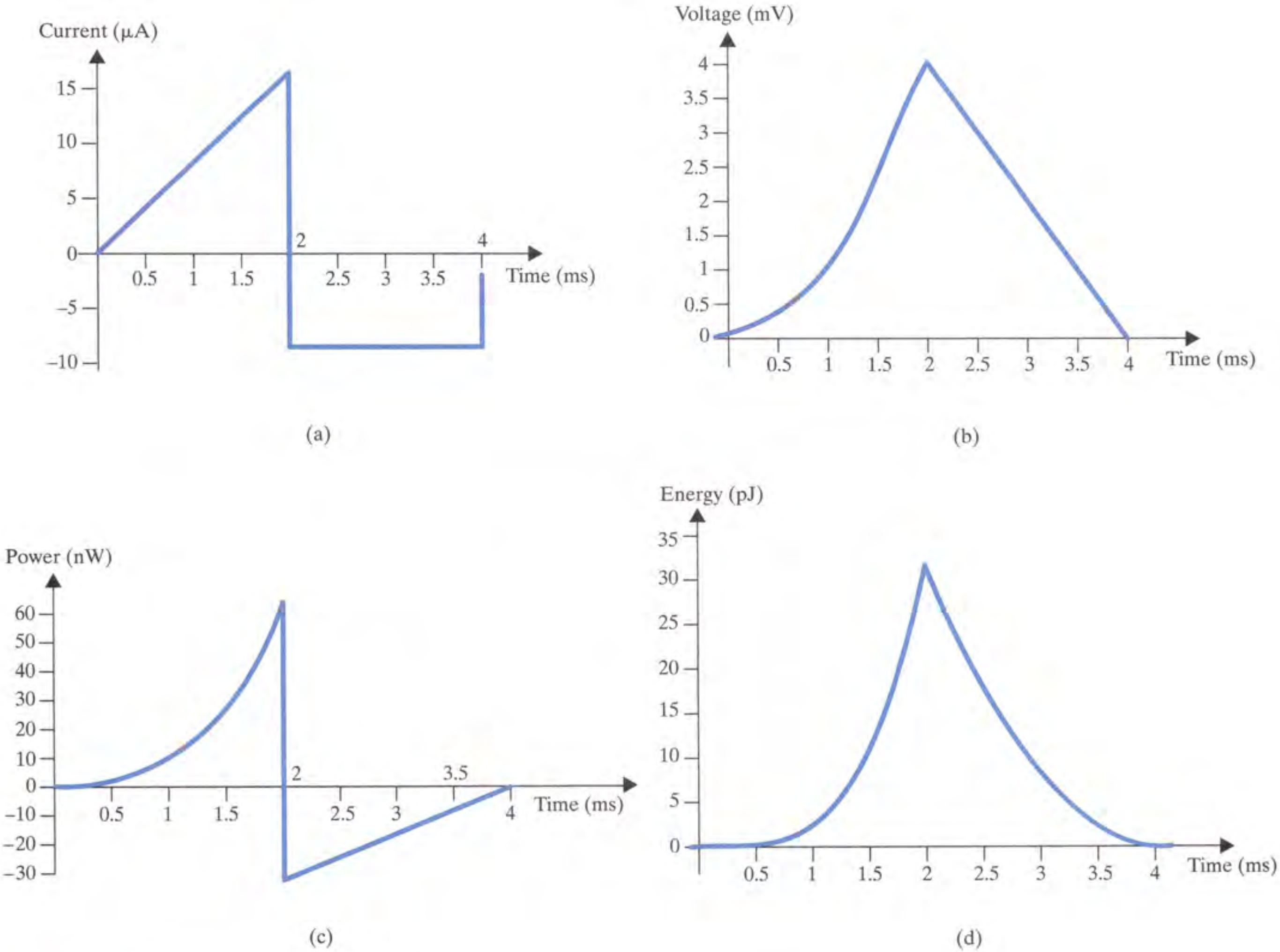


Figure 5.5 *Waveforms used in Example 5.4.*

SOLUTION The equations for the current waveform in the specific time intervals are

$$i(t) = \frac{16 \times 10^{-6} t}{2 \times 10^{-3}} \qquad 0 \le t \le 2 \text{ ms}$$
$$= -8 \times 10^{-6} \qquad 2 \text{ ms} \le t \le 4 \text{ ms}$$
$$= 0 \qquad 4 \text{ ms} < t$$

Since $v(0) = 0$, the equation for $v(t)$ in the time interval $0 \le t \le 2$ ms is

$$v(t) = \frac{1}{(4)(10^{-6})} \int_0^t 8(10^{-3})x\,dx = 10^3 t^2$$

and hence,

$$v(2 \text{ ms}) = 10^3(2 \times 10^{-3})^2 = 4 \text{ mV}$$

In the time interval 2 ms $\le t \le$ 4 ms,

$$v(t) = \frac{1}{(4)(10^{-6})} \int_{2(10^{-3})}^t -(8)(10^{-6})dx + (4)(10^{-3})$$
$$= -2t + 8 \times 10^{-3}$$

The waveform for the voltage is shown in Fig. 5.5b.

Since the power is $p(t) = v(t)i(t)$, the expression for the power in the time interval $0 \leq t \leq 2$ ms is $p(t) = 8t^3$. In the time interval 2 ms $\leq t \leq$ 4 ms, the equation for the power is

$$\begin{aligned} p(t) &= -(8)(10^{-6})(-2t + 8 \times 10^{-3}) \\ &= 16(10^{-6})t - 64(10^{-9}) \end{aligned}$$

The power waveform is shown in Fig. 5.5c. Note that during the time interval $0 \leq t \leq 2$ ms, the capacitor is absorbing energy and during the interval 2 ms $\leq t \leq$ 4 ms, it is delivering energy.

The energy is given by the expression

$$w(t) = \int_{t_0}^{t} p(x)\,dx + w(t_0)$$

In the time interval $0 \leq t \leq 2$ ms,

$$w(t) = \int_0^t 8x^3\,dx = 2t^4$$

Hence,

$$w(2 \text{ ms}) = 32 \text{ pJ}$$

In the time interval $2 \leq t \leq 4$ ms,

$$\begin{aligned} w(t) &= \int_{2\times10^{-3}}^{t} \left[(16 \times 10^{-6})x - (64 \times 10^{-9})\right] dx + 32 \times 10^{-12} \\ &= \left[(8 \times 10^{-6})x^2 - (64 \times 10^{-9})x\right]_{2\times10^{-3}}^{t} + 32 \times 10^{-12} \\ &= (8 \times 10^{-6})t^2 - (64 \times 10^{-9})t + 128 \times 10^{-12} \end{aligned}$$

From this expression we find that $w(2 \text{ ms}) = 32$ pJ and $w(4 \text{ ms}) = 0$. The energy waveform is shown in Fig. 5.5d.

LEARNING EXTENSIONS

E5.2 The voltage across a 2-μF capacitor is shown in Fig. E5.2. Determine the waveform for the capacitor current.

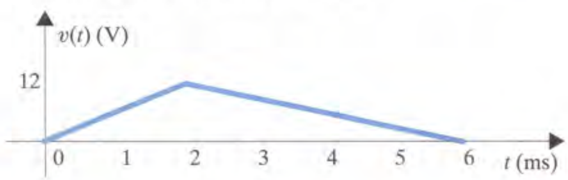


Figure E5.2

ANSWER

E5.3 Compute the energy stored in the electric field of the capacitor in Extension Exercise E5.2 at $t = 2$ ms.

ANSWER $w = 144$ μJ.

5.2 Inductors

An *inductor* is a circuit element that consists of a conducting wire usually in the form of a coil. Two typical inductors and their electrical symbols are shown in Fig. 5.6. Inductors are typically categorized by the type of core on which they are wound. For example, the core material may be air or any nonmagnetic material, iron, or ferrite. Inductors made with air or nonmagnetic materials are widely used in radio, television, and filter circuits. Iron-core inductors are used in electrical power supplies and filters. Ferrite-core inductors are widely used in high-frequency applications. Note that in contrast to the magnetic core that confines the flux, as shown in Fig. 5.6b, the flux lines for nonmagnetic inductors extend beyond the inductor itself, as illustrated in Fig. 5.6a. Like stray capacitance, stray inductance can result from any element carrying current surrounded by flux linkages. Figure 5.7 shows a variety of typical inductors.

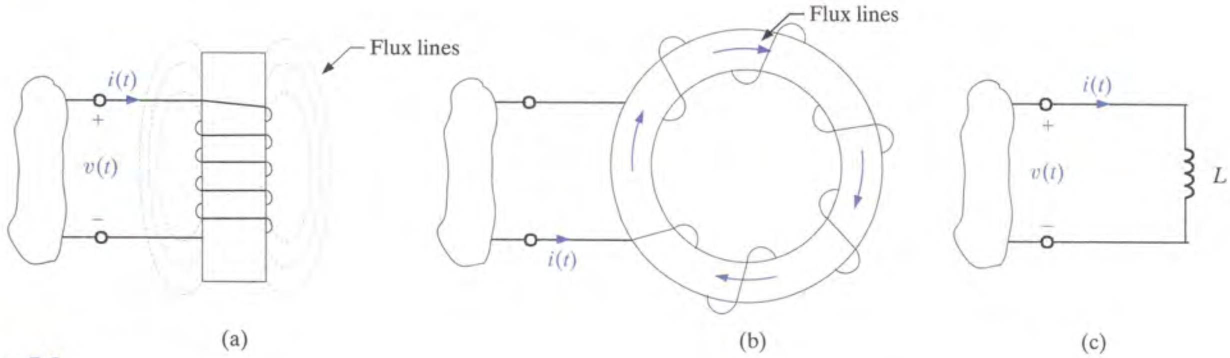


Figure 5.6
Two inductors and their electrical symbol.

LEARNING Hint

Note the use of the passive sign convention, as illustrated in Fig. 5.6.

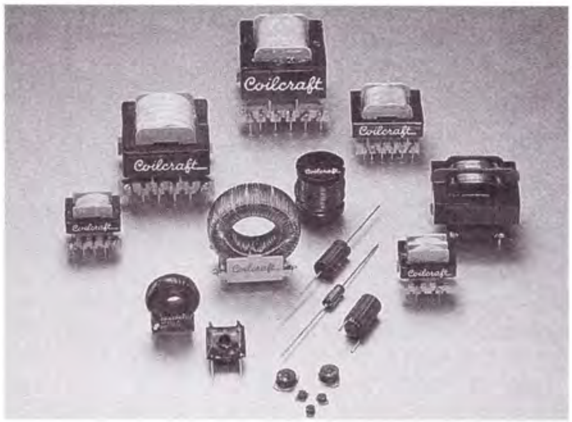
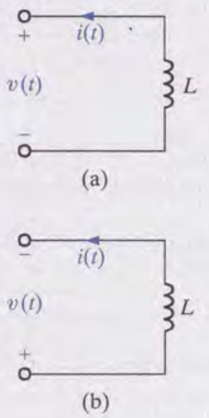


Figure 5.7
Some typical inductors (courtesy of Coilcraft).

LEARNING by Doing

D 5.2 Write the i-v relationship for the following inductors.



ANSWER

(a) $v(t) = -L\dfrac{di(t)}{dt}$

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

From a historical standpoint, developments that led to the mathematical model we employ to represent the inductor are as follows. It was first shown that a current-carrying conductor would produce a magnetic field. It was later found that the magnetic field and the current that produced it were linearly related. Finally, it was shown that a changing magnetic field produced a voltage that was proportional to the time rate of change of the current that produced the magnetic field; that is,

$$v(t) = L\frac{di(t)}{dt} \tag{5.8}$$

The constant of proportionality L is called the inductance and is measured in the unit henry, named after the American inventor Joseph Henry, who discovered the relationship. As seen in Eq. (5.8), 1 henry (H) is dimensionally equal to 1 volt-second per ampere.

Following the development of the mathematical equations for the capacitor, we find that the expression for the current in an inductor is

$$i(t) = \frac{1}{L}\int_{-\infty}^{t} v(x)\,dx \tag{5.9}$$

which can also be written as

$$i(t) = i(t_0) + \frac{1}{L}\int_{t_0}^{t} v(x)\,dx \tag{5.10}$$

The power delivered to the inductor can be used to derive the energy stored in the element. This power is equal to

$$\begin{aligned} p(t) &= v(t)i(t) \\ &= \left[L\frac{di(t)}{dt}\right]i(t) \end{aligned} \tag{5.11}$$

Therefore, the energy stored in the magnetic field is

$$w_L(t) = \int_{-\infty}^{t}\left[L\frac{di(x)}{dx}\right]i(x)\,dx$$

Following the development of Eq. (5.6), we obtain

$$w_L(t) = \tfrac{1}{2}Li^2(t)\text{ J} \qquad \textbf{5.12}$$

The inductor, like the resistor and capacitor, is a passive element. The polarity of the voltage across the inductor is shown in Fig. 5.6.

Practical inductors typically range from a few microhenrys to tens of henrys. From a circuit design standpoint it is important to note that inductors cannot be easily fabricated on an integrated circuit chip, and therefore chip designs typically employ only active electronic devices, resistors, and capacitors that can be easily fabricated in microcircuit form.

LEARNING Example 5.5

The current in a 10-mH inductor has the waveform shown in Fig. 5.8a. Determine the voltage waveform.

SOLUTION Using Eq. (5.8) and noting that

$$i(t) = \frac{20 \times 10^{-3}t}{2 \times 10^{-3}} \qquad 0 \le t \le 2\text{ ms}$$

$$i(t) = \frac{-20 \times 10^{-3}t}{2 \times 10^{-3}} + 40 \times 10^{-3} \qquad 2 \le t \le 4\text{ ms}$$

and

$$i(t) = 0 \qquad 4\text{ ms} < t$$

we find that

$$v(t) = (10 \times 10^{-3})\frac{20 \times 10^{-3}}{2 \times 10^{-3}} \qquad 0 \le t \le 2\text{ ms}$$
$$= 100\text{ mV}$$

and

$$v(t) = (10 \times 10^{-3})\frac{-20 \times 10^{-3}}{2 \times 10^{-3}} \qquad 2 \le t \le 4\text{ ms}$$
$$= -100\text{ mV}$$

and $v(t) = 0$ for $t > 4$ ms. Therefore, the voltage waveform is shown in Fig. 5.8b.

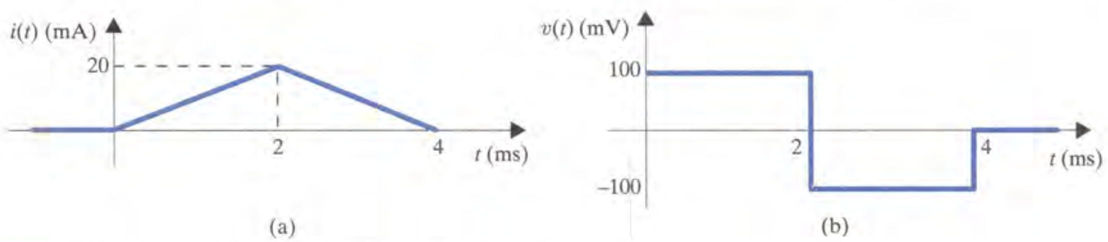


Figure 5.8 Current and voltage waveforms for a 10-mH inductor.

LEARNING Example 5.6

The current in a 2-mH inductor is

$$i(t) = 2\sin 377t\text{ A}$$

Determine the voltage across the inductor and the energy stored in the inductor.

SOLUTION From Eq. (5.8), we have

$$v(t) = L\frac{di(t)}{dt}$$
$$= (2 \times 10^{-3})\frac{d}{dt}(2\sin 377t)$$
$$= 1.508\cos 377t\text{ V}$$

and from Eq. (5.12),

$$w_L(t) = \tfrac{1}{2}Li^2(t)$$
$$= \tfrac{1}{2}(2 \times 10^{-3})(2\sin 377t)^2$$
$$= 0.004\sin^2 377t\text{ J}$$

LEARNING Example 5.7

The voltage across a 200-mH inductor is given by the expression

$$v(t) = (1 - 3t)e^{-3t}\text{mV} \quad t \geq 0$$
$$= 0 \quad t < 0$$

Let us derive the waveforms for the current, energy, and power.

SOLUTION The waveform for the voltage is shown in Fig. 5.9a. The current is derived from Eq. (5.10) as

$$i(t) = \frac{10^3}{200}\int_0^t (1 - 3x)e^{-3x}\,dx$$
$$= 5\left\{\int_0^t e^{-3x}\,dx - 3\int_0^t xe^{-3x}\,dx\right\}$$
$$= 5\left\{\frac{e^{-3x}}{-3}\bigg|_0^t - 3\left[-\frac{e^{-3x}}{9}(3x + 1)\right]_0^t\right\}$$
$$= 5te^{-3t}\text{ mA} \quad t \geq 0$$
$$= 0 \quad t < 0$$

A plot of the current waveform is shown in Fig. 5.9b.

The power is given by the expression

$$p(t) = v(t)i(t)$$
$$= 5t(1 - 3t)e^{-6t}\ \mu\text{W} \quad t \geq 0$$
$$= 0 \quad t < 0$$

The equation for the power is plotted in Fig. 5.9c.

The expression for the energy is

$$w(t) = \tfrac{1}{2}Li^2(t)$$
$$= 2.5t^2e^{-6t}\ \mu\text{J} \quad t \geq 0$$
$$= 0 \quad t < 0$$

This equation is plotted in Fig. 5.9d.

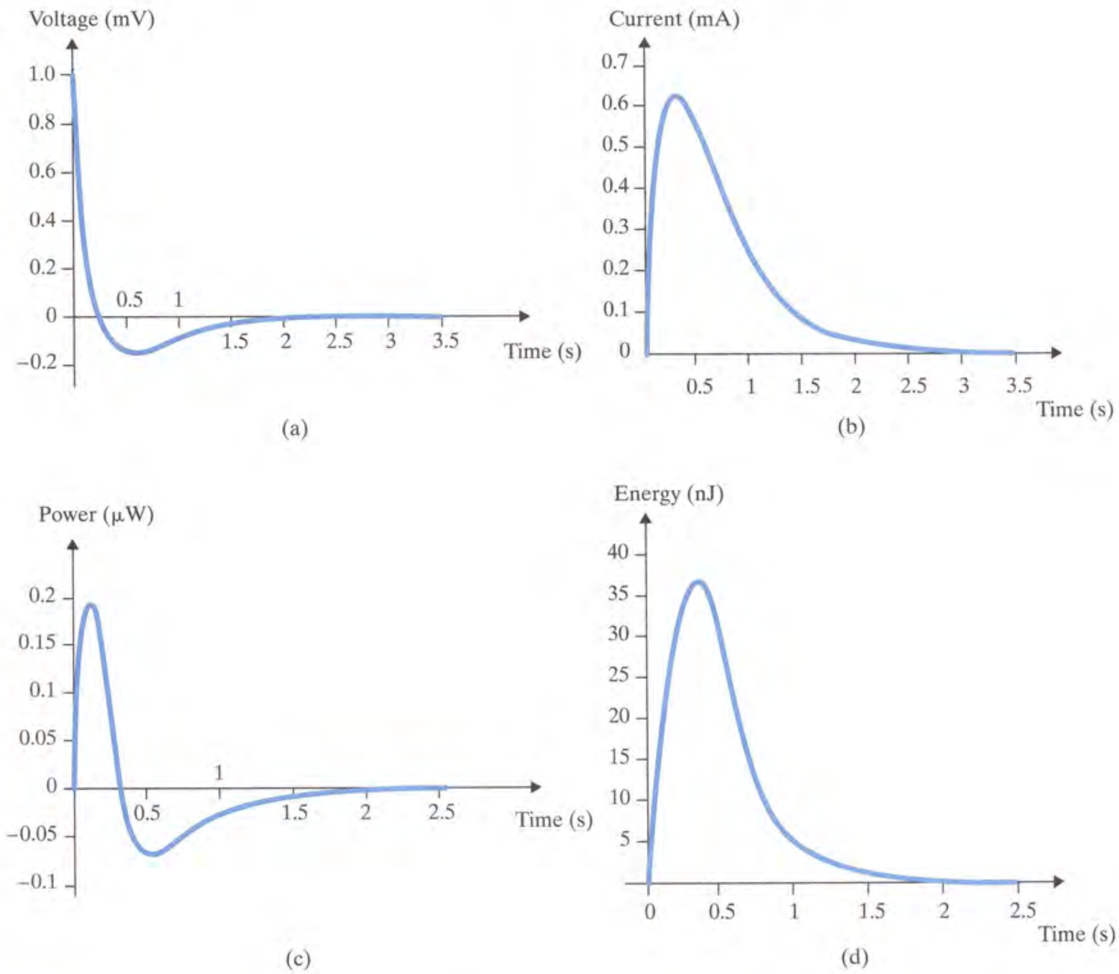


Figure 5.9 Waveforms used in Example 5.7.

LEARNING EXTENSIONS

E5.4 The current in a 5-mH inductor has the waveform shown in Fig. E5.4. Compute the waveform for the inductor voltage.

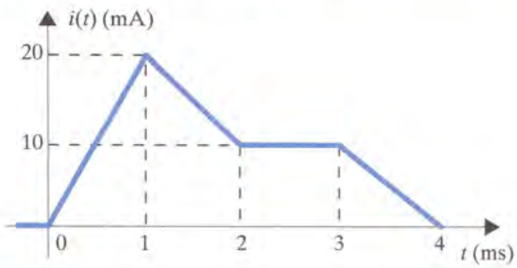
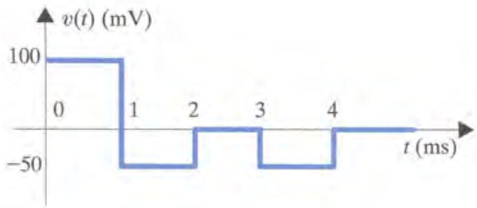


Figure E5.4

ANSWER



E5.5 Compute the energy stored in the magnetic field of the inductor in Extension Exercise E5.4 at $t = 1.5$ ms.

ANSWER $W = 562.5$ nJ.

CAPACITOR AND INDUCTOR SPECIFICATIONS There are a couple of important parameters that are used to specify capacitors and inductors. In the case of capacitors, the capacitance value, working voltage, and tolerance are issues that must be considered in their application. Standard capacitor values range from a few pF to about 50 mF. Capacitors larger than 1 F are available, but will not be discussed here. Table 5.1 is a list of standard capacitor values, which are typically given in picofarads or microfarads. Although both smaller and larger ratings are available, the standard working voltage, or dc voltage rating, is typically between 6.3 V and 500 V. Manufacturers specify this working voltage since it is critical to keep the applied voltage below the breakdown point of the dielectric. Tolerance is an adjunct to the capacitance value and is usually listed as a percentage of the nominal value. Standard tolerance values are ±5%, ±10%, and ±20%. Occasionally, tolerances for single-digit pF capacitors are listed in pF. For example, 5 pF ±0.25 pF.

Table 5.1 Standard capacitor values

pF	pF	pF	pF	μF	μF	μF	μF	μF	μF	μF
1	10	100	1000	0.010	0.10	1.0	10	100	1000	10,000
	12	120	1200	0.012	0.12	1.2	12	120	1200	12,000
1.5	15	150	1500	0.015	0.15	1.5	15	150	1500	15,000
	18	180	1800	0.018	0.18	1.8	18	180	1800	18,000
2	20	200	2000	0.020	0.20	2.0	20	200	2000	20,000
	22	220	2200	0.022	0.20	2.2	22	220	2200	22,000
	27	270	2700	0.027	0.27	2.7	27	270	2700	27,000
3	33	330	3300	0.033	0.33	3.3	33	330	3300	33,000
4	39	390	3900	0.039	0.39	3.9	39	390	3900	39,000
5	47	470	4700	0.047	0.47	4.7	47	470	4700	47,000
6	51	510	5100	0.051	0.51	5.1	51	510	5100	51,000
7	56	560	5600	0.056	0.56	5.6	56	560	5600	56,000
8	68	680	6800	0.068	0.68	6.8	68	680	6800	68,000
9	82	820	8200	0.082	0.82	8.2	82	820	8200	82,000

Table 5.2 Standard inductor values

nH	nH	nH	μH	μH	μH	mH	mH	mH
1	10	100	1.0	10	100	1.0	10	100
1.2	12	120	1.2	12	120	1.2	12	
1.5	15	150	1.5	15	150	1.5	15	
1.8	18	180	1.8	18	180	1.8	18	
2	20	200	2.0	20	200	2.0	20	
2.2	22	220	2.2	22	200	2.2	22	
2.7	27	270	2.7	27	270	2.7	27	
3	33	330	3.3	33	330	3.3	33	
4	39	390	3.9	39	390	3.9	39	
5	47	470	4.7	47	470	4.7	47	
6	51	510	5.1	51	510	5.1	51	
7	56	560	5.6	56	560	5.6	56	
8	68	680	6.8	68	680	6.8	68	
9	82	820	8.2	82	820	8.2	82	

There are two principle inductor specifications, inductance and resistance. Standard commercial inductances range from about 1 nH to around 100 mH. Larger inductances can, of course, be custom built for a price. Table 5.2 lists the standard inductor values. The current rating for inductors typically extends from a few dozen mA's to about 1 A. Tolerances are typically 5% or 10% of the specified value.

As indicated in Chapter 2, wire-wound resistors are simply coils of wire, and therefore it is only logical that inductors will have some resistance. The major difference between wire-wound resistors and inductors is the wire material. High-resistance materials such as Nichrome are used in resistors and low-resistance copper is used in inductors. The resistance of the copper wire is dependent on the length and diameter of the wire. Table 5.3 lists the American Wire Gauge (AWG) standard wire diameters and the resulting resistance per foot for copper wire.

Table 5.3 Resistance per foot of solid copper wire

AWG No.	Diameter (in.)	mΩ/ft
12	0.0808	1.59
14	0.0641	2.54
16	0.0508	4.06
18	0.0400	6.50
20	0.0320	10.4
22	0.0253	16.5
24	0.0201	26.2
26	0.0159	41.6
28	0.0126	66.2
30	0.0100	105
32	0.0080	167
34	0.0063	267
36	0.0049	428
38	0.0039	684
40	0.0031	1094

LEARNING Example 5.8

We wish to find the possible range of capacitance values for a 51-mF capacitor that has a tolerance of 20%.

SOLUTION The minimum capacitor value is $0.8C = 40.8$ mF, and the maximum capacitor value is $1.2C = 61.2$ mF.

LEARNING Example 5.9

The capacitor in Fig. 5.10a is a 100-nF capacitor with a tolerance of 20%. If the voltage waveform is as shown in Fig. 5.10b, let us graph the current waveform for the minimum and maximum capacitor values.

SOLUTION The maximum capacitor value is $1.2C = 120$ nF, and the minimum capacitor value is $0.8C = 80$ nF. The maximum and minimum capacitor currents, obtained from the equation

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

are shown in Fig. 5.10c.

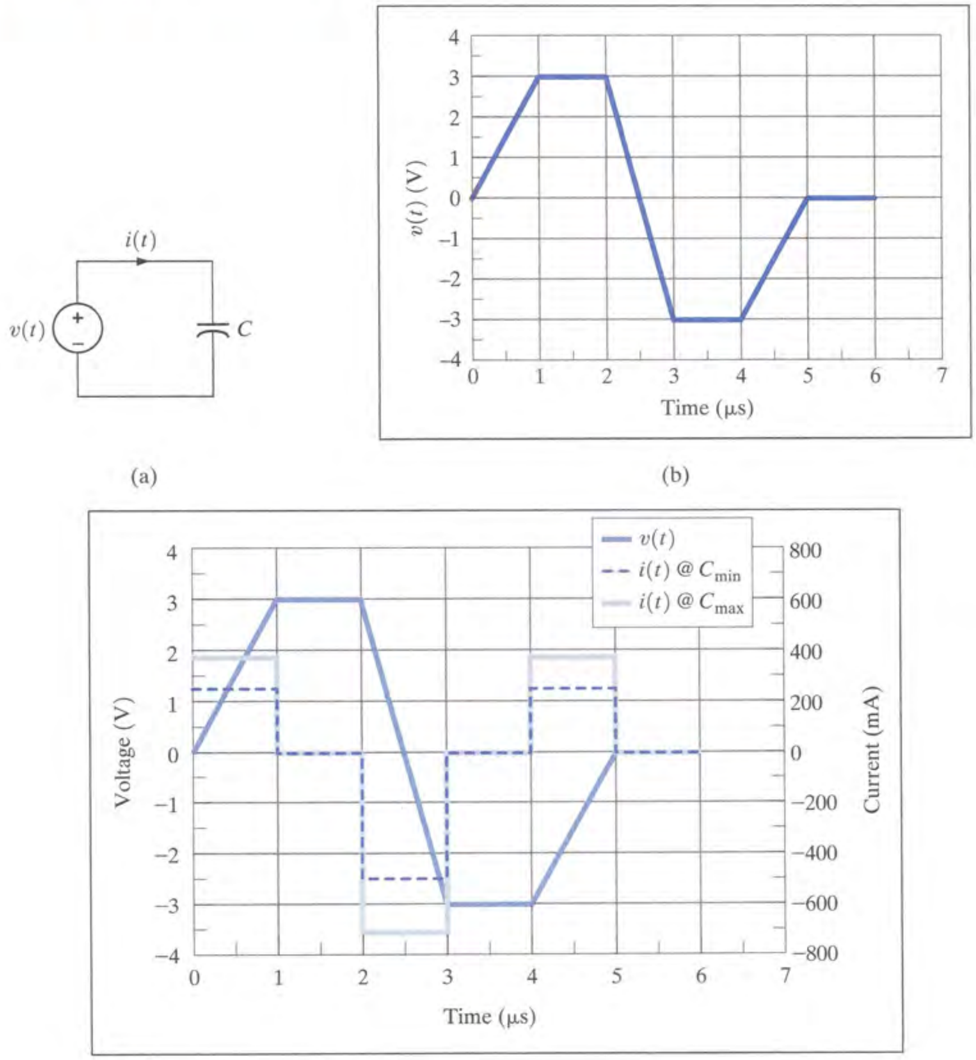
LEARNING by Doing

D 5.3 Find the possible range of capacitance values for the following capacitors:

(a) 100 pF with a tolerance of 10%

(b) 8.2 nF with a tolerance of 5%

ANSWER *(a) 90 pF to 110 pF, (b) 7.79 nF to 8.61 nF*



(c)

Figure 5.10
Circuit and graphs used in Example 5.9.

LEARNING Example 5.10

The inductor in Fig. 5.11a is a 100-μH inductor with a tolerance of 10%. If the current waveform is as shown in Fig. 5.11b, let us graph the voltage waveform for the minimum and maximum inductor values.

SOLUTION The maximum inductor value is $1.1L = 110\ \mu\text{H}$, and the minimum inductor value is $0.9L = 90\ \mu\text{H}$. The maximum and minimum inductor voltages, obtained from the equation

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

are shown in Fig. 5.11c.

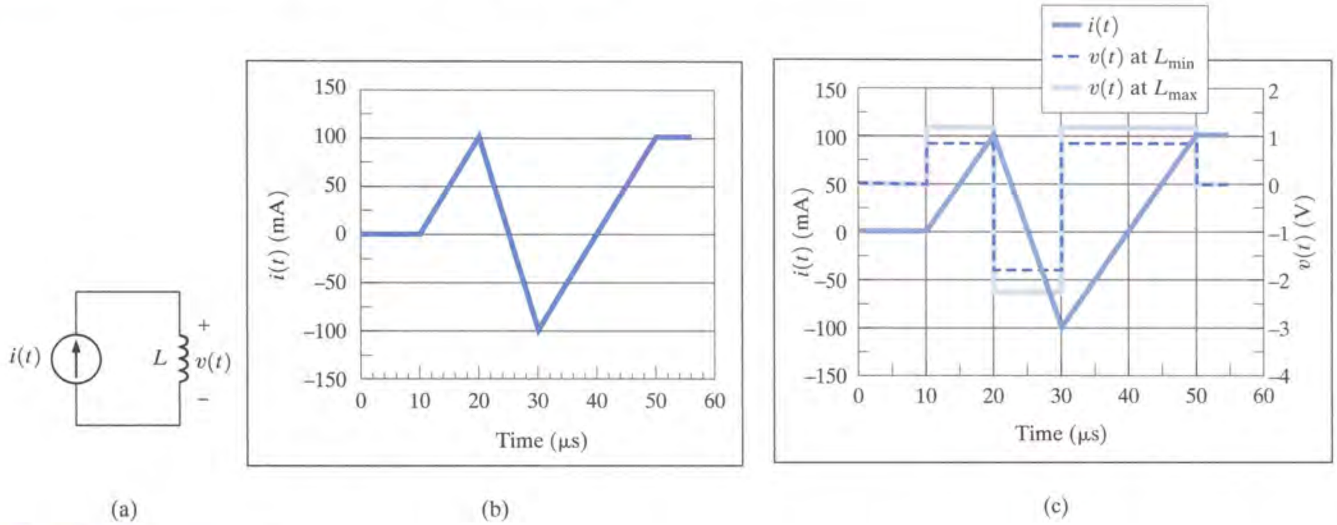


Figure 5.11
Circuit and graphs used in Example 5.10.

5.3 Capacitor and Inductor Combinations

SERIES CAPACITORS If a number of capacitors are connected in series, their equivalent capacitance can be calculated using KVL. Consider the circuit shown in Fig. 5.12a. For this circuit

$$v(t) = v_1(t) + v_2(t) + v_3(t) + \cdots + v_N(t) \qquad \textbf{5.13}$$

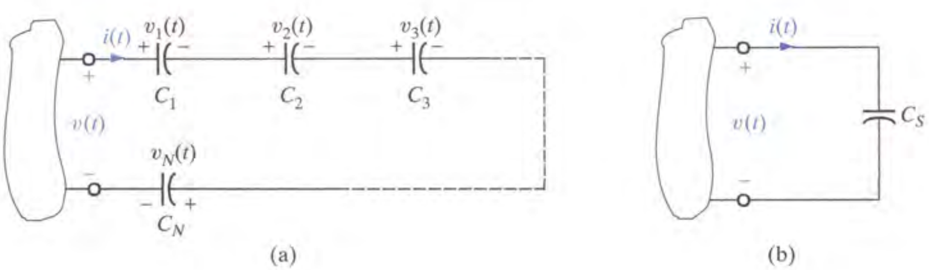


Figure 5.12
Equivalent circuit for N series-connected capacitors.

but

$$v_i(t) = \frac{1}{C_i}\int_{t_0}^{t} i(t)\,dt + v_i(t_0) \tag{5.14}$$

Therefore, Eq. (5.13) can be written as follows using Eq. (5.14):

$$v(t) = \left(\sum_{i=1}^{N}\frac{1}{C_i}\right)\int_{t_0}^{t} i(t)\,dt + \sum_{i=1}^{N} v_i(t_0) \tag{5.15}$$

$$= \frac{1}{C_S}\int_{t_0}^{t} i(t)\,dt + v(t_0) \tag{5.16}$$

where

$$v(t_0) = \sum_{i=1}^{N} v_i(t_0)$$

and

$$\frac{1}{C_S} = \sum_{i=1}^{N}\frac{1}{C_i} = \frac{1}{C_1} + \frac{1}{C_2} + \cdots + \frac{1}{C_N} \tag{5.17}$$

LEARNING Hint

Capacitors in series combine like resistors in parallel.

Thus, the circuit in Fig. 5.12b is equivalent to that in Fig. 5.12a under the conditions stated previously.

It is also important to note that since the same current flows in each of the series capacitors, each capacitor gains the same charge in the same time period. The voltage across each capacitor will depend on this charge and the capacitance of the element.

LEARNING Example 5.11

Determine the equivalent capacitance and the initial voltage for the circuit shown in Fig. 5.13.

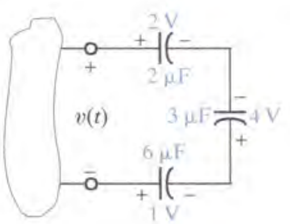


Figure 5.13 *Circuit containing multiple capacitors with initial voltages.*

SOLUTION Note that these capacitors must have been charged before they were connected in series or else the charge of each would be equal and the voltages would be in the same direction.

The equivalent capacitance is

$$\frac{1}{C_S} = \frac{1}{2} + \frac{1}{3} + \frac{1}{6}$$

where all capacitance values are in microfarads.

Therefore, $C_S = 1$ μF and, as seen from the figure, $v(t_0) = -3$ V. Note that the total energy stored in the circuit is

$$w(t_0) = \tfrac{1}{2}\left[2\times10^{-6}(2)^2 + 3\times10^{-6}(-4)^2 + 6\times10^{-6}(-1)^2\right]$$
$$= 31\ \mu\text{J}$$

However, the energy recoverable at the terminals is

$$w_C(t_0) = \tfrac{1}{2}C_S v^2(t)$$
$$= \tfrac{1}{2}\left[1\times10^{-6}(-3)^2\right]$$
$$= 4.5\ \mu\text{J}$$

LEARNING Example 5.12

Two previously uncharged capacitors are connected in series and then charged with a 12-V source. One capacitor is 30 μF and the other is unknown. If the voltage across the 30-μF capacitor is 8 V, find the capacitance of the unknown capacitor.

SOLUTION The charge on the 30-μF capacitor is

$$Q = CV = (30\ \mu\text{F})(8\ \text{V}) = 240\ \mu\text{C}$$

Since the same current flows in each of the series capacitors, each capacitor gains the same charge in the same time period.

$$C = \frac{Q}{V} = \frac{240\ \mu\text{C}}{4\text{V}} = 60\ \mu\text{F}$$

PARALLEL CAPACITORS To determine the equivalent capacitance of N capacitors connected in parallel, we employ KCL. As can be seen from Fig. 5.14a,

$$\begin{aligned} i(t) &= i_1(t) + i_2(t) + i_3(t) + \cdots + i_N(t) \\ &= C_1 \frac{dv(t)}{dt} + C_2 \frac{dv(t)}{dt} + C_3 \frac{dv(t)}{dt} + \cdots + C_N \frac{dv(t)}{dt} \\ &= \left(\sum_{i=1}^{N} C_i \right) \frac{dv(t)}{dt} \\ &= C_p \frac{dv(t)}{dt} \end{aligned} \qquad \begin{matrix} 5.18 \\ \\ \\ 5.19 \end{matrix}$$

LEARNING Hint

Capacitors in parallel combine like resistors in series.

where

$$C_p = C_1 + C_2 + C_3 + \cdots + C_N \qquad 5.20$$

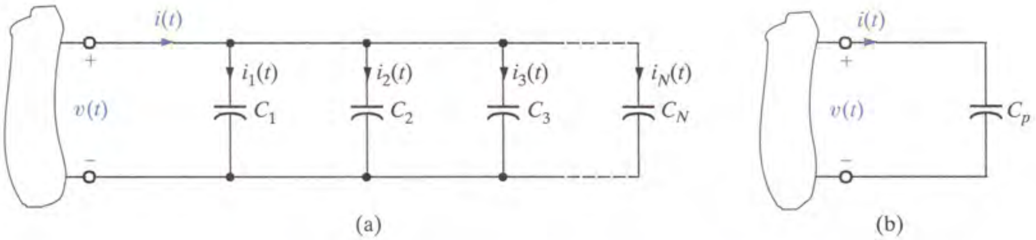


Figure 5.14
Equivalent circuit for N capacitors connected in parallel.

LEARNING Example 5.13

Determine the equivalent capacitance at terminals A-B of the circuit shown in Fig. 5.15.

SOLUTION $C_p = 15\ \mu F$

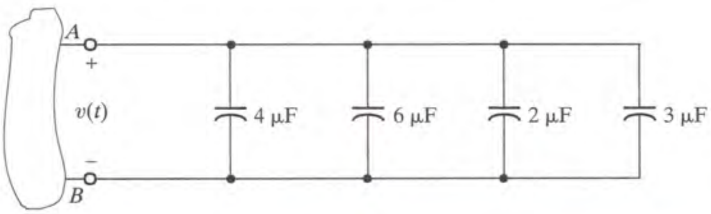


Figure 5.15
Circuit containing multiple capacitors in parallel.

LEARNING EXTENSION

E5.6 Two initially uncharged capacitors are connected as shown in Fig. E5.6. After a period of time, the voltage reaches the value shown. Determine the value of C_1.

ANSWER $C_1 = 4\ \mu F$.

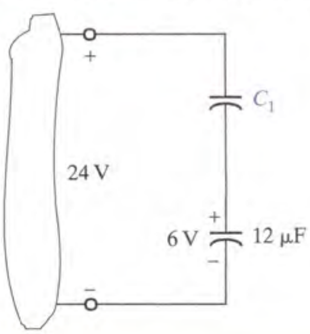


Figure E5.6

LEARNING EXTENSION

E5.7 Compute the equivalent capacitance of the network in Fig. E5.7.

ANSWER $C_{eq} = 1.5\ \mu F$.

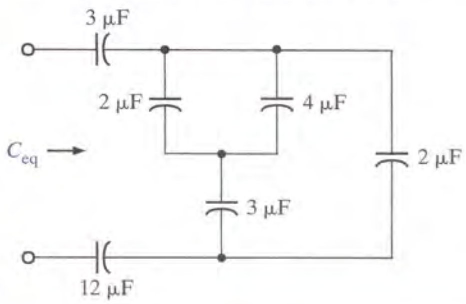


Figure E5.7

SERIES INDUCTORS If N inductors are connected in series, the equivalent inductance of the combination can be determined as follows. Referring to Fig. 5.16a and using KVL, we see that

$$v(t) = v_1(t) + v_2(t) + v_3(t) + \cdots + v_N(t) \quad \mathbf{5.21}$$

and therefore,

$$v(t) = L_1 \frac{di(t)}{dt} + L_2 \frac{di(t)}{dt} + L_3 \frac{di(t)}{dt} + \cdots + L_N \frac{di(t)}{dt} \quad \mathbf{5.22}$$

$$= \left(\sum_{i=1}^{N} L_i\right) \frac{di(t)}{dt}$$

$$= L_S \frac{di(t)}{dt} \quad \mathbf{5.23}$$

where

$$L_S = \sum_{i=1}^{N} L_i = L_1 + L_2 + \cdots + L_N \quad \mathbf{5.24}$$

Therefore, under this condition the network in Fig. 5.16b is equivalent to that in Fig. 5.16a.

LEARNING Hint

Inductors in series combine like resistors in series.

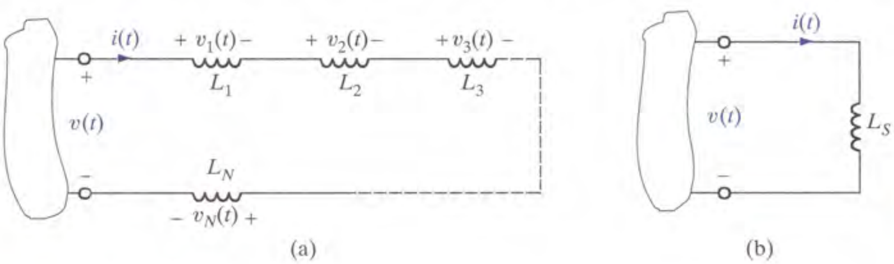


Figure 5.16
Equivalent circuit for N series-connected inductors.

LEARNING Example 5.14

Find the equivalent inductance of the circuit shown in Fig. 5.17.

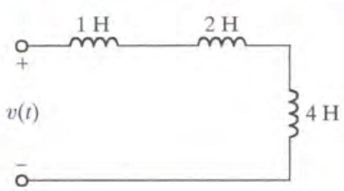


Figure 5.17
Circuit containing multiple inductors.

SOLUTION The equivalent inductance of the circuit shown in Fig. 5.17 is

$$L_S = 1\text{H} + 2\text{H} + 4\text{H}$$
$$= 7\text{H}$$

PARALLEL INDUCTORS Consider the circuit shown in Fig. 5.18a, which contains N parallel inductors. Using KCL, we can write

$$i(t) = i_1(t) + i_2(t) + i_3(t) + \cdots + i_N(t) \quad 5.25$$

However,

$$i_j(t) = \frac{1}{L_j}\int_{t_0}^{t} v(x)\,dx + i_j(t_0) \quad 5.26$$

Substituting this expression into Eq. (5.25) yields

$$i(t) = \left(\sum_{j=1}^{N}\frac{1}{L_j}\right)\int_{t_0}^{t} v(x)\,dx + \sum_{j=1}^{N} i_j(t_0) \quad 5.27$$

$$= \frac{1}{L_p}\int_{t_0}^{t} v(x)\,dx + i(t_0) \quad 5.28$$

where

$$\frac{1}{L_p} = \frac{1}{L_1} + \frac{1}{L_2} + \frac{1}{L_3} + \cdots + \frac{1}{L_N} \quad 5.29$$

and $i(t_0)$ is equal to the current in L_p at $t = t_0$. Thus, the circuit in Fig. 5.18b is equivalent to that in Fig. 5.18a under the conditions stated previously.

LEARNING Hint

Inductors in parallel combine like resistors in parallel.

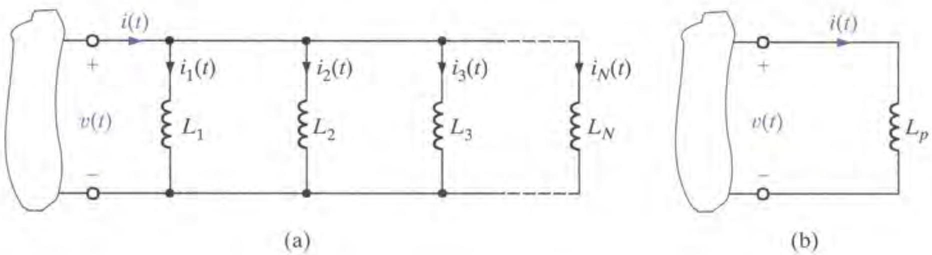


Figure 5.18
Equivalent circuits for N inductors connected in parallel.

LEARNING Example 5.15

Determine the equivalent inductance and the initial current for the circuit shown in Fig. 5.19.

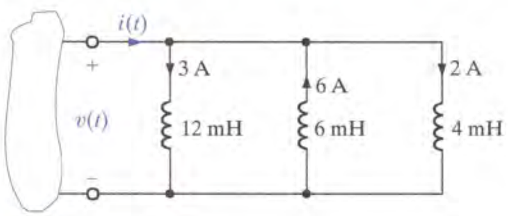


Figure 5.19
Circuit containing multiple inductors with initial currents.

SOLUTION The equivalent inductance is

$$\frac{1}{L_p} = \frac{1}{12} + \frac{1}{6} + \frac{1}{4}$$

where all inductance values are in millihenrys

$$L_p = 2 \text{ mH}$$

and the initial current is $i(t_0) = -1\ A$.

The previous material indicates that capacitors combine like conductances, whereas inductances combine like resistances.

LEARNING EXTENSION

E5.8 Determine the equivalent inductance of the network in Fig. E5.8 if all inductors are 6 mH. **ANSWER** 9.429 mH.

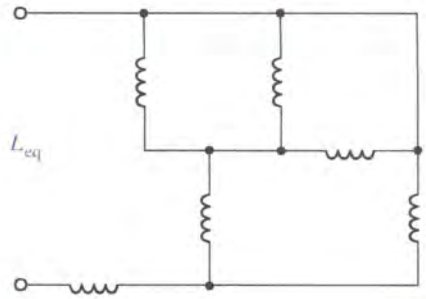


Figure E5.8

5.4 *RC* Operational Amplifier Circuits

Two very important *RC* op-amp circuits are the differentiator and the integrator. These circuits are derived from the circuit for an inverting op-amp by replacing the resistors R_1 and R_2, respectively, by a capacitor. Consider, for example, the circuit shown in Fig. 5.20a. The circuit equations are

LEARNING Hint

The properties of the ideal op-amp are $v_+ = v_-$ and $i_+ = i_- = 0$.

$$C_1 \frac{d}{dt}(v_1 - v_-) + \frac{v_o - v_-}{R_2} = i_-$$

However, $v_- = 0$ and $i_- = 0$. Therefore,

$$v_o(t) = -R_2 C_1 \frac{dv_1(t)}{dt} \qquad \textbf{5.30}$$

Thus, the output of the op-amp circuit is proportional to the derivative of the input.

The circuit equations for the op-amp configuration in Fig. 5.20b are

$$\frac{v_1 - v_-}{R_1} + C_2 \frac{d}{dt}(v_o - v_-) = i_-$$

but since $v_- = 0$ and $i_- = 0$, the equation reduces to

$$\frac{v_1}{R_1} = -C_2 \frac{dv_o}{dt}$$

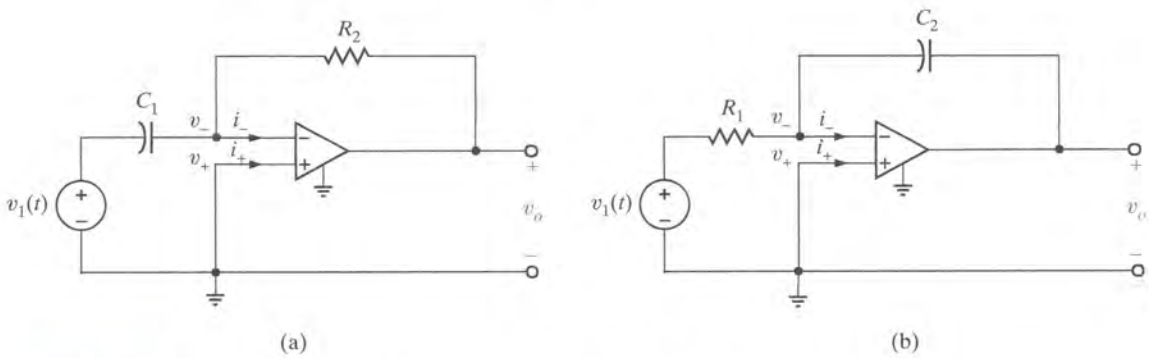


Figure 5.20 *Differentiator and integrator operational amplifier circuits.*

or

$$v_o(t) = \frac{-1}{R_1C_2}\int_{-\infty}^{t} v_1(x)\,dx$$
$$= \frac{-1}{R_1C_2}\int_{0}^{t} v_1(x)\,dx + v_o(0) \qquad \textbf{5.31}$$

If the capacitor is initially discharged, then $v_o(0) = 0$; hence,

$$v_o(t) = \frac{-1}{R_1C_2}\int_{0}^{t} v_1(x)\,dx \qquad \textbf{5.32}$$

Thus, the output voltage of the op-amp circuit is proportional to the integral of the input voltage.

LEARNING Example 5.16

The waveform in Fig. 5.21a is applied at the input of the differentiator circuit shown in Fig. 5.20a. If $R_2 = 1\ \text{k}\Omega$ and $C_1 = 2\ \mu\text{F}$, determine the waveform at the output of the op-amp.

SOLUTION Using Eq. (5.30), we find that the op-amp output is

$$v_o(t) = -R_2C_1\frac{dv_1(t)}{dt}$$
$$= -(2)10^{-3}\frac{dv_1(t)}{dt}$$

$dv_1(t)/dt = (2)10^3$ for $0 \le t < 5$ ms, and therefore,

$$v_o(t) = -4\text{ V} \qquad 0 \le t < 5\text{ ms}$$

$dv_1(t)/dt = -(2)10^3$ for $5 \le t < 10$ ms, and therefore,

$$v_o(t) = 4\text{ V} \qquad 5 \le t < 10\text{ ms}$$

Hence, the output waveform of the differentiator is shown in Fig. 5.21b.

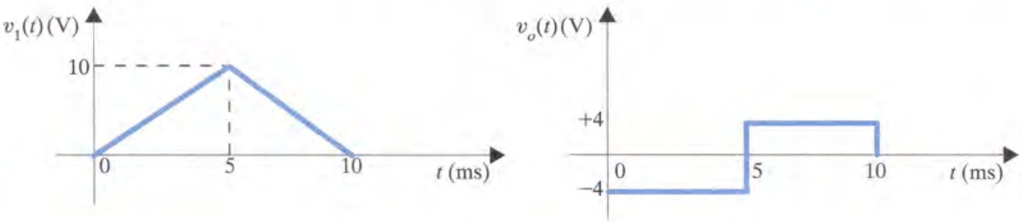
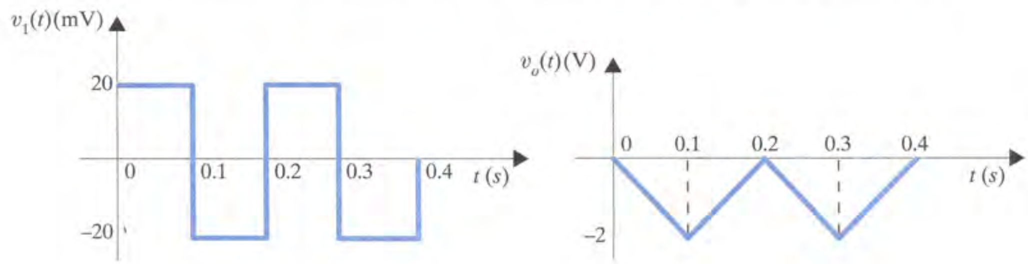


Figure 5.21 *Input and output waveforms for a differentiator circuit.*

LEARNING Example 5.17

If the integrator shown in Fig. 5.20b has the parameters $R_1 = 5\ \text{k}\Omega$ and $C_2 = 0.2\ \mu\text{F}$, determine the waveform at the op-amp output if the input waveform is given as in Fig. 5.22a and the capacitor is initially discharged.



(a) (b)

Figure 5.22 *Input and output waveforms for an integrator circuit.*

SOLUTION The integrator output is given by the expression

$$v_o(t) = \frac{-1}{R_1C_2}\int_0^t v_1(x)\,dx$$

which with the given circuit parameters is

$$v_o(t) = -10^3\int_0^t v_1(x)\,dx$$

In the interval $0 \le t < 0.1$ s, $v_1(t) = 20$ mV. Hence,

$$\begin{aligned} v_o(t) &= -10^3(20)10^{-3}t \qquad 0 \le t < 0.1 \text{ s} \\ &= -20t \end{aligned}$$

At $t = 0.1$ s, $v_o(t) = -2$ V. In the interval from 0.1 to 0.2 s, the integrator produces a positive slope output of $20t$ from $v_o(0.1) = -2$ V to $v_o(0.2) = 0$ V. This waveform from $t = 0$ to $t = 0.2$ s is repeated in the interval $t = 0.2$ to $t = 0.4$ s, and therefore, the output waveform is shown in Fig. 5.22b.

LEARNING EXTENSION

E5.9 The waveform in Fig. E5.9 is applied to the input terminals of the op-amp differentiator circuit. Determine the differentiator output waveform if the op-amp circuit parameters are $C_1 = 2$ F and $R_2 = 2\ \Omega$.

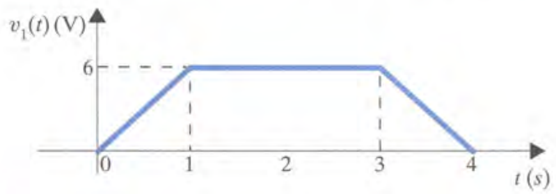
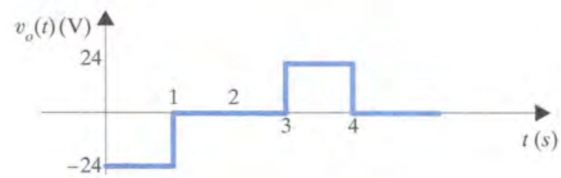


Figure E5.9

ANSWER



Learning by Application

LEARNING Example 5.18

An excellent example of capacitor operation is the memory inside a personal computer. This memory, called dynamic random access memory (DRAM) contains as many as half a billion data storage sites called cells (circa 2000). Expect this number to roughly double every two years for the next decade or two. Let us examine in some detail the operation of a single DRAM cell.

SOLUTION Figure 5.23a shows a simple model for a DRAM cell. Data are stored on the cell capacitor in true/false (or 1/0) format, where a large capacitor voltage represents a true condition and a low voltage represents a false condition. The switch closes to allow access from the processor to the DRAM cell. Current source I_{leak} is an unintentional, or parasitic, current

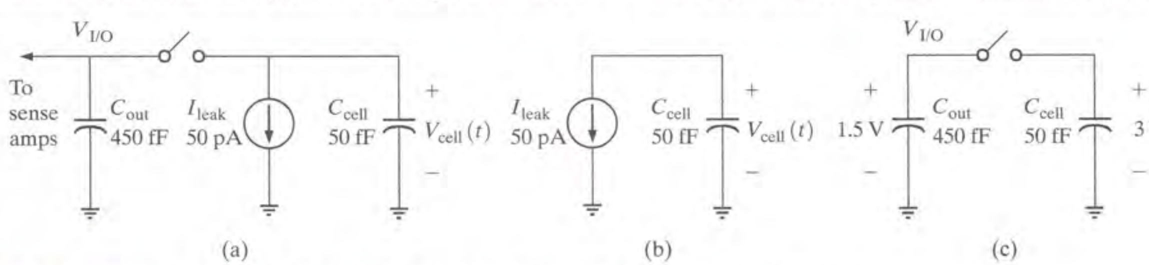


Figure 5.23 *A simple circuit model showing (a) the DRAM memory cell, (b) the effect of charge leakage from the cell capacitor, and (c) cell conditions at the beginning of a read operation.*

(continued)

that models charge leakage from the capacitor. Another parasitic model element is the capacitance, C_{out}, the capacitance of the wiring connected to the output side of the cell. Both I_{leak} and C_{out} have enormous impacts on DRAM performance and design.

Consider storing a true condition in the cell. A high voltage of 3.0 V is applied at node I/O and the switch is closed, causing the voltage on C_{cell} to quickly rise to 3.0 V. We open the switch and the data are stored. During the store operation the charge, energy and number of electrons, n, used are

$$Q = CV = (50 \times 10^{-15})(3) = 150 \text{ fC}$$
$$W = \tfrac{1}{2}CV^2 = (0.5)(50 \times 10^{-15})(3^2) = 225 \text{ fJ}$$
$$n = Q/q = 150 \times 10^{-15}/(1.6 \times 10^{-19}) = 937{,}500 \text{ electrons}$$

Once data are written, the switch opens and the capacitor begins to discharge through I_{leak}. A measure of DRAM quality is the time required for the data voltage to drop by half, from 3.0 V to 1.5 V. Let us call this time t_H. For the capacitor, we know

$$v_{cell}(t) = \frac{1}{C_{cell}} \int i_{cell}\, dt \text{ V}$$

where, from Fig. 5.23b, $i_{cell}(t) = -I_{leak}$. Performing the integral yields

$$v_{cell}(t) = \frac{1}{C_{cell}} \int (-I_{leak})\, dt = -\frac{I_{leak}}{C_{cell}} t + K$$

We know that at $t = 0$, $v_{cell} = 3$ V. Thus, $K = 3$ and the cell voltage is

$$v_{cell}(t) = 3 - \frac{I_{leak}}{C_{cell}} t \text{ V} \qquad \textbf{5.33}$$

Substituting $t = t_H$ and $v_{cell}(t_H) = 1.5$ V into Eq. 5.33 and solving for t_H yields $t_H = 15$ ms. Thus, the cell data are gone in only a few milliseconds! The solution is rewriting the data before it can disappear. This technique, called *refresh*, is a must for all DRAM using this one-transistor cell.

To see the affect of C_{out}, consider reading a fully charged $(v_{cell} = 3.0\text{ V})$ true condition. The I/O line is usually precharged to half the data voltage. In this example, that would be 1.5 V as seen in Fig. 5.23c. (To isolate the effect of C_{out}, we have removed I_{leak}.) Next, the switch is closed. What happens next is best viewed as a conservation of charge. Just before the switch closes, the total stored charge in the circuit is

$$Q_T = Q_{out} + Q_{cell} = V_{I/O}C_{out} + V_{cell}C_{cell} = (1.5)(450 \times 10^{-15}) + (3)(50 \times 10^{-15}) = 825 \text{ fC}$$

When the switch closes, the capacitor voltages are the same (let us call it V_o) and the total charge is unchanged.

$$Q_T = 825 \text{ fC} = V_oC_{out} + V_oC_{cell} = V_o(450 \times 10^{-15} + 50 \times 10^{-15})$$

and

$$V_o = 1.65 \text{ V}$$

Thus, the change in voltage at $V_{I/O}$ during the read operation is only 0.15 V. A very sensitive amplifier is required to quickly detect such a small change. In DRAMs, these amplifiers are called *sense amps*. How can v_{cell} change instantaneously when the switch closes? It cannot. In an actual DRAM cell, a transistor, which has a small equivalent resistance, acts as the switch. The resulting RC time constant is very small, indicating a very fast circuit. Recall that we are not analyzing the cell's speed—only the final voltage value, V_o. As long as the power lost in the switch is small compared to the capacitor energy, we can be comfortable in neglecting the switch resistance. By the way, if a false condition (zero volts) were read from the cell, then V_o would drop from its precharged value of 1.5 V to 1.35 V—a negative change of 0.15 V. This symmetric voltage change is the reason for precharging the I/O node to half the data voltage. Review the effects of I_{leak} and C_{out}. You will find that eliminating them would greatly simplify the refresh requirement and improve the voltage swing at node I/O when reading data. DRAM designers earn a very good living trying to do just that.

LEARNING Example 5.19

Traditionally, integrated circuits, or ICs, are built on silicon die and packaged by gluing the die to a mechanical base and connecting small wires called wirebonds (1 μm diameter) from the die to the package's external leads as seen in Fig. 5.24a. The wirebond, like any piece of wire, has inductance (about 2 nH), which can affect circuit performance. Since

$$v_L(t) = L\frac{di_L(t)}{dt} \qquad \textbf{5.34}$$

any IC that has large current transients has undesirable voltage transients across its wirebonds. This is particularly troublesome in the ground wirebond of digital switching ICs since the current there can be large. The voltage transient on the ground connection is called *ground bounce*. From (Eq. 5.34), there are three ways to decrease ground bounce: lower the inductance, decrease the current level, or operate at slower speeds. The last solution is no solution at all since we all want faster and faster electronics. However, the first two ideas have merit. In fact, manufacturers of very-high-speed ICs are presently (circa 2000) changing from wirebond to flip-chip connections where small solder balls provide electrical contact and the die is mounted upside down as seen in

Figs. 5.24b and c. This decreases the parasitic inductance to around 0.1 nH. Let us investigate the effects of flip-chip ball inductance on ground bounce.

SOLUTION Consider the circuit in Fig. 5.25a where $i_G(t)$ and L_{ball} model the current required by a single digital gate (smallest subcircuit)and the flip-chip ball inductance, respectively. The resulting ground bounce, $v_{GB}(t)$, shown in Fig. 5.25b, is ± 0.1 mV, only 0.004% of a typical 2.5-V supply voltage. Now consider a more realistic scenario where the IC contains 100,000 gates with 10% of them switching at the same time, 10,000 switching gates. The current through L_{ball} increases 10,000 times and, from Eq. (5.34), ground bounce increases by the same factor to ± 1 V, an unacceptable level. Thus, we resort to the second solution: less current. Instead of having all the logic gates connected to the same ground connection, we use many ground connections (balls) and divide the gates among them. For example, if we divide our 100,000 gates equally between 100 ground balls, then there are only 1000 gates/ball. If only 10% of those switch at the same time (100 gates), then the ground bounce, shown in Fig. 5.25b, is only 10 mV, 0.4% of the supply voltage, which can be tolerated.

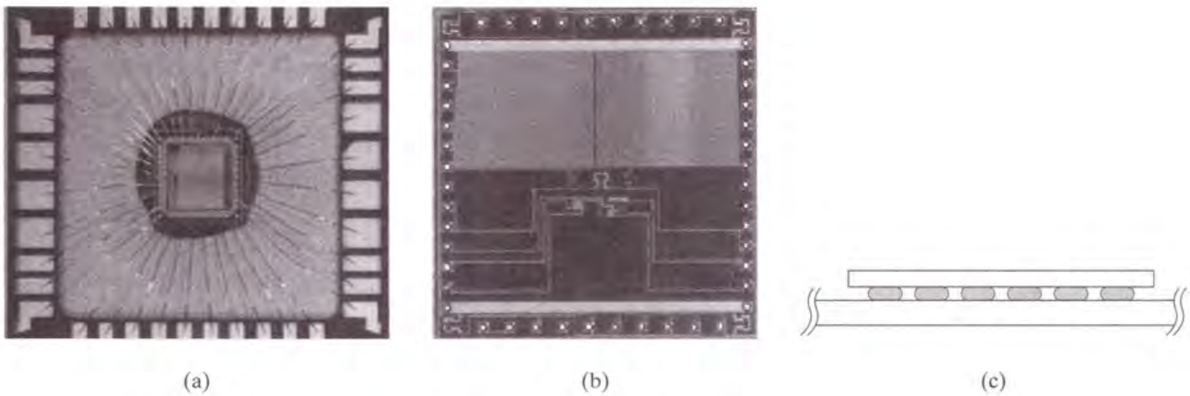


Figure 5.24
In wirebonded packages, (a) small wirebonds connect the die to the package. In flip-chip attachment, small solder balls are placed on the die's contact points (b), then the die is flipped onto the package (c). Under heat, the solder flows, forming the electrical contacts.

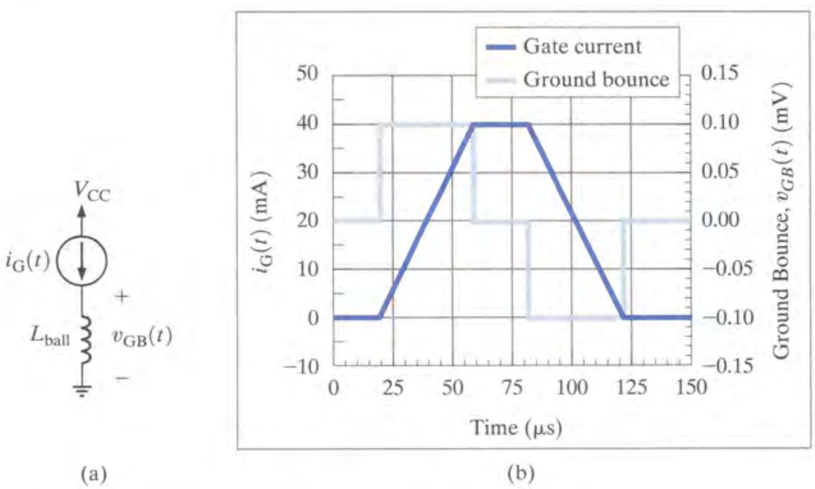


Figure 5.25
(a) An equivalent circuit for a single logic gate and (b) the resulting ground bounce.

LEARNING Check

Summary

- The important (dual) relationships for capacitors and inductors are as follows:

$$q = Cv$$

$$i(t) = C\frac{dv(t)}{dt} \qquad v(t) = L\frac{di(t)}{dt}$$

$$v(t) = \frac{1}{C}\int_{-\infty}^{t} i(x)\,dx \qquad i(t) = \frac{1}{L}\int_{-\infty}^{t} v(x)dx$$

$$p(t) = Cv(t)\frac{dv(t)}{dt} \qquad p(t) = Li(t)\frac{di(t)}{dt}$$

$$w_C(t) = \tfrac{1}{2}Cv^2(t) \qquad w_L(t) = \tfrac{1}{2}Li^2(t)$$

- The passive sign convention is used with capacitors and inductors.
- In dc steady state a capacitor looks like an open circuit and an inductor looks like a short circuit.
- Leakage resistance is present in practical capacitors and inductors.
- When capacitors are interconnected, their equivalent capacitance is determined as follows: Capacitors in series combine like resistors in parallel and capacitors in parallel combine like resistors in series.
- When inductors are interconnected, their equivalent inductance is determined as follows: Inductors in series combine like resistors in series, and inductors in parallel combine like resistors in parallel.
- *RC* operational amplifier circuits can be used to differentiate or integrate an electrical signal.

Problems

For solutions and additional help on problems marked with ► go to www.wiley.com/college/irwin

SECTION 5.1

5.1 ► **An uncharged 100-μF capacitor is charged by a constant current of 1 mA. Find the voltage across the capacitor after 4 s.**

5.2 A 12-μF capacitor has an accumulated charge of 480 μC. Determine the voltage across the capacitor.

5.3 A capacitor has an accumulated charge of 600 μC with 5 V across it. What is the value of capacitance?

5.4 A 25-μF capacitor initially charged to −10 V is charged by a constant current of 2.5 μA. Find the voltage across the capacitor after $2\frac{1}{2}$ min.

5.5 ► **The energy that is stored in a 25-μF capacitor is $w(t) = 12\sin^2 377t$ J. Find the current in the capacitor.**

5.6 An uncharged 10-mF capacitor is charged by the current $i(t) = 10\cos 377t$ mA. Find (a) the expression for the voltage across the capacitor and (b) the expression for the power.

5.7 The voltage across a 100-μF capacitor is given by the expression $v(t) = 120\sin 377t$ V. Find (a) the current in the capacitor and (b) the expression for the energy stored in the element.

5.8 A capacitor is charged by a constant current of 2 mA and results in a voltage increase of 12 V in a 10-s interval. What is the value of the capacitance?

5.9 ► **The current in a 100-μF capacitor is shown in Fig. P5.9. Determine the waveform for the voltage across the capacitor if it is initially uncharged.**

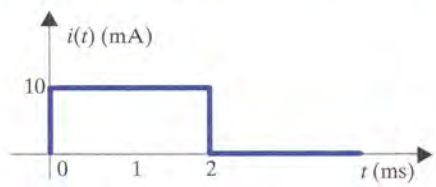


Figure P5.9

5.10 The voltage across a 100-μF capacitor is shown in Fig. P5.10. Compute the waveform for the current in the capacitor.

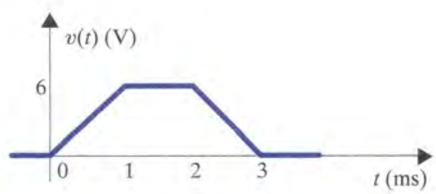


Figure P5.10

5.11 The voltage across a 6-μF capacitor is shown in Fig. P5.11. Compute the waveform for the current in the capacitor.

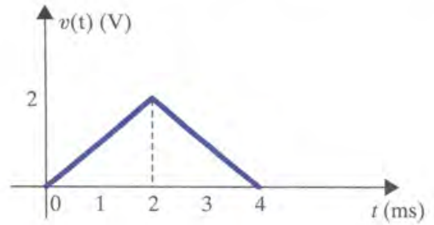


Figure P5.11

5.12 The voltage across a 50-μF capacitor is shown in Fig. P5.12. Determine the current waveform.

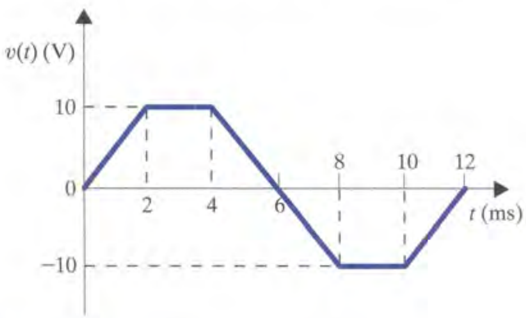


Figure P5.12

5.13 **The voltage across a 2-μF capacitor is given by the waveform in Fig. P5.13. Compute the current waveform.**

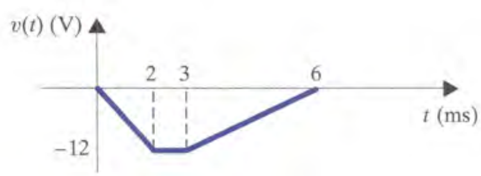


Figure P5.13

5.14 The voltage across a 0.1-F capacitor is given by the waveform in Fig. P5.14. Find the waveform for the current in the capacitor.

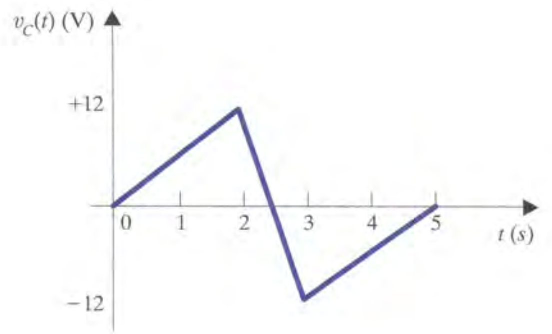


Figure P5.14

5.15 The waveform for the current in a 200-μF capacitor is shown in Fig. P5.15. Determine the waveform for the capacitor voltage.

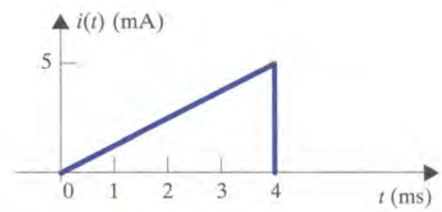


Figure P5.15

5.16 Draw the waveform for the current in a 12-μF capacitor when the capacitor voltage is as described in Fig. P5.16.

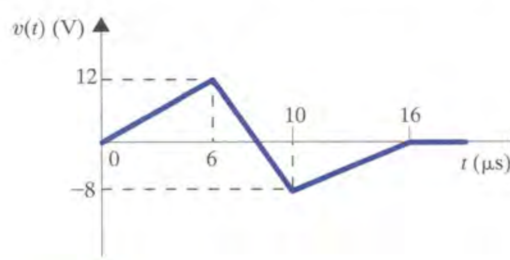


Figure P5.16

5.17 ► **Draw the waveform for the current in a 3-μF capacitor when the voltage across the capacitor is given in Fig. P5.17.**

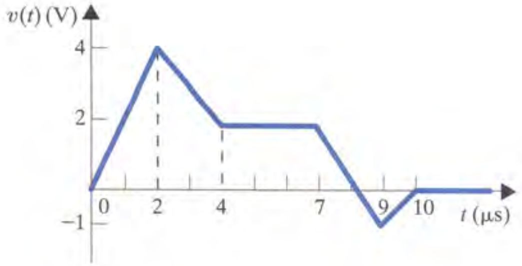


Figure P5.17

5.18 The waveform for the current in a 100-μF initially uncharged capacitor is shown in Fig. P5.18. Determine the waveform for the capacitor's voltage.

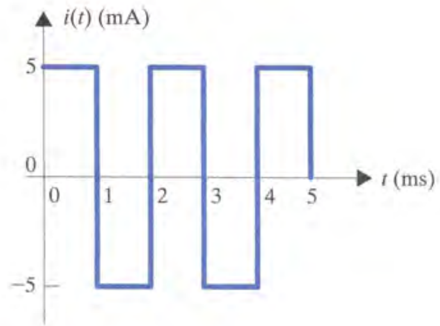


Figure P5.18

5.19 The voltage across a 6-μF capacitor is given by the waveform in Fig. P5.19. Plot the waveform for the capacitor current.

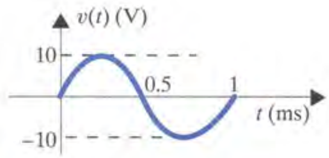


Figure P5.19

SECTION 5.2

5.20 The current in an inductor changes from 0 to 200 mA in 4 ms and induces a voltage of 100 mV. What is the value of the inductor?

5.21 ► **The current in a 100-mH inductor is $i(t) = 2\sin 377t$ A. Find (a) the voltage across the inductor and (b) the expression for the energy stored in the element.**

5.22 A 10-mH inductor has a sudden current change from 200 mA to 100 mA in 1 ms. Find the induced voltage.

5.23 The induced voltage across a 10-mH inductor is $v(t) = 120\cos 377t$ V. Find (a) the expression for the inductor current and (b) the expression for the power.

5.24 The current in a 25-mH inductor is given by the expressions

$$i(t) = 0 \qquad t < 0$$
$$i(t) = 10(1 - e^{-t})\text{mA} \qquad t > 0$$

Find (a) the voltage across the inductor and (b) the expression for the energy stored in it.

5.25 ► **Given the data in the previous problem, find the voltage across the inductor and the energy stored in it after 1 s.**

5.26 The current in a 50-μH inductor is specified as follows:

$$i(t) = 0 \qquad t < 0$$
$$i(t) = 2te^{-4t}\text{A} \qquad t > 0$$

Find (a) the voltage across the inductor, (b) the time at which the current is a maximum, and (c) the time at which the voltage is a minimum.

5.27 The current

$$i(t) = 0 \qquad t < 0$$
$$i(t) = 100e^{-t/10}\text{A} \qquad t > 0$$

is present in a 150-mH inductor. Find both the voltage across the inductor and the energy stored in it after 5 sec.

5.28 The current in a 10-mH inductor is shown in Fig. P5.28. Find the voltage across the inductor.

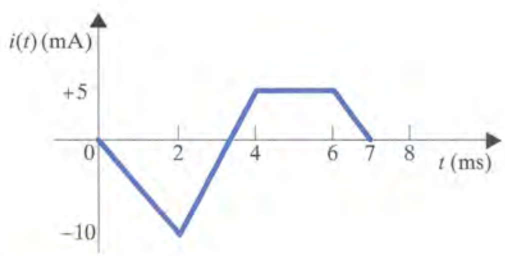


Figure P5.28

5.29 ► **The current in a 50-mH inductor is given in Fig. P5.29. Sketch the inductor voltage.**

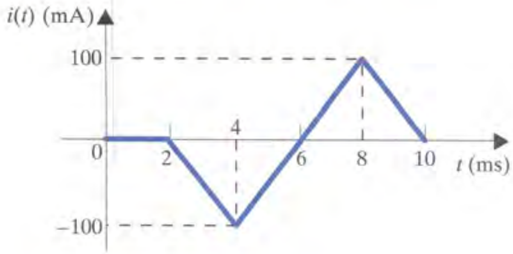


Figure P5.29

5.30 The current in a 16-mH inductor is given by the waveform in Fig. P5.30. Find the waveform for the voltage across the inductor.

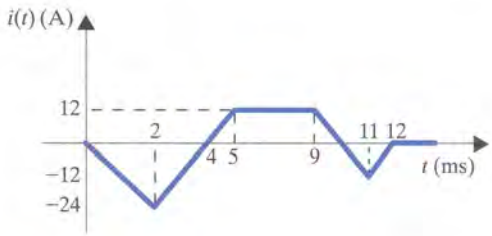


Figure P5.30

5.31 Draw the waveform for the voltage across a 10-mH inductor when the inductor current is given by the waveform shown in Fig. P5.31.

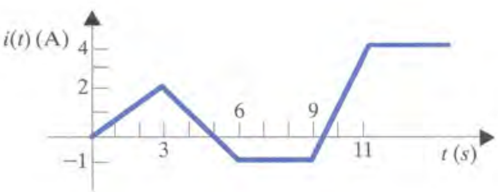


Figure P5.31

5.32 The voltage across a 10-mH inductor is shown in Fig. P5.32. Determine the waveform for the inductor current. $v(t) = 0$, $t < 0$.

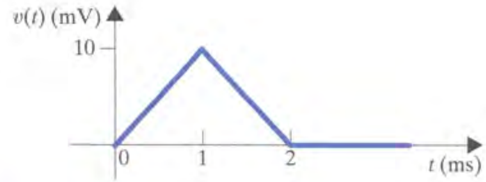


Figure P5.32

5.33 ► **The waveform for the voltage across a 20-mH inductor is shown in Fig. P5.33. Compute the waveform for the inductor current. $v(t) = 0, t < 0$.**

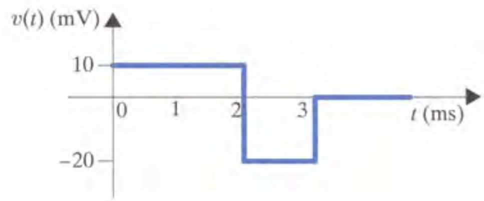


Figure P5.33

5.34 The voltage across a 2-H inductor is given by the waveform shown in Fig. P5.34. Find the waveform for the current in the inductor. $v(t) = 0, t < 0$.

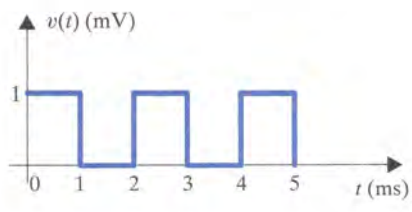


Figure P5.34

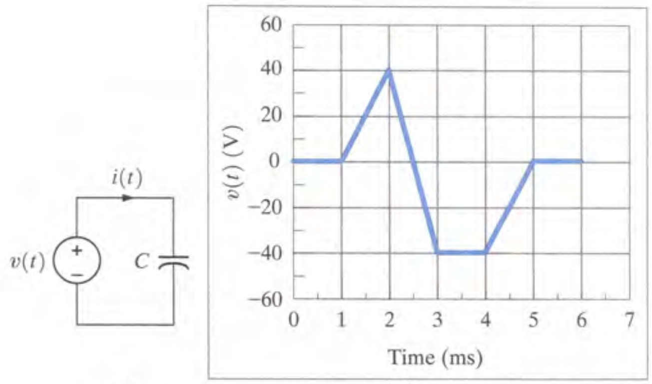
5.35 Find the possible capacitance range of the following capacitors.

(a) 0.068 μF with a tolerance of 10%

(b) 120 pF with a tolerance of 20%

(c) 39 μF with a tolerance of 20%

5.36 The capacitor in Fig. P5.36a is 51 nF with a tolerance of 10%. Given the voltage waveform in Fig. P5.36b, graph the current $i(t)$ for the minimum and maximum capacitor values.



(a) (b)

Figure P5.36

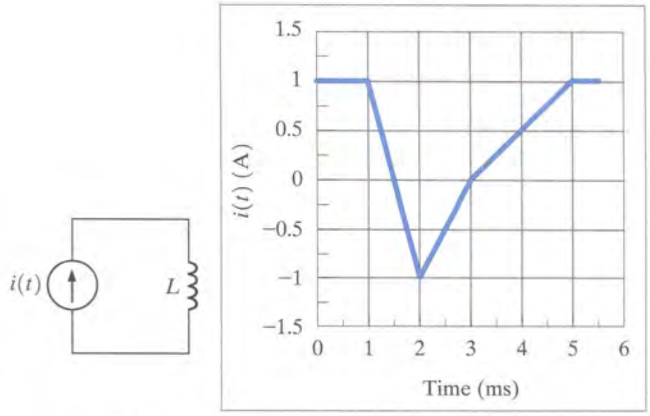
5.37 **Find the possible inductance range of the following capacitors.**

(a) 10 mH with a tolerance of 10%

(b) 2.0 nH with a tolerance of 5%

(c) 68 μH with a tolerance of 10%

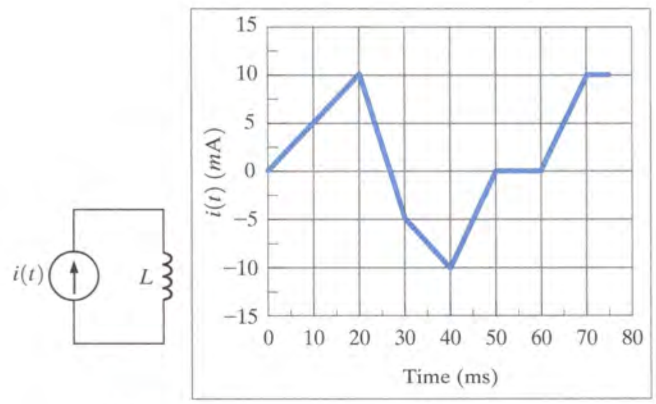
5.38 The inductor in Fig. P5.38a is 330 μH with a tolerance of 5%. Given the current waveform in Fig. 5.38b, graph the voltage $v(t)$ for the minimum and maximum inductor values.



(a) (b)

Figure P5.38

5.39 The inductor in Fig. P5.39a is 4.7 μH with a tolerance of 20%. Given the current waveform in Fig. 5.39b, graph the voltage $v(t)$ for the minimum and maximum inductor values.



(a) (b)

Figure P5.39

SECTION 5.3

5.40 What values of capacitance can be obtained by interconnecting a 4-μF capacitor, a 6-μF capacitor, and a 12-μF capacitor?

5.41 ► **Given a 1-, 3-, and 4-μF capacitor, can they be interconnected to obtain an equivalent 2-μF capacitor?**

5.42 Given four 2-μF capacitors, find the maximum value and minimum value that can be obtained by interconnecting the capacitors in series/parallel combinations.

5.43 The two capacitors in Fig. P5.43 were charged and then connected as shown. Determine the equivalent capacitance, the initial voltage at the terminals, and the total energy stored in the network.

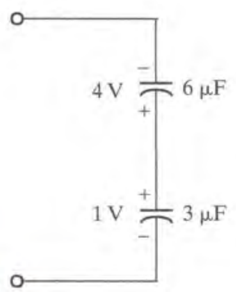


Figure P5.43

5.44 The two capacitors shown in Fig. P5.44 have been connected for some time and have reached their present values. Find V_o.

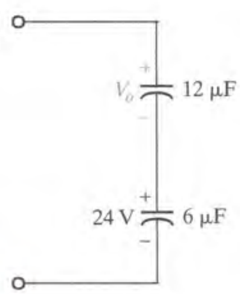


Figure P5.44

5.45 ► **The three capacitors shown in Fig. P5.45 have been connected for some time and have reached their present values. Find V_1 and V_2.**

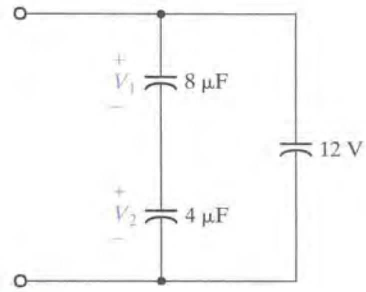


Figure P5.45

5.46 Select the value of C to produce the desired total capacitance of $C_T = 2$ μF in the circuit in Fig. P5.46.

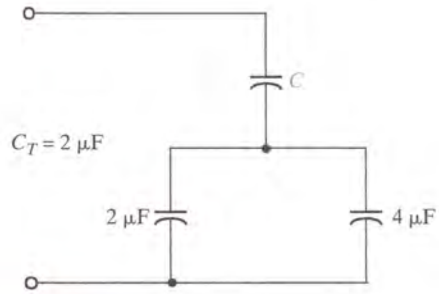


Figure P5.46

5.47 Select the value of C to produce the desired total capacitance of $C_T = 1$ μF in the circuit in Fig. P5.47.

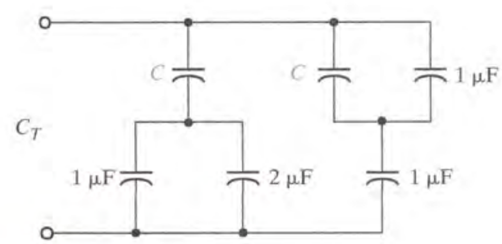


Figure P5.47

5.48 Find the equivalent capacitance at terminals A-B in Fig. P5.48.

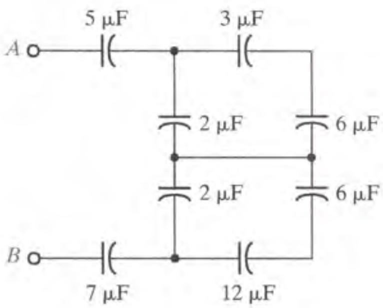


Figure P5.48

5.49 **Determine the total capacitance of the network in Fig. P5.49.**

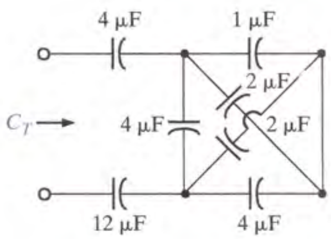


Figure P5.49

5.50 Find C_T in the network in Fig. P5.50 if (a) the switch is open and (b) the switch is closed.

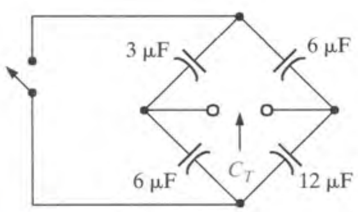


Figure P5.50

5.51 Find the total capacitance C_T of the network in Fig. P5.51.

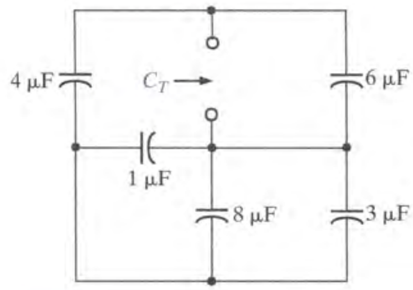


Figure P5.51

5.52 Compute the equivalent capacitance of the network in Fig. P5.52 if all the capacitors are 4 μF.

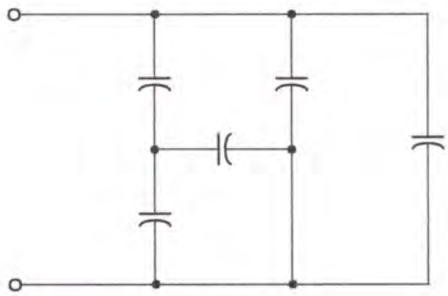


Figure P5.52

5.53 **If all the capacitors in Fig. P5.53 are 6 μF, find C_{eq}.**

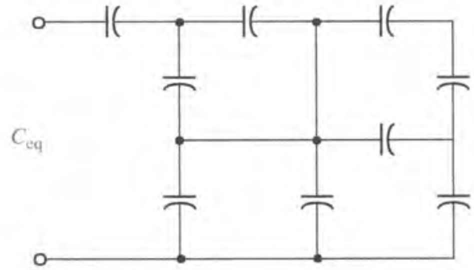


Figure P5.53

5.54 Given the capacitors in Fig. P5.54 are $C_1 = 2.0\ \mu F$ with a tolerance of 2% and $C_2 = 2.0\ \mu F$ with a tolerance of 20%, find the following.

(a) The nominal value of C_{eq}.

(b) The minimum and maximum possible values of C_{eq}.

(c) The percent errors of the minimum and maximum values.

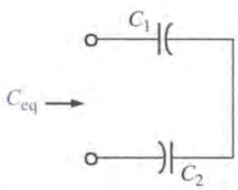


Figure P5.54

5.55 The capacitor values for the network in Fig. P5.55 are $C_1 = 0.1\ \mu F$ with a tolerance of 10%, $C_2 = 0.33\ \mu F$ with a tolerance of 20%, and $C_3 = 1\ \mu F$ with a tolerance of 10%. Find the following.

(a) The nominal value of C_{eq}.

(b) The minimum and maximum possible values of C_{eq}.

(c) The percent errors of the minimum and maximum values.

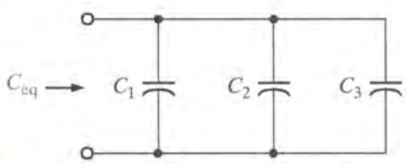


Figure P5.55

5.56 Select the value of L that produces a total inductance of $L_T = 10$ mH in the circuit in Fig. P5.56.

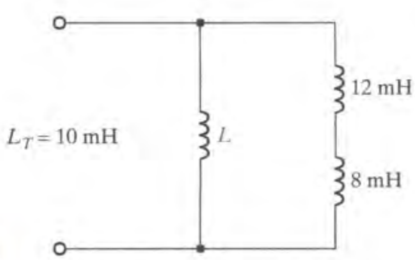


Figure P5.56

5.57 ► **Find the value of L in the network in Fig. P5.57 so that the total inductance L_T will be 2 mH.**

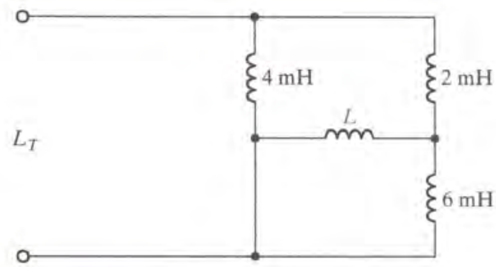


Figure P5.57

5.58 Find the value of L in the network in Fig. P5.58 so that the value of L_T will be 2 mH.

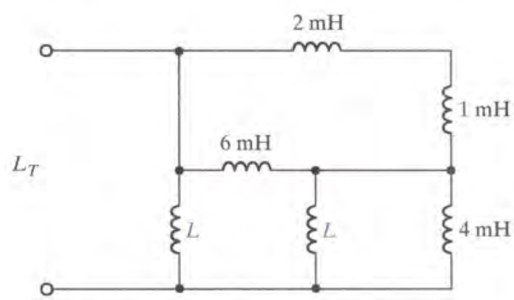


Figure P5.58

5.59 Determine the inductance at terminals A-B in the network in Fig. P5.59.

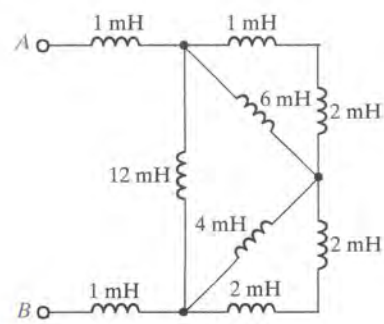


Figure P5.59

5.60 Compute the equivalent inductance of the network in Fig. P5.60 if all inductors are 4 mH.

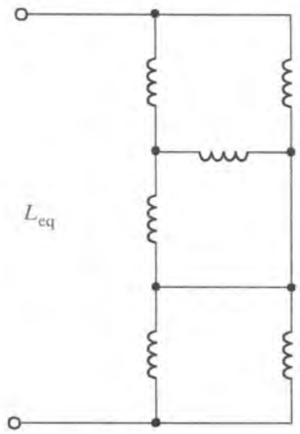


Figure P5.60

5.61 ► **Determine the inductance at terminals *A-B* in the network in Fig P5.61.**

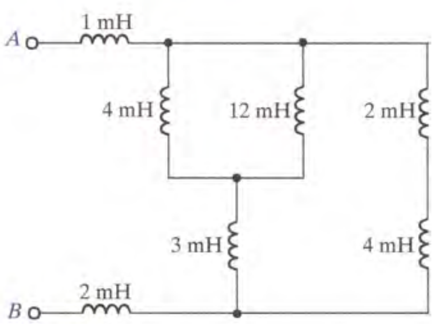


Figure P5.61

5.62 Find the total inductance at the terminals of the network in Fig. P5.62.

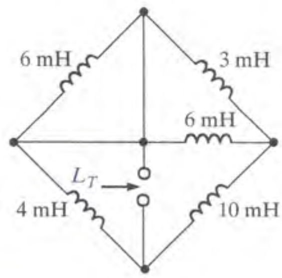


Figure P5.62

5.63 Given the network shown in Fig. P5.63, find (a) the equivalent inductance at terminals *A-B* with terminals *C-D* short circuited, and (b) the equivalent inductance at terminals *C-D* with terminals *A-B* open circuited.

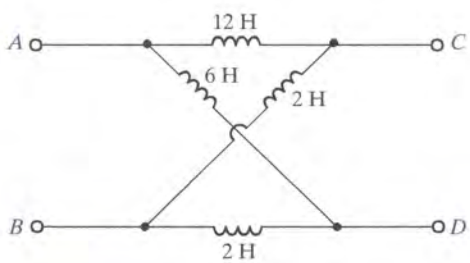


Figure P5.63

SECTION 5.4

5.64 For the network in Fig. P5.64, choose C such that

$$v_o = -10 \int v_S \, dt$$

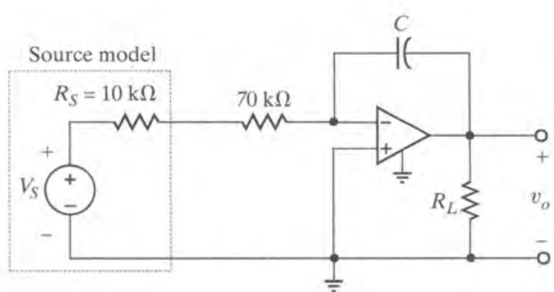


Figure P5.64

5.65 For the network in Fig. P5.65, $v_S(t) = 120 \cos 377t$ V. Find $v_o(t)$.

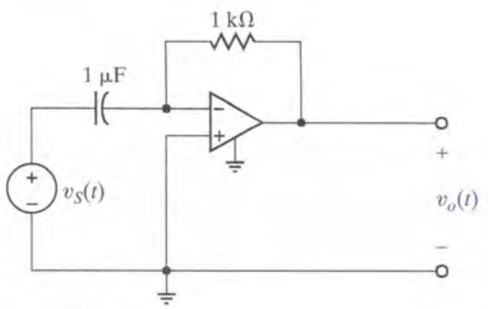


Figure P5.65

5.66 For the network in Fig. P5.66, $v_S(t) = 115 \sin 377t$ V. Find $v_o(t)$.

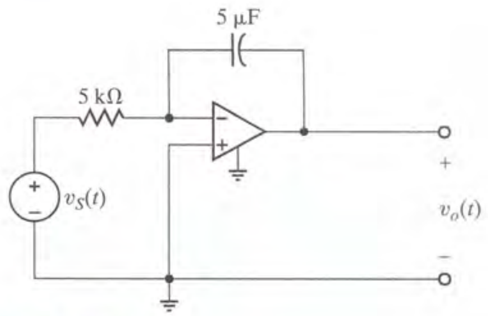


Figure P5.66

Typical Problems Found on the FE Exam

5FE-1 ► **Given three capacitors with values 2 μF, 4 μF, and 6 μF, can the capacitors be interconnected so that the combination is an equivalent 3 μF?**

5FE-2 The current pulse shown in Fig. 5PFE-2 is applied to a 1-μF capacitor. Determine the charge on the capacitor and the energy stored.

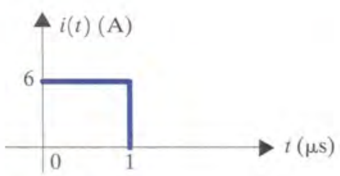


Figure 5PFE-2

5FE-3 ► **The two capacitors shown in Fig. 5PFE-3 have been connected for some time and have reached their present values. Determine the energy stored in the unknown capacitor C_x.**

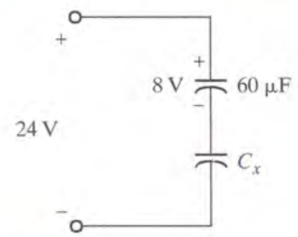


Figure 5PFE-3

6 First- and Second-Order Transient Circuits

LEARNING Goals

We now consider circuits that are in transition from one state to another; that is, a circuit may have no forcing function and we suddenly apply one, or we instantly remove from a circuit the source of energy. The study of the circuit behavior in this transition phase we call a transient analysis. This transition is affected by the presence of a capacitor or an inductor, or both, since these two elements are capable of storing energy and releasing it over some interval of time. Our analysis includes both first-order circuits, those containing a single capacitor or inductor, and second-order circuits, those in which both a capacitor and inductor are present.

6.1 Introduction

In this chapter we perform what is normally referred to as a transient analysis. We begin our analysis with first-order circuits—that is, those that contain only a single storage element. When only a single storage element is present in the network, the network can be described by a first-order differential equation.

Our analysis involves an examination and description of the behavior of a circuit as a function of time after a sudden change in the network occurs due to switches opening or closing. Because of the presence of one or more storage elements, the circuit response to a sudden change will go through a transition period prior to settling down to a steady-state value. It is this transition period that we will examine carefully in our transient analysis.

One of the important parameters that we will examine in our transient analysis is the circuit's time constant. This is a very important network parameter because it tells us how fast the circuit will respond to changes. We can contrast two very different systems to obtain a feel for the parameter. For example, consider the model for a room air-conditioning system and the model for a single-transistor stage of amplification in a computer chip. If we change the setting for the air conditioner from 70 degrees to 60 degrees, the unit will come on and the room will begin to cool. However, the temperature measured by a thermometer in the room will fall very slowly and, thus, the time required to reach the desired temperature is long. However, if we send a trigger signal to a transistor to change state, the action may take only a few nanoseconds. These two systems will have vastly different time constants.

Our analysis of first-order circuits begins with the presentation of two techniques for performing a transient analysis: the differential equation approach, in which a differential equation is written and solved for each network, and a step-by-step approach, which takes advantage of the known form of the solution in every case. In the second-order case, both an inductor and capacitor are present simultaneously and the network is described by a second-order differential equation. Although the *RLC* circuits are more complicated than the first-order single storage circuits, we will follow a development similar to that used in the first-order case.

Our presentation will deal only with very simple circuits, since the analysis can quickly become complicated for networks that contain more than one loop or one nonreference node. Furthermore, we will demonstrate a much simpler method for handling these circuits when we cover the Laplace transform later in this book. We will analyze several networks in which the parameters have been chosen to illustrate the different types of circuit response. In addition, we will extend our PSPICE analysis techniques to the analysis of transient circuits. Finally, a number of application-oriented examples are presented and discussed.

We begin our discussion by recalling that in Chapter 5 we found that capacitors and inductors were capable of storing electric energy. In the case of a charged capacitor, the energy is stored in the electric field that exists between the positively and negatively charged plates. This stored energy can be released if a circuit is somehow connected across the capacitor that provides a path through which the negative charges move to the positive charges. As we know, this movement of charge constitutes a current. The rate at which the energy is discharged is a direct function of the parameters in the circuit that is connected across the capacitor's plates.

As an example, consider the flash circuit in a camera. Recall that the operation of the flash circuit, from a user standpoint, involves depressing the push button on the camera that triggers both the shutter and the flash and then waiting a few seconds before repeating the process to take the next picture. This operation can be modeled using the circuit in Fig. 6.1a. The voltage source and resistor R_S model the batteries that power the camera and flash. The capacitor models the energy storage, the switch models the push button, and finally the resistor R models the xenon flash lamp. Thus, if the capacitor is charged, when the switch is closed, the capacitor voltage drops and energy is released through the xenon lamp, producing the flash. In practice this

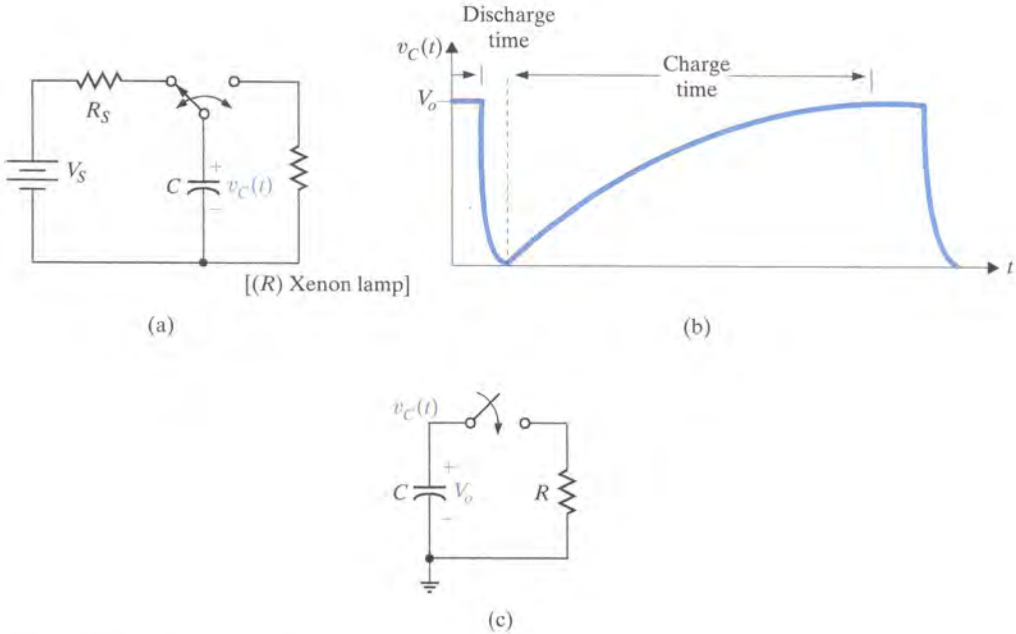


Figure 6.1
Diagrams used to describe a camera's flash circuit.

energy release takes about a millisecond and the discharge time is a function of the elements in the circuit. When the push button is released and the switch is then opened, the battery begins to recharge the capacitor. Once again, the time required to charge the capacitor is a function of the circuit elements. The discharge and charge cycles are graphically illustrated in Fig. 6.1b. Although the discharge time is very fast, it is not instantaneous. To provide further insight into this phenomenon, consider what we might call a *free-body diagram* of the right half of the network in Fig. 6.1a as shown in Fig. 6.1c (that is, a charged capacitor that is discharged through a resistor). When the switch is closed, KCL for the circuit is

$$C\frac{dv_C(t)}{dt} + \frac{v_C(t)}{R} = 0$$

or

$$\frac{dv_C(t)}{dt} + \frac{1}{RC}v_C(t) = 0$$

We will demonstrate in the next section that the solution of this equation is

$$v_C(t) = V_o e^{-t/RC}$$

Note that this function is a decaying exponential and the rate at which it decays is a function of the values of R and C. The product RC is a very important parameter, and we will give it a special name in the following discussions.

6.2 First-Order Circuits

GENERAL FORM OF THE RESPONSE EQUATIONS In our study of first-order transient circuits we will show that the solution of these circuits (i.e., finding a voltage or current) requires us to solve a first-order differential equation of the form

$$\frac{dx(t)}{dt} + ax(t) = f(t) \qquad \textbf{6.1}$$

Although there are a number of techniques for solving an equation of this type, we will obtain a general solution that we will then employ in two different approaches to transient analysis.

A fundamental theorem of differential equations states that if $x(t) = x_p(t)$ is any solution to Eq. (6.1), and $x(t) = x_c(t)$ is any solution to the homogeneous equation

$$\frac{dx(t)}{dt} + ax(t) = 0 \qquad \textbf{6.2}$$

then

$$x(t) = x_p(t) + x_c(t) \qquad \textbf{6.3}$$

is a solution to the original Eq. (6.1). The term $x_p(t)$ is called *the particular integral solution*, or forced response, and $x_c(t)$ is called the *complementary solution*, or natural response.

At the present time we confine ourselves to the situation in which $f(t) = A$ (i.e., some constant). The general solution of the differential equation then consists of two parts that are obtained by solving the two equations

$$\frac{dx_p(t)}{dt} + ax_p(t) = A \qquad \textbf{6.4}$$

$$\frac{dx_c(t)}{dt} + ax_c(t) = 0 \qquad \textbf{6.5}$$

Since the right-hand side of Eq. (6.4) is a constant, it is reasonable to assume that the solution $x_p(t)$ must also be a constant. Therefore, we assume that

$$x_p(t) = K_1 \qquad \textbf{6.6}$$

Substituting this constant into Eq. (6.4) yields

$$K_1 = \frac{A}{a} \qquad \textbf{6.7}$$

Examining Eq. (6.5), we note that

$$\frac{dx_c(t)/dt}{x_c(t)} = -a \qquad \textbf{6.8}$$

This equation is equivalent to

$$\frac{d}{dt}\left[\ln x_c(t)\right] = -a$$

Hence,

$$\ln x_c(t) = -at + c$$

and therefore,

$$x_c(t) = K_2 e^{-at} \qquad 6.9$$

Thus, a solution of Eq. (6.1) is

$$\begin{aligned} x(t) &= x_p(t) + x_c(t) \\ &= \frac{A}{a} + K_2 e^{-at} \qquad 6.10 \end{aligned}$$

The constant K_2 can be found if the value of the independent variable $x(t)$ is known at one instant of time.

Equation (6.10) can be expressed in general in the form

$$x(t) = K_1 + K_2 e^{-t/\tau} \qquad 6.11$$

Once the solution in Eq. (6.11) is obtained, certain elements of the equation are given names that are commonly employed in electrical engineering. For example, the term K_1 is referred to as the *steady-state solution*: the value of the variable $x(t)$ as $t \to \infty$ when the second term becomes negligible. The constant τ is called the *time constant* of the circuit. Note that the second term in Eq. (6.11) is a decaying exponential that has a value, if $\tau > 0$, of K_2 for $t = 0$ and a value of 0 for $t = \infty$. The rate at which this exponential decays is determined by the time constant τ. A graphical picture of this effect is shown in Fig. 6.2a. As can be seen from the figure, the value of $x_c(t)$ has fallen from K_2 to a value of $0.368K_2$ in one time constant, a drop of 63.2%. In two time constants the value of $x_c(t)$ has fallen to $0.135K_2$, a drop of 63.2% from the value at time $t = \tau$. This means that the gap between a point on the curve and the final value of the curve is closed by 63.2% each time constant. Finally, after five time constants, $x_c(t) = 0.0067K_2$, which is less than 1%.

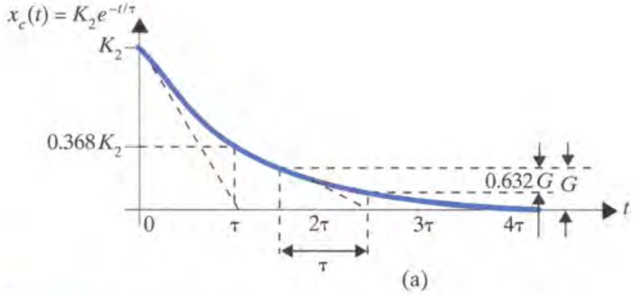
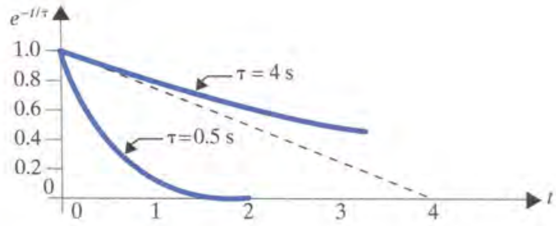


Figure 6.2 *Time-constant illustrations.*



(b)

Figure 6.2 *Continued*

An interesting property of the exponential function shown in Fig. 6.2a is that the initial slope of the curve intersects the time axis at a value of $t = \tau$. In fact, we can take any point on the curve, not just the initial value, and find the time constant by finding the time required to close the gap by 63.2%. Finally, the difference between a small time constant (i.e., fast response) and a large time constant (i.e., slow response) is shown in Fig. 6.2b. These curves indicate that if the circuit has a small time constant, it settles down quickly to a steady-state value. Conversely, if the time constant is large, more time is required for the circuit to settle down or reach steady state. In any case, note that the circuit response essentially reaches steady state within five time constants (i.e., 5τ).

Note that the previous discussion has been very general in that no particular form of the circuit has been assumed—except that it results in a first-order differential equation.

ANALYSIS TECHNIQUES

The Differential Equation Approach Equation (6.11) defines the general form of the solution of first-order transient circuits; that is, it represents the solution of the differential equation that describes an unknown current or voltage *anywhere in the network*. One of the ways that we can arrive at this solution is to solve the equations that describe the network behavior using what is often called the *state-variable approach*. In this technique we write the equation for the voltage across the capacitor and/or the equation for the current through the inductor. Recall from Chapter 5 that these quantities cannot change instantaneously. Let us first illustrate this technique in the general sense and then examine two specific examples.

Consider the circuit shown in Fig. 6.3a. At time $t = 0$, the switch closes. The KCL equation that describes the capacitor voltage for time $t > 0$ is

$$C\frac{dv(t)}{dt} + \frac{v(t) - V_S}{R} = 0$$

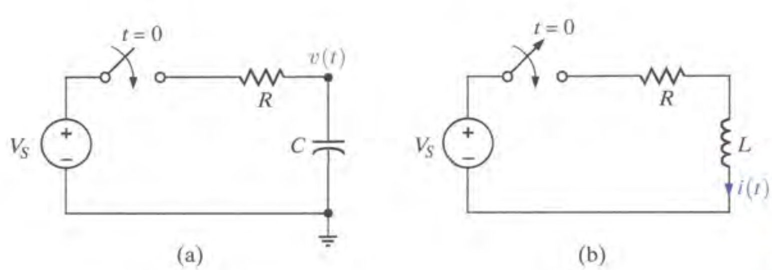


Figure 6.3
RC and RL circuits.

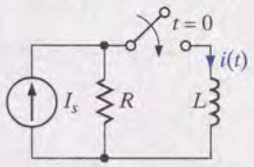
LEARNING by Doing

D 6.1 If the capacitor in the network in Fig. 6.3a is initially charged to $V_S/2$, find the complete solutions for $v(t)$.

ANSWER

$v(t) = V_S - \frac{V_S}{2} e^{-t/RC}$

D 6.2 Find $i(t)$ for $t > 0$ in the following network:



ANSWER

$i(t) = I_S(1 - e^{-\frac{R}{L}t})$

or

$$\frac{dv(t)}{dt} + \frac{v(t)}{RC} = \frac{V_S}{RC}$$

From our previous development, we assume that the solution of this first-order differential equation is of the form

$$v(t) = K_1 + K_2 e^{-t/\tau}$$

Substituting this solution into the differential equation yields

$$-\frac{K_2}{\tau} e^{-t/\tau} + \frac{K_1}{RC} + \frac{K_2}{RC} e^{-t/\tau} = \frac{V_S}{RC}$$

Equating the constant and exponential terms, we obtain

$$K_1 = V_S$$

$$\tau = RC$$

Therefore,

$$v(t) = V_S + K_2 e^{-t/RC}$$

where V_S is the steady-state value and RC is the network's time constant. K_2 is determined by the initial condition of the capacitor. For example, if the capacitor is initially uncharged (that is, the voltage across the capacitor is zero at $t = 0$), then

$$0 = V_S + K_2$$

or

$$K_2 = -V_S$$

Hence, the complete solution for the voltage $v(t)$ is

$$v(t) = V_S - V_S e^{-t/RC}$$

The circuit in Fig. 6.3b can be examined in a similar manner. The KVL equation that describes the inductor current for $t > 0$ is

$$L \frac{di(t)}{dt} + Ri(t) = V_S$$

A development identical to that just used yields

$$i(t) = \frac{V_S}{R} + K_2 e^{-\left(\frac{R}{L}\right)t}$$

where V_S/R is the steady-state value and L/R is the circuit's time constant. If there is no initial current in the inductor, then at $t = 0$

$$0 = \frac{V_S}{R} + K_2$$

and

$$K_2 = \frac{-V_S}{R}$$

Hence,

$$i(t) = \frac{V_S}{R} - \frac{V_S}{R} e^{-\frac{R}{L}t}$$

is the complete solution. Note that if we wish to calculate the voltage across the resistor, then

$$\begin{aligned} v_R(t) &= Ri(t) \\ &= V_S\left(1 - e^{-\frac{R}{L}t}\right) \end{aligned}$$

LEARNING Example 6.1

Consider the circuit shown in Fig. 6.4a. Assuming that the switch has been in position 1 for a long time, at time $t = 0$ the switch is moved to position 2. We wish to calculate the current $i(t)$ for $t > 0$.

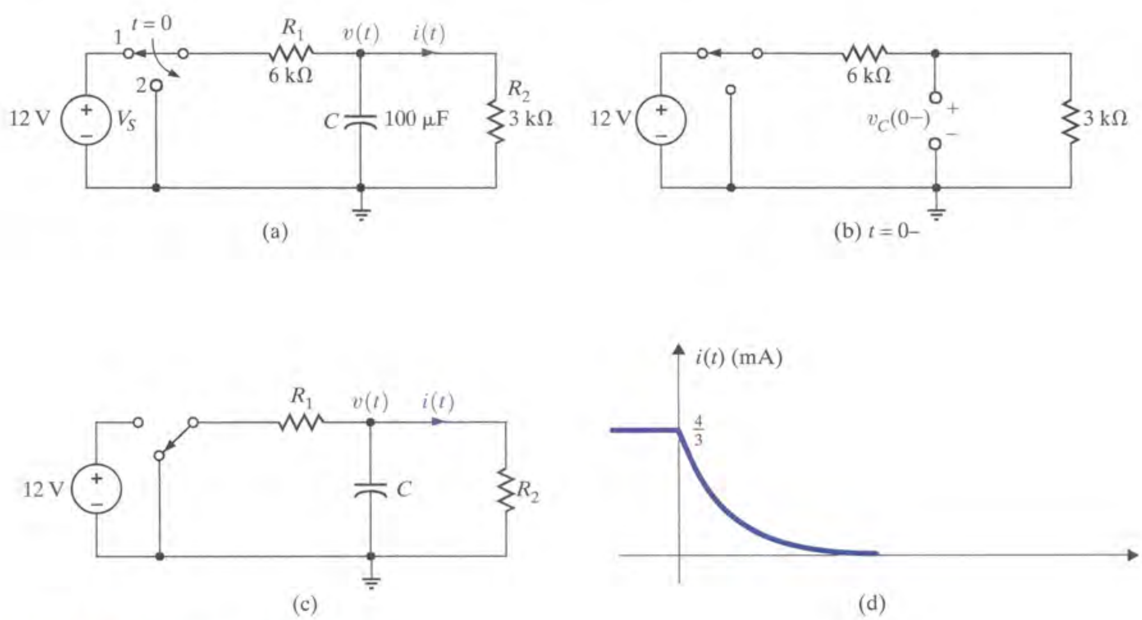


Figure 6.4 *Analysis of RC circuits.*

(continued)

SOLUTION At $t = 0-$ the capacitor is fully charged and conducts no current since the capacitor acts like an open circuit to dc. The initial voltage across the capacitor can be found using voltage division. As shown in Fig. 6.4b,

$$v_C(0-) = 12\left(\frac{3\text{k}}{6\text{k} + 3\text{k}}\right) = 4 \text{ V}$$

The network for $t > 0$ is shown in Fig. 6.4c. The KCL equation for the voltage across the capacitor is

$$\frac{v(t)}{R_1} + C\frac{dv(t)}{dt} + \frac{v(t)}{R_2} = 0$$

Using the component values, the equation becomes

$$\frac{dv(t)}{dt} + 5v(t) = 0$$

The form of the solution to this homogeneous equation is

$$v(t) = K_2 e^{-t/\tau}$$

If we substitute this solution into the differential equation, we find that $\tau = 0.2$ s. Thus,

$$v(t) = K_2 e^{-t/0.2} \text{ V}$$

Using the initial condition $v_C(0-) = v_C(0+) = 4$ V, we find that the complete solution is

$$v(t) = 4e^{-t/0.2} \text{ V}$$

Then $i(t)$ is simply

$$i(t) = \frac{v(t)}{R_2}$$

or

$$i(t) = \frac{4}{3}e^{-t/0.2} \text{ mA}$$

LEARNING Example 6.2

The switch in the network in Fig. 6.5a opens at $t = 0$. Let us find the output voltage $v_o(t)$ for $t > 0$.

SOLUTION At $t = 0-$ the circuit is in steady state and the inductor acts like a short circuit. The initial current through the inductor can be found in many ways; however, we will form a Thévenin equivalent for the part of the network to the left of the inductor, as shown in Fig. 6.5b. From this network we find that $I_1 = 4$ A and $V_{oc} = 4$ V. In addition, $R_{Th} = 1\ \Omega$. Hence, $i_L(0-)$ obtained from Fig. 6.5c is $i_L(0-) = 4/3$ A.

The network for $t > 0$ is shown in Fig. 6.5d. Note that the 4-V independent source and the 2-ohm resistor in series with it no longer have any impact on the resulting circuit. The KVL equation for the circuit is

$$-V_{S_1} + R_1 i(t) + L\frac{di(t)}{dt} + R_3 i(t) = 0$$

which with the component values reduces to

$$\frac{di(t)}{dt} + 2i(t) = 6$$

The solution to this equation is of the form

$$i(t) = K_1 + K_2 e^{-t/\tau}$$

which when substituted into the differential equation yields

$$K_1 = 3$$

$$\tau = 1/2$$

Therefore,

$$i(t) = \left(3 + K_2 e^{-2t}\right) \text{A}$$

Evaluating this function at the initial condition, which is

$$i_L(0-) = i_L(0+) = i(0) = 4/3 \text{ A}$$

we find that

$$K_2 = \frac{-5}{3}$$

Hence,

$$i(t) = \left(3 - \frac{5}{3}e^{-2t}\right) \text{A}$$

and then

$$v_o(t) = 6 - \frac{10}{3}e^{-2t} \text{ V}$$

A plot of the voltage $v_o(t)$ is shown in Fig. 6.5e.

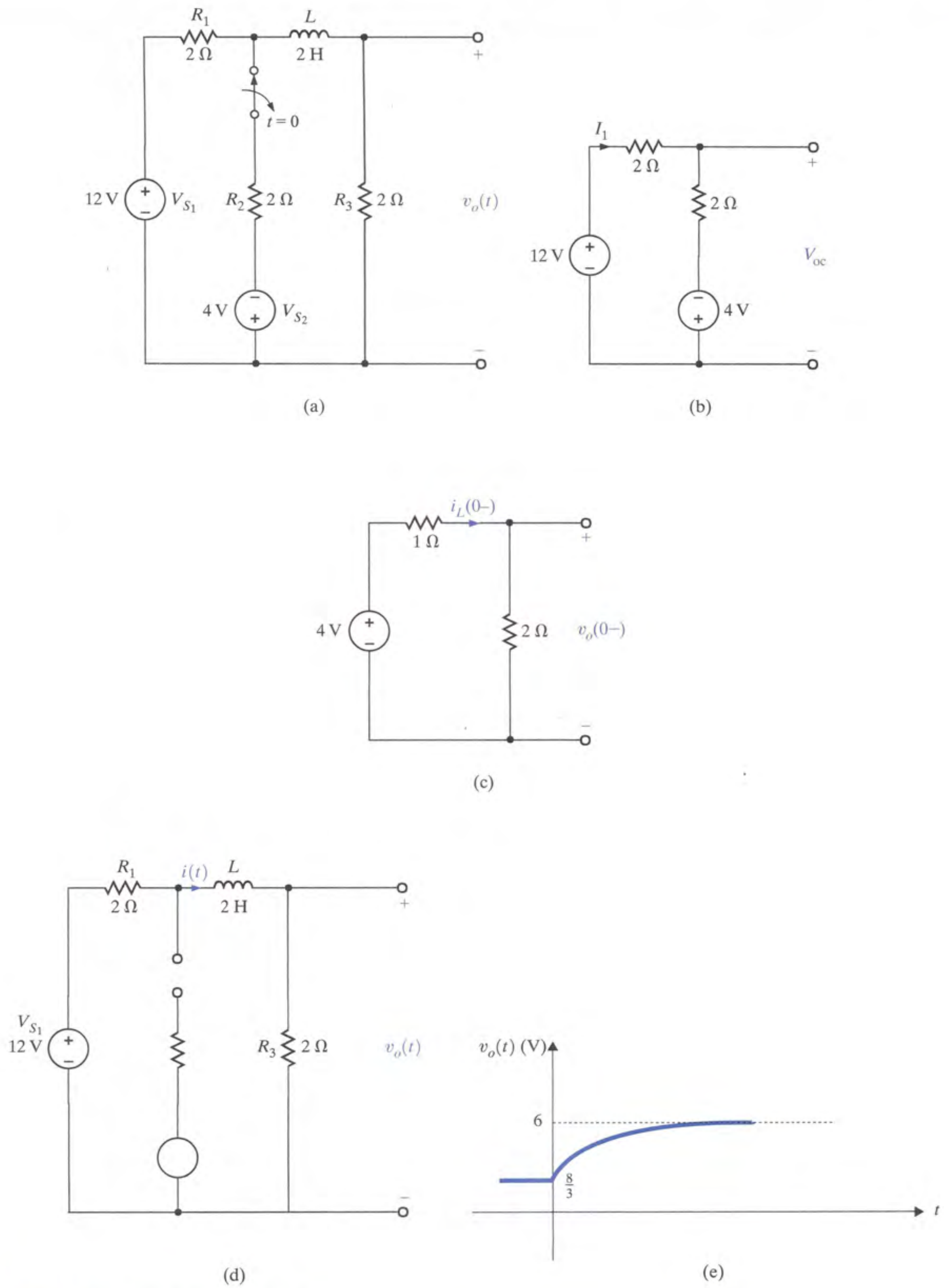


Figure 6.5 *Analysis of an RL circuit.*

LEARNING EXTENSIONS

E6.1 Find $v_C(t)$ for $t > 0$ in the circuit shown in Fig. E6.1.

ANSWER $v_C(t) = 8e^{-t/0.6}$ V.

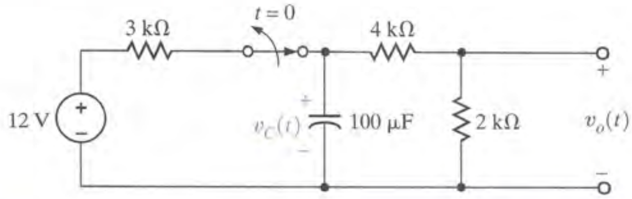


Figure E6.1

E6.2 In the circuit shown in Fig. E6.2, the switch opens at $t = 0$. Find $i_1(t)$ for $t > 0$.

ANSWER $i_1(t) = 1e^{-9t}$ A.

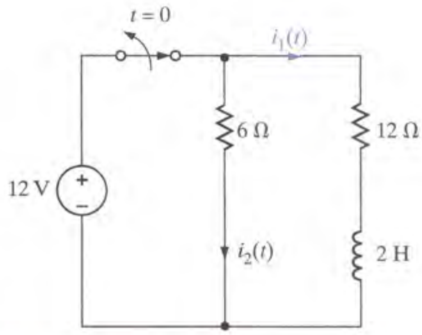


Figure E6.2

The Step-by-Step Approach In the previous analysis technique we derived the differential equation for the capacitor voltage or inductor current, solved the differential equation, and used the solution to find the unknown variable in the network. In the very methodical technique that we will now describe we will use the fact that Eq. (6.11) is the form of the solution and employ circuit analysis to determine the constants K_1, K_2, and τ.

From Eq. (6.11) we note that as $t \to \infty$, $e^{-at} \to 0$ and $x(t) = K_1$. Therefore, if the circuit is solved for the variable $x(t)$ in steady state (i.e., $t \to \infty$) with the capacitor replaced by an open circuit [v is constant and therefore $i = C(dv/dt) = 0$] or the inductor replaced by a short circuit [i is constant and therefore $v = L(di/dt) = 0$], then the variable $x(t) = K_1$. Note that since the capacitor or inductor has been removed, the circuit is a dc circuit with constant sources and resistors, and therefore only dc analysis is required in the steady-state solution.

The constant K_2 in Eq. (6.11) can also be obtained via the solution of a dc circuit in which a capacitor is replaced by a voltage source or an inductor is replaced by a current source. The value of the voltage source for the capacitor or the current source for the inductor is a known value at one instant of time. In general, we will use the initial condition value since it is generally the one known, but the value at any instant could be used. This value can be obtained in numerous ways and is often specified as input data in a statement of the problem. However, a more likely situation is one in which a switch is thrown in the circuit and the initial value of the capacitor voltage or inductor current is determined from the previous circuit (i.e., the circuit before the switch is thrown). It is normally assumed that the previous circuit has reached steady state, and therefore the voltage across the capacitor or the current through the inductor can be found in exactly the same manner as was used to find K_1.

Finally, the value of the time constant can be found by determining the Thévenin equivalent resistance at the terminals of the storage element. Then $\tau = R_{Th}C$ for an RC circuit, and $\tau = L/R_{Th}$ for an RL circuit.

Let us now reiterate this procedure in a step-by-step fashion.

Problem-Solving Strategy Using the Step-by-Step Approach

Step 1. We assume a solution for the variable $x(t)$ of the form $x(t) = K_1 + K_2e^{-t/\tau}$.

Step 2. Assuming that the original circuit has reached steady state before a switch was thrown (thereby producing a new circuit), draw this previous circuit with the capacitor replaced by an open circuit or the inductor replaced by a short circuit. Solve for the voltage across the capacitor, $v_C(0-)$, or the current through the inductor, $i_L(0-)$, prior to switch action.

Step 3. Assuming that the energy in the storage element cannot change in zero time, draw the circuit, valid only at $t = 0+$. The switches are in their new positions and the capacitor is replaced by a voltage source with a value of $v_C(0+) = v_C(0-)$ or the inductor is replaced by a current source with value $i_L(0+) = i_L(0-)$. Solve for the initial value of the variable $x(0+)$.

Step 4. Assuming that steady state has been reached after the switches are thrown, draw the equivalent circuit, valid for $t > 5\tau$, by replacing the capacitor by an open circuit or the inductor by a short circuit. Solve for the steady-state value of the variable

$$x(t)\big|_{t>5\tau} \doteq x(\infty)$$

Step 5. Since the time constant for all voltages and currents in the circuit will be the same, it can be obtained by reducing the entire circuit to a simple series circuit containing a voltage source, resistor, and a storage element (i.e., capacitor or inductor) by forming a simple Thévenin equivalent circuit at the terminals of the storage element. This Thévenin equivalent circuit is obtained by looking into the circuit from the terminals of the storage element. The time constant for a circuit containing a capacitor is $\tau = R_{Th}C$, and for a circuit containing an inductor it is $\tau = L/R_{Th}$.

Step 6. Using the results of steps 3, 4, and 5, we can evaluate the constants in step 1 as

$$x(0+) = K_1 + K_2$$

$$x(\infty) = K_1$$

and therefore, $K_1 = x(\infty)$, $K_2 = x(0+) - x(\infty)$, and hence the solution is

$$x(t) = x(\infty) + [x(0+) - x(\infty)]e^{-t/\tau}$$

Keep in mind that this solution form applies only to a first-order circuit having constant, dc sources. If the sources are not dc, the forced response will be different. Generally, the forced response is of the same form as the forcing functions (sources) and their derivatives.

LEARNING Example 6.3

Consider the circuit shown in Fig. 6.6a. The circuit is in steady state prior to time $t = 0$, when the switch is closed. Let us calculate the current $i(t)$ for $t > 0$.

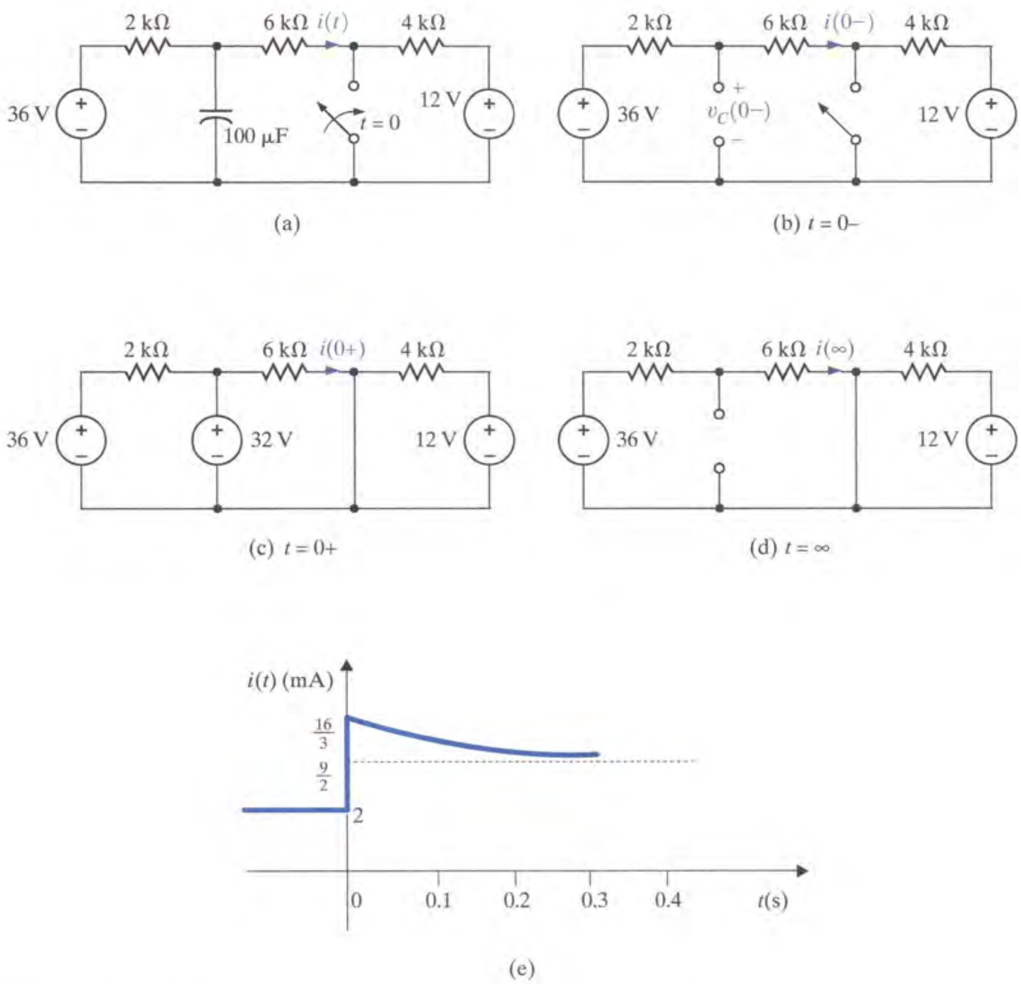


Figure 6.6 *Analysis of an RC transient circuit with a constant forcing function.*

SOLUTION

Step 1. $i(t)$ is of the form $K_1 + K_2 e^{-t/\tau}$.

Step 2. The initial voltage across the capacitor is calculated from Fig. 6.6b as

$$v_C(0-) = 36 - (2)(2)$$
$$= 32 \text{ V}$$

Step 3. The new circuit, valid only for $t = 0+$, is shown in Fig. 6.6c. The value of the voltage source that replaces the capacitor is $v_C(0-) = v_C(0+) = 32$ V. Hence,

$$i(0+) = \frac{32}{6\text{k}}$$
$$= \frac{16}{3} \text{ mA}$$

Step 4. The equivalent circuit, valid for $t > 5\tau$, is shown in Fig. 6.6d. The current $i(\infty)$ caused by the 36-V source is

$$i(\infty) = \frac{36}{2\text{k} + 6\text{k}}$$
$$= \frac{9}{2} \text{ mA}$$

Step 5. The Thévenin equivalent resistance, obtained by looking into the open-circuit terminals of the capacitor in Fig. 6.6d, is

$$R_{\text{Th}} = \frac{(2\text{k})(6\text{k})}{2\text{k} + 6\text{k}} = \frac{3}{2} \text{ k}\Omega$$

Therefore, the circuit time constant is

$$\tau = R_{\text{Th}}C$$
$$= \left(\frac{3}{2}\right)(10^3)(100)(10^{-6})$$
$$= 0.15 \text{ s}$$

Step 6.

$$K_1 = i(\infty) = \frac{9}{2} \text{ mA}$$

$$K_2 = i(0+) - i(\infty) = i(0+) - K_1$$

$$= \frac{16}{3} - \frac{9}{2}$$
$$= \frac{5}{6} \text{ mA}$$

Therefore,

$$i(t) = \frac{36}{8} + \frac{5}{6} e^{-t/0.15} \text{ mA}$$

Let us now employ MATLAB to plot the function. First, an interval for the variable t must be specified. The beginning of the interval will be chosen to be $t = 0$. The end of the interval will be chosen to be 10 times the time constant. This is realized in MATLAB as follows:

```
>>tau = 0.15
>>tend = 10*tau
```

Once the time interval has been specified, we can use MATLAB's linspace function to generate an array of evenly spaced points in the interval. The linspace command has the following syntax: `linspace(x1,x2,N)` where `x1` and `x2` denote the beginning and ending points in the interval and `N` represents the number of points. Thus, to generate an array containing 150 points in the interval [0, tend], we execute the following command:

```
>>t = linspace(0, tend, 150)
```

The MATLAB program for generating a plot of the function is

```
>>tau = 0.15;
>>tend = 10*tau;
>>t = linspace(0, tend, 150);
>>i = 9/2 + (5/6)*exp(-t/tau);
>>plot(t,i)
>>xlabel('Time (s)')
>>ylabel('Current (mA)')
```

The MATLAB plot is shown in Fig. 6.7 and can be compared to the sketch in Fig. 6.6(e). Examination of Fig. 6.6(e) indicates once again that although the voltage across the capacitor is continuous at $t = 0$, the current $i(t)$ in the 6-kΩ resistor jumps at $t = 0$ from 2 mA to $5\frac{1}{3}$mA, and finally decays to $4\frac{1}{2}$mA.

(continued)

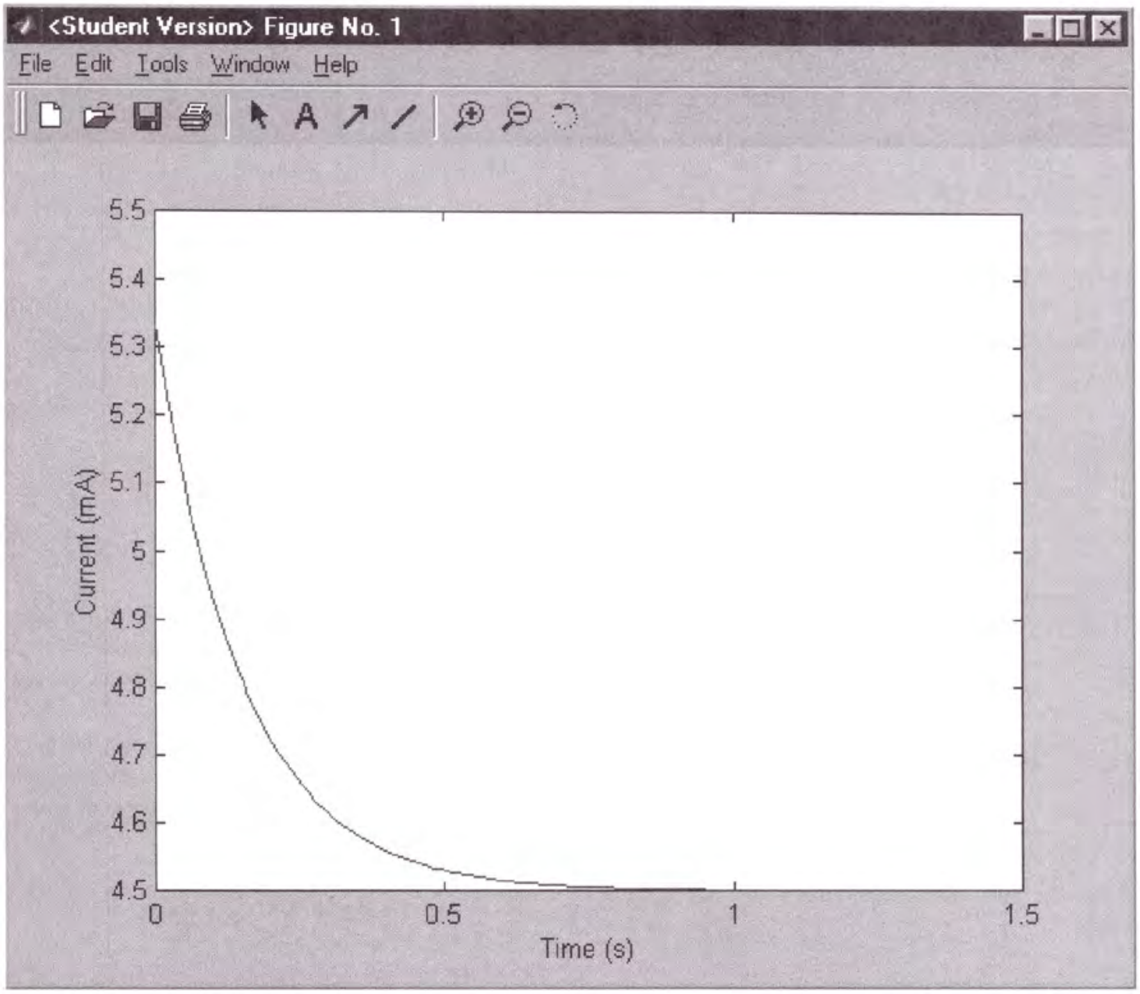


Figure 6.7 *MATLAB plot for Example 6.3.*

LEARNING Example 6.4

The circuit shown in Fig. 6.8a is assumed to have been in a steady-state condition prior to switch closure at $t = 0$. We wish to calculate the voltage $v(t)$ for $t > 0$.

SOLUTION

Step 1. $v(t)$ is of the form $K_1 + K_2 e^{-t/\tau}$.

Step 2. In Fig. 6.8b we see that

$$i_L(0-) = \frac{24}{4 + \dfrac{(6)(3)}{6+3}}\left(\frac{6}{6+3}\right) = \frac{8}{3}\text{ A}$$

Step 3. The new circuit, valid only for $t = 0+$, is shown in Fig. 6.8c, which is equivalent to the circuit shown in Fig. 6.8d. The value of the current source that replaces the inductor is $i_L(0-) = i_L(0+) = \frac{8}{3}$A. The node voltage $v_1(0+)$ can be determined from the circuit in Fig. 6.8d using a single-node equation, and $v(0+)$ is equal to the difference between the source voltage and $v_1(0+)$. The equation for $v_1(0+)$ is

$$\frac{v_1(0+) - 24}{4} + \frac{v_1(0+)}{6} + \frac{8}{3} + \frac{v_1(0+)}{12} = 0$$

or

$$v_1(0+) = \frac{20}{3}\text{ V}$$

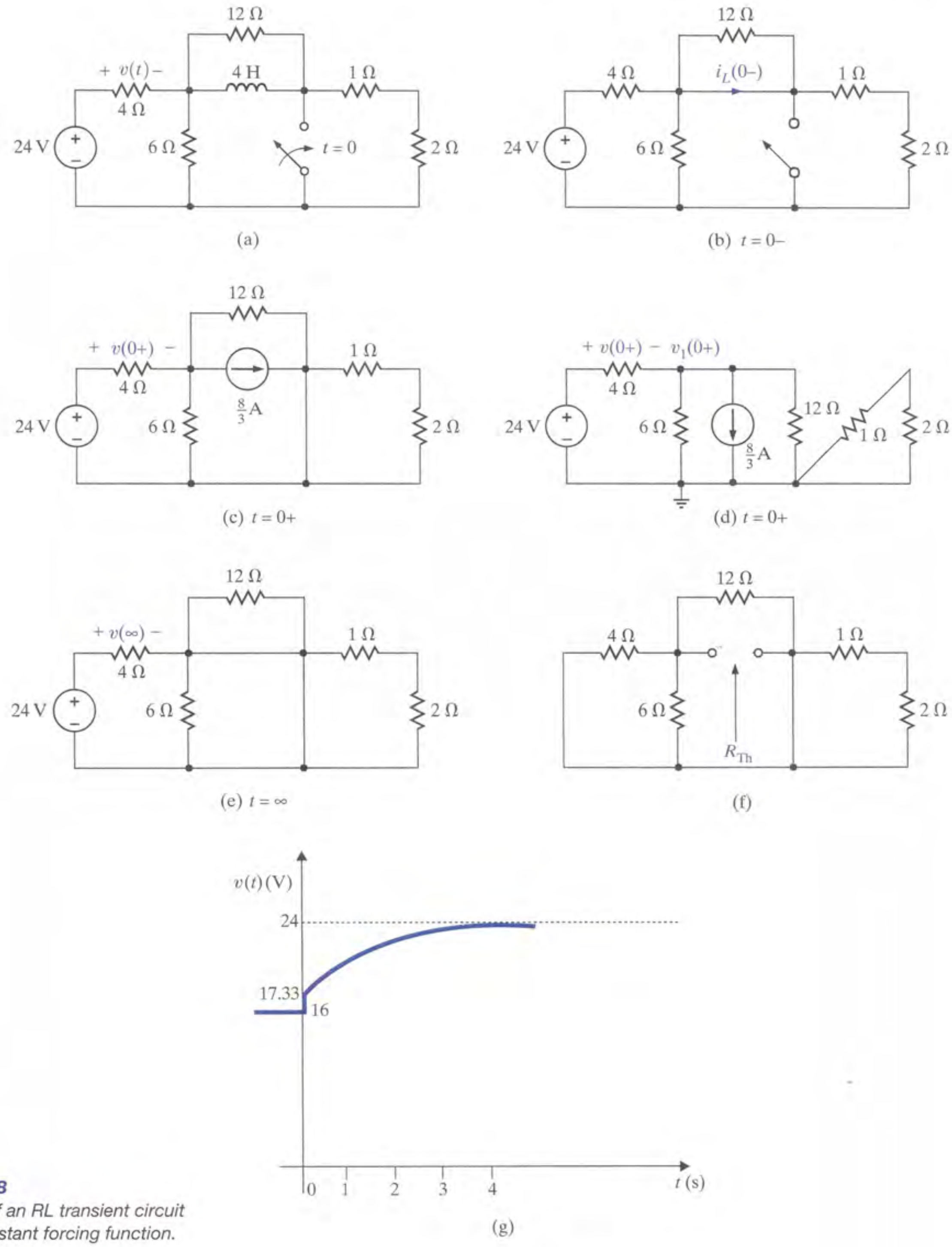


Figure 6.8
Analysis of an RL transient circuit with a constant forcing function.

Then

$$v(0+) = 24 - v_1(0+)$$
$$= \frac{52}{3}\text{ V}$$

Step 4. The equivalent circuit for the steady-state condition after switch closure is given in Fig. 6.8e. Note that the 6-, 12-, 1, and 2-Ω resistors are shorted, and therefore $v(\infty) = 24$ V.

(continued)

Step 5. The Thévenin equivalent resistance is found by looking into the circuit from the inductor terminals. This circuit is shown in Fig. 6.8f. Note carefully that R_{Th} is equal to the 4-, 6-, and 12-Ω resistors in parallel. Therefore, $R_{Th} = 2\ \Omega$, and the circuit time constant is

$$\tau = \frac{L}{R_{Th}} = \frac{4}{2} = 2 \text{ s}$$

Step 6. From the previous analysis we find that

$$K_1 = v(\infty) = 24$$

$$K_2 = v(0+) - v(\infty) = -\frac{20}{3}$$

and hence that

$$v(t) = 24 - \frac{20}{3} e^{-t/2} \text{ V}$$

From Fig. 6.8b we see that the value of $v(t)$ before switch closure is 16 V. This value jumps to 17.33 V at $t = 0$. The MATLAB program for generating the plot (shown in Fig. 6.9) of this function for $t > 0$, is listed next.

```
>>tau = 2;
>>tend = 10*tau;
>>t = linspace(0, tend, 150);
>>v = 24 - (20/3)*exp(-t/tau);
>>plot (t,v)
>>xlabel('Time (s)')
>>ylabel('Voltage (V)')
```

This plot can be compared to the sketch shown in Fig. 6.8g.

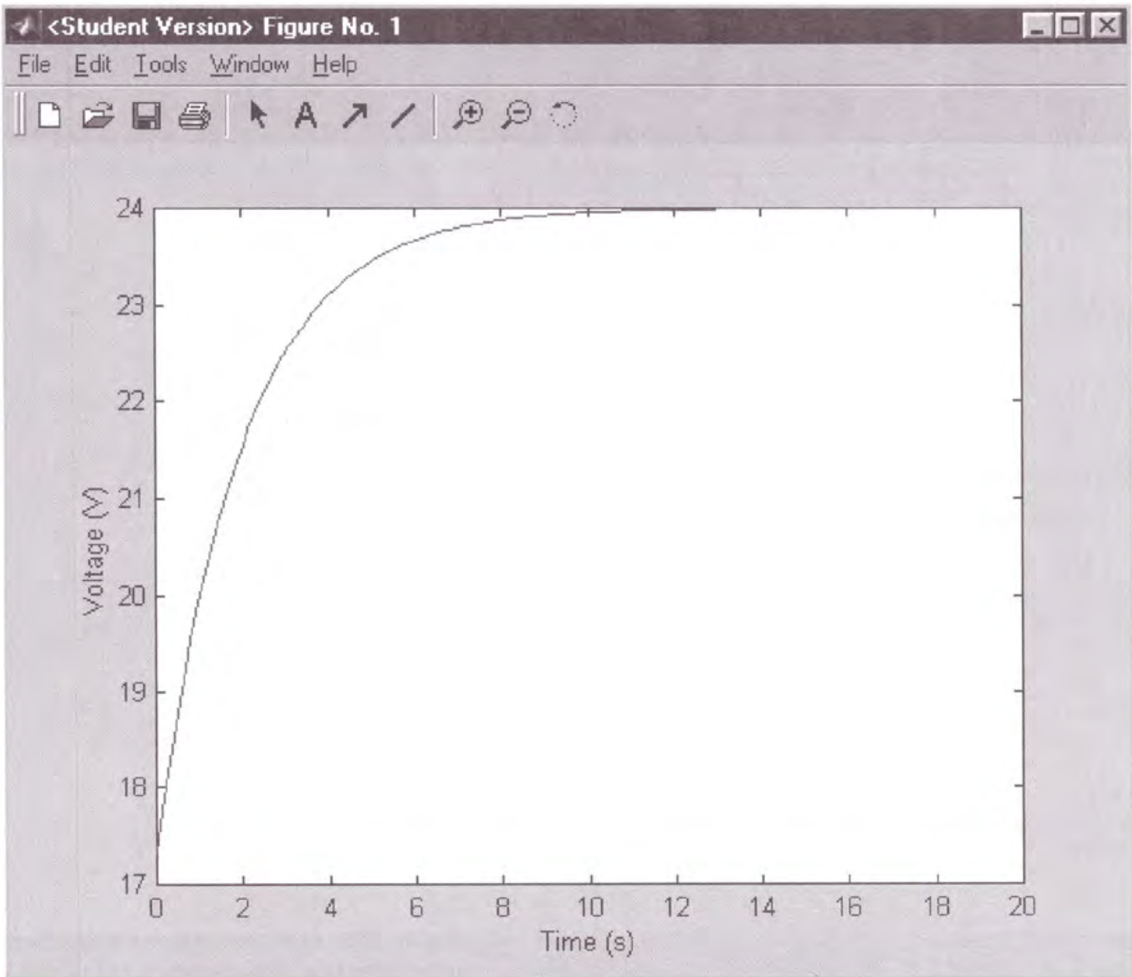


Figure 6.9 *MATLAB plot for Example 6.4.*

LEARNING EXTENSIONS

E6.3 Consider the network in Fig. E6.3. The switch opens at $t = 0$. Find $v_o(t)$ for $t > 0$.

ANSWER

$v_o(t) = \frac{24}{5} + \frac{1}{5}e^{-(5/8)t}$ V.

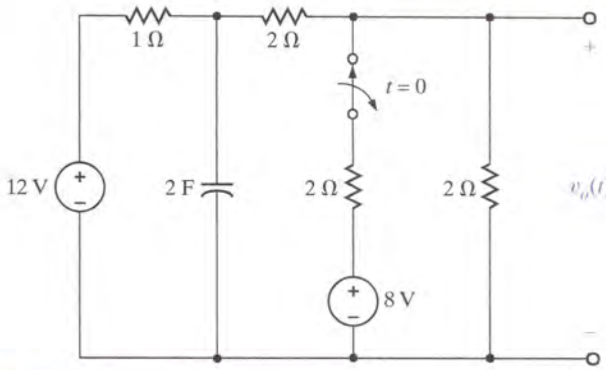


Figure E6.3

E6.4 Consider the network in Fig. E6.4. If the switch opens at $t = 0$, find the output voltage $v_o(t)$ for $t > 0$.

ANSWER

$v_o(t) = 6 - \frac{10}{3}e^{-2t}$ V.

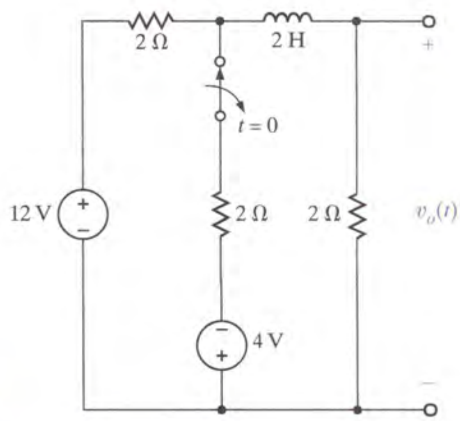


Figure E6.4

LEARNING Example 6.5

The circuit shown in Fig. 6.10a has reached steady state with the switch in position 1. At time $t = 0$ the switch moves from position 1 to position 2. We want to calculate $v_o(t)$ for $t > 0$.

SOLUTION

Step 1. $v_o(t)$ is of the form $K_1 + K_2e^{-t/\tau}$.

Step 2. Using the circuit in Fig. 6.10b, we can calculate $i_L(0-)$

$$i_A = \frac{12}{4} = 3 \text{ A}$$

Then

$$i_L(0-) = \frac{12 + 2i_A}{6} = \frac{18}{6} = 3 \text{ A}$$

Step 3. The new circuit, valid only for $t = 0+$, is shown in Fig. 6.10c. The value of the current source that replaces the inductor is $i_L(0-) = i_L(0+) = 3$ A. Because of the current source

$$v_o(0+) = (3)(6) = 18 \text{ V}$$

Step 4. The equivalent circuit, for the steady-state condition after switch closure, is given in Fig. 6.10d. Using the voltages

(continued)

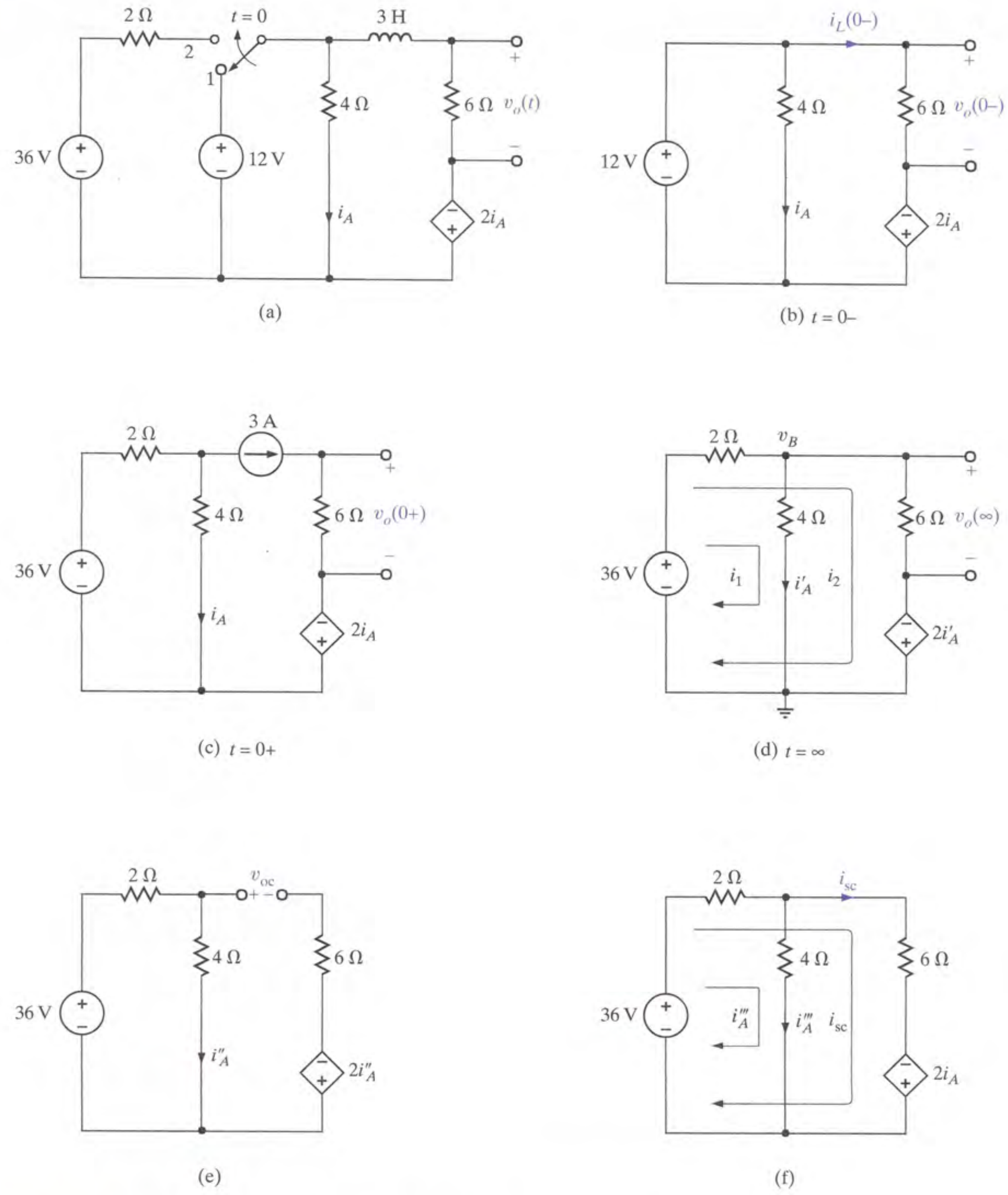


Figure 6.10 *Analysis of an RL transient circuit containing a dependent source.*

and currents defined in the figure, we can compute $v_o(\infty)$ in a variety of ways. For example, using node equations we can find $v_o(\infty)$ from

$$\frac{v_B - 36}{2} + \frac{v_B}{4} + \frac{v_B + 2i'_A}{6} = 0$$

$$i'_A = \frac{v_B}{4}$$

$$v_o(\infty) = v_B + 2i'_A$$

or, using loop equations,

$$36 = 2(i_1 + i_2) + 4i_1$$

$$36 = 2(i_1 + i_2) + 6i_2 - 2i_1$$

$$v_o(\infty) = 6i_2$$

Using either approach, we find that $v_o(\infty) = 27$ V.

Step 5. The Thévenin equivalent resistance can be obtained via v_{oc} and i_{sc} because of the presence of the dependent source. From Fig. 6.10e we note that

$$i''_A = \frac{36}{2 + 4} = 6 \text{ A}$$

Therefore,

$$v_{oc} = (4)(6) + 2(6)$$

$$= 36 \text{ V}$$

From Fig. 6.10f we can write the following loop equations:

$$36 = 2(i'''_A + i_{sc}) + 4i'''_A$$

$$36 = 2(i'''_A + i_{sc}) + 6i_{sc} - 2i'''_A$$

Solving these equations for i_{sc} yields

$$i_{sc} = \frac{9}{2} \text{ A}$$

Therefore,

$$R_{Th} = \frac{v_{oc}}{i_{sc}} = \frac{36}{9/2} = 8\ \Omega$$

Hence, the circuit time constant is

$$\tau = \frac{L}{R_{Th}} = \frac{3}{8} \text{ s}$$

Step 6. Using the information just computed, we can derive the final equation for $v_o(t)$:

$$K_1 = v_o(\infty) = 27$$

$$K_2 = v_o(0+) - v_o(\infty) = 18 - 27 = -9$$

Therefore,

$$v_o(t) = 27 - 9e^{-t/(3/8)} \text{ V}$$

LEARNING EXTENSION

E6.5 If the switch in the network in Fig. E6.5 closes at $t = 0$, find $v_o(t)$ for $t > 0$.

ANSWER
$v_o(t) = 24 + 36e^{-(t/12)}$ V.

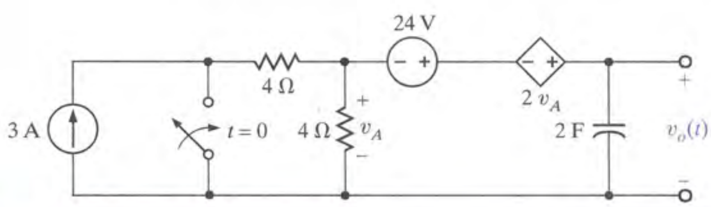


Figure E6.5

At this point it is appropriate to state that not all switch action will always occur at time $t = 0$. It may occur at any time t_0. In this case the results of the step-by-step analysis yield the following equations:

$$x(t_0) = K_1 + K_2$$

$$x(\infty) = K_1$$

and

$$x(t) = x(\infty) + \left[x(t_0) - x(\infty)\right]e^{-(t-t_0)/\tau} \qquad t > t_0$$

The function is essentially time-shifted by t_0 seconds.

Finally, note that if more than one independent source is present in the network, we can simply employ superposition to obtain the total response.

PULSE RESPONSE Thus far we have examined networks in which a voltage or current source is suddenly applied. As a result of this sudden application of a source, voltages or currents in the circuit are forced to change abruptly. A forcing function whose value changes in a discontinuous manner or has a discontinuous derivative is called a *singular function*. Two such singular functions that are very important in circuit analysis are the unit impulse function and the unit step function. We will defer a discussion of the former until a later chapter and concentrate on the latter.

The *unit step function* is defined by the following mathematical relationship:

$$u(t) = \begin{cases} 0 & t < 0 \\ 1 & t > 0 \end{cases}$$

In other words, this function, which is dimensionless, is equal to zero for negative values of the argument and equal to 1 for positive values of the argument. It is undefined for a zero argument where the function is discontinuous. A graph of the unit step is shown in Fig. 6.11a. The unit step is dimensionless, and therefore a voltage step of V_o volts or a current step of I_o amperes is written as $V_o u(t)$ and $I_o u(t)$, respectively. Equivalent circuits for a voltage step are shown in Figs. 6.11b and c. Equivalent circuits for a current step are shown in Figs. 6.11d and e. If we use the definition of the unit step, it is easy to generalize this function by replacing the argument t by $t - t_0$. In this case

$$u(t - t_0) = \begin{cases} 0 & t < t_0 \\ 1 & t > t_0 \end{cases}$$

A graph of this function is shown in Fig. 6.11f. Note that $u(t - t_0)$ is equivalent to delaying $u(t)$ by t_0 seconds, so that the abrupt change occurs at time $t = t_0$.

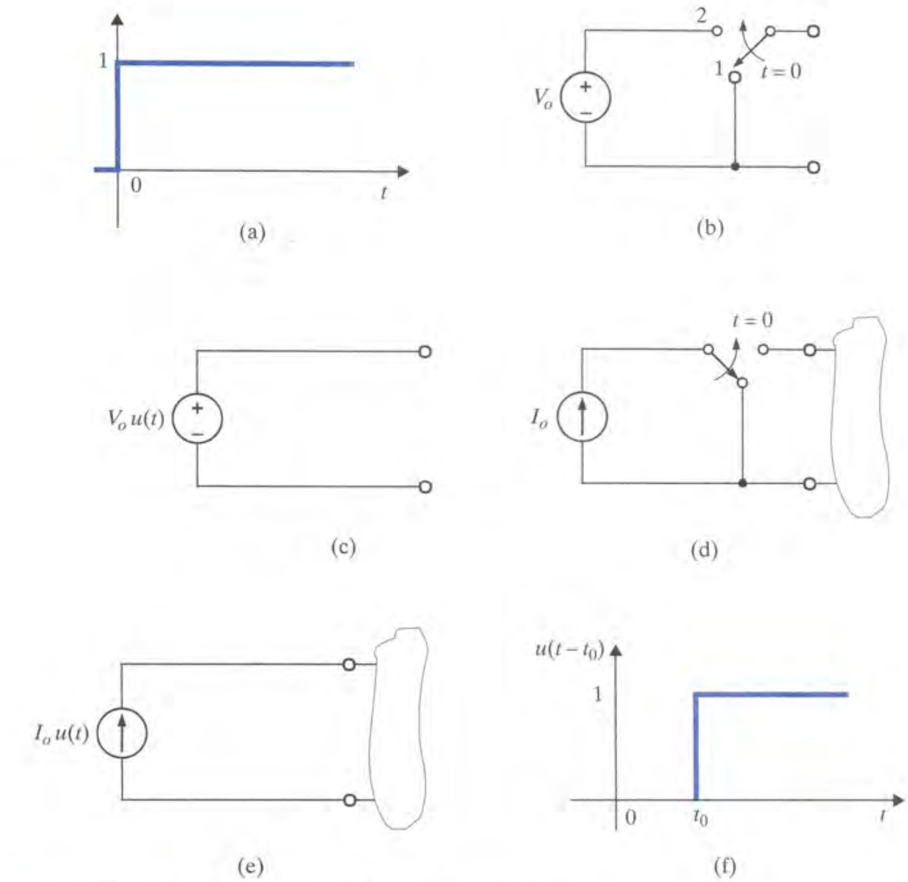


Figure 6.11 *Graphs and models of the unit step function.*

Step functions can be used to construct one or more pulses. For example, the voltage pulse shown in Fig. 6.12a can be formulated by initiating a unit step at $t = 0$ and subtracting one that starts at $t = T$, as shown in Fig. 6.12b. The equation for the pulse is

$$v(t) = A[u(t) - u(t - T)]$$

If the pulse is to start at $t = t_0$ and have width T, the equation would be

$$v(t) = A\{u(t - t_0) - u[t - (t_0 + T)]\}$$

Using this approach, we can write the equation for a pulse starting at any time and ending at any time. Similarly, using this approach, we could write the equation for a series of pulses, called a *pulse train*, by simply forming a summation of pulses constructed in the manner illustrated previously.

The following example will serve to illustrate many of the concepts we have just presented.

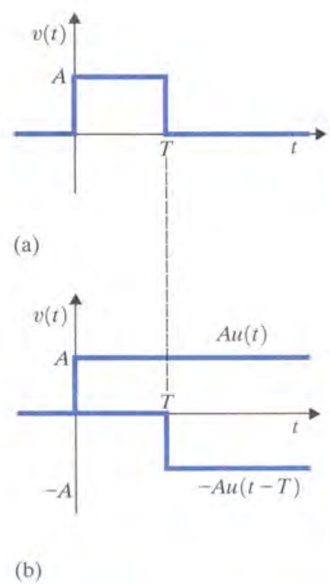


Figure 6.12
Construction of a pulse via two step functions.

LEARNING Example 6.6

Consider the circuit shown in Fig. 6.13a. The input function is the voltage pulse shown in Fig. 6.13b. Since the source is zero for all negative time, the initial conditions for the network are zero $[\text{i.e., } v_C(0-) = 0]$. The response $v_o(t)$ for $0 < t < 0.3$ s is due to the application of the constant source at $t = 0$ and is not influenced by any source changes that will occur later. At $t = 0.3$ *s* the forcing function becomes zero and therefore $v_o(t)$ for $t > 0.3$ s is the source-free or natural response of the network.

Let us determine the expression for the voltage $v_o(t)$.

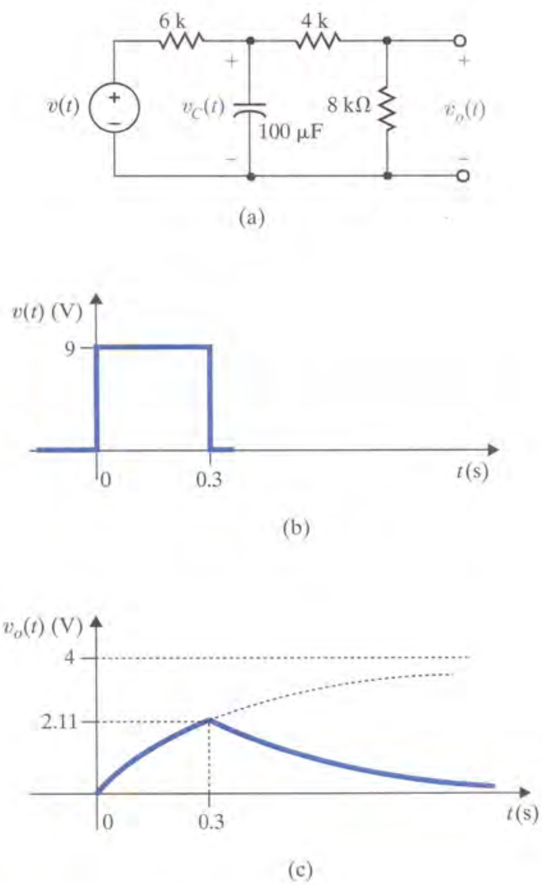


Figure 6.13 *Pulse response of a network.*

SOLUTION Since the output voltage $v_o(t)$ is a voltage division of the capacitor voltage, and the initial voltage across the capacitor is zero, we know that $v_o(0+) = 0$.

If no changes were made in the source after $t = 0$, the steady-state value of $v_o(t)$ $[\text{i.e., } v_o(\infty)]$ due to the application of the unit step at $t = 0$ would be

$$\begin{aligned} v_o(\infty) &= \frac{9}{6\text{k} + 4\text{k} + 8\text{k}}(8\text{k}) \\ &= 4 \text{ V} \end{aligned}$$

The Thévenin equivalent resistance is

$$\begin{aligned} R_{\text{Th}} &= \frac{(6\text{k})(12\text{k})}{6\text{k} + 12\text{k}} \\ &= 4 \text{ k}\Omega \end{aligned}$$

Therefore, the circuit time constant τ is

$$\begin{aligned} \tau &= R_{\text{Th}}C \\ &= (4)(10^3)(100)(10^{-6}) \\ &= 0.4 \text{ s} \end{aligned}$$

Therefore, the response $v_o(t)$ for the period $0 < t < 0.3$ s is

$$v_o(t) = 4 - 4e^{-t/0.4} \text{ V} \quad 0 < t < 0.3 \text{ s}$$

The capacitor voltage can be calculated by realizing that using voltage division, $v_o(t) = \frac{2}{3}v_C(t)$. Therefore,

$$v_C(t) = \tfrac{3}{2}\left(4 - 4e^{-t/0.4}\right) \text{ V}$$

Since the capacitor voltage is continuous,

$$v_C(0.3-) = v_C(0.3+)$$

and therefore,

$$\begin{aligned} v_o(0.3+) &= \tfrac{2}{3}v_C(0.3-) \\ &= 4\left(1 - e^{-0.3/0.4}\right) \\ &= 2.11 \text{ V} \end{aligned}$$

Since the source is zero for $t > 0.3$ s, the final value for $v_o(t)$ as $t \to \infty$ is zero. Therefore, the expression for $v_o(t)$ for $t > 0.3$ s is

$$v_o(t) = 2.11e^{-(t-0.3)/0.4} \text{ V} \quad t > 0.3 \text{ s}$$

The term $e^{-(t-0.3)/0.4}$ indicates that the exponential decay starts at $t = 0.3$ s. The complete solution can be written by means of superposition as

$$v_o(t) = 4\left(1 - e^{-t/0.4}\right)u(t) - 4\left(1 - e^{-(t-0.3)/0.4}\right)u(t - 0.3) \text{ V}$$

LEARNING by Doing

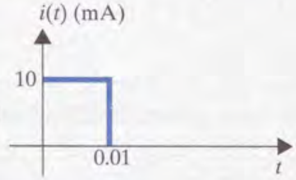
D 6.3 Plot the following functions:

(a) $i(t) = 10[u(t) - u(t - 0.01)]$ mA

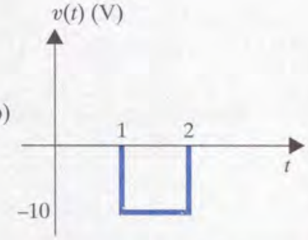
(b) $v(t) = -10[u(t - 1) - u(t - 2)]$ V

ANSWER

(a)



(b)



or, equivalently, the complete solution is

$$v_o(t) = \begin{cases} 0 & t < 0 \\ 4(1 - e^{-t/0.4})\text{ V} & 0 < t < 0.3\text{ s} \\ 2.11e^{-(t-0.3)/0.4}\text{ V} & 0.3\text{ s} < t \end{cases}$$

which in mathematical form is

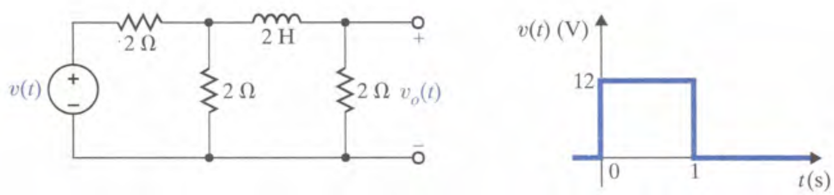
$$v_o(t) = 4(1 - e^{-t/0.4})[u(t) - u(t - 0.3)] + 2.11e^{-(t-0.3)/0.4}u(t - 0.3)\text{ V}$$

Note that the term $[u(t) - u(t - 0.3)]$ acts like a gating function that captures only the part of the step response that exists in the time interval $0 < t < 0.3$ s. The output as a function of time is shown in Fig. 6.13c.

LEARNING EXTENSION

E6.6 The voltage source in the network in Fig. E6.6a is shown in Fig. E6.6b. The initial current in the inductor must be zero. (Why?) Determine the output voltage $v_o(t)$ for $t > 0$.

ANSWER $v_o(t) = 0$ for $t < 0$,$4(1 - e^{-(3/2)t})$ V for $0 \leqq t \leqq 1$, and $3.11e^{-(3/2)(t-1)}$ V for $1 < t$.



(a) (b)

Figure E6.6

6.3 Second-Order Circuits

THE BASIC CIRCUIT EQUATION To begin our development, let us consider the two basic *RLC* circuits shown in Fig. 6.14. We assume that energy may be initially stored in both the inductor and capacitor. The node equation for the parallel *RLC* circuit is

$$\frac{v}{R} + \frac{1}{L}\int_{t_0}^{t} v(x)\,dx + i_L(t_0) + C\frac{dv}{dt} = i_S(t)$$

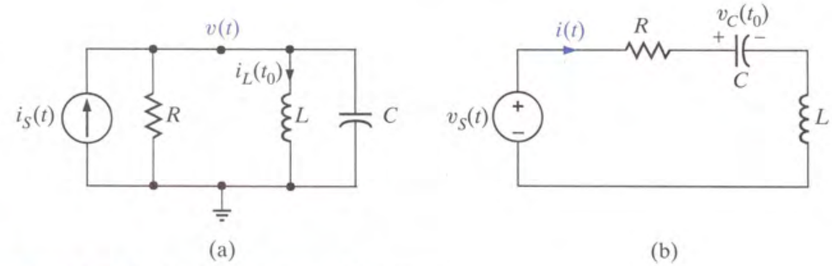


Figure 6.14 *Parallel and series RLC circuits.*

Similarly, the loop equation for the series *RLC* circuit is

$$Ri + \frac{1}{C}\int_{t_0}^{t} i(x)\,dx + v_C(t_0) + L\frac{di}{dt} = v_S(t)$$

Note that the equation for the node voltage in the parallel circuit is of the same form as that for the loop current in the series circuit. Therefore, the solution of these two circuits is dependent on solving one equation. If the two preceding equations are differentiated with respect to time, we obtain

$$C\frac{d^2v}{dt^2} + \frac{1}{R}\frac{dv}{dt} + \frac{v}{L} = \frac{di_S}{dt}$$

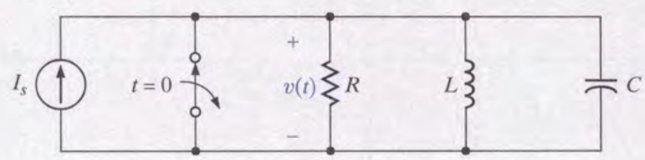
and

$$L\frac{d^2i}{dt^2} + R\frac{di}{dt} + \frac{i}{C} = \frac{dv_S}{dt}$$

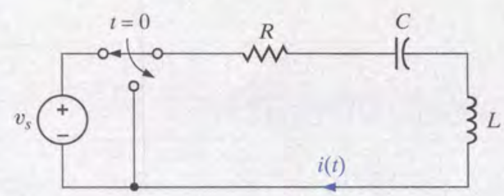
Since both circuits lead to a second-order differential equation with constant coefficients, we will concentrate our analysis on this type of equation.

LEARNING by Doing

D 6.4 Write the differential equation that describes the node voltage in (a) and the loop current in (b).



(a)



(b)

ANSWER (a) $\frac{d^2v}{dt^2} + \frac{1}{RC}\frac{dv}{dt} + \frac{v}{LC} = 0$ (b) $\frac{d^2i}{dt^2} + \frac{R}{L}\frac{di}{dt} + \frac{i}{LC} = 0$

THE RESPONSE EQUATIONS In concert with our development of the solution of a first-order differential equation that results from the analysis of either an *RL* or an *RC* circuit as outlined earlier, we will now employ the same approach here to obtain the solution of a second-order differential equation that results from the analysis of *RLC* circuits. As a general rule, for this case we are confronted with an equation of the form

$$\frac{d^2x(t)}{dt^2} + a_1\frac{dx(t)}{dt} + a_2x(t) = f(t) \qquad 6.12$$

Once again we use the fact that if $x(t) = x_p(t)$ is a solution to Eq. (6.12), and if $x(t) = x_c(t)$ is a solution to the homogeneous equation

$$\frac{d^2x(t)}{dt^2} + a_1\frac{dx(t)}{dt} + a_2x(t) = 0$$

then

$$x(t) = x_p(t) + x_c(t)$$

is a solution to the original Eq. (6.12). If we again confine ourselves to a constant forcing function $[\text{i.e., } f(t) = A]$, the development at the beginning of this chapter shows that the solution of Eq. (6.12) will be of the form

$$x(t) = \frac{A}{a_2} + x_c(t) \qquad 6.13$$

Let us now turn our attention to the solution of the homogeneous equation

$$\frac{d^2x(t)}{dt^2} + a_1\frac{dx(t)}{dt} + a_2x(t) = 0$$

where a_1 and a_2 are constants. For simplicity we will rewrite the equation in the form

$$\frac{d^2x(t)}{dt^2} + 2\zeta\omega_0\frac{dx(t)}{dt} + \omega_0^2x(t) = 0 \qquad 6.14$$

where we have made the following simple substitutions for the constants $a_1 = 2\zeta\omega_0$ and $a_2 = \omega_0^2$.

Following the development of a solution for the first-order homogeneous differential equation earlier in this chapter, the solution of Eq. (6.14) must be a function whose first- and second-order derivatives have the same form, so that the left-hand side of Eq. (6.14) will become identically zero for all t. Again we assume that

$$x(t) = Ke^{st}$$

Substituting this expression into Eq. (6.14) yields

$$s^2Ke^{st} + 2\zeta\omega_0 sKe^{st} + \omega_0^2Ke^{st} = 0$$

LEARNING by Doing

D 6.5 Given the homogeneous differential equation

$$4\frac{d^2x(t)}{dt^2} + 8\frac{dx(t)}{dt} + 16x(t) = 0$$

determine the characteristic equation, the damping ratio, and the undamped natural frequency.

ANSWER *Characteristic equation:* $s^2 + 2s + 4 = 0$

$\zeta = \frac{1}{2}$

$\omega_0 = 2$

Dividing both sides of the equation by Ke^{st} yields

$$s^2 + 2\zeta\omega_0 s + \omega_0^2 = 0 \tag{6.15}$$

This equation is commonly called the *characteristic equation*; ζ is called the exponential *damping ratio*, and ω_0 is referred to as the *undamped natural frequency*. The importance of this terminology will become clear as we proceed with the development. If this equation is satisfied, our assumed solution $x(t) = Ke^{st}$ is correct. Employing the quadratic formula, we find that Eq. (6.15) is satisfied if

$$\begin{aligned} s &= \frac{-2\zeta\omega_0 \pm \sqrt{4\zeta^2\omega_0^2 - 4\omega_0^2}}{2} \\ &= -\zeta\omega_0 \pm \omega_0\sqrt{\zeta^2 - 1} \end{aligned} \tag{6.16}$$

Therefore, there are two values of s, s_1 and s_2 that satisfy Eq. (6.15):

$$\begin{aligned} s_1 &= -\zeta\omega_0 + \omega_0\sqrt{\zeta^2 - 1} \\ s_2 &= -\zeta\omega_0 - \omega_0\sqrt{\zeta^2 - 1} \end{aligned} \tag{6.17}$$

Therefore, in general, the complementary solution of Eq. (6.14) is of the form

$$x_c(t) = K_1 e^{s_1 t} + K_2 e^{s_2 t} \tag{6.18}$$

K_1 and K_2 are constants that can be evaluated via the initial conditions $x(0)$ and $dx(0)/dt$. For example, since

$$x(t) = K_1 e^{s_1 t} + K_2 e^{s_2 t}$$

then

$$x(0) = K_1 + K_2$$

and

$$\left.\frac{dx(t)}{dt}\right|_{t=0} = \frac{dx(0)}{dt} = s_1 K_1 + s_2 K_2$$

Hence, $x(0)$ and $dx(0)/dt$ produce two simultaneous equations, which when solved yield the constants K_1 and K_2.

Close examination of Eqs. (6.17) and (6.18) indicates that the form of the solution of the homogeneous equation is dependent on the value ζ. For example, if $\zeta > 1$, the roots of the characteristic equation, s_1 and s_2, also called the *natural frequencies* because they determine the natural (unforced) response of the network, are real and unequal; if $\zeta < 1$, the roots are complex numbers; and finally, if $\zeta = 1$, the roots are real and equal.

Let us now consider the three distinct forms of the unforced response—that is, the response due to an initial capacitor voltage or initial inductor current.

Case 1, $\zeta > 1$ This case is commonly called *overdamped*. The natural frequencies s_1 and s_2 are real and unequal, and therefore the natural response of the network described by the second-order differential equation is of the form

$$x_c(t) = K_1 e^{-(\zeta\omega_0 - \omega_0\sqrt{\zeta^2-1})t} + K_2 e^{-(\zeta\omega_0 + \omega_0\sqrt{\zeta^2-1})t} \qquad \textbf{6.19}$$

where K_1 and K_2 are found from the initial conditions. This indicates that the natural response is the sum of two decaying exponentials.

LEARNING by Doing

D 6.6 Determine the general form of the solution of the equation in Learning by Doing 6.5.

ANSWER $x_c(t) = e^{-t}(A_1 \cos\sqrt{3}t + A_2 \sin\sqrt{3}t)$.

Case 2, $\zeta < 1$ This case is called *underdamped*. Since $\zeta < 1$, the roots of the characteristic equation given in Eq. (6.17) can be written as

$$s_1 = -\zeta\omega_0 + j\omega_0\sqrt{1-\zeta^2} = -\sigma + j\omega_d$$
$$s_2 = -\zeta\omega_0 - j\omega_0\sqrt{1-\zeta^2} = -\sigma - j\omega_d$$

where $j = \sqrt{-1}$, $\sigma = \zeta\omega_0$ and $\omega_d = \omega_0\sqrt{1-\zeta^2}$. Thus, the natural frequencies are complex numbers (briefly discussed in the Appendix). The natural response is then of the form

$$x_c(t) = e^{-\zeta\omega_0 t}(A_1 \cos\omega_0\sqrt{1-\zeta^2}\,t + A_2 \sin\omega_0\sqrt{1-\zeta^2}\,t) \qquad \textbf{6.20}$$

where A_1 and A_2, like K_1 and K_2, are constants, which are evaluated using the initial conditions $x(0)$ and $dx(0)/dt$. This illustrates that the natural response is an exponentially damped oscillatory response.

Case 3, $\zeta = 1$ This case, called *critically damped*, results in

$$s_1 = s_2 = -\zeta\omega_0$$

In the case where the characteristic equation has repeated roots, the general solution is of the form

$$x_c(t) = B_1 e^{-\zeta\omega_0 t} + B_2 t e^{-\zeta\omega_0 t} \qquad \textbf{6.21}$$

where B_1 and B_2 are constants derived from the initial conditions.

LEARNING by Doing

D 6.7 Determine the general form of the solution of the equation

$$\frac{d^2x(t)}{dt^2} + \frac{4\,dx(t)}{dt} + 4x(t) = 0$$

ANSWER
$x_c(t) = B_1 e^{-2t} + B_2 t e^{-2t}$

It is informative to sketch the natural response for the three cases we have discussed: overdamped, Eq. (6.19); underdamped, Eq. (6.20); and critically damped, Eq. (6.21). Figure 6.15 graphically illustrates the three cases for the situations in which $x_c(0) = 0$. Note that the critically damped response peaks and decays faster than the overdamped response. The underdamped response is an exponentially damped sinusoid whose rate of decay is dependent on the factor ζ. Actually, the terms $\pm e^{-\zeta\omega_0 t}$ define what is called the *envelope* of the response, and the damped oscillations (i.e., the oscillations of decreasing amplitude) exhibited by the waveform in Fig. 6.15b are called *ringing*.

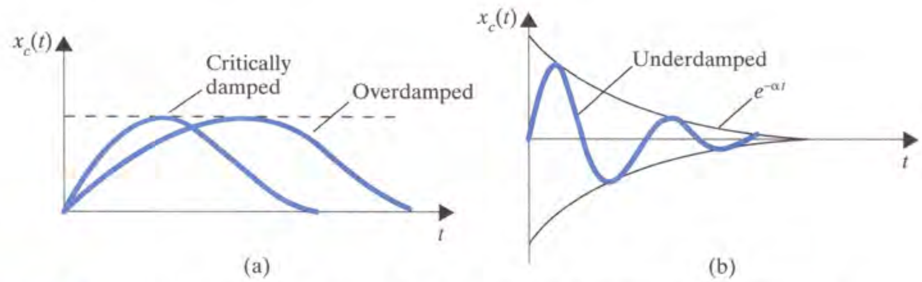


Figure 6.15 *Comparison of overdamped, critically damped, and underdamped responses.*

LEARNING EXTENSIONS

E6.7 A parallel *RLC* circuit has the following circuit parameters: $R = 1\ \Omega$, $L = 2$ H, and $C = 2$ F. Compute the damping ratio and the undamped natural frequency of this network.

ANSWER $\zeta = 0.5$, $\omega_0 = 0.5$ rad/s.

E6.8 A series *RLC* circuit consists of $R = 2\ \Omega$, $L = 1$ H, and a capacitor. Determine the type of response exhibited by the network if (a) $C = \frac{1}{2}$ F, (b) $C = 1$ F, and (c) $C = 2$ F.

ANSWER (a) underdamped, (b) critically damped, (c) overdamped.

THE NETWORK RESPONSE We will now analyze a number of simple *RLC* networks that contain both nonzero initial conditions and constant forcing functions. Circuits that exhibit overdamped, underdamped, and critically damped responses will be considered.

Problem-Solving Strategy
Second-Order Transient Circuits

- Write the differential equation that describes the circuit.
- Derive the characteristic equation, which can be written in the form $s^2 + 2\zeta\omega_0 s + \omega_0^2 = 0$, where ζ is the damping ratio and ω_0 is the undamped natural frequency.

- The two roots of the characteristic equation will determine the type of response. If the roots are real and unequal (i.e., $\zeta > 1$), the network response is overdamped. If the roots are real and equal (i.e., $\zeta = 1$), the network response is critically damped. If the roots are complex (i.e., $\zeta < 1$), the network response is underdamped.
- The damping condition and corresponding response for the aforementioned three cases outlined are as follows:
  Overdamped: $x(t) = K_1 e^{-(\zeta\omega_0 - \omega_0\sqrt{\zeta^2-1})t} + K_2 e^{-(\zeta\omega_0 + \omega_0\sqrt{\zeta^2-1})t}$
  Critically damped: $x(t) = B_1 e^{-\zeta\omega_0 t} + B_2 t e^{-\zeta\omega_0 t}$
  Underdamped: $x(t) = e^{-\sigma t}(A_1 \cos\omega_d t + A_2 \sin\omega_d t)$, where $\sigma = \zeta\omega_0$, and $\omega_d = \omega_0\sqrt{1-\zeta^2}$
- Two initial conditions, either given or derived, are required to obtain the two unknown coefficients in the response equation.

The following examples will serve to demonstrate the analysis techniques.

LEARNING Example 6.7

Consider the parallel RLC circuit shown in Fig. 6.16. The second-order differential equation that describes the voltage $v(t)$ is

$$\frac{d^2v}{dt^2} + \frac{1}{RC}\frac{dv}{dt} + \frac{v}{LC} = 0$$

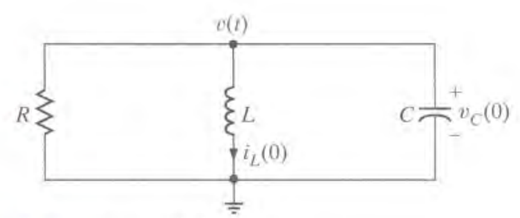


Figure 6.16 Parallel RLC circuit.

A comparison of this equation with Eqs. (6.14) and (6.15) indicates that for the parallel RLC circuit the damping term is $1/2\,RC$ and the undamped natural frequency is $1/\sqrt{LC}$. If the circuit parameters are $R = 2\ \Omega$, $C = \frac{1}{5}$ F, and $L = 5$ H, the equation becomes

$$\frac{d^2v}{dt^2} + 2.5\frac{dv}{dt} + v = 0$$

Let us assume that the initial conditions on the storage elements are $i_L(0) = -1$ A and $v_C(0) = 4$ V. Let us find the node voltage $v(t)$ and the inductor current.

SOLUTION The characteristic equation for the network is

$$s^2 + 2.5s + 1 = 0$$

and the roots are

$$s_1 = -2$$
$$s_2 = -0.5$$

Since the roots are real and unequal, the circuit is overdamped, and $v(t)$ is of the form

$$v(t) = K_1 e^{-2t} + K_2 e^{-0.5t}$$

The initial conditions are now employed to determine the constants K_1 and K_2. Since $v(t) = v_C(t)$,

$$v_C(0) = v(0) = 4 = K_1 + K_2$$

The second equation needed to determine K_1 and K_2 is normally obtained from the expression

$$\frac{dv(t)}{dt} = -2K_1 e^{-2t} - 0.5K_2 e^{-0.5t}$$

However, the second initial condition is not $dv(0)/dt$. If this were the case, we would simply evaluate the equation at $t = 0$. This would produce a second equation in the unknowns K_1 and K_2. We can, however, circumvent this problem by noting that the node equation for the circuit can be written as

$$C\frac{dv(t)}{dt} + \frac{v(t)}{R} + i_L(t) = 0$$

(continued)

or

$$\frac{dv(t)}{dt} = \frac{-1}{RC}v(t) - \frac{i_L(t)}{C}$$

At $t = 0$,

$$\begin{aligned}\frac{dv(0)}{dt} &= \frac{-1}{RC}v(0) - \frac{1}{C}i_L(0)\\ &= -2.5(4) - 5(-1)\\ &= -5\end{aligned}$$

However, since

$$\frac{dv(t)}{dt} = -2K_1e^{-2t} - 0.5K_2e^{-0.5t}$$

then when $t = 0$

$$-5 = -2K_1 - 0.5K_2$$

This equation, together with the equation

$$4 = K_1 + K_2$$

produces the constants $K_1 = 2$ and $K_2 = 2$. Therefore, the final equation for the voltage is

$$v(t) = 2e^{-2t} + 2e^{-0.5t} \text{ V}$$

Note that the voltage equation satisfies the initial condition $v(0) = 4$ V. The response curve for this voltage $v(t)$ is shown in Fig. 6.17.

The inductor current is related to $v(t)$ by the equation

$$i_L(t) = \frac{1}{L}\int v(t)\,dt$$

Substituting our expression for $v(t)$ yields

$$i_L(t) = \frac{1}{5}\int\left[2e^{-2t} + 2e^{-0.5t}\right]dt$$

or

$$i_L(t) = -\frac{1}{5}e^{-2t} - \frac{4}{5}e^{-0.5t} \text{ A}$$

Note that in comparison with the *RL* and *RC* circuits, the response of this *RLC* circuit is controlled by two time constants. The first term has a time constant of $\frac{1}{2}$, and the second term has a time constant of 2.

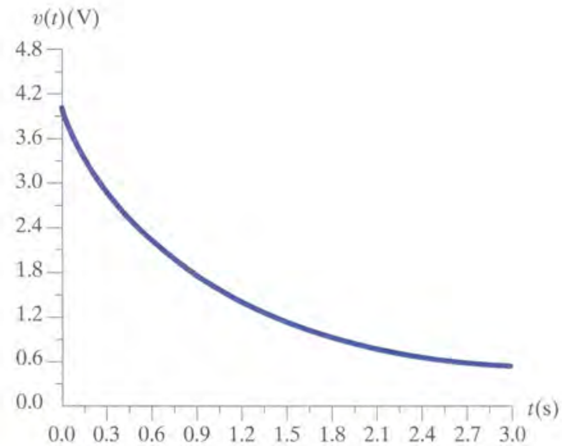


Figure 6.17 *Overdamped response.*

LEARNING Example 6.8

The series *RLC* circuit shown in Fig. 6.18 has the following parameters: $C = 0.04$ F, $L = 1$ H, $R = 6\ \Omega$, $i_L(0) = 4$ A, and $v_C(0) = -4$ V. The equation for the current in the circuit is given by the expression

$$\frac{d^2i}{dt^2} + \frac{R}{L}\frac{di}{dt} + \frac{i}{LC} = 0$$

A comparison of this equation with Eqs. (6.14) and (6.15) illustrates that for a series *RLC* circuit the damping term is $R/2L$ and the undamped natural frequency is $1/\sqrt{LC}$. Substituting the circuit element values into the preceding equation yields

$$\frac{d^2i}{dt^2} + 6\frac{di}{dt} + 25i = 0$$

Let us determine the expression for both the current and the capacitor voltage.

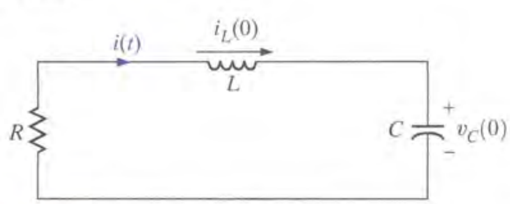


Figure 6.18 *Series RLC circuit.*

SOLUTION The characteristic equation is then

$$s_2 + 6s + 25 = 0$$

and the roots are

$$s_1 = -3 + j4$$
$$s_2 = -3 - j4$$

Since the roots are complex, the circuit is underdamped, and the expression for $i(t)$ is

$$i(t) = K_1 e^{-3t} \cos 4t + K_2 e^{-3t} \sin 4t$$

Using the initial conditions, we find that

$$i(0) = 4 = K_1$$

and

$$\frac{di}{dt} = -4K_1 e^{-3t} \sin 4t - 3K_1 e^{-3t} \cos 4t + 4K_2 e^{-3t} \cos 4t - 3K_2 e^{-3t} \sin 4t$$

and thus

$$\frac{di(0)}{dt} = -3K_1 + 4K_2$$

Although we do not know $di(0)/dt$, we can find it via KVL. From the circuit we note that

$$Ri(0) + L\frac{di(0)}{dt} + v_C(0) = 0$$

or

$$\frac{di(0)}{dt} = -\frac{R}{L}i(0) - \frac{v_C(0)}{L}$$
$$= -\frac{6}{1}(4) + \frac{4}{1}$$
$$= -20$$

Therefore,

$$-3K_1 + 4K_2 = -20$$

and since $K_1 = 4$, $K_2 = -2$, the expression then for $i(t)$ is

$$i(t) = 4e^{-3t} \cos 4t - 2e^{-3t} \sin 4t \text{ A}$$

Note that this expression satisfies the initial condition $i(0) = 4$. The voltage across the capacitor could be determined via KVL using this current:

$$Ri(t) + L\frac{di(t)}{dt} + v_C(t) = 0$$

or

$$v_C(t) = -Ri(t) - L\frac{di(t)}{dt}$$

Substituting the preceding expression for $i(t)$ into this equation yields

$$v_C(t) = -4e^{-3t} \cos 4t + 22e^{-3t} \sin 4t \text{ V}$$

Note that this expression satisfies the initial condition $v_C(0) = -4$ V.

The MATLAB program for plotting this function in the time interval $t > 0$ is listed as follows:

```
>>tau = 1/3;
>>tend = 10*tau;
>>t = linspace(0, tend, 150);
>>v1 = -4*exp(-3*t).*cos(4*t);
>>v2 = 22*exp(-3*t).*sin(4*t);
>>v = v1+v2;
>>plot (t,v)
>>xlabel('Time (s)')
>>ylabel('Voltage (V)')
```

Note that the functions `exp(-3*t)` and `cos (4*t)` produce arrays the same size as `t`. Hence, when we multiply these functions together, we need to specify that we want the arrays to be multiplied element by element. Element-by-element multiplication of an array is denoted in MATLAB by the `.*` notation. Multiplying an array by a scalar `(-4*exp(-3*t)`, for example, can be performed by using the `*` by itself.

The plot generated by this program is shown in Fig. 6.19.

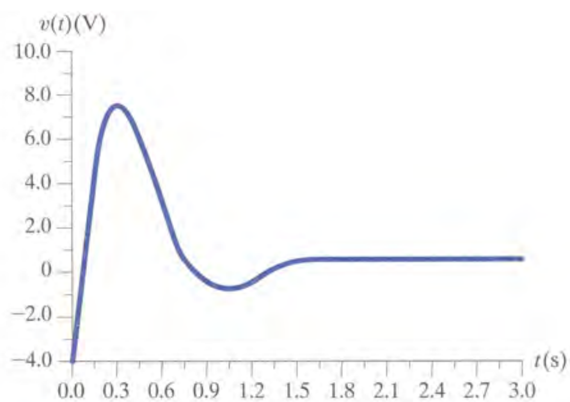


Figure 6.19 *Underdamped response.*

LEARNING Example 6.9

Let us examine the circuit in Fig. 6.20, which is slightly more complicated than the two we have considered earlier. The two equations that describe the network are

$$L\frac{di(t)}{dt} + R_1 i(t) + v(t) = 0$$

$$i(t) = C\frac{dv(t)}{dt} + \frac{v(t)}{R_2}$$

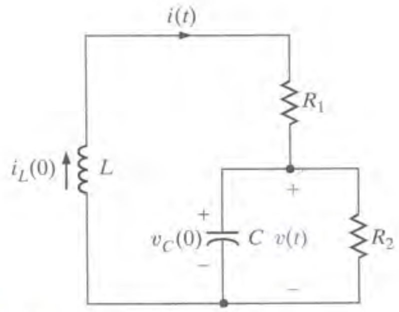


Figure 6.20 Series-parallel RLC circuit.

Substituting the second equation into the first yields

$$\frac{d^2v}{dt^2} + \left(\frac{1}{R_2C} + \frac{R_1}{L}\right)\frac{dv}{dt} + \frac{R_1 + R_2}{R_2LC}v = 0$$

If the circuit parameters and initial conditions are

$$R_1 = 10\ \Omega \qquad C = \tfrac{1}{8}\text{F} \qquad v_C(0) = 1\text{ V}$$
$$R_2 = 8\ \Omega \qquad L = 2\text{ H} \qquad i_L(0) = \tfrac{1}{2}\text{A}$$

the differential equation becomes

$$\frac{d^2v}{dt^2} + 6\frac{dv}{dt} + 9v = 0$$

We wish to find expressions for the current $i(t)$ and the voltage $v(t)$.

SOLUTION The characteristic equation is then

$$s^2 + 6s + 9 = 0$$

and hence the roots are

$$s_1 = -3$$
$$s_2 = -3$$

Since the roots are real and equal, the circuit is critically damped. The term $v(t)$ is then given by the expression

$$v(t) = K_1e^{-3t} + K_2te^{-3t}$$

Since $v(t) = v_C(t)$,

$$v(0) = v_C(0) = 1 = K_1$$

In addition,

$$\frac{dv(t)}{dt} = -3K_1e^{-3t} + K_2e^{-3t} - 3K_2te^{-3t}$$

However,

$$\frac{dv(t)}{dt} = \frac{i(t)}{C} - \frac{v(t)}{R_2C}$$

Setting these two expressions equal to one another and evaluating the resultant equation at $t = 0$ yields

$$\frac{1/2}{1/8} - \frac{1}{1} = -3K_1 + K_2$$
$$3 = -3K_1 + K_2$$

Since $K_1 = 1$, $K_2 = 6$ and the expression for $v(t)$ is

$$v(t) = e^{-3t} + 6te^{-3t}\text{ V}$$

Note that the expression satisfies the initial condition $v(0) = 1$.

The current $i(t)$ can be determined from the nodal analysis equation at $v(t)$.

$$i(t) = C\frac{dv(t)}{dt} + \frac{v(t)}{R_2}$$

Substituting $v(t)$ from the preceding equation, we find

$$i(t) = \frac{1}{8}\left[-3e^{-3t} + 6e^{-3t} - 18te^{-3t}\right] + \frac{1}{8}\left[e^{-3t} + 6te^{-3t}\right]$$

or

$$i(t) = \frac{1}{2}e^{-3t} - \frac{3}{2}te^{-3t} \text{ A}$$

If this expression for the current is employed in the circuit equation,

$$v(t) = -L\frac{di(t)}{dt} - R_1 i(t)$$

we obtain

$$v(t) = e^{-3t} + 6te^{-3t} \text{ V}$$

which is identical to the expression derived earlier.

The MATLAB program for generating a plot of this function, shown in Fig. 6.21, is listed here.

```
>>tau = 1/3;
>>tend = 10*tau;
>>t = linspace(0, tend, 150);
>>v = exp(-3*t) + 6*t.*exp(-3*t);
>>plot(t,v)
>>xlabel('Time (s)')
>>ylabel('Voltage (V)')
```

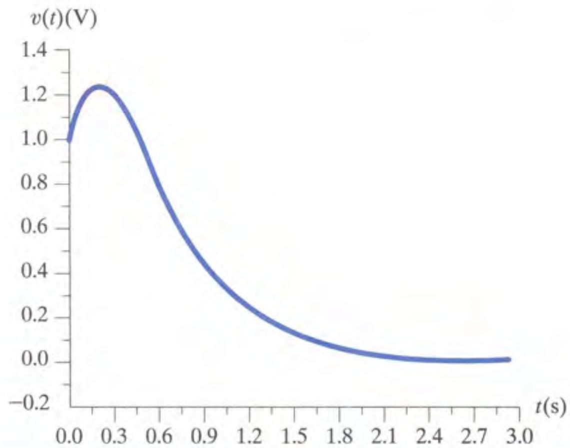


Figure 6.21
Critically damped response.

LEARNING EXTENSION

E6.9 The switch in the network in Fig. E6.9 opens at $t = 0$. Find $i(t)$ for $t > 0$.

ANSWER
$i(t) = -2e^{-t/2} + 4e^{-t}$ A.

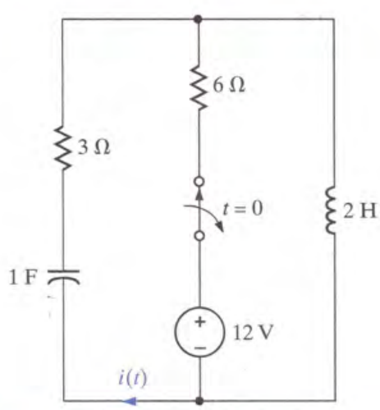


Figure E6.9

LEARNING EXTENSION

E6.10 The switch in the network in Fig. E6.10 moves from position 1 to position 2 at $t = 0$. Find $v_o(t)$ for $t > 0$.

ANSWER $v_o(t) = 2(e^{-t} - 3e^{-3t})$ V.

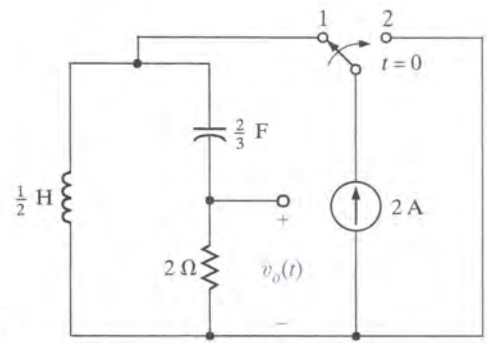


Figure E6.10

LEARNING Example 6.10

Consider the circuit shown in Fig. 6.22. This circuit is the same as that analyzed in Example 6.8, except that a constant forcing function is present. The circuit parameters are the same as those used in Example 6.8:

$$C = 0.04\text{ F} \qquad i_L(0) = 4\text{ A}$$
$$L = 1\text{ H} \qquad v_C(0) = -4\text{ V}$$
$$R = 6\,\Omega$$

We want to find an expression for $v_C(t)$ for $t > 0$.

SOLUTION From our earlier mathematical development we know that the general solution of this problem will consist of a particular solution plus a complementary solution. From Example 6.8 we know that the complementary solution is of the form $K_3e^{-3t}\cos 4t + K_4e^{-3t}\sin 4t$. The particular solution is a constant, since the input is a constant and therefore the general solution is

$$v_C(t) = K_3e^{-3t}\cos 4t + K_4e^{-3t}\sin 4t + K_5$$

An examination of the circuit shows that in the steady state the final value of $v_C(t)$ is 12 V, since in the steady-state condition, the inductor is a short circuit and the capacitor is an open circuit. Thus, $K_5 = 12$. The steady-state value could also be immediately calculated from the differential equation. The form of the general solution is then

$$v_C(t) = K_3e^{-3t}\cos 4t + K_4e^{-3t}\sin 4t + 12$$

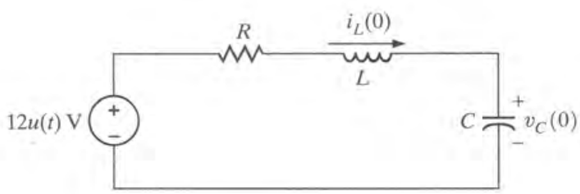


Figure 6.22
Series RLC circuit with a step function input.

The initial conditions can now be used to evaluate the constants K_3 and K_4.

$$v_C(0) = -4 = K_3 + 12$$
$$-16 = K_3$$

Since the derivative of a constant is zero, the results of Example 6.8 show that

$$\frac{dv_C(0)}{dt} = \frac{i(0)}{C} = 100 = -3K_3 + 4K_4$$

and since $K_3 = -16$, $K_4 = 13$. Therefore, the general solution for $v_C(t)$ is

$$v_C(t) = 12 - 16e^{-3t}\cos 4t + 13e^{-3t}\sin 4t \text{ V}$$

Note that this equation satisfies the initial condition $v_C(0) = -4$ and the final condition $v_C(\infty) = 12$ V.

LEARNING EXTENSION

E6.11 The switch in the network in Fig. E6.11 moves from position 1 to position 2 at $t = 0$. Compute $i_o(t)$ for $t > 0$ and use this current to determine $v_o(t)$ for $t > 0$.

ANSWER
$i_o(t) = -\frac{11}{6}e^{-3t} + \frac{14}{6}e^{-6t}$ A,
$v_o(t) = 12 + 18i_o(t)$ V.

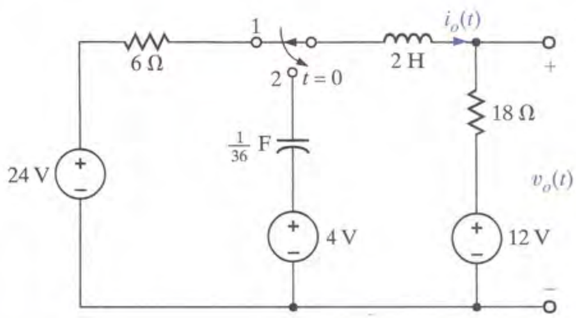


Figure E6.11

6.4 Transient PSPICE Analysis Using Schematic Capture

INTRODUCTION In transient analyses, we determine voltages and currents as functions of time. Typically, the time dependence is demonstrated by plotting the waveforms using time as the independent variable. PSPICE can perform this kind of analysis, called a *Transient* simulation, in which all voltages and currents are determined over a specified time duration. To facilitate plotting, PSPICE uses what is known as the PROBE utility, which will be described later. As an introduction to transient analysis, let us simulate the circuit in Fig. 6.23, plot the voltage $v_C(t)$ and the current $i(t)$, and extract the time constant. Although we will introduce some new PSPICE topics in this section, *Schematics* fundamentals such as getting parts, wiring, and editing part names and values have already been covered in Chapter 4. Also, uppercase text refers to PSPICE utilities and dialog boxes whereas boldface denotes keyboard or mouse inputs.

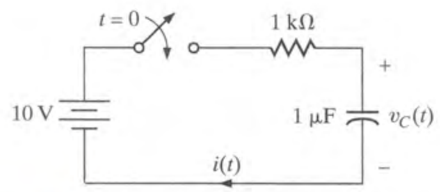


Figure 6.23 *A circuit used for Transient simulation.*

THE SWITCH PARTS The inductor and capacitor parts are called L and C, respectively, and are in the ANALOG library. The switch, called SW_TCLOSE, is in the EVAL library. There is also a SW_TOPEN part that models an opening switch. After placing and wiring the switch along with the other parts, the *Schematics* circuit appears as that shown in Fig. 6.24.

To edit the switch's attributes, double-click on the switch symbol and the ATTRIBUTES box in Fig. 6.25 will appear. Deselecting the **Include Non-changeable Attributes** and **Include System-defined Attributes** fields limits the attribute list to those we can edit and is highly recommended.

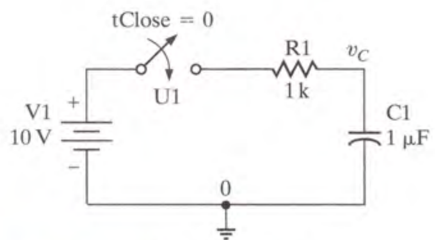


Figure 6.24 *The Schematics circuit.*

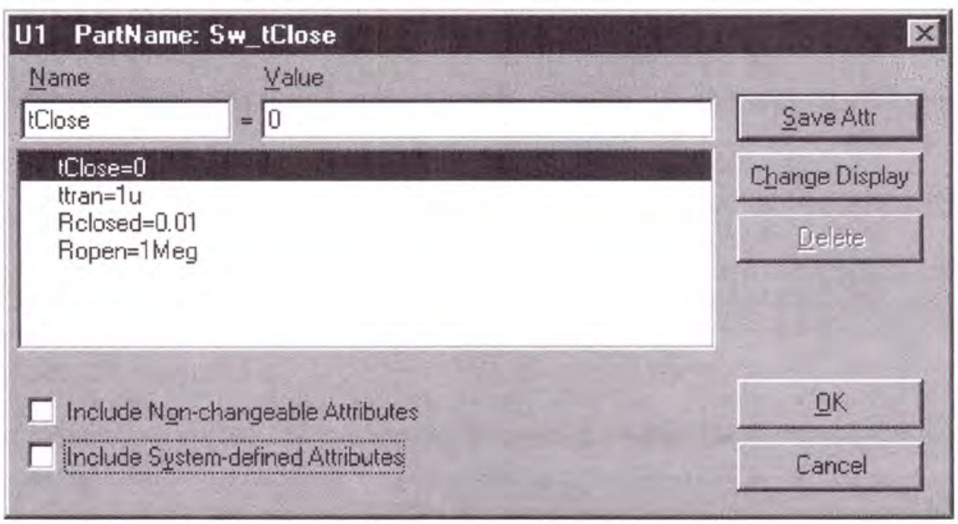


Figure 6.25
The switch's ATTRIBUTES box.

The attribute **tClose** is the time at which the switch begins to close, and **ttran** is the time required to complete the closure. Switch attributes **Rclosed** and **Ropen** are the switch's resistance in the closed and open positions, respectively. During simulations, the resistance of the switch changes linearly from **Ropen** at t = **tClose** to **Rclosed** at t = **tClose** + **ttran.**

When using the SW_TCLOSE and SW_TOPEN parts to simulate ideal switches, care should be taken to ensure that the values for **ttran**, **Rclosed**, and **Ropen** are appropriate for valid simulation results. In our present example, we see that the switch and R_1 are in series, thus, their resistances add. Using the default values listed in Fig. 6.25, we find that when the switch is closed, the switch resistance, **Rclosed**, is 0.01 Ω, 100,000 times smaller than that of the resistor. The resulting series-equivalent resistance is essentially that of the resistor. Alternatively, when the switch is open, the switch resistance is 1 MΩ, 1,000 times larger than that of the resistor. Now, the equivalent resistance is much larger than that of the resistor. Both are desirable scenarios.

To determine a reasonable value for **ttran**, we first estimate the duration of the transient response. The component values yield a time constant of 1 ms, and thus all voltages and currents will reach steady state in about 5 ms. For accurate simulations, **ttran** should be much less than 5 ms. Therefore, the default value of 1 μs, is viable in this case.

THE IMPORTANCE OF PIN NUMBERS As mentioned in Chapter 4, each component within the various Parts libraries has two or more terminals. Within PSPICE, these terminals are called pins and are numbered sequentially starting with pin 1, as shown in Fig. 6.26 for several two-terminal parts. The significance of the pin numbers is their effect on currents plotted using the PROBE utility. PROBE always plots the current entering pin 1 and exiting pin 2. Thus, if the current through an element is to be plotted, the part should be oriented in the *Schematics* diagram such that the defined current direction enters the part at pin 1. This can be done by using the ROTATE command in the EDIT menu. ROTATE causes the part to spin 90° counterclockwise. In our example, we will plot the current $i(t)$ by plotting the current through the capacitor, I(C1). Therefore, when the *Schematics* circuit in Fig. 6.24 was created, the capacitor was rotated 270°. As a result, pin 1 is at the top of the diagram and the assigned current direction in Fig. 6.23 matches the direction presumed by PROBE. If a component's current direction in PROBE is opposite the desired direction, simply go to the *Schematics* circuit, rotate the part in question 180°, and re-simulate.

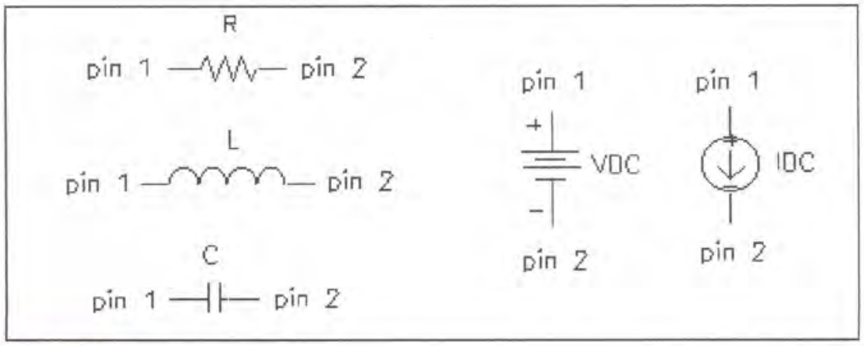


Figure 6.26
Pin numbers for common PSPICE parts.

SETTING INITIAL CONDITIONS To set the initial condition of the capacitor voltage, double-click on the capacitor symbol in Fig. 6.24 to open its ATTRIBUTE box, as shown in Fig. 6.27. Click on the **IC** field and set the value to the desired voltage, 0 V in this example. Setting the initial condition on an inductor current is done in a similar fashion. Be forewarned that the initial condition for a capacitor voltage is positive at pin 1 versus pin 2. Similarly, the initial condition for an inductor's current will flow into pin 1 and out of pin 2.

Figure 6.27
Setting the capacitor initial condition.

SETTING UP A TRANSIENT ANALYSIS The simulation duration is selected using **SETUP** from the **ANALYSIS** menu. When the SETUP window shown in Fig. 6.28 appears, double-click on the text **TRANSIENT** and the TRANSIENT window in Fig. 6.29 will appear. The simulation period described by **Final time** is selected as 6 milliseconds. All simulations start at $t = 0$. The **No-Print Delay** field sets the time the simulation runs before data collection begins. **Print Step** is the interval used for printing data to the output file. **Print Step** has no effect on the data used to create PROBE plots. The **Detailed Bias Pt.** option is useful when simulating circuits containing transistors and diodes, and thus will not be used here. When **Skip initial transient solution** is enabled, all capacitors and inductors that do not have specific initial condition values in their ATTRIBUTES boxes will use zero initial conditions.

Sometimes, plots created in PROBE are not smooth. This is caused by an insufficient number of data points. More data points can be requested by inputting a **Step Ceiling** value. A reasonable first guess would be a hundredth of the **Final Time**. If the resulting PROBE plots are still unsatisfactory, reduce the **Step Ceiling** further. As soon as the TRANSIENT window is complete, simulate the circuit by selecting **Simulate** from the **Analysis** menu.

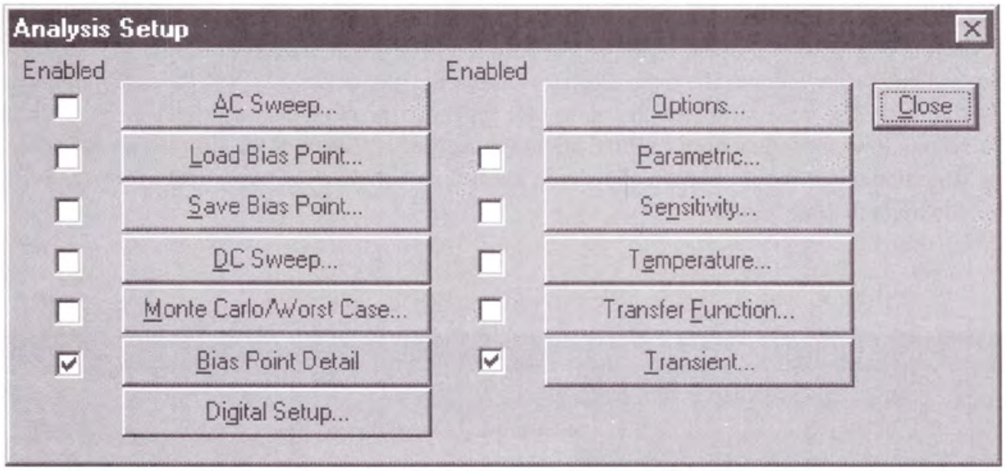


Figure 6.28
The ANALYSIS SETUP window.

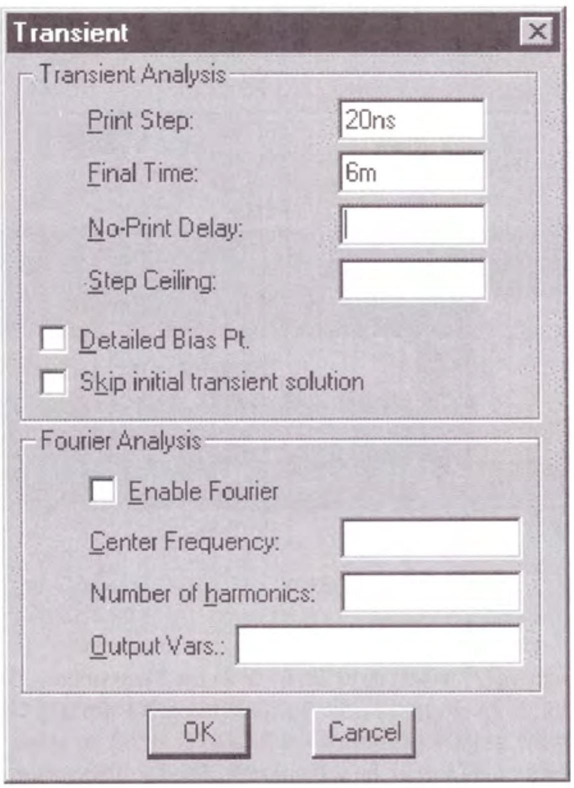


Figure 6.29
The TRANSIENT window.

PLOTTING IN PROBE When the PSPICE simulation is finished, the PROBE window shown in Fig. 6.30 will open. If not, select **Run Probe** from the **Analysis** menu. In Fig. 6.30, we see three subwindows: the main display window, the output window, and the simulation status window. The waveforms we choose to plot appear in the main display window. The output window shows messages from PSPICE about the success or failure of the simulation. Run-time information about the simulation appears in the simulation status window. Here we will focus on the main display window.

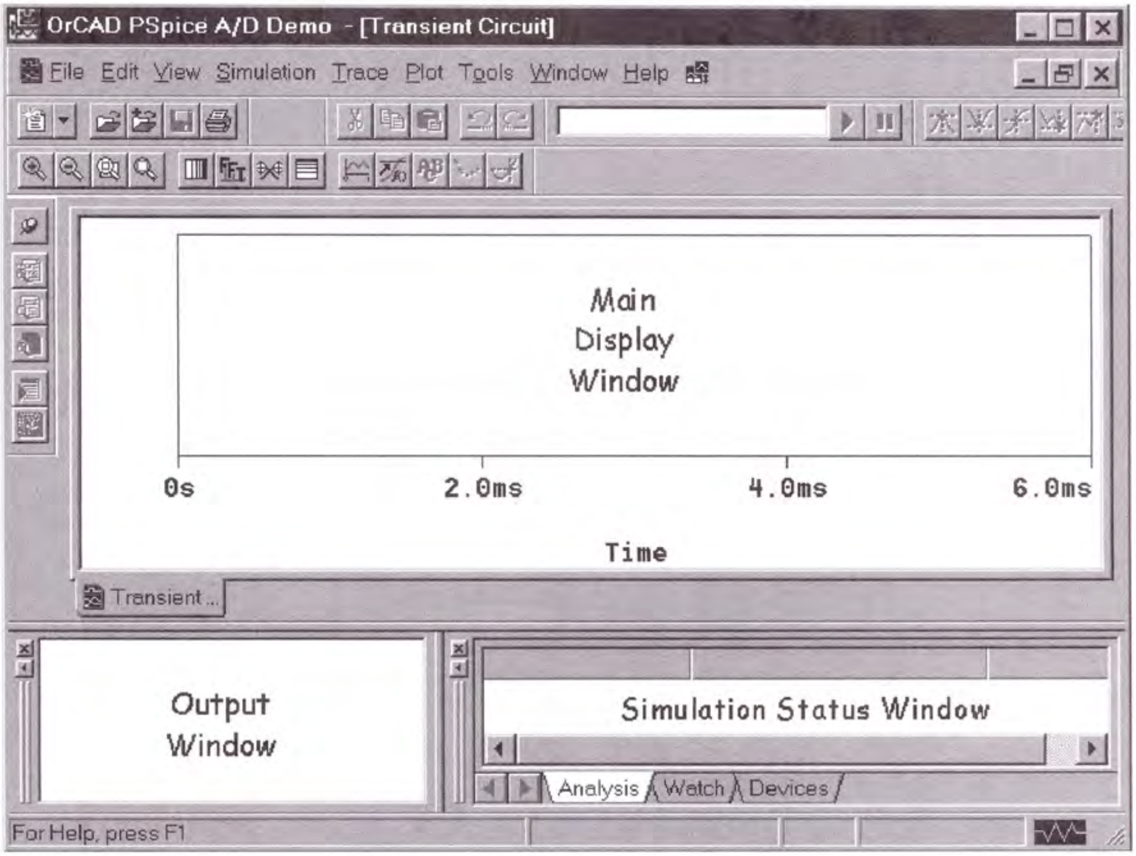


Figure 6.30
The PROBE window.

To plot the voltage, $v_C(t)$, select **Add Trace** from the **Trace** menu. The ADD TRACES window is shown in Fig. 6.31. Note that the options **Alias Names** and **Subcircuit Nodes** have been deselected, which greatly simplifies the ADD TRACES window. The capacitor voltage is obtained by clicking on **V(Vc)** in the left column. The PROBE window should look like that shown in Fig. 6.32.

Before adding the current $i(t)$ to the plot, we note that the dc source is 10 V and the resistance is 1 kΩ, which results in a loop current of a few milliamps. Since the capacitor voltage span is much greater, we will plot the current on a second y axis. From the **Plot** menu, select

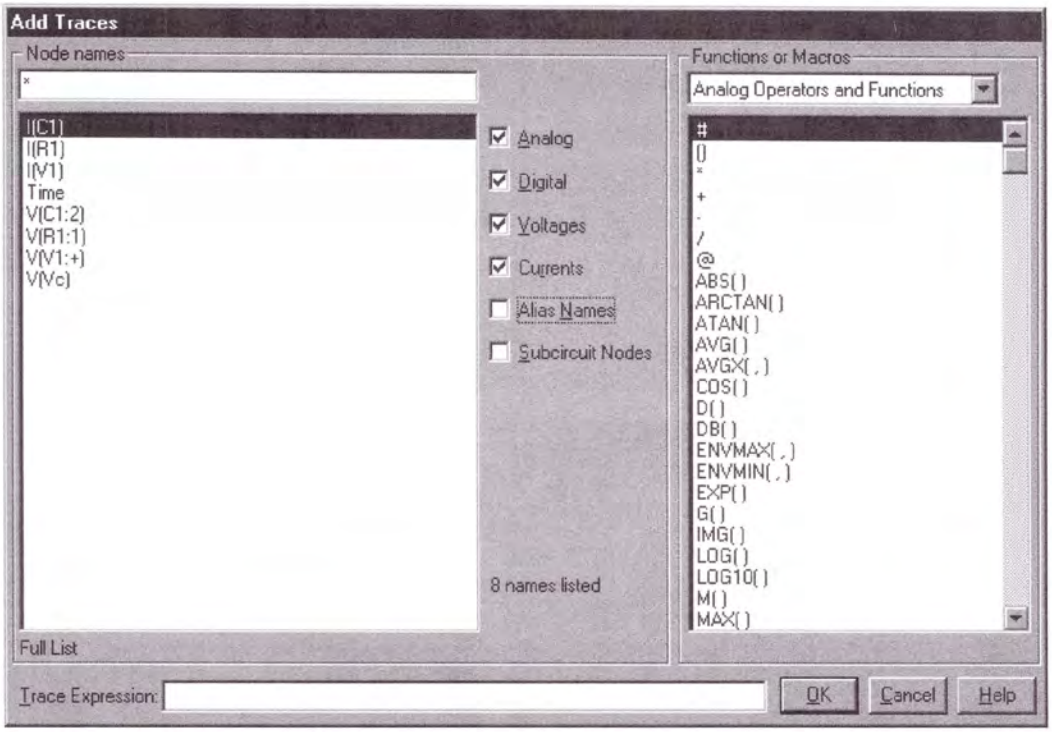


Figure 6.31
The Add Traces window.

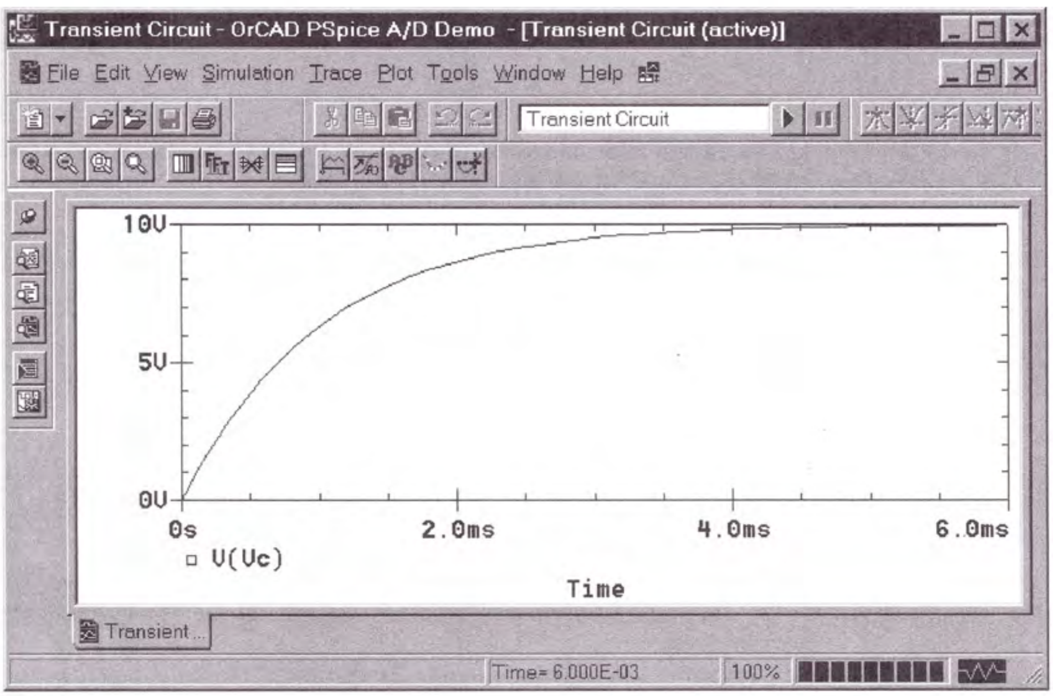


Figure 6.32
The capacitor voltage.

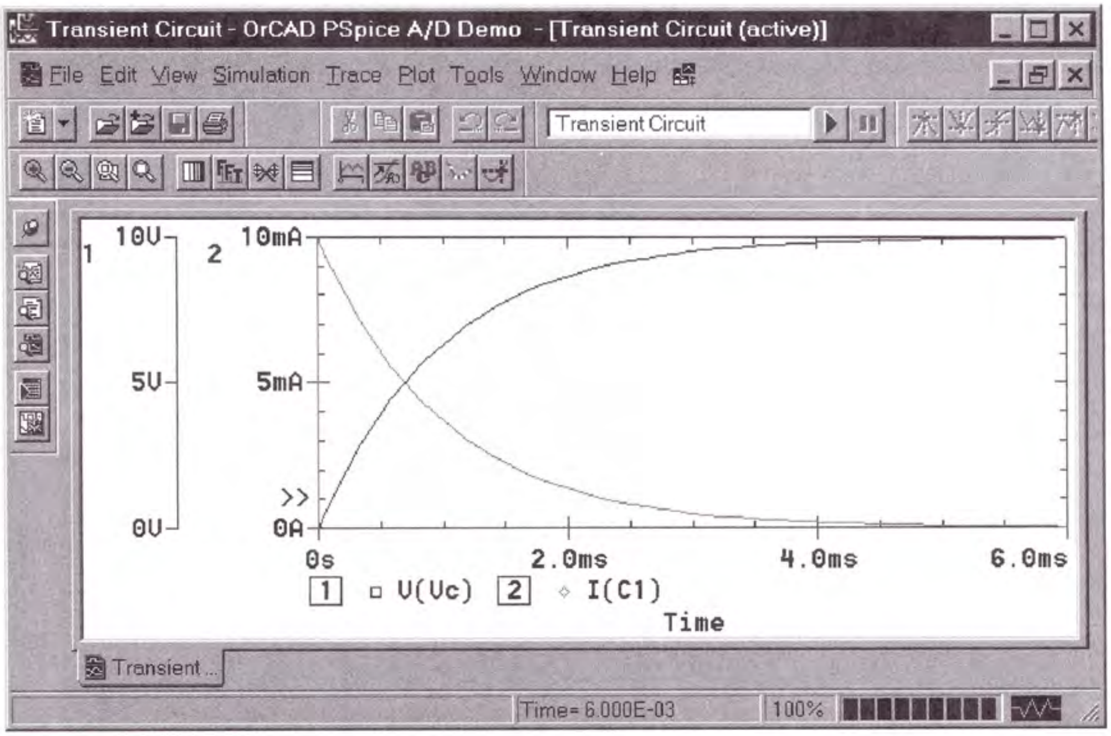


Figure 6.33
The capacitor voltage and the clockwise loop current.

Add Y Axis. To add the current to the plot, select **Add Trace** from the **Trace** menu, then select **I(C1)**. Figure 6.33 shows the PROBE plot for $v_C(t)$ and $i(t)$.

FINDING THE TIME CONSTANT Given that the final value of $v_C(t)$ is 10 V, we can write

$$v_C(t) = 10\left[1 - e^{-t/\tau}\right] \text{ V}$$

When $t = \tau$,

$$v_C(\tau) = 10\left(1 - e^{-1}\right) \text{ V} = 6.32 \text{ V}$$

To determine the time at which the capacitor voltage is 6.32 V, we activate the cursors by selecting **Cursor/Display** in the **Trace** menu. There are two cursors that can be used to extract x-y data from the plots. Use the ← and → arrow keys to move the first cursor. By holding the SHIFT key down, the arrow keys move the second cursor. Moving the first cursor along the voltage plot, as shown in Fig. 6.34, we find that 6.32 V occurs at a time of 1 ms. Therefore, the time constant is 1 ms—exactly the RC product.

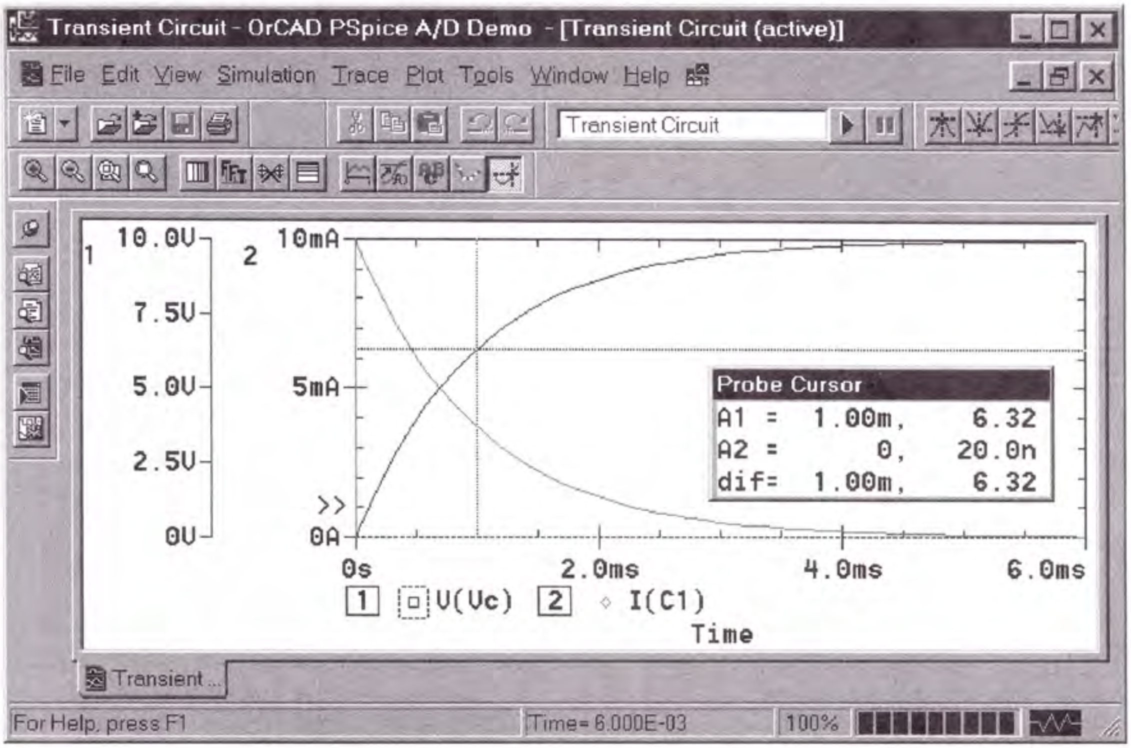


Figure 6.34
PROBE plot for $v_C(t)$ and time constant extraction.

SAVING AND PRINTING PROBE PLOTS Saving plots within PROBE requires the use of the **Display Control** command in the **Windows** menu. Using the SAVE/RESTORE DISPLAY window shown in Fig. 6.35, we simply name the plot and click on **Save**. Figure 6.35 shows that one plot has already been saved, TranExample. This procedure saves the plot attributes such as axes settings, additional text, and cursor settings in a file with a .prb extension. Additionally, the .prb file contains a reference to the appropriate data file, which has a .dat extension, and contains the actual simulation results. Therefore, in using **Display Control** to save a plot, we do not save the plot itself, only the plot settings and the .dat file's name. To access an old PROBE plot, enter PROBE, and from the **File** menu, open the appropriate .dat file. Next, access the DISPLAY CONTROL window, select the file of interest and click on **RESTORE**. Use the **Save As** option in Fig. 6.35 to save the .prb file to any directory on any disk, hard or floppy.

To copy the PROBE plot to other documents such as word processors, select the **Copy to Clipboard** command in the **Window** menu. The window in Fig. 6.36 will appear showing several options. If your PROBE display screen background is black, it is recommended that you choose the **make window and plot backgrounds transparent** and **change white to black** options. When the plot is pasted, it will have a white background with black text—a better scenario for printing. To print a PROBE plot, select **Print** from the **File** menu and the PRINT window in Fig. 6.37 will open. The options in this window are self-explanatory. Note that printed plots have white backgrounds with black text.

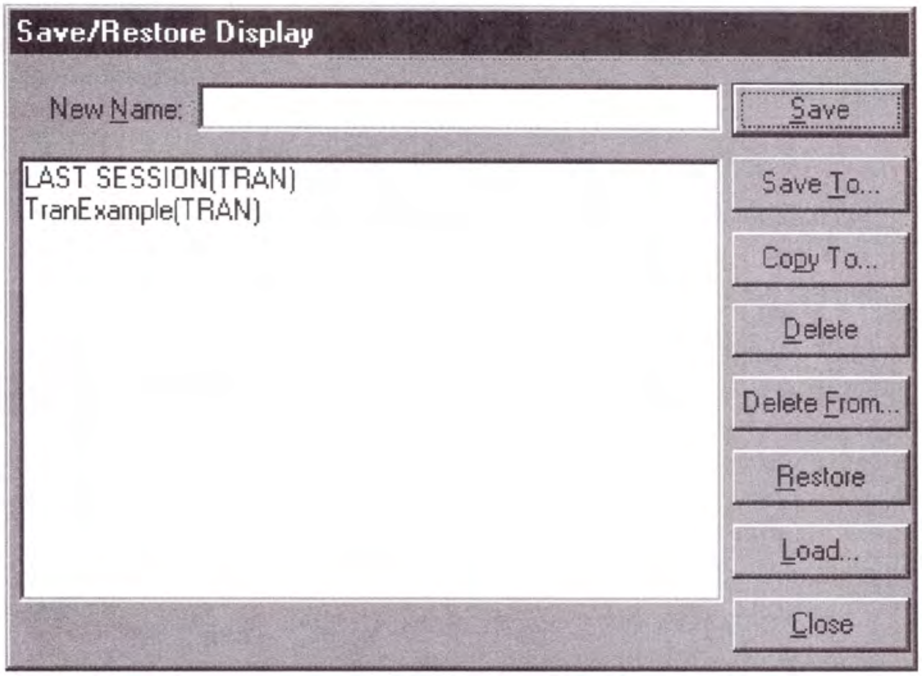


Figure 6.35
The SAVE/RESTORE DISPLAY window used in PROBE to save plots.

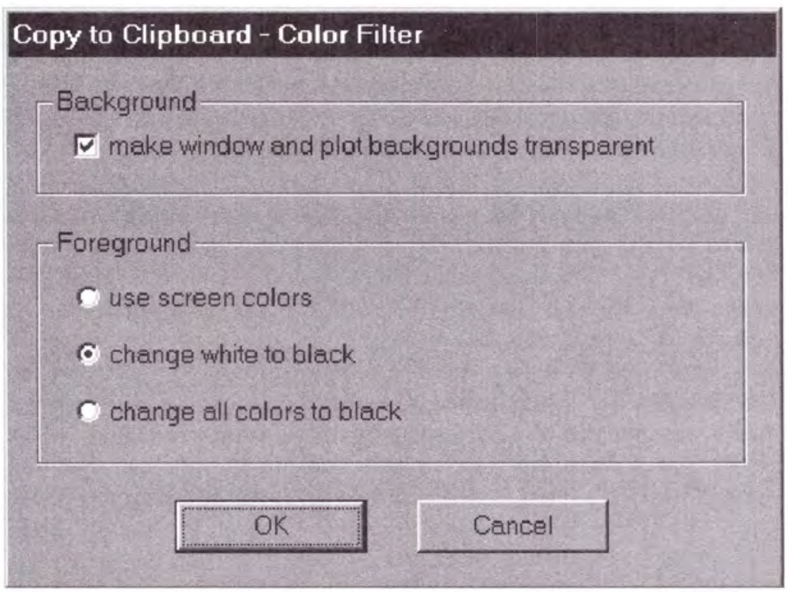


Figure 6.36
The SAVE/RESTORE DISPLAY window used in PROBE to save plots.

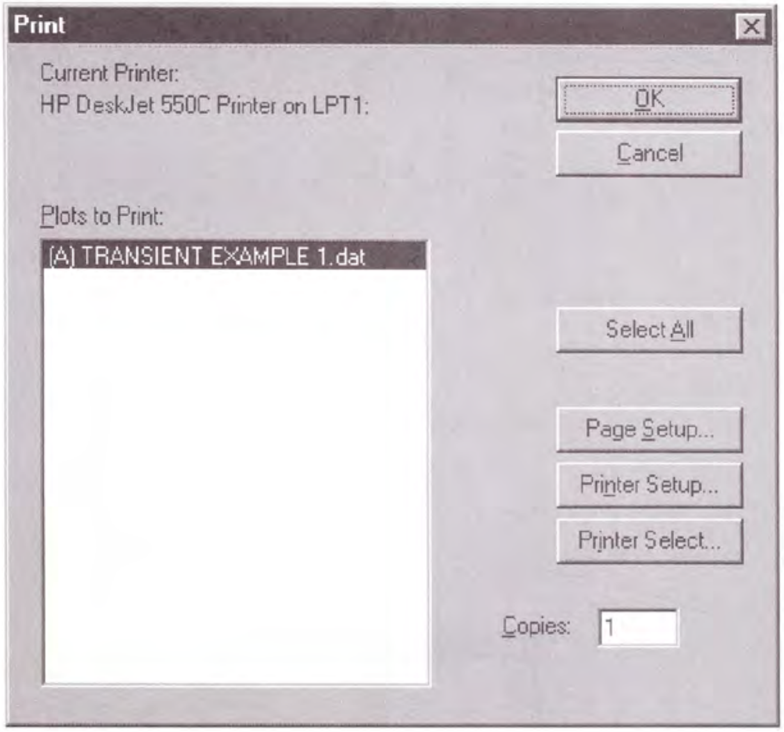


Figure 6.37
The PRINT window used in PROBE.

OTHER PROBE FEATURES There are several features within PROBE for plot manipulation and data extraction. Within the **Plot** menu resides commands for editing the plot itself. These include altering the axes, adding axes, and adding additional plots to the page. Also in the **Tools** menu is the **Label** command, which allows one to add marks (data point values), explanatory text, lines, and shapes to the plot.

LEARNING Example 6.11

Using the PSPICE *Schematics* editor, draw the circuit in Fig. 6.38 and use the PROBE utility to find the time at which the capacitor and inductor current are equal.

SOLUTION Figure 6.39a shows the *Schematics* circuit, and the simulation results are shown in Fig. 6.39b. Based on the PROBE plot, the currents are equal at 561.8 ns.

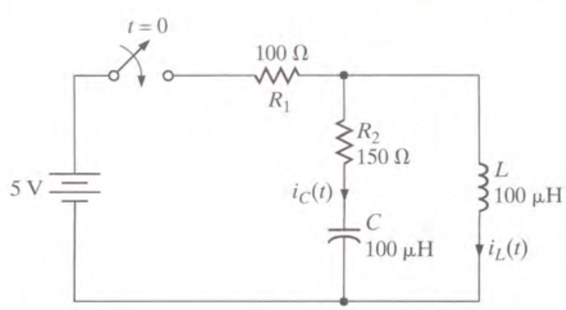


Figure 6.38
Circuit used in Example 6.11.

(continued)

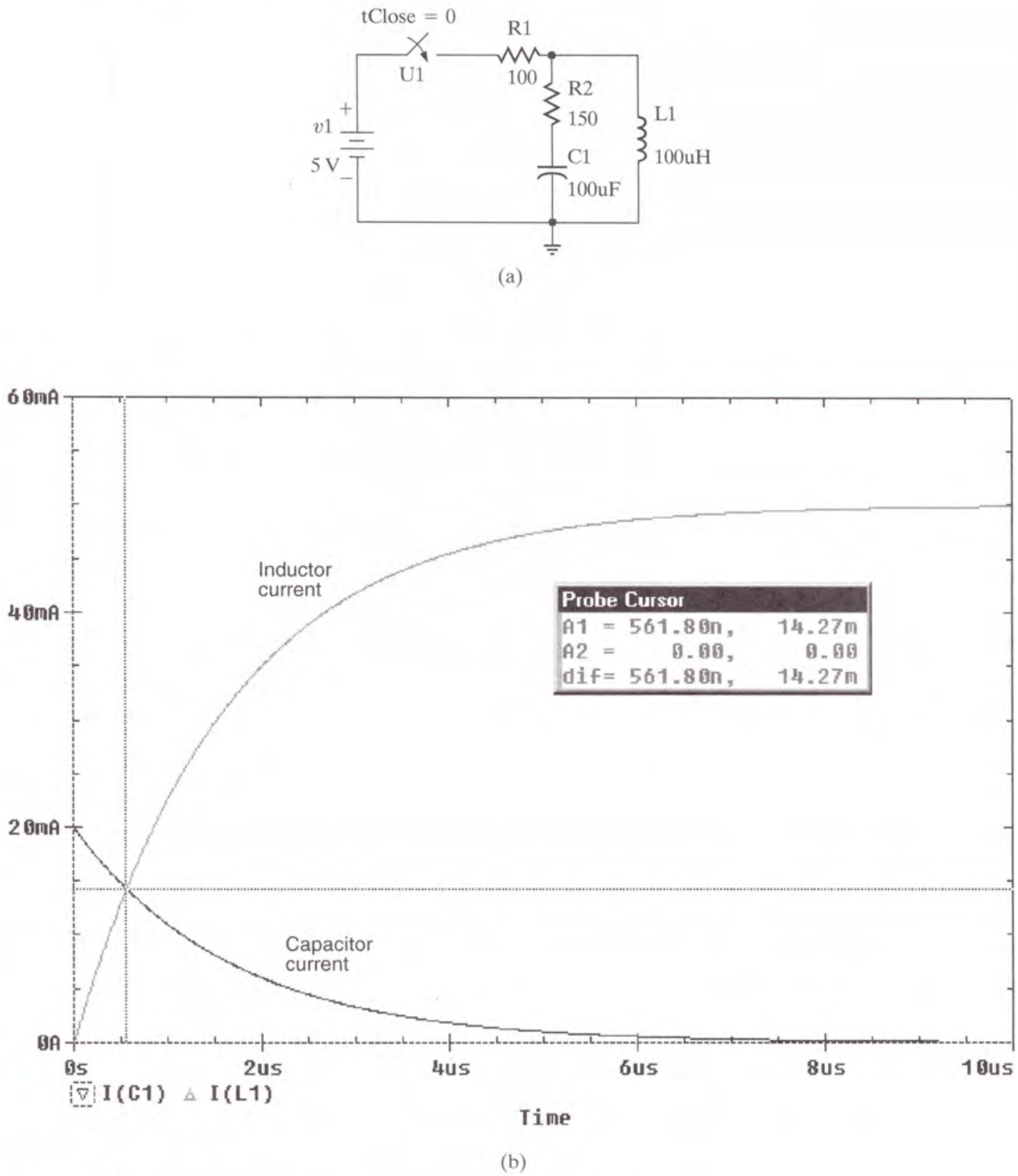


Figure 6.39
(a) PSPICE network and (b) simulation results for $i_C(t)$ and $i_L(t)$ in Example 6.11.

Learning by Application

There are a wide variety of applications for transient circuits. The following examples will serve to demonstrate some of them.

LEARNING Example 6.12

A heart pacemaker circuit is shown in Fig. 6.40. The SCR (silicon-controlled rectifier) is a solid-state device that has two distinct modes of operation. When the voltage across the SCR is increasing but less than 5 V, the SCR behaves like an open circuit, as shown in Fig. 6.41a. Once the voltage across the SCR reaches 5 V, the device functions like a current source, as shown in Fig. 6.41b. This behavior will continue as long as the SCR voltage remains above 0.2 V. At this voltage, the SCR shuts off and again becomes an open circuit.

Assume that at $t = 0$, $v_C(t)$ is 0 V and the 1-μF capacitor begins to charge toward the 6-V source voltage. Find the resistor value such that $v_C(t)$ will equal 5 V (the SCR firing voltage) at 1 s. At $t = 1$ s, the SCR fires and begins discharging the capacitor. Find the time required for $v_C(t)$ to drop from 5 V to 0.2 V. Finally, plot $v_C(t)$ for the three cycles.

SOLUTION For $t < 1$ s, the equivalent circuit for the pacemaker is shown in Fig. 6.42. As indicated earlier, the capacitor voltage has the form

$$v_C(t) = 6 - 6e^{-t/RC}$$

A voltage of 0.2 V occurs at

$$t_1 = 0.034RC$$

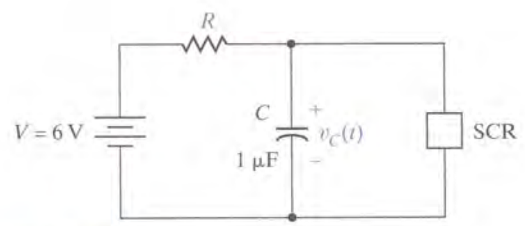


Figure 6.40
Heart pacemaker equivalent circuit.

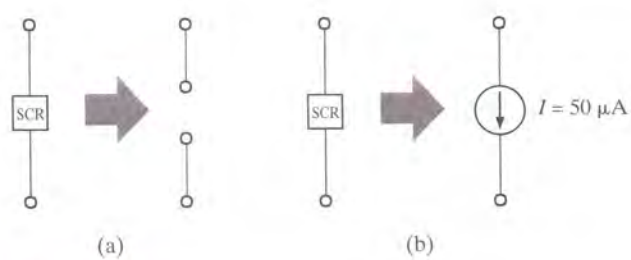


Figure 6.41
Equivalent circuits for silicon-controlled rectifer.

(continued)

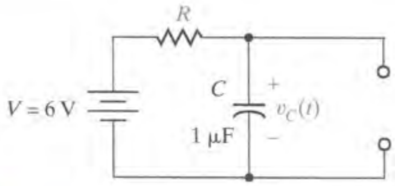


Figure 6.42
Pacemaker equivalent network during capacitor charge cycle.

whereas a voltage of 5 V occurs at

$$t_2 = 1.792RC$$

We desire that $t_2 - t_1 = 1$ s. Therefore,

$$t_2 - t_1 = 1.758RC = 1 \text{ s}$$

and

$$RC = 0.569 \text{ s} \quad \text{and} \quad R = 569 \text{ k}\Omega$$

At $t = 1$ s the SCR fires and the pacemaker is modeled by the circuit in Fig. 6.43. The form of the discharge waveform is

$$v(t) = K_1 + K_2 e^{-(t-1)/RC}$$

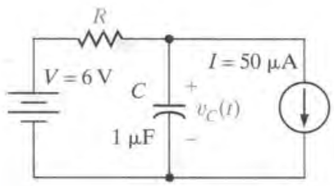


Figure 6.43
Pacemaker equivalent network during capacitor discharge cycle.

The term $(t - 1)$ appears in the exponential to shift the function 1 s, since during that time the capacitor was charging. Just after the SCR fires at $t = 1^+$ s, $v_C(t)$ is still 5 V, whereas at $t = \infty$, $v_C(t) = 6 - IR$. Therefore,

$$K_1 + K_2 = 5 \quad \text{and} \quad K_1 = 6 - IR$$

Our solution, then, is of the form

$$v_C(t) = 6 - IR + (IR - 1)e^{-(t-1)/RC}$$

Let T be the time beyond 1 s necessary for $v(t)$ to drop to 0.2 V. We write

$$v_C(T + 1) = 6 - IR + (IR - 1)e^{-T/RC} = 0.2$$

Substituting for I, R, and C, we find

$$T = 0.11 \text{ s}$$

The output waveform is shown in Fig. 6.44.

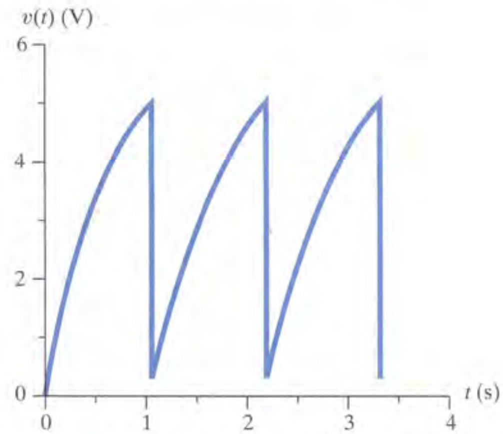


Figure 6.44 *Heart pacemaker output voltage waveform.*

LEARNING Example 6.13

One of the most common and necessary subcircuits that appears in a wide variety of electronic systems—for example, stereos, TVs, radios, and computers—is a quality dc voltage source or power supply. The standard wall socket supplies an alternating current (ac) voltage waveform shown in Fig. 6.45a, and the conversion of this voltage to a desired dc level is done as illustrated in Fig. 6.45b. The ac waveform is converted to a quasi-dc voltage by an inexpensive ac–dc converter whose output contains remnants of the ac input and is unregulated. A higher quality dc output is created by a switching dc–dc converter. Of the several versions of dc–dc converters, we will focus on a topology called the boost converter, shown in Fig. 6.46. Let us develop an equation relating the output voltage to the switching characteristics.

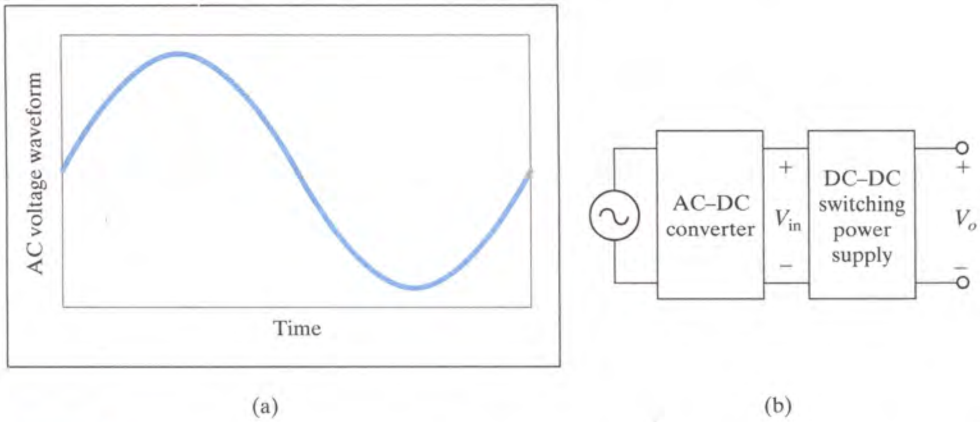


Figure 6.45
(a) The ac voltage waveform at a standard wall outlet and (b) a block diagram of a modern dc power supply.

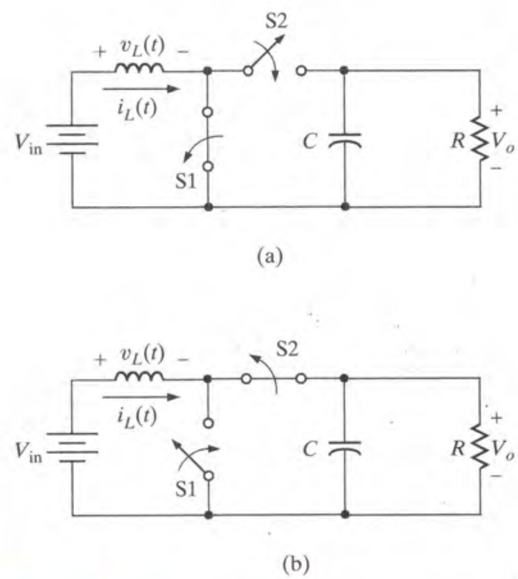


Figure 6.46
The boost converter with switch settings for time intervals (a) t_{on} and (b) t_{off}.

SOLUTION Consider the boost converter in Fig. 6.46a, where switch 1 (S1) is closed and S2 is open for a time interval t_{on}. This isolates the inductor from the capacitor, creating two subcircuits that can be analyzed independently. Note that during t_{on} the inductor current and stored energy are increasing while at the output node, the capacitor voltage discharges exponentially into the load. If the capacitor's time constant ($\tau = RC$) is large, then the output voltage will decrease slowly. Thus, during t_{on} energy is stored in the inductor and the capacitor provides energy to the load.

(continued)

Next, we change both switch positions so that S1 is open and S2 is closed for a time interval t_{off}, as seen in Fig. 6.46b. Since the inductor current cannot change instantaneously, current flows into the capacitor and the load, recharging the capacitor. During t_{off} the energy that was added to the inductor during t_{on} is used to recharge the capacitor and drive the load. When t_{off} has elapsed, the cycle is repeated.

Note that the energy added to the inductor during t_{on} must go to the capacitor and load during t_{off}, otherwise, the inductor energy would increase to the point that the inductor would fail. This requires that the energy stored in the inductor must be the same at the end of each switching cycle. Recalling that the inductor energy is related to the current by

$$w(t) = \frac{1}{2}Li^2(t)$$

we can state that the inductor current must also be the same at the end of each switching cycle, as shown in Fig. 6.47. The inductor current during t_{on} and t_{off} can be written as

$$i_L(t) = \frac{1}{L}\int_0^{t_{on}} v_L(t)\,dt = \frac{1}{L}\int_0^{t_{on}} V_{in}\,dt = \left[\frac{V_{in}}{L}\right]t_{on} + I_0 \qquad 0 < t < t_{on}$$

$$i_L(t) = \frac{1}{L}\int_{t_{on}}^{t_{on}+t_{off}} v_L(t)\,dt = \frac{1}{L}\int_{t_{on}}^{t_{on}+t_{off}} (V_{in} - V_o)\,dt =$$

$$\left[\frac{V_{in} - V_o}{L}\right]t_{off} + I_0 \qquad t_{on} < t < t_{off} \qquad \textbf{6.22}$$

where I_0 is the initial current at the beginning of each switching cycle. If the inductor current is the same at the beginning and end of each switching cycle, then the integrals in Eq. (6.22) must sum to zero. Or,

$$V_{in}t_{on} = (V_o - V_{in})t_{off} = (V_o - V_{in})(T - t_{on})$$

where T is the period $(T = t_{on} + t_{off})$. Solving for V_o yields

$$V_o = V_{in}\left[\frac{T}{T - t_{on}}\right] = V_{in}\left[\frac{1}{(T - t_{on})/T}\right] =$$

$$V_{in}\left[\frac{1}{(1 - t_{on}/T)}\right] = V_{in}\left[\frac{1}{1 - D}\right]$$

where D is the duty cycle $(D = t_{on}/T)$. Thus, by controlling the duty cycle, we control the output voltage. Since D is always a positive fraction, V_o is always bigger than V_{in}—thus the name, boost converter. A plot of V_o/V_{in} versus duty cycle is shown in Fig. 6.48.

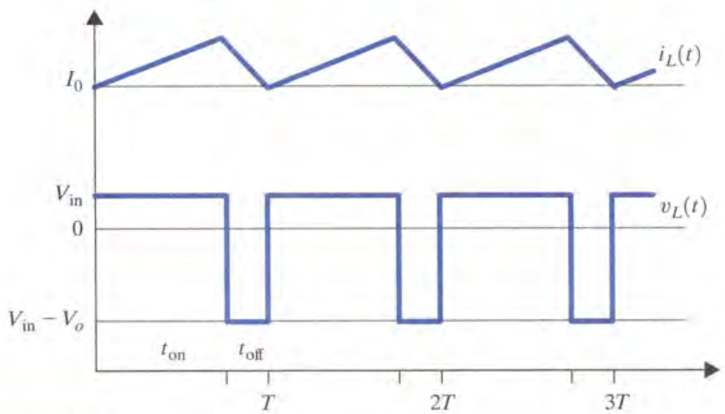


Figure 6.47
Waveform sketches for the inductor voltage and current.

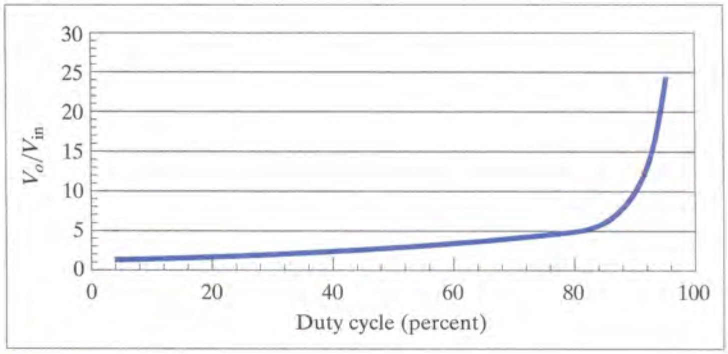


Figure 6.48
Effect of duty cycle on boost converter gain.

Learning by Design

The following example provides an introduction to some typical strategies for transient circuits. In this example the problem involves the selection of the proper circuit parameters to achieve a specified transient response.

LEARNING Example 6.14

The network in Fig. 6.49 models an automobile ignition system. The voltage source represents the standard 12-V battery. The inductor is the ignition coil, which is magnetically coupled to the starter (not shown). The inductor's internal resistance is modeled by the resistor, and the switch is the keyed ignition switch. Initially the switch connects the ignition circuitry to the battery, and thus the capacitor is charged to 12 V. To start the motor, we close the switch, thereby discharging the capacitor through the inductor. Assuming that optimum starter operation requires an overdamped response for $i_L(t)$ that reaches at least 1 A within 100 ms after switching and remains above 1 A for between 1 and 1.5 s, let us find a value for the capacitor that will produce such a current waveform. In addition, let us plot the response including the time interval just prior to moving the switch and verify our design.

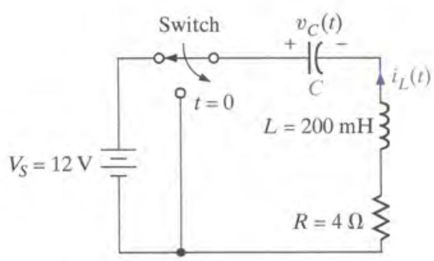


Figure 6.49
Circuit model for ignition system.

(continued)

SOLUTION Before the switch is moved at $t = 0$, the capacitor looks like an open circuit and the inductor acts like a short circuit. Thus,

$$i_L(0^-) = i_L(0^+) = 0 \text{ A} \quad \text{and} \quad v_C(0^-) = v_C(0^+) = 12 \text{ V}$$

After switching, the circuit is a series RLC unforced network described by the characteristic equation

$$s^2 + \frac{R}{L}s + \frac{1}{LC} = 0$$

with roots at $s = -s_1$ and $-s_2$. The characteristic equation is of the form

$$(s + s_1)(s + s_2) = s^2 + (s_1 + s_2)s + s_1 s_2 = 0$$

Comparing the two expressions, we see that

$$\frac{R}{L} = s_1 + s_2 = 20$$

and

$$\frac{1}{LC} = s_1 s_2$$

Since the network must be overdamped, the inductor current is of the form

$$i_L(t) = K_1 e^{-s_1 t} + K_2 e^{-s_2 t}$$

Just after switching,

$$i_L(0^+) = K_1 + K_2 = 0$$

or

$$K_2 = -K_1$$

Also, at $t = 0^+$, the inductor voltage equals the capacitor voltage because $i_L = 0$ and therefore $i_L R = 0$. Thus, we can write

$$v_L(0^+) = L\frac{di_L(0^+)}{dt} \Rightarrow -s_1 K_1 + s_2 K_1 = \frac{12}{L}$$

or

$$K_1 = \frac{60}{s_2 - s_1}$$

At this point, let us arbitrarily choose $s_1 = 3$ and $s_2 = 17$, which satisfies the condition $s_1 + s_2 = 20$, and furthermore,

$$K_1 = \frac{60}{s_2 - s_1} = \frac{60}{14} = 4.29$$

$$C = \frac{1}{Ls_1 s_2} = \frac{1}{(0.2)(3)(17)} = 98 \text{ mF}$$

Hence, $i_L(t)$ is

$$i_L(t) = 4.29[e^{-3t} - e^{-17t}] \text{ A}$$

Figure 6.50a shows a plot of $i_L(t)$. At 100 ms the current has increased to 2.39 A, which meets the initial magnitude specifications. However, one second later at $t = 1.1$ s, $i_L(t)$ has fallen to only 0.16 A—well below the magnitude-over-time requirement. Simply put, the current falls too quickly. To make an informed estimate for s_1 and s_2, let us investigate the effect the roots exhibit on the current waveform when $s_2 > s_1$.

Since $s_2 > s_1$, the exponential associated with s_2 will decay to zero faster than that associated with s_1. This causes $i_L(t)$ to rise—the larger the value of s_2, the faster the rise. After $5(1/s_2)$ seconds have elapsed, the exponential associated with s_2 is approximately zero and $i_L(t)$ decreases exponentially with a time constant of $\tau = 1/s_1$. Thus, to slow the fall of $i_L(t)$ we should reduce s_1. Hence, let us choose $s_1 = 1$. Since $s_1 + s_2$ must equal 20, $s_2 = 19$. Under these conditions

$$C = \frac{1}{Ls_1 s_2} = \frac{1}{(0.2)(1)(19)} = 263 \text{ mF}$$

and

$$K_1 = \frac{60}{s_2 - s_1} = \frac{60}{18} = 3.33$$

Thus, the current is

$$i_L(t) = 3.33[e^{-t} - e^{-19t}] \text{ A}$$

which is shown in Fig. 6.50b. At 100 ms the current is 2.52 A. Also, at $t = 1.1$ s, the current is 1.11 A—above the 1-A requirement. Therefore, the choice of $C = 263$ mF meets all starter specifications.

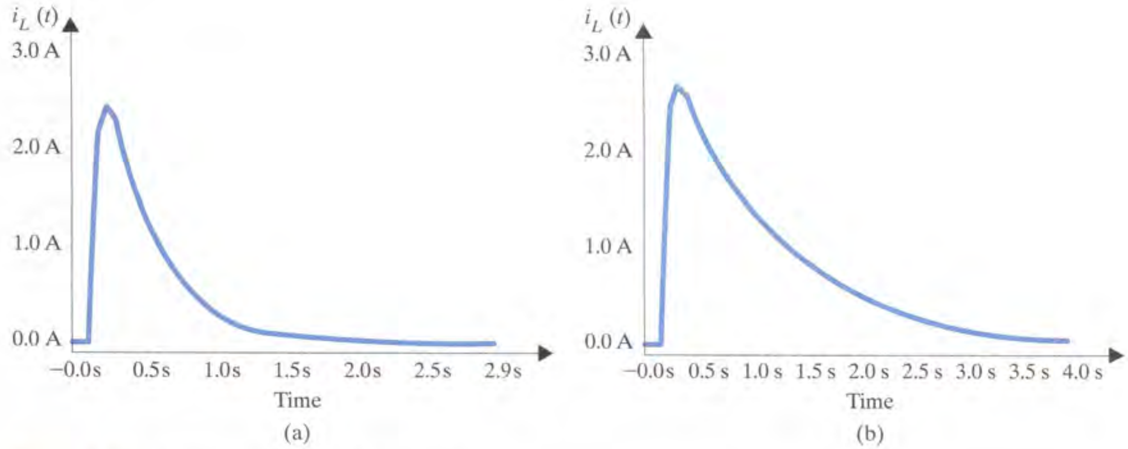


Figure 6.50 *Ignition current as a function of time.*

LEARNING Check

Summary

First-Order Circuits

- An *RC* or *RL* transient circuit is said to be first order if it contains only a single capacitor or single inductor. The voltage or current anywhere in the network can be obtained by solving a first-order differential equation.
- The form of a first-order differential equation with a constant forcing function is

$$\frac{dx(t)}{dt} + \frac{x(t)}{\tau} = A$$

and the solution is

$$x(t) = A\tau + K_2 e^{-t/\tau}$$

where $A\tau$ is referred to as the steady-state solution and τ is called the time constant.

- The function $e^{-t/\tau}$ decays to a value that is less than 1% of its initial value after a period of 5τ. Therefore, the time constant, τ, determines the time required for the circuit to reach steady state.
- The time constant for an *RC* circuit is $R_{Th}C$ and for an *RL* circuit is L/R_{Th}, where R_{Th} is the Thévenin equivalent resistance looking into the circuit at the terminals of the storage element (i.e., capacitor or inductor).
- The two approaches proposed for solving first-order transient circuits are the differential equation approach and the step-by-step method. In the former case, the differential equation that describes the dynamic behavior of the circuit is solved to determine the desired solution. In the latter case, the initial conditions and the steady-state value of the voltage across the capacitor or current in the inductor are used in conjunction with the circuit's time constant and the known form of the desired variable to obtain a solution.
- The response of a first-order transient circuit to an input pulse can be obtained by treating the pulse as a combination of two step-function inputs.

Second-Order Circuits

- The voltage or current in an *RLC* transient circuit can be described by a constant coefficient differential equation of the form

$$\frac{d^2x(t)}{dt^2} + 2\zeta\omega_0\frac{dx(t)}{dt} + \omega_0^2 x(t) = f(t)$$

where $f(t)$ is the network forcing function.

- The characteristic equation for a second-order circuit is $s^2 + 2\zeta\omega_0 s + \omega_0^2 = 0$, where ζ is the damping ratio and ω_0 is the undamped natural frequency.
- If the two roots of the characteristic equation are
  - real and unequal, then $\zeta > 1$ and the network response is overdamped
  - real and equal, then $\zeta = 1$ and the network response is critically damped
  - complex conjugates, then $\zeta < 1$ and the network response is underdamped
- The three types of damping together with the corresponding network response are as follows:
  1. Overdamped: $x(t) = K_1 e^{-(\zeta\omega_0 - \omega_0\sqrt{\zeta^2-1})t} + K_2 e^{-(\zeta\omega_0 + \omega_0\sqrt{\zeta^2-1})t}$
  2. Critically damped: $x(t) = B_1 e^{-\zeta\omega_0 t} + B_2 t e^{-\zeta\omega_0 t}$
  3. Underdamped: $x(t) = e^{-\sigma t}(A_1 \cos\omega_d t + A_2 \sin\omega_d t)$, where $\sigma = \zeta\omega_0$ and $\omega_d = \omega_0\sqrt{1-\zeta^2}$
- Two initial conditions are required to derive the two unknown coefficients in the network response equations.

Problems

For solutions and additional help on problems marked with ► go to www.wiley.com/college/irwin

SECTION 6.2

6.1 ► **Use the differential equation approach to find $v_C(t)$ for $t > 0$ in the circuit in Fig. P6.1.**

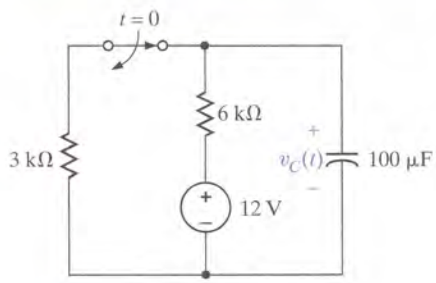


Figure P6.1

6.2 Use the differential equation approach to find $i(t)$ for $t > 0$ in the network in Fig. P6.2.

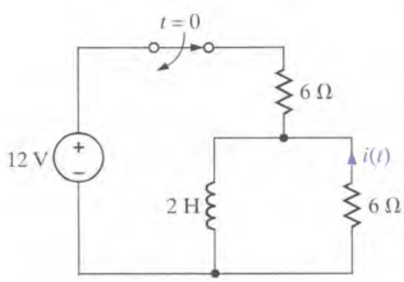


Figure P6.2

6.3 Use the differential equation approach to find $v_C(t)$ for $t > 0$ in the circuit in Fig. P6.3.

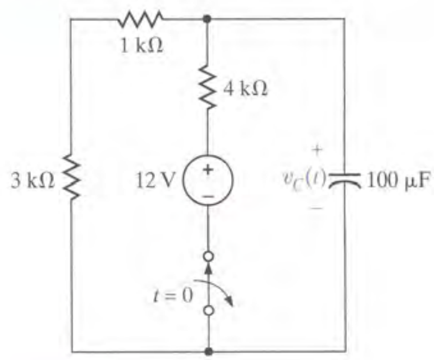


Figure P6.3

6.4 Use the differential equation approach to find $i_o(t)$ for $t > 0$ in the network in Fig. P6.4.

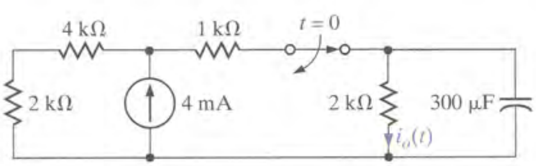


Figure P6.4

6.5 In the network in Fig. P6.5, find $i_o(t)$ for $t > 0$ using the differential equation approach.

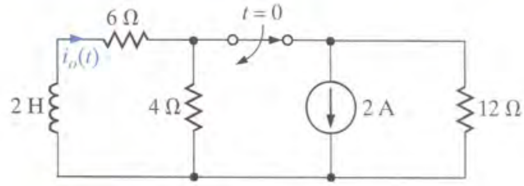


Figure P6.5

6.6 In the circuit in Fig. P6.6, find $i_o(t)$ for $t > 0$ using the differential equation approach.

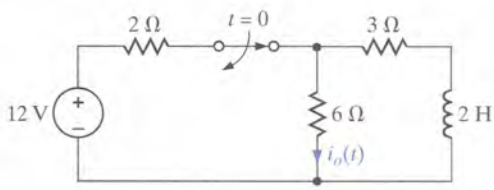


Figure P6.6

6.7 Use the differential equation approach to find $v_o(t)$ for $t > 0$ in the circuit in Fig. P6.7 and plot the response including the time interval just prior to switch action.

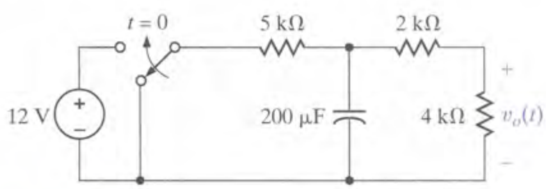


Figure P6.7

6.8 Use the differential equation approach to find $i(t)$ for $t > 0$ in the circuit in Fig. P6.8 and plot the response including the time interval just prior to opening the switch.

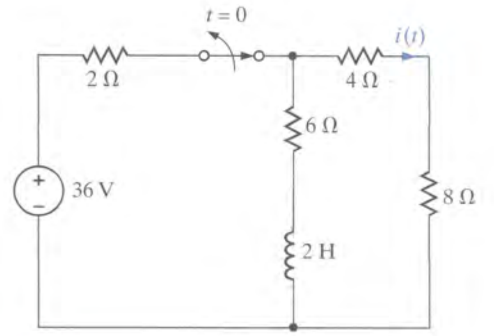


Figure P6.8

6.9 Use the differential equation approach to find $v_C(t)$ for $t > 0$ in the circuit in Fig. P6.9 and plot the response including the time interval just prior to opening the switch.

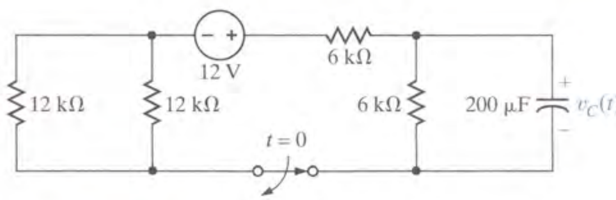


Figure P6.9

6.10 Use the differential equation approach to find $i_L(t)$ for $t > 0$ in the circuit in Fig. P6.10 and plot the response including the time interval just prior to opening the switch.

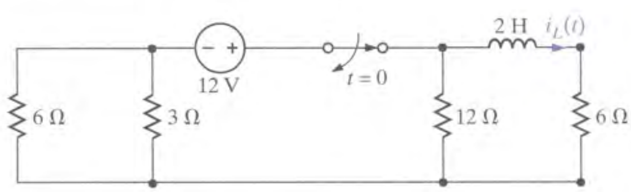


Figure P6.10

6.11 Use the differential equation approach to find $i(t)$ for $t > 0$ in the circuit in Fig. P6.11 and plot the response including the time interval just prior to switch movement.

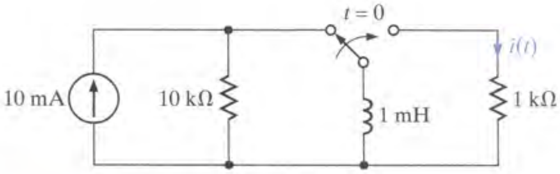


Figure P6.11

6.12 Use the differential equation approach to find $v_C(t)$ for $t > 0$ in the circuit in Fig. P6.12 and plot the response including the time interval just prior to closing the switch.

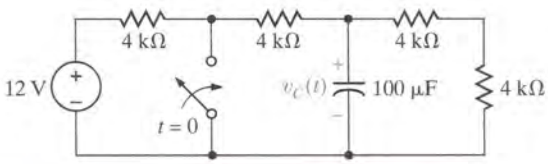


Figure P6.12

6.13 ► **Use the differential equation approach to find $v_o(t)$ for $t > 0$ in the circuit in Fig. P6.13 and plot the response including the time interval just prior to opening the switch.**

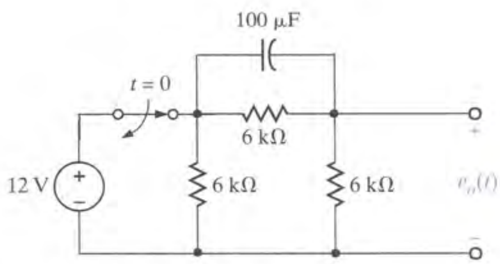


Figure P6.13

6.14 Use the differential equation approach to find $i_o(t)$ for $t > 0$ in the circuit in Fig. P6.14 and plot the response including the time interval just prior to closing the switch.

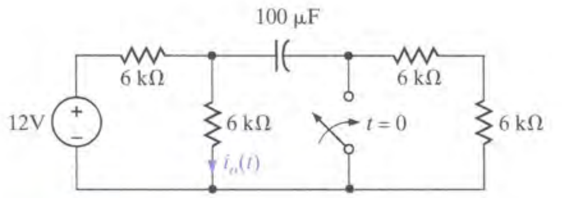


Figure P6.14

6.15 Use the differential equation approach to find $i(t)$ for $t > 0$ in the circuit in Fig. P6.15 and plot the response including the time interval just prior to switch movement.

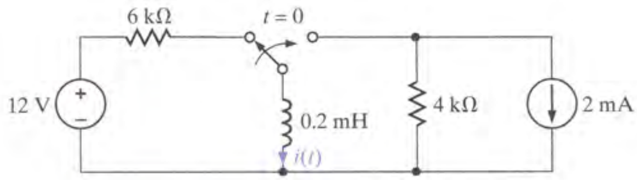


Figure P6.15

6.16 Use the differential equation approach to find $v_o(t)$ for $t > 0$ in the circuit in Fig. P6.16 and plot the response including the time interval just prior to switch action.

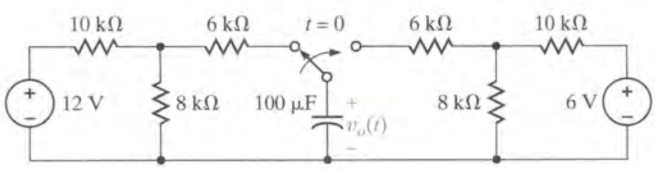


Figure P6.16

6.17 Use the differential equation approach to find $v_o(t)$ for $t > 0$ in the circuit in Fig. P6.17 and plot the response including the time interval just prior to closing the switch.

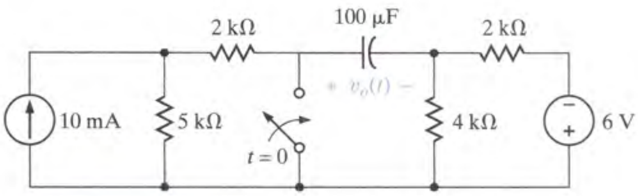


Figure P6.17

6.18 Use the differential equation approach to find $v_o(t)$ for $t > 0$ in the circuit in Fig. P6.18 and plot the response including the time interval just prior to opening the switch.

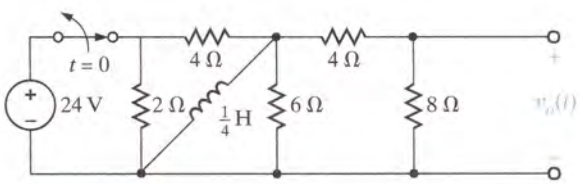


Figure P6.18

6.19 Use the differential equation approach to find $v_o(t)$ for $t > 0$ in the circuit in Fig. P6.19 and plot the response including the time interval just prior to opening the switch.

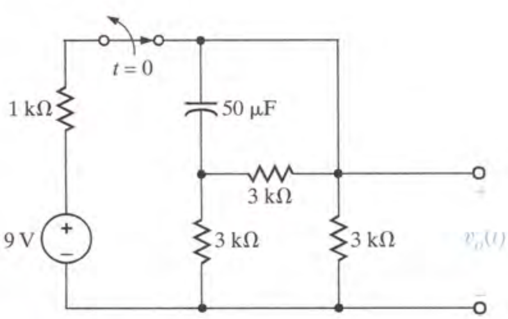


Figure P6.19

6.20 Use the differential equation approach to find $i_o(t)$ for $t > 0$ in the circuit in Fig. P6.20 and plot the response including the time interval just prior to opening the switch.

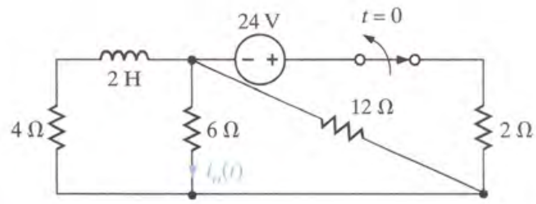


Figure P6.20

6.21 Use the differential equation approach to find $i_o(t)$ for $t > 0$ in the circuit in Fig. P6.21 and plot the response including the time interval just prior to opening the switch.

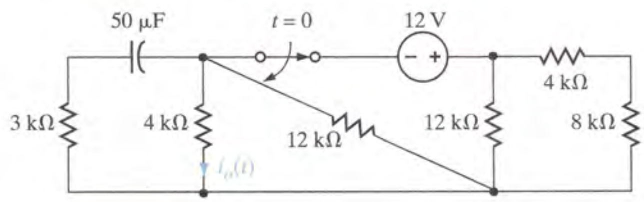


Figure P6.21

6.22 Use the differential equation approach to find $i_o(t)$ for $t > 0$ in the circuit in Fig. P6.22 and plot the response including the time interval just prior to opening the switch.

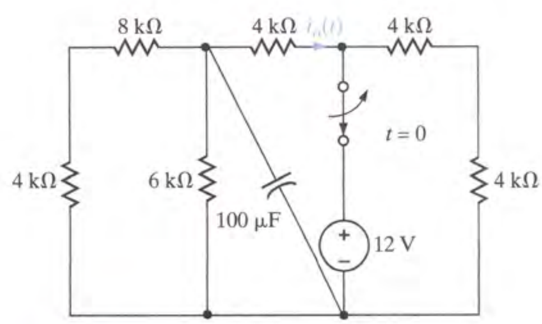


Figure P6.22

6.23 Using the differential equation approach, find $i_o(t)$ for $t > 0$ in the circuit in Fig. P6.23 and plot the response including the time interval just prior to opening the switch.

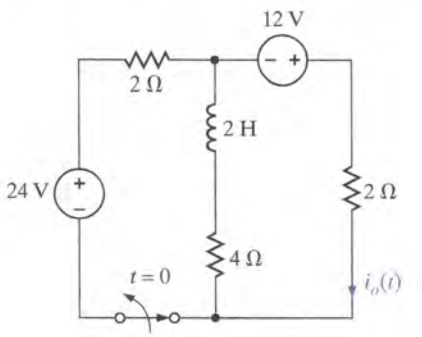


Figure P6.23

6.24 Use the differential equation approach to find $v_o(t)$ for $t > 0$ in the circuit in Fig. P6.24 and plot the response including the time interval just prior to opening the switch.

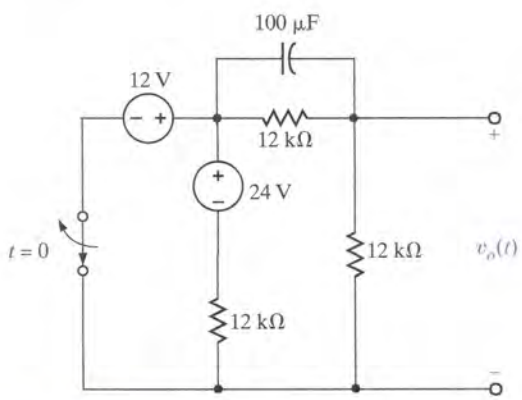


Figure P6.24

6.25 ► **Find $v_C(t)$ for $t > 0$ in the network in Fig. P6.25 using the step-by-step method.**

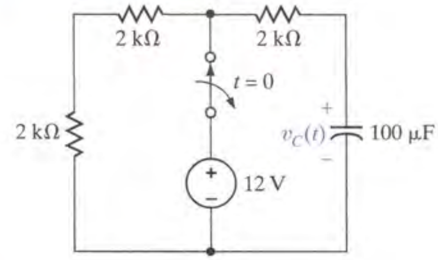


Figure P6.25

6.26 Use the step-by-step method to find $i_o(t)$ for $t > 0$ in the circuit in Fig. P6.26.

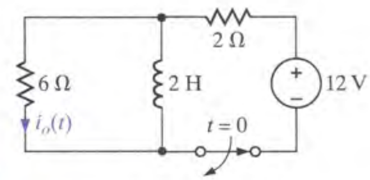


Figure P6.26

6.27 Find $i_o(t)$ for $t > 0$ in the network in Fig. P6.27 using the step-by-step method.

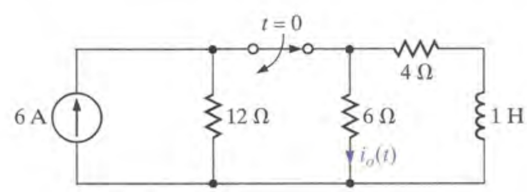


Figure P6.27

6.28 Use the step-by-step method to find $i_o(t)$ for $t > 0$ in the circuit in Fig. P6.28.

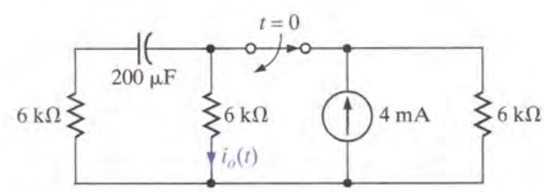


Figure P6.28

6.29 ► **Use the step-by-step technique to find $i_o(t)$ for $t > 0$ in the network in Fig. P6.29.**

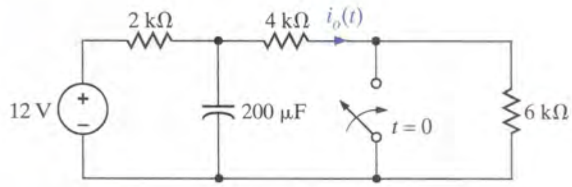


Figure P6.29

6.30 Use the step-by-step method to find $v_o(t)$ for $t > 0$ in the network in Fig. P6.30.

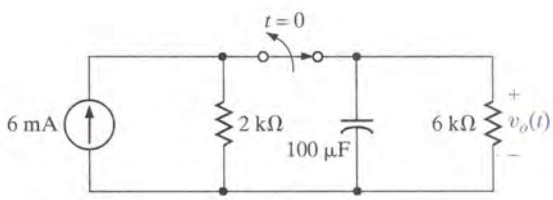


Figure P6.30

6.31 Use the step-by-step method to find $i_o(t)$ for $t > 0$ in the circuit in Fig. P6.31.

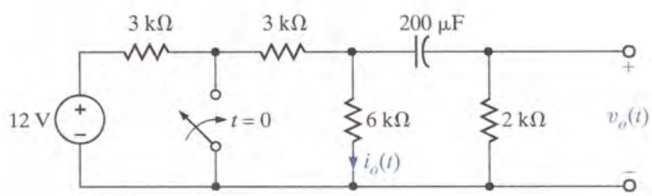


Figure P6.31

6.32 Find $v_o(t)$ for $t > 0$ in the network in Fig. P6.31 using the step-by-step technique.

6.33 ► **Find $i_o(t)$ for $t > 0$ in the network in Fig. P6.33 using the step-by-step method.**

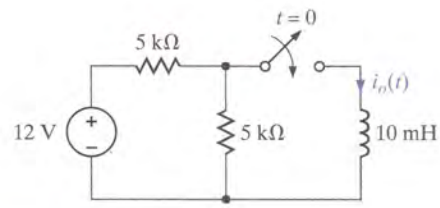


Figure P6.33

6.34 Find $v_o(t)$ for $t > 0$ in the network in Fig. P6.34 using the step-by-step method.

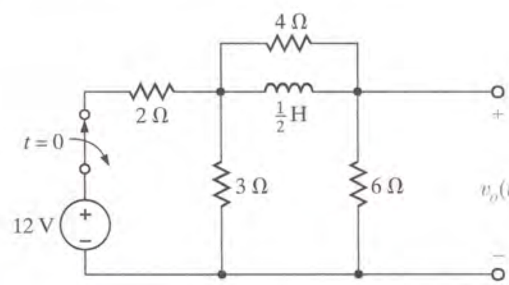


Figure P6.34

6.35 Use the step-by-step method to find $v_o(t)$ for $t > 0$ in the network in Fig. P6.35.

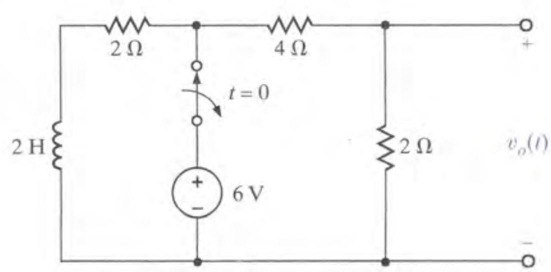


Figure P6.35

6.36 Use the step-by-step technique to find $i_o(t)$ for $t > 0$ in the network in Fig. P6.36.

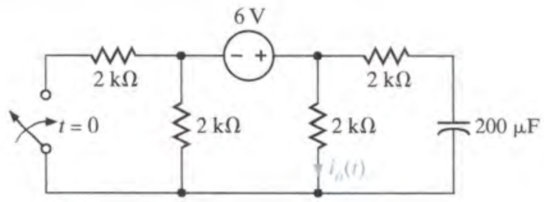


Figure P6.36

6.37 Find $i_o(t)$ for $t > 0$ in the network in Fig. P6.37 using the step-by-step method.

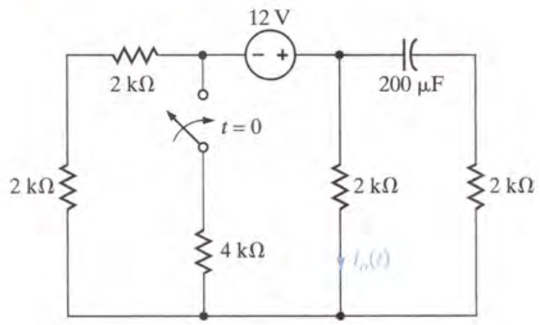


Figure P6.37

6.38 Use the step-by-step method to find $i_o(t)$ for $t > 0$ in the network in Fig. P6.38.

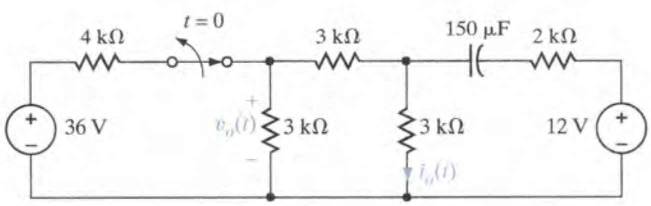


Figure P6.38

6.39 Find $v_o(t)$ for $t > 0$ in the circuit in Fig. P6.38 using the step-by-step method.

6.40 Find $i_o(t)$ for $t > 0$ in the network in Fig. P6.40 using the step-by-step method.

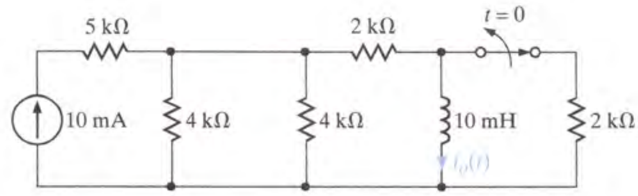


Figure P6.40

6.41 Find $v_o(t)$ for $t > 0$ in the network in Fig. P6.41 using the step-by-step method.

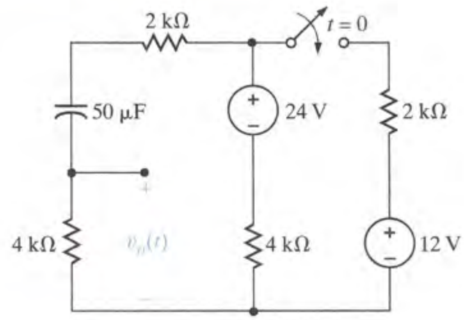


Figure P6.41

6.42 Find $v_o(t)$ for $t > 0$ in the network in Fig. P6.42 using the step-by-step method.

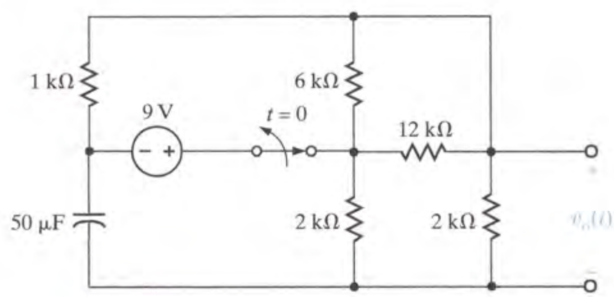


Figure P6.42

6.43 Find $i_o(t)$ for $t > 0$ in the circuit in Fig. P6.43 using the step-by-step technique.

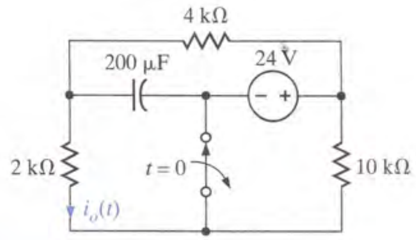


Figure P6.43

6.44 Find $v_o(t)$ for $t > 0$ in the network in Fig. P6.44 using the step-by-step method.

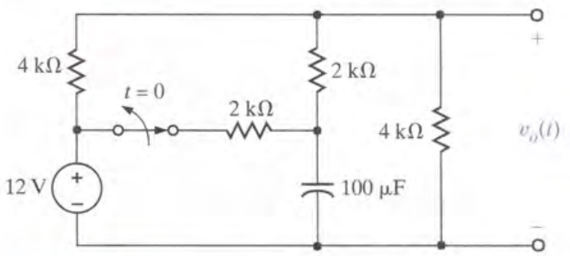


Figure P6.44

6.45 ► **Use the step-by-step method to find $i_o(t)$ for $t > 0$ in the network in Fig. P6.45.**

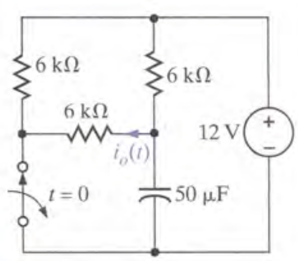


Figure P6.45

6.46 Find $v_o(t)$ for $t > 0$ in the circuit in Fig. P6.46 using the step-by-step method.

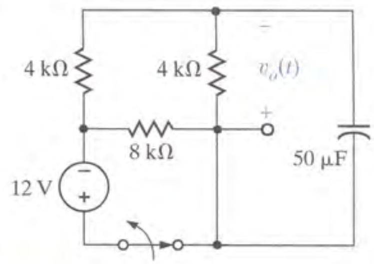


Figure P6.46

6.47 Use the step-by-step technique to find $v_o(t)$ for $t > 0$ in the circuit in Fig. P6.47.

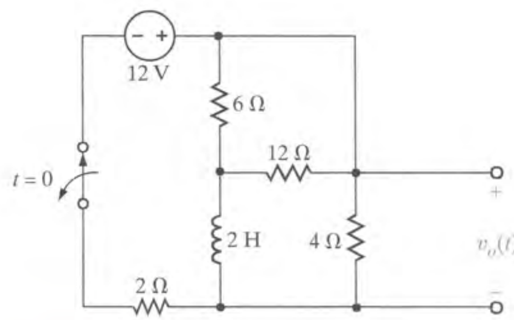


Figure P6.47

6.48 Find $v_o(t)$ for $t > 0$ in the circuit in Fig. P6.48 using the step-by-step method.

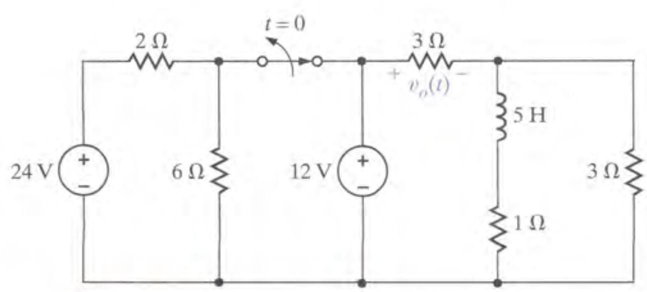


Figure P6.48

6.49 ► **Use the step-by-step method to find $v_o(t)$ for $t > 0$ in the network in Fig. P6.49.**

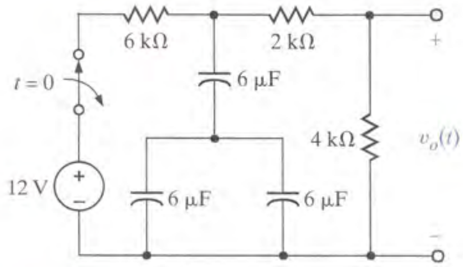


Figure P6.49

6.50 Find $v_o(t)$ for $t > 0$ in the network in Fig. P6.50 using the step-by-step method.

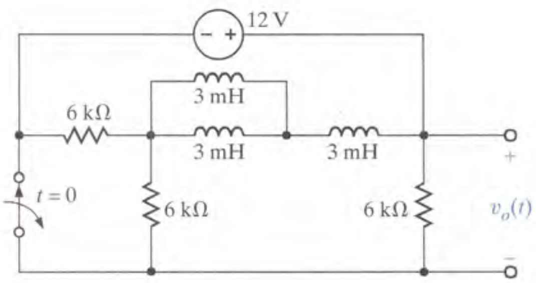


Figure P6.50

6.51 Use the step-by-step technique to find $i_o(t)$ for $t > 0$ in the network in Fig. P6.51.

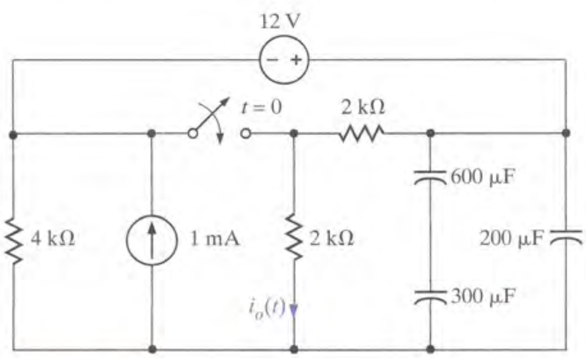


Figure P6.51

6.52 Use the step-by-step method to find $i_o(t)$ for $t > 0$ in the network in Fig. P6.52.

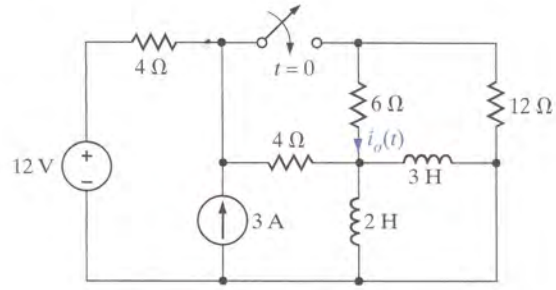


Figure P6.52

6.53 ► **Find $i_o(t)$ for $t > 0$ in the circuit in Fig. P6.53 using the step-by-step method.**

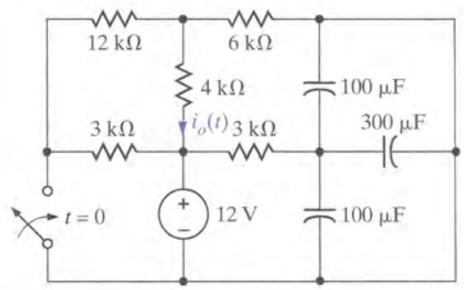


Figure P6.53

6.54 Find $i_o(t)$ for $t > 0$ in the circuit in Fig. P6.54 using the step-by-step technique.

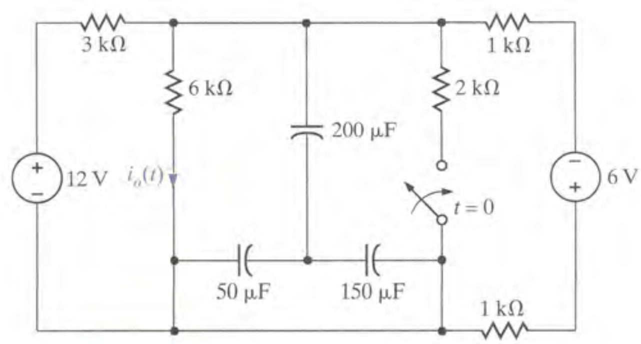


Figure P6.54

SECTION 6.3

6.55 The differential equation that describes the current $i_o(t)$ in a network is

$$\frac{d^2 i_o(t)}{dt^2} + 6\left[\frac{di_o(t)}{dt}\right] + 8i_o(t) = 0$$

Find

(a) the characteristic equation of the network.

(b) the network's natural frequencies.

(c) the expression for $i_o(t)$.

6.56 The terminal current in a network is described by the equation

$$\frac{d^2 i_o(t)}{dt^2} + 10\left[\frac{di_o(t)}{dt}\right] + 25i_o(t) = 0$$

Find

(a) the characteristic equation of the network.

(b) the network's natural frequencies.

(c) the equation for $i_o(t)$.

6.57 **The voltage $v_1(t)$ in a network is defined by the equation**

$$\frac{d^2 v_1(t)}{dt^2} + 2\left[\frac{dv_1(t)}{dt}\right] + 5v_1(t) = 0$$

Find

(a) the characteristic equation of the network.

(b) the circuit's natural frequencies.

(c) the expression for $v_1(t)$.

6.58 The output voltage of a circuit is described by the differential equation

$$\frac{d^2 v_o(t)}{dt^2} + 6\left[\frac{dv_o(t)}{dt}\right] + 10v_o(t) = 0$$

Find

(a) the characteristic equation of the circuit.

(b) the network's natural frequencies.

(c) the equation for $v_o(t)$.

6.59 The parameters for a parallel RLC circuit are $R = 1\ \Omega$, $L = \frac{1}{5}$ H, and $C = \frac{1}{4}$ F. Determine the type of damping exhibited by the circuit.

6.60 A series RLC circuit contains a resistor $R = 2\ \Omega$ and a capacitor $C = \frac{1}{8}$ F. Select the value of the inductor so that the circuit is critically damped.

6.61 **For the underdamped circuit shown in Fig. 6.61, determine the voltage $v(t)$ if the initial conditions on the storage elements are $i_L(0) = 1$ A and $v_C(0) = 10$ V.**

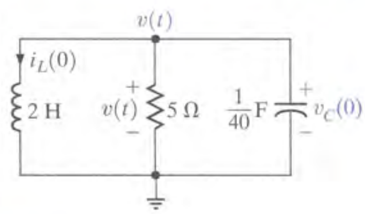


Figure P6.61

6.62 Given the circuit and the initial conditions of Problem 6.61, determine the current through the inductor.

6.63 Find $v_C(t)$ for $t > 0$ in the circuit in Fig. P6.63 if $v_C(0) = 0$.

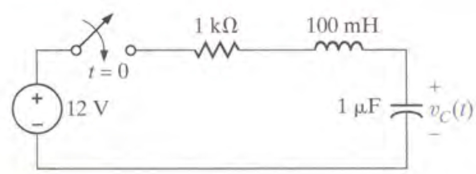


Figure P6.63

6.64 Find $v_o(t)$ for $t > 0$ in the circuit in Fig. P6.64 and plot the response including the time interval just prior to moving the switch.

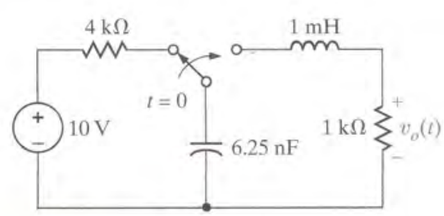


Figure P6.64

6.65 **Find $v_C(t)$ for $t > 0$ in the circuit in Fig. P6.65.**

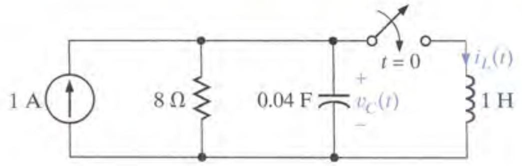


Figure P6.65

6.66 Find $i_L(t)$ for $t > 0$ in the circuit in Fig. P6.65.

6.67 Given the circuit in Fig. P6.67, find the equation for $i(t)$, $t > 0$.

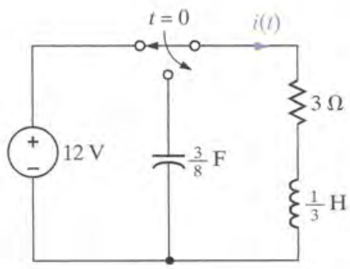


Figure P6.67

6.68 In the circuit shown in Fig. P6.68, find $v(t)$, $t > 0$.

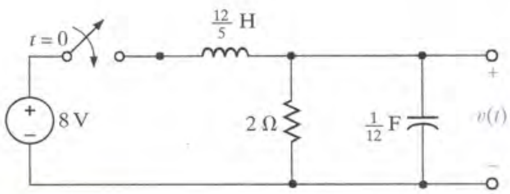


Figure P6.68

6.69 Find $i_o(t)$ for $t > 0$ in the circuit in Fig. P6.69 and plot the response including the time interval just prior to opening the switch.

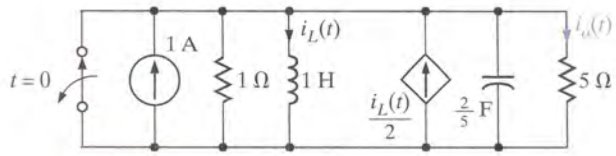


Figure P6.69

6.70 Find $v_o(t)$ for $t > 0$ in the circuit in Fig. P6.70 and plot the response including the time interval just prior to closing the switch.

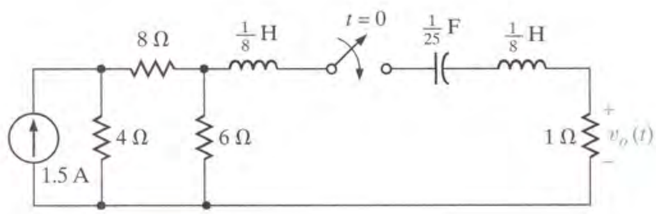


Figure P6.70

6.71 Find $v_o(t)$ for $t > 0$ in the circuit in Fig. P6.71 and plot the response including the time interval just prior to moving the switch.

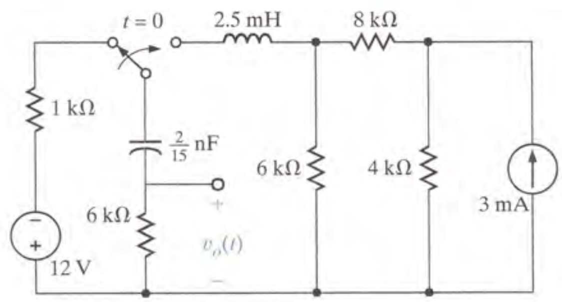


Figure P6.71

SECTION 6.4

6.72 Using the PSPICE *Schematics* editor, draw the circuit in Fig. P6.72, and use the PROBE utility to plot $v_C(t)$ and determine the time constants for $0 < t < 1$ ms and $1 \text{ ms} < t < \infty$. Also, find the maximum voltage on the capacitor.

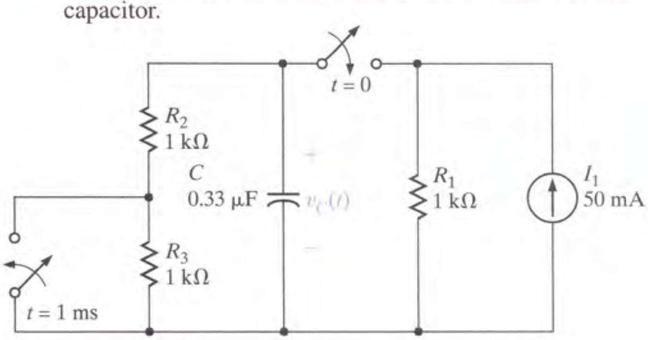


Figure P6.72

6.73 Using the PSPICE *Schematics* editor, draw the circuit in Fig. P6.73, and use the PROBE utility to find the maximum values of $v_L(t)$, $i_C(t)$, and $i(t)$.

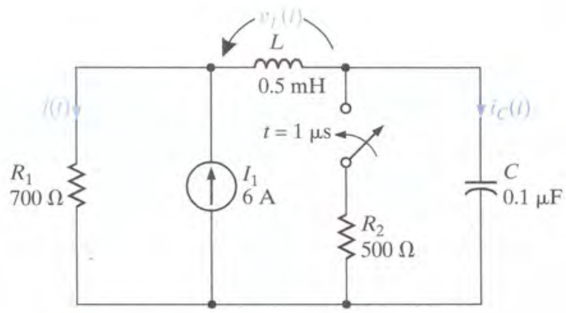


Figure P6.73

SECTION 6.6

6.74 Design a series *RCL* circuit with $R \geqq 1$ kΩ that has the characteristic equation

$$s^2 + 4 \times 10^7 s + 4 \times 10^{14} = 0$$

6.75 Design a parallel *RLC* circuit with $R \geqq 1$ kΩ that has the characteristic equation

$$s^2 + 4 \times 10^7 s + 3 \times 10^{14} = 0$$

Typical Problems Found on the FE Exam

6FE-1 ► **In the circuit in Fig. 6PFE-1, the switch, which has been closed for a long time, opens at $t = 0$. Find the value of the capacitor voltage $v_C(t)$ at $t = 2$ s.**

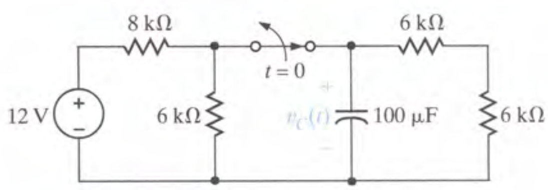


Figure 6PFE-1

6FE-2 In the network in Fig. 6PFE-2, the switch closes at $t = 0$. Find $v_o(t)$ at $t = 1$ s.

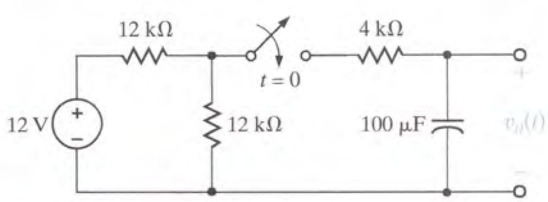


Figure 6PFE-2

6FE-3 ► **Assume that the switch in the network in Fig. 6PFE-3 has been closed for some time. At $t = 0$ the switch opens. Determine the time required for the capacitor voltage to decay to one-half of its initially charged value.**

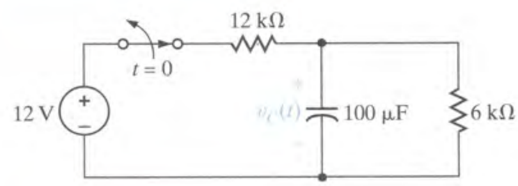


Figure 6PFE-3

7 AC Steady-State Analysis

LEARNING Goals

In the preceding chapters we have considered in some detail both the natural and forced response of a network. We found that the natural response was a characteristic of the network and was independent of the forcing function. The forced response, however, depends directly on the type of forcing function, which until now has generally been a constant. At this point we will diverge from this tack to consider an extremely important excitation: the *sinusoidal forcing function*. Nature is replete with examples of sinusoidal phenomena, and although this is important to us as we examine many different types of physical systems, one reason that we can appreciate at this point for studying this forcing function is that it is the dominant waveform in the electric power industry. The signal present at the ac outlets in our home, office, laboratory, and so on is sinusoidal. In addition, it can be shown that via Fourier analysis we can represent any periodic electrical signal by a sum of sinusoids.

In this chapter we concentrate on the steady-state forced response of networks with sinusoidal driving functions. We will ignore the initial conditions and the transient or natural response, which will eventually vanish for the type of circuits with which we will be dealing. We refer to this as an *ac steady-state analysis*.

Our approach will be to begin by first studying the characteristics of a sinusoidal function as a prelude to using it as a forcing function for a circuit. We will mathematically relate this sinusoidal forcing function to a complex forcing function, which will lead us to define a phasor. By employing phasors we effectively transform a set of differential equations with sinusoidal forcing functions in the time domain into a set of algebraic equations containing complex numbers in the frequency domain. We will show that in this frequency domain Kirchhoff's laws are valid and, thus, all the analysis techniques that we have learned for dc analysis are applicable in ac steady-state analysis. Finally, we demonstrate the power of PSPICE in the solution of ac steady-state circuits.

7.1 Sinusoids

Let us begin our discussion of sinusoidal functions by considering the sine wave

$$x(\omega t) = X_M \sin \omega t \quad \text{7.1}$$

where $x(t)$ could represent either $v(t)$ or $i(t)$. X_M is the *amplitude* or *maximum value*, ω is the *radian* or *angular frequency*, and ωt is the *argument* of the sine function. A plot of the function in Eq. (7.1) as a function of its argument is shown in Fig. 7.1a. Obviously, the function repeats itself every 2π radians. This condition is described mathematically as $x(\omega t + 2\pi) = x(\omega t)$ or in general for period T as

$$x[\omega(t + T)] = x(\omega t) \quad \text{7.2}$$

meaning that the function has the same value at time $t + T$ as it does at time t.

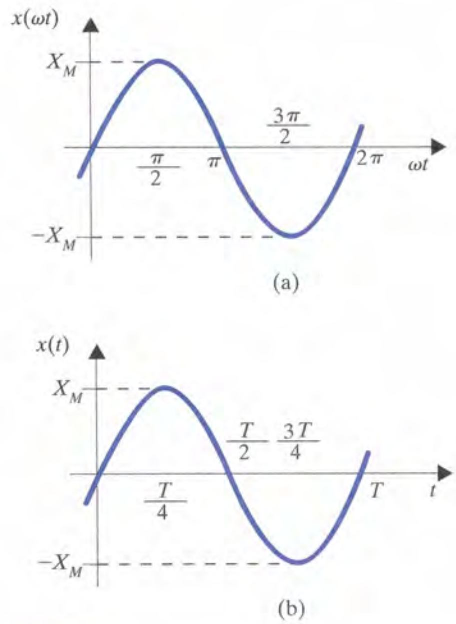


Figure 7.1
Plots of a sine wave as a function of both ωt and t.

LEARNING Hint
The relationship between frequency and period

The waveform can also be plotted as a function of time, as shown in Fig. 7.1b. Note that this function goes through one period every T seconds, or in other words, in 1 second it goes through $1/T$ periods or cycles. The number of cycles per second, called Hertz, is the frequency f, where

$$f = \frac{1}{T} \quad \text{7.3}$$

Now since $\omega T = 2\pi$, as shown in Fig. 7.1a, we find that

$$\omega = \frac{2\pi}{T} = 2\pi f \qquad 7.4$$

LEARNING Hint

The relationship between frequency, period, and radian frequency

which is, of course, the general relationship among period in seconds, frequency in Hertz, and radian frequency.

Now that we have discussed some of the basic properties of a sine wave, let us consider the following general expression for a sinusoidal function:

$$x(t) = X_M \sin(\omega t + \theta) \qquad 7.5$$

LEARNING Hint

Phase lag defined

In this case $(\omega t + \theta)$ is the argument of the sine function, and θ is called the *phase angle*. A plot of this function is shown in Fig. 7.2, together with the original function in Eq. (7.1) for comparison. Because of the presence of the phase angle, any point on the waveform $X_M \sin(\omega t + \theta)$ occurs θ radians earlier in time than the corresponding point on the waveform $X_M \sin \omega t$. Therefore, we say that $X_M \sin \omega t$ *lags* $X_M \sin(\omega t + \theta)$ by θ radians. In the more general situation, if

$$x_1(t) = X_{M_1} \sin(\omega t + \theta)$$

LEARNING Hint

Phase lead graphically illustrated

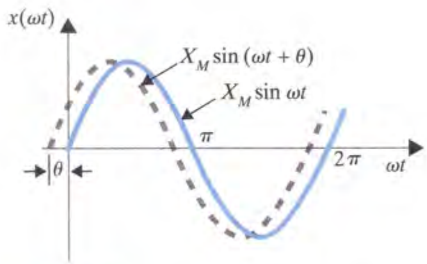


Figure 7.2
Graphical illustration of $X_M \sin(\omega t + \theta)$ leading $X_M \sin \omega t$ by θ radians.

LEARNING Hint

In phase and out of phase defined

and

$$x_2(t) = X_{M_2} \sin(\omega t + \phi)$$

then $x_1(t)$ leads $x_2(t)$ by $\theta - \phi$ radians and $x_2(t)$ lags $x_1(t)$ by $\theta - \phi$ radians. If $\theta = \phi$, the waveforms are identical and the functions are said to be *in phase*. If $\theta \neq \phi$, the functions are *out of phase*.

The phase angle is normally expressed in degrees rather than radians, and therefore we will simply state at this point that we will use the two forms interchangeably; that is,

$$x(t) = X_M \sin\left(\omega t + \frac{\pi}{2}\right) = X_M \sin(\omega t + 90°) \qquad 7.6$$

Rigorously speaking, since ωt is in radians, the phase angle should be also. However, it is common practice and convenient to use degrees for phase, and therefore, that will be our practice in this text.

LEARNING Hint

A very important point

In addition, it should be noted that adding to the argument integer multiples of either 2π radians or 360° does not change the original function. This can easily be shown mathematically but is visibly evident when examining the waveform, as shown in Fig. 7.2.

Although our discussion has centered on the sine function, we could just as easily have used the cosine function, since the two waveforms differ only by a phase angle; that is,

$$\cos \omega t = \sin\left(\omega t + \frac{\pi}{2}\right) \quad \textbf{7.7}$$

$$\sin \omega t = \cos\left(\omega t - \frac{\pi}{2}\right) \quad \textbf{7.8}$$

LEARNING Hint

Some trigonometric identities that are useful in phase angle calculations

It should be noted that when comparing one sinusoidal function with another *of the same frequency* to determine the phase difference, it is necessary to express both functions as either sines or cosines with positive amplitudes. Once in this format, the phase angle between the functions can be computed as outlined previously. Two other trigonometric identities that normally prove useful in phase angle determination are

$$-\cos(\omega t) = \cos(\omega t \pm 180°) \quad \textbf{7.9}$$

$$-\sin(\omega t) = \sin(\omega t \pm 180°) \quad \textbf{7.10}$$

Finally, the angle-sum and angle-difference relationships for sines and cosines may be useful in the manipulation of sinusoidal functions. These relations are

$$\begin{aligned} \sin(\alpha + \beta) &= \sin\alpha\cos\beta + \cos\alpha\sin\beta \\ \cos(\alpha + \beta) &= \cos\alpha\cos\beta - \sin\alpha\sin\beta \\ \sin(\alpha - \beta) &= \sin\alpha\cos\beta - \cos\alpha\sin\beta \\ \cos(\alpha - \beta) &= \cos\alpha\cos\beta + \sin\alpha\sin\beta \end{aligned} \quad \textbf{7.11}$$

LEARNING Example 7.1

We wish to plot the waveforms for the following functions:

(a) $v(t) = 1\cos(\omega t + 45°)$,

(b) $v(t) = 1\cos(\omega t + 225°)$, and

(c) $v(t) = 1\cos(\omega t - 315°)$.

SOLUTION Figure 7.3a shows a plot of the function $v(t) = 1\cos\omega t$. Figure 7.3b is a plot of the function $v(t) = 1\cos(\omega t + 45°)$. Figure 7.3c is a plot of the function $v(t) = 1\cos(\omega t + 225°)$. Note that since

$$v(t) = 1\cos(\omega t + 225°) = 1\cos(\omega t + 45° + 180°)$$

this waveform is 180° out of phase with the waveform in Fig. 7.3b; that is, $\cos(\omega t + 225°) = -\cos(\omega t + 45°)$, and Fig. 7.3c is the negative of Fig. 7.3b. Finally, since the function

$$v(t) = 1\cos(\omega t - 315°) = 1\cos(\omega t - 315° + 360°) = 1\cos(\omega t + 45°)$$

this function is identical to that shown in Fig. 7.3b.

(continued)

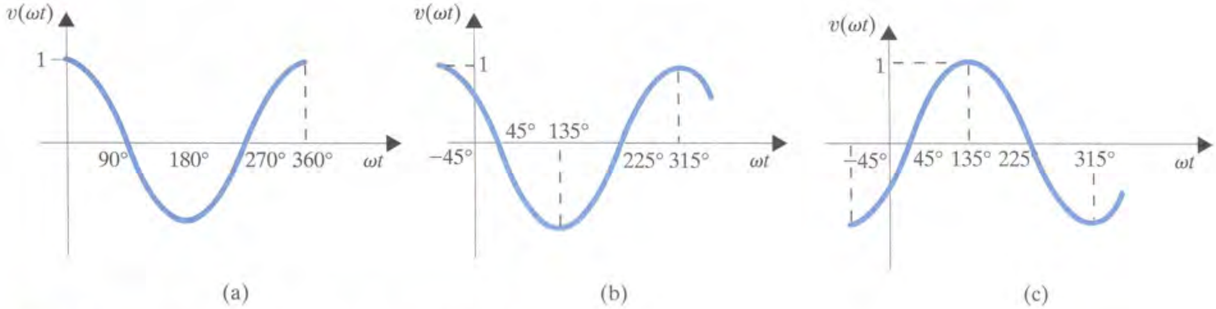


Figure 7.3 Cosine waveforms with various phase angles.

LEARNING Example 7.2

Determine the frequency and the phase angle between the two voltages $v_1(t) = 12\sin(1000t + 60°)$ V and $v_2(t) = -6\cos(1000t + 30°)$ V.

SOLUTION The frequency in Hertz (Hz) is given by the expression

$$f = \frac{\omega}{2\pi} = \frac{1000}{2\pi} = 159.2 \text{ Hz}$$

Using Eq. (7.9), $v_2(t)$ can be written as

$$v_2(t) = -6\cos(\omega t + 30°) = 6\cos(\omega t + 210°) \text{ V}$$

Then employing Eq. (7.7), we obtain

$$6\sin(\omega t + 300°) \text{ V} = 6\sin(\omega t - 60°) \text{ V}$$

Now that both voltages of the same frequency are expressed as sine waves with positive amplitudes, the phase angle between $v_1(t)$ and $v_2(t)$ is $60° - (-60°) = 120°$; that is, $v_1(t)$ leads $v_2(t)$ by 120° or $v_2(t)$ lags $v_1(t)$ by 120°.

LEARNING by Doing

D 7.1 The local power company supplies voltage to the home at a radian frequency of $\omega = 377$ r/s. What is the frequency in Hertz, or equivalently, cycles per second?

ANSWER *60 Hz*

LEARNING EXTENSIONS

E7.1 Given the voltage $v(t) = 120\cos(314t + \pi/4)$ V, determine the frequency of the voltage in Hertz and the phase angle in degrees.

ANSWER $f = 50$ Hz, $\theta = 45°$.

E7.2 Three branch currents in a network are known to be

$$i_1(t) = 2\sin(377t + 45°) \text{ A}$$
$$i_2(t) = 0.5\cos(377t + 10°) \text{ A}$$
$$i_3(t) = -0.25\sin(377t + 60°) \text{ A}$$

Determine the phase angles by which $i_1(t)$ leads $i_2(t)$ and $i_1(t)$ leads $i_3(t)$.

ANSWER i_1 leads i_2 by −55°, i_1 leads i_3 by 165°.

7.2 Sinusoidal and Complex Forcing Functions

In the preceding chapters we applied a constant forcing function to a network and found that the steady-state response was also constant.

In a similar manner, if we apply a sinusoidal forcing function to a linear network, the steady-state voltages and currents in the network will also be sinusoidal. This should also be clear from the KVL and KCL equations. For example, if one branch voltage is a sinusoid of some frequency, the other branch voltages must be sinusoids of the same frequency if KVL is to apply around any closed path. This means, of course, that the forced solutions of the differential equations that describe a network with a sinusoidal forcing function are sinusoidal functions of time. For example, if we assume that our input function is a voltage $v(t)$ and our output response is a current $i(t)$ as shown in Fig. 7.4, then if $v(t) = A\sin(\omega t + \theta)$, $i(t)$ will be of the form $i(t) = B\sin(\omega t + \phi)$. The critical point here is that we know the form of the output response, and therefore the solution involves simply determining the values of the two parameters B and ϕ.

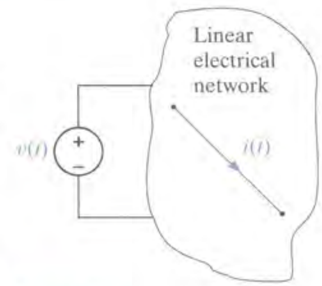


Figure 7.4
Current response to an applied voltage in an electrical network.

LEARNING Example 7.3

Consider the circuit in Fig. 7.5. Let us derive the expression for the current.

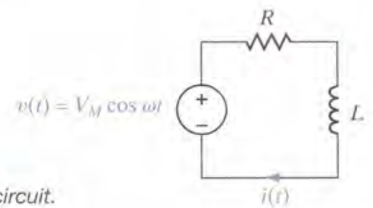


Figure 7.5
A simple RL circuit.

SOLUTION The KVL equation for this circuit is

$$L\frac{di(t)}{dt} + Ri(t) = V_M \cos\omega t$$

Since the input forcing function is $V_M \cos\omega t$, we assume that the forced response component of the current $i(t)$ is of the form

$$i(t) = A\cos(\omega t + \phi)$$

which can be written using Eq. (7.11) as

$$\begin{aligned} i(t) &= A\cos\phi\cos\omega t - A\sin\phi\sin\omega t \\ &= A_1\cos\omega t + A_2\sin\omega t \end{aligned}$$

Note that this is, as we observed in Chapter 6, of the form of the forcing function $\cos\omega t$ and its derivative $\sin\omega t$. Substituting this form for $i(t)$ into the preceding differential equation yields

$$L\frac{d}{dt}(A_1\cos\omega t + A_2\sin\omega t) + R(A_1\cos\omega t + A_2\sin\omega t) = V_M\cos\omega t$$

Evaluating the indicated derivative produces

$$-A_1\omega L\sin\omega t + A_2\omega L\cos\omega t + RA_1\cos\omega t + RA_2\sin\omega t = V_M\cos\omega t$$

By equating coefficients of the sine and cosine functions, we obtain

$$-A_1\omega L + A_2R = 0$$
$$A_1R + A_2\omega L = V_M$$

that is, two simultaneous equations in the unknowns A_1 and A_2. Solving these two equations for A_1 and A_2 yields

$$A_1 = \frac{RV_M}{R^2 + \omega^2L^2}$$
$$A_2 = \frac{\omega LV_M}{R^2 + \omega^2L^2}$$

Therefore,

$$i(t) = \frac{RV_M}{R^2 + \omega^2L^2}\cos\omega t + \frac{\omega LV_M}{R^2 + \omega^2L^2}\sin\omega t$$

which, using the last identity in Eq. (7.11), can be written as

$$i(t) = A\cos(\omega t + \phi)$$

(continued)

where A and ϕ are determined as follows:

$$A\cos\phi = \frac{RV_M}{R^2 + \omega^2L^2}$$

$$A\sin\phi = \frac{-\omega LV_M}{R^2 + \omega^2L^2}$$

Hence,

$$\tan\phi = \frac{A\sin\phi}{A\cos\phi} = -\frac{\omega L}{R}$$

and therefore,

$$\phi = -\tan^{-1}\frac{\omega L}{R}$$

and since

$$(A\cos\phi)^2 + (A\sin\phi)^2 = A^2(\cos^2\phi + \sin^2\phi) = A^2$$

$$A^2 = \frac{R^2V_M^2}{(R^2 + \omega^2L^2)^2} + \frac{(\omega L)^2V_M^2}{(R^2 + \omega^2L^2)^2} = \frac{V_M^2}{R^2 + \omega^2L^2}$$

$$A = \frac{V_M}{\sqrt{R^2 + \omega^2L^2}}$$

Hence, the final expression for $i(t)$ is

$$i(t) = \frac{V_M}{\sqrt{R^2 + \omega^2L^2}}\cos\left(\omega t - \tan^{-1}\frac{\omega L}{R}\right)$$

The preceding analysis indicates that ϕ is zero if $L = 0$ and hence $i(t)$ is in phase with $v(t)$. If $R = 0$, $\phi = -90°$, and the current lags the voltage by 90°. If L and R are both present, the current lags the voltage by some angle between 0° and 90°.

This example illustrates an important point—solving even a simple one-loop circuit containing one resistor and one inductor is very complicated when compared to the solution of a single-loop circuit containing only two resistors. Imagine for a moment how laborious it would be to solve a more complicated circuit using the procedure we have employed in Example 7.3. To circumvent this approach, we will establish a correspondence between sinusoidal time functions and complex numbers. We will then show that this relationship leads to a set of algebraic equations for currents and voltages in a network (e.g., loop currents or node voltages) in which the coefficients of the variables are complex numbers. Hence, once again we will find that determining the currents or voltages in a circuit can be accomplished by solving a set of algebraic equations; however, in this case, their solution is complicated by the fact that variables in the equations have complex, rather than real, coefficients.

The vehicle we will employ to establish a relationship between time-varying sinusoidal functions and complex numbers is Euler's equation, which for our purposes is written as

$$e^{j\omega t} = \cos\omega t + j\sin\omega t \qquad \textbf{7.12}$$

This complex function has a real part and an imaginary part:

$$\text{Re}(e^{j\omega t}) = \cos\omega t$$
$$\text{Im}(e^{j\omega t}) = \sin\omega t \qquad \textbf{7.13}$$

where Re(·) and Im(·) represent the real part and the imaginary part, respectively, of the function in the parentheses.

Now suppose that we select as our forcing function in Fig. 7.4 the nonrealizable voltage

$$v(t) = V_M e^{j\omega t} \tag{7.14}$$

which because of Euler's identity can be written as

$$v(t) = V_M \cos \omega t + jV_M \sin \omega t \tag{7.15}$$

The real and imaginary parts of this function are each realizable. We think of this complex forcing function as two forcing functions, a real one and an imaginary one, and as a consequence of linearity, the principle of superposition applies and thus the current response can be written as

$$i(t) = I_M \cos(\omega t + \phi) + jI_M \sin(\omega t + \phi) \tag{7.16}$$

where $I_M \cos(\omega t + \phi)$ is the response due to $V_M \cos \omega t$ and $jI_M \sin(\omega t + \phi)$ is the response due to $jV_M \sin \omega t$. This expression for the current containing both a real and an imaginary term can be written via Euler's equation as

$$i(t) = I_M e^{j(\omega t + \phi)} \tag{7.17}$$

Because of the preceding relationships we find that rather than applying the forcing function $V_M \cos \omega t$ and calculating the response $I_M \cos(\omega t + \phi)$, we can apply the complex forcing function $V_M e^{j\omega t}$ and calculate the response $I_M e^{j(\omega t + \phi)}$, the real part of which is the desired response $I_M \cos(\omega t + \phi)$. Although this procedure may initially appear to be more complicated, it is not. It is via this technique that we will convert the differential equation to an algebraic equation that is much easier to solve.

LEARNING Example 7.4

Once again, let us determine the current in the RL circuit examined in Example 7.3. However, rather than applying $V_M \cos \omega t$ we will apply $V_M e^{j\omega t}$.

SOLUTION The forced response will be of the form

$$i(t) = I_M e^{j(\omega t + \phi)}$$

where only I_M and ϕ are unknown. Substituting $v(t)$ and $i(t)$ into the differential equation for the circuit, we obtain

$$RI_M e^{j(\omega t + \phi)} + L\frac{d}{dt}\left(I_M e^{j(\omega t + \phi)}\right) = V_M e^{j\omega t}$$

Taking the indicated derivative, we obtain

$$RI_M e^{j(\omega t + \phi)} + j\omega L I_M e^{j(\omega t + \phi)} = V_M e^{j\omega t}$$

Dividing each term of the equation by the common factor $e^{j\omega t}$ yields

$$RI_M e^{j\phi} + j\omega L I_M e^{j\phi} = V_M$$

which is an algebraic equation with complex coefficients. This equation can be written as

$$I_M e^{j\phi} = \frac{V_M}{R + j\omega L}$$

Converting the right-hand side of the equation to exponential or polar form produces the equation

$$I_M e^{j\phi} = \frac{V_M}{\sqrt{R^2 + \omega^2 L^2}} e^{j[-\tan^{-1}(\omega L/R)]}$$

(A quick refresher on complex numbers is given in the Appendix for readers who need to sharpen their skills in this area.) The

(continued)

preceding form clearly indicates that the magnitude and phase of the resulting current are

$$I_M = \frac{V_M}{\sqrt{R^2 + \omega^2 L^2}}$$

and

$$\phi = -\tan^{-1}\frac{\omega L}{R}$$

However, since our actual forcing function was $V_M \cos \omega t$ rather than $V_M e^{j\omega t}$, our actual response is the real part of the complex response:

$$\begin{aligned} i(t) &= I_M \cos(\omega t + \phi) \\ &= \frac{V_M}{\sqrt{R^2 + \omega^2 L^2}} \cos\left(\omega t - \tan^{-1}\frac{\omega L}{R}\right) \end{aligned}$$

Note that this is identical to the response obtained in the previous example by solving the differential equation for the current $i(t)$.

LEARNING Hint

Summary of complex number relationships:

$$x + jy = re^{j\theta}$$
$$r = \sqrt{x^2 + y^2}$$
$$\theta = \tan^{-1}\frac{y}{x}$$
$$x = r\cos\theta$$
$$y = r\sin\theta$$
$$\frac{1}{e^{j\theta}} = e^{-j\theta}$$

7.3 Phasors

Once again let us assume that the forcing function for a linear network is of the form

$$v(t) = V_M e^{j\omega t}$$

Then every steady-state voltage or current in the network will have the same form and the same frequency ω; for example, a current $i(t)$ will be of the form $i(t) = I_M e^{j(\omega t + \phi)}$.

As we proceed in our subsequent circuit analyses, we will simply note the frequency and then drop the factor $e^{j\omega t}$ since it is common to every term in the describing equations. Dropping the term $e^{j\omega t}$ indicates that every voltage or current can be fully described by a magnitude and phase. For example, a voltage $v(t)$ can be written in exponential form as

$$v(t) = V_M \cos(\omega t + \theta) = \text{Re}\left[V_M e^{j(\omega t + \theta)}\right] \qquad \textbf{7.18}$$

or as a complex number

$$v(t) = \text{Re}\left(V_M \angle \theta \, e^{j\omega t}\right) \qquad \textbf{7.19}$$

LEARNING Hint

If $v(t) = V_M \cos(\omega t + \theta)$ and $i(t) = I_M \cos(\omega t + \phi)$, then in phasor notation

$$\mathbf{V} = V_M \angle \theta$$

and

$$\mathbf{I} = I_M \angle \phi$$

Since we are working with a complex forcing function, the real part of which is the desired answer, and each term in the equation will contain $e^{j\omega t}$, we can drop Re(·) and $e^{j\omega t}$ and work only with the complex number $V_M \angle \theta$. This complex representation is commonly called a *phasor*. As a distinguishing feature, phasors will be written in boldface type. In a completely identical manner a voltage $v(t) = V_M \cos(\omega t + \theta) = \text{Re}\left[V_M e^{j(\omega t + \theta)}\right]$ and a current $i(t) = I_M \cos(\omega t + \phi) = \text{Re}\left[I_M e^{j(\omega t + \phi)}\right]$ are written in phasor notation as $\mathbf{V} = V_M \angle \theta$ and $\mathbf{I} = I_M \angle \phi$, respectively.

LEARNING Example 7.5

Again, we consider the RL circuit in Example 7.3. Let us use phasors to determine the expression for the current.

SOLUTION The differential equation is

$$L\frac{di(t)}{dt} + Ri(t) = V_M \cos \omega t$$

The forcing function can be replaced by a complex forcing function that is written as $\mathbf{V}e^{j\omega t}$ with phasor $\mathbf{V} = V_M \angle 0°$. Similarly, the forced response component of the current $i(t)$ can be replaced by a complex function that is written as $\mathbf{I}e^{j\omega t}$ with phasor $\mathbf{I} = I_M \angle \phi$. From our previous discussions we recall that the solution of the differential equation is the real part of this current.

Using the complex forcing function, we find that the differential equation becomes

$$L\frac{d}{dt}(\mathbf{I}e^{j\omega t}) + R\mathbf{I}e^{j\omega t} = \mathbf{V}e^{j\omega t}$$

$$j\omega L\mathbf{I}e^{j\omega t} + R\mathbf{I}e^{j\omega t} = \mathbf{V}e^{j\omega t}$$

Note that $e^{j\omega t}$ is a common factor and, as we have already indicated, can be eliminated, leaving the phasors; that is,

$$j\omega L\mathbf{I} + R\mathbf{I} = \mathbf{V}$$

Therefore,

$$\mathbf{I} = \frac{\mathbf{V}}{R + j\omega L} = I_M \angle \phi = \frac{V_M}{\sqrt{R^2 + \omega^2 L^2}} \angle -\tan^{-1}\frac{\omega L}{R}$$

Thus,

$$i(t) = \frac{V_M}{\sqrt{R^2 + \omega^2 L^2}} \cos\left(\omega t - \tan^{-1}\frac{\omega L}{R}\right)$$

which once again is the function we obtained earlier.

LEARNING Hint

The differential equation is reduced to a phasor equation.

We define relations between phasors after the $e^{j\omega t}$ term has been eliminated as "phasor, or frequency domain, analysis." Thus we have transformed a set of differential equations with sinusoidal forcing functions in the time domain into a set of algebraic equations containing complex numbers in the frequency domain. In effect, we are now faced with solving a set of algebraic equations for the unknown phasors. The phasors are then simply transformed back to the time domain to yield the solution of the original set of differential equations. In addition, we note that the solution of sinusoidal steady-state circuits would be relatively simple if we could write the phasor equation directly from the circuit description. In Section 7.4 we will lay the groundwork for doing just that.

Note that in our discussions we have tacitly assumed that sinusoidal functions would be represented as phasors with a phase angle based on a cosine function. Therefore, if sine functions are used, we will simply employ the relationship in Eq. (7.7) to obtain the proper phase angle.

In summary, while $v(t)$ represents a voltage in the time domain, the phasor $\mathbf{V}$ represents the voltage in the frequency domain. The phasor contains only magnitude and phase information, and the frequency is implicit in this representation. The transformation from the time domain to the frequency domain, as well as the reverse transformation, is shown in Table 7.1.

LEARNING Hint

Phasor analysis

1. Using phasors, transform a set of differential equations in the time domain into a set of algebraic equations in the frequency domain.
2. Solve the algebraic equations for the unknown phasors.
3. Transform the now-known phasors back to the time domain.

Table 7.1 Phasor representation

Time Domain	Frequency Domain
$A\cos(\omega t \pm \theta)$	$A \angle \pm\theta$
$A\sin(\omega t \pm \theta)$	$A \angle \pm\theta - 90°$

Recall that the phase angle is based on a cosine function and, therefore, if a sine function is involved, a 90° shift factor must be employed, as shown in the table.

The following examples illustrate the use of the phasor transformation.

It is important to note, however, that if a network contains only sine sources, there is no need to perform the 90° shift. We simply perform the normal phasor analysis and then the *imaginary* part of the time-varying complex solution is the desired response. Simply put, cosine sources generate a cosine response and sine sources generate a sine response.

LEARNING EXTENSIONS

E7.3 Convert the following voltage functions to phasors.

$$v_1(t) = 12\cos(377t - 425^\circ)\text{ V}$$

$$v_2(t) = 18\sin(2513t + 4.2^\circ)\text{ V}$$

ANSWER $\mathbf{V}_1 = 12\angle -425^\circ$ V, $\mathbf{V}_2 = 18\angle -85.8^\circ$ V.

E7.4 Convert the following phasors to the time domain if the frequency is 400 Hz.

$$\mathbf{V}_1 = 10\angle 20^\circ\text{ V}$$

$$\mathbf{V}_2 = 12\angle -60^\circ\text{ V}$$

ANSWER $v_1(t) = 10\cos(800\pi t + 20^\circ)$ V, $v_2(t) = 12\cos(800\pi t - 60^\circ)$ V.

7.4 Phasor Relationships for Circuit Elements

LEARNING Hint

Current and voltage are in phase.

As we proceed in our development of the techniques required to analyze circuits in the sinusoidal steady-state, we are now in a position to establish the phasor relationships between voltage and current for the three passive elements *R*, *L*, and *C*.

In the case of a resistor as shown in Fig. 7.6a, the voltage–current relationship is known to be

$$v(t) = Ri(t) \qquad 7.20$$

Applying the complex voltage $V_M e^{j(\omega t+\theta_v)}$ results in the complex current $I_M e^{j(\omega t+\theta_i)}$, and therefore Eq. (7.20) becomes

$$V_M e^{j(\omega t+\theta_v)} = RI_M e^{j(\omega t+\theta_i)}$$

which reduces to

$$V_M e^{j\theta_v} = RI_M e^{j\theta_i} \qquad 7.21$$

Equation (7.21) can be written in phasor form as

$$\mathbf{V} = R\mathbf{I} \qquad 7.22$$

where

$$\mathbf{V} = V_M e^{j\theta_v} = V_M\angle\theta_v \quad\text{and}\quad \mathbf{I} = I_M e^{j\theta_i} = I_M\angle\theta_i$$

From Eq. (7.21) we see that $\theta_v = \theta_i$ and thus the current and voltage for this circuit are *in phase*.

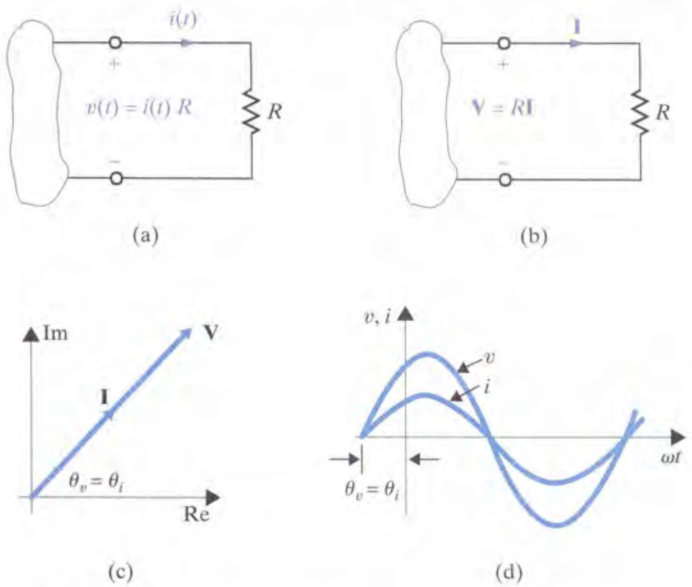


Figure 7.6
Voltage–current relationships for a resistor.

Historically, complex numbers have been represented as points on a graph in which the x-axis represents the real axis and the y-axis the imaginary axis. The line segment connecting the origin with the point provides a convenient representation of the magnitude and angle when the complex number is written in a polar form. A review of the Appendix will indicate how these complex numbers or line segments can be added, subtracted, and so on. Since phasors are complex numbers, it is convenient to represent the phasor voltage and current graphically as line segments. A plot of the line segments representing the phasors is called a *phasor diagram*. This pictorial representation of phasors provides immediate information on the relative magnitude of one phasor with another, the angle between two phasors, and the relative position of one phasor with respect to another (i.e., leading or lagging). A phasor diagram and the sinusoidal waveforms for the resistor are shown in Figs. 7.6c and d, respectively. A phasor diagram will be drawn for each of the other circuit elements in the remainder of this section.

LEARNING Example 7.6

If the voltage $v(t) = 24\cos(377t + 75°)$ V is applied to a 6-Ω resistor as shown in Fig. 7.6a, we wish to determine the resultant current.

SOLUTION Since the phasor voltage is

$$\mathbf{V} = 24\underline{/75°} \text{ V}$$

the phasor current from Eq. (7.22) is

$$\mathbf{I} = \frac{24\underline{/75°}}{6} = 4\underline{/75°} \text{ A}$$

which in the time domain is

$$i(t) = 4\cos(377t + 75°) \text{ A}$$

LEARNING EXTENSION

E7.5 The current in a 4-Ω resistor is known to be $\mathbf{I} = 12\angle 60°$ A. Express the voltage across the resistor as a time function if the frequency of the current is 4 kHz.

ANSWER
$v(t) = 48\cos(8000\pi t + 60°)$ V.

The voltage–current relationship for an inductor, as shown in Fig. 7.7a, is

$$v(t) = L\frac{di(t)}{dt} \qquad 7.23$$

Substituting the complex voltage and current into this equation yields

$$V_M e^{j(\omega t + \theta_v)} = L\frac{d}{dt} I_M e^{j(\omega t + \theta_i)}$$

which reduces to

$$V_M e^{j\theta_v} = j\omega L I_M e^{j\theta_i} \qquad 7.24$$

LEARNING Hint

The derivative process yields a frequency-dependent impedance.

Equation (7.24) in phasor notation is

$$\mathbf{V} = j\omega L\mathbf{I} \qquad 7.25$$

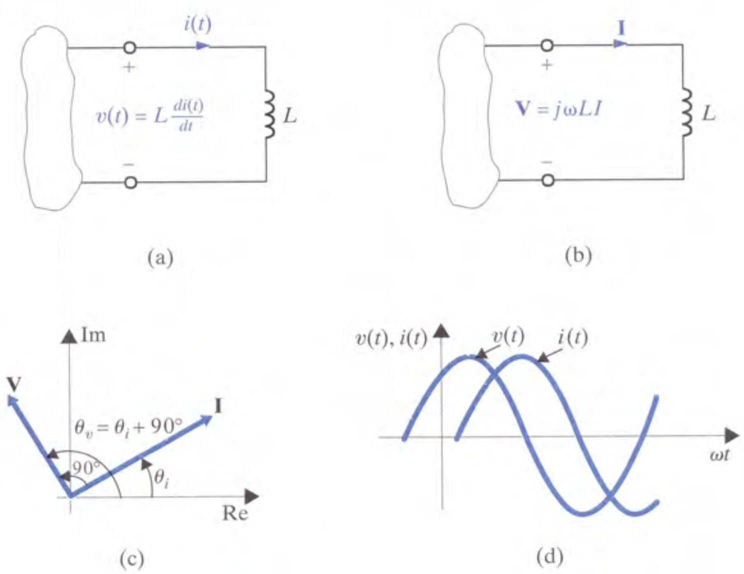


Figure 7.7
Voltage–current relationships for an inductor.

Note that the differential equation in the time domain (7.23) has been converted to an algebraic equation with complex coefficients in the frequency domain. This relationship is shown in Fig. 7.7b. Since the imaginary operator $j = 1e^{j90^\circ} = 1\underline{/90^\circ} = \sqrt{-1}$, Eq. (7.24) can be written as

$$V_M e^{j\theta_v} = \omega L I_M e^{j(\theta_i + 90^\circ)} \quad 7.26$$

Therefore, the voltage and current are *90° out of phase*, and in particular the voltage leads the current by 90° or the current lags the voltage by 90°. The phasor diagram and the sinusoidal waveforms for the inductor circuit are shown in Figs. 7.7c and d, respectively.

LEARNING Hint

The voltage leads the current or the current lags the voltage.

LEARNING Example 7.7

The voltage $v(t) = 12\cos(377t + 20^\circ)$ V is applied to a 20-mH inductor as shown in Fig. 7.7a. Find the resultant current.

SOLUTION The phasor current is

$$\mathbf{I} = \frac{\mathbf{V}}{j\omega L} = \frac{12\underline{/20^\circ}}{\omega L\underline{/90^\circ}}$$

$$= \frac{12\underline{/20^\circ}}{(377)(20 \times 10^{-3})\underline{/90^\circ}}$$

$$= 1.59\underline{/-70^\circ} \text{ A}$$

or

$$i(t) = 1.59\cos(377t - 70^\circ) \text{ A}$$

LEARNING Hint

Applying $\mathbf{V} = j\omega L\mathbf{I}$

$$\frac{x_1\underline{/\theta_1}}{x_2\underline{/\theta_2}} = \frac{x_1}{x_2}\underline{/\theta_1 - \theta_2}$$

LEARNING EXTENSION

E7.6 The current in a 0.05-H inductor is $\mathbf{I} = 4\underline{/-30^\circ}$ A. If the frequency of the current is 60 Hz, determine the voltage across the inductor.

ANSWER $v_L(t) = 75.4\cos(377t + 60^\circ)$ V.

The voltage–current relationship for our last passive element, the capacitor, as shown in Fig. 7.8a, is

$$i(t) = C\frac{dv(t)}{dt} \quad 7.27$$

LEARNING Hint

The current leads the voltage or the voltage lags the current.

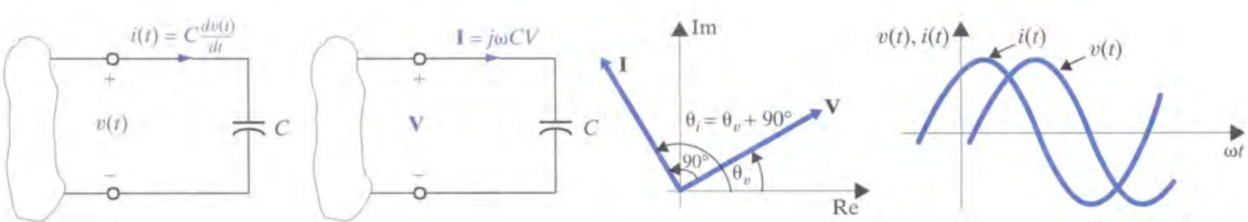


Figure 7.8 Voltage–current relationships for a capacitor.

Once again employing the complex voltage and current, we obtain

$$I_M e^{j(\omega t+\theta_i)} = C \frac{d}{dt} V_M e^{j(\omega t+\theta_v)}$$

LEARNING Hint

The derivative process yields a frequency-dependent impedance.

which reduces to

$$I_M e^{j\theta_i} = j\omega C V_M e^{j\theta_v} \tag{7.28}$$

In phasor notation this equation becomes

$$\mathbf{I} = j\omega C \mathbf{V} \tag{7.29}$$

Equation (7.27), a differential equation in the time domain, has been transformed into Eq. (7.29), an algebraic equation with complex coefficients in the frequency domain. The phasor relationship is shown in Fig. 7.8b. Substituting $j = 1e^{j90°}$ into Eq. (7.28) yields

$$I_M e^{j\theta_i} = \omega C V_M e^{j(\theta_v + 90°)} \tag{7.30}$$

Note that the voltage and current are *90° out of phase*. Equation (7.30) states that the current leads the voltage by 90° or the voltage lags the current by 90°. The phasor diagram and the sinusoidal waveforms for the capacitor circuit are shown in Figs. 7.8c and d, respectively.

LEARNING Example 7.8

The voltage $v(t) = 100\cos(314t + 15°)$ V is applied to a 100-μF capacitor as shown in Fig. 7.8a. Find the current.

SOLUTION The resultant phasor current is

$$\begin{aligned}\mathbf{I} &= j\omega C(100\,\underline{/15°})\\ &= (314)(100 \times 10^{-6}\,\underline{/90°})(100\,\underline{/15°})\\ &= 3.14\,\underline{/105°}\text{ A}\end{aligned}$$

Therefore, the current written as a time function is

$$i(t) = 3.14\cos(314t + 105°)\text{ A}$$

LEARNING Hint

Applying $\mathbf{I} = j\omega C\mathbf{V}$

LEARNING EXTENSION

E7.7 The current in a 150-μF capacitor is $\mathbf{I} = 3.6\,\underline{/-145°}$ A. If the frequency of the current is 60 Hz, determine the voltage across the capacitor.

ANSWER
$v_C(t) = 63.66\cos(377t - 235°)$ V

7.5 Impedance and Admittance

We have examined each of the circuit elements in the frequency domain on an individual basis. We now wish to treat these passive circuit elements in a more general fashion. We now define the two-terminal input *impedance* **Z**, also referred to as the driving point impedance, in exactly the same manner in which we defined resistance earlier. Later we will examine another type of impedance, called transfer impedance.

Impedance is defined as the ratio of the phasor voltage **V** to the phasor current **I**:

$$\mathbf{Z} = \frac{\mathbf{V}}{\mathbf{I}} \quad 7.31$$

at the two terminals of the element related to one another by the passive sign convention, as illustrated in Fig. 7.9. Since **V** and **I** are complex, the impedance **Z** is complex and

$$\mathbf{Z} = \frac{V_M \underline{/\theta_v}}{I_M \underline{/\theta_i}} = \frac{V_M}{I_M} \underline{/\theta_v - \theta_i} = Z \underline{/\theta_z} \quad 7.32$$

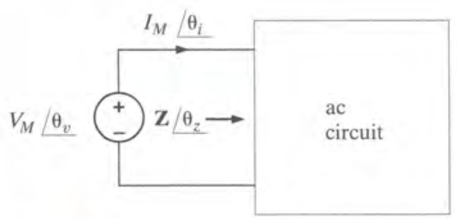


Figure 7.9
General impedance relationship.

Since **Z** is the ratio of **V** to **I**, the units of **Z** are ohms. Thus, impedance in an ac circuit is analogous to resistance in a dc circuit. In rectangular form, impedance is expressed as

$$\mathbf{Z}(\omega) = R(\omega) + jX(\omega) \quad 7.33$$

where $R(\omega)$ is the real, or resistive, component and $X(\omega)$ is the imaginary, or reactive, component. In general, we simply refer to R as the resistance and X as the reactance. It is important to note that R and X are real functions of ω and therefore $\mathbf{Z}(\omega)$is frequency dependent. Equation (7.33) clearly indicates that **Z** is a complex number; however, it is not a phasor, since phasors denote sinusoidal functions.

Equations (7.32) and (7.33) indicate that

$$Z \underline{/\theta_z} = R + jX \quad 7.34$$

Therefore,

$$Z = \sqrt{R^2 + X^2}$$
$$\theta_z = \tan^{-1}\frac{X}{R} \qquad \textbf{7.35}$$

where

$$R = Z\cos\theta_z$$
$$X = Z\sin\theta_z$$

For the individual passive elements the impedance is as shown in Table 7.2. However, just as it was advantageous to know how to determine the equivalent resistance in dc circuits, we want to learn how to determine the equivalent impedance in ac circuits.

Table 7.2 Passive element impedance

Passive element	Impedance
R	$\mathbf{Z} = R$
L	$\mathbf{Z} = j\omega L = jX_L = \omega L\,\underline{/90^\circ}$, $X_L = \omega L$
C	$\mathbf{Z} = \dfrac{1}{j\omega C} = jX_C = -\dfrac{1}{\omega C}\,\underline{/90^\circ}$, $X_C = -\dfrac{1}{\omega C}$

KCL and KVL are both valid in the frequency domain and we can use this fact, as was done in Chapter 2 for resistors, to show that impedances can be combined using the same rules that we established for resistor combinations. That is, if $\mathbf{Z}_1, \mathbf{Z}_2, \mathbf{Z}_3, \ldots, \mathbf{Z}_n$ are connected in series, the equivalent impedance $\mathbf{Z}_s$ is

$$\mathbf{Z}_s = \mathbf{Z}_1 + \mathbf{Z}_2 + \mathbf{Z}_3 + \cdots + \mathbf{Z}_n \qquad \textbf{7.36}$$

and if $\mathbf{Z}_1, \mathbf{Z}_2, \mathbf{Z}_3, \ldots, \mathbf{Z}_n$ are connected in parallel, the equivalent impedance is given by

$$\frac{1}{\mathbf{Z}_p} = \frac{1}{\mathbf{Z}_1} + \frac{1}{\mathbf{Z}_2} + \frac{1}{\mathbf{Z}_3} + \cdots + \frac{1}{\mathbf{Z}_n} \qquad \textbf{7.37}$$

LEARNING Example 7.9

Determine the equivalent impedance of the network shown in Fig. 7.10 if the frequency is $f = 60$ Hz. Then compute the current $i(t)$ if the voltage source is $v(t) = 50\cos(\omega t + 30^\circ)$ V. Finally, calculate the equivalent impedance if the frequency is $f = 400$ Hz.

SOLUTION The impedances of the individual elements at 60 Hz are

$$\mathbf{Z}_R = 25\ \Omega$$
$$\mathbf{Z}_L = j\omega L = j(2\pi \times 60)(20 \times 10^{-3}) = j7.54\ \Omega$$
$$\mathbf{Z}_C = \frac{-j}{\omega C} = \frac{-j}{(2\pi \times 60)(50 \times 10^{-6})} = -j53.05\ \Omega$$

Since the elements are in series,

$$\mathbf{Z} = \mathbf{Z}_R + \mathbf{Z}_L + \mathbf{Z}_C = 25 - j45.51\ \Omega$$

The current in the circuit is given by

$$\mathbf{I} = \frac{\mathbf{V}}{\mathbf{Z}} = \frac{50\angle 30^\circ}{25 - j45.51} = \frac{50\angle 30^\circ}{51.93\angle -61.22^\circ} = 0.96\angle 91.22^\circ\ \text{A}$$

or in the time domain, $i(t) = 0.96\cos(377t + 91.22^\circ)$ A.

If the frequency is 400 Hz, the impedance of each element is

$$\mathbf{Z}_R = 25\ \Omega$$
$$\mathbf{Z}_L = j\omega L = j50.27\ \Omega$$
$$\mathbf{Z}_C = \frac{-j}{\omega C} = -j7.96\ \Omega$$

The total impedance is then

$$\mathbf{Z} = 25 + j42.31 = 49.14\angle 59.42^\circ\ \Omega$$

It is important to note that at the frequency $f = 60$ Hz, the reactance of the circuit is capacitive; that is, if the impedance is written as $R + jX$, $X < 0$; however, at $f = 400$ Hz the reactance is inductive since $X > 0$.

LEARNING Hint

Technique

1. Express $v(t)$ as a phasor and determine the impedance of each passive element.
2. Combine impedances and solve for the phasor **I**.
3. Convert the phasor **I** to $i(t)$.

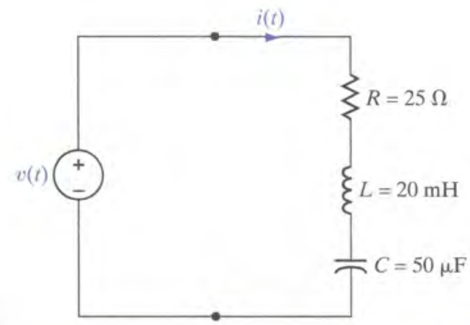


Figure 7.10
Series ac circuit.

LEARNING EXTENSION

E7.8 Find the current $i(t)$ in the network in Fig. E7.8.

ANSWER
$i(t) = 3.88\cos(377t - 39.2^\circ)$ A.

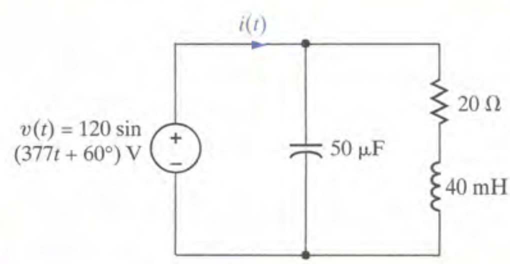


Figure E7.8

Another quantity that is very useful in the analysis of ac circuits is the two-terminal input *admittance*, which is the reciprocal of impedance; that is,

$$\mathbf{Y} = \frac{1}{\mathbf{Z}} = \frac{\mathbf{I}}{\mathbf{V}} \qquad 7.38$$

The units of **Y** are siemens, and this quantity is analogous to conductance in resistive dc circuits. Since **Z** is a complex number, **Y** is also a complex number.

$$\mathbf{Y} = Y_M \underline{/\theta_y} \qquad 7.39$$

which is written in rectangular form as

$$\mathbf{Y} = G + jB \qquad 7.40$$

where G and B are called *conductance* and *susceptance*, respectively. Because of the relationship between **Y** and **Z**, we can express the components of one quantity as a function of the components of the other. From the expression

$$G + jB = \frac{1}{R + jX} \qquad 7.41$$

LEARNING Hint

Technique for taking the reciprocal:

$$\frac{1}{R + jX} = \frac{R - jX}{(R + jX)(R - jX)} = \frac{R - jX}{R^2 + X^2}$$

we can show that

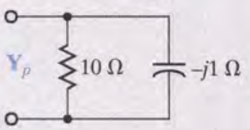
$$G = \frac{R}{R^2 + X^2}, \qquad B = \frac{-X}{R^2 + X^2} \qquad 7.42$$

and in a similar manner, we can show that

$$R = \frac{G}{G^2 + B^2}, \qquad X = \frac{-B}{G^2 + B^2} \qquad 7.43$$

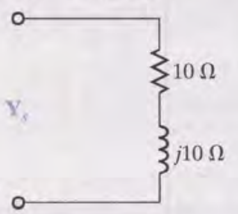
LEARNING by Doing

D 7.2 Find $\mathbf{Y}_p$



ANSWER
$\mathbf{Y}_p = 0.1 + j1$ S.

D 7.3 Find $\mathbf{Y}_s$



ANSWER
$\mathbf{Y}_s = 0.05 - j0.05$ S.

It is very important to note that in general R and G are *not* reciprocals of one another. The same is true for X and B. The purely resistive case is an exception. In the purely reactive case the quantities are negative reciprocals of one another.

The admittance of the individual passive elements are

$$\begin{aligned} \mathbf{Y}_R &= \frac{1}{R} = G \\ \mathbf{Y}_L &= \frac{1}{j\omega L} = -\frac{1}{\omega L}\underline{/90^\circ} \\ \mathbf{Y}_C &= j\omega C = \omega C\underline{/90^\circ} \end{aligned} \qquad 7.44$$

Once again, since KCL and KVL are valid in the frequency domain, we can show, using the same approach outlined in Chapter 2 for conductance in resistive circuits, that the rules for combining admittances are the same as those for combining conductances; that is, if $\mathbf{Y}_1, \mathbf{Y}_2, \mathbf{Y}_3, \ldots, \mathbf{Y}_n$ are connected in parallel, the equivalent admittance is

$$\mathbf{Y}_p = \mathbf{Y}_1 + \mathbf{Y}_2 + \cdots + \mathbf{Y}_n \qquad 7.45$$

and if $\mathbf{Y}_1, \mathbf{Y}_2, \ldots, \mathbf{Y}_n$ are connected in series, the equivalent admittance is

$$\frac{1}{\mathbf{Y}_S} = \frac{1}{\mathbf{Y}_1} + \frac{1}{\mathbf{Y}_2} + \cdots + \frac{1}{\mathbf{Y}_n} \qquad 7.46$$

LEARNING Example 7.10

Calculate the equivalent admittance $\mathbf{Y}_p$ for the network in Fig. 7.11 and use it to determine the current $\mathbf{I}$ if $\mathbf{V}_S = 60\,\underline{/45°}$ V.

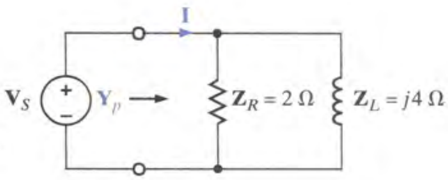


Figure 7.11 *An example parallel circuit.*

SOLUTION From Fig. 7.11 we note that

$$\mathbf{Y}_R = \frac{1}{\mathbf{Z}_R} = \frac{1}{2}\text{ S}$$

$$\mathbf{Y}_L = \frac{1}{\mathbf{Z}_L} = \frac{-j}{4}\text{ S}$$

Therefore,

$$\mathbf{Y}_p = \frac{1}{2} - j\frac{1}{4}\text{ S}$$

and hence,

$$\mathbf{I} = \mathbf{Y}_p\mathbf{V}_S$$

$$= \left(\frac{1}{2} - j\frac{1}{4}\right)(60\,\underline{/45°})$$

$$= 33.5\,\underline{/18.43°}\text{ A}$$

LEARNING Hint

Admittances add in parallel.

LEARNING EXTENSION

E7.9 Find the current $\mathbf{I}$ in the network in Fig. E7.9.

ANSWER $\mathbf{I} = 9.01\underline{/53.7°}$ A.

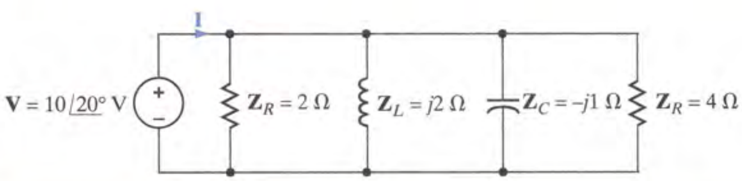


Figure E7.9

As a prelude to our analysis of more general ac circuits, let us examine the techniques for computing the impedance or admittance of circuits in which numerous passive elements are interconnected. The following example illustrates that our technique is analogous to our earlier computations of equivalent resistance.

LEARNING Example 7.11

Consider the network shown in Fig. 7.12a. The impedance of each element is given in the figure. We wish to calculate the equivalent impedance of the network $\mathbf{Z}_{eq}$ at terminals *A-B*.

SOLUTION The equivalent impedance $\mathbf{Z}_{eq}$ could be calculated in a variety of ways; we could use only impedances, or only

LEARNING Hint

Technique:

1. Add the admittances of elements in parallel.
2. Add the impedances of elements in series.
3. Convert back and forth between admittance and impedance in order to combine neighboring elements.

(continued)

admittances, or a combination of the two. We will use the latter. We begin by noting that the circuit in Fig. 7.12a can be represented by the circuit in Fig. 7.12b.

Note that

$$\mathbf{Y}_4 = \mathbf{Y}_L + \mathbf{Y}_C = \frac{1}{j4} + \frac{1}{-j2} = j\frac{1}{4}\text{ S}$$

Therefore,

$$\mathbf{Z}_4 = -j4\ \Omega$$

Now

$$\mathbf{Z}_{34} = \mathbf{Z}_3 + \mathbf{Z}_4 = (4 + j2) + (-j4) = 4 - j2\ \Omega$$

and hence,

$$\mathbf{Y}_{34} = \frac{1}{\mathbf{Z}_{34}} = \frac{1}{4 - j2} = 0.20 + j0.10\text{ S}$$

Since

$$\mathbf{Z}_2 = 2 + j6 - j2 = 2 + j4\ \Omega$$

then

$$\mathbf{Y}_2 = \frac{1}{2 + j4} = 0.10 - j0.20\text{ S}$$

$$\mathbf{Y}_{234} = \mathbf{Y}_2 + \mathbf{Y}_{34} = 0.30 - j0.10\text{ S}$$

The reader should note carefully our approach—we are adding impedances in series and adding admittances in parallel.

From $\mathbf{Y}_{234}$ we can compute $\mathbf{Z}_{234}$ as

$$\mathbf{Z}_{234} = \frac{1}{\mathbf{Y}_{234}} = \frac{1}{0.30 - j0.10} = 3 + j1\ \Omega$$

Now

$$\mathbf{Y}_1 = \mathbf{Y}_R + \mathbf{Y}_C = \frac{1}{1} + \frac{1}{-j2} = 1 + j\frac{1}{2}\text{ S}$$

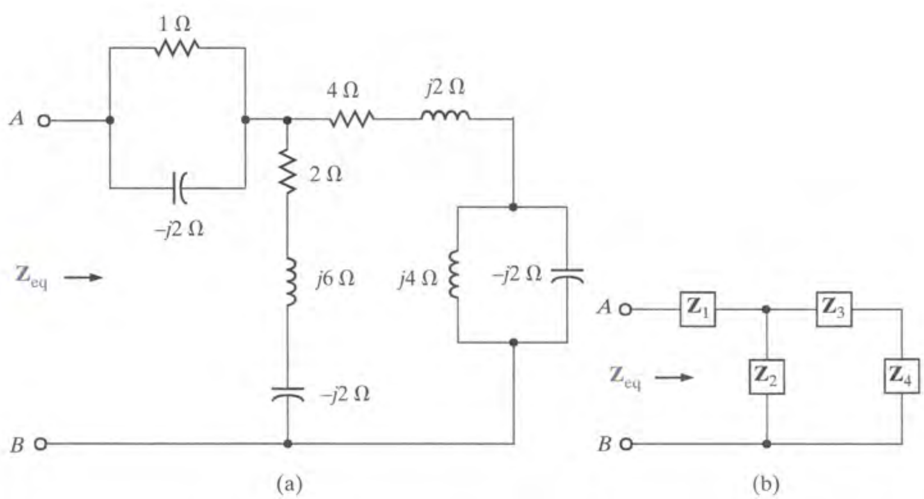


Figure 7.12
Example circuit for determining equivalent impedance in two steps.

and then

$$\mathbf{Z}_1 = \frac{1}{1 + j\frac{1}{2}}$$

$$= 0.8 - j0.4\ \Omega$$

Therefore,

$$\mathbf{Z}_{eq} = \mathbf{Z}_1 + \mathbf{Z}_{234}$$

$$= 0.8 - j0.4 + 3 + j1$$

$$= 3.8 + j0.6\ \Omega$$

LEARNING EXTENSION

E7.10 Compute the impedance $\mathbf{Z}_T$ in the network in Fig. E7.10.

ANSWER
$\mathbf{Z}_T = 3.38 + j1.08\ \Omega$.

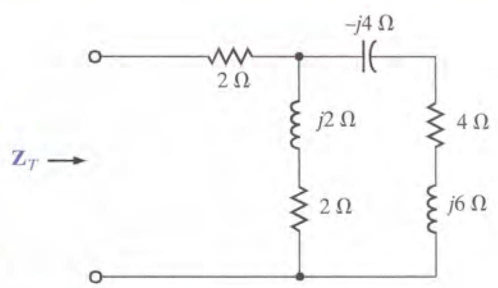


Figure E7.10

7.6 Phasor Diagrams

Impedance and admittance are functions of frequency, and therefore their values change as the frequency changes. These changes in **Z** and **Y** have a resultant effect on the current–voltage relationships in a network. This impact of changes in frequency on circuit parameters can be easily seen via a phasor diagram. The following examples will serve to illustrate these points.

LEARNING Example 7.12

Let us sketch the phasor diagram for the network shown in Fig. 7.13.

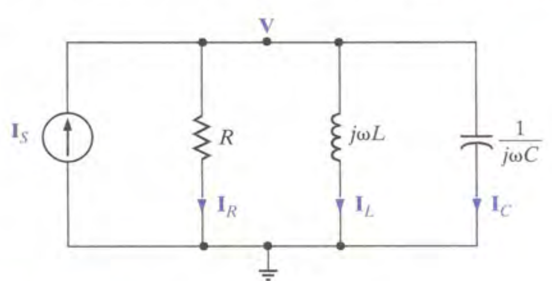


Figure 7.13 *Example parallel circuit.*

SOLUTION The pertinent variables are labeled on the figure. For convenience in forming a phasor diagram, we select **V** as a reference phasor and arbitrarily assign it a 0° phase angle. We will, therefore, measure all currents with respect to this phasor. We suffer no loss of generality by assigning **V** a 0° phase angle, since if it is actually 30°, for example, we will simply rotate the entire phasor diagram by 30° because all the currents are measured with respect to this phasor.

At the upper node in the circuit KCL is

$$\mathbf{I}_S = \mathbf{I}_R + \mathbf{I}_L + \mathbf{I}_C = \frac{\mathbf{V}}{R} + \frac{\mathbf{V}}{j\omega L} + \frac{\mathbf{V}}{1/j\omega C}$$

(continued)

Since $\mathbf{V} = V_M \angle 0°$, then

$$\mathbf{I}_S = \frac{V_M \angle 0°}{R} + \frac{V_M \angle -90°}{\omega L} + V_M \omega C \angle 90°$$

The phasor diagram that illustrates the phase relationship between $\mathbf{V}$, $\mathbf{I}_R$, $\mathbf{I}_L$, and $\mathbf{I}_C$ is shown in Fig. 7.14a. For small values of ω such that the magnitude of $\mathbf{I}_L$ is greater than that of $\mathbf{I}_C$, the phasor diagram for the currents is shown in Fig. 7.14b. In the case of large values of ω—that is, those for which $\mathbf{I}_C$ is greater than $\mathbf{I}_L$—the phasor diagram for the currents is shown in Fig. 7.14c. Note that as ω increases, the phasor $\mathbf{I}_S$ moves from $\mathbf{I}_{S_1}$ to $\mathbf{I}_{S_n}$ along a locus of points specified by the dashed line shown in Fig. 7.14d.

Note that $\mathbf{I}_S$ is in phase with $\mathbf{V}$ when $\mathbf{I}_C = \mathbf{I}_L$ or, in other words, when $\omega L = 1/\omega C$. Hence, the node voltage $\mathbf{V}$ is in phase with the current source $\mathbf{I}_S$ when

$$\omega = \frac{1}{\sqrt{LC}}$$

This can also be seen from the KCL equation

$$\mathbf{I} = \left[\frac{1}{R} + j\left(\omega C - \frac{1}{\omega L}\right)\right]\mathbf{V}$$

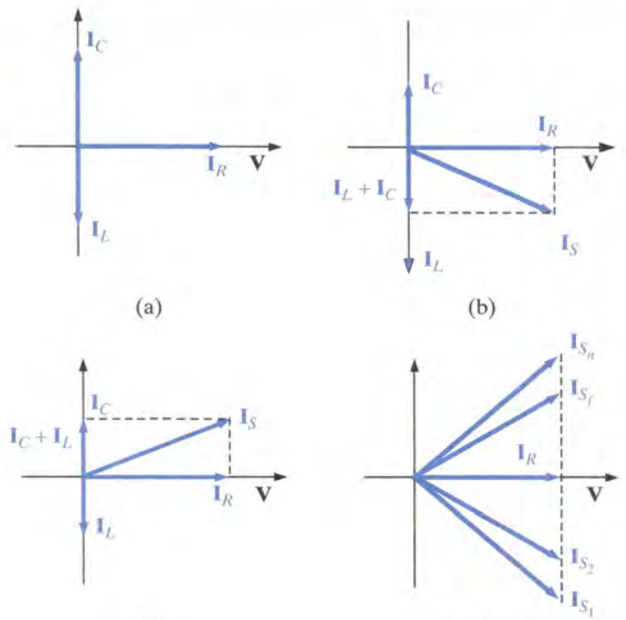
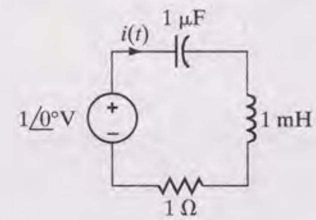


Figure 7.14 Phasor diagrams for the circuit in Fig. 7.13.

LEARNING by Doing

D 7.4 Find the frequency at which $v(t)$ and $i(t)$ are in phase.



ANSWER $f = 5.03 \times 10^3$ Hz.

LEARNING Hint

From a graphical standpoint, phasors can be manipulated like vectors.

LEARNING Example 7.13

Let us determine the phasor diagram for the series circuit shown in Fig. 7.15a.

SOLUTION KVL for this circuit is of the form

$$\mathbf{V}_S = \mathbf{V}_R + \mathbf{V}_L + \mathbf{V}_C$$

$$= \mathbf{I}R + \omega L\mathbf{I} \angle 90° + \frac{\mathbf{I}}{\omega C} \angle -90°$$

If we select $\mathbf{I}$ as a reference phasor so that $\mathbf{I} = I_M \angle 0°$, then if $\omega L I_M > I_M/\omega C$, the phasor diagram will be of the form shown in Fig. 7.15b. Specifically, if $\omega = 377$ rad/s (i.e., $f = 60$ Hz), then $\omega L = 6$ and $1/\omega C = 2$. Under these conditions the phasor diagram is as shown in Fig. 7.15c. If, however, we select $\mathbf{V}_S$ as reference with, for example,

$$v_S(t) = 12\sqrt{2}\cos(377t + 90°) \text{ V}$$

then

$$\mathbf{I} = \frac{\mathbf{V}}{\mathbf{Z}} = \frac{12\sqrt{2} \angle 90°}{4 + j6 - j2}$$

$$= \frac{12\sqrt{2} \angle 90°}{4\sqrt{2} \angle 45°}$$

$$= 3 \angle 45° \text{ A}$$

and the entire phasor diagram, as shown in Figs. 7.15b and c, is rotated 45°, as shown in Fig. 7.15d.

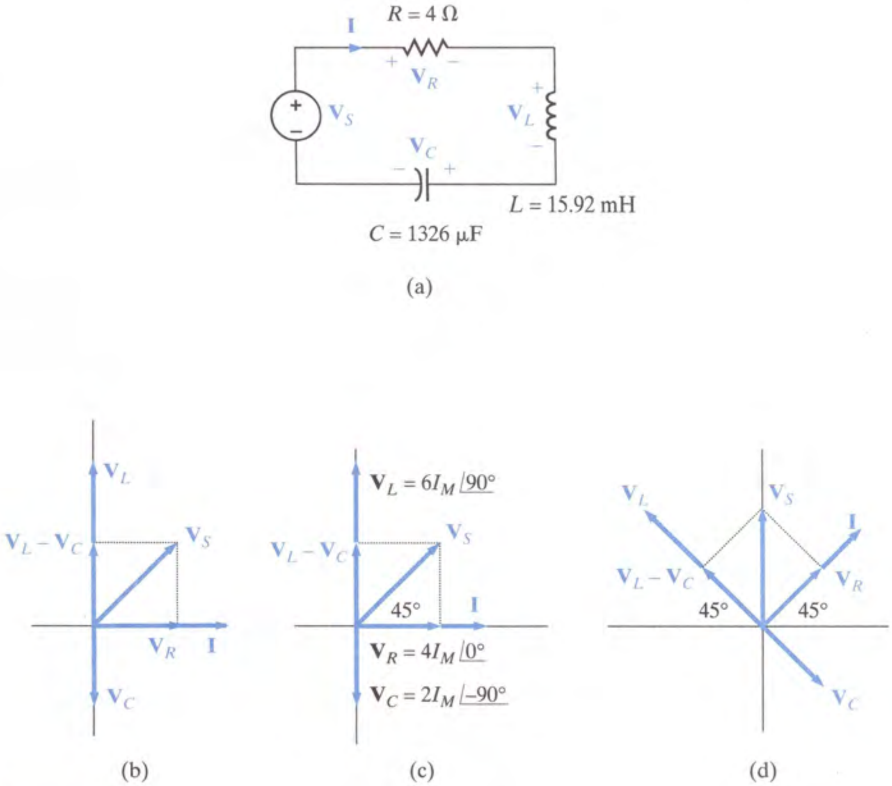


Figure 7.15
Series circuit and certain specific phasor diagrams (plots are not drawn to scale).

LEARNING EXTENSION

E7.11 Draw a phasor diagram illustrating all currents and voltages for the network in Fig. E7.11.

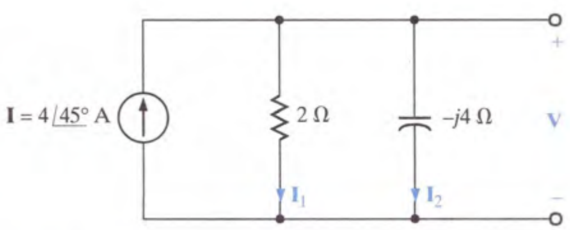
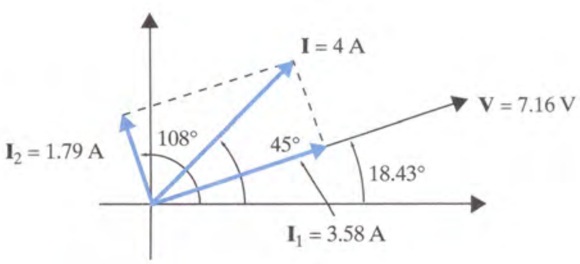


Figure E7.11

ANSWER



7.7 Basic Analysis Using Kirchhoff's Laws

We have shown that Kirchhoff's laws apply in the frequency domain, and therefore they can be used to compute steady-state voltages and currents in ac circuits. This approach involves expressing these voltages and currents as phasors, and once this is done, the ac steady-state analysis employing phasor equations is performed in an identical fashion to that used in the dc analysis of resistive circuits. Complex number algebra is the tool that is used for the mathematical manipulation of the phasor equations, which, of course, have complex coefficients. We will begin by illustrating that the techniques we have applied in the solution of dc resistive circuits are valid in ac circuit analysis also—the only difference being that in steady-state ac circuit analysis the algebraic phasor equations have complex coefficients.

Problem-Solving Strategy AC Steady-State Analysis

- For relatively simple circuits (e.g., those with a single source), use
  - Ohm's law for ac analysis, i.e., $\mathbf{V} = \mathbf{IZ}$
  - The rules for combining $\mathbf{Z}_s$ and $\mathbf{Y}_p$
  - KCL and KVL
  - Current and voltage division
- For more complicated circuits with multiple sources, use
  - Nodal analysis
  - Loop or mesh analysis
  - Superposition
  - Source exchange
  - Thévenin's and Norton's theorems
  - MATLAB
  - PSPICE

At this point, it is important for the reader to understand that in our manipulation of algebraic phasor equations with complex coefficients we will, for the sake of simplicity, normally carry only two digits to the right of the decimal point. In doing so, we will introduce round-off errors in our calculations. Nowhere are these errors more evident than when two or more approaches are used to solve the same problem, as is done in the following example.

LEARNING Example 7.14

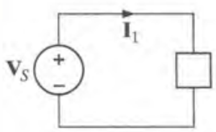
We wish to calculate all the voltages and currents in the circuit shown in Fig. 7.16a.

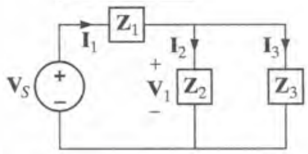
SOLUTION Our approach will be as follows. We will calculate the total impedance seen by the source $\mathbf{V}_S$. Then we will use this to determine $\mathbf{I}_1$. Knowing $\mathbf{I}_1$, we can compute $\mathbf{V}_1$ using KVL. Knowing $\mathbf{V}_1$, we can compute $\mathbf{I}_2$ and $\mathbf{I}_3$, and so on.

LEARNING Hint

Technique

1. Compute $\mathbf{I}_1$.





2. Determine $\mathbf{V}_1 = \mathbf{V}_S - \mathbf{I}_1\mathbf{Z}_1$.

Then $\mathbf{I}_2 = \dfrac{\mathbf{V}_1}{\mathbf{Z}_2}$ and $\mathbf{I}_3 = \dfrac{\mathbf{V}_1}{\mathbf{Z}_3}$

Current and voltage division are also applicable.

The total impedance seen by the source $\mathbf{V}_S$ is

$$\mathbf{Z}_{eq} = 4 + \frac{(j6)(8 - j4)}{j6 + 8 - j4}$$

$$= 4 + \frac{24 + j48}{8 + j2}$$

$$= 4 + 4.24 + j4.94$$

$$= 9.61\,\underline{/30.94^\circ}\ \Omega$$

Then

$$\mathbf{I}_1 = \frac{\mathbf{V}_S}{\mathbf{Z}_{eq}} = \frac{24\,\underline{/60^\circ}}{9.61\,\underline{/30.94^\circ}}$$

$$= 2.5\,\underline{/29.06^\circ}\ \text{A}$$

$\mathbf{V}_1$ can be determined using KVL:

$$\mathbf{V}_1 = \mathbf{V}_S - 4\mathbf{I}_1$$

$$= 24\,\underline{/60^\circ} - 10\,\underline{/29.06^\circ}$$

$$= 3.26 + j15.93$$

$$= 16.26\,\underline{/78.43^\circ}\ \text{V}$$

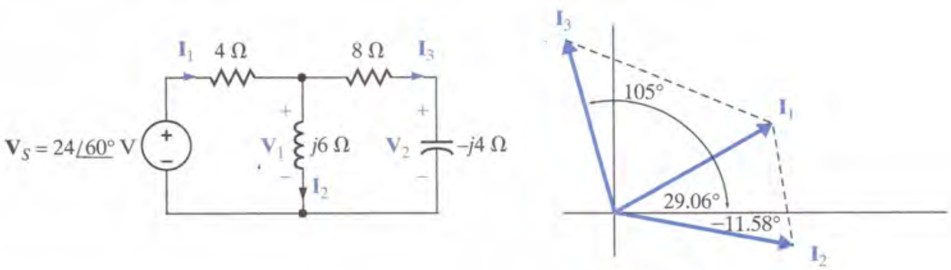
Note that $\mathbf{V}_1$ could also be computed via voltage division:

$$\mathbf{V}_1 = \frac{\mathbf{V}_S \dfrac{(j6)(8 - j4)}{j6 + 8 - j4}}{4 + \dfrac{(j6)(8 - j4)}{j6 + 8 - 4}}\ \text{V}$$

which from our previous calculation is

$$\mathbf{V}_1 = \frac{(24\,\underline{/60^\circ})(6.51\,\underline{/49.36^\circ})}{9.61\,\underline{/30.94^\circ}}$$

$$= 16.26\,\underline{/78.42^\circ}\ \text{V}$$



(a) (b)

Figure 7.16

(a) Example ac circuit, (b) phasor diagram for the currents (plots are not drawn to scale).

(continued)

Knowing $\mathbf{V}_1$, we can calculate both $\mathbf{I}_2$ and $\mathbf{I}_3$:

$$\mathbf{I}_2 = \frac{\mathbf{V}_1}{j6} = \frac{16.26\angle 78.43^\circ}{6\angle 90^\circ} = 2.71\angle -11.58^\circ \text{ A}$$

and

$$\mathbf{I}_3 = \frac{\mathbf{V}_1}{8 - j4} = 1.82\angle 105^\circ \text{ A}$$

Note that $\mathbf{I}_2$ and $\mathbf{I}_3$ could have been calculated by current division. For example, $\mathbf{I}_2$ could be determined by

$$\mathbf{I}_2 = \frac{\mathbf{I}_1(8 - j4)}{8 - j4 + j6} = \frac{(2.5\angle 29.06^\circ)(8.94\angle -26.57^\circ)}{8 + j2} = 2.71\angle -11.55^\circ \text{ A}$$

Finally, $\mathbf{V}_2$ can be computed as

$$\mathbf{V}_2 = \mathbf{I}_3(-j4) = 7.28\angle 15^\circ \text{ V}$$

This value could also have been computed by voltage division. The phasor diagram for the currents $\mathbf{I}_1$, $\mathbf{I}_2$, and $\mathbf{I}_3$ is shown in Fig. 7.16b and is an illustration of KCL.

Finally, the reader is encouraged to work the problem in reverse; that is, given $\mathbf{V}_2$, find $\mathbf{V}_S$. Note that if $\mathbf{V}_2$ is known, $\mathbf{I}_3$ can be computed immediately using the capacitor impedance. Then $\mathbf{V}_2 + \mathbf{I}_3(8)$ yields $\mathbf{V}_1$. Knowing $\mathbf{V}_1$ we can find $\mathbf{I}_2$. Then $\mathbf{I}_2 + \mathbf{I}_3 = \mathbf{I}_1$, and so on. Note that this analysis, which is the subject of Learning Extensions Exercise E7.12, involves simply a repeated application of Ohm's law, KCL, and KVL.

LEARNING EXTENSION

E7.12 In the network in Fig. E7.12, $\mathbf{V}_o$ is known to be $8\angle 45^\circ$ V. Compute $\mathbf{V}_S$.

ANSWER
$\mathbf{V}_S = 17.89\angle -18.43^\circ$ V.

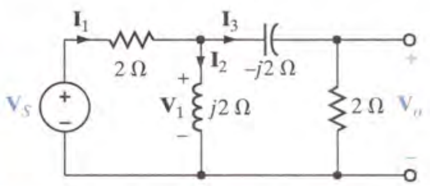


Figure E7.12

7.8 Analysis Techniques

In this section we revisit the circuit analysis methods that were successfully applied earlier to dc circuits and illustrate their applicability to ac steady-state analysis. The vehicle we employ to present these techniques is examples in which all the theorems, together with nodal analysis and loop analysis, are used to obtain a solution.

LEARNING Example 7.15

Let us determine the current $\mathbf{I}_o$ in the network in Fig. 7.17a using nodal analysis, loop analysis, superposition, source exchange, Thévenin's theorem, and Norton's theorem.

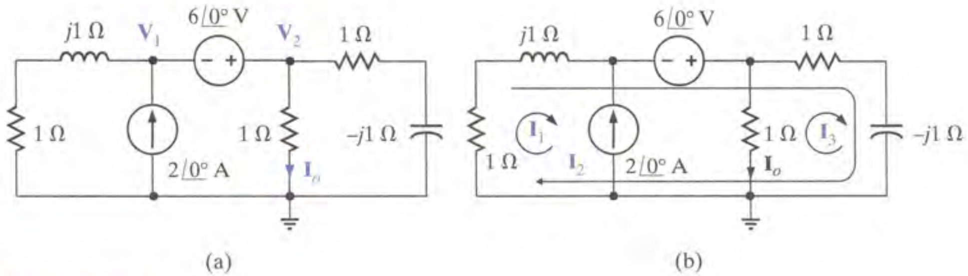


Figure 7.17
Circuits used in Example 7.15 for node and loop analysis.

SOLUTION

LEARNING Hint

Summing the current, leaving the supernode. Outbound currents have a positive sign.

(1) Nodal Analysis We begin with a nodal analysis of the network. The KCL equation for the supernode that includes the voltage source is

$$\frac{\mathbf{V}_1}{1+j} - 2\underline{/0^\circ} + \frac{\mathbf{V}_2}{1} + \frac{\mathbf{V}_2}{1-j} = 0$$

and the associated KVL constraint equation is

$$\mathbf{V}_1 + 6\underline{/0^\circ} = \mathbf{V}_2$$

Solving for $\mathbf{V}_1$ in the second equation and using this value in the first equation yields

$$\frac{\mathbf{V}_2 - 6\underline{/0^\circ}}{1+j} - 2\underline{/0^\circ} + \mathbf{V}_2 + \frac{\mathbf{V}_2}{1-j} = 0$$

or

$$\mathbf{V}_2\left[\frac{1}{1+j} + 1 + \frac{1}{1-j}\right] = \frac{6+2+2j}{1+j}$$

Solving for $\mathbf{V}_2$, we obtain

$$\mathbf{V}_2 = \left(\frac{4+j}{1+j}\right)\text{V}$$

Therefore,

$$\mathbf{I}_o = \frac{4+j}{1+j} = \left(\frac{5}{2} - \frac{3}{2}j\right)\text{A}$$

LEARNING Hint

Just as in a dc analysis, the loop equations assume that a decrease in potential level is + and an increase is −.

(2) Loop Analysis The network in Fig. 7.17b is used to perform a loop analysis. Note that one loop current is selected that passes through the independent current source. The three loop equations are

$$\mathbf{I}_1 = -2\underline{/0^\circ}$$

$$1(\mathbf{I}_1+\mathbf{I}_2) + j1(\mathbf{I}_1+\mathbf{I}_2) - 6\underline{/0^\circ} + 1(\mathbf{I}_2+\mathbf{I}_3) - j1(\mathbf{I}_2+\mathbf{I}_3) = 0$$

$$1\mathbf{I}_3 + 1(\mathbf{I}_2+\mathbf{I}_3) - j1(\mathbf{I}_2+\mathbf{I}_3) = 0$$

Combining the first two equations yields

$$\mathbf{I}_2(2) + \mathbf{I}_3(1-j) = 8+2j$$

The third loop equation can be simplified to the form

$$\mathbf{I}_2(1-j) + \mathbf{I}_3(2-j) = 0$$

Solving this last equation for $\mathbf{I}_2$ and substituting the value into the previous equation yields

$$\mathbf{I}_3\left[\frac{-4+2j}{1-j} + 1 - j\right] = 8+2j$$

(continued)

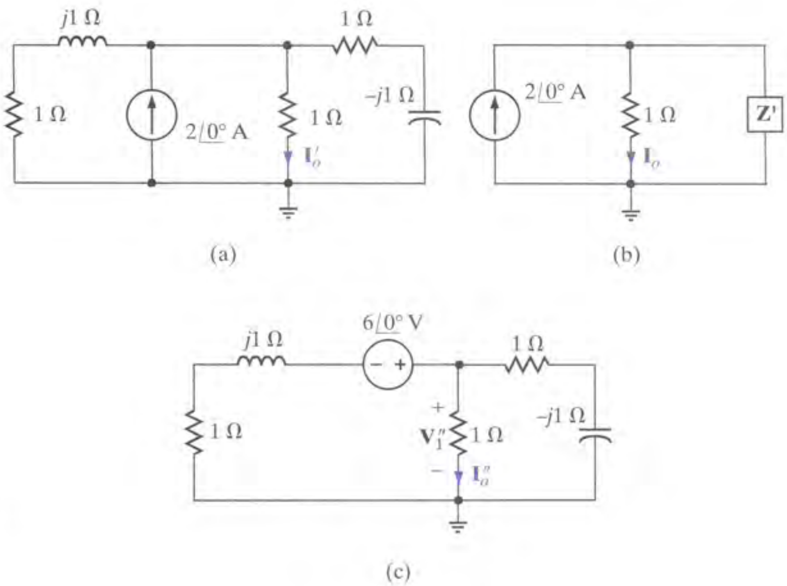


Figure 7.18
Circuits used in Example 7.15 for a superposition analysis.

or

$$\mathbf{I}_3 = \frac{-10 + 6j}{4}$$

and finally

$$\mathbf{I}_o = -\mathbf{I}_3 = \left(\frac{5}{2} - \frac{3}{2}j\right) \text{A}$$

LEARNING Hint

In applying superposition in this case, each source is applied independently and the results are added to obtain the solution.

(3) Superposition In using superposition, we apply one independent source at a time. The network in which the current source acts alone is shown in Fig. 7.18a. By combining the two parallel impedances on each end of the network, we obtain the circuit in Fig. 7.18b, where

$$\mathbf{Z}' = \frac{(1 + j)(1 - j)}{(1 + j) + (1 - j)} = 1\ \Omega$$

Therefore, using current division

$$\mathbf{I}'_o = 1\underline{/0^\circ}\ \text{A}$$

The circuit in which the voltage source acts alone is shown in Fig. 7.18c. The voltage $\mathbf{V}''_1$ obtained using voltage division is

$$\mathbf{V}''_1 = \frac{(6\underline{/0^\circ})\left[\dfrac{1(1 - j)}{1 + 1 - j}\right]}{1 + j + \left[\dfrac{1(1 - j)}{1 + 1 - j}\right]} = \frac{6(1 - j)}{4} \text{V}$$

and hence,

$$\mathbf{I}''_0 = \frac{6}{4}(1 - j)\ \text{A}$$

Then

$$\mathbf{I}_o = \mathbf{I}'_o + \mathbf{I}''_o = 1 + \frac{6}{4}(1 - j) = \left(\frac{5}{2} - \frac{3}{2}j\right) \text{A}$$

LEARNING Hint

In source exchange, a voltage source in series with an impedance can be exchanged for a current source in parallel with the impedance, and vice versa. Repeated application systematically reduces the number of circuit elements.

(4) Source Exchange As a first step in the source exchange approach, we exchange the current source and parallel impedance for a voltage source in series with the impedance, as shown in Fig. 7.19a.

Adding the two voltage sources and transforming them and the series impedance into a current source in parallel with that impedance are shown in Fig. 7.19b. Combining the two impedances that are in parallel with the 1-Ω resistor produces the network in Fig. 7.19c, where

$$\mathbf{Z} = \frac{(1 + j)(1 - j)}{1 + j + 1 - j} = 1\ \Omega$$

Therefore, using current division,

$$\mathbf{I}_o = \left(\frac{8 + 2j}{1 + j}\right)\left(\frac{1}{2}\right) = \frac{4 + j}{1 + j} = \left(\frac{5}{2} - \frac{3}{2}j\right) \text{A}$$

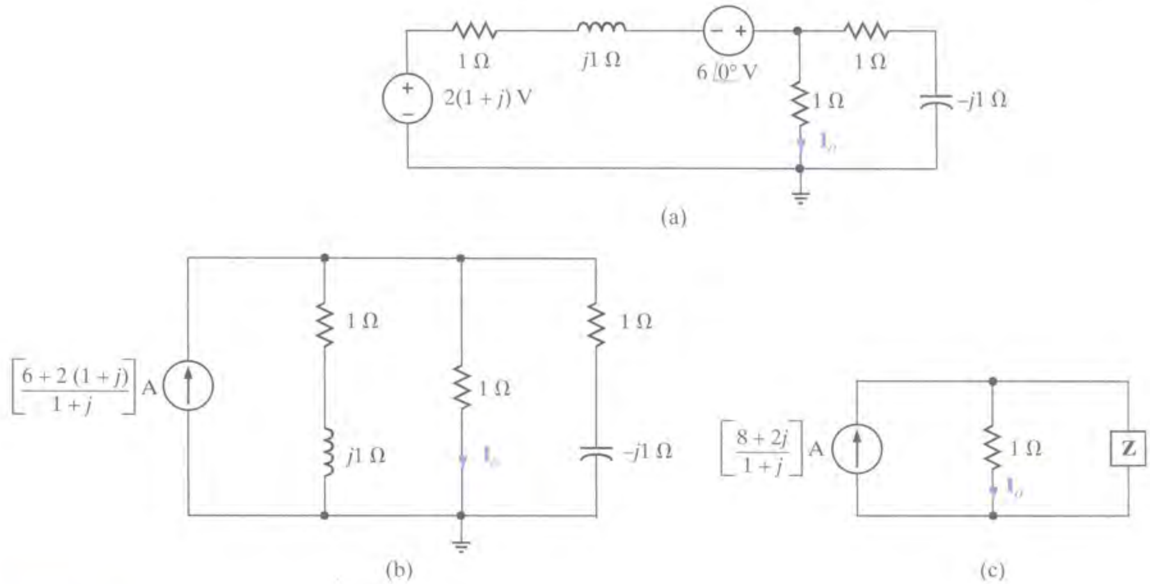
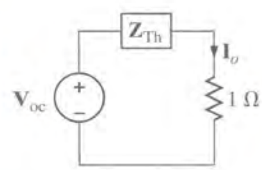


Figure 7.19
Circuits used in Example 7.15 for a sourrce exchange analysis.

LEARNING Hint

In this Thévenin analysis,

1. Remove the 1-Ω load and find the voltage across the open terminals, V_{oc}.
2. Determine the impedance Z_{Th} at the open terminals with all sources made zero.
3. Construct the following circuit and determine I_o.



(5) Thévenin Analysis In applying Thévenin's theorem to the circuit in Fig. 7.17a, we first find the open-circuit voltage, $\mathbf{V}_{oc}$, as shown in Fig. 7.20a. To simplify the analysis, we perform a source exchange on the left end of the network, which results in the circuit in Fig. 7.20b. Now using voltage division,

$$\mathbf{V}_{oc} = [6 + 2(1 + j)]\left[\frac{1 - j}{1 - j + 1 + j}\right]$$

or

$$\mathbf{V}_{oc} = (5 - 3j)\ \text{V}$$

The Thévenin equivalent impedance, $\mathbf{Z}_{Th}$, obtained at the open-circuit terminals when the current source is replaced with an open circuit and the voltage source is replaced with a short circuit is shown in Fig. 7.20c and calculated to be

$$\mathbf{Z}_{Th} = \frac{(1 + j)(1 - j)}{1 + j + 1 - j} = 1\ \Omega$$

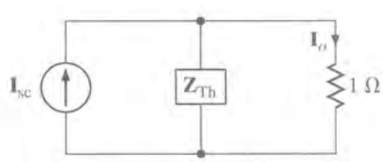
Connecting the Thévenin equivalent circuit to the 1-Ω resistor containing $\mathbf{I}_o$ in the original network yields the circuit in Fig. 7.20d. The current $\mathbf{I}_o$ is then

$$\mathbf{I}_o = \left(\frac{5}{2} - \frac{3}{2}j\right)\text{A}$$

LEARNING Hint

In this Norton analysis,

1. Remove the 1-Ω load and find the current I_{sc} through the short-circuited terminals.
2. Determine the impedance Z_{Th} at the open load terminals with all sources made zero.
3. Construct the following circuit and determine I_o.



(6) Norton Analysis Finally, in applying Norton's theorem to the circuit in Fig. 7.17a, we calculate the short-circuit current, $\mathbf{I}_{sc}$, using the network in Fig. 7.21a. Note that because of the short circuit, the voltage source is directly across the impedance in the left-most branch. Therefore,

$$\mathbf{I}_1 = \frac{6\,\angle 0^\circ}{1 + j}$$

(continued)

Then using KCL,

$$\mathbf{I}_{sc} = \mathbf{I}_1 + 2\underline{/0^\circ} = 2 + \frac{6}{1 + j}$$
$$= \left(\frac{8 + 2j}{1 + j}\right) \text{A}$$

The Thévenin equivalent impedance, $\mathbf{Z}_{Th}$, is known to be 1 Ω and, therefore, connecting the Norton equivalent to the 1-Ω resistor containing $\mathbf{I}_o$ yields the network in Fig. 7.21b. Using current division, we find that

$$\mathbf{I}_o = \frac{1}{2}\left(\frac{8 + 2j}{1 + j}\right)$$
$$= \left(\frac{5}{2} - \frac{3}{2}j\right) \text{A}$$

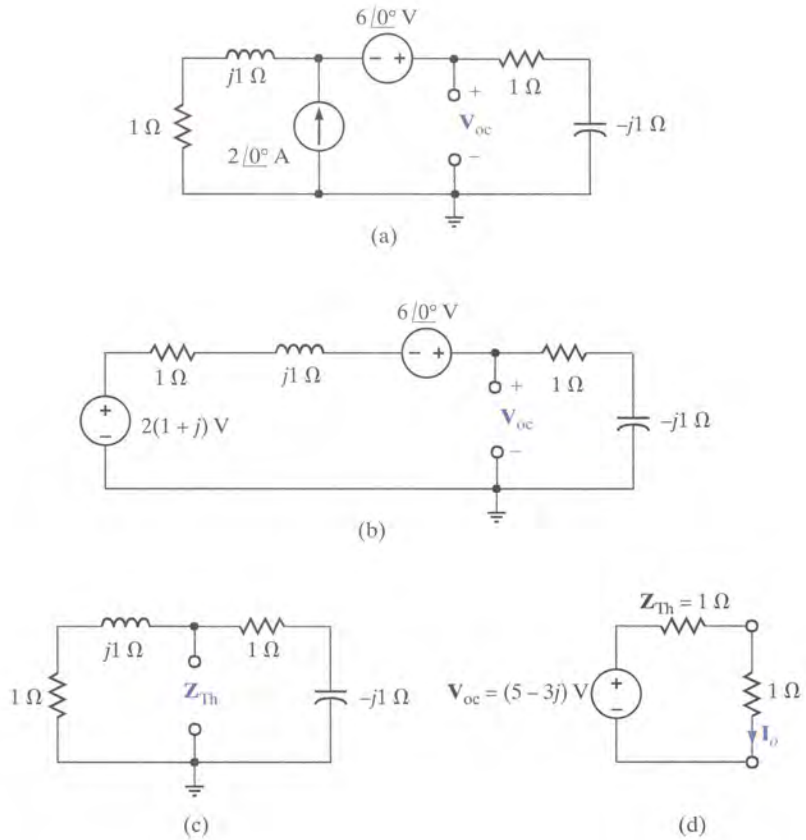


Figure 7.20
Circuits used in Example 7.15 for a Thévenin analysis.

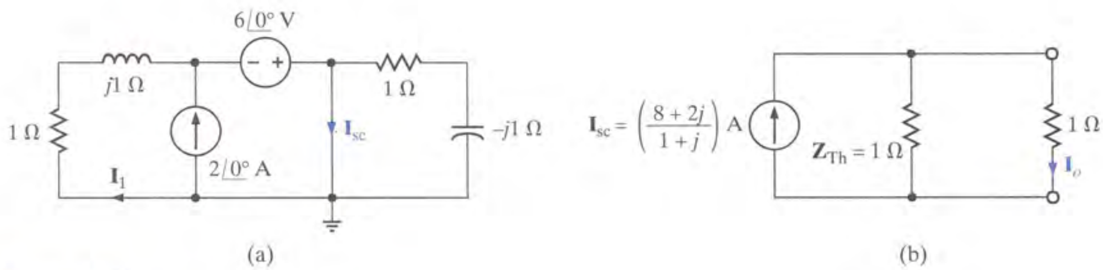


Figure 7.21
Circuits used in Example 7.15 for a Norton analysis.

Let us now consider an example containing a dependent source.

LEARNING Example 7.16

Let us determine the voltage $\mathbf{V}_o$ in the circuit in Fig. 7.22a. In this example we will use node equations, loop equations, Thévenin's theorem, and Norton's theorem. We will omit the techniques of superposition and source transformation. Why?

SOLUTION

LEARNING Hint

How does the presence of a dependent source affect superposition and source exchange?

(1) Nodal Analysis To perform a nodal analysis, we label the node voltages and identify the supernode as shown in Fig. 7.22b. The constraint equation for the supernode is

$$\mathbf{V}_3 + 12\underline{/0^\circ} = \mathbf{V}_1$$

and the KCL equations for the nodes of the network are

$$\frac{\mathbf{V}_1 - \mathbf{V}_2}{-j1} + \frac{\mathbf{V}_3 - \mathbf{V}_2}{1} - 4\underline{/0^\circ} + \frac{\mathbf{V}_3 - \mathbf{V}_o}{1} + \frac{\mathbf{V}_3}{j1} = 0$$

$$\frac{\mathbf{V}_2 - \mathbf{V}_1}{-j1} + \frac{\mathbf{V}_2 - \mathbf{V}_3}{1} - 2\left(\frac{\mathbf{V}_3 - \mathbf{V}_o}{1}\right) = 0$$

$$4\underline{/0^\circ} + \frac{\mathbf{V}_o - \mathbf{V}_3}{1} + \frac{\mathbf{V}_o}{1} = 0$$

At this point we can solve the foregoing equations using a matrix analysis or, for example, substitute the first and last equations into the remaining two equations, which yields

$$3\mathbf{V}_o - (1 + j)\mathbf{V}_2 = -(4 + j12)$$
$$-(4 + j2)\mathbf{V}_o + (1 + j)\mathbf{V}_2 = 12 + j16$$

Solving these equations for $\mathbf{V}_o$ yields

$$\mathbf{V}_o = \frac{-(8 + j4)}{1 + j2} = +4\underline{/143.13^\circ}\text{ V}$$

(2) Loop Analysis The mesh currents for the network are defined in Fig. 7.22c. The constraint equations for the circuit are

$$\mathbf{I}_2 = -4\underline{/0^\circ}$$
$$\mathbf{I}_x = \mathbf{I}_4 - \mathbf{I}_2 = \mathbf{I}_4 + 4\underline{/0^\circ}$$
$$\mathbf{I}_3 = 2\mathbf{I}_x = 2\mathbf{I}_4 + 8\underline{/0^\circ}$$

The KVL equations for mesh 1 and mesh 4 are

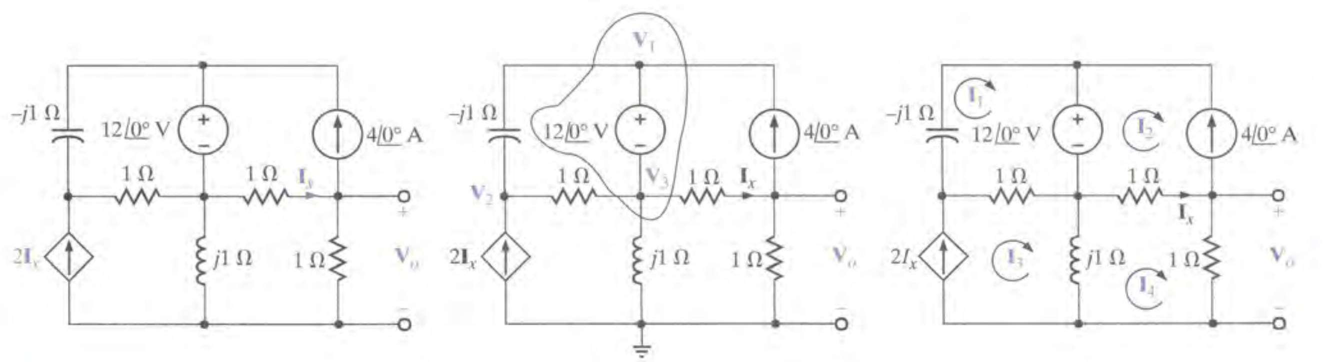
$$-j1\mathbf{I}_1 + 1(\mathbf{I}_1 - \mathbf{I}_3) = -12\underline{/0^\circ}$$
$$j1(\mathbf{I}_4 - \mathbf{I}_3) + 1(\mathbf{I}_4 - \mathbf{I}_2) + 1\mathbf{I}_4 = 0$$

Note that if the constraint equations are substituted into the second KVL equation, the only unknown in the equation is $\mathbf{I}_4$. This substitution yields

$$\mathbf{I}_4 = +4\underline{/143.13^\circ}\text{ A}$$

and hence,

$$\mathbf{V}_o = +4\underline{/143.13^\circ}\text{ V}$$



(a) (b) (c)

Figure 7.22 Circuits used in Example 7.16 for nodal and loop analysis.

(continued)

(3) Thévenin's Theorem In applying Thévenin's theorem, we will find the open-circuit voltage and then determine the Thévenin equivalent impedance using a test source at the open-circuit terminals. We could determine the Thévenin equivalent impedance by calculating the short-circuit current; however, we will determine this current when we apply Norton's theorem.

The open-circuit voltage is determined from the network in Fig. 7.23a. Note that $\mathbf{I}'_x = 4\angle 0°$ A and since $2\mathbf{I}'_x$ flows through the inductor, the open-circuit voltage $\mathbf{V}_{oc}$ is

$$\mathbf{V}_{oc} = -1(4\angle 0°) + j1(2\mathbf{I}'_x)$$
$$= -4 + j8 \text{ V}$$

To determine the Thévenin equivalent impedance, we turn off the independent sources, apply a test voltage source to the output terminals, and compute the current leaving the test source. As shown in Fig. 7.23b, since $\mathbf{I}''_x$ flows in the test source, KCL requires that the current in the inductor be $\mathbf{I}''_x$ also. KVL around the mesh containing the test source indicates that

$$j1\mathbf{I}''_x - 1\mathbf{I}''_x - \mathbf{V}_{\text{test}} = 0$$

Therefore,

$$\mathbf{I}''_x = \frac{-\mathbf{V}_{\text{test}}}{1 - j}$$

Then

$$\mathbf{Z}_{\text{Th}} = \frac{\mathbf{V}_{\text{test}}}{-\mathbf{I}''_x}$$
$$= 1 - j\,\Omega$$

If the Thévenin equivalent network is now connected to the load, as shown in Fig. 7.23c, the output voltage $\mathbf{V}_o$ is found to be

$$\mathbf{V}_o = \frac{-4 + 8j}{2 - j1}(1)$$
$$= +4\angle 143.13° \text{ V}$$

(4) Norton's Theorem In using Norton's theorem, we will find the short-circuit current from the network in Fig. 7.24a. Once again, using the supernode, the constraint and KCL equations are

$$\mathbf{V}_3 + 12\angle 0° = \mathbf{V}_1$$

$$\frac{\mathbf{V}_2 - \mathbf{V}_1}{-j1} + \frac{\mathbf{V}_2 - \mathbf{V}_3}{1} - 2\mathbf{I}'''_x = 0$$

$$\frac{\mathbf{V}_1 - \mathbf{V}_2}{-j1} + \frac{\mathbf{V}_3 - \mathbf{V}_2}{1} - 4\angle 0° + \frac{\mathbf{V}_3}{j1} + \mathbf{I}'''_x = 0$$

$$\mathbf{I}'''_x = \frac{\mathbf{V}_3}{1}$$

Substituting the first and last equations into the remaining equations yields

$$(1 + j)\mathbf{V}_2 - (3 + j)\mathbf{I}'''_x = j12$$
$$-(1 + j)\mathbf{V}_2 + (2)\mathbf{I}'''_x = 4 - j12$$

Solving these equations for $\mathbf{I}'''_x$ yields

$$\mathbf{I}'''_x = \frac{-4}{1 + j} \text{ A}$$

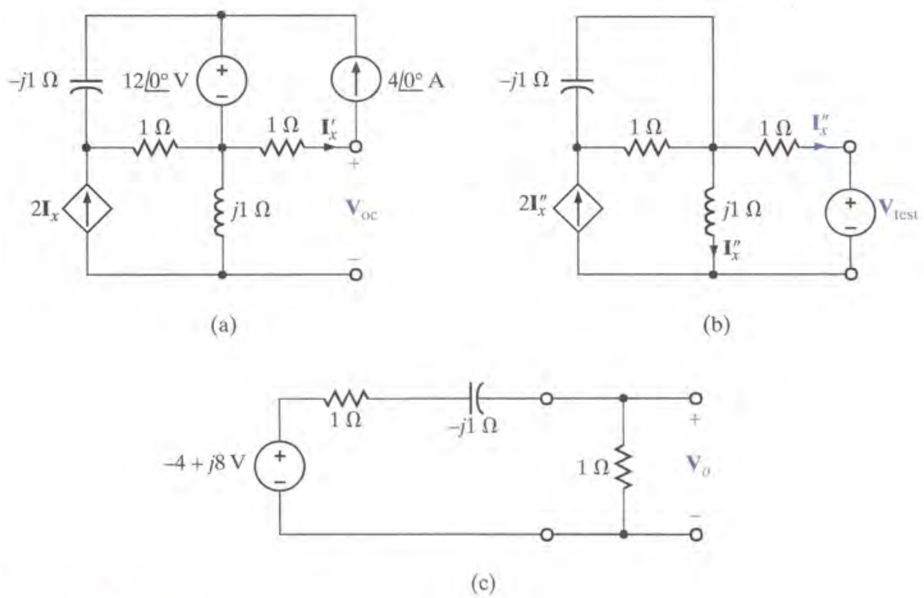


Figure 7.23
Circuits used in Example 7.16 when applying Thévenin's theorem.

The KCL equation at the right-most node in the network in Fig. 7.24a is

$$\mathbf{I}_x''' = 4\underline{/0^\circ} + \mathbf{I}_{sc}$$

Solving for $\mathbf{I}_{sc}$, we obtain

$$\mathbf{I}_{sc} = \frac{-(8 + j4)}{1 + j}\text{ A}$$

The Thévenin equivalent impedance was found earlier to be

$$\mathbf{Z}_{Th} = 1 - j\,\Omega$$

Using the Norton equivalent network, the original network is reduced to that shown in Fig. 7.24b. The voltage $\mathbf{V}_o$ is then

$$\begin{aligned}\mathbf{V}_o &= \frac{-(8 + j4)}{1 + j}\left[\frac{(1)(1 - j)}{1 + 1 - j}\right] \\ &= -4\left[\frac{3 - j}{3 + j}\right] \\ &= +4\underline{/-143.13^\circ}\text{ V}\end{aligned}$$

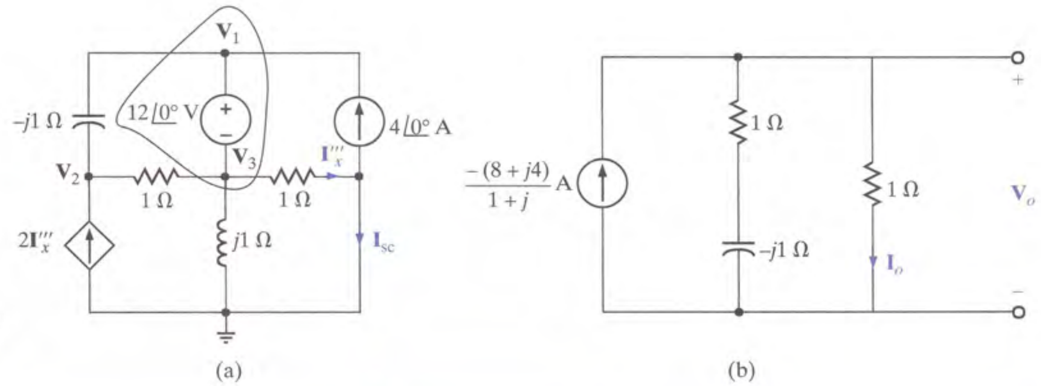


Figure 7.24 Circuits used in Example 7.16 when applying Norton's theorem.

LEARNING EXTENSIONS

E7.13 Use nodal analysis to find $\mathbf{V}_o$ in the network in Fig. E7.13.

ANSWER $\mathbf{V}_o = 2.12\underline{/75^\circ}$ V.

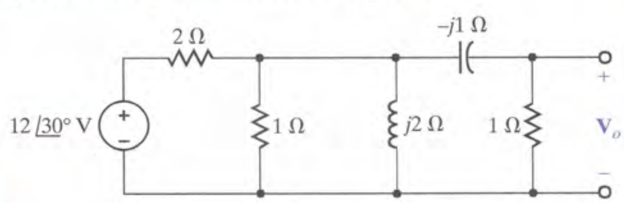


Figure E7.13

E7.14 Use (a) mesh equations and (b) Thévenin's theorem to find $\mathbf{V}_o$ in the network in Fig. E7.14.

ANSWER $\mathbf{V}_o = 10.88\underline{/36^\circ}$ V.

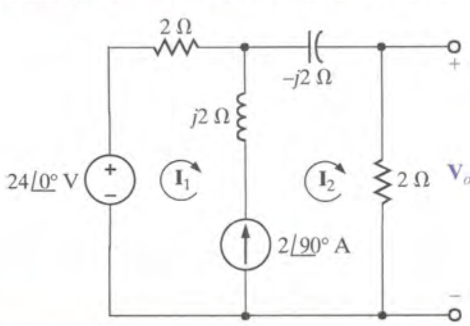


Figure E7.14

E7.15 Use (a) superposition, (b) source transformation, and (c) Norton's theorem to find $\mathbf{V}_o$ in the network in Fig. E7.15.

ANSWER $\mathbf{V}_o = 12\angle 90°$ V.

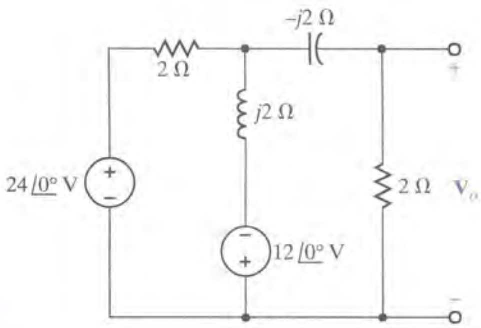


Figure E7.15

MATLAB Analysis

When dealing with large networks it is impractical to determine the currents and voltages within the network without the help of a mathematical software or CAD package. Thus, we will once again demonstrate how to bring this computing power to bear when solving more complicated ac networks.

Earlier we used MATLAB to solve a set of simultaneous equations, which yielded the node voltages or loop currents in dc circuits. We now apply this technique to ac circuits. In the ac case where the number-crunching involves complex numbers, we use j to represent the imaginary part of a complex number (unless it has been previously defined as something else), and the complex number $x + jy$ is expressed in MATLAB as `x+j*y`. Although we use j in defining a complex number, MATLAB will list the complex number using i.

Complex sources are expressed in rectangular form, and we use the fact that 360° equals 2 pi radians. For example, a source V = $10\angle 45°$ will be entered into MATLAB data as

$$V = 10\angle 45° = x+j*y$$

where the real and imaginary components are

```
x = 10*cos(45*pi/180)
  = 7.07
```

and

```
y = 10*sin(45*pi/180)
  = 7.07
```

When using MATLAB to determine the node voltages in ac circuits, we enter the Y matrix, the I vector, and then the solution equation

```
V = inv(Y)*I
```

as was done in the dc case. The following example will serve to illustrate the use of MATLAB in the solution of ac circuits.

LEARNING Example 7.17

Consider the network in Fig. 7.25. We wish to find all the node voltages in this network. The five simultaneous equations describing the node voltages are

$$V_1 = 12\angle 30^\circ$$

$$\frac{V_2 - V_1}{1} + \frac{V_2 - V_5}{j2} + \frac{V_2 - V_3}{-j1} = 0$$

$$\frac{V_3 - V_2}{-j1} + \frac{V_3 - V_5}{2} + \frac{V_3}{1} = 0$$

$$\frac{V_4 - V_1}{2} + \frac{V_4 - V_5}{1} + \frac{V_4}{-j1} = 0$$

$$\frac{V_5 - V_2}{j2} + \frac{V_5 - V_3}{2} + \frac{V_5}{-j1} + \frac{V_5 - V_4}{1} = 2\angle 45^\circ$$

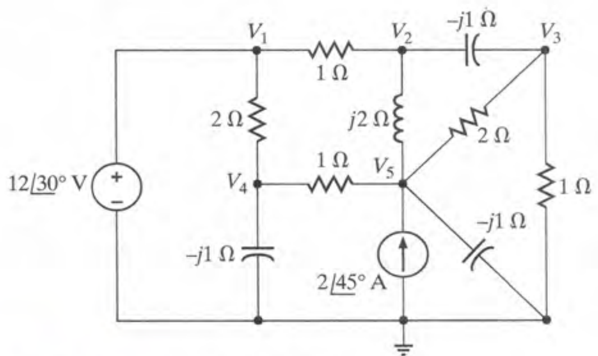


Figure 7.25 Circuit used in Example 7.17.

Expressing the equations in matrix form, we obtain

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 0 \\ -1 & 1 + j0.5 & -j1 & 0 & j0.5 \\ 0 & -j1 & 1.5 + j1 & 0 & -0.5 \\ -0.5 & 0 & 0 & 1.5 + j1 & -1 \\ 0 & j0.5 & -0.5 & -1 & 1.5 + j0.5 \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \\ V_3 \\ V_4 \\ V_5 \end{bmatrix} = \begin{bmatrix} 12\angle 30^\circ \\ 0 \\ 0 \\ 0 \\ 2\angle 45^\circ \end{bmatrix}$$

The following MATLAB data consist of the conversion of the sources to rectangular form, the coefficient matrix Y, the vector I, the solution equation V = inv(Y)*I, and the solution vector V.

```
>> x = 12*cos(30*pi/180)
x =
 10.3923
>> y = 12*sin(30*pi/180)
y =
 6.0000
>> v1 = x+j*y
v1 =
 10.3923 + 6.0000i
>> I2 = 2*cos(45*pi/180)
+ j*2*sin(45*pi/180)
I2 =
 1.4142 + 1.4142i
>> Y = [1 0 0 0 0; -1 1+j*0.5 -j*1 0
j*0.5; 0 -j*1 1.5+j*1 0 -0.5; -0.5 0 0
1.5+j*1 -1; 0 j*0.5 -0.5 -1 1.5+j*0.5]
Y =
Columns 1 through 4

 1.0000        0                 0                0
-1.0000 1.0000 +0.5000i          0 -1.0000i       0
      0        0 -1.0000i 1.5000 +1.0000i         0
-0.5000        0                 0         1.5000 +1.0000i
      0        0 +0.5000i -0.5000          -1.0000

Column 5
       0
       0  +0.5000i
 -0.5000
 -1.0000
  1.5000  +0.5000i
>> I = [v1 ; 0; 0; 0; I2]
I =
  10.3923  +6.0000i
        0
        0
        0
   1.4142  +1.4142i
>> V = inv(Y) *I
V =
  10.3923  +6.0000i
   7.0766  +2.1580i
   1.4038  +2.5561i
   3.7661  -2.9621i
   3.4151  -3.6771i
```

7.9 AC PSPICE Analysis Using Schematic Capture

INTRODUCTION In this chapter we found that an ac steady-state analysis is facilitated by the use of phasors. PSPICE can perform ac steady-state simulations, outputting magnitude and phase data for any voltage or current phasors of interest. Additionally, PSPICE can perform an AC SWEEP in which the frequency of the sinusoidal sources is varied over a user-defined range. In this case, the simulation results are the magnitude and phase of every node voltage and branch current as a function of frequency.

We will introduce five new *Schematics*/PSPICE topics in this section: defining AC sources, simulating at a single frequency, simulating over a frequency range, using the PROBE feature to create plots and, finally, saving and printing these plots. *Schematics* fundamentals such as getting parts, wiring, and changing part names and values were already discussed in Chapter 4. As in Chapter 4, we will use the following font conventions. Uppercase text refers to programs and utilities within PSPICE such as the AC SWEEP feature and PROBE graphing utility. All boldface text, whether upper or lowercase, denotes keyboard or mouse entries. For example, when placing a resistor into a circuit schematic, one must specify the resistor **VALUE** using the keyboard. The case of the boldface text matches that used in PSPICE.

DEFINING AC SOURCES Figure 7.26 shows the circuit we will simulate at a frequency of 60 Hz. We will continue to follow the flowchart shown in Fig. 4.20 in performing this simulation. Inductor and capacitor parts are in the ANALOG library and are called L and C, respectively. The AC source, VAC, is in the SOURCE library. Figure 7.27 shows the resulting *Schematics* diagram after wiring and editing the part's names and values.

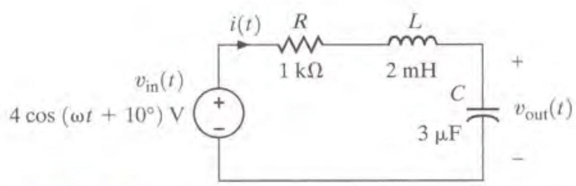


Figure 7.26
Circuit for ac simulation.

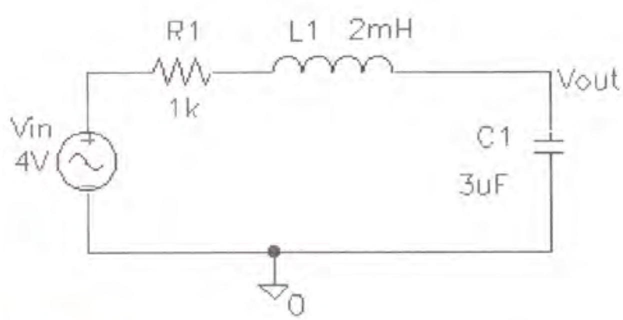


Figure 7.27
The Schematics diagram for the circuit in Figure 7.26.

To set up the AC source for simulation, double-click on the source symbol to open its ATTRIBUTES box, which is shown, after editing, in Fig. 7.28. As discussed in Chapter 6, we deselect the fields **Include Non-changeable Attributes** and **Include System-defined Attributes**. Each line in the ATTRIBUTES box is called an attribute of the ac source. Each attribute has a name and a value. The **DC** attribute is the dc value of the source for dc analyses. The **ACMAG** and **ACPHASE** attributes set the magnitude and phase of the phasor representing Vin for ac analyses. Each of these attributes defaults to zero. The value of the **ACMAG** attribute was set to 4 V when we created the schematic in Fig. 7.27. To set the **ACPHASE** attribute to 10°, click on the **ACPHASE** attribute line, enter 10 in the **Value** field, press **Save Attr** and **OK**. When the ATTRIBUTE box looks like that shown in Fig. 7.28, the source is ready for simulation.

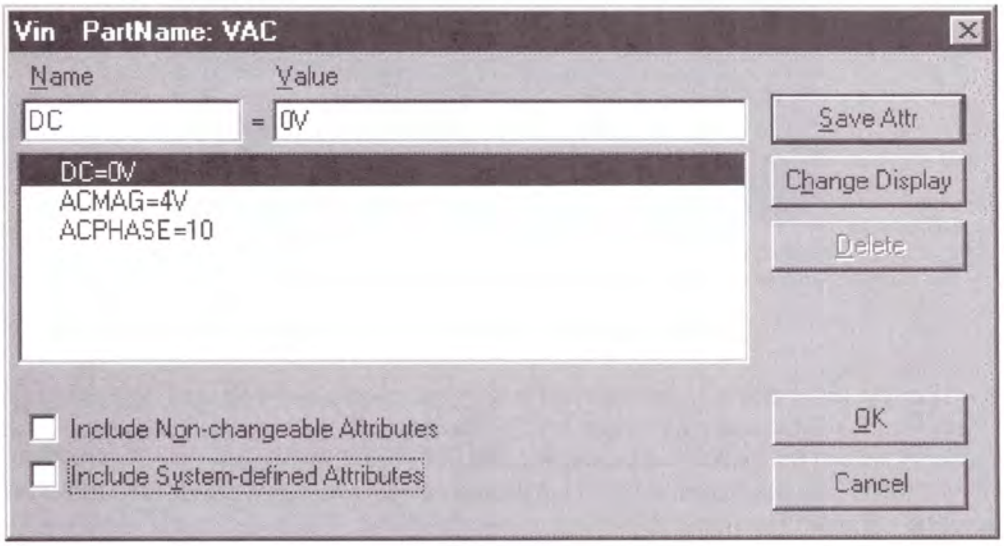


Figure 7.28
Setting the ac source phase angle.

SINGLE FREQUENCY AC SIMULATIONS Next, we must specify the frequency for simulation. This is done by selecting **Setup** from the **Analysis** menu. The SETUP box in Fig. 7.29 should appear. If we double-click on the text **AC Sweep**, the AC SWEEP AND NOISE ANALYSIS window in Fig. 7.30 will open. All of the fields in Fig. 7.30 have been set for our 60-Hz simulation.

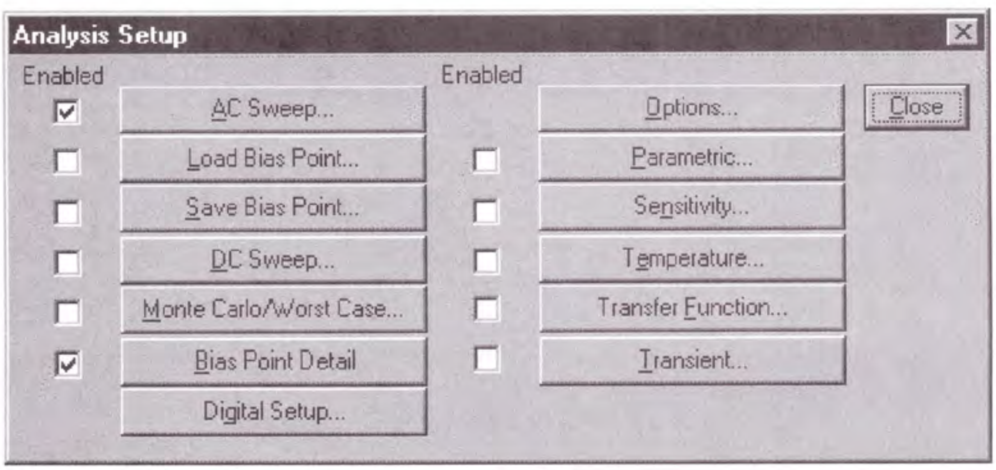


Figure 7.29
The ANALYSIS SETUP window.

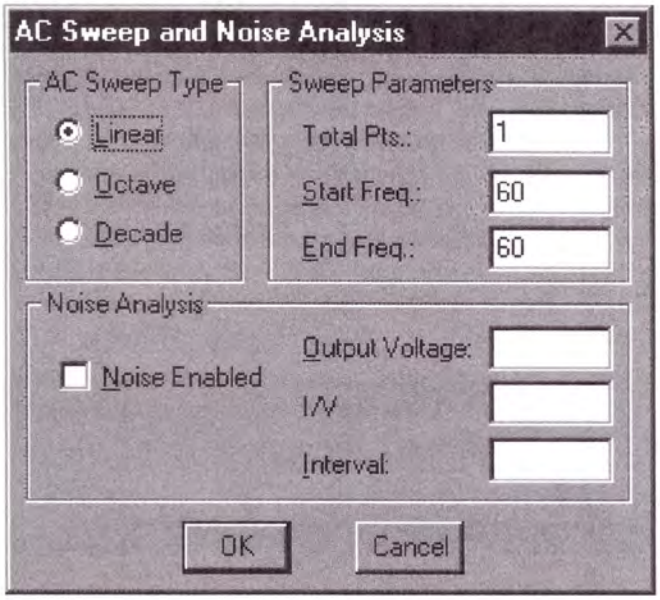


Figure 7.30
Setting the frequency range for a single frequency simulation.

Since the simulation will be performed at only one frequency, 60 Hz, graphing the simulation results is not an attractive option. Instead, we will write the magnitude and phase of the phasors **Vout** and **I** to the output file using the VPRINT1 and IPRINT parts, from the SPECIAL library, which have been added to the circuit diagram as shown in Fig. 7.31. The VPRINT1 part acts as a voltmeter, measuring the voltage at any single node with respect to the ground node. There is also a VPRINT2 part, which measures the voltage between any two nonreference nodes. Similarly, the IPRINT part acts as an ammeter and must be placed in series with the branch current of interest. By convention, current in the IPRINT part is assumed to exit from its negatively marked terminal. To find the clockwise loop current, as defined in Fig. 7.26, the IPRINT part has been flipped. The FLIP command is in the EDIT menu.

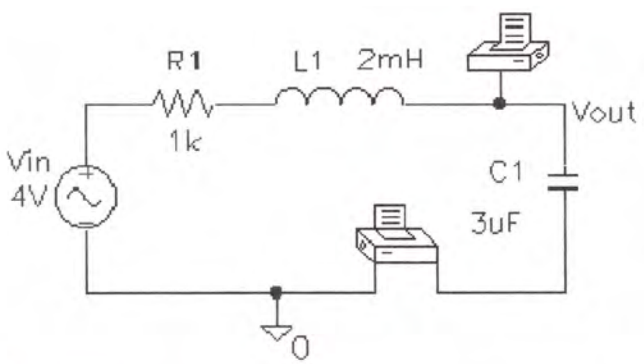


Figure 7.31
A Schematics diagram ready for single-frequency ac simulation.

After placing the VPRINT1 part, double-click on it to open its ATTRIBUTES box, shown in Fig. 7.32. The VPRINT1 part can be configured to meter the node voltage in any kind of simulation: dc, ac, or transient. Since an ac analysis was specified in the SETUP window in Fig. 7.29, the values of the **AC**, **MAG**, and **PHASE** attributes are set to **Y**, where Y stands for YES. This process is repeated for the IPRINT part. When we return to *Schematics*, the simulation is ready to run.

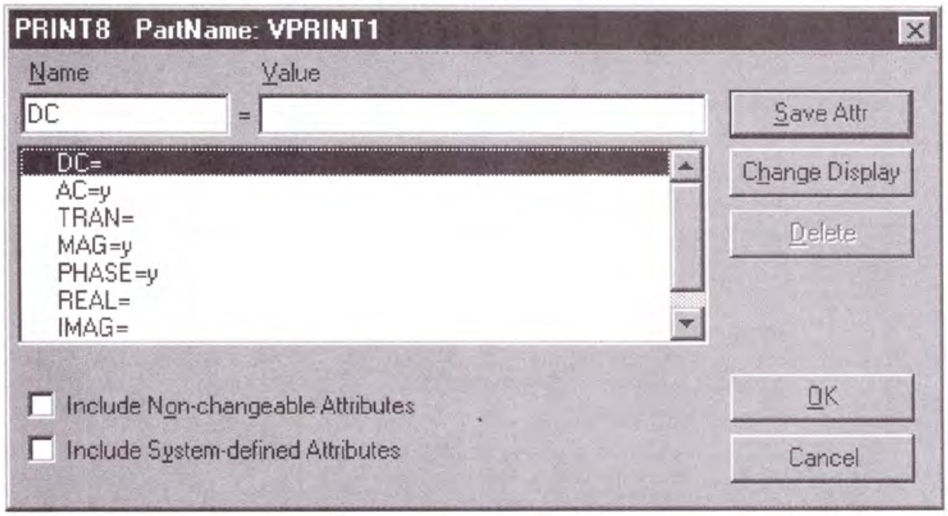


Figure 7.32
Setting the VPRINT1 measurements for ac magnitude and phase.

When an AC SWEEP is performed, PSPICE, unless instructed otherwise, will attempt to plot the results using the PROBE plotting program. To turn off this feature, select **Probe Setup** in the **Analysis** menu. When the PROBE SETUP window shown in Fig. 7.33 appears, select **Do Not Auto-Run Probe** and **OK**.

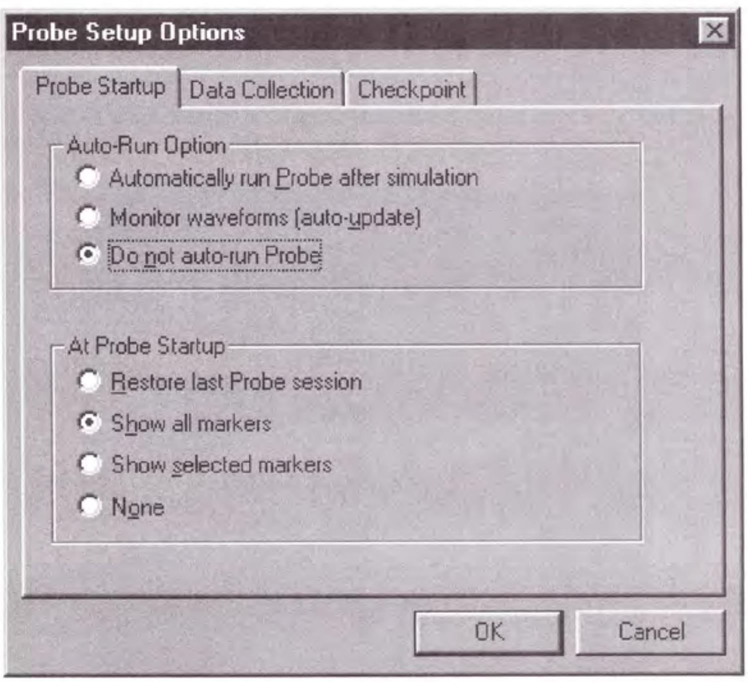


Figure 7.33
The PROBE SETUP window.

The circuit is simulated by selecting **Simulate** from the **Analysis** menu. Since the results are in the output file, select **Examine Output** from the **Analysis** menu to view the data. At the bottom of the file, we find the results as seen in Fig. 7.34: **Vout** $= 2.651 \angle -38.54^\circ$ V and **I** $= 2.998 \angle 51.46^\circ$ mA.

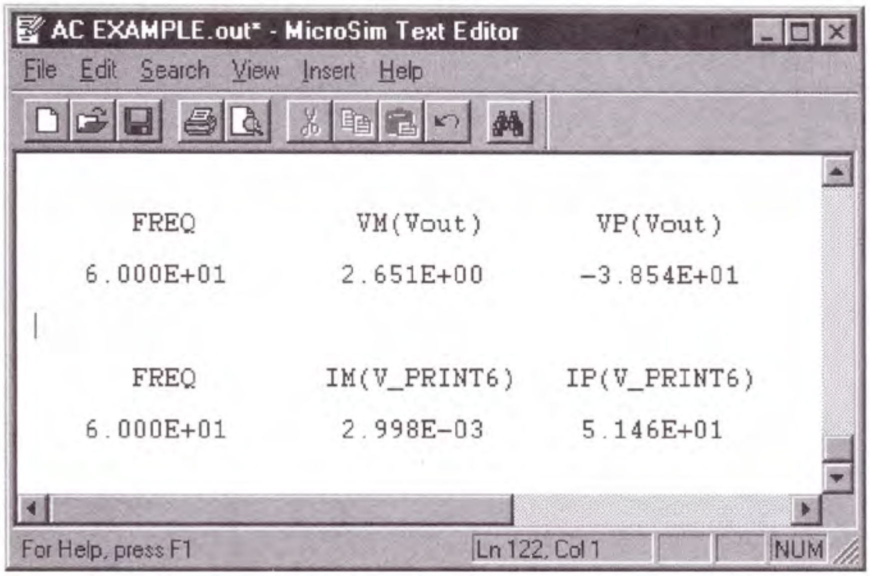


Figure 7.34 *Magnitude and phase data for* **Vout** *and* **I** *are at the bottom of the output file.*

VARIABLE FREQUENCY AC SIMULATIONS To sweep the frequency over a range, 1 Hz to 10 MHz, for example, return to the AC SWEEP AND NOISE ANALYSIS box shown in Fig. 7.30. Change the fields to those shown in Fig. 7.35. Since the frequency range is so large, we have chosen a log axis for frequency with 50 data points in each decade. We can now plot the data using the PROBE utility. This procedure requires two steps. First, we remove the VPRINT1 and IPRINT parts in Fig. 7.31. Second, we return to the PROBE SETUP window shown in Fig. 7.33, and select **Automatically Run Probe After Simulation** and **OK**.

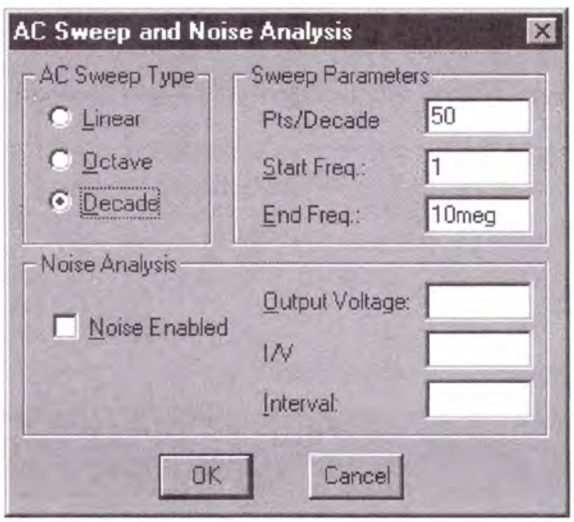


Figure 7.35 *Setting the frequency range for a swept frequency simulation.*

CREATING PLOTS IN PROBE When the PSPICE simulation is finished, the PROBE window shown in Fig. 7.36 will open. Actually, there are three windows here: the main display window, the output window, and the simulation status window. The latter two can be toggled off and on in the **View** menu. We will focus on the main display window, where the frequency is displayed on a log axis, as requested. To plot the magnitude and phase of **Vout**, select **Add** from the **Trace** menu. The ADD TRACES window shown in Fig. 7.37 will appear. To display the magnitude of **Vout**, we select **V(Vout)** from the left column. When a voltage or current is selected, the magnitude of the phasor will be plotted. Now the PROBE window should look like that shown in Fig. 7.38.

Before adding the phase to the plot, we note that **Vout** spans a small range—that is, 0 to 4 V. Since the phase change could span a much greater range, we will plot the phase on a second y-axis. From the **Plot** menu, select **Add Y Axis**. To add the phase to the plot, select **Add** from the **Trace** menu. On the right side of the ADD TRACES window in Fig. 7.37, scroll down to the entry, **P()**. Click on that, and then click on **V(Vout)** in the left column. The TRACE EXPRESSION line at the bottom of the window will contain the expression **P(V(Vout))**—the phase of **Vout**. Figure 7.39 shows the PROBE plot for both magnitude and phase of **Vout**.

To plot the current, **I**, on a new plot, we select **New** from the **Window** menu. Then, add the traces for the magnitude and phase of the current through R1 (PSPICE calls it **I(R1)**) using the process described above for plotting the magnitude and phase of **Vout**. The results are shown in Fig. 7.40.

The procedures for saving and printing PROBE plots, as well as the techniques for plot manipulation and data extraction, are described in Chapter 6.

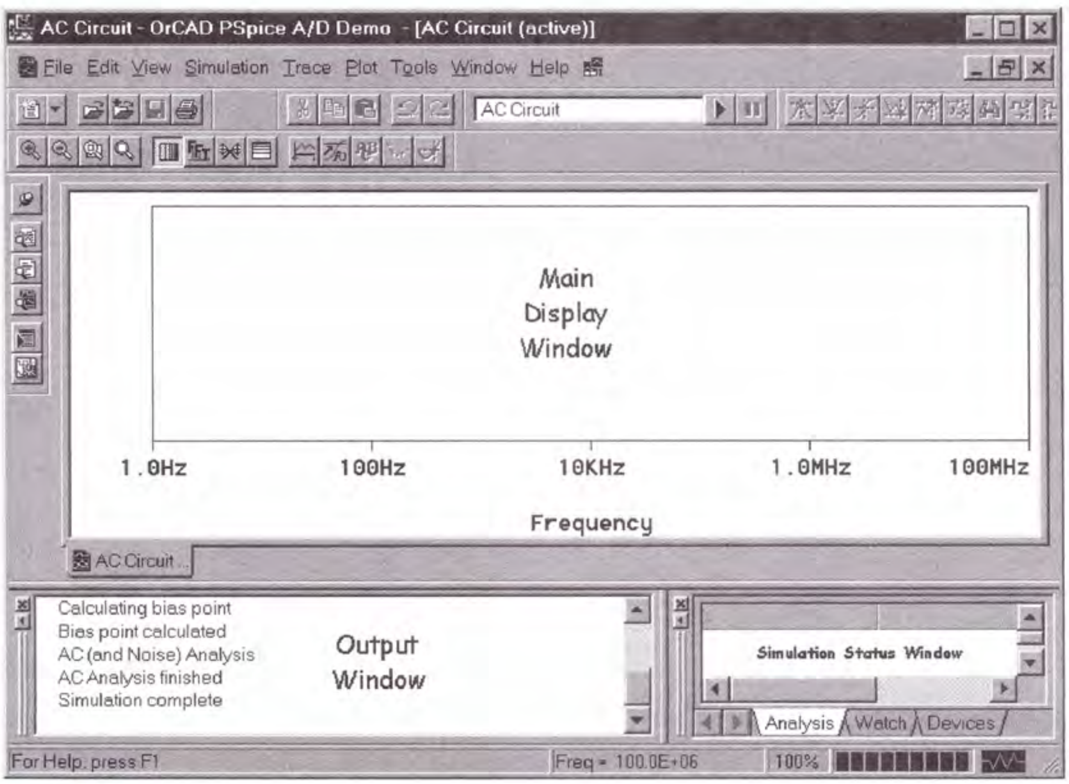


Figure 7.36 *The PROBE window.*

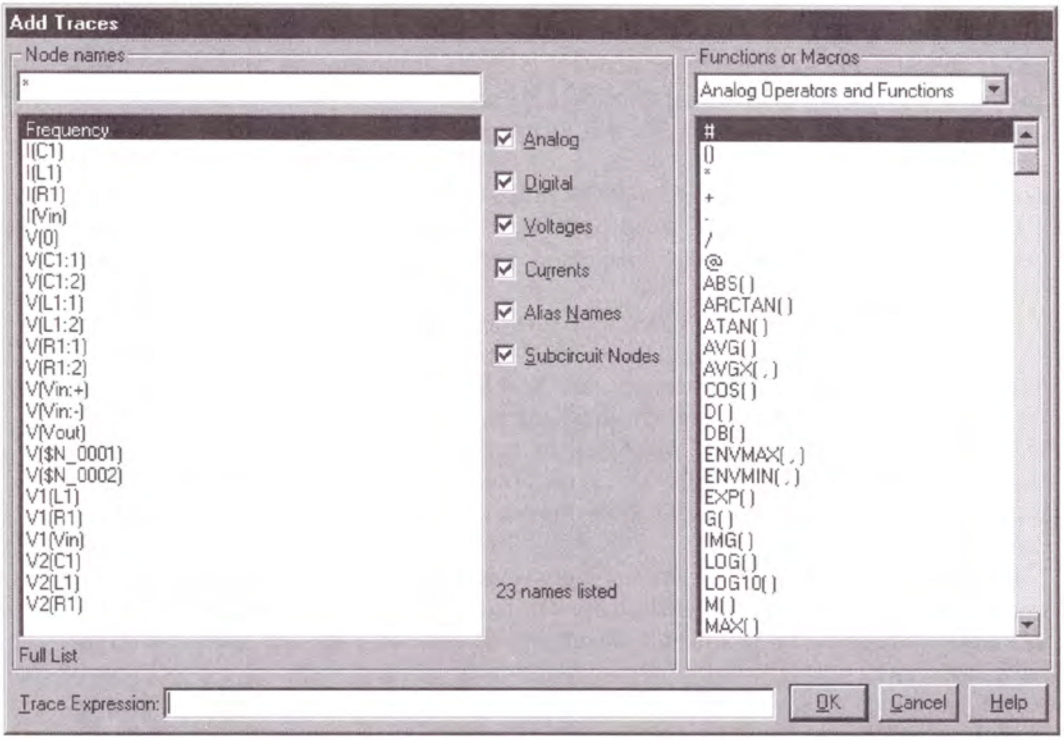


Figure 7.37 *The Add Traces window.*

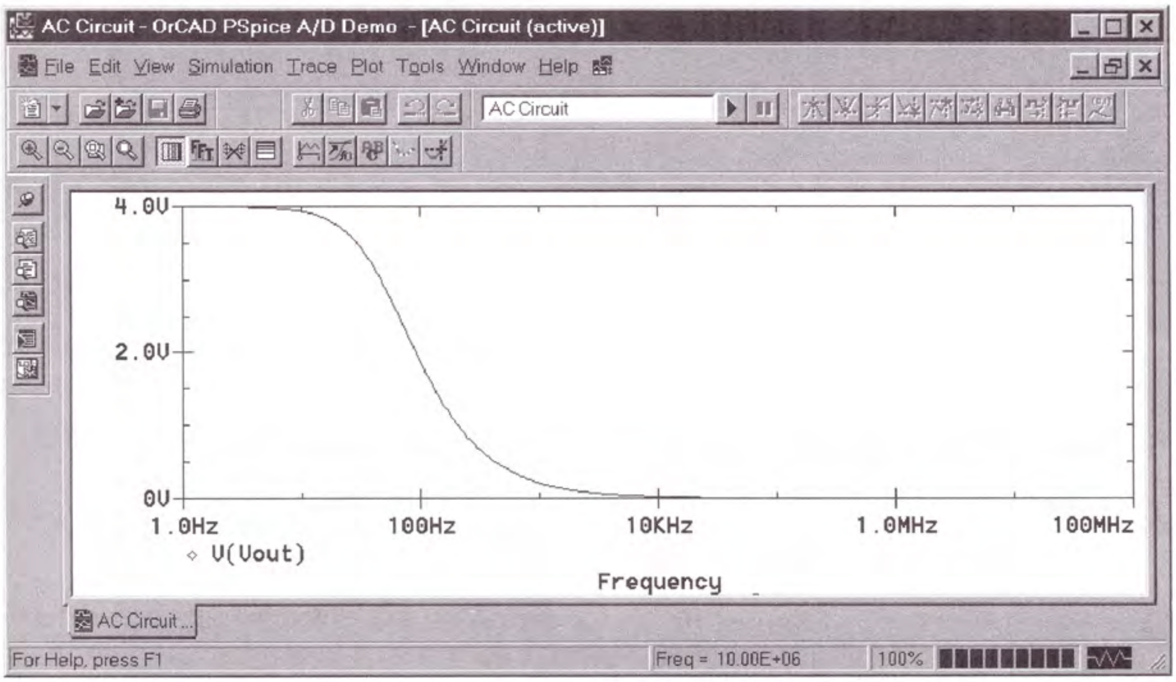


Figure 7.38 *The magnitude of* **Vout**.

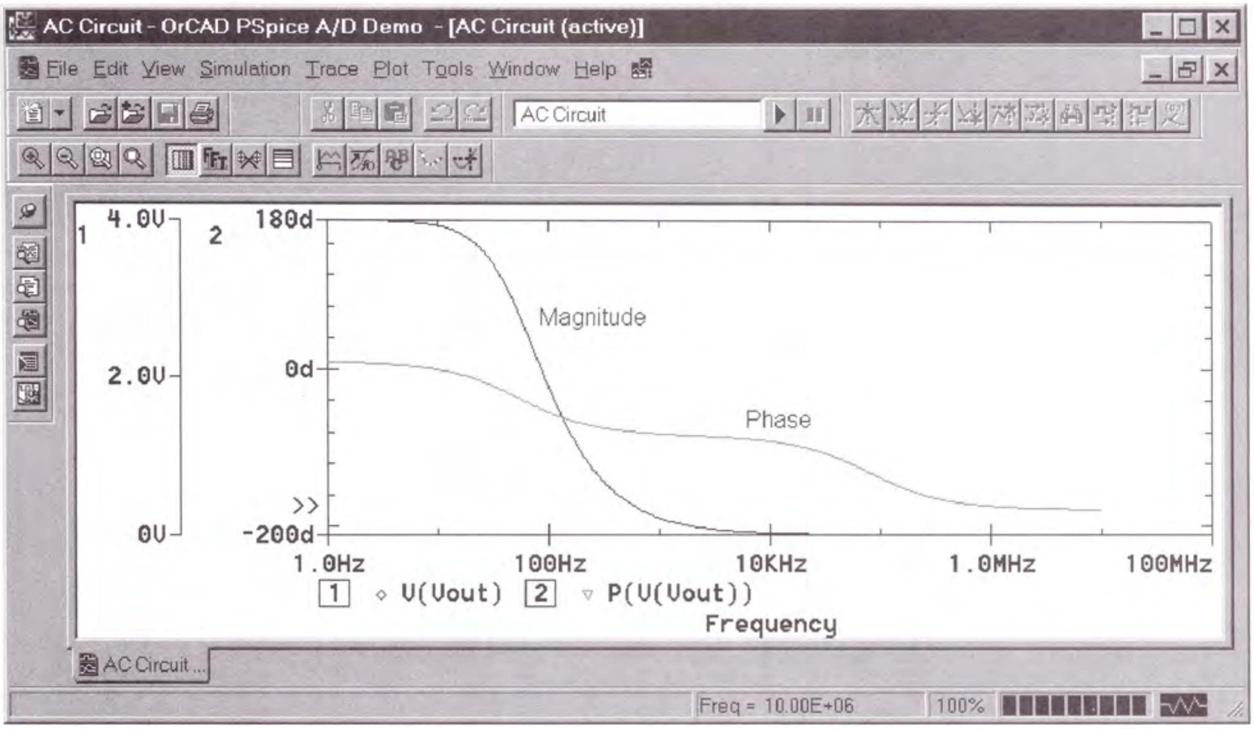


Figure 7.39 *The magnitude and phase of* **Vout**.

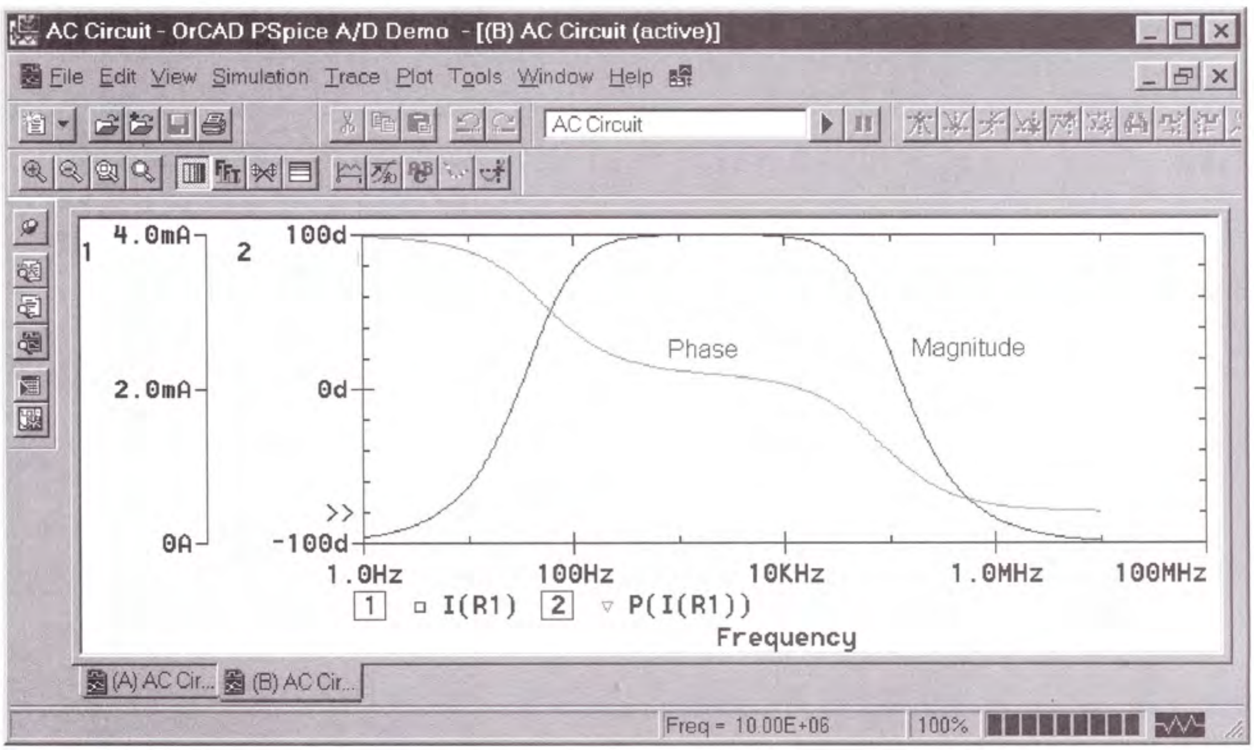


Figure 7.40 *The magnitude and phase of the current,* **I**.

LEARNING Example 7.18

Using the PSPICE *Schematics* editor, draw the circuit in Fig. 7.41, and use the PROBE utility to create plots for the magnitude and phase of $\mathbf{V}_{out}$ and $\mathbf{I}$. At what frequency does maximum $|\mathbf{I}|$ occur? What are the phasors $\mathbf{V}_{out}$ and $\mathbf{I}$ at that frequency?

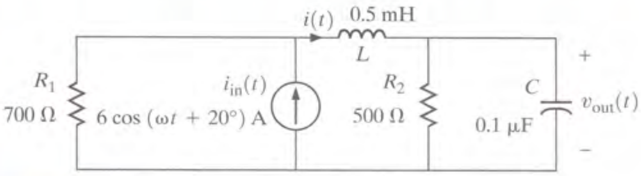


Figure 7.41 *Circuit for Example 7.18.*

SOLUTION The *Schematics* diagram for the simulation is shown in Fig. 7.42, where an AC SWEEP has been set up for the frequency range 10 Hz to 10 MHz at 100 data points per decade. Plots for $\mathbf{V}_{out}$ and $\mathbf{I}$ magnitudes and phases are given in Figs. 7.43a and b, respectively. From Fig. 7.43b we see that the maximum inductor current magnitude occurs at 31.42 kHz. At that frequency, the phasors of interest are $\mathbf{I} = 5.94\underline{/16.06°}$ A and $\mathbf{V}_{out} = 299.5\underline{/-68.15°}$ V.

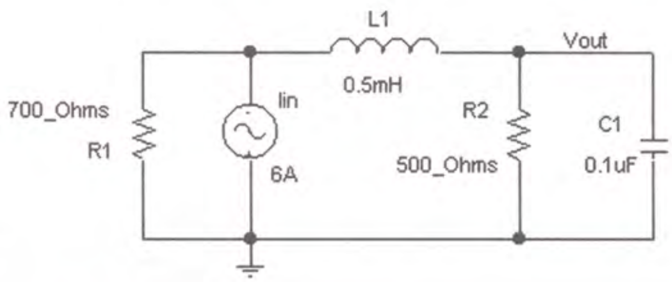
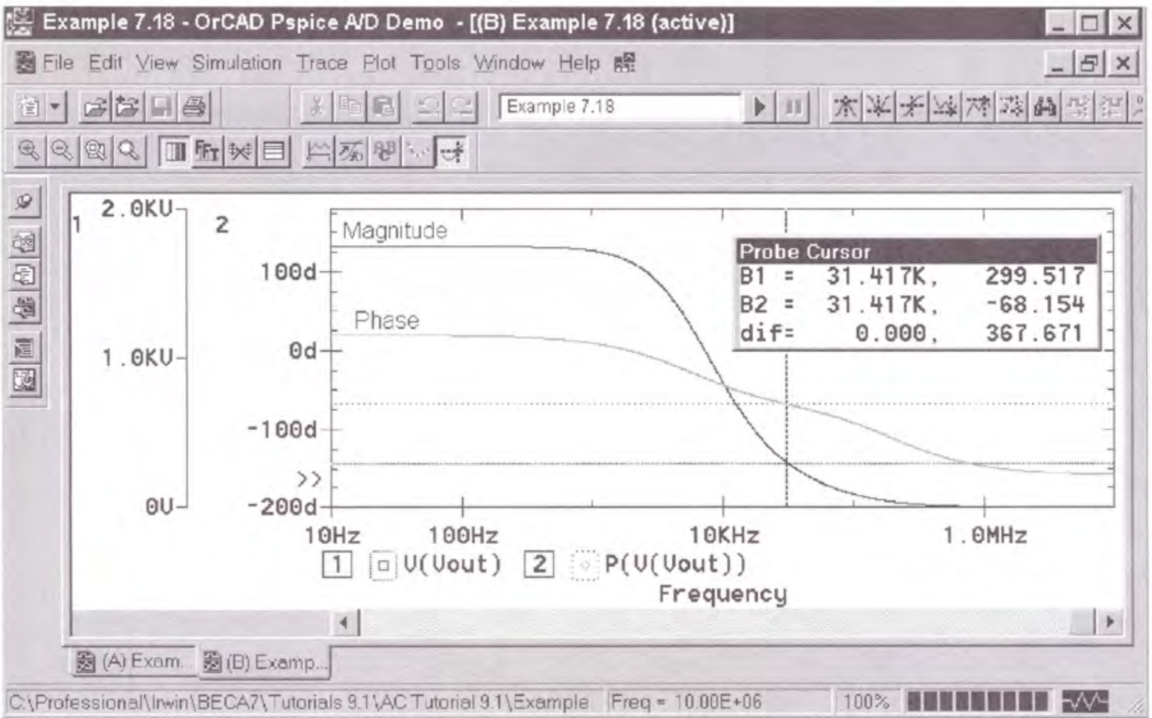
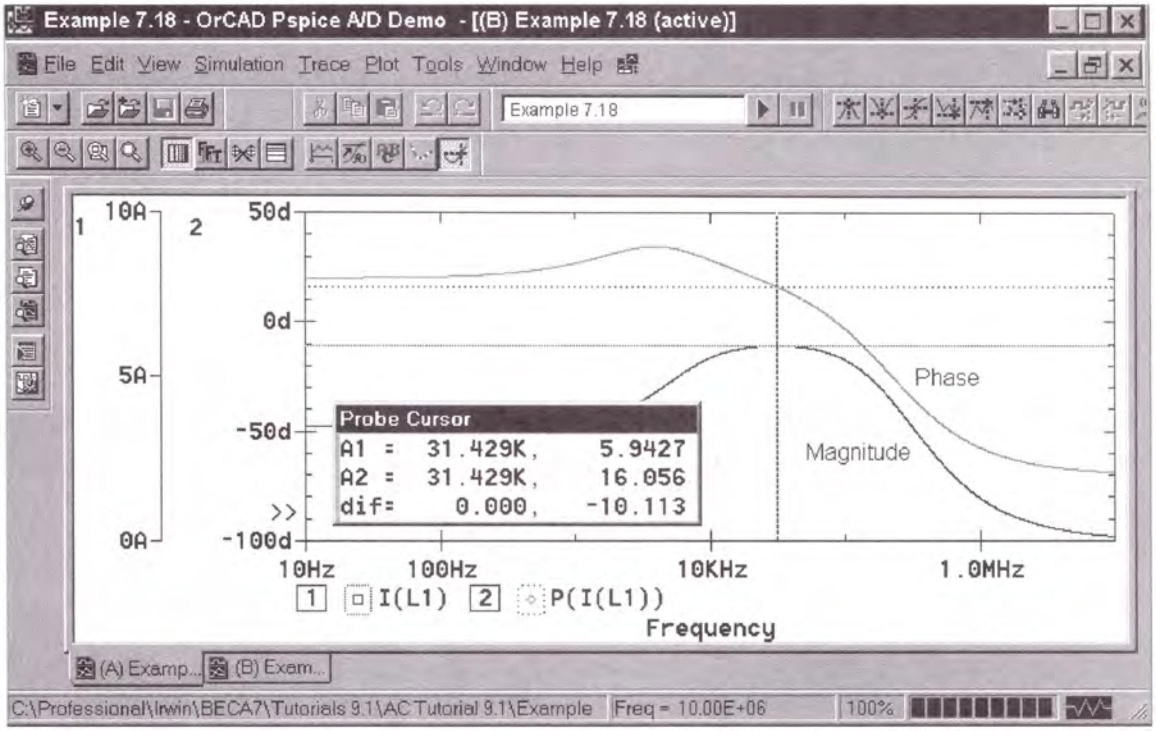


Figure 7.42
PSPICE Schematics diagram for Example 7.18.



(a)

Figure 7.43 *Simulation results for Example 7.18, (a)* **Vout** *and (b)* **I**.



(b)

Figure 7.43 (continued)

LEARNING Check

Summary

- ***The sinusoidal function definition*** The sinusoidal function $x(t) = X_M \sin(\omega t + \theta)$ has an amplitude of X_M, a radian frequency of ω, a period of $2\pi/\omega$, and a phase angle of θ.

- ***The phase lead and phase lag definitions*** If $x_1(t) = X_{M_1} \sin(\omega t + \theta)$ and $x_2(t) = X_{M_2} \sin(\omega t + \phi)$, $x_1(t)$ leads $x_2(t)$ by $\theta - \phi$ radians and $x_2(t)$ lags $x_1(t)$ by $\theta - \phi$ radians.

- ***The phasor definition*** The sinusoidal voltage $v(t) = V_M \cos(\omega t + \theta)$ can be written in exponential form as $v(t) = \text{Re}\left[V_M e^{j(\omega t + \theta)}\right]$ and in phasor form as $\mathbf{V} = V_M \underline{/\theta}$.

- ***The phase relationship in θ_v and θ_i for elements R, L, and C*** If θ_v and θ_i represent the phase angles of the voltage across and the current through a circuit element, then $\theta_i = \theta_v$ if the element is a resistor, θ_i lags θ_v by 90° if the element is an inductor, θ_i leads θ_v by 90° if the element is a capacitor.

- ***The impedances of R, L, and C*** Impedance, $\mathbf{Z}$, is defined as the ratio of the phasor voltage, $\mathbf{V}$, to the phasor current, $\mathbf{I}$, where $\mathbf{Z} = R$ for a resistor, $\mathbf{Z} = j\omega L$ for an inductor, and $\mathbf{Z} = 1/j\omega C$ for a capacitor.

- ***The phasor diagrams*** Phasor diagrams can be used to display the magnitude and phase relationships of various voltages and currents in a network.

Frequency domain analysis:

1. Represent all voltages, $v_i(t)$, and all currents, $i_j(t)$, as phasors and represent all passive elements by their impedance or admittance.
2. Solve for the unknown phasors in the frequency (ω) domain.
3. Transform the now-known phasors back to the time domain.

Solution techniques for ac steady-state problems:

Ohm's law
KCL and KVL
PSPICE
MATLAB
Nodal and loop analysis
Superposition and source exchange
Thévenin's Theorem
Norton's Theorem

Problems

For solutions and additional help on problems marked with ► go to www.wiley.com/college/irwin

SECTION 7.1

7.1 ► **Given $i(t) = 5\cos(400t - 120°)$ A, determine the period of the current and the frequency in hertz.**

7.2 Determine the relative position of the two sine waves.

$$v_1(t) = 12\sin(377t - 45°)$$
$$v_2(t) = 6\sin(377t + 675°)$$

7.3 Given the following currents

$$i_1(t) = 4\sin(377t - 10°)\text{ A}$$
$$i_2(t) = -2\cos(377t - 195°)\text{ A}$$
$$i_3(t) = -1\sin(377t - 250°)\text{ A}$$

compute the phase angle between each pair of currents.

7.4 Determine the phase angles by which $v_1(t)$ leads $i_1(t)$ and $v_1(t)$ leads $i_2(t)$, where

$$v_1(t) = 4\sin(377t + 25°)\text{ V}$$
$$i_1(t) = 0.05\cos(377t - 10°)\text{ A}$$
$$i_2(t) = -0.1\sin(377t - 75°)\text{ A}$$

SECTIONS 7.2, 7.3, 7.4, 7.5

7.5 ► **Calculate the current in the resistor in Fig. P7.5 if the voltage input is**

(a) $v_1(t) = 10\cos(377t + 180°)$ V.

(b) $v_2(t) = 12\sin(377t + 45°)$ V.

Give the answers in both the time and frequency domains.

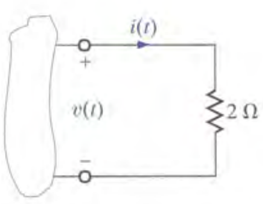


Figure P7.5

7.6 Calculate the current in the capacitor shown in Fig. P7.6 if the voltage input is

(a) $v_1(t) = 16\cos(377t - 22°)$ V.

(b) $v_2(t) = 8\sin(377t + 64°)$ V.

Give the answers in both the time and frequency domains.

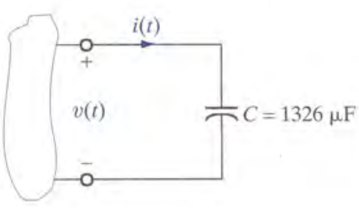


Figure P7.6

7.7 Calculate the current in the inductor shown in Fig. P7.7 if the voltage input is

(a) $v_1(t) = 24\cos(377t + 12°)$ V.

(b) $v_2(t) = 18\sin(377t + 48°)$ V.

Give the answers in both the time and frequency domains.

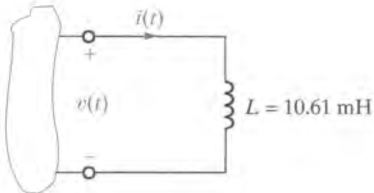


Figure P7.5

7.8 Find the frequency-domain impedance, Z, for the network in Fig. P7.8.

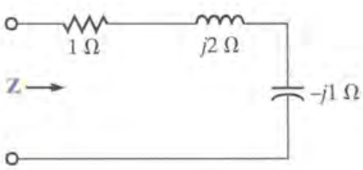


Figure P7.8

7.9 ► **Find the frequency-domain impedance, Z, as shown in Fig. P7.9.**

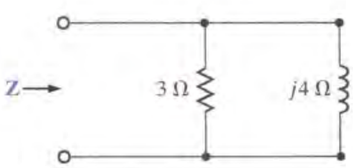


Figure P7.9

7.10 Find the frequency-domain impedance, **Z**, for the circuit in Fig. P7.10.

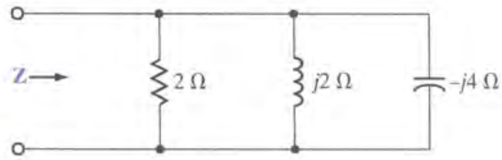


Figure P7.10

7.11 Find the frequency-domain impedance, **Z**, as shown in Fig. P7.11.

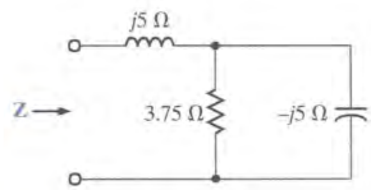


Figure P7.11

7.12 Find the frequency-domain impedance, **Z**, as shown in Fig. P7.12.

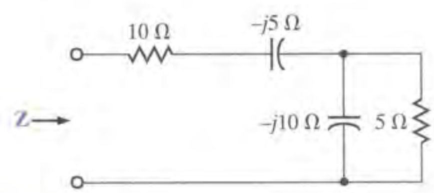


Figure P7.12

7.13 ► **In the network in Fig. P7.13, find $\mathbf{Z}(j\omega)$ at a frequency of 60 Hz.**

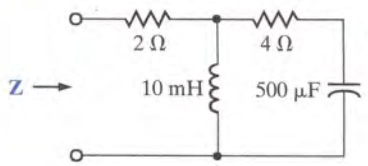


Figure P7.13

7.14 Calculate the equivalent impedance at terminals *A-B* in the circuit shown in Fig. P7.14.

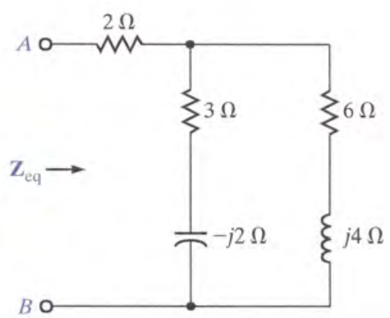


Figure P7.14

7.15 Find **Z** in the network in Fig. P7.15.

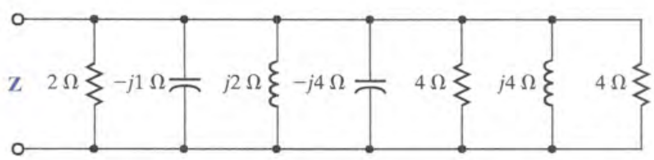


Figure P7.15

7.16 Find **Z** in the network in Fig. P7.16.

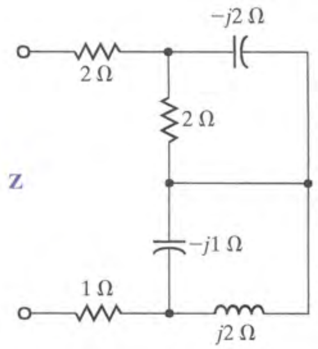


Figure P7.16

7.17 ► **Find Z in the network in Fig. P7.17.**

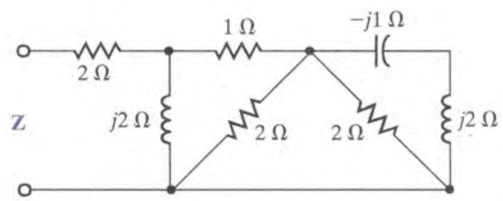


Figure P7.17

7.18 The impedance of the network in Fig. P7.18 is found to be purely real at $f = 60$ Hz. What is the value of C?

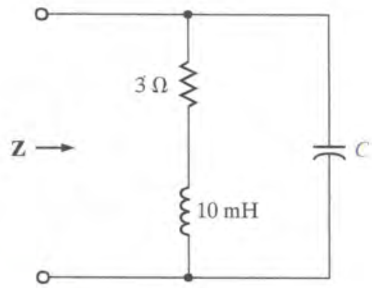


Figure P7.18

7.19 In the circuit shown in Fig. P7.19, determine the value of the inductance such that the current is in phase with the source voltage.

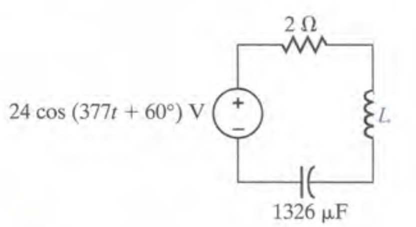


Figure P7.19

7.20 Draw the frequency-domain circuit and calculate $i(t)$ for the circuit shown in Fig. P7.20 if $v_s(t) = 10\cos(377t + 30°)$ V.

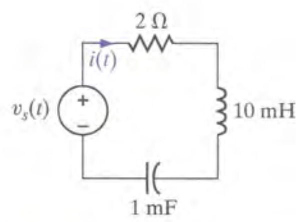


Figure P7.20

7.21 ► **Draw the frequency-domain circuit and calculate $v(t)$ for the circuit shown in Fig. P7.21 if $i_s(t) = 20\cos(377t + 120°)$ A.**

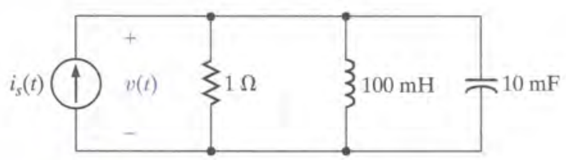


Figure P7.21

SECTION 7.6

7.22 The voltages $v_R(t)$, $v_L(t)$, and $v_C(t)$ in the circuit shown in Fig. P7.20 can be drawn as phasors in a phasor diagram. Show that $v_R(t) + v_L(t) + v_C(t) = v_S(t)$.

7.23 The currents $i_R(t)$, $i_L(t)$, and $i_C(t)$ in the circuit shown in Fig. P7.21 can be drawn as phasors in a phasor diagram. Show that $i_R(t) + i_L(t) + i_C(t) = i_S(t)$.

7.24 The voltages $v_R(t)$ and $v_L(t)$ in the circuit shown in Fig. P7.24 can be drawn as phasors in a phasor diagram. Use a phasor diagram to show that $v_R(t) + v_L(t) = v_S(t)$.

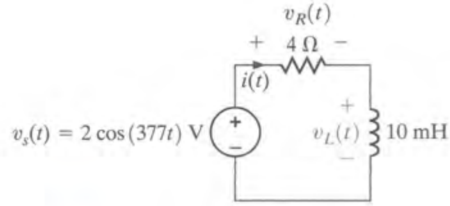


Figure P7.24

7.25 ► **The currents $i_R(t)$ and $i_C(t)$ in the circuit shown in Fig. P7.25 can be drawn as phasors in a phasor diagram. Use the diagram to show that $i_R(t) + i_C(t) = i_S(t)$.**

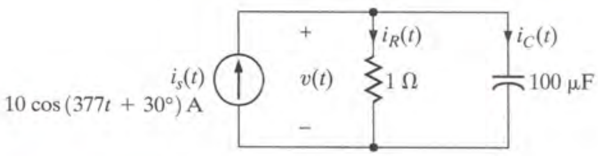


Figure P7.25

7.26 The currents $i_L(t)$ and $i_C(t)$ of the inductor and capacitor in the circuit shown in Fig. P7.26 can be drawn as phasors in a phasor diagram. Show that $i_L(t) + i_C(t) = i(t)$.

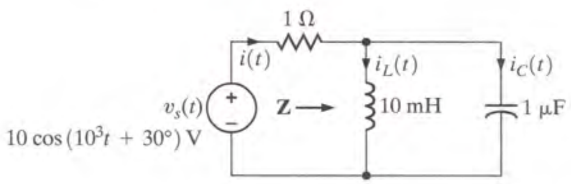


Figure P7.26

7.27 In the circuit shown in Fig. P7.27, determine the frequency at which $i(t)$ is in phase with $v_s(t)$.

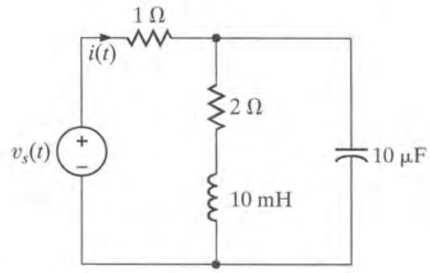


Figure P7.27

SECTION 7.7

7.28 Find the frequency-domain voltage $\mathbf{V}_o$, as shown in Fig. P7.28.

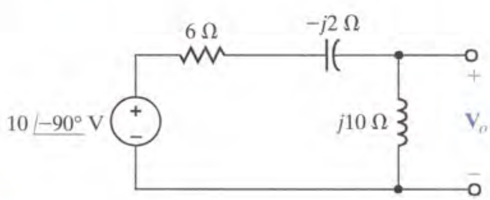


Figure P7.28

7.29 ► **Find the frequency-domain voltage $\mathbf{V}_o$, as shown in Fig. P7.29.**

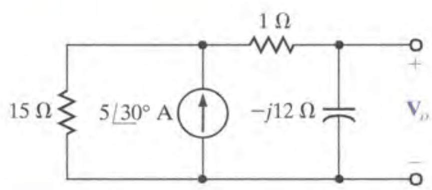


Figure P7.29

7.30 Find the frequency-domain current, $\mathbf{I}_o$, as shown in Fig. P7.30.

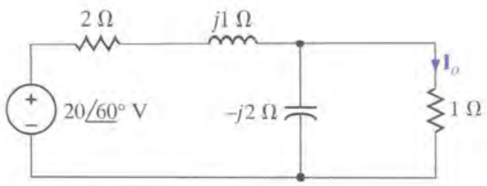


Figure P7.30

7.31 Find the frequency-domain voltage $\mathbf{V}_o$, as shown in Fig. P7.31.

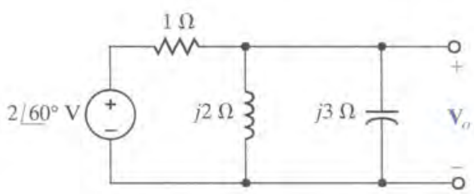


Figure P7.31

7.32 Draw the frequency-domain network and calculate $v_o(t)$ in the circuit shown in Fig. P7.32 if $i_S(t)$ is $100\cos(5000t + 8.13°)$ mA. Also, using a phasor diagram, show that $i_L(t) + i_R(t) = i_S(t)$.

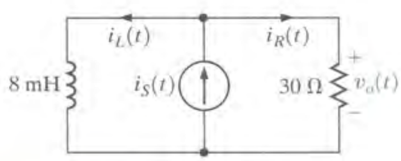


Figure P7.32

7.33 ► **Draw the frequency-domain network and calculate $v_o(t)$ in the circuit shown in Fig. P7.33 if $i_S(t)$ is $300\sin(10^4 t - 45°)$ mA. Also, using a phasor diagram, show that $i_1(t) + i_2(t) = i_S(t)$.**

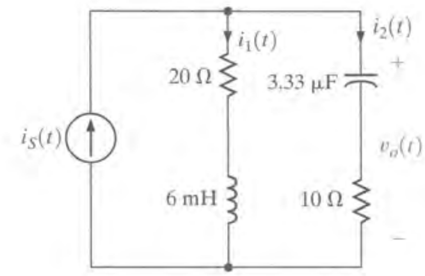


Figure P7.33

7.34 Find $\mathbf{I}_o$ in the network in Fig. P7.34.

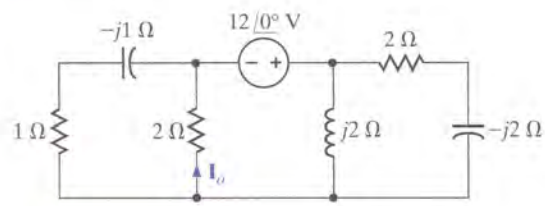


Figure P7.34

7.35 Find $\mathbf{I}_o$ in the network in Fig. P7.35.

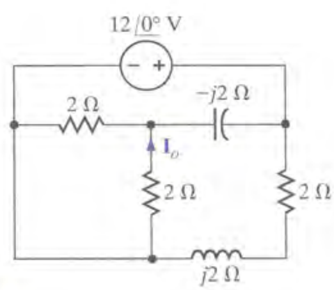


Figure P7.35

7.36 In the circuit in Fig. P7.36, if $\mathbf{V}_o = 4\underline{/45^\circ}$ V, find $\mathbf{I}_1$.

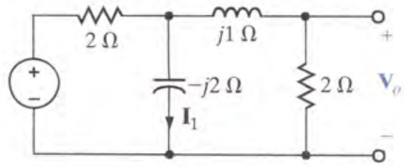


Figure P7.36

7.37 Find $\mathbf{V}_S$ in the network in Fig. P7.37, if $\mathbf{V}_1 = 4\underline{/0^\circ}$ V.

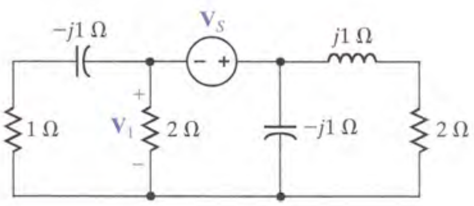


Figure P7.37

7.38 Find $\mathbf{V}_S$ in the network in Fig. P7.38 if $\mathbf{I}_1 = 2\underline{/0^\circ}$ A.

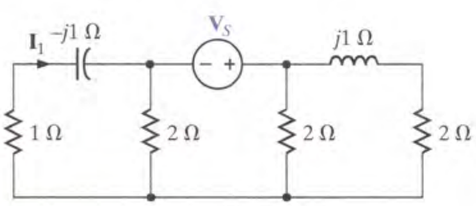


Figure P7.38

7.39 Find $\mathbf{V}_S$ in the network in Fig. P7.39 if $\mathbf{I}_o = 2\underline{/0^\circ}$ A.

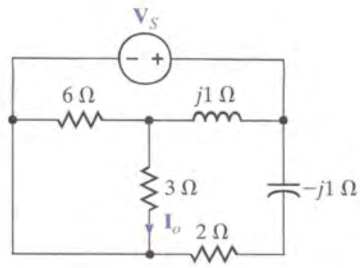


Figure P7.39

7.40 Find $\mathbf{I}_S$ in the network in Fig. P7.40 if $\mathbf{V}_1 = 8\underline{/0^\circ}$ V.

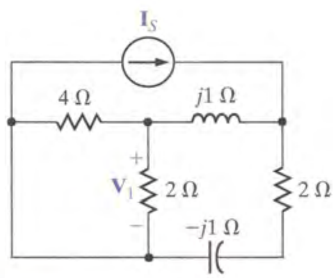


Figure P7.40

7.41 If $\mathbf{V}_1 = 4\underline{/0^\circ}$ V, find $\mathbf{I}_o$ in Fig. P7.41.

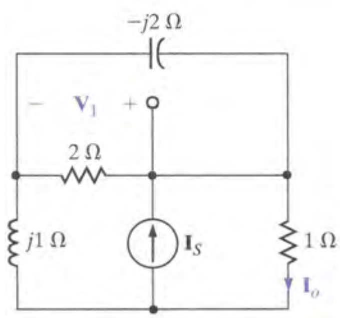


Figure P7.41

7.42 In the network in Fig. P7.42, $\mathbf{V}_o$ is known to be $4\,/45°$ V. Find $\mathbf{Z}$.

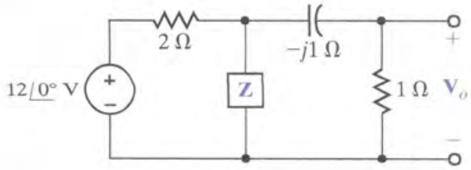


Figure P7.42

7.43 In the network in Fig. P7.43, $\mathbf{V}_1 = 2\,/45°$ V. Find $\mathbf{Z}$.

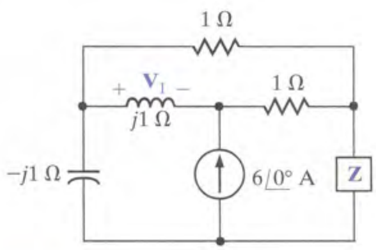


Figure P7.43

7.44 Determine $\mathbf{I}_o$ in the network in Fig. P7.44 if $\mathbf{I}_S = 12\,/0°$ A.

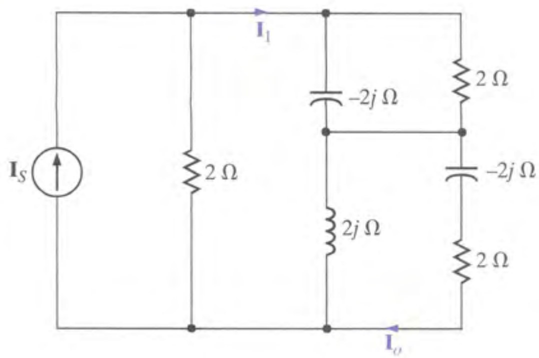


Figure P7.44

SECTION 7.8

7.45 Using nodal analysis, find $\mathbf{I}_o$ in the circuit in Fig. P7.45.

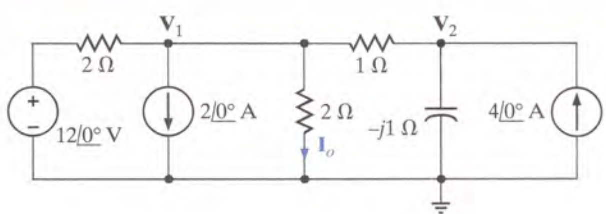


Figure P7.45

7.46 Use nodal analysis to find $\mathbf{V}_o$ in the circuit in Fig. P7.46.

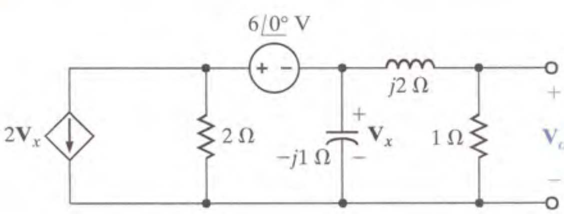


Figure P7.46

7.47 Find $\mathbf{V}_o$ in the network in Fig. P7.47 using nodal analysis.

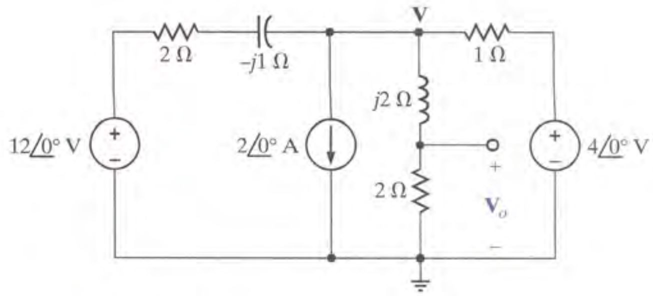


Figure P7.47

7.48 Use nodal analysis to determine $\mathbf{I}_o$ in the network in Fig. P7.48. In addition, solve the problem using MATLAB.

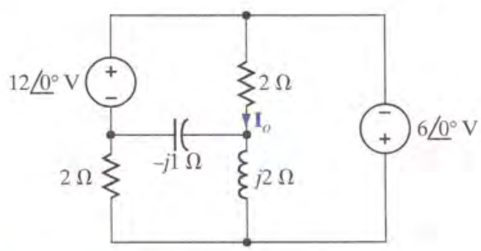


Figure P7.48

7.49 Find $\mathbf{V}_o$ in the network in Fig. P7.49.

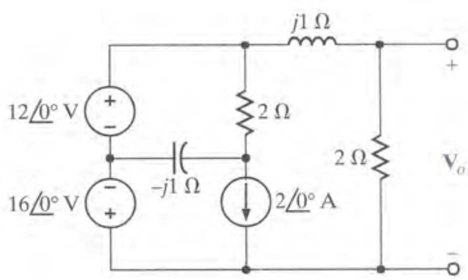


Figure P7.49

7.50 Find the voltage across the inductor in the circuit shown in Fig. P7.50 using nodal analysis.

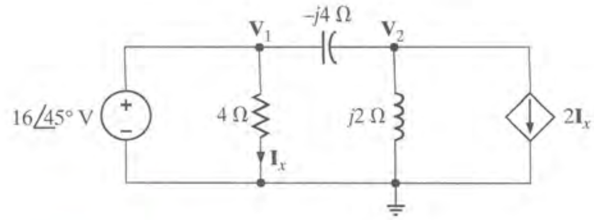


Figure P7.50

7.51 Use mesh analysis to find $\mathbf{V}_o$ in the circuit shown in Fig. P7.51.

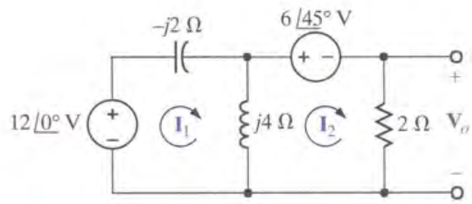


Figure P7.51

7.52 Use mesh analysis to find $\mathbf{V}_o$ in the circuit shown in Fig. P7.52.

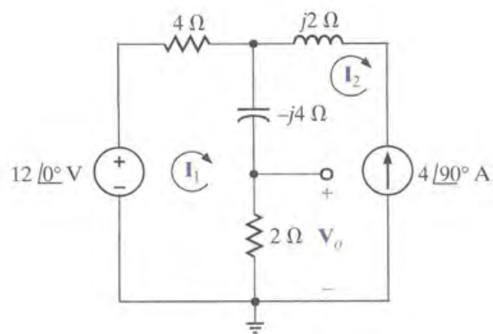


Figure P7.52

7.53 ► **Using both loop analysis and MATLAB, find $\mathbf{I}_o$ in the network in Fig. P7.53.**

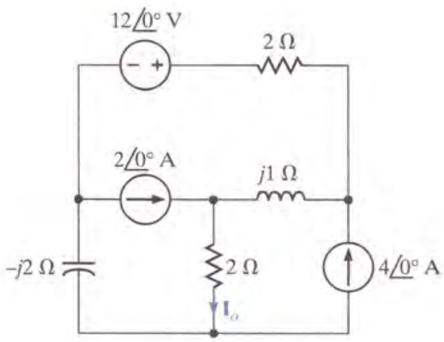


Figure P7.53

7.54 Find $\mathbf{V}_o$ in the network in Fig. P7.54.

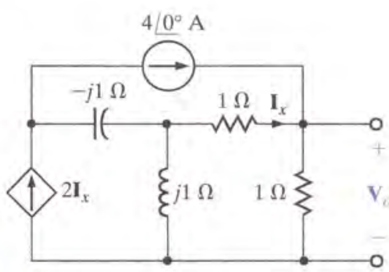


Figure P7.54

7.55 Find $\mathbf{V}_o$ in the network in Fig. P7.55.

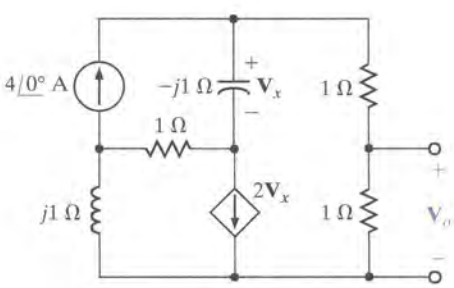


Figure P7.55

7.56 Use superposition to find $\mathbf{V}_o$ in the network in Fig. P7.56.

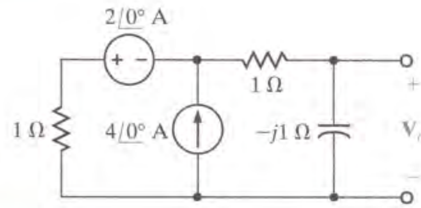


Figure P7.56

7.57 ► **Find $\mathbf{V}_o$ in the network in Fig. P7.57 using superposition.**

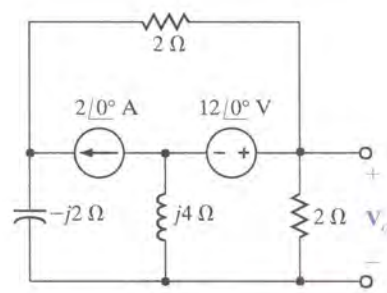


Figure P7.57

7.58 Using superposition, find $\mathbf{V}_o$ in the circuit in Fig. P7.58.

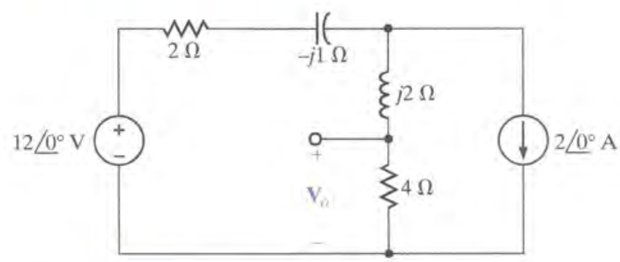


Figure P7.58

7.59 Use both superposition and MATLAB to determine $\mathbf{V}_o$ in the circuit in Fig. P7.59.

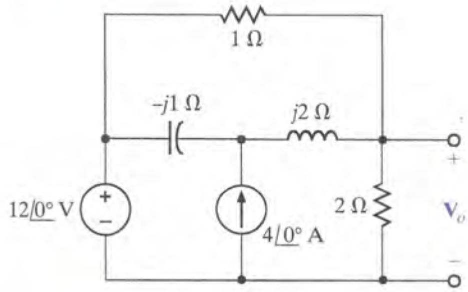


Figure P7.59

7.60 Use source exchange to determine $\mathbf{V}_o$ in the network in Fig. P7.60.

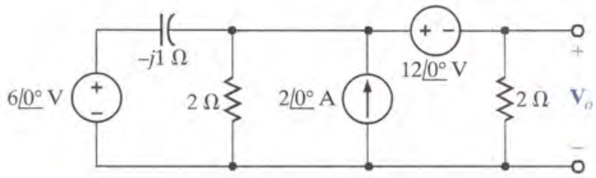


Figure P7.60

7.61 ► **Use source exchange to find the current $\mathbf{I}_o$ in the network in Fig. P7.61.**

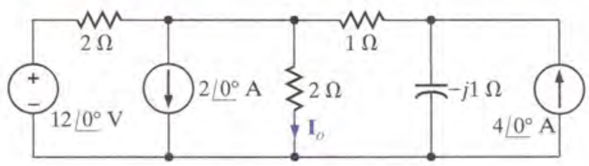


Figure P7.61

7.62 Use source transformation to determine $\mathbf{I}_o$ in the network in Fig. P7.45.

7.63 Use Thévenin's theorem to find $\mathbf{V}_o$ in the circuit in Fig. P7.63.

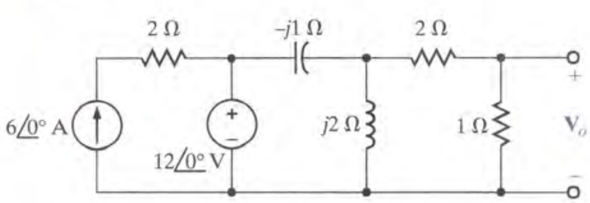


Figure P7.63

7.64 Using Thévenin's theorem, find $\mathbf{V}_o$ in the network in Fig. P7.51.

7.65 ► **Use Thévenin's theorem to find $\mathbf{V}_o$ in the circuit in Fig. P7.52.**

7.66 Solve Problem 7.49 using Thévenin's theorem.

7.67 Apply Thévenin's theorem twice to find $\mathbf{V}_o$ in the circuit in Fig. P7.67.

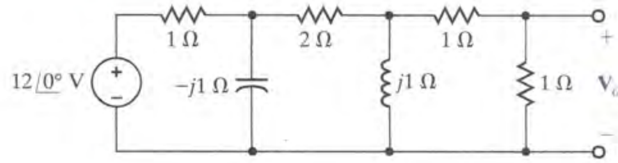


Figure P7.67

7.68 Find $\mathbf{V}_o$ in the network in Fig. P7.68 using Thévenin's theorem.

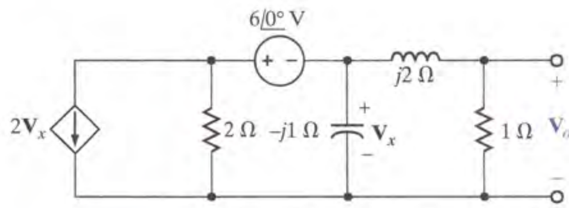


Figure P7.68

7.69 ► **Find the Thévenin's equivalent for the network in Fig. P7.69 at the terminals *A*-*B*.**

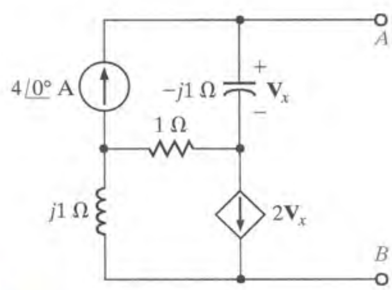


Figure P7.69

7.70 Find $\mathbf{V}_x$ in the circuit in Fig. P7.70 using Norton's theorem.

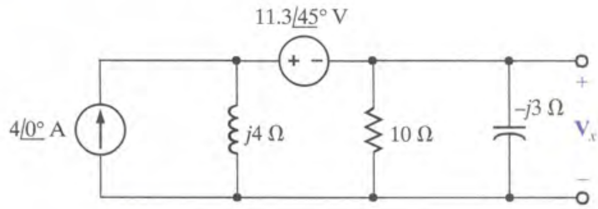


Figure P7.70

7.71 Find $\mathbf{I}_o$ in the network in Fig. P7.71 using Norton's theorem.

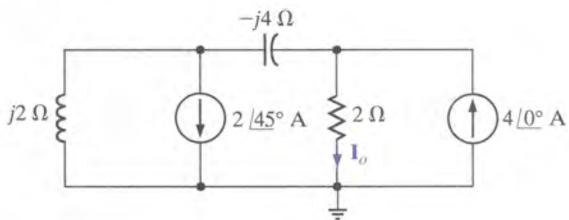


Figure P7.71

7.72 Apply both Norton's theorem and MATLAB to find $\mathbf{V}_o$ in the network in Fig. P7.72.

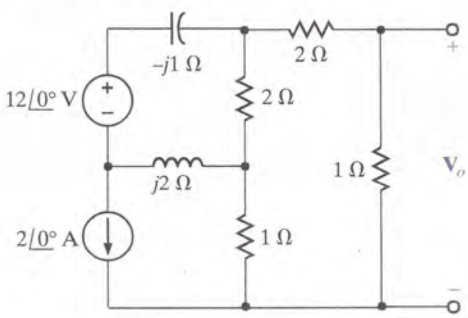


Figure P7.72

7.73 ► **Find $\mathbf{V}_o$ using Norton's theorem for the circuit in Fig. P7.73.**

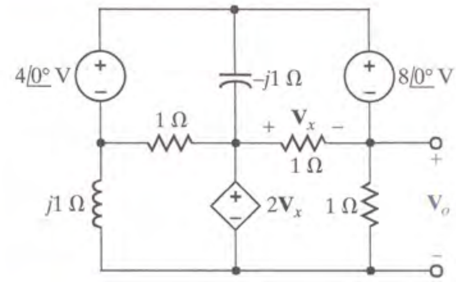


Figure P7.73

7.74 Use Norton's theorem to find $\mathbf{V}_o$ in the network in Fig. P7.74.

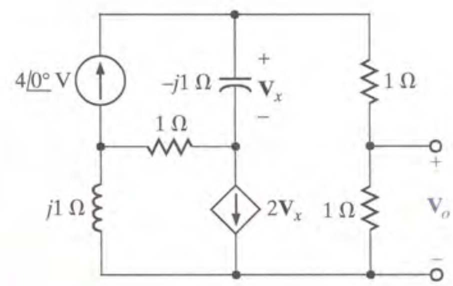


Figure P7.74

7.75 Use MATLAB to find the node voltages in the network in Fig. P7.75.

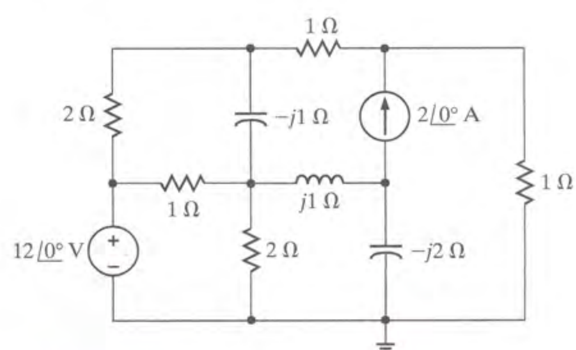


Figure P7.75

7.76 Using the PSPICE *Schematics* editor, draw the circuit in Fig. P7.76. At what frequency are the magnitudes of $i_C(t)$ and $i_L(t)$ equal?

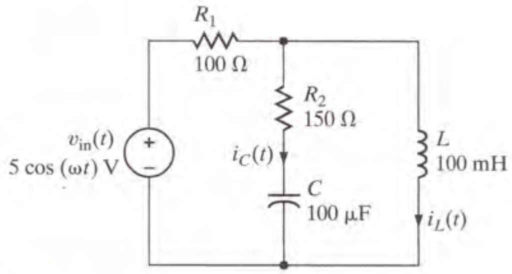


Figure P7.76

7.77 Using the PSPICE *Schematics* editor, draw the circuit in Fig. P7.77. At what frequency are the phases of $i_1(t)$ and $v_x(t)$ equal?

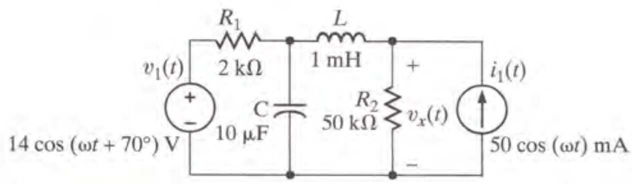


Figure P7.77

Typical Problems Found on the FE Exam

7FE-1 Find $\mathbf{V}_o$ in the network in Fig. 7PFE-1.

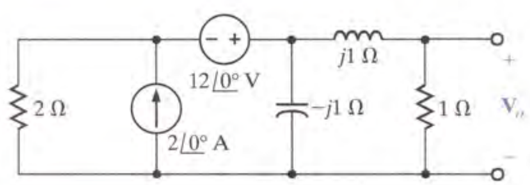


Figure 7PFE-1

7FE-2 Find $\mathbf{V}_o$ in the circuit in Fig. 7PFE-2.

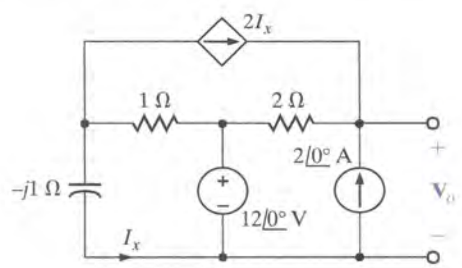


Figure 7PFE-2

7FE-3 Find $\mathbf{V}_o$ in the network in Fig. 7PFE-3.

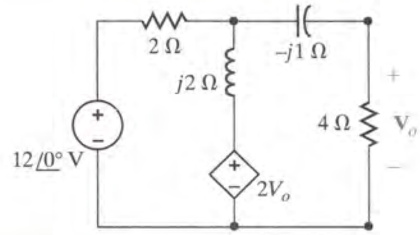


Figure 7PFE-3

7FE-4 Determine the midband (where the coupling capacitors can be ignored) gain of the single-stage transistor amplifier shown in Fig. 7PFE-4.

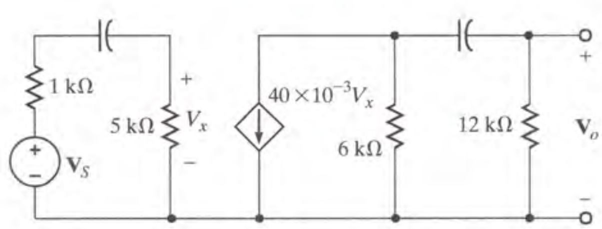


Figure 7PFE-4

Variable-Frequency Network Performance

8

In Chapter 7 we demonstrated that a network containing a capacitor and an inductor operated differently if the frequency was changed from the U.S. power grid frequency of 60 Hz to the aircraft frequency of 400 Hz. This phenomenon, although not surprising since the impedance of both these circuit elements is frequency dependent, indicates that if the frequency of the network sources is varied over some range, we can also expect the network to undergo variations in response to these changes in frequency.

Consider for a moment your stereo amplifier. The input signal contains sound waves of frequencies that range from soup to nuts; and yet, the amplifier must amplify each frequency component exactly the same amount in order to achieve perfect sound reproduction. Achieving perfect sound reproduction is a nontrivial task; and when you buy a very good amplifier, part of the price reflects the design necessary to achieve constant amplification over a wide range of frequencies.

In this chapter we examine the performance of electrical networks when excited from variable-frequency sources. Such effects are important in the analysis and design of networks such as filters, tuners, and amplifiers that have wide application in communication and control systems. Terminology and techniques for frequency-response analysis are introduced including standard plots (Bode plots) for describing network performance graphically. In particular, Bode plot construction and interpretation are discussed in detail.

The concept of resonance is introduced in reference to frequency selectivity and tuning. The various parameters used to define selectivity such as bandwidth, cutoff frequency, and quality factor are defined and discussed. Network scaling for both magnitude and phase is also presented.

Networks with special filtering properties are examined. Specifically, low-pass, high-pass, band-pass, and band-elimination filters are discussed. Techniques for designing active filters (containing op-amps) are presented.

LEARNING Goals

8.1 Variable Frequency-Response Analysis

In previous chapters we investigated the response of *RLC* networks to sinusoidal inputs. In particular, we considered 60-Hz sinusoidal inputs. In this chapter we allow the frequency of excitation to become a variable and evaluate network performance as a function of frequency. To begin, let us consider the effect of varying frequency on elements with which we are already quite familiar—the resistor, inductor, and capacitor. The frequency-domain impedance of the resistor shown in Fig. 8.1a is

$$\mathbf{Z}_{\mathrm{R}} = R = R\angle 0^{\circ}$$

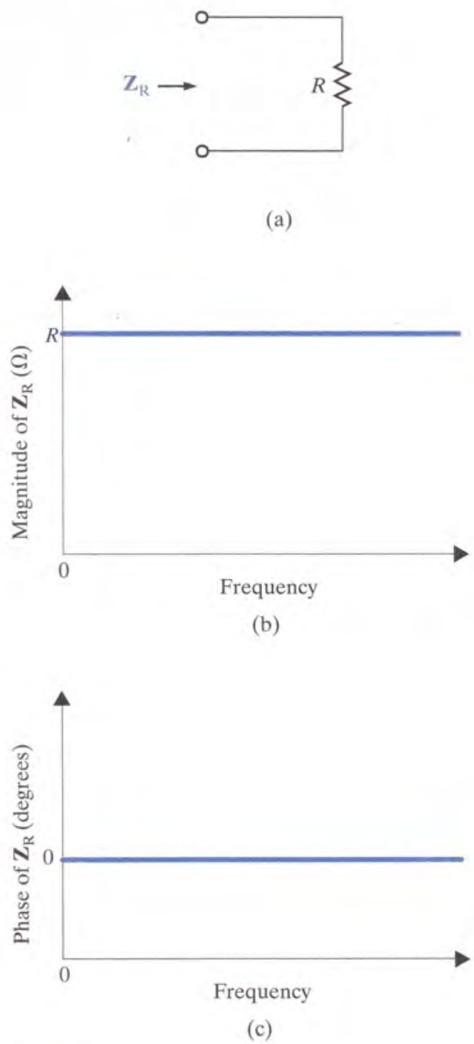


Figure 8.1
Frequency-dependent impedance of a resistor.

The magnitude and phase are constant and independent of frequency. Sketches of the magnitude and phase of $\mathbf{Z}_R$ are shown in Figs. 8.1b and c. Obviously, this is a very simple situation.

For the inductor in Fig. 8.2a, the frequency-domain impedance $\mathbf{Z}_L$ is

$$\mathbf{Z}_L = j\omega L = \omega L \underline{/90^\circ}$$

The phase is constant at 90° but the magnitude of $\mathbf{Z}_L$ is directly proportional to frequency. Figures 8.2b and c show sketches of the magnitude and phase of $\mathbf{Z}_L$ versus frequency. Note that at low frequencies the inductor's impedance is quite small. In fact, at dc, $\mathbf{Z}_L$ is zero and the inductor appears as a short circuit. Conversely, as frequency increases, the impedance also increases.

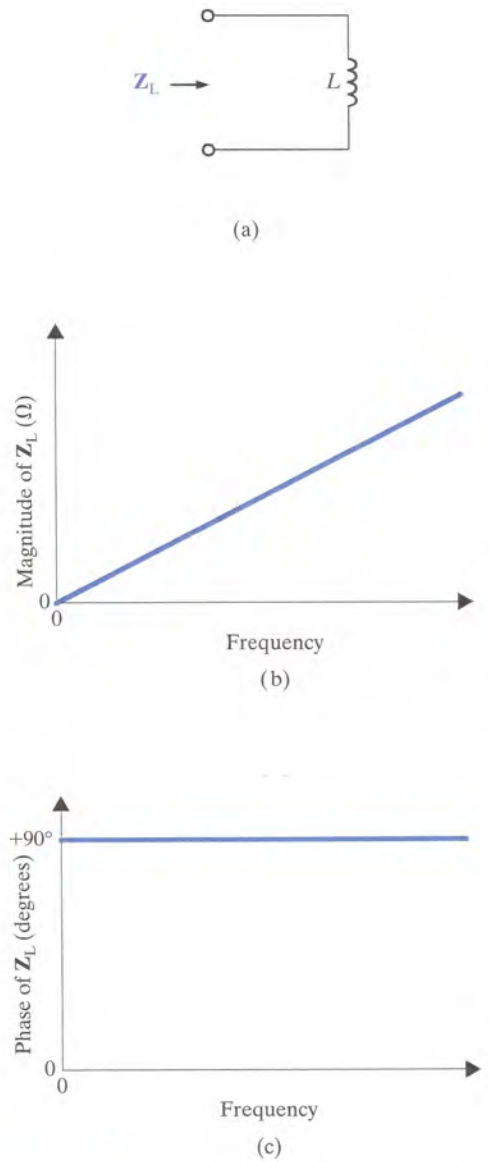


Figure 8.2
Frequency-dependent impedance of an inductor.

Next consider the capacitor of Fig. 8.3a. The impedance is

$$\mathbf{Z}_C = \frac{1}{j\omega C} = \frac{1}{\omega C} \angle -90^\circ$$

Once again the phase of the impedance is constant but now the magnitude is inversely proportional to frequency, as shown in Figs. 8.3b and c. Note that the impedance approaches infinity, or an open circuit, as ω approaches zero and $\mathbf{Z}_C$ approaches zero as ω approaches infinity.

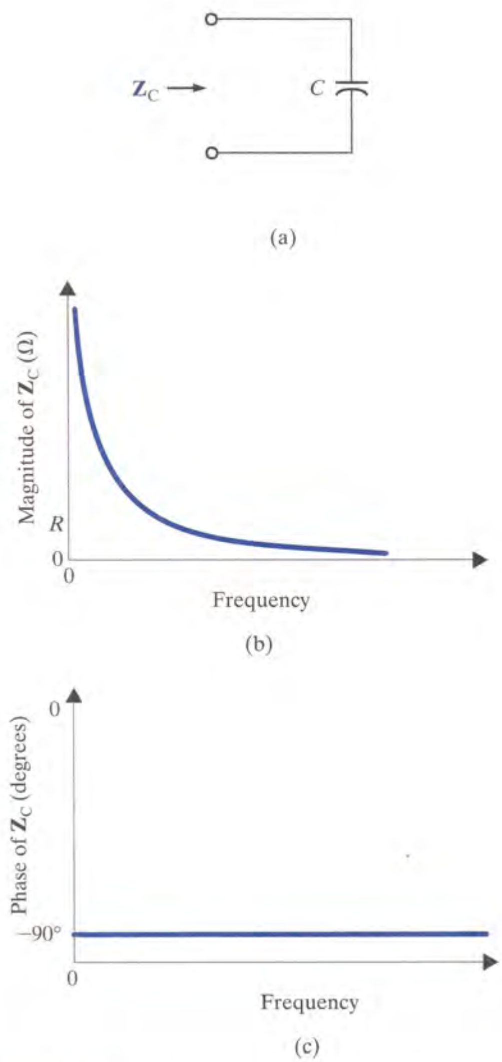


Figure 8.3
Frequency-dependent impedance of a capacitor.

Now let us investigate a more complex circuit: the *RLC* series network in Fig. 8.4a. The equivalent impedance is

$$\mathbf{Z}_{eq} = R + j\omega L + \frac{1}{j\omega C}$$

or

$$\mathbf{Z}_{eq} = \frac{(j\omega)^2 LC + j\omega RC + 1}{j\omega C}$$

Sketches of the magnitude and phase of this function are shown in Figs. 8.4b and c.

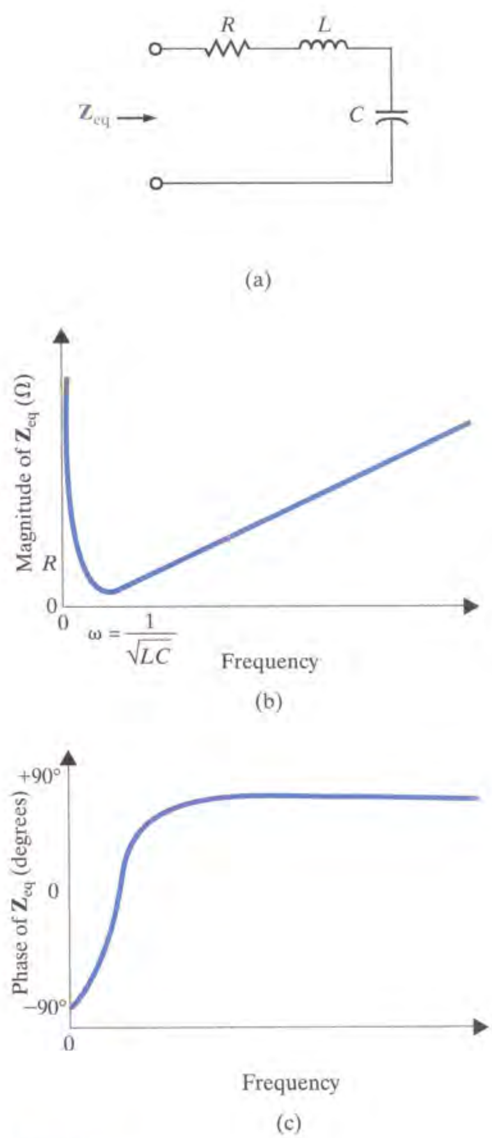


Figure 8.4
Frequency-dependent impedance of an RLC series network.

Note that at very low frequencies, the capacitor appears as an open circuit and, therefore, the impedance is very large in this range. At high frequencies, the capacitor has very little effect and the impedance is dominated by the inductor, whose impedance keeps rising with frequency.

As the circuits become more complicated, the equations become more cumbersome. In an attempt to simplify them, let us make the substitution $j\omega = s$. With this substitution, the expression for $\mathbf{Z}_{eq}$ becomes

$$\mathbf{Z}_{eq} = \frac{s^2LC + sRC + 1}{sC}$$

If we review the four circuits we investigated thus far, we will find that in every case the impedance is the ratio of two polynomials in s and is of the general form

$$\mathbf{Z}(s) = \frac{N(s)}{D(s)} = \frac{a_m s^m + a_{m-1}s^{m-1} + \cdots + a_1 s + a_0}{b_n s^n + b_{n-1}s^{n-1} + \cdots + b_1 s + b_0} \qquad \mathbf{8.1}$$

where $N(s)$ and $D(s)$ are polynomials of order m and n, respectively. An extremely important aspect of Eq. (8.1) is that it holds not only for impedances but also for all voltages, currents, admittances, and gains in the network. The only restriction is that the values of all circuit elements (resistors, capacitors, inductors, and dependent sources) must be real numbers.

Let us now demonstrate the manner in which the voltage across an element in a series *RLC* network varies with frequency.

LEARNING Example 8.1

Consider the network in Fig. 8.5a. We wish to determine the variation of the output voltage as a function of frequency over the range from 0 to 1 kHz.

SOLUTION Using voltage division, the output can be expressed as

$$\mathbf{V}_o = \left(\frac{R}{R + j\omega L + \dfrac{1}{j\omega C}}\right)\mathbf{V}_S$$

or, equivalently,

$$\mathbf{V}_o = \left(\frac{j\omega CR}{(j\omega)^2 LC + j\omega CR + 1}\right)\mathbf{V}_S$$

Using the element values, the equation becomes

$$\mathbf{V}_o = \left(\frac{(j\omega)(37.95 \times 10^{-3})}{(j\omega)^2(2.53 \times 10^{-4}) + j\omega(37.95 \times 10^{-3}) + 1}\right) 10\underline{/0^\circ}$$

MATLAB can be effectively employed here to determine the magnitude and phase of the voltages as a function of frequency. The program required to generate the magnitude and phase plots consists of three simple statements. If, in general, the function is of the form

$$\mathbf{V}_o(s) = \frac{a_m s^m + a_{m-1}s^{m-1} + \cdots + a_1 s + a_0}{b_n s^n + b_{n-1}s^{n-1} + \cdots + b_1 s + b_0}$$

then the plots, generated by the statements (in the following exact format)

```
>>B = [a_m  a_m-1 ........... a_1  a_0]
>>A = [b_n  b_n-1 ........... b_1  b_0]
>>freqs(B,A)
```

will produce the magnitude and phase characteristics as a function of frequency. The resultant characteristics are semilog plots in which the frequency is displayed on the log axis. The program for the output voltage in this example is

```
>>B = [ 37.95e-3 0];
>>A = [ 2.53e-4 37.95e-3 1];
>>freqs(B,A)
```

and the results are shown in Fig. 8.5b.

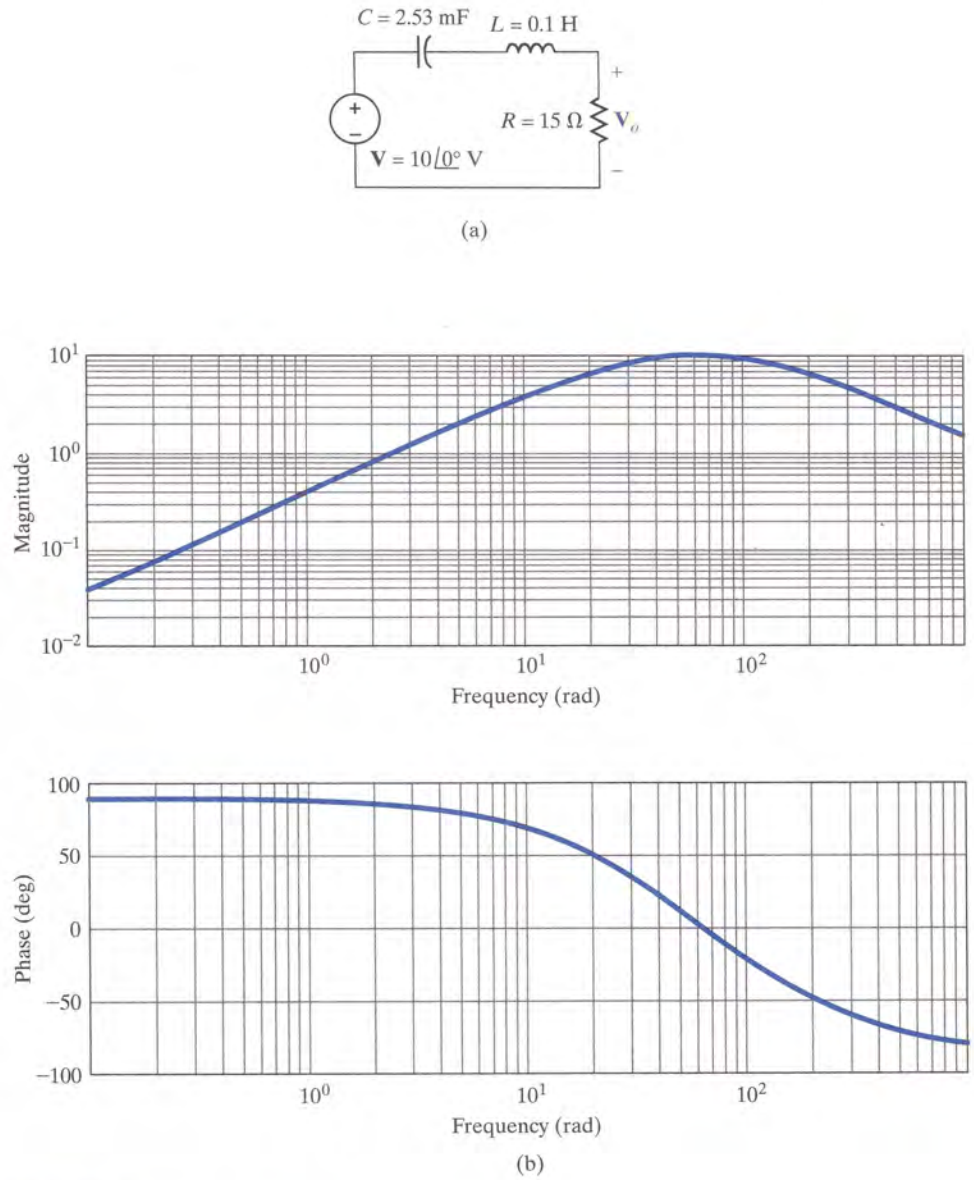


Figure 8.5
(a) Network and (b) its frequency-response simulation.

We will illustrate in subsequent sections that the use of a semilog plot is a very useful tool in deriving frequency-response information.

As an introductory application of variable frequency-response analysis and characterization, let us consider a stereo amplifier. In particular we should consider first the frequency range over which the amplifier must perform and then exactly what kind of performance we desire. The frequency range of the amplifier must exceed that of the human ear, which is roughly 50 Hz to 15,000 Hz. Accordingly, typical stereo amplifiers are designed to operate in the frequency range from 50 Hz to 20,000 Hz. Furthermore, we want to preserve the fidelity of the signal as it passes through the amplifier. Thus, the output signal should be an exact duplicate of the input

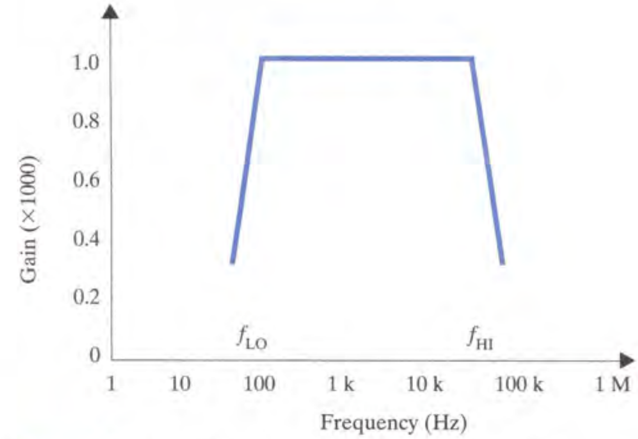


Figure 8.6 *Amplifier frequency-response requirements.*

signal times a gain factor. This requires that the gain be independent of frequency over the specified frequency range of 50 Hz to 20,000 Hz. An ideal sketch of this requirement for a gain of 1000 is shown in Fig. 8.6, where the midband region is defined as that portion of the plot where the gain is constant and is bounded by two points, which we will refer to as f_{LO} and f_{HI}. Notice once again that the frequency axis is a log axis and, thus, the frequency response is displayed on a semilog plot.

A model for the amplifier described graphically in Fig. 8.6 is shown in Fig. 8.7a with the frequency-domain equivalent circuit in Fig. 8.7b.

If the input is a steady-state sinusoid, we can use frequency-domain analysis to find the gain

$$\mathbf{G}_v(j\omega) = \frac{\mathbf{V}_o(j\omega)}{\mathbf{V}_S(j\omega)}$$

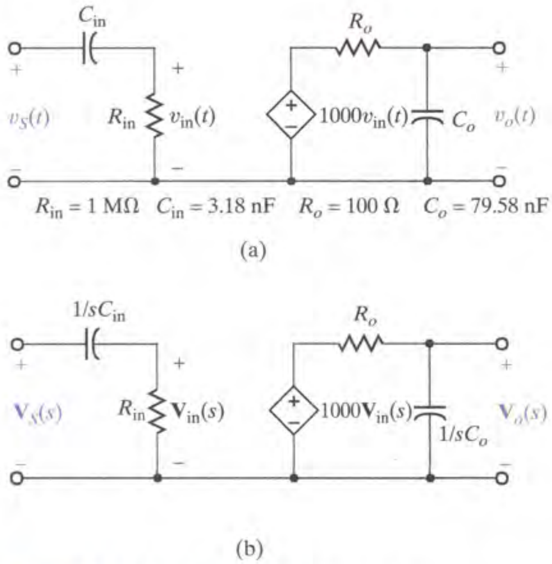


Figure 8.7 *Amplifier equivalent network.*

which with the substitution $s = j\omega$ can be expressed as

$$\mathbf{G}_v(s) = \frac{\mathbf{V}_o(s)}{\mathbf{V}_S(s)}$$

Using voltage division, we find that the gain is

$$\mathbf{G}_v(s) = \frac{\mathbf{V}_o(s)}{\mathbf{V}_S(s)} = \frac{\mathbf{V}_{\text{in}}(s)}{\mathbf{V}_S(s)}\frac{\mathbf{V}_o(s)}{\mathbf{V}_{\text{in}}(s)} = \left[\frac{R_{\text{in}}}{R_{\text{in}} + 1/sC_{\text{in}}}\right](1000)\left[\frac{1/sC_o}{R_o + 1/sC_o}\right]$$

or

$$\mathbf{G}_v(s) = \left[\frac{sC_{\text{in}}R_{\text{in}}}{1 + sC_{\text{in}}R_{\text{in}}}\right](1000)\left[\frac{1}{1 + sC_oR_o}\right]$$

Using the element values in Fig. 8.8a,

$$\mathbf{G}_v(s) = \left[\frac{s}{s + 100\pi}\right](1000)\left[\frac{40{,}000\pi}{s + 40{,}000\pi}\right]$$

where 100π and $40{,}000\pi$ are the radian equivalents of 50 Hz and 20,000 Hz, respectively. Since $s = j\omega$, the network function is indeed complex. An exact plot of $\mathbf{G}_v(s)$ is shown in Fig. 8.8 superimposed over the sketch of Fig. 8.6. The exact plot exhibits smooth transitions at f_{LO} and f_{HI}; otherwise the plots match fairly well.

Let us examine our expression for $\mathbf{G}_v(s)$ more closely with respect to the plot in Fig. 8.8. Assume that f is well within the midband frequency range; that is,

$$f_{\text{LO}} \ll f \ll f_{\text{HI}}$$

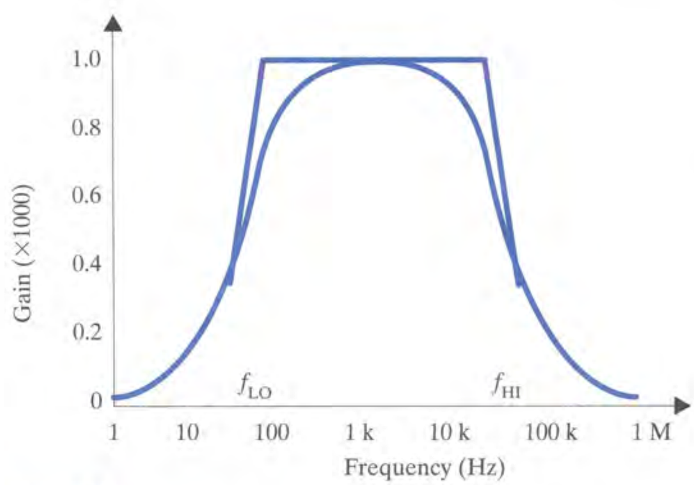


Figure 8.8
Exact and approximate amplifier gain versus frequency plots.

or

$$100\pi \ll |s| \ll 40{,}000\pi$$

Under these conditions, the network function becomes

$$\mathbf{G}_v(s) \approx \left[\frac{s}{s}\right](1000)\left[\frac{1}{1+0}\right]$$

or

$$\mathbf{G}_v(s) = 1000$$

Thus, well within midband, the gain is constant. However, if the frequency of excitation decreases toward f_{LO}, then $|s|$ is comparable to 100π and

$$\mathbf{G}_v(s) \approx \left[\frac{s}{s+100\pi}\right](1000)$$

Since $R_{\text{in}}C_{\text{in}} = 1/100\pi$, we see that C_{in} causes the rolloff in gain at low frequencies. Similarly, when the frequency approaches f_{HI}, the gain rolloff is due to C_o.

Through this amplifier example, we have introduced the concept of frequency-dependent networks and have demonstrated that frequency-dependent network performance is caused by the reactive elements in a network.

NETWORK FUNCTIONS In the previous section, we introduced the term *voltage gain*, $\mathbf{G}_v(s)$. This term is actually only one of several network functions, designated generally as $\mathbf{H}(s)$, which define the ratio of response to input. Since the function describes a reaction due to an excitation at some other point in the circuit, network functions are also called *transfer functions*. Furthermore, transfer functions are not limited to voltage ratios. Since in electrical networks inputs and outputs can be either voltages or currents, there are four possible network functions, as listed in Table 8.1.

There are also *driving point functions*, which are impedances or admittances defined at a single pair of terminals. For example, the input impedance of a network is a driving point function.

Table 8.1 Network transfer functions

Input	Output	Transfer Function	Symbol
Voltage	Voltage	Voltage gain	$\mathbf{G}_v(s)$
Current	Voltage	Transimpedance	$\mathbf{Z}(s)$
Current	Current	Current gain	$\mathbf{G}_i(s)$
Voltage	Current	Transadmittance	$\mathbf{Y}(s)$

LEARNING Example 8.2

We wish to determine the transfer admittance $[\mathbf{I}_2(s)/\mathbf{V}_1(s)]$ and the voltage gain of the network shown in Fig. 8.9.

SOLUTION The mesh equations for the network are

$$(R_1 + sL)\mathbf{I}_1(s) - sL\mathbf{I}_2(s) = \mathbf{V}_1(s)$$

$$-sL\mathbf{I}_1(s) + \left(R_2 + sL + \frac{1}{sC}\right)\mathbf{I}_2(s) = 0$$

$$\mathbf{V}_2(s) = \mathbf{I}_2(s)R_2$$

Solving the equations for $\mathbf{I}_2(s)$ yields

$$\mathbf{I}_2(s) = \frac{sL\mathbf{V}_1(s)}{(R_1 + sL)(R_2 + sL + 1/sC) - s^2L^2}$$

Therefore, the transfer admittance $[\mathbf{I}_2(s)/\mathbf{V}_1(s)]$ is

$$\mathbf{Y}_T(s) = \frac{\mathbf{I}_2(s)}{\mathbf{V}_1(s)} = \frac{LCs^2}{(R_1 + R_2)LCs^2 + (L + R_1R_2C)s + R_1}$$

and the voltage gain is

$$\mathbf{G}_v(s) = \frac{\mathbf{V}_2(s)}{\mathbf{V}_1(s)} = \frac{LCR_2s^2}{(R_1 + R_2)LCs^2 + (L + R_1R_2C)s + R_1}$$

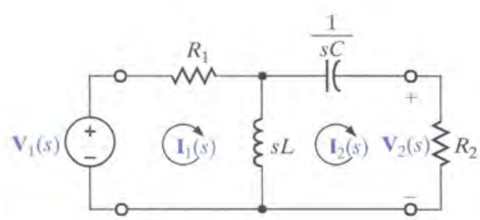


Figure 8.9
Circuit employed in Example 8.2.

POLES AND ZEROS As we have indicated, the network function can be expressed as the ratio of the two polynomials in s. In addition, we note that since the values of our circuit elements, or controlled sources, are real numbers, the coefficients of the two polynomials will be real. Therefore, we will express a network function in the form

$$\mathbf{H}(s) = \frac{N(s)}{D(s)} = \frac{a_m s^m + a_{m-1}s^{m-1} + \cdots + a_1 s + a_0}{b_n s^n + b_{n-1}s^{n-1} + \cdots + b_1 s + b_0} \qquad 8.2$$

where $N(s)$ is the numerator polynomial of degree m and $D(s)$ is the denominator polynomial of degree n. Equation (8.2) can also be written in the form

$$\mathbf{H}(s) = \frac{K_0(s - z_1)(s - z_2)\cdots(s - z_m)}{(s - p_1)(s - p_2)\cdots(s - p_n)} \qquad 8.3$$

where K_0 is a constant, $z_1, \ldots, z_m$ are the roots of $N(s)$, and $p_1, \ldots, p_n$ are the roots of $D(s)$. Note that if $s = z_1$, or $z_2, \ldots, z_m$, then $\mathbf{H}(s)$ becomes zero and hence $z_1, \ldots, z_m$ are called zeros of the transfer function. Similarly, if $s = p_1$, or $p_2, \ldots, p_n$, then $\mathbf{H}(s)$ becomes infinite and, therefore, $p_1, \ldots, p_n$ are called poles of the function. The zeros or poles may actually be complex. However, if they are complex, they must occur in conjugate pairs since the coefficients of the polynomial are real. The representation of the network function specified in Eq. (8.3) is extremely important and is generally employed to represent any linear time-invariant system.

The importance of this form stems from the fact that the dynamic properties of a system can be gleaned from an examination of the system poles.

LEARNING EXTENSIONS

E8.1 Find the driving point impedance at $\mathbf{V}_S(s)$ in the amplifier shown in Fig. 8.7b.

ANSWER

$$\mathbf{Z}(s) = R_{\text{in}} + \frac{1}{sC_{\text{in}}}$$
$$= \left[1 + \left(\frac{100\pi}{s}\right)\right] \text{M}\Omega$$

E8.2 Find the pole and zero locations in hertz and the value of K_0 for the amplifier network in Fig. 8.7.

ANSWER $z_1 = 0$ Hz (dc), $p_1 = -50$ Hz, $p_2 = -20{,}000$ Hz, $K_0 = (4 \times 10^7)\pi$.

8.2 Sinusoidal Frequency Analysis

Although there are specific cases in which a network operates at only one frequency (e.g., a power system network), in general we are interested in the behavior of a network as a function of frequency. In a sinusoidal steady-state analysis, the network function can be expressed as

$$\mathbf{H}(j\omega) = M(\omega)e^{j\phi(\omega)} \tag{8.4}$$

where $M(\omega) = |\mathbf{H}(j\omega)|$ and $\phi(\omega)$ is the phase. A plot of these two functions, which are commonly called the *magnitude* and *phase characteristics,* displays the manner in which the response varies with the input frequency ω. We will now illustrate the manner in which to perform a frequency-domain analysis by simply evaluating the function at various frequencies within the range of interest.

FREQUENCY RESPONSE USING A BODE PLOT If the network characteristics are plotted on a semilog scale (that is, a linear scale for the ordinate and a logarithmic scale for the abscissa), they are known as *Bode plots* (named after Hendrik W. Bode). This graph is a powerful tool in both the analysis and design of frequency-dependent systems and networks such as filters, tuners, and amplifiers. In using the graph, we plot $20 \log_{10} M(\omega)$ versus $\log_{10}(\omega)$ instead of $M(\omega)$ versus ω. The advantage of this technique is that rather than plotting the characteristic point by point, we can employ straight-line approximations to obtain the characteristic very efficiently. The ordinate for the magnitude plot is the decibel (dB). This unit was originally employed to measure the ratio of powers; that is,

$$\text{number of dB} = 10 \log_{10} \frac{P_2}{P_1} \tag{8.5}$$

If the powers are absorbed by two equal resistors, then

$$\text{number of dB} = 10 \log_{10} \frac{|\mathbf{V}_2|^2/R}{|\mathbf{V}_1|^2/R} = 10 \log_{10} \frac{|\mathbf{I}_2|^2 R}{|\mathbf{I}_1|^2 R}$$
$$= 20 \log_{10} \frac{|\mathbf{V}_2|}{|\mathbf{V}_1|} = 20 \log_{10} \frac{|\mathbf{I}_2|}{|\mathbf{I}_1|} \quad \textbf{8.6}$$

The term "dB" has become so popular that it now is used for voltage and current ratios, as illustrated in Eq. (8.6), without regard to the impedance employed in each case.

In the sinusoidal steady-state case, $\mathbf{H}(j\omega)$ in Eq. (8.3) can be expressed in general as

$$\mathbf{H}(j\omega) = \frac{K_0(j\omega)^{\pm N}(1 + j\omega\tau_1)[1 + 2\zeta_3(j\omega\tau_3) + (j\omega\tau_3)^2]\cdots}{(1 + j\omega\tau_a)[1 + 2\zeta_b(j\omega\tau_b) + (j\omega\tau_b)^2]\cdots} \quad \textbf{8.7}$$

Note that this equation contains the following typical factors:

1. A frequency-independent factor $K_0 > 0$
2. Poles or zeros at the origin of the form $j\omega$; that is, $(j\omega)^{+N}$ for zeros and $(j\omega)^{-N}$ for poles
3. Poles or zeros of the form $(1 + j\omega\tau)$
4. Quadratic poles or zeros of the form $1 + 2\zeta(j\omega\tau) + (j\omega\tau)^2$

Taking the logarithm of the magnitude of the function $\mathbf{H}(j\omega)$ in Eq. (8.7) yields

$$\begin{aligned}20 \log_{10}|\mathbf{H}(j\omega)| &= 20 \log_{10} K_0 \pm 20N \log_{10}|j\omega| \\ &+ 20 \log_{10}|1 + j\omega\tau_1| \\ &+ 20 \log_{10}|1 + 2\zeta_3(j\omega\tau_3) + (j\omega\tau_3)^2| \\ &+ \cdots - 20 \log_{10}|1 + j\omega\tau_a| \\ &- 20 \log_{10}|1 + 2\zeta_b(j\omega\tau_b) + (j\omega\tau_b)^2| \cdots\end{aligned} \quad \textbf{8.8}$$

Note that we have used the fact that the log of the product of two or more terms is equal to the sum of the logs of the individual terms, the log of the quotient of two terms is equal to the difference of the logs of the individual terms, and $\log_{10} A^n = n \log_{10} A$.

The phase angle for $\mathbf{H}(j\omega)$ is

$$\begin{aligned}\underline{/\mathbf{H}(j\omega)} &= 0 \pm N(90°) + \tan^{-1}\omega\tau_1 + \tan^{-1}\left(\frac{2\zeta_3\omega\tau_3}{1 - \omega^2\tau_3^2}\right) \\ &+ \cdots - \tan^{-1}\omega\tau_a - \tan^{-1}\left(\frac{2\zeta_b\omega\tau_b}{1 - \omega^2\tau_b^2}\right)\cdots\end{aligned} \quad \textbf{8.9}$$

As Eqs. (8.8) and (8.9) indicate, we will simply plot each factor individually on a common graph and then sum them algebraically to obtain the total characteristic. Let us examine some of the individual terms and illustrate an efficient manner in which to plot them on the Bode diagram.

Constant Term The term $20 \log_{10} K_0$ represents a constant magnitude with zero phase shift, as shown in Fig. 8.10a.

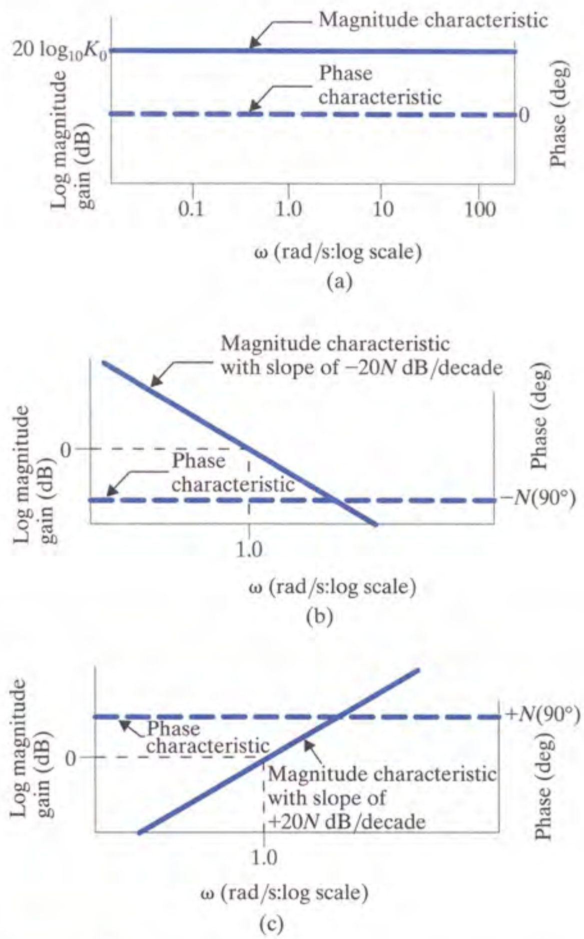


Figure 8.10
Magnitude and phase characteristics for a constant term and poles and zeros at the origin.

Poles or Zeros at the Origin Poles or zeros at the origin are of the form $(j\omega)^{\pm N}$, where + is used for a zero and − is used for a pole. The magnitude of this function is $\pm 20N \log_{10} \omega$, which is a straight line on semilog paper with a slope of $\pm 20N$ dB/decade; that is, the value will change by $20N$ each time the frequency is multiplied by 10, and the phase of this function is a constant $\pm N(90°)$. The magnitude and phase characteristics for poles and zeros at the origin are shown in Figs. 8.10b and c, respectively.

Simple Pole or Zero Linear approximations can be employed when a simple pole or zero of the form $(1 + j\omega\tau)$ is present in the network function. For $\omega\tau \ll 1$, $(1 + j\omega\tau) \approx 1$, and therefore, $20 \log_{10}|(1 + j\omega\tau)| = 20 \log_{10} 1 = 0$ dB. Similarly, if $\omega\tau \gg 1$, then $(1 + j\omega\tau) \approx j\omega\tau$, and hence $20 \log_{10}|(1 + j\omega\tau)| \approx 20 \log_{10} \omega\tau$. Therefore, for $\omega\tau \ll 1$ the response is 0 dB and for $\omega\tau \gg 1$ the response has a slope that is the same as that of a simple pole or zero at the origin. The intersection of these two asymptotes, one for $\omega\tau \ll 1$ and one for $\omega\tau \gg 1$, is the point where $\omega\tau = 1$ or $\omega = 1/\tau$, which is called the *break frequency*. At this break frequency, where $\omega = 1/\tau$, $20 \log_{10}|(1 + j1)| = 20 \log_{10}(2)^{1/2} = 3$ dB. Therefore, the actual curve deviates from the asymptotes by 3 dB at the break frequency. It can be shown that at one-half and twice the break frequency, the deviations are 1 dB. The phase angle associated with a simple pole or zero is $\phi = \tan^{-1} \omega\tau$, which is a simple arctangent curve. Therefore, the phase shift is 45° at the break frequency and 26.6° and 63.4° at one-half and twice the break frequency, respectively. The actual magnitude curve for a pole of this form is shown in Fig. 8.11a. For a zero the magnitude curve and the asymptote for $\omega\tau \gg 1$ have a positive slope, and the phase curve extends from 0° to +90°, as shown in Fig. 8.11b. If multiple poles or zeros of the form $(1 + j\omega\tau)^N$ are present, then the slope of the high-frequency asymptote is multiplied by

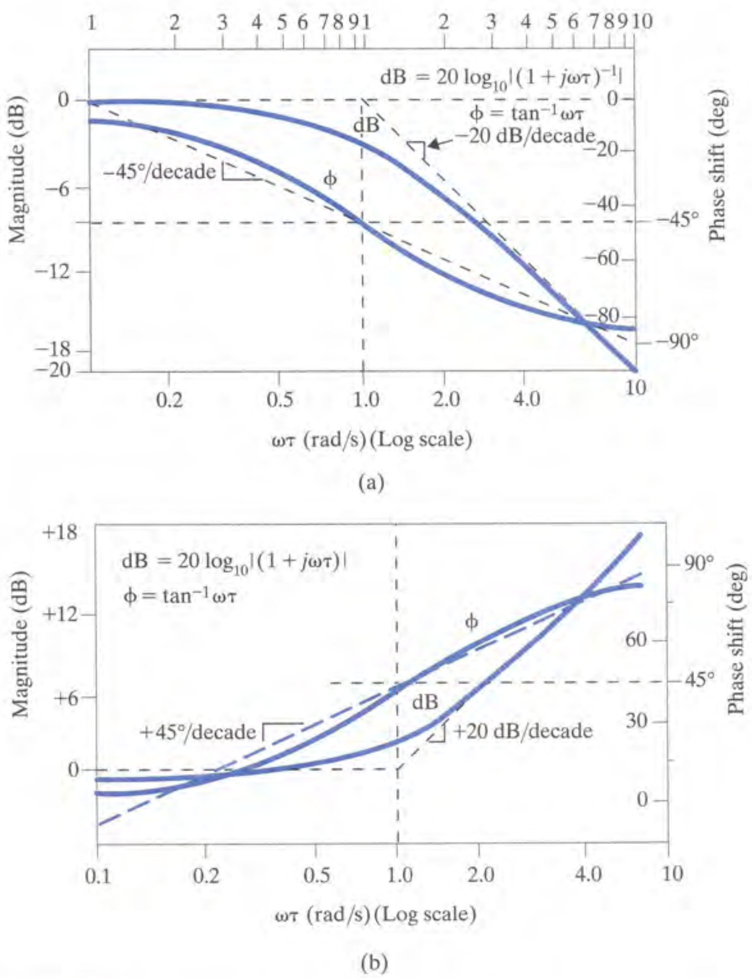


Figure 8.11
Magnitude and phase plot (a) for a simple pole, and (b) for a simple zero.

N, the deviation between the actual curve and the asymptote at the break frequency is $3N$ dB, and the phase curve extends from 0 to $N(90°)$ and is $N(45°)$ at the break frequency.

Quadratic Poles or Zeros Quadratic poles or zeros are of the form $1 + 2\zeta(j\omega\tau) + (j\omega\tau)^2$. This term is a function not only of ω, but also of the dimensionless term ζ, which is called the *damping ratio*. If $\zeta > 1$ or $\zeta = 1$, the roots are real and unequal or real and equal, respectively, and these two cases have already been addressed. If $\zeta < 1$, the roots are complex conjugates, and it is this case that we will examine now. Following the preceding argument for a simple pole or zero, the log magnitude of the quadratic factor is 0 dB for $\omega\tau \ll 1$. For $\omega\tau \gg 1$.

$$20 \log_{10}\left|1 - (\omega\tau)^2 + 2j\zeta(\omega\tau)\right| \approx 20 \log_{10}\left|(\omega\tau)^2\right| = 40 \log_{10}|\omega\tau|$$

and therefore, for $\omega\tau \gg 1$, the slope of the log magnitude curve is +40 dB/decade for a quadratic zero and −40 dB/decade for a quadratic pole. Between the two extremes, $\omega\tau \ll 1$ and $\omega\tau \gg 1$, the behavior of the function is dependent on the damping ratio ζ. Figure 8.12a

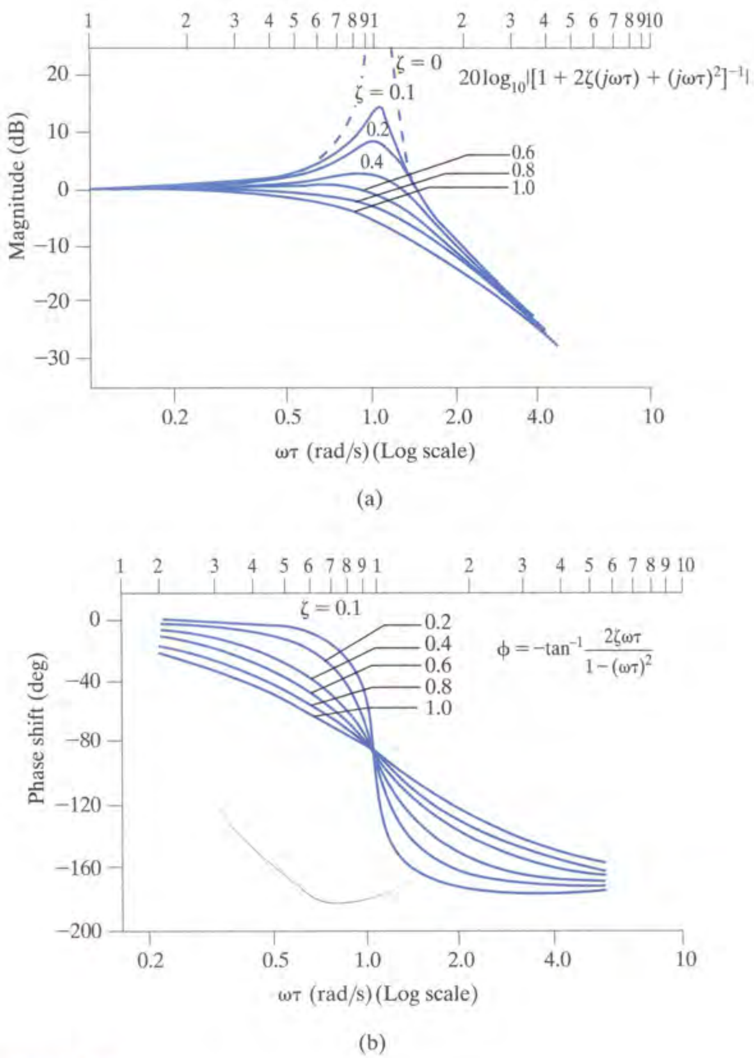


Figure 8.12
Magnitude and phase characteristics for quadratic poles.

illustrates the manner in which the log magnitude curve for a quadratic *pole* changes as a function of the damping ratio. The phase shift for the quadratic factor is $\tan^{-1} 2\zeta\omega\tau/[1 - (\omega\tau)^2]$. The phase plot for quadratic *poles* is shown in Fig. 8.12b. Note that in this case the phase changes from 0° at frequencies for which $\omega\tau \ll 1$ to −180° at frequencies for which $\omega\tau \gg 1$. For quadratic zeros the magnitude and phase curves are inverted; that is, the log magnitude curve has a slope of +40 dB/decade for $\omega\tau \gg 1$, and the phase curve is 0° for $\omega\tau \ll 1$ and +180° for $\omega\tau \gg 1$.

LEARNING Example 8.3

We want to generate the magnitude and phase plots for the transfer function

$$\mathbf{G}_v(j\omega) = \frac{10(0.1j\omega + 1)}{(j\omega + 1)(0.02j\omega + 1)}$$

SOLUTION Note that this function is in standard form, since every term is of the form $(j\omega\tau + 1)$. To determine the composite magnitude and phase characteristics, we will plot the individual asymptotic terms and then add them as specified in Eqs. (8.8) and (8.9). Let us consider the magnitude plot first. Since $K_0 = 10$, $20 \log_{10} 10 = 20$ dB, which is a constant independent of frequency, as shown in Fig. 8.13a. The zero of the transfer function contributes a term of the form $+20 \log_{10}|1 + 0.1j\omega|$, which is 0 dB for $0.1\omega \ll 1$, has a slope of +20 dB/decade for $0.1\omega \gg 1$, and has a break frequency at $\omega = 10$ rad/s. The poles have break frequencies at $\omega = 1$ and $\omega = 50$ rad/s. The pole with break frequency at $\omega = 1$ rad/s contributes a term of the form $-20 \log_{10}|1 + j\omega|$, which is 0 dB for $\omega \ll 1$ and has a slope of −20 dB/decade for $\omega \gg 1$. A similar argument can be made for the pole that has a break frequency at $\omega = 50$ rad/s. These factors are all plotted individually in Fig. 8.13a.

Consider now the individual phase curves. The term K_0 is not a function of ω and does not contribute to the phase of the transfer function. The phase curve for the zero is $+\tan^{-1} 0.1\omega$,

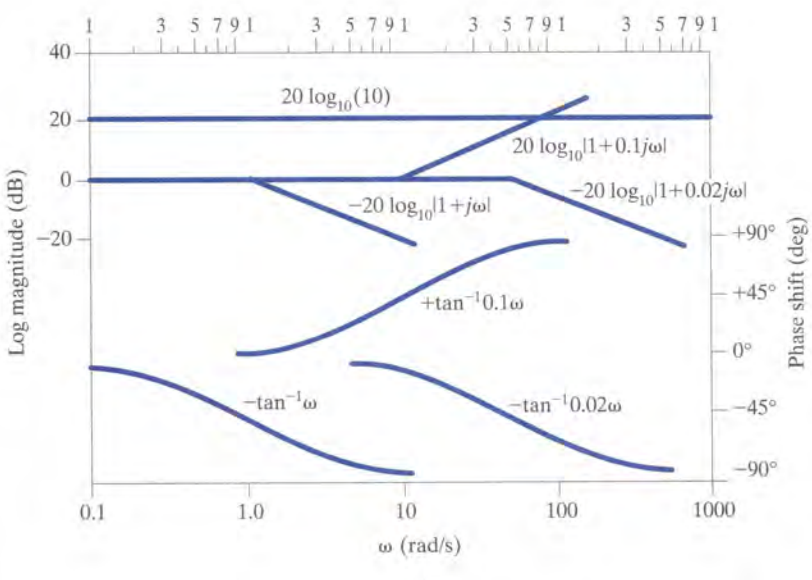


Figure 8.13

(a) Magnitude and phase components for the poles and zeros of the transfer function in Example 8.3; (b) Bode plot for the transfer function in Example 8.3.

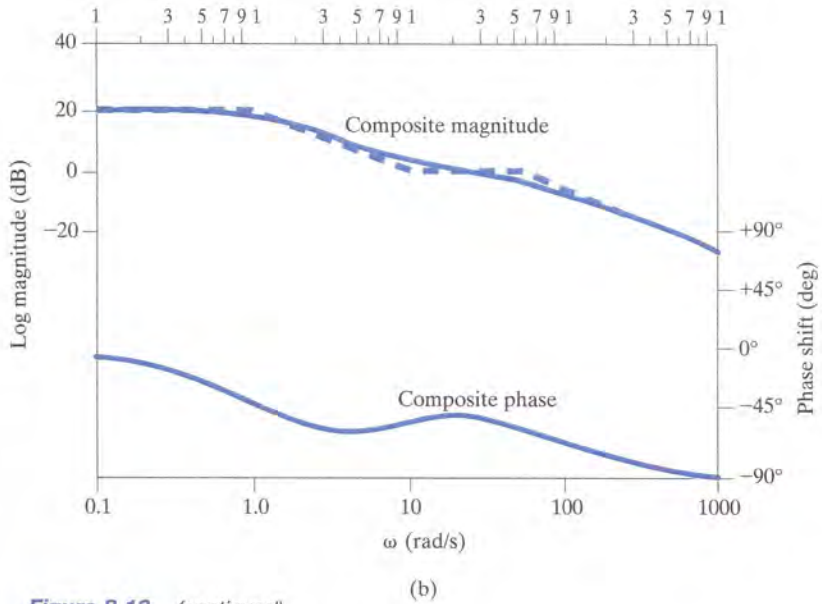


Figure 8.13 (continued)

which is an arctangent curve that extends from 0° for $0.1\omega \ll 1$ to +90° for $0.1\omega \gg 1$ and has a phase of +45° at the break frequency. The phase curves for the two poles are $-\tan^{-1}\omega$ and $-\tan^{-1}0.02\omega$. The term $-\tan^{-1}\omega$ is 0° for $\omega \ll 1$, −90° for $\omega \gg 1$, and −45° at the break frequency $\omega = 1$. The phase curve for the remaining pole is plotted in a similar fashion. All the individual phase curves are shown in Fig. 8.13a.

As specified in Eqs. (8.8) and (8.9), the composite magnitude and phase of the transfer function are obtained simply by adding the individual terms. The composite curves are plotted in Fig. 8.13b. Note that the actual magnitude curve (solid line) differs from the straight-line approximation (dashed line) by 3 dB at the break frequencies and 1 dB at one-half and twice the break frequencies.

LEARNING Example 8.4

Let us draw the Bode plot for the following transfer function:

$$\mathbf{G}_v(j\omega) = \frac{25(j\omega + 1)}{(j\omega)^2(0.1j\omega + 1)}$$

SOLUTION Once again all the individual terms for both magnitude and phase are plotted in Fig. 8.14a. The straight line with a slope of −40 dB/decade is generated by the double pole at the origin. This line is a plot of $-40\log_{10}\omega$ versus ω and therefore passes through 0 dB at $\omega = 1$ rad/s. The phase for the double pole is a constant −180° for all frequencies. The remainder of the terms are plotted as illustrated in Example 8.3.

The composite plots are shown in Fig. 8.14b. Once again they are obtained simply by adding the individual terms in Fig. 8.14a. Note that for frequencies for which $\omega \ll 1$, the slope of the magnitude curve is −40 dB/decade. At $\omega = 1$ rad/s, which is the break frequency of the zero, the magnitude curve changes slope to −20 dB/decade. At $\omega = 10$ rad/s, which is the break frequency of the pole, the slope of the magnitude curve changes back to −40 dB/decade.

The composite phase curve starts at −180° due to the double pole at the origin. Since the first break frequency encountered is a zero, the phase curve shifts toward −90°. However, before the composite phase reaches −90°, the pole with break frequency $\omega = 10$ rad/s begins to shift the composite curve back toward −180°.

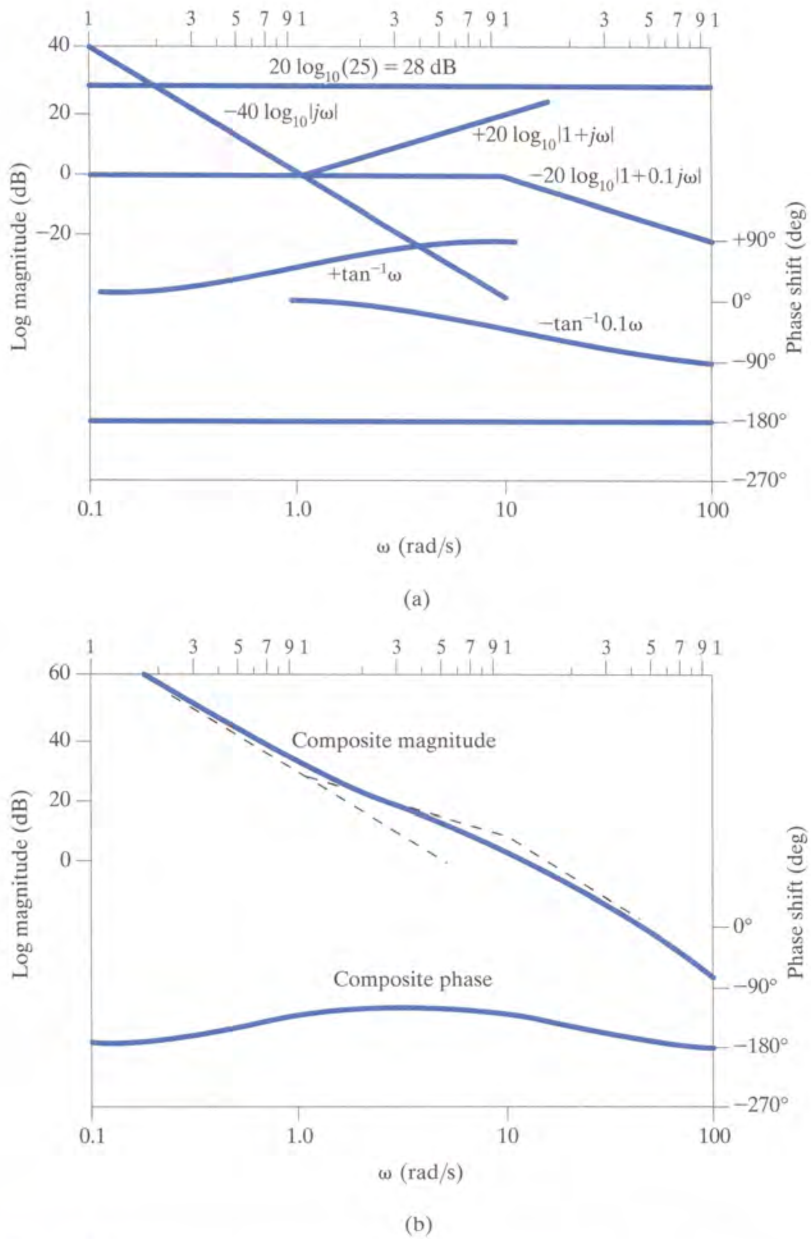


Figure 8.14

(a) Magnitude and phase components for the poles and zeros of the transfer function in Example 8.4; (b) Bode plot for the transfer function in Example 8.4.

Example 8.4 illustrates the manner in which to plot directly terms of the form $K_0/(j\omega)^N$. For terms of this form, the initial slope of $-20N$ dB/decade will intersect the 0-dB axis at a frequency of $(K_0)^{1/N}$ rad/s; that is, $-20\log_{10}|K_0/(j\omega)^N| = 0$ dB implies that $K_0/(j\omega)^N = 1$, and therefore, $\omega = (K_0)^{1/N}$ rad/s. Note that the projected slope of the magnitude curve in Example 8.4 intersects the 0-dB axis at $\omega = (25)^{1/2} = 5$ rad/s.

Similarly, it can be shown that for terms of the form $K_0(j\omega)^N$, the initial slope of $+20N$ dB/decade will intersect the 0-dB axis at a frequency of $\omega = (1/K_0)^{1/N}$ rad/s; that is, $+20\log_{10}|K_0/(j\omega)^N| = 0$ dB implies that $K_0/(j\omega)^N = 1$, and therefore $\omega = (1/K_0)^{1/N}$ rad/s.

By applying the concepts we have just demonstrated, we can normally plot the log magnitude characteristic of a transfer function directly in one step.

LEARNING EXTENSIONS

E8.3 Sketch the magnitude characteristic of the Bode plot, labeling all critical slopes and points for the function

$$\mathbf{G}(j\omega) = \frac{10^4(j\omega + 2)}{(j\omega + 10)(j\omega + 100)}$$

ANSWER

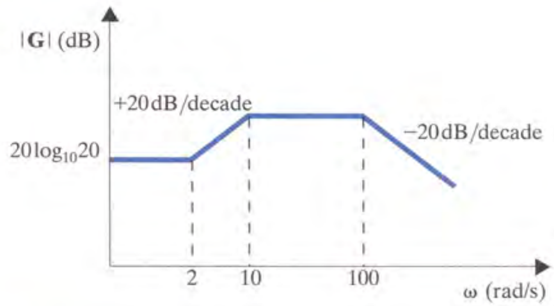


Figure E8.3

E8.4 Sketch the magnitude characteristic of the Bode plot, labeling all critical slopes and points for the function

$$\mathbf{G}(j\omega) = \frac{100(0.02j\omega + 1)}{(j\omega)^2}$$

ANSWER

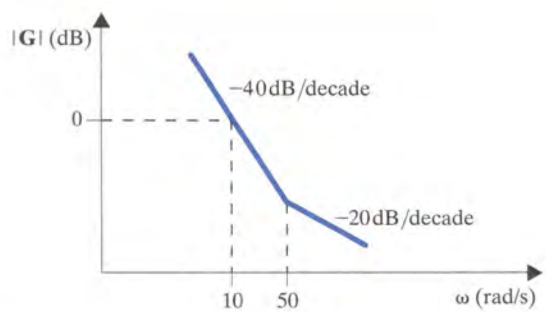


Figure E8.4

E8.5 Sketch the magnitude characteristic of the Bode plot, labeling all critical slopes and points for the function

$$\mathbf{G}(j\omega) = \frac{10j\omega}{(j\omega + 1)(j\omega + 10)}$$

ANSWER

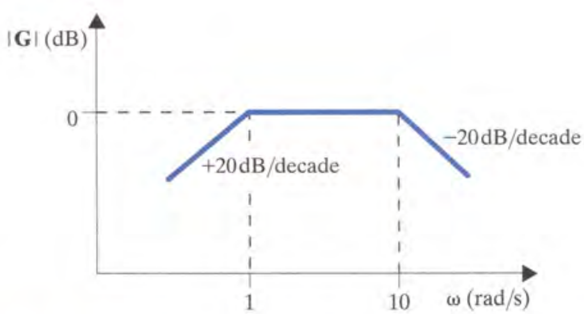


Figure E8.5

LEARNING Example 8.5

We wish to generate the Bode plot for the following transfer function:

$$\mathbf{G}_v(j\omega) = \frac{25j\omega}{(j\omega + 0.5)[(j\omega)^2 + 4j\omega + 100]}$$

SOLUTION Expressing this function in standard form, we obtain

$$\mathbf{G}_v(j\omega) = \frac{0.5j\omega}{(2j\omega + 1)[(j\omega/10)^2 + j\omega/25 + 1]}$$

The Bode plot is shown in Fig. 8.15. The initial low-frequency slope due to the zero at the origin is +20 dB/decade, and this slope intersects the 0-dB line at $\omega = 1/K_0 = 2$ rad/s. At $\omega = 0.5$ rad/s the slope changes from +20 dB/decade to 0 dB/decade due to the presence of the pole with a break frequency at $\omega = 0.5$ rad/s. The quadratic term has a center frequency of $\omega = 10$ rad/s (i.e., $\tau = 1/10$). Since

$$2\zeta\tau = \frac{1}{25}$$

and

$$\tau = 0.1$$

then

$$\zeta = 0.2$$

Plotting the curve in Fig. 8.12a with a damping ratio of $\zeta = 0.2$ at the center frequency $\omega = 10$ rad/s completes the composite magnitude curve for the transfer function.

The initial low-frequency phase curve is +90°, due to the zero at the origin. This curve and the phase curve for the simple pole and the phase curve for the quadratic term, as defined in Fig. 8.12b, are combined to yield the composite phase curve.

MATLAB could also be used to generate the magnitude and phase plots. The transfer function can be expressed in the form

$$\mathbf{G} = \frac{25j\omega}{(j\omega)^3 + 4.5(j\omega)^2 + 102j\omega + 50}$$

The MATLAB program for plotting the function is

```
>>B = [25 0];
>>A = [ 1 4.5 102 50];
>>freqs(B,A)
```

and the results are shown in Fig. 8.16.

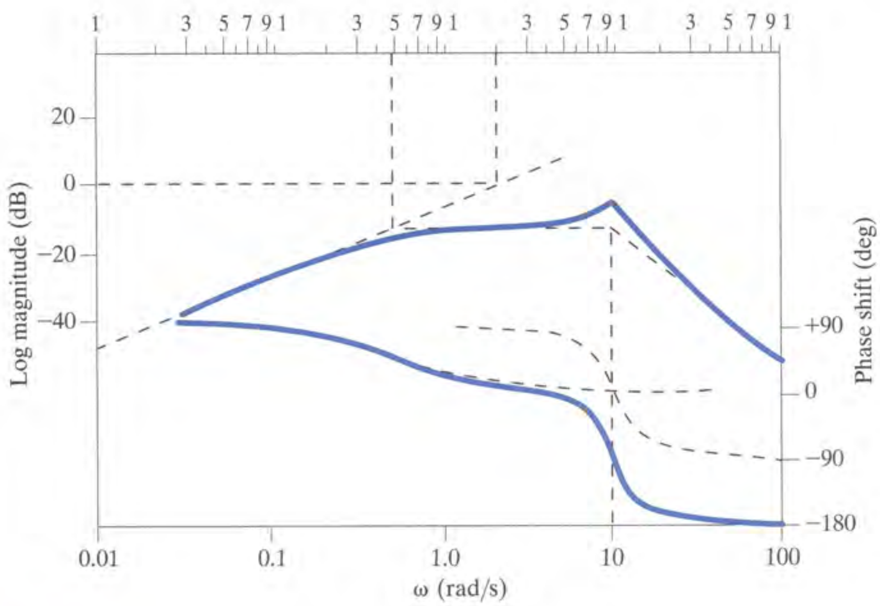


Figure 8.15
Bode plot for the transfer function in Example 8.5.

(continued)

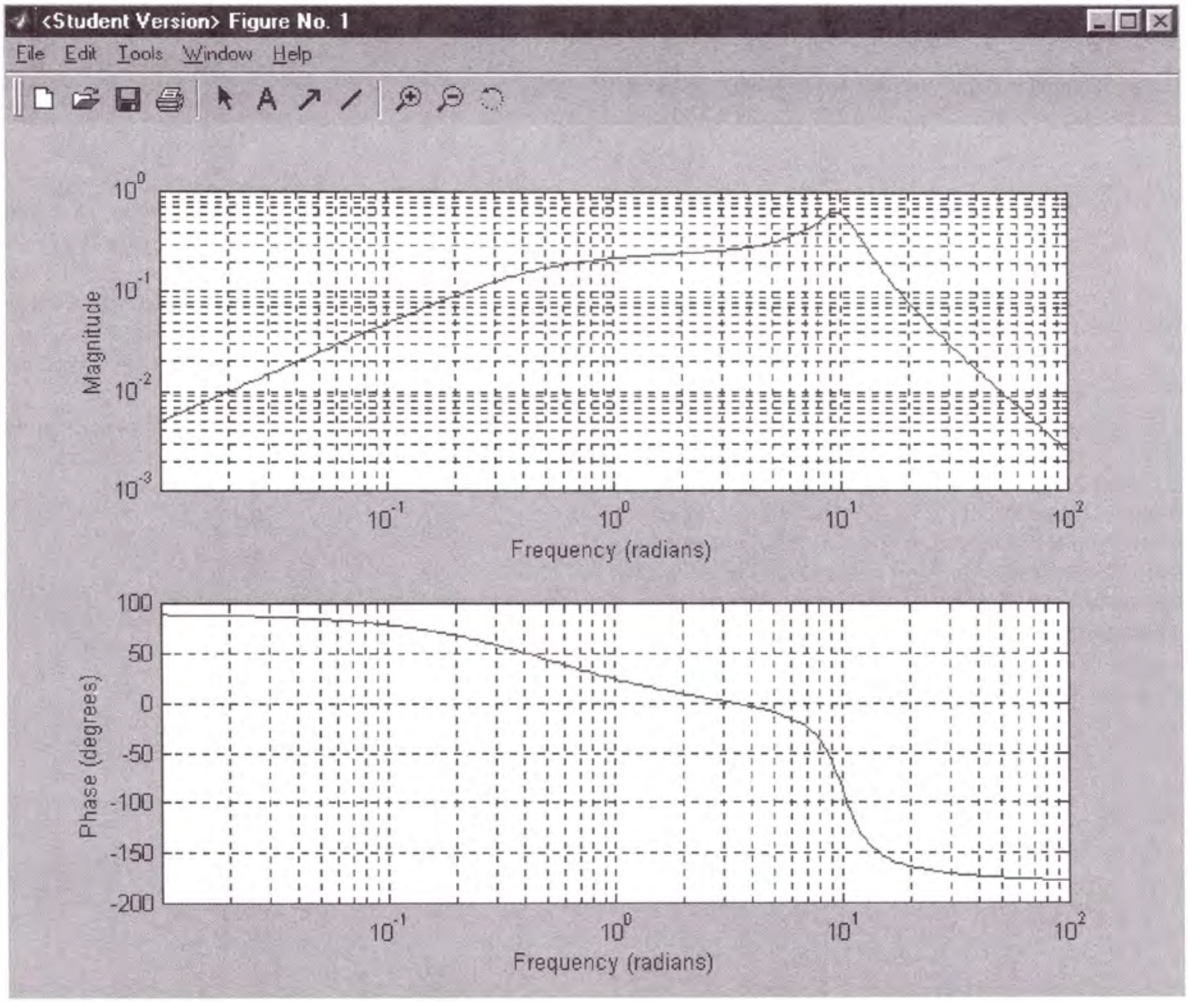


Figure 8.16 MATLAB-generated Bode plot.

LEARNING EXTENSION

E8.6 Given the following function $\mathbf{G}(j\omega)$, sketch the magnitude characteristic of the Bode plot, labeling all critical slopes and points.

$$\mathbf{G}(j\omega) = \frac{0.2(j\omega + 1)}{j\omega\left[(j\omega/12)^2 + j\omega/36 + 1\right]}$$

ANSWER

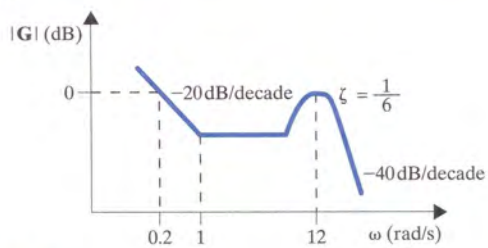


Figure E8.6

DERIVING THE TRANSFER FUNCTION FROM THE BODE PLOT

LEARNING Example 8.6

Given the asymptotic magnitude characteristic shown in Fig. 8.17, we wish to determine the transfer function $\mathbf{G}_v(j\omega)$.

SOLUTION Since the initial slope is 0 dB/decade, and the level of the characteristic is 20 dB, the factor K_0 can be obtained from the expression

$$20 \text{ dB} = 20 \log_{10} K_0$$

and hence

$$K_0 = 10$$

The −20-dB/decade slope starting at $\omega = 0.1$ rad/s indicates that the first pole has a break frequency at $\omega = 0.1$ rad/s, and therefore one of the factors in the denominator is $(10j\omega + 1)$. The slope changes by +20 dB/decade at $\omega = 0.5$ rad/s, indicating that there is a zero present with a break frequency at $\omega = 0.5$ rad/s, and therefore the numerator has a factor of $(2j\omega + 1)$. Two additional poles are present with break frequencies at $\omega = 2$ rad/s and $\omega = 20$ rad/s. Therefore, the composite transfer function is

$$\mathbf{G}_v(j\omega) = \frac{10(2j\omega + 1)}{(10j\omega + 1)(0.5j\omega + 1)(0.05j\omega + 1)}$$

Note carefully the ramifications of this example with regard to network design.

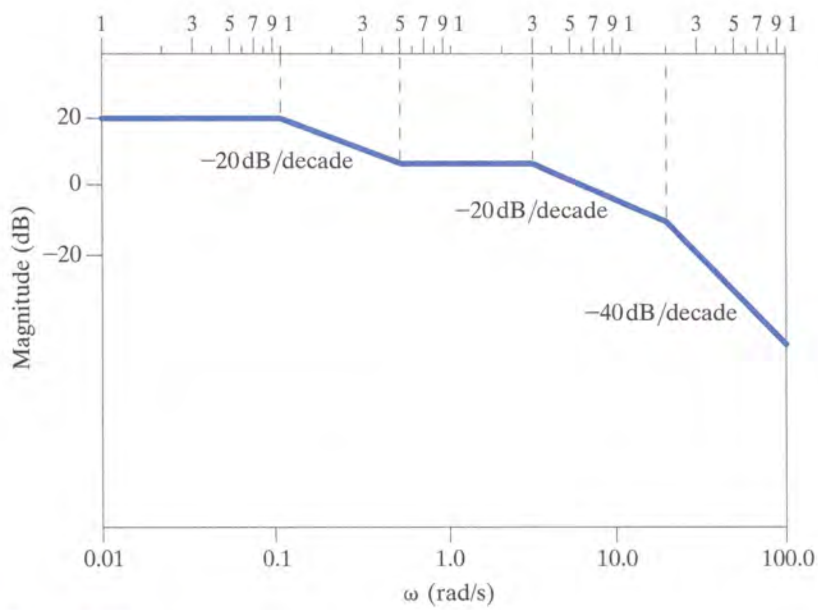


Figure 8.17
Straight-line magnitude plot employed in Example 8.6.

LEARNING EXTENSION

E8.7 Determine the transfer function $\mathbf{G}(j\omega)$ if the straight-line magnitude characteristic approximation for this function is as shown in Fig. E8.7.

ANSWER

$$\mathbf{G}(j\omega) = \frac{5\left(\frac{j\omega}{5}+1\right)\left(\frac{j\omega}{50}+1\right)}{j\omega\left(\frac{j\omega}{20}+1\right)\left(\frac{j\omega}{100}+1\right)}$$

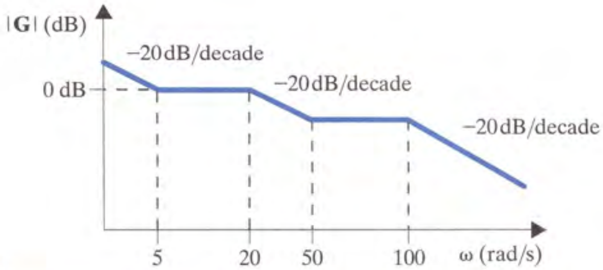


Figure E8.7

8.3 Resonant Circuits

Two circuits with extremely important frequency characteristics are shown in Fig. 8.18. The input impedance for the series *RLC* circuit is

$$\mathbf{Z}(j\omega) = R + j\omega L + \frac{1}{j\omega C} \tag{8.10}$$

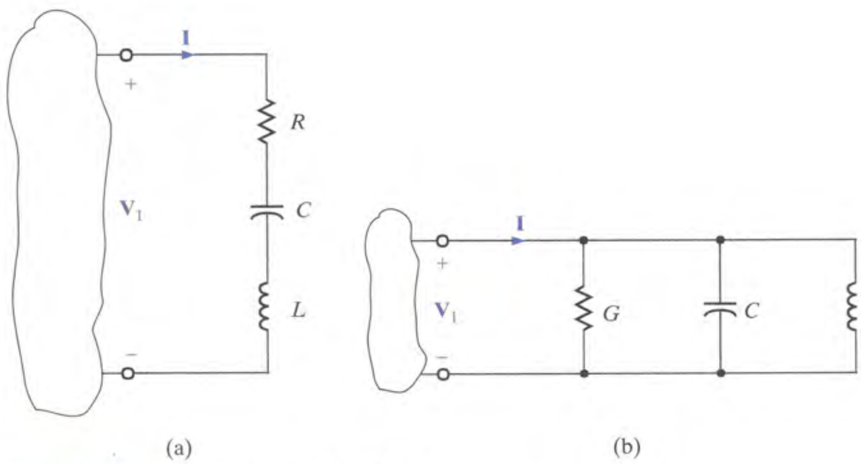


Figure 8.18 Series and parallel RLC circuits.

and the input admittance for the parallel *RLC* circuit is

$$\mathbf{Y}(j\omega) = G + j\omega C + \frac{1}{j\omega L} \tag{8.11}$$

Note that these two equations have the same general form. The imaginary terms in both of the preceding equations will be zero if

$$\omega L = \frac{1}{\omega C}$$

The value of ω that satisfied this equation is

$$\omega_0 = \frac{1}{\sqrt{LC}} \tag{8.12}$$

and at this value of ω the impedance of the series circuit becomes

$$\mathbf{Z}(j\omega_0) = R \tag{8.13}$$

and the admittance of the parallel circuit is

$$\mathbf{Y}(j\omega_0) = G \tag{8.14}$$

LEARNING Hint

Recall that ω_0 is the undamped natural frequency.

This frequency ω_0, at which the impedance of the series circuit or the admittance of the parallel circuit is purely real, is also called the *resonant frequency,* and the circuits themselves, at this frequency, are said to be *in resonance.* Resonance is a very important consideration in engineering design. For example, engineers designing the attitude control system for the Saturn vehicles had to ensure that the control system frequency did not excite the body bending (resonant) frequencies of the vehicle. Excitation of the bending frequencies would cause oscillations that, if continued unchecked, would result in a buildup of stress until the vehicle would finally break apart.

At resonance the voltage and current are in phase and, therefore, the phase angle is zero and the power factor is unity. In the series case, at resonance the impedance is a minimum and, therefore, the current is maximum for a given voltage. Figure 8.19 illustrates the frequency response of both the series and parallel *RLC* circuits. Note that at low frequencies the impedance of the series circuit is dominated by the capacitive term and the admittance of the parallel circuit is dominated by the inductive term. At high frequencies the impedance of the series circuit is dominated by the inductive term, and the admittance of the parallel circuit is dominated by the capacitive term.

Resonance can be viewed from another perspective—that of the phasor diagram. Once again we will consider the series and parallel cases together to illustrate the similarities between them. In the series case the current is common to every element, and in the parallel case the voltage is a common variable. Therefore, the current in the series circuit and the voltage in the parallel circuit are employed as references. Phasor diagrams for both circuits are shown in Fig. 8.20 for the three frequency values $\omega < \omega_0$, $\omega = \omega_0$, $\omega > \omega_0$.

In the series case when $\omega < \omega_0$, $\mathbf{V}_C > \mathbf{V}_L$, θ_Z is negative and the voltage $\mathbf{V}_1$ lags the current. If $\omega = \omega_0$, $\mathbf{V}_L = \mathbf{V}_C$, θ_Z is zero, and the voltage $\mathbf{V}_1$ is in phase with the current. If $\omega > \omega_0$, $\mathbf{V}_L > \mathbf{V}_C$, θ_Z is positive, and the voltage $\mathbf{V}_1$ leads the current. Similar statements can be made for the parallel case in Fig. 8.20b. Because of the close relationship between series and parallel resonance, as illustrated by the preceding material, we will concentrate most of our discussion on the series case in the following developments.

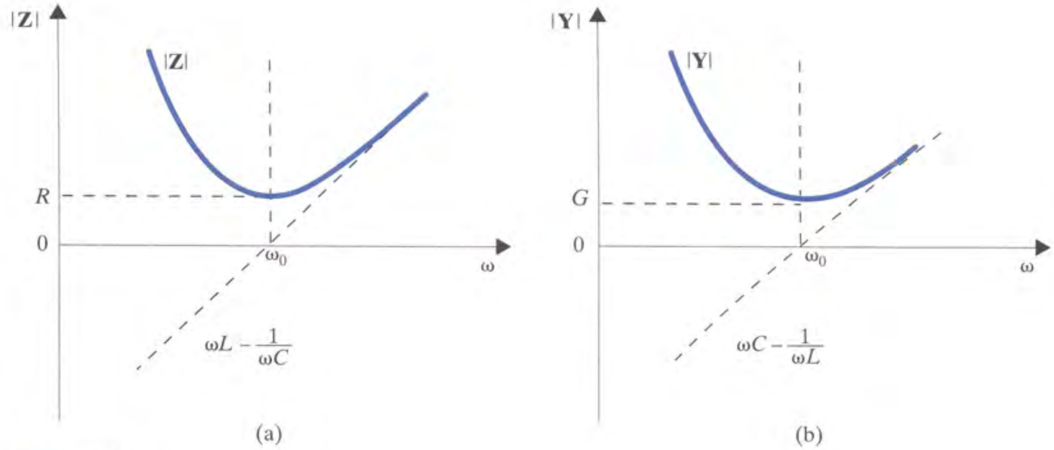


Figure 8.19
Frequency response of (a) a series and (b) a parallel RLC circuit.

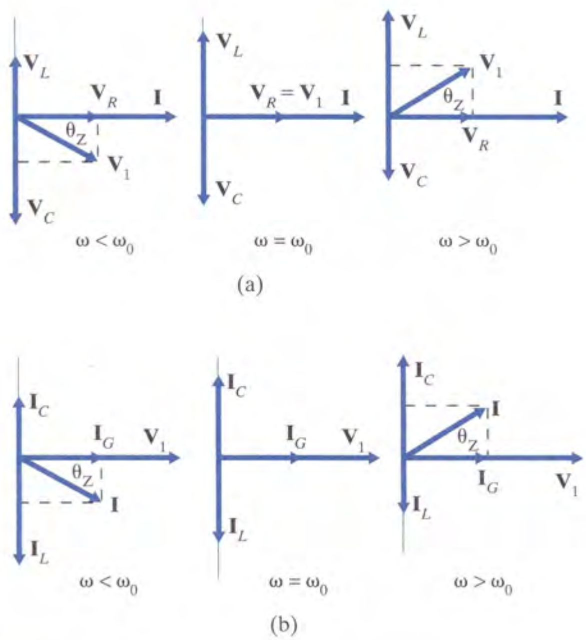


Figure 8.20
Phasor diagrams for (a) a series RLC circuit, and (b) a parallel GLC circuit.

LEARNING Hint

The quality factor is an important descriptor for resonant circuits.

For the series circuit we define what is commonly called the *quality factor Q* as

$$Q = \frac{\omega_0 L}{R} = \frac{1}{\omega_0 CR} = \frac{1}{R}\sqrt{\frac{L}{C}} \qquad \textbf{8.15}$$

Q is a very important factor in resonant circuits, and its ramifications will be illustrated throughout the remainder of this section.

LEARNING Example 8.7

Consider the network shown in Fig. 8.21. Let us determine the resonant frequency, the voltage across each element at resonance, and the value of the quality factor.

SOLUTION The resonant frequency is obtained from the expression

$$\begin{aligned}\omega_0 &= \frac{1}{\sqrt{LC}}\\ &= \frac{1}{\sqrt{(25)(10^{-3})(10)(10^{-6})}}\\ &= 2000 \text{ rad/s}\end{aligned}$$

At this resonant frequency

$$\mathbf{I} = \frac{\mathbf{V}}{\mathbf{Z}} = \frac{\mathbf{V}}{R} = 5\underline{/0^\circ} \text{ A}$$

Therefore,

$$\begin{aligned}\mathbf{V}_R &= (5\underline{/0^\circ})(2) = 10\underline{/0^\circ} \text{ V}\\ \mathbf{V}_L &= j\omega_0 L\mathbf{I} = 250\underline{/90^\circ} \text{ V}\\ \mathbf{V}_C &= \frac{\mathbf{I}}{j\omega_0 C} = 250\underline{/-90^\circ} \text{ V}\end{aligned}$$

Note the magnitude of the voltages across the inductor and capacitor with respect to the input voltage. Note also that these voltages are equal and are 180° out of phase with one another. Therefore, the phasor diagram for this condition is shown in Fig. 8.20a for $\omega = \omega_0$. The quality factor Q derived from Eq. (8.15) is

$$Q = \frac{\omega_0 L}{R} = \frac{(2)(10^3)(25)(10^{-3})}{2} = 25$$

It is interesting to note that the voltages across the inductor and capacitor can be written in terms of Q as

$$|\mathbf{V}_L| = \omega_0 L|\mathbf{I}| = \frac{\omega_0 L}{R}\mathbf{V}_S = Q\mathbf{V}_S$$

and

$$|\mathbf{V}_C| = \frac{|\mathbf{I}|}{\omega_0 C} = \frac{1}{\omega_0 CR}\mathbf{V}_S = Q\mathbf{V}_S$$

This analysis indicates that for a given current there is a resonant voltage rise across the inductor and capacitor that is equal to the product of Q and the applied voltage.

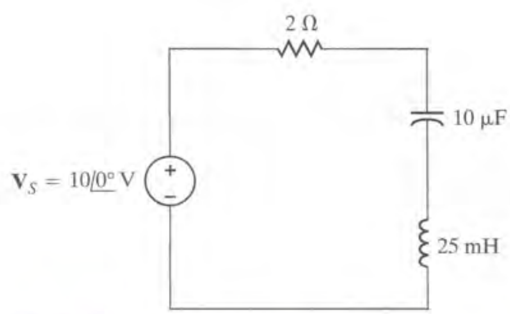


Figure 8.21 Series circuit.

LEARNING Example 8.8

In an undergraduate circuits laboratory, students are asked to construct an *RLC* network that will demonstrate resonance at $f = 1000$ Hz given a 0.02 H inductor that has a Q of 200. One student produced the circuit shown in Fig. 8.22, where the inductor's internal resistance is represented by R.

If the capacitor chosen to demonstrate resonance was an oil-impregnated paper capacitor rated at 300 V, let us determine the network parameters and the effect of this choice of capacitor.

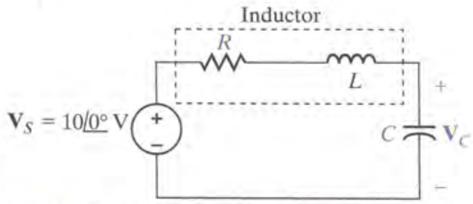


Figure 8.22 RLC series resonant network.

SOLUTION For resonance at 1000 Hz, the student found the required capacitor value using the expression

$$\omega_0 = 2\pi f_0 = \frac{1}{\sqrt{LC}}$$

which yields

$$C = 1.27\ \mu\text{F}$$

The student selected an oil-impregnated paper capacitor rated at 300 V. The resistor value was found using the expression for Q

$$Q = \frac{\omega_0 L}{R} = 200$$

or

$$R = 1.59\ \Omega$$

At resonance, the current would be

$$\mathbf{I} = \frac{\mathbf{V}_S}{R}$$

or

$$\mathbf{I} = 6.28\angle 0°\ \text{A}$$

When constructed, the current was measured to be only

$$\mathbf{I} \sim 1\angle 0°\ \text{mA}$$

This measurement clearly indicated that the impedance seen by the source was about 10 kΩ of resistance instead of 1.59 Ω—quite a drastic difference. Suspecting that the capacitor that was selected was the source of trouble, the student calculated what the capacitor voltage should be. If operated as designed, then at resonance,

$$\mathbf{V}_C = \frac{\mathbf{V}_S}{R}\left(\frac{1}{j\omega C}\right) = Q\mathbf{V}_S$$

or

$$\mathbf{V}_C = 2000\angle -90°\ \text{V}$$

which is more than six times the capacitor's rated voltage! This overvoltage had damaged the capacitor so that it did not function properly. When a new capacitor was selected and the source voltage reduced by a factor of 10, the network performed properly as a high Q circuit.

LEARNING EXTENSIONS

E8.8 Given the network in Fig. E8.8, find the value C that will place the circuit in resonance at 1800 rad/s.

ANSWER $C = 3.09\ \mu\text{F}$.

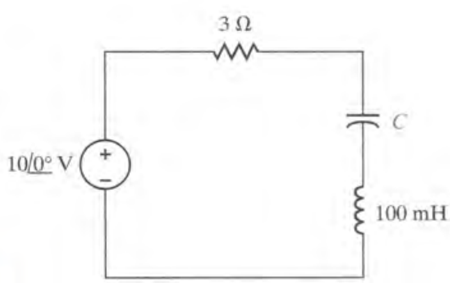


Figure E8.8

E8.9 Given the network in E8.8, determine the Q of the network and the magnitude of the voltage across the capacitor.

ANSWER $Q = 60$, $|\mathbf{V}_C| = 600$ V.

The impedance of the circuit in Fig. 8.18a is given by Eq. (8.10), which can be expressed as an admittance,

$$\mathbf{Y}(j\omega) = \frac{1}{R[1 + j(1/R)(\omega L - 1/\omega C)]}$$

$$= \frac{1}{R[1 + j(\omega L/R - 1/\omega CR)]}$$

$$= \frac{1}{R[1 + jQ(\omega L/RQ - 1/\omega CRQ)]} \qquad \mathbf{8.16}$$

Using the fact that $Q = \omega_0 L/R = 1/\omega_0 CR$ Eq. (8.16) becomes

$$\mathbf{Y}(j\omega) = \frac{1}{R[1 + jQ(\omega/\omega_0 - \omega_0/\omega)]} \qquad \mathbf{8.17}$$

Since $\mathbf{I} = \mathbf{Y}\mathbf{V}_1$ and the voltage across the resistor is $\mathbf{V}_R = \mathbf{I}R$, then

$$\frac{\mathbf{V}_R}{\mathbf{V}_1} = \mathbf{G}_v(j\omega) = \frac{1}{1 + jQ(\omega/\omega_0 - \omega_0/\omega)} \qquad \mathbf{8.18}$$

and the magnitude and phase are

$$M(\omega) = \frac{1}{[1 + Q^2(\omega/\omega_0 - \omega_0/\omega)^2]^{1/2}} \qquad \mathbf{8.19}$$

and

$$\phi(\omega) = -\tan^{-1} Q\left(\frac{\omega}{\omega_0} - \frac{\omega_0}{\omega}\right) \qquad \mathbf{8.20}$$

The sketches for these functions are shown in Fig. 8.23. Note that the circuit has the form of a band-pass filter. The bandwidth as shown is the difference between the two half-power frequencies. Since power is proportional to the square of the magnitude, these two frequencies may be derived by setting the magnitude $M(\omega) = 1/\sqrt{2}$; that is,

$$\left|\frac{1}{1 + jQ(\omega/\omega_0 - \omega_0/\omega)}\right| = \frac{1}{\sqrt{2}}$$

Therefore,

$$Q\left(\frac{\omega}{\omega_0} - \frac{\omega_0}{\omega}\right) = \pm 1 \qquad \mathbf{8.21}$$

Solving this equation, we obtain four frequencies,

$$\omega = \pm\frac{\omega_0}{2Q} \pm \omega_0\sqrt{\left(\frac{1}{2Q}\right)^2 + 1} \qquad \mathbf{8.22}$$

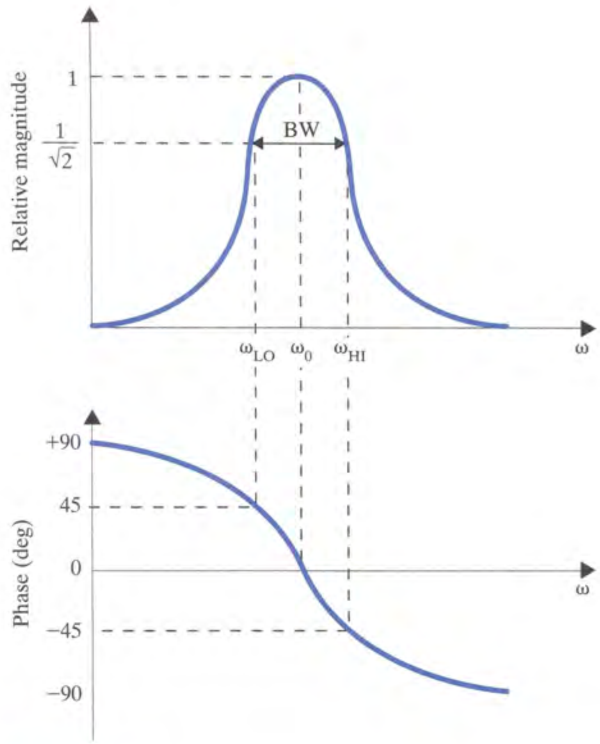


Figure 8.23
Magnitude and phase curves for Eq. (8.18).

LEARNING Hint

Half-power frequencies and their dependence on ω_0 and Q

Taking only the positive values, we obtain

$$\omega_{LO} = \omega_0\left[-\frac{1}{2Q} + \sqrt{\left(\frac{1}{2Q}\right)^2 + 1}\right] \qquad 8.23$$

$$\omega_{HI} = \omega_0\left[\frac{1}{2Q} + \sqrt{\left(\frac{1}{2Q}\right)^2 + 1}\right]$$

LEARNING Hint

The bandwidth is the difference between the half-power frequencies and a function of ω_0 and Q.

Subtracting these two equations yields the bandwidth as shown in Fig. 8.23:

$$\text{BW} = \omega_{HI} - \omega_{LO} = \frac{\omega_0}{Q} \qquad 8.24$$

and multiplying the two equations yields

$$\omega_0^2 = \omega_{LO}\omega_{HI} \qquad 8.25$$

which illustrates that the resonant frequency is the geometric mean of the two half-power frequencies. Recall that the half-power frequencies are the points at which the log-magnitude curve is down 3 dB from its maximum value. Therefore, the difference between the 3-dB frequencies, which is, of course, the bandwidth, is often called the 3-dB bandwidth.

LEARNING EXTENSION

E8.10 For the network in Fig. E8.8, compute the two half-power frequencies and the bandwidth of the network.

ANSWER
$\omega_{HI} = 1815$ rad/s,
$\omega_{LO} = 1785$ rad/s,
BW = 30 rad/s.

Equation (8.15) indicates the dependence of Q on R. A high-Q series circuit has a small value of R, and, as we will illustrate later, a high-Q parallel circuit has a relatively large value of R.

Equation (8.24) illustrates that the bandwidth is inversely proportional to Q. Therefore, the frequency selectivity of the circuit is determined by the value of Q. A high-Q circuit has a small bandwidth and, therefore, the circuit is very selective. The manner in which Q affects the frequency selectivity of the network is graphically illustrated in Fig. 8.24. Hence, if we pass a signal with a wide frequency range through a high-Q circuit, only the frequency components within the bandwidth of the network will not be attenuated; that is, the network acts like a band-pass filter.

Q has a more general meaning that we can explore via an energy analysis of the series circuit. Recall from Chapter 5 that an inductor stores energy in its magnetic field and a capacitor stores energy in its electric field. When a network is in resonance, there is a continuous exchange of energy between the magnetic field of the inductor and the electric field of the capacitor. During each half-cycle the energy stored in the inductor's magnetic field will vary from zero to a maximum value and back to zero again. The capacitor operates in a similar manner. The energy exchange takes place in the following way. During one quarter-cycle the capacitor absorbs energy as quickly as the inductor gives it up, and during the following one quarter-cycle the inductor absorbs energy as fast as it is released by the capacitor. Although the energy stored in each element is continuously varying, the total energy stored in the resonant circuit is constant and therefore not changing with time.

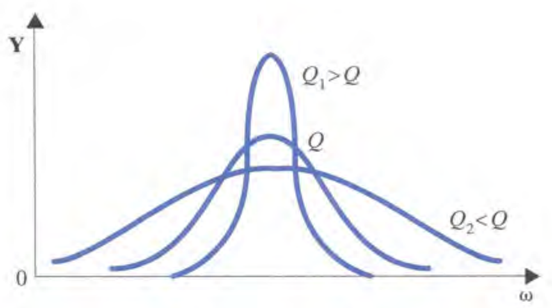


Figure 8.24
Network frequency response as a function of Q.

Q can also be defined as

$$Q = 2\pi \frac{W_S}{W_D} \tag{8.26}$$

where W_S is the maximum energy stored at resonance and W_D is the energy dissipated per cycle. The importance of this definition of Q stems from the fact that this expression is applicable to acoustic, electrical, and mechanical systems and therefore is generally considered to be the basic definition of Q.

LEARNING Example 8.9

Given a series circuit with $R = 2\ \Omega$, $L = 2$ mH, and $C = 5\ \mu$F, we wish to determine the resonant frequency, the quality factor, and the bandwidth for the circuit. Then we will determine the change in Q and the BW if R is changed from 2 to 0.2 Ω.

SOLUTION Using Eq. (8.12), we have

$$\omega_0 = \frac{1}{\sqrt{LC}} = \frac{1}{[(2)(10^{-3})(5)(10^{-6})]^{1/2}}$$
$$= 10^4 \text{ rad/s}$$

and therefore, the resonant frequency is $10^4/2\pi = 1592$ Hz.

The quality factor is

$$Q = \frac{\omega_0 L}{R} = \frac{(10^4)(2)(10^{-3})}{2}$$
$$= 10$$

and the bandwidth is

$$\text{BW} = \frac{\omega_0}{Q} = \frac{10^4}{10}$$
$$= 10^3 \text{ rad/s}$$

If R is changed to $R = 0.2\ \Omega$, the new value of Q is 100, and therefore the new BW is 10^2 rad/s.

LEARNING EXTENSIONS

E8.11 A series circuit is composed of $R = 2\ \Omega$, $L = 40$ mH, and $C = 100\ \mu$F. Determine the bandwidth of this circuit about its resonant frequency.

ANSWER BW = 50 rad/s, $\omega_0 = 500$ rad/s.

E8.12 A series *RLC* circuit has the following properties: $R = 4\ \Omega$, $\omega_0 = 4000$ rad/s, and the BW = 100 rad/s. Determine the values of L and C.

ANSWER $L = 40$ mH, $C = 1.56\ \mu$F.

LEARNING Example 8.10

We wish to determine the parameters R, L, and C so that the circuit shown in Fig. 8.25 operates as a band-pass filter with an ω_0 of 1000 rad/s and a bandwidth of 100 rad/s.

SOLUTION The voltage gain for the network is

$$\mathbf{G}_v(j\omega) = \frac{(R/L)j\omega}{(j\omega)^2 + (R/L)j\omega + 1/LC}$$

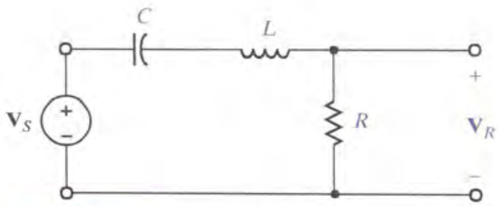


Figure 8.25 Series RLC circuit.

Hence,

$$\omega_0 = \frac{1}{\sqrt{LC}}$$

and since $\omega_0 = 10^3$,

$$\frac{1}{LC} = 10^6$$

The bandwidth is

$$\text{BW} = \frac{\omega_0}{Q}$$

Then

$$Q = \frac{\omega_0}{\text{BW}} = \frac{1000}{100} = 10$$

However,

$$Q = \frac{\omega_0 L}{R}$$

Therefore,

$$\frac{1000L}{R} = 10$$

Note that we have two equations in the three unknown circuit parameters R, L, and C. Hence, if we select $C = 1\ \mu\text{F}$, then

$$L = \frac{1}{10^6 C} = 1\text{ H}$$

and

$$\frac{1000(1)}{R} = 10$$

yields

$$R = 100\ \Omega$$

Therefore, the parameters $R = 100\ \Omega$, $L = 1$ H, and $C = 1\ \mu$F will produce the proper filter characteristics.

In Examples 8.7 and 8.8 we found that the voltage across the capacitor or inductor in the series resonant circuit could be quite high. In fact, it was equal to Q times the magnitude of the source voltage. With this in mind, let us reexamine this network as shown in Fig. 8.26. The output voltage for the network is

$$\mathbf{V}_o = \left(\frac{1/j\omega C}{R + j\omega L + 1/j\omega C}\right)\mathbf{V}_S$$

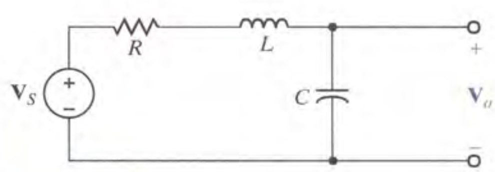


Figure 8.26
Series resonant circuit.

which can be written as

$$\mathbf{V}_o = \frac{\mathbf{V}_S}{1 - \omega^2 LC + j\omega CR}$$

The magnitude of this voltage can be expressed as

$$|\mathbf{V}_o| = \frac{|\mathbf{V}_S|}{\sqrt{(1 - \omega^2 LC)^2 + (\omega CR)^2}} \quad \textbf{8.27}$$

In view of the previous discussion, we might assume that the maximum value of the output voltage would occur at the resonant frequency ω_0. Let us see whether this assumption is correct. The frequency at which $|\mathbf{V}_o|$ is maximum is the nonzero value of ω, which satisfies the equation

$$\frac{d|\mathbf{V}_o|}{d\omega} = 0 \quad \textbf{8.28}$$

If we perform the indicated operation and solve for the nonzero ω_{max}, we obtain

$$\omega_{max} = \sqrt{\frac{1}{LC} - \frac{1}{2}\left(\frac{R}{L}\right)^2} \quad \textbf{8.29}$$

By employing the relationships $\omega_0^2 = 1/LC$ and $Q = \omega_0 L/R$, the expression for ω_{max} can be written as

$$\begin{aligned}\omega_{max} &= \sqrt{\omega_0^2 - \frac{1}{2}\left(\frac{\omega_0}{Q}\right)^2} \\ &= \omega_0\sqrt{1 - \frac{1}{2Q^2}}\end{aligned} \quad \textbf{8.30}$$

Clearly, $\omega_{max} \neq \omega_0$; however, ω_0 closely approximates ω_{max} if the Q is high. In addition, if we substitute Eq. (8.30) into Eq. (8.27) and use the relationships $\omega_0^2 = 1/LC$ and $\omega_0^2 C^2 R^2 = 1/Q^2$, we find that

$$|\mathbf{V}_o|_{max} = \frac{Q|\mathbf{V}_S|}{\sqrt{1 - 1/4Q^2}} \quad \textbf{8.31}$$

Again, we see that $|\mathbf{V}_o|_{max} \approx Q|\mathbf{V}_S|$ if the network has a high Q.

LEARNING Example 8.11

Given the network in Fig. 8.26, we wish to determine ω_0 and ω_{max} for $R = 50\ \Omega$ and $R = 1\ \Omega$ if $L = 50$ mH and $C = 5\ \mu$F.

SOLUTION The network parameters yield

$$\begin{aligned}\omega_0 &= \frac{1}{\sqrt{LC}} \\ &= \frac{1}{\sqrt{(5)(10^{-2})(5)(10^{-6})}} \\ &= 2000 \text{ rad/s}\end{aligned}$$

If $R = 50\ \Omega$, then

$$Q = \frac{\omega_0 L}{R} = \frac{(2000)(0.05)}{50} = 2$$

and

$$\omega_{max} = \omega_0\sqrt{1 - \frac{1}{2Q^2}} = 2000\sqrt{1 - \frac{1}{8}} = 1871 \text{ rad/s}$$

If $R = 1\ \Omega$, then $Q = 100$ and $\omega_{max} = 2000$ rad/s.

MATLAB can be used to plot the frequency response of the network transfer function for $R = 50\ \Omega$ and $R = 1\ \Omega$. The transfer function is

$$\frac{\mathbf{V}_o}{\mathbf{V}_s} = \frac{1}{2.5 \times 10^{-7}(j\omega)^2 + 2.5 \times 10^{-4}(j\omega) + 1}$$

for $R = 50\ \Omega$ and

$$\frac{\mathbf{V}_o}{\mathbf{V}_s} = \frac{1}{2.5 \times 10^{-7}(j\omega)^2 + 5 \times 10^{-6}(j\omega) + 1}$$

for $R = 1\ \Omega$. The corresponding MATLAB programs are

```
>>B = [1];
>>A = [ 2.5e-7 2.5e-4 1];
>>freqs(B, A)
```

and

```
>>B = [ 1 ];
>>A = [2.5e-7 5e-6 1]
>>freqs(B,A)
```

The magnitude and phase characteristics for the network with $R = 50\ \Omega$ and $R = 1\ \Omega$ are shown in Figs. 8.27a and b, respectively.

Note that when the Q of the network is small, the frequency response is not selective and $\omega_0 \neq \omega_{max}$. However, if the Q is large, the frequency response is very selective and $\omega_0 \simeq \omega_{max}$.

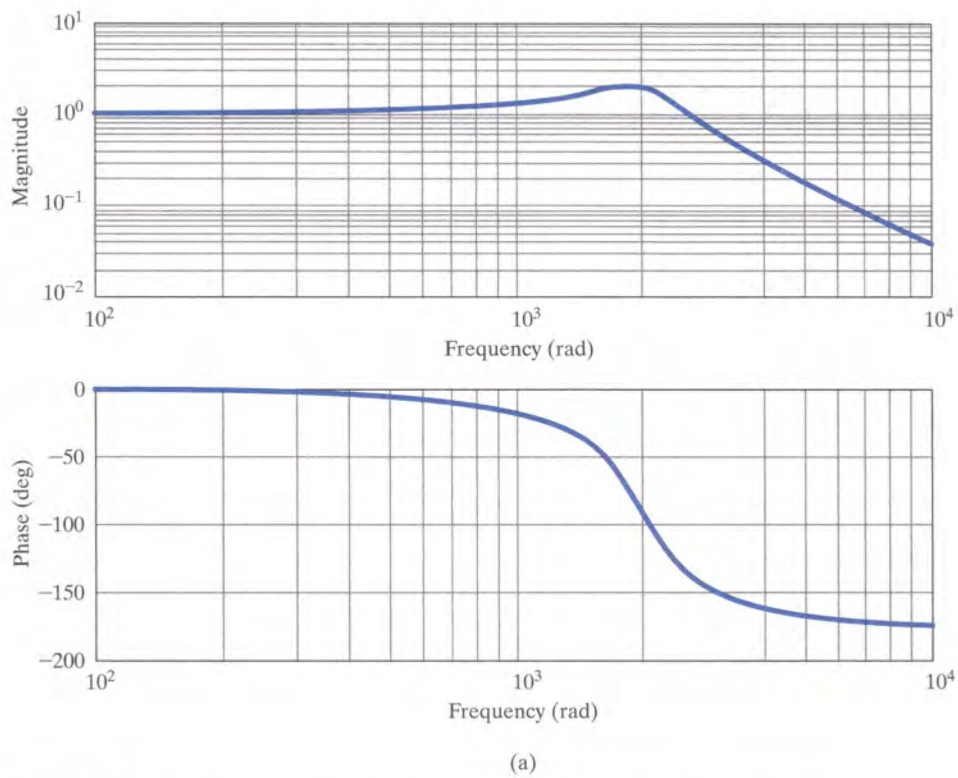
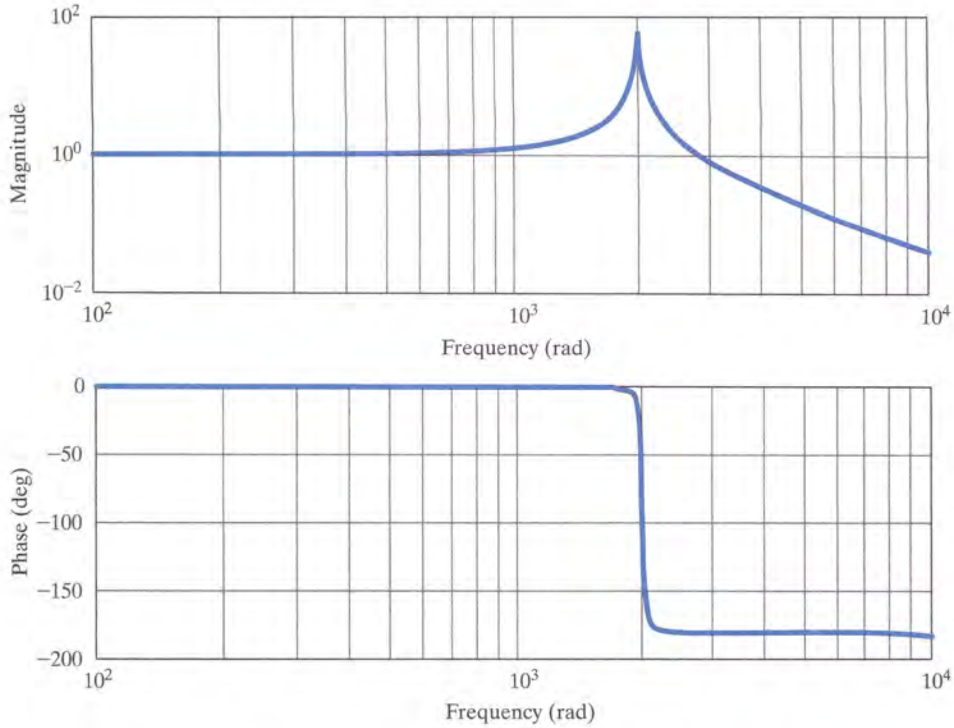


Figure 8.27
Frequency response plots for the network in Fig. 8.26 with (a) $R = 50\ \Omega$ and (b) $R = 1\ \Omega$.



(b)

Figure 8.27 *Continued*

LEARNING Example 8.12

On July 1, 1940, the third longest bridge in the nation, the Tacoma Narrows Bridge, was opened to traffic across Puget Sound in Washington. On November 7, 1940, the structure collapsed in what has become the most celebrated structural failure of that century. A photograph of the bridge, taken as it swayed back and forth just before breaking apart, is shown in Fig. 8.28. Explaining the disaster in quantitative terms is a feat for civil engineers and structures experts, and several theories have been presented. However, the one common denominator in each explanation is that wind blowing across the bridge caused the entire structure to resonate to such an extent that the bridge tore itself apart. One can theorize that the wind, fluctuating at a frequency near the natural frequency of the bridge (0.2 Hz), drove the structure into resonance. Thus, the bridge can be roughly modeled as a second-order system. Let us design an *RLC* resonance network to demonstrate the bridge's vertical movement and investigate the effect of the wind's frequency.

SOLUTION The *RLC* network shown in Fig. 8.29 is a second-order system in which $v_{\text{in}}(t)$ is analogous to vertical deflection of the bridge's roadway (1 volt = 1 foot). The values of C, L, R_A, and R_B can be derived from the data taken at the site and from scale models, as follows:

$$\text{vertical deflection at failure} \approx 4 \text{ feet}$$
$$\text{wind speed at failure} \approx 42 \text{ mph}$$
$$\text{resonant frequency} = f_0 \approx 0.2 \text{ Hz}$$

The output voltage can be expressed as

$$\mathbf{V}_o(j\omega) = \frac{j\omega\left(\dfrac{R_B}{L}\right)\mathbf{V}_{\text{in}}(j\omega)}{-\omega^2 + j\omega\left(\dfrac{R_A + R_B}{L}\right) + \dfrac{1}{LC}}$$

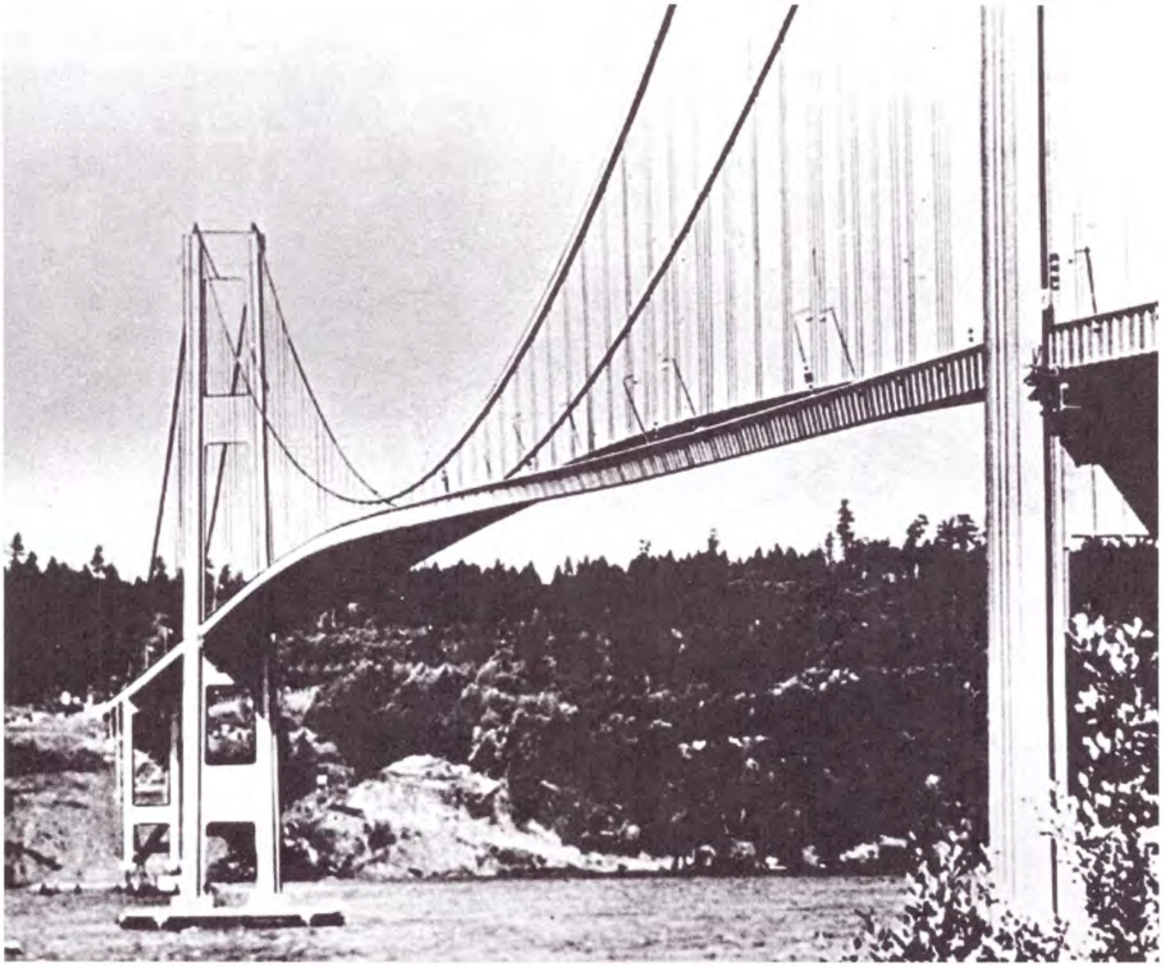


Figure 8.28 Tacoma Narrows Bridge on the verge of collapse. (Used with permission from Special Collection Division, University of Washington Libraries. Photo by Farguharson, negative number 12.)

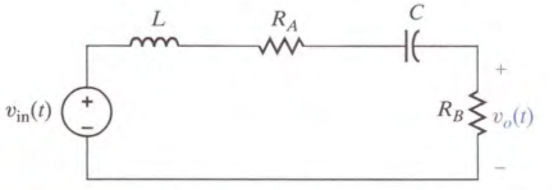


Figure 8.29 RLC resonance network for a simple Tacoma Narrows Bridge simulation.

from which we can easily extract the following expressions:

$$\omega_0 = \frac{1}{\sqrt{LC}} = 2\pi(0.2) \text{ rad/s}$$

$$2\zeta\omega_0 = \frac{R_A + R_B}{L}$$

and

$$\frac{\mathbf{V}_o(j\omega_0)}{\mathbf{V}_{in}(j\omega_0)} = \frac{R_B}{R_A + R_B} \approx \frac{4 \text{ feet}}{42 \text{ mph}}$$

Let us choose $R_B = 1\ \Omega$ and $R_A = 9.5\ \Omega$. Having no data for the damping ratio, ζ, we will select $L = 20$ H, which yields $\zeta = 0.209$ and $Q = 2.39$, which seem reasonable for such a large structure. Given the aforementioned choices, the required capacitor value is $C = 31.66$ mF. Using these circuit values, we now simulate the effect of 42 mph winds fluctuating at 0.05 Hz, 0.1 Hz, and 0.2 Hz using an ac analysis at the three frequencies of interest.

The results are shown in Fig. 8.30. Note that at 0.05 Hz the vertical deflection (1 ft/V) is only 0.44 feet, whereas at 0.1 Hz the bridge undulates about 1.07 feet. Finally, at the bridge's resonant frequency of 0.2 Hz, the bridge is oscillating 3.77 feet—catastrophic failure.

Clearly, we have used an extremely simplistic approach to modeling something as complicated as the Tacoma Narrows Bridge. A more accurate model is provided by K. Y. Billah and R. H. Scalan, "Resonance, Tacoma Narrows Bridge Failure, and Undergraduate Physics Textbooks," *American Journal of Physics* vol. 59, no. 2, pp. 118–124.

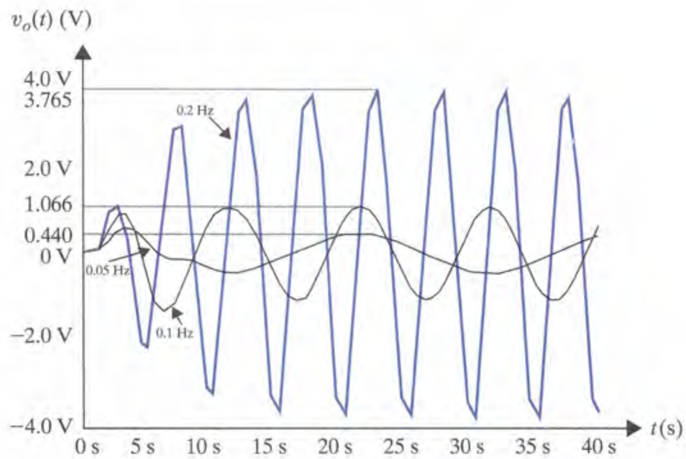


Figure 8.30
Simulated vertical deflection (1 volt = 1 foot) for the Tacoma Narrows Bridge for wind shift frequencies of 0.05, 0.1, and 0.2 Hz.

In our presentation of resonance thus far, we have focused most of our discussion on the series resonant circuit. We should recall, however, that the equations for the impedance of the series circuit and the admittance of the parallel circuit are similar. Therefore, the networks possess similar properties, as we illustrate in the following examples.

Consider the network shown in Fig. 8.31. The source current $\mathbf{I}_S$ can be expressed as

$$\begin{aligned}\mathbf{I}_S &= \mathbf{I}_G + \mathbf{I}_C + \mathbf{I}_L \\ &= \mathbf{V}_S G + j\omega C \mathbf{V}_S + \frac{\mathbf{V}_S}{j\omega L} \\ &= \mathbf{V}_S\left[G + j\left(\omega C - \frac{1}{\omega L}\right)\right]\end{aligned}$$

When the network is in resonance,

$$\mathbf{I}_S = G\mathbf{V}_S$$

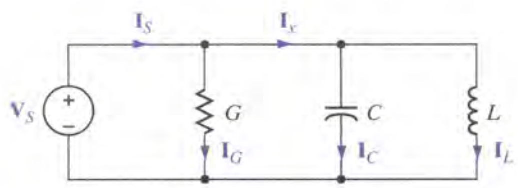


Figure 8.31
Parallel RLC circuit.

that is, all the source current flows through the conductance G. Does this mean that there is no current in L or C? Definitely not! $\mathbf{I}_C$ and $\mathbf{I}_L$ are equal in magnitude but 180° out of phase with one another. Therefore, $\mathbf{I}_x$, as shown in Fig. 8.31, is zero. In addition, if $G = 0$, the source current is zero. What is actually taking place, however, is an energy exchange between the electric field of the capacitor and the magnetic field of the inductor. As one increases, the other decreases, and vice versa.

LEARNING Example 8.13

The network in Fig. 8.31 has the following parameters:

$$\mathbf{V}_S = 120\underline{/0^\circ}\text{ V}, \qquad G = 0.01\ S,$$
$$C = 600\ \mu\text{F}, \qquad \text{and} \qquad L = 120\text{ mH}$$

If the source operates at the resonant frequency of the network, compute all the branch currents.

SOLUTION The resonant frequency for the network is

$$\begin{aligned}\omega_0 &= \frac{1}{\sqrt{LC}} \\ &= \frac{1}{\sqrt{(120)(10^{-3})(600)(10^{-6})}} \\ &= 117.85\text{ rad/s}\end{aligned}$$

At this frequency

$$\mathbf{Y}_C = j\omega_0 C = j7.07 \times 10^{-2}\text{ S}$$

and

$$\mathbf{Y}_L = -j\left(\frac{1}{\omega_0 L}\right) = -j7.07 \times 10^{-2}\text{ S}$$

The branch currents are then

$$\begin{aligned}\mathbf{I}_G &= G\mathbf{V}_S = 1.2\underline{/0^\circ}\text{ A} \\ \mathbf{I}_C &= \mathbf{Y}_C\mathbf{V}_S = 8.49\underline{/90^\circ}\text{ A} \\ \mathbf{I}_L &= \mathbf{Y}_L\mathbf{V}_S = 8.49\underline{/-90^\circ}\text{ A}\end{aligned}$$

and

$$\begin{aligned}\mathbf{I}_S &= \mathbf{I}_G + \mathbf{I}_C + \mathbf{I}_L \\ &= \mathbf{I}_G = 1.2\underline{/0^\circ}\text{ A}\end{aligned}$$

As the analysis indicates, the source supplies only the losses in the resistive element. In addition, the source voltage and current are in phase and, therefore, the power factor is unity.

LEARNING Example 8.14

Given the parallel RLC circuit in Fig. 8.32,

(a) Derive the expression for the resonant frequency, the half-power frequencies, the bandwidth, and the quality factor for the transfer characteristic $\mathbf{V}_{\text{out}}/\mathbf{I}_{\text{in}}$ in terms of the circuit parameters R, L, and C.

(b) Compute the quantities in part (a) if $R = 1\text{ k}\Omega$, $L = 10\text{ mH}$, and $C = 100\ \mu\text{F}$.

SOLUTION (a) The output voltage can be written as

$$\mathbf{V}_{\text{out}} = \frac{\mathbf{I}_{\text{in}}}{\mathbf{Y}_T}$$

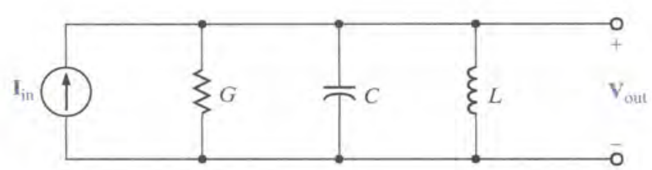


Figure 8.32
Circuit used in Example 8.14.

(continued)

and, therefore, the magnitude of the transfer characteristic can be expressed as

$$\left|\frac{\mathbf{V}_{out}}{\mathbf{I}_{in}}\right| = \frac{1}{\sqrt{(1/R^2) + (\omega C - 1/\omega L)^2}}$$

The transfer characteristic is a maximum at the resonant frequency

$$\omega_0 = \frac{1}{\sqrt{LC}} \qquad \textbf{8.32}$$

and at this frequency

$$\left|\frac{\mathbf{V}_{out}}{\mathbf{I}_{in}}\right|_{max} = R \qquad \textbf{8.33}$$

As demonstrated earlier, at the half-power frequencies the magnitude is equal to $1/\sqrt{2}$ of its maximum value, and hence the half-power frequencies can be obtained from the expression

$$\frac{1}{\sqrt{(1/R^2) + (\omega C - 1/\omega L)^2}} = \frac{R}{\sqrt{2}}$$

Solving this equation and taking only the positive values of ω yields

$$\omega_{LO} = -\frac{1}{2RC} + \sqrt{\frac{1}{(2RC)^2} + \frac{1}{LC}} \qquad \textbf{8.34}$$

and

$$\omega_{HI} = \frac{1}{2RC} + \sqrt{\frac{1}{(2RC)^2} + \frac{1}{LC}} \qquad \textbf{8.35}$$

Subtracting these two half-power frequencies yields the bandwidth

$$\begin{aligned} \text{BW} &= \omega_{HI} - \omega_{LO} \\ &= \frac{1}{RC} \end{aligned} \qquad \textbf{8.36}$$

Therefore, the quality factor is

$$\begin{aligned} Q &= \frac{\omega_0}{\text{BW}} \\ &= \frac{RC}{\sqrt{LC}} \\ &= R\sqrt{\frac{C}{L}} \end{aligned} \qquad \textbf{8.37}$$

Using Eqs. (8.32), (8.36), and (8.37), we can write Eqs. (8.34) and (8.35) as

$$\omega_{LO} = \omega_0\left[\frac{-1}{2Q} + \sqrt{\frac{1}{(2Q)^2} + 1}\right] \qquad \textbf{8.38}$$

$$\omega_{HI} = \omega_0\left[\frac{1}{2Q} + \sqrt{\frac{1}{(2Q)^2} + 1}\right] \qquad \textbf{8.39}$$

(b) Using the values given for the circuit components, we find that

$$\omega_0 = \frac{1}{\sqrt{(10^{-2})(10^{-4})}} = 10^3 \text{ rad/s}$$

The half-power frequencies are

$$\begin{aligned} \omega_{LO} &= \frac{-1}{(2)(10^3)(10^{-4})} + \sqrt{\frac{1}{[(2)(10^{-1})]^2} + 10^6} \\ &= 995 \text{ rad/s} \end{aligned}$$

and

$$\omega_{HI} = 1005 \text{ rad/s}$$

Therefore, the bandwidth is

$$\text{BW} = \omega_{HI} = \omega_{LO} = 10 \text{ rad/s}$$

and

$$\begin{aligned} Q &= 10^3\sqrt{\frac{10^{-4}}{10^{-2}}} \\ &= 100 \end{aligned}$$

LEARNING Example 8.15

Two radio stations, WHEW and WHAT, broadcast in the same listening area: WHEW broadcasts at 100 MHz and WHAT at 98 MHz. A single-stage tuned amplifier, such as that shown in Fig. 8.33, can be used as a tuner to filter out one of the stations. However, single-stage tuned amplifiers have poor selectivity due to their wide bandwidths. To reduce the bandwidth (increase the quality factor) of single-stage tuned amplifiers, designers employ a technique called synchronous tuning. In this process, identical tuned amplifiers are cascaded. To demonstrate this phenomenon, let us generate a Bode plot for the amplifier shown in Fig. 8.33 when it is tuned to WHEW (100 MHz), using one, two, three, and four stages of amplification.

SOLUTION Using the circuit for a single-stage amplifier shown in Fig. 8.33, we can cascade the stages to form a four-stage synchronously tuned amplifier. If we now plot the frequency response over the range from 90 MHz to 110 MHz, which is easily done using PSPICE, we obtain the Bode plot shown in Fig. 8.34.

From the Bode plot in Fig. 8.34 we see that increasing the number of stages does indeed decrease the bandwidth without altering the center frequency. As a result, the quality factor and selectivity increase. Accordingly, as we add stages, the gain at 98 MHz (WHAT's frequency) decreases and that station is "tuned out."

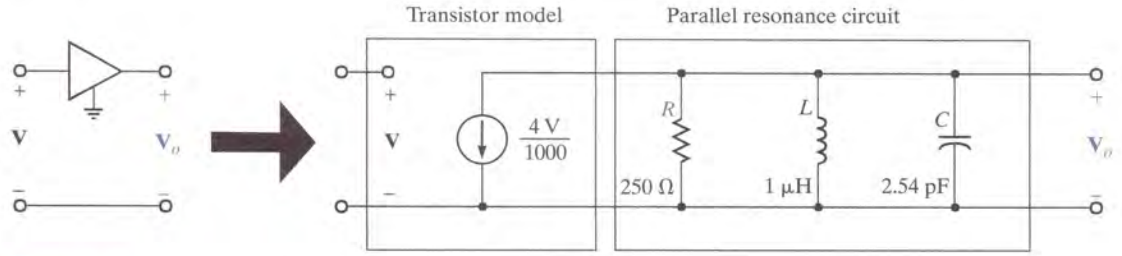


Figure 8.33
Single-stage tuned amplifier.

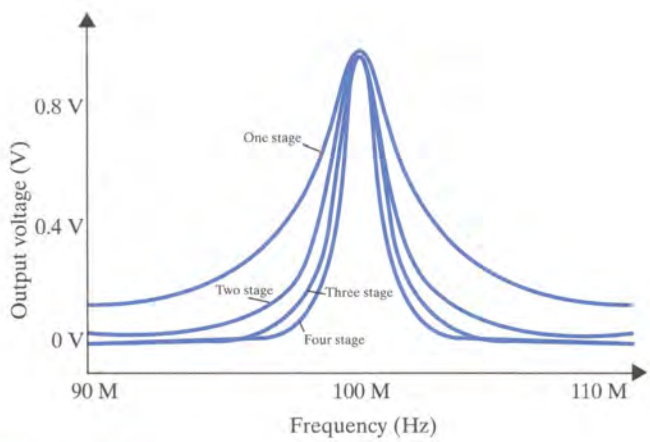


Figure 8.34
Bode plots for one-, two-, three-, and four-stage tuned amplifiers.

LEARNING EXTENSIONS

E8.13 A parallel *RLC* circuit has the following parameters: $R = 2\ \text{k}\Omega$, $L = 20$ mH, and $C = 150\ \mu\text{F}$. Determine the resonant frequency, the Q, and the bandwidth of the circuit.

ANSWER $\omega_0 = 577$ rad/s, $Q = 173$, and BW $= 3.33$ rad/s.

E8.14 A parallel *RLC* circuit has the following parameters: $R = 6\ \text{k}\Omega$, BW $= 1000$ rad/s, and $Q = 120$. Determine the values of L, C, and ω_0.

ANSWER $L = 417.5\ \mu\text{H}$, $C = 0.167\ \mu\text{F}$, and $\omega_0 = 119{,}760$ rad/s.

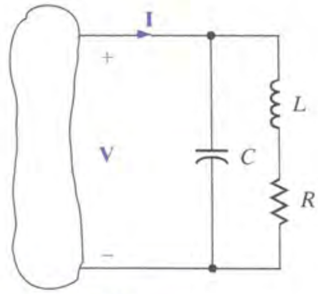


Figure 8.35
Practical parallel resonant circuit.

In general, the resistance of the winding of an inductor cannot be neglected, and hence a more practical parallel resonant circuit is the one shown in Fig. 8.35. The input admittance of this circuit is

$$\begin{aligned}\mathbf{Y}(j\omega) &= j\omega C + \frac{1}{R + j\omega L} \\ &= j\omega C + \frac{R - j\omega L}{R^2 + \omega^2 L^2} \\ &= \frac{R}{R^2 + \omega^2 L^2} + j\left(\omega C - \frac{\omega L}{R^2 + \omega^2 L^2}\right)\end{aligned}$$

The frequency at which the admittance is purely real is

$$\omega_r C - \frac{\omega_r L}{R^2 + \omega_r^2 L^2} = 0$$

$$\omega_r = \sqrt{\frac{1}{LC} - \frac{R^2}{L^2}} \qquad \textbf{8.40}$$

LEARNING Example 8.16

Given the tank circuit in Fig. 8.36, let us determine ω_0 and ω_r for $R = 50\ \Omega$ and $R = 5\ \Omega$.

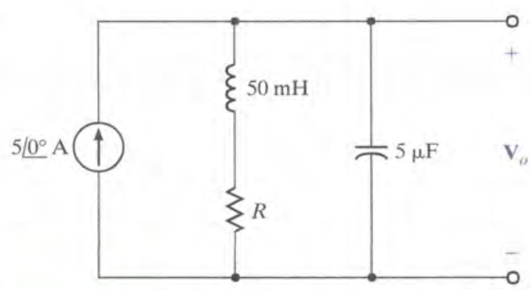


Figure 8.36 *Tank circuit used in Example 8.16.*

SOLUTION Using the network parameter values, we obtain

$$\begin{aligned}\omega_0 &= \frac{1}{\sqrt{LC}} \\ &= \frac{1}{\sqrt{(0.05)(5)(10^{-6})}} \\ &= 2000 \text{ rad/s} \\ f_0 &= 318.3 \text{ Hz}\end{aligned}$$

If $R = 50\ \Omega$, then

$$\omega_r = \sqrt{\frac{1}{LC} - \frac{R^2}{L^2}}$$
$$= \sqrt{\frac{1}{(0.05)(5)(10^{-6})} - \left(\frac{50}{0.05}\right)^2}$$
$$= 1732 \text{ rad/s}$$
$$f_r = 275.7 \text{ Hz}$$

If $R = 5\ \Omega$, then

$$\omega_r = \sqrt{\frac{1}{(0.05)(5)(10^{-6})} - \left(\frac{5}{0.05}\right)^2}$$
$$= 1997 \text{ rad/s}$$
$$f_r = 317.9 \text{ Hz}$$

Note that as $R \to 0$, $\omega_r \to \omega_0$. This fact is also illustrated in the frequency-response curves in Figs. 8.37a and b, where we have plotted $|\mathbf{V}_o|$ versus frequency for $R = 50\ \Omega$ and $R = 5\ \Omega$, respectively.

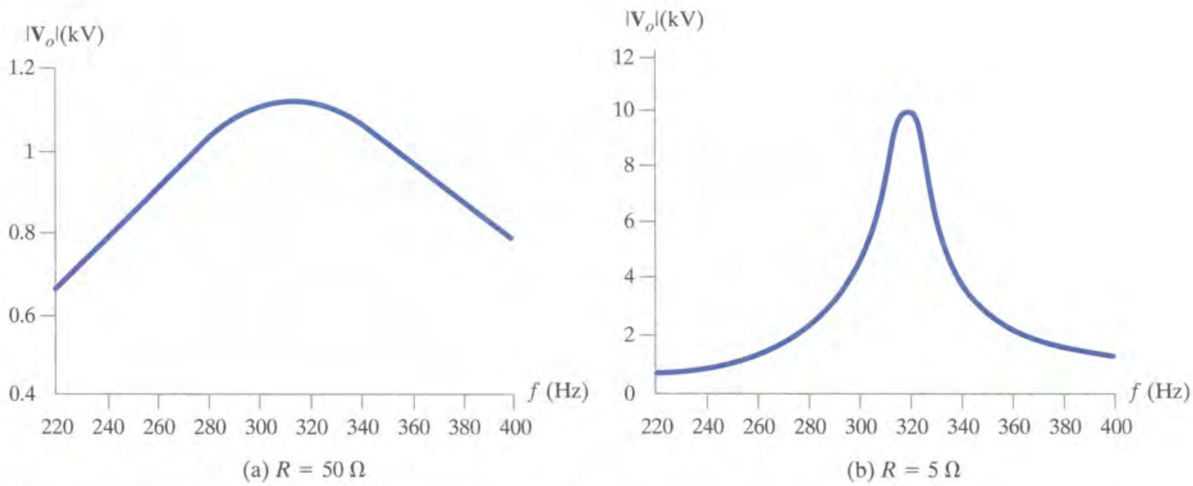


Figure 8.37 *Frequency-response curves for Example 8.16.*

Let us now try to relate some of the things we have learned about resonance to the Bode plots we presented earlier. The admittance for the series resonant circuit is

$$\mathbf{Y}(j\omega) = \frac{1}{R + j\omega L + 1/j\omega C}$$
$$= \frac{j\omega C}{(j\omega)^2 LC + j\omega CR + 1} \qquad \textbf{8.41}$$

The standard form for the quadratic factor is

$$(j\omega\tau)^2 + 2\zeta\omega\tau j + 1$$

where $\tau = 1/\omega_0$, and hence in general the quadratic factor can be written as

$$\frac{(j\omega)^2}{\omega_0^2} + \frac{2\zeta\omega}{\omega_0} j + 1 \qquad \textbf{8.42}$$

If we now compare this form of the quadratic factor with the denominator of $\mathbf{Y}(j\omega)$, we find that

$$\omega_0^2 = \frac{1}{LC}$$

$$\frac{2\zeta}{\omega_0} = CR$$

and therefore,

$$\zeta = \frac{R}{2}\sqrt{\frac{C}{L}}$$

However, from Eq. (8.15),

$$Q = \frac{1}{R}\sqrt{\frac{L}{C}}$$

and hence,

$$Q = \frac{1}{2\zeta} \qquad \textbf{8.43}$$

To illustrate the significance of this equation, consider the Bode plot for the function $\mathbf{Y}(j\omega)$. The plot has an initial slope of +20 dB/decade due to the zero at the origin. If $\zeta > 1$, the poles represented by the quadratic factor in the denominator will simply roll off the frequency response, as illustrated in Fig. 8.12a, and at high frequencies the slope of the composite characteristic will be −20 dB/decade. Note from Eq. (8.43) that if $\zeta > 1$, the Q of the circuit is very small. However, if $0 < \zeta < 1$, the frequency response will peak as shown in Fig. 8.12a, and the sharpness of the peak will be controlled by ζ. If ζ is very small, the peak of the frequency response is very narrow, the Q of the network is very large, and the circuit is very selective in filtering the input signal. Equation (8.43) and Fig. 8.24 illustrate the connections among the frequency response, the Q, and the ζ of a network.

8.4 Scaling

Throughout this book we have employed a host of examples to illustrate the concepts being discussed. In many cases the actual values of the parameters were unrealistic in a practical sense, even though they may have simplified the presentation. In this section we illustrate how to *scale* the circuits to make them more realistic.

LEARNING Hint

Magnitude or impedance scaling

There are two ways to scale a circuit: *magnitude* or *impedance scaling* and *frequency scaling*. To magnitude scale a circuit, we simply multiply the impedance of each element by a scale factor K_M. Therefore, a resistor R becomes $K_M R$. Multiplying the impedance of an inductor $j\omega L$ by K_M yields a new inductor $K_M L$, and multiplying the impedance of a capacitor $1/j\omega C$ by K_M yields a new capacitor C/K_M. Therefore, in magnitude scaling,

$$\begin{aligned} R' &\rightarrow K_M R \\ L' &\rightarrow K_M L \\ C' &\rightarrow \frac{C}{K_M} \end{aligned} \qquad \textbf{8.44}$$

since

$$\omega_0' = \frac{1}{\sqrt{L'C'}} = \frac{1}{\sqrt{K_M LC/K_M}} = \omega_0$$

and Q' is

$$Q' = \frac{\omega_0 L'}{R'} = \frac{\omega_0 K_M L}{K_M R} = Q$$

The resonant frequency, the quality factor, and therefore the bandwidth are unaffected by magnitude scaling.

In frequency scaling the scale factor is denoted as K_F. The resistor is frequency independent and, therefore, unaffected by this scaling. The new inductor L', which has the same impedance at the scaled frequency ω_1', must satisfy the equation

$$j\omega_1 L = j\omega_1' L'$$

where $\omega_1' = k_F \omega_1$. Therefore,

$$j\omega_1 L = jK_F \omega_1 L'$$

Hence, the new inductor value is

$$L' = \frac{L}{K_F}$$

Using a similar argument, we find that

$$C' = \frac{C}{K_F}$$

Therefore, to frequency scale by a factor K_F,

LEARNING Hint

Frequency scaling

$$R' \to R$$
$$L' \to \frac{L}{K_F} \qquad \textbf{8.45}$$
$$C' \to \frac{C}{K_F}$$

Note that

$$\omega_0' = \frac{1}{\sqrt{(L/K_F)(C/K_F)}} = K_F \omega_0$$

and

$$Q' = \frac{K_F \omega_0 L}{RK_F} = Q$$

and therefore,

$$\text{BW}' = K_F(\text{BW})$$

Hence, the resonant frequency and bandwidth of the circuit are affected by frequency scaling.

LEARNING Example 8.17

If the values of the circuit parameters in Fig 8.35 are $R = 2\ \Omega$, $L = 1$ H, and $C = \frac{1}{2}$F, let us determine the values of the elements if the circuit is magnitude scaled by a factor $K_M = 10^2$ and frequency scaled by a factor $K_F = 10^2$.

SOLUTION The magnitude scaling yields

$$R' = 2K_M = 200\ \Omega$$

$$L' = (1)K_M = 100\text{ H}$$

$$C' = \frac{1}{2}\frac{1}{K_M} = \frac{1}{200}\text{F}$$

Applying frequency scaling to these values yields the final results:

$$R'' = 200\ \Omega$$

$$L'' = \frac{100}{K_F} = 100\ \mu\text{H}$$

$$C'' = \frac{1}{200}\frac{1}{K_F} = 0.005\ \mu\text{F}$$

LEARNING EXTENSION

E8.15 An *RLC* network has the following parameter values: $R = 10\ \Omega$, $L = 1$ H, and $C = 2$ F. Determine the values of the circuit elements if the circuit is magnitude scaled by a factor of 100 and frequency scaled by a factor of 10,000.

ANSWER $R = 1\text{ k}\Omega$, $L = 10$ mH, $C = 2\ \mu$F.

8.5 Filter Networks

PASSIVE FILTERS A filter network is generally designed to pass signals with a specific frequency range and reject or attenuate signals whose frequency spectrum is outside this passband. The most common filters are *low-pass* filters, which pass low frequencies and reject high frequencies; *high-pass* filters, which pass high frequencies and block low frequencies; *band-pass* filters, which pass some particular band of frequencies and reject all frequencies outside the range; and *band-rejection* filters, which are specifically designed to reject a particular band of frequencies and pass all other frequencies.

The ideal frequency characteristic for a low-pass filter is shown in Fig. 8.38a. Also shown is a typical or physically realizable characteristic. Ideally, we would like the low-pass filter to pass all frequencies to some frequency ω_0 and pass no frequency above that value; however, it is not possible to design such a filter with linear circuit elements. Hence, we must be content to employ filters that we can actually build in the laboratory, and these filters have frequency characteristics that are simply not ideal.

A simple low-pass filter network is shown in Fig. 8.38b. The voltage gain for the network is

$$\mathbf{G}_v(j\omega) = \frac{1}{1 + j\omega RC} \tag{8.46}$$

which can be written as

$$\mathbf{G}_v(j\omega) = \frac{1}{1 + j\omega\tau} \qquad \textbf{8.47}$$

where $\tau = RC$, the time constant. The amplitude characteristic is

$$M(\omega) = \frac{1}{\left[1 + (\omega\tau)^2\right]^{1/2}} \qquad \textbf{8.48}$$

and the phase characteristic is

$$\phi(\omega) = -\tan^{-1} \omega\tau \qquad \textbf{8.49}$$

Note that at the break frequency, $\omega = \frac{1}{\tau}$, the amplitude is

$$M\left(\omega = \frac{1}{\tau}\right) = \frac{1}{\sqrt{2}} \qquad \textbf{8.50}$$

The break frequency is also commonly called the *half-power frequency*. This name is derived from the fact that if the voltage or current is $1/\sqrt{2}$ of its maximum value, then the power, which is proportional to the square of the voltage or current, is one-half its maximum value.

The magnitude, in decibels, and phase curves for this simple low-pass circuit are shown in Fig. 8.38c. Note that the magnitude curve is flat for low frequencies and rolls off at high frequencies. The phase shifts from 0° at low frequencies to −90° at high frequencies.

The ideal frequency characteristic for a high-pass filter is shown in Fig. 8.39a, together with a typical characteristic that we could achieve with linear circuit components. Ideally, the high-pass filter passes all frequencies above some frequency ω_0 and no frequencies below that value.

A simple high-pass filter network is shown in Fig. 8.39b. This is the same network as shown in Fig. 8.38b, except that the output voltage is taken across the resistor. The voltage gain for this network is

$$\mathbf{G}_v(j\omega) = \frac{j\omega\tau}{1 + j\omega\tau} \qquad \textbf{8.51}$$

where once again $\tau = RC$. The magnitude of this function is

$$M(\omega) = \frac{\omega\tau}{\left[1 + (\omega\tau)^2\right]^{1/2}} \qquad \textbf{8.52}$$

and the phase is

$$\phi(\omega) = \frac{\pi}{2} - \tan^{-1} \omega\tau \qquad \textbf{8.53}$$

The half-power frequency is $\omega = 1/\tau$, and the phase at this frequency is 45°.

The magnitude and phase curves for this high-pass filter are shown in Fig. 8.39c. At low frequencies the magnitude curve has a slope of +20 dB/decade due to the term $\omega\tau$ in the numerator of Eq. (8.52). Then at the break frequency the curve begins to flatten out. The phase curve is derived from Eq. (8.53).

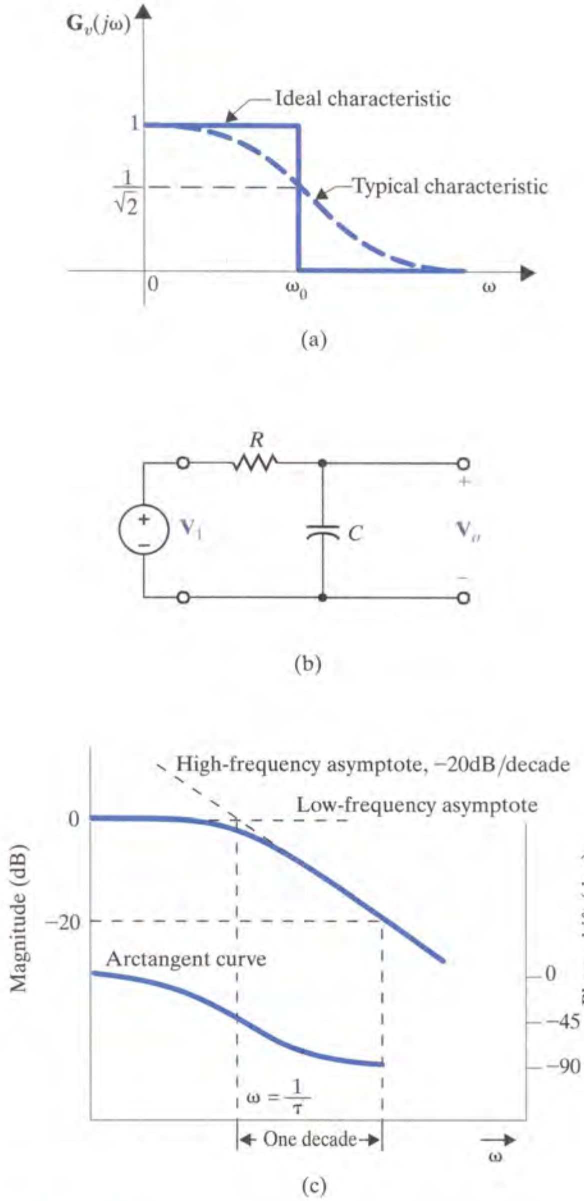


Figure 8.38
Low-pass filter circuit and its frequency characteristics.

Ideal and typical amplitude characteristics for simple band-pass and band-rejection filters are shown in Figs. 8.40a and b, respectively. Simple networks that are capable of realizing the typical characteristics of each filter are shown below the characteristics in Figs. 8.40c and d. ω_0 is the center frequency of the pass or rejection band and the frequency at which the maximum or minimum amplitude occurs. ω_{LO} and ω_{HI} are the lower and upper break frequencies or *cut-off frequencies*, where the amplitude is $1/\sqrt{2}$ of the maximum value. The width of the pass or rejection band is called *bandwidth*, and hence

$$\text{BW} = \omega_{HI} - \omega_{LO} \tag{8.54}$$

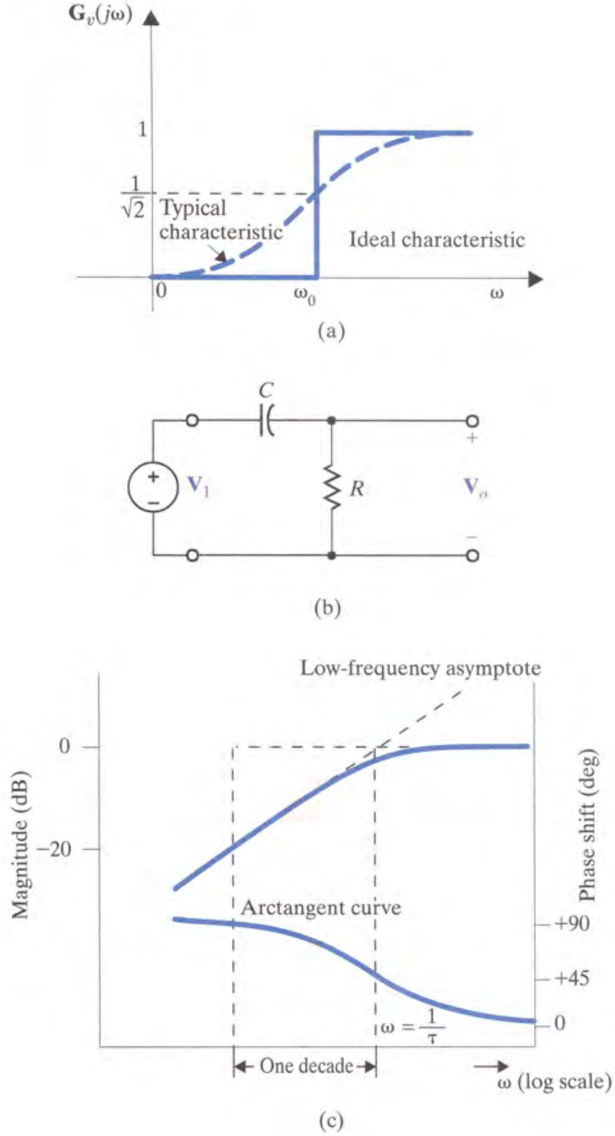


Figure 8.39
High-pass filter circuit and frequency characteristics.

To illustrate these points, let us consider the band-pass filter. The voltage transfer function is

$$\mathbf{G}_v(j\omega) = \frac{R}{R + j(\omega L - 1/\omega C)}$$

and, therefore, the amplitude characteristic is

$$M(\omega) = \frac{RC\omega}{\sqrt{(RC\omega)^2 + (\omega^2 LC - 1)^2}}$$

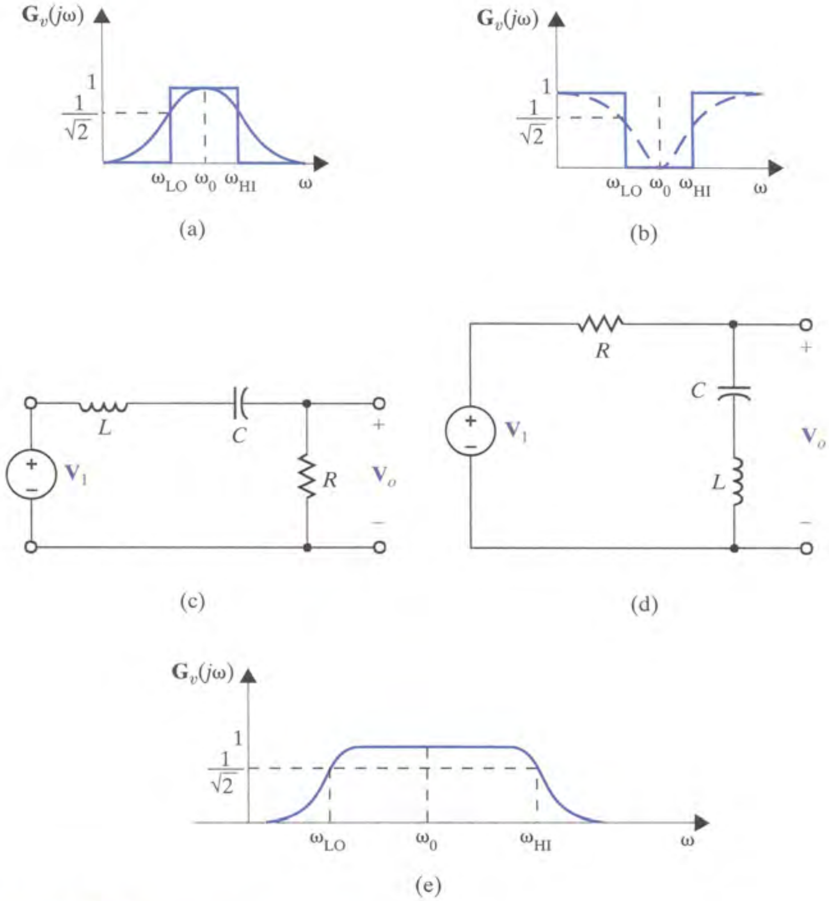


Figure 8.40
Band-pass and band-rejection filters and characteristics.

At low frequencies

$$M(\omega) \approx \frac{RC\omega}{1} \approx 0$$

At high frequencies

$$M(\omega) \simeq \frac{RC\omega}{\omega^2 LC} \approx \frac{R}{\omega L} \approx 0$$

In the midfrequency range $(RC\omega)^2 \gg (\omega^2 LC - 1)^2$, and thus $M(\omega) \approx 1$. Therefore, the frequency characteristic for this filter is shown in Fig. 8.40e. The center frequency is $\omega_0 = 1/\sqrt{LC}$. At the lower cutoff frequency

$$\omega^2 LC - 1 = -RC\omega$$

or

$$\omega^2 + \frac{R\omega}{L} - \omega_0^2 = 0$$

Solving this expression for ω_{LO}, we obtain

$$\omega_{LO} = \frac{-(R/L) + \sqrt{(R/L)^2 + 4\omega_0^2}}{2}$$

At the upper cutoff frequency

$$\omega^2 LC - 1 = +RC\omega$$

or

$$\omega^2 - \frac{R}{L}\omega - \omega_0^2 = 0$$

Solving this expression for ω_{HI}, we obtain

$$\omega_{HI} = \frac{+(R/L) + \sqrt{(R/L)^2 + 4\omega_0^2}}{2}$$

Therefore, the bandwidth of the filter is

$$BW = \omega_{HI} - \omega_{LO} = \frac{R}{L}$$

LEARNING Example 8.18

Consider the frequency-dependent network in Fig. 8.41. Given the following circuit parameter values: $L = 159\ \mu H$, $C = 159\ \mu F$, and $R = 10\ \Omega$, let us demonstrate that this one network can be used to produce a low-pass, high-pass, or band-pass filter.

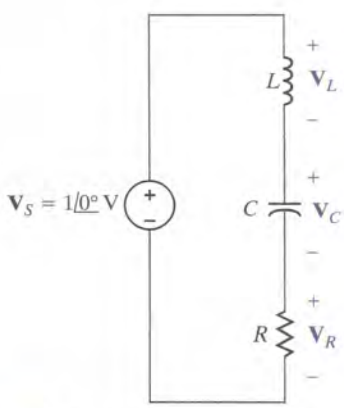


Figure 8.41 *Circuit used in Example 8.18.*

SOLUTION The voltage gain $\mathbf{V}_R/\mathbf{V}_S$ is found by voltage division to be

$$\frac{\mathbf{V}_R}{\mathbf{V}_S} = \frac{R}{j\omega L + R + 1/(j\omega C)} = \frac{j\omega\left(\frac{R}{L}\right)}{(j\omega)^2 + j\omega\left(\frac{R}{L}\right) + \frac{1}{LC}}$$

$$= \frac{(62.9 \times 10^3)j\omega}{-\omega^2 + (62.9 \times 10^3)j\omega + 39.6 \times 10^6}$$

which is the transfer function for a band-pass filter. At resonance, $\omega^2 = 1/LC$, and hence

$$\frac{\mathbf{V}_R}{\mathbf{V}_S} = 1$$

Now consider the gain $\mathbf{V}_L/\mathbf{V}_S$:

$$\frac{\mathbf{V}_L}{\mathbf{V}_S} = \frac{j\omega L}{j\omega L + R + 1/(j\omega C)} = \frac{-\omega^2}{(j\omega)^2 + j\omega\left(\frac{R}{L}\right) + \frac{1}{LC}}$$

(continued)

$$= \frac{-\omega^2}{-\omega^2 + (62.9 \times 10^3)j\omega + 39.6 \times 10^6}$$

which is a second-order high-pass filter transfer function. Again, at resonance,

$$\frac{\mathbf{V}_L}{\mathbf{V}_S} = \frac{j\omega L}{R} = jQ = j0.1$$

Similarly, the gain $\mathbf{V}_C/\mathbf{V}_S$ is

$$\frac{\mathbf{V}_C}{\mathbf{V}_S} = \frac{1/(j\omega C)}{j\omega L + R + 1/(j\omega C)} = \frac{\frac{1}{LC}}{(j\omega)^2 + j\omega\left(\frac{R}{L}\right) + \frac{1}{LC}}$$

$$= \frac{39.6 \times 10^6}{-\omega^2 + (62.9 \times 10^3)j\omega + 39.6 \times 10^6}$$

which is a second-order low-pass filter transfer function. At resonant frequency,

$$\frac{\mathbf{V}_C}{\mathbf{V}_S} = \frac{1}{j\omega CR} = -jQ = -j0.1$$

Thus, one circuit produces three different filters depending on where the output is taken. This can be seen in the Bode plot for each of the three voltages in Fig. 8.42, where $\mathbf{V}_S$ is set to $1\underline{/0°}$ V.

We know that Kirchhoff's voltage law must be satisfied at all times. Note from the Bode plot that the $\mathbf{V}_R + \mathbf{V}_C + \mathbf{V}_L$ also equals $\mathbf{V}_S$ at all frequencies! Finally, let us demonstrate KVL by adding $\mathbf{V}_R$, $\mathbf{V}_L$, and $\mathbf{V}_C$.

$$\mathbf{V}_L + \mathbf{V}_R + \mathbf{V}_C = \frac{\left((j\omega)^2 + j\omega\left(\frac{R}{L}\right) + \frac{1}{\sqrt{LC}}\right)\mathbf{V}_S}{(j\omega)^2 + j\omega\left(\frac{R}{L}\right) + \frac{1}{\sqrt{LC}}} = \mathbf{V}_S$$

Thus, even though $\mathbf{V}_S$ is distributed between the resistor, capacitor, and inductor based on frequency, the sum of the three voltages completely reconstructs $\mathbf{V}_S$.

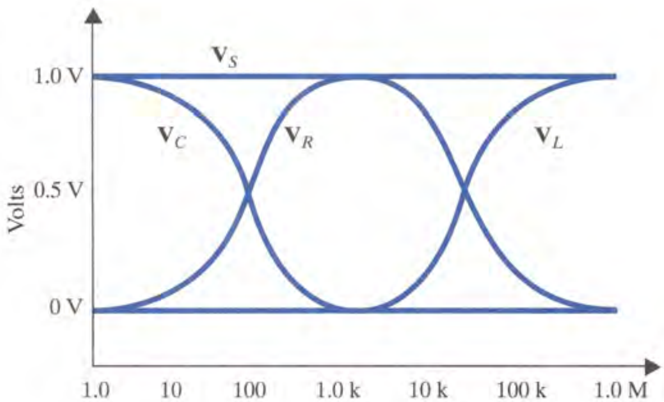


Figure 8.42
Bode plots for network in Fig. 8.41.

LEARNING Example 8.19

A telephone transmission system suffers from 60-Hz interference caused by nearby power utility lines. Let us use the network in Fig. 8.43 to design a simple notch filter to eliminate the 60-Hz interference.

SOLUTION The resistor R_{eq} represents the equivalent resistance of the telephone system to the right of the LC combination. The LC parallel combination has an equivalent impedance of

$$\mathbf{Z} = (j\omega L)//(1/j\omega C) = \frac{(L/C)}{j\omega L + 1/(j\omega C)}$$

Now the voltage transfer function is

$$\frac{\mathbf{V}_o}{\mathbf{V}_{in}} = \frac{R_{eq}}{R_{eq} + \mathbf{Z}} = \frac{R_{eq}}{R_{eq} + \frac{(L/C)}{j\omega L + (1/j\omega C)}}$$

which can be written

$$\frac{\mathbf{V}_o}{\mathbf{V}_{in}} = \frac{(j\omega)^2 + \frac{1}{LC}}{(j\omega)^2 + \left(\frac{j\omega}{R_{eq}C}\right) + \frac{1}{LC}}$$

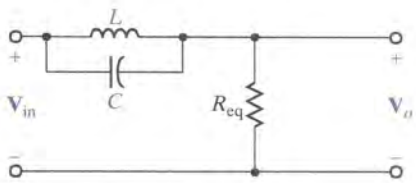


Figure 8.43 *Circuit used in Example 8.19.*

Note that at resonance, the numerator and thus $\mathbf{V}_o$ go to zero. We want resonance to occur at 60 Hz. Thus,

$$\omega_0 = \frac{1}{\sqrt{LC}} = 2\pi(60) = 120\pi$$

If we select $C = 100\ \mu F$, then the required value for L is 70.3 mH—both are reasonable values. To demonstrate the effectiveness of the filter, let the input voltage consist of a 60-Hz sinusoid and a 1000-Hz sinusoid of the form

$$v_{in}(t) = 1 \sin[(2\pi)60t] + 0.2 \sin[(2\pi)1000t]$$

The input and output waveforms are both shown in Fig. 8.44. Note that the output voltage, as desired, contains none of the 60-Hz interference.

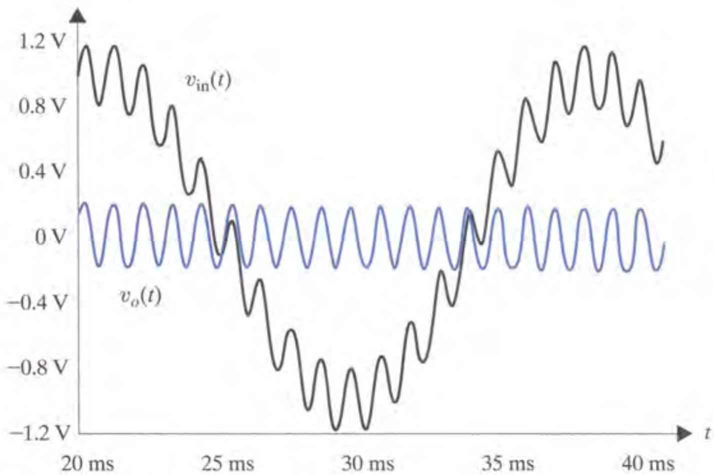


Figure 8.44
Transient analysis of the network in Fig. 8.43.

LEARNING EXTENSIONS

E8.16 Given the filter network shown in Fig. E8.16, sketch the magnitude characteristic of the Bode plot for $\mathbf{G}_v(j\omega)$.

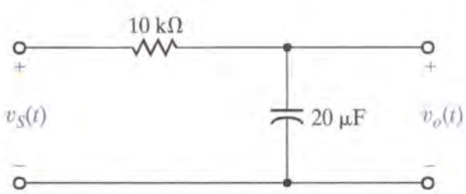
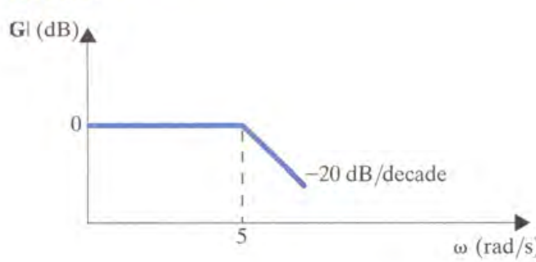


Figure E8.16

ANSWER



E8.17 Given the filter network in Fig. E8.17, sketch the magnitude characteristic of the Bode plot for $\mathbf{G}_v(j\omega)$.

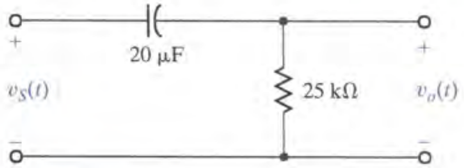
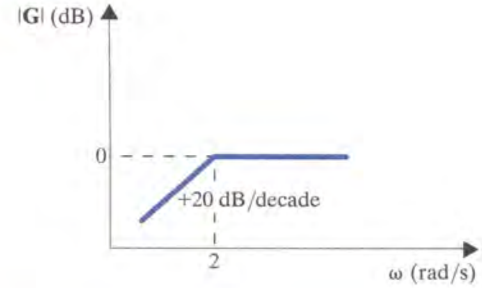


Figure E8.17

ANSWER



E8.18 A band-pass filter network is shown in Fig. E8.18. Sketch the magnitude characteristic of the Bode plot for $\mathbf{G}_v(j\omega)$.

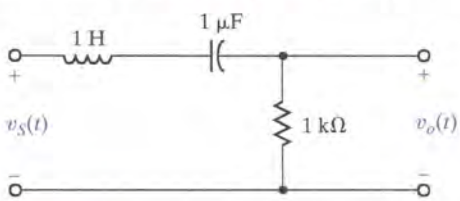
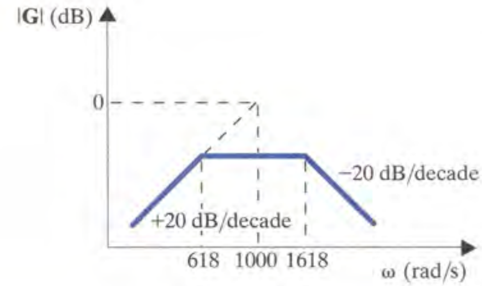


Figure E8.18

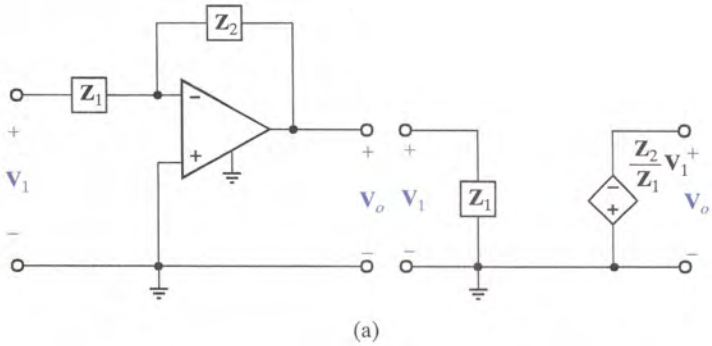
ANSWER



ACTIVE FILTERS In the preceding section we saw that the four major classes of filters (*low pass, high pass, band pass,* and *band rejection*) are realizable with simple, passive element circuits. However, passive filters do have some serious drawbacks. One obvious problem is the inability to generate a network with a gain > 1 since a purely passive network cannot add energy to a signal. Another serious drawback of passive filters is the need in many topologies for inductive elements. Inductors are generally expensive and are not usually available in precise values. In addition, inductors usually come in odd shapes (toroids, bobbins, E-cores, etc.) and are not easily handled by existing automated printed circuit board assembly machines. By applying operational amplifiers in linear feedback circuits, one can generate all of the primary filter types using only resistors, capacitors, and the op-amp integrated circuits themselves.

The equivalent circuits for the operational amplifiers derived in Chapter 3 are valid in the sinusoidal steady-state case also, when we replace the attendant resistors with impedances. The equivalent circuits for the basic inverting and noninverting op-amp circuits are shown in Figs. 8.45a and b, respectively. Particular filter characteristics are obtained by judiciously selecting the impedances $\mathbf{Z}_1$ and $\mathbf{Z}_2$. However, due to the nature of the op-amp's internal circuitry, it has limitations and is not always the best amplifier choice. Two examples are high-frequency active filters and low-voltage (< 3 V) circuitry. Given the evolution of the wireless market (cell phones, pagers, etc.) these applications will only grow in prominence. There is, however, an op-amp variant called the *operational transconductance amplifier*, or OTA, that performs excellently in these scenarios, allowing, for example, very advanced filters to be implemented on a single chip. In this text we will restrict our treatment of the OTA to this single application.

Advantages of the OTA over the op-amp can be deduced from the diagrams in Fig. 8.46. In the three-stage op-amp model, the input stage provides the large input resistance, converts



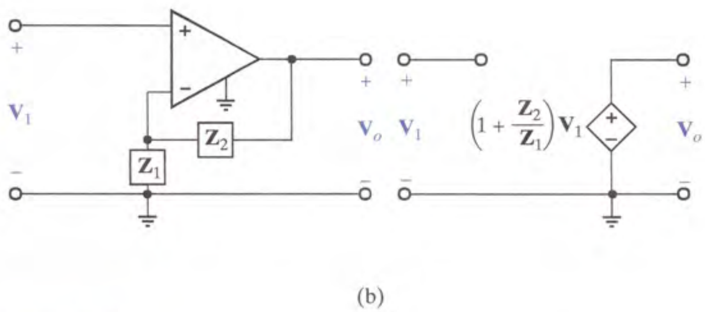
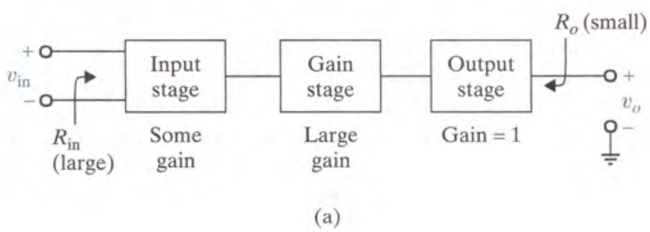


Figure 8.45
Equivalent circuits for the (a) inverting and (b) noninverting operational amplifier circuits.



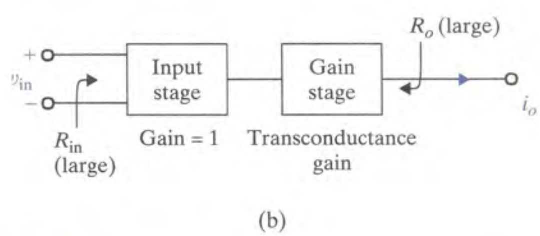


Figure 8.46
Block diagrams depicting the physical construction of the (a) op-amp and (b) the OTA.

the differential input voltage $v_{\text{in}}(t)$ to a single-ended (referenced to ground) voltage and produces some voltage gain. The gain stage provides the bulk of the op-amp's voltage gain. Finally, the output stage, has little if any voltage gain, but produces a low output resistance. This three-stage model accurately depicts the physical design of most op-amps.

Now consider the two-stage OTA model. As in the op-amp, the input stage provides a large input resistance, but its voltage gain is minimal. The gain stage is very similar to that of the op-amp in that the value of R_o is large. Unlike the op-amp, the output signal is a current rather than a voltage, yielding an overall transconductance gain in A/V or siemens. With no output stage, the OTA is more compact and consumes less power than the op-amp and has an overall output resistance of R_o—a large value. Having all of the OTA's gain in a single stage further simplifies the internal design, resulting in a simple, fast, compact amplifier that can be efficiently replicated many times on a single silicon chip. The schematic symbol for the OTA and a simpler model are shown in Fig. 8.47a and b, respectively.

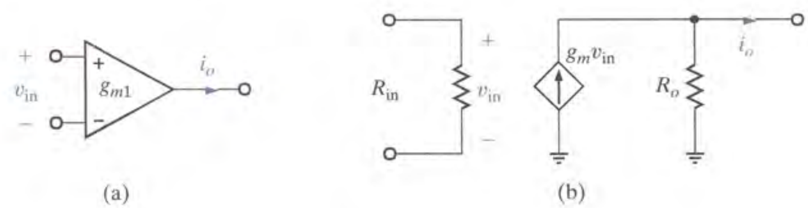


Figure 8.47
The OTA schematic symbol (a) and (b) simple model.

To compare the performance of the op-amp and OTA, consider the circuits in Fig. 8.48. For the op-amp, the overall voltage gain is

$$A = \frac{v_o}{v_{\text{in}}} = \left[\frac{R_{\text{in}}}{R_S + R_{\text{in}}}\right] A_V \left[\frac{R_L}{R_L + R_o}\right]$$

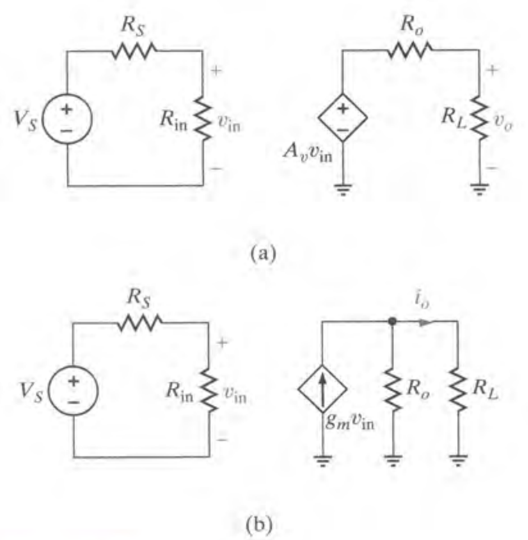


Figure 8.48
Simple circuits that demonstrate the relative strengths of the (a) op-amp and (b) OTA.

Ideally, $R_{\text{in}} \to \infty$, $R_o \to 0$, and the output voltage is independent of external components R_S and R_L. The overall gain of the OTA is

$$G_m = \frac{i_o}{v_{\text{in}}} = \left[\frac{R_{\text{in}}}{R_S + R_{\text{in}}}\right] g_m \left[\frac{R_o}{R_L + R_o}\right]$$

For an ideal OTA, both R_{in} and $R_o \to \infty$, yielding an output current that is independent of R_S and R_L. Similarities and differences between ideal OTAs and ideal op-amps are listed in Table 8.2.

Table 8.2 A comparison of ideal Op-amps and OTA features

Amplifier Type	Ideal R_{in}	Ideal R_o	Ideal Gain	Input Currents	Input Voltage
Op-amp	∞	0	∞	0	0
OTA	∞	∞	g_m	0	nonzero

As with op-amps, OTAs can be used to create mathematical circuits. We will focus on three OTA circuits used extensively in active filters: the integrator, the simulated resistor, and the summer. To simplify our analyses, we assume that the OTA is ideal with infinite input and output resistances. The integrator in Fig. 8.49, which forms the heart of our OTA active filters, can be analyzed as follows.

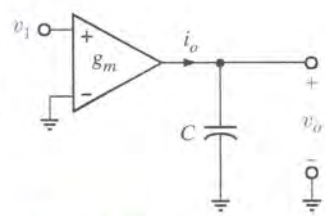


Figure 8.49
The OTA integrator.

$$i_o = g_m v_1 \qquad v_o = \frac{1}{C}\int i_o\,dt \qquad v_o = \frac{g_m}{C}\int v_1\,dt$$

Or, in the frequency domain,

$$\mathbf{I}_o = g_m \mathbf{V}_1 \qquad \mathbf{V}_o = \frac{\mathbf{I}_o}{j\omega C} \qquad \mathbf{V}_o = \frac{g_m}{j\omega C}\mathbf{V}_1$$

An interesting aspect of IC fabrication is that resistors (especially large valued ones) are physically very large compared to other devices such as transistors. Additionally, producing accurate values is quite difficult. This has motivated designers to use OTAs to simulate resistors. One such circuit is the grounded resistor, shown in Fig. 8.50. Applying the ideal OTA conditions in Table 8.2,

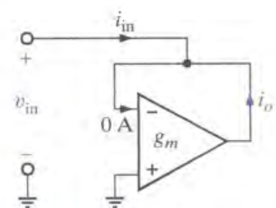
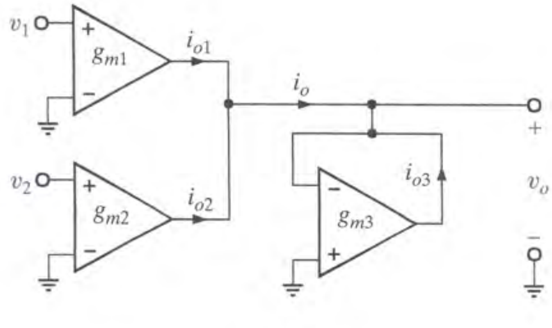


Figure 8.50
The OTA simulated resistor.

$$i_o = g_m(0 - v_{\text{in}}) = -g_m v_{\text{in}} \qquad i_{\text{in}} = -i_o \qquad R_{\text{eq}} = \frac{v_{\text{in}}}{i_{\text{in}}} = \frac{1}{g_m} \qquad \textbf{8.55}$$

A simple summer circuit is shown in Fig. 8.51a, where OTA 3 is a simulated resistor. Based on Eq. (8.55), we produce the equivalent circuit in Fig. 8.51b. The analysis is straightforward.

$$i_{o1} = g_{m1} v_1 \qquad i_{o2} = g_{m2} v_2 \qquad i_o = i_{o1} + i_{o2} \qquad v_o = \frac{i_o}{g_{m3}} = \frac{g_{m1}}{g_{m3}} v_1 + \frac{g_{m2}}{g_{m3}} v_2 \qquad \textbf{8.56}$$



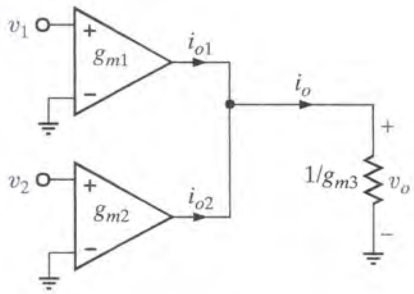


Figure 8.51
An OTA voltage summer.

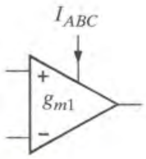


Figure 8.52
A modified OTA schematic symbol showing the input bias current.

At this point, we introduce our last important feature of the OTA—programmability. The transconductance, g_m, is linearly controlled by a current called the amplifier bias current, or I_{ABC}, as seen in Fig. 8.52. Unfortunately, the I_{ABC} input is not part of the schematic symbol. The sensitivity of g_m to I_{ABC} is typically 20 S/A but the range of g_m and its maximum value depend on the OTA design. Typical values are 10 mS for the maximum g_m and 3 to 7 powers of ten, or decades, for the transconductance range. For example, if the maximum g_m were 10 mS and the range were 4 decades, then the minimum g_m is 1 μS and the usable range of I_{ABC} is 0.05 μA to 0.5 mA.

LEARNING Example 8.20

An ideal OTA has a $g_m - I_{ABC}$ sensitivity of 20, a maximum g_m of 4 mS, and a g_m range of 4 decades. Using the circuit in Fig. 8.50, produce an equivalent resistance of 25 kΩ, giving both g_m and I_{ABC}.

SOLUTION From Eq. (8.55), the equivalent resistance is $R_{eq} = 1/g_m = 25\text{ k}\Omega$, yielding $g_m = 40\ \mu\text{S}$. Since $g_m = 20I_{ABC}$, the required amplifier bias current is $I_{ABC} = 2\ \mu\text{A}$.

LEARNING Example 8.21

The circuit in Fig. 8.53 is a floating simulated resistor. For an ideal OTA, find an expression for $R_{eq} = v_1/i_1$. Using the OTA described in Example 8.20, produce an 80-kΩ resistance. Repeat for a 10-MΩ resistor.

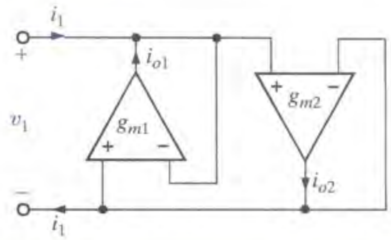


Figure 8.53 *The floating simulated resistor.*

SOLUTION For OTA 1, we have $i_{o1} = g_{m1}(-v_1)$ and $i_1 = -i_{o1}$. Thus, $R_{eq} = v_1/i_1 = 1/g_{m1}$. We must also consider the return current which is contributed by OTA 2, where $i_{o2} = g_{m2}(v_1)$ and $i_{o2} = i_1$. Now $R_{eq} = v_1/i_1 = 1/g_{m2}$. For proper operation, we must ensure that $g_{m1} = g_{m2}$.

For $R_{eq} = 1/g_m = 80$ kΩ, we have $g_{m1} = g_{m2} = g_m = 12.5$ μS. Since $g_m = 20I_{ABC}$, the required bias current for both OTAs is $I_{ABC} = 0.625$ μA. Changing to $R_{eq} = 1/g_m = 10$ MΩ, the transconductance becomes $g_m = 0.1$ μS. However, the minimum g_m for these OTAs is specified at 0.4 μS. We must find either suitable OTAs or a better circuit.

LEARNING Example 8.22

Using the summer in Fig. 8.51 and the OTAs specified in Example 8.20, produce the following function:

$$v_o = 10v_1 + 2v_2$$

Repeat for the function

$$v_o = 10v_1 - 2v_2$$

SOLUTION Comparing Eq. (8.56) to the desired function, we see that $g_{m1}/g_{m3} = 10$ and $g_{m2}/g_{m3} = 2$. With only two equations and three unknowns, we must choose one g_m value. Arbitrarily selecting $g_{m3} = 0.1$ mS yields $g_{m1} = 1$ mS and $g_{m2} = 0.2$ mS. The corresponding bias currents are $I_{ABC1} = 50$ μA, $I_{ABC2} = 10$ μA and $I_{ABC3} = 5$ μA.

For the second case, we simply invert the sign of v_2 as shown in Fig. 8.54. This is yet another advantage of OTA versus op-amps. Again choosing $g_{m3} = 0.1$ mS yields the same bias current as the first case.

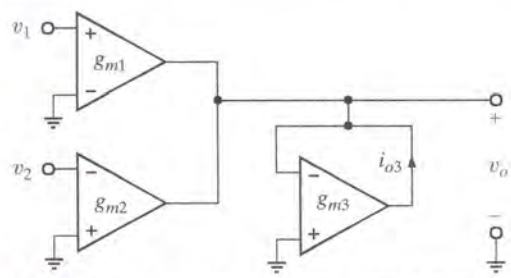


Figure 8.54 *A slight modification of the adder in Fig. 8.51 yields this subtracting circuit.*

Using the subcircuits in Fig. 8.49 and 8.50, one can create active filters called OTA-C filters, which contain only OTAs and capacitors. The lack of resistors makes OTA-C filters ideal for single-chip, or monolithic, implementations. As an introduction, consider the circuit in Fig. 8.55. For ideal OTAs, the transfer function can be determined as follows.

$$\mathbf{I}_{o1} = g_{m1}\mathbf{V}_{i1} \qquad \mathbf{I}_{o2} = -g_{m2}\mathbf{V} \qquad \mathbf{I}_C = \mathbf{V}_o(j\omega C) = \mathbf{I}_{o1} + \mathbf{I}_{o2}$$

$$\mathbf{V}_o = \frac{\mathbf{I}_{o1} + \mathbf{I}_{o2}}{j\omega C} = \frac{g_{m1}}{j\omega C}\mathbf{V}_{i1} - \frac{g_{m2}}{j\omega C}\mathbf{V}_o$$

Solving the transfer function yields the low-pass function

$$\frac{\mathbf{V}_o}{\mathbf{V}_{i1}} = \frac{g_{m1}/g_{m2}}{\dfrac{j\omega C}{g_{m2}} + 1} \qquad \textbf{8.57}$$

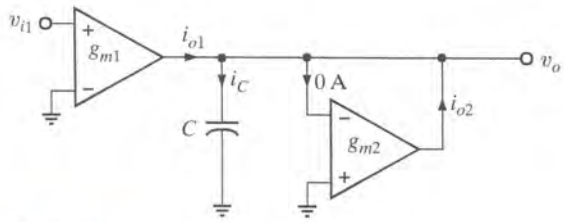


Figure 8.55
A simple first-order low-pass OTA-C filter.

From Eq. (8.57), the circuit is a first-order low-pass filter with the asymptotic Bode plot shown in Fig. 8.56. Both the corner frequency, $f_C = g_{m2}/(2\pi C)$, and dc gain, $A_{dc} = g_{m1}/g_{m2}$, are programmable.

In monolithic OTA-C filters, the capacitors and OTAs are fabricated on a single chip. Typical OTA capacitor values range from about 1 pF up to 50 pF.

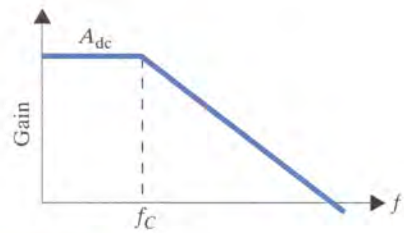


Figure 8.56
Asymptotic Bode plot for a first-order low-pass filter.

LEARNING Example 8.23

The low-pass filter in Fig. 8.55 is implemented using a 25-pF capacitor and OTAs with a $g_m - I_{ABC}$ sensitivity of 20, a maximum g_m of 1 mS and 3 decades of range. Find the required bias currents for the filter transfer function

$$\frac{\mathbf{V}_o}{\mathbf{V}_{in}} = \frac{4}{\dfrac{j\omega}{2\pi(10^5)} + 1} \qquad 8.58$$

SOLUTION Comparing Eq. (8.57) to the desired function, $g_{m2}/C = (2\pi)10^5$. For $C = 25$ pF, $g_{m2} = 15.7$ µS. Since $g_m = 20I_{ABC}$, the bias current for OTA 2 is $I_{ABC2} = 0.785$ µA. Finally $g_{m1}/g_{m2} = 4$ yields $I_{ABC1} = 3.14$ µA.

Of the dozens of OTA filter topologies, a very popular one is the two-integrator biquad filter. The term biquad is short for biquadratic, which, in filter terminology, means the filter gain is a ratio of two quadratic functions such as

$$\frac{\mathbf{V}_o}{\mathbf{V}_{in}} = \frac{A(j\omega)^2 + B(j\omega) + C}{(j\omega)^2 + \dfrac{\omega_0}{Q}(j\omega) + \omega_0^2} \qquad 8.59$$

By selecting appropriate values for A, B, and C, low-pass, band-pass, and high-pass functions can be created, as listed in Table 8.3. Figure 8.57 shows the most popular two-integrator biquad used in practice—the Tow–Thomas filter. Assuming ideal OTAs, we can derive the filter's transfer function. For OTA 1, an integrator,

$$\frac{\mathbf{V}_{o1}}{\mathbf{V}_{i1} - \mathbf{V}_{o2}} = \frac{g_{m1}}{j\omega C_1}$$

The output current of OTA 2 is

$$\mathbf{I}_{o2} = g_{m2}[\mathbf{V}_{o1} - \mathbf{V}_{i2}]$$

Applying KCL at the second output node, we find

$$\mathbf{I}_{o3} + \mathbf{I}_{o2} = (j\omega C_2)\mathbf{V}_{o2}$$

where

$$\mathbf{I}_{o3} = [\mathbf{V}_{i3} - \mathbf{V}_{o2}]g_{m3}$$

Solving for $\mathbf{V}_{o1}$ and $\mathbf{V}_{o2}$ yields

$$\mathbf{V}_{o1} = \frac{\left[\dfrac{j\omega C_2}{g_{m2}} + \dfrac{g_{m3}}{g_{m2}}\right]\mathbf{V}_{i1} + \mathbf{V}_{i2} - \left[\dfrac{g_{m3}}{g_{m2}}\right]\mathbf{V}_{i3}}{\left[\dfrac{C_1 C_2}{g_{m1} g_{m2}}\right](j\omega)^2 + \left[\dfrac{g_{m3} C_1}{g_{m2} g_{m1}}\right](j\omega) + 1}$$

Table 8.3 Various Tow–Thomas biquad filter possibilities

Filter Type	A	B	C
Low-pass	0	0	nonzero
Band-pass	0	nonzero	0
High-pass	nonzero	0	0

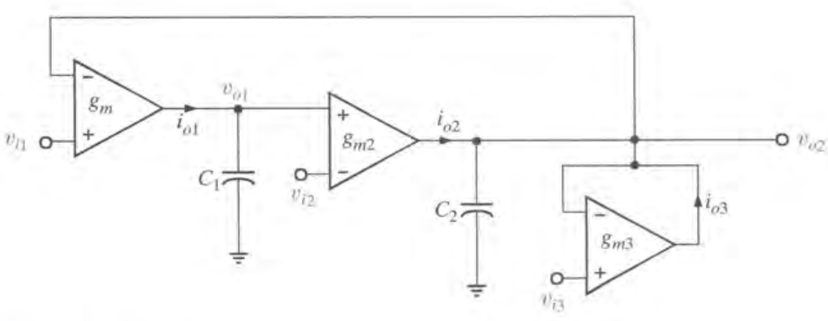


Figure 8.57
The Tow–Thomas OTA-C biquad filter.

and

$$\mathbf{V}_{o2} = \frac{\mathbf{V}_{i1} - \left[\frac{j\omega C_1}{g_{m1}}\right]\mathbf{V}_{i2} + \left[\frac{j\omega C_1 g_{m3}}{g_{m1}g_{m2}}\right]\mathbf{V}_{i3}}{\left[\frac{C_1C_2}{g_{m1}g_{m2}}\right](j\omega)^2 + \left[\frac{g_{m3}C_1}{g_{m2}g_{m1}}\right](j\omega) + 1} \qquad \mathbf{8.60}$$

Note that this single circuit can implement both low-pass and band-pass filters depending on where the input is applied! Table 8.4 lists the possibilities. Comparing Eqs. (8.59) and (8.60), design equations for ω_0, Q, and the bandwidth can be written as

$$\omega_0 = \sqrt{\frac{g_{m1}g_{m2}}{C_1C_2}} \qquad \frac{\omega_0}{Q} = \text{BW} = \frac{g_{m3}}{C_2} \qquad Q = \sqrt{\frac{g_{m1}g_{m2}}{g_{m3}^2}}\sqrt{\frac{C_2}{C_1}} \qquad \mathbf{8.61}$$

Consider a Tow–Thomas band-pass filter. From Eq. (8.61), if $g_{m1} = g_{m2} = g_m$ and $C_1 = C_2 = C$, the following relationships are easily derived.

$$\omega_0 = \frac{g_m}{C} = \frac{k}{C}I_{ABC} \qquad \frac{\omega_0}{Q} = \text{BW} = \frac{g_{m3}}{C} = \frac{k}{C}I_{ABC3} \qquad Q = \frac{g_m}{g_{m3}} = \frac{I_{ABC}}{I_{ABC3}} \qquad \mathbf{8.62}$$

where k is the $g_m - I_{ABC}$ sensitivity. Based on Eq. (8.62) we have efficient control over the filter characteristics. In particular, tuning I_{ABC} with I_{ABC3} fixed scales both the center frequency and Q directly without affecting bandwidth. Tuning I_{ABC3} only changes the bandwidth but not the center frequency. Finally, tuning all three bias currents scales both the center frequency and bandwidth proportionally, producing a constant Q factor.

Table 8.4 Low-pass and band-pass combinations for the Tow–Thomas biquad filter in Fig. 8.57

Filter Type	Input	Output	Sign
Low-pass	v_{i2}	v_{o1}	positive
	v_{i3}	v_{o1}	negative
	v_{i1}	v_{o2}	positive
Band-pass	v_{i2}	v_{o2}	negative
	v_{i3}	v_{o2}	positive

LEARNING Example 8.24

Using the OTAs specified in Example 8.20 and 50-pF capacitors, design a Tow–Thomas band-pass filter with a center frequency of 500 kHz, a bandwidth of 75 kHz, and a center frequency gain, A_C, of –5.

SOLUTION From Eq. (8.61), the filter characteristics for the v_{i2} –v_{o2} input-output pair are

$$\text{BW} = \frac{g_{m3}}{C} \qquad \omega_0^2 = \frac{g_{m1}g_{m2}}{C^2} \qquad A_C = -\frac{g_{m2}}{g_{m3}}$$

For the specifications given,

$$g_{m3} = (\text{BW})(C) = (2\pi)(7.5 \times 10^4)(50 \times 10^{-12}) = 23.56\ \mu\text{S}$$

$$g_{m2} = -A_C g_{m3} = 117.8\ \mu\text{S}$$

$$g_{m1} = \frac{\omega_0^2 C^2}{g_{m2}} = \frac{(2\pi)^2(5 \times 10^5)^2(50 \times 10^{-12})^2}{117.8 \times 10^{-6}} = 209.5\ \mu\text{S}$$

The required bias currents are $I_{ABC1} = 10.47\ \mu$A, $I_{ABC} = 5.89\ \mu$A, and $I_{ABC3} = 1.18\ \mu$A. From the filter's Bode plot in Fig. 8.58, we see that the gain, bandwidth, and center frequency specifications are met.

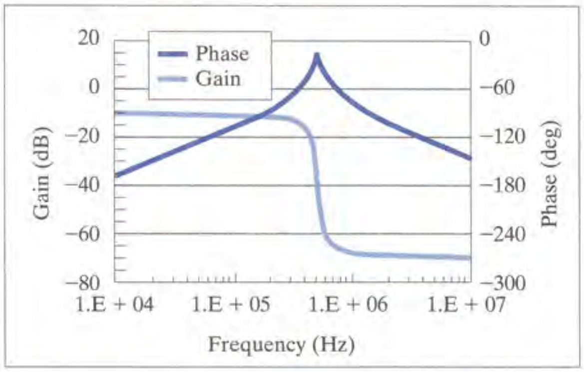


Figure 8.58
Bode plot for the Tow–Thomas band-pass filter of Example 8.24.

Learning by Application

LEARNING Example 8.25

The ac-dc converter in Fig. 8.59a is designed for use with a hand-held calculator. We will ignore the output load for now; it is addressed in Problem 8.70. Ideally, the circuit should convert a 120-V rms sinusoidal voltage to a 9-V dc output. In actuality the output is

$$v_o(t) = 9 + 0.5 \sin 377t$$

Let us use a low-pass filter to reduce the 60-Hz component of $v_o(t)$.

SOLUTION The Thévenin equivalent circuit for the converter is shown in Fig. 8.59b. By placing a capacitor across the output terminals, as shown in Fig. 8.59c, we create a low-pass filter at the output. The transfer function of the filtered converter is

$$\frac{\mathbf{V}_{\text{OF}}}{\mathbf{V}_{\text{Th}}} = \frac{1}{1 + sR_{\text{Th}}C}$$

which has a pole at a frequency of $f = 1/2\pi R_{\text{Th}}C$. To obtain significant attenuation at 60 Hz, we choose to place the pole at 6 Hz, yielding the equation

$$\frac{1}{2\pi R_{\text{Th}}C} = 6$$

or

$$C = 53.05\ \mu\text{F}$$

A transient simulation of the converter is used to verify performance.

Figure 8.59d shows the output without filtering $v_o(t)$, and with filtering, $v_{\text{OF}}(t)$. The filter has successfully reduced the unwanted 60-Hz component by a factor of roughly six.

(continued)

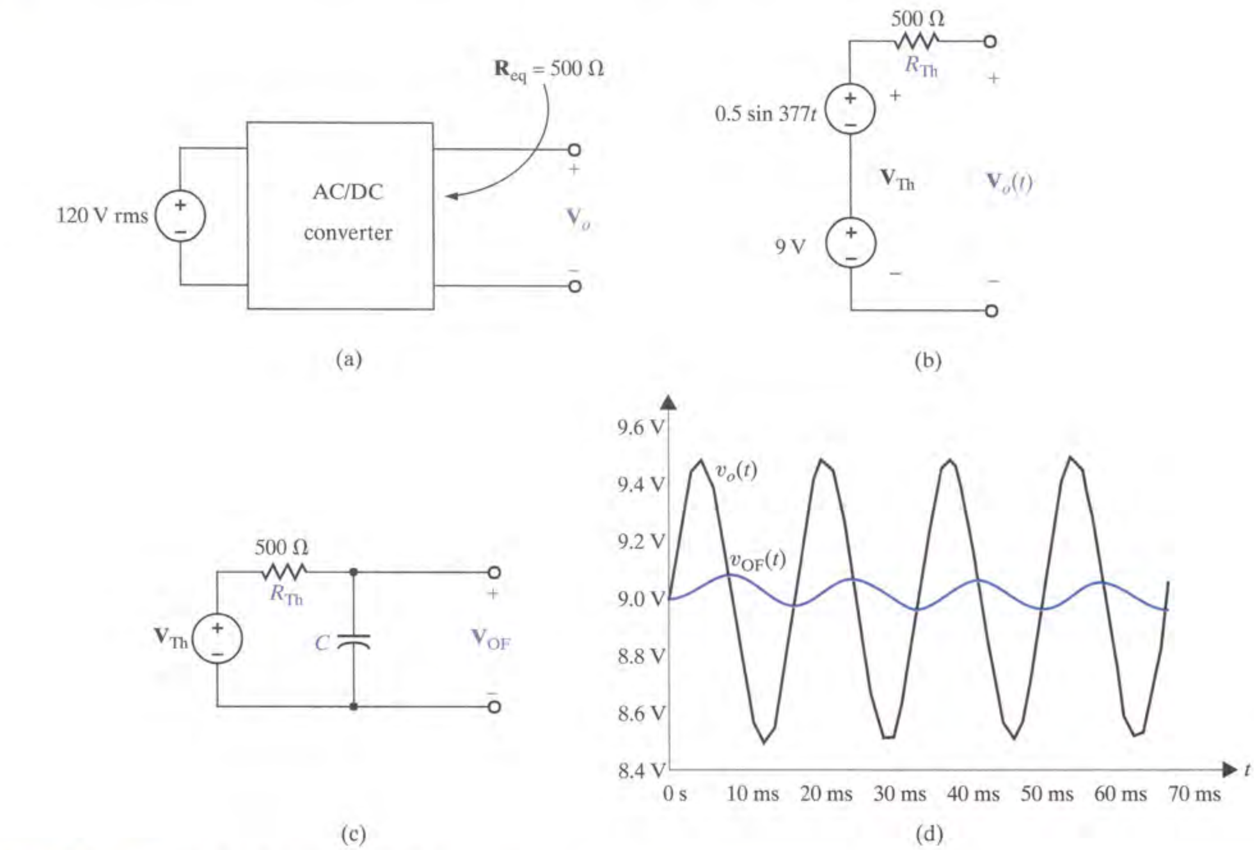


Figure 8.59 *Circuits and output plots for ac/dc converter.*

LEARNING Example 8.26

The antenna of an FM radio picks up stations across the entire FM frequency range—approximately 87.5 MHz to 108 MHz. The radio's circuitry must have the capability to first reject all of the stations except the one that the listener wants to hear and then to boost the minute antenna signal. A tuned amplifier incorporating parallel resonance can perform both tasks simultaneously.

The network in Fig. 8.60a is a circuit model for a single-stage tuned transistor amplifier where the resistor, capacitor, and inductor are discrete elements. Let us find the transfer function $\mathbf{V}_o(s)/\mathbf{V}_A(s)$, where $\mathbf{V}_A(s)$ is the antenna voltage and the value of C for maximum gain at 91.1 MHz. Finally, we will simulate the results.

SOLUTION Since $\mathbf{V}(s) = \mathbf{V}_A(s)$, the transfer function is

$$\frac{\mathbf{V}_o(s)}{\mathbf{V}_A(s)} = -\frac{4}{1000}\left[R//sL//\frac{1}{sC}\right]$$

$$\frac{\mathbf{V}_o(s)}{\mathbf{V}_A(s)} = -\frac{4}{1000}\left[\frac{s/C}{s^2 + \frac{s}{RC} + \frac{1}{LC}}\right]$$

The parallel resonance network is actually a band-pass filter. Maximum gain occurs at the center frequency, f_0. This condition corresponds to a minimum value in the denominator. Isolating the denominator polynomial, $D(s)$, and letting $s = j\omega$, we have

$$D(j\omega) = \frac{R}{LC} - \omega^2 + \frac{j\omega}{C}$$

which has a minimum value when the real part goes to zero, or

$$\frac{1}{LC} - \omega_0^2 = 0$$

yielding a center frequency of

$$\omega_0 = \frac{1}{\sqrt{LC}}$$

Thus, for a center frequency of 91.1 MHz, we have

$$2\pi(91.1 \times 10^6) = \frac{1}{\sqrt{LC}}$$

and the required capacitor value is

$$C = 3.05 \text{ pF}$$

The Bode plot for the tuned amplifier, as shown in Fig. 8.60b, confirms the design, since the center frequency is 91.1 MHz, as specified.

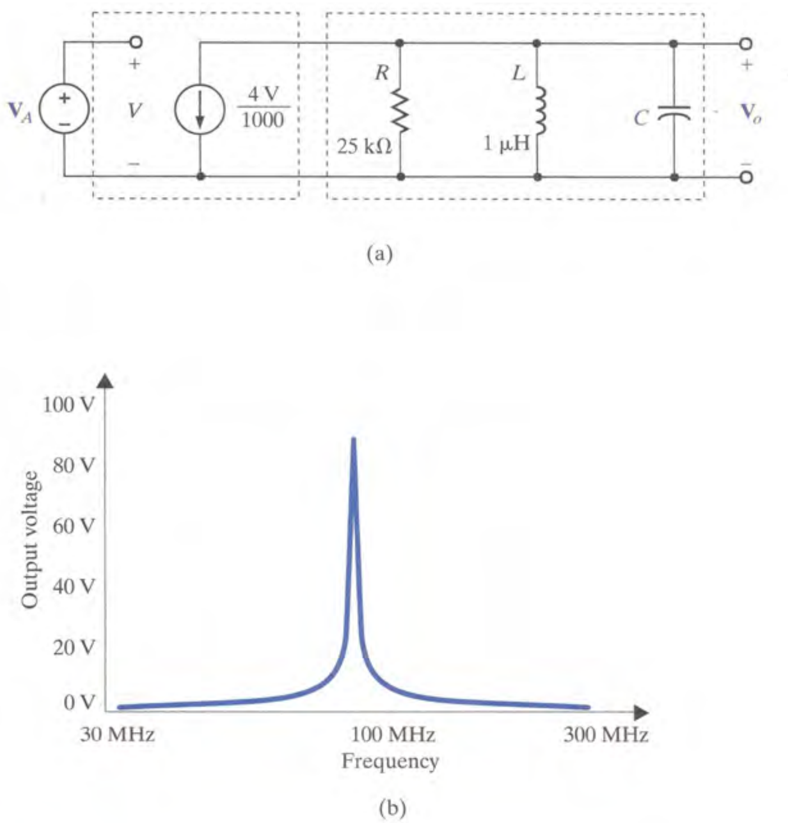


Figure 8.60
Circuit and Bode plot for parallel resonance tuned amplifier.

Learning by Design

Throughout this chapter we have presented a number of design examples. In this section we consider some additional ones that also have practical ramifications.

LEARNING Example 8.27

Compact disks (CDs) have become a very popular medium for recording and playing music. CDs store information in a digital manner; that is, the music is sampled at a very high rate, and the samples are recorded on the disc. The trick is to sample so quickly that the reproduction sounds continuous. The industry standard sampling rate is 44.1 kHz—one sample every 22.7 μs.

One interesting aspect regarding the analog-to-digital conversion that takes place inside the unit recording a CD is called the Nyquist criterion. This criterion states that in the analog conversion, any signal components at frequencies above half the sampling rate (22.05 kHz in this case) cannot be faithfully reproduced. Therefore, recording technicians filter these frequencies out before any sampling occurs, yielding higher fidelity to the listener.

Let us design a series of low-pass filters to perform this task.

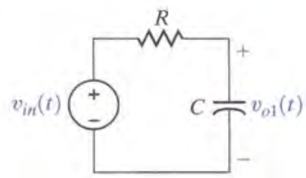


Figure 8.61
Single-pole low-pass filter.

SOLUTION Suppose, for example, that our specification for the filter is unity gain at dc and 20 dB of attenuation at 22.05 kHz. Let us consider first the simple RC filter in Fig. 8.61.

The transfer function is easily found to be

$$\mathbf{G}_{v1}(s) = \frac{\mathbf{V}_{o1}}{\mathbf{V}_{in}} = \frac{1}{1 + sRC}$$

Since a single-pole transfer function attenuates at 20 dB/decade, we should place the pole frequency one decade before the −20 dB point of 22.05 kHz.

Thus,

$$f_p = \frac{1}{2\pi RC} = 2.205 \text{ kHz}$$

If we arbitrarily choose $C = 1$ nF, the resulting value for R is 72.18 kΩ, which is reasonable. A Bode plot of the magnitude of $\mathbf{G}_{v1}(s)$ is shown in Fig. 8.62. All specifications are met but at

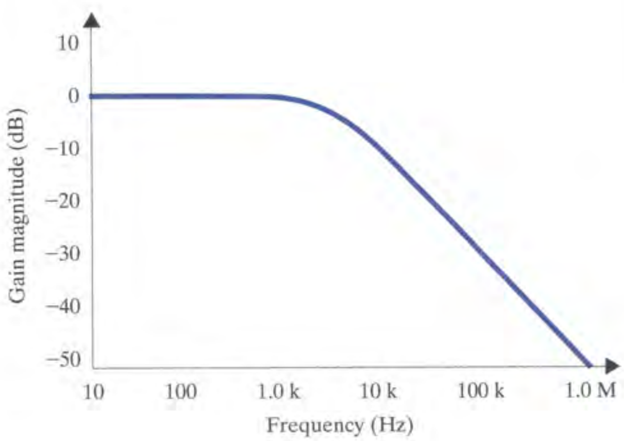


Figure 8.62
Bode plot for single-pole filter.

the cost of severe attenuation in the audible frequency range. This is undesirable.

An improved filter is shown in Fig. 8.63. It is a two-stage low-pass filter with identical filter stages separated by a unity-gain buffer.

The presence of the op-amp permits us to consider the stages independently. Thus, the transfer function becomes

$$\mathbf{G}_{v2}(s) = \frac{\mathbf{V}_{o2}}{\mathbf{V}_{\text{in}}} = \frac{1}{[1 + sRC]^2}$$

To find the required pole frequencies, let us employ the equation for $\mathbf{G}_{v2}(s)$ at 22.05 kHz, since we know that the gain must be 0.1 (attenuated 20 dB) at that frequency. Using the substitution $s = j\omega$, we can express the magnitude of $\mathbf{G}_{v2}(s)$ as

$$|\mathbf{G}_{v2}| = \left\{\frac{1}{1 + (22{,}050/f_p)^2}\right\} = 0.1$$

and the pole frequency is found to be 7.35 kHz. The corresponding resistor value is 21.65 kΩ. Bode plots for $\mathbf{G}_{v1}$ and $\mathbf{G}_{v2}$ are shown in Fig. 8.64. Note that the two-stage filter has a wider bandwidth, which improves the fidelity of the recording.

Let us try one more improvement—expanding the two-stage filter to a four-stage filter. Again, the gain magnitude is 0.1 at 22.05 kHz and can be written

$$|\mathbf{G}_{v3}| = \left\{\frac{1}{[1 + (22{,}050/f_p)^2]^2}\right\} = 0.1$$

The resulting pole frequencies are at 15 kHz, and the required resistor value is 10.61 kΩ. Figure 8.65 shows all three Bode plots. Obviously, the four-stage filter, having the widest bandwidth, is the best option (discounting any extra cost associated with the additional active and passive circuit elements).

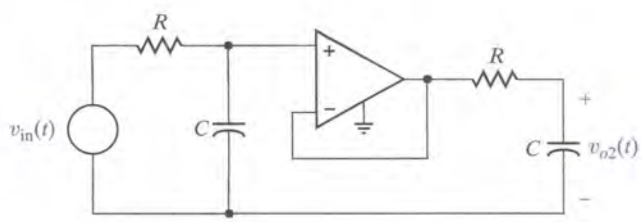


Figure 8.63
Two-stage buffered filter.

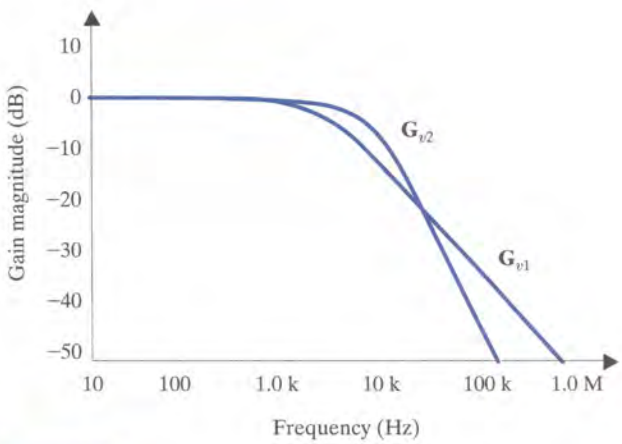


Figure 8.64
Bode plot for single- and two-stage filters.

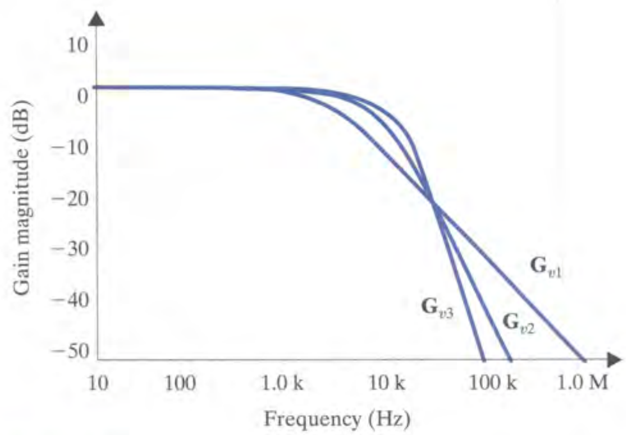


Figure 8.65
Bode plots for single-, two-, and four-stage filters.

LEARNING Example 8.28

The circuit in Fig. 8.66a is called a notch filter. From a sketch of its Bode plot in Fig. 8.66b, we see that at the notch frequency, f_n, the transfer function gain is zero, while at frequencies above and below f_n the gain is unity. Let us design a notch filter to remove an annoying 60-Hz hum from the output voltage of a cassette tape player and generate its Bode plot.

SOLUTION Figure 8.66c shows a block diagram for the filter implementation. The tape output contains both the desired music and the undesired hum. After filtering, the voltage $\mathbf{V}_{\text{amp}}$ will have no 60-Hz component as well as some attenuation at frequencies around 60 Hz. An equivalent circuit for the block diagram including a Thévenin equivalent for the tape deck and an equivalent resistance for the power amp is shown in Fig. 8.66d. Applying voltage division, the transfer function is found to be

$$\frac{\mathbf{V}_{\text{amp}}}{\mathbf{V}_{\text{tape}}} = \frac{R_{\text{amp}}}{R_{\text{amp}} + R_{\text{tape}} + \left(sL//\dfrac{1}{Cs}\right)}$$

After some manipulation, the transfer function can be written as

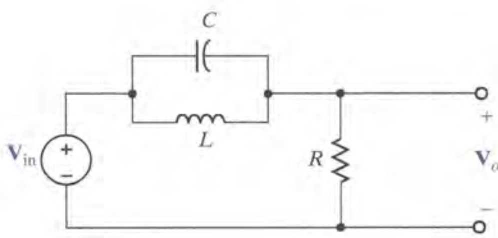
$$\frac{\mathbf{V}_{\text{amp}}}{\mathbf{V}_{\text{tape}}} = \frac{R_{\text{amp}}}{R_{\text{amp}} + R_{\text{tape}}}\left[\frac{s^2LC + 1}{s^2LC + s\left(\dfrac{L}{R_{\text{tape}} + R_{\text{amp}}}\right) + 1}\right]$$

We see that the transfer function contains two zeros and two poles. Letting $s = j\omega$, the zero frequencies, ω_z, are found to be at

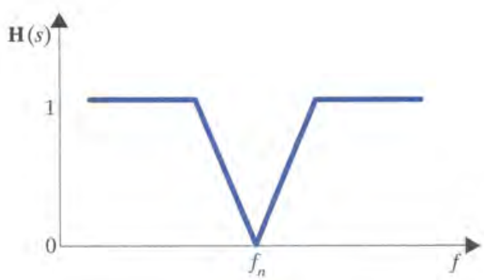
$$\omega_z = \pm\frac{1}{\sqrt{LC}}$$

Obviously, we would like the zero frequencies to be at 60 Hz. If we arbitrarily choose $C = 10\ \mu\text{F}$, then $L = 0.704$ mH.

The Bode plot, shown in Fig. 8.66e, confirms that there is indeed zero transmission at 60 Hz.



(a)



(b)

Figure 8.66
Circuits and Bode plots for 60-Hz notch filter.

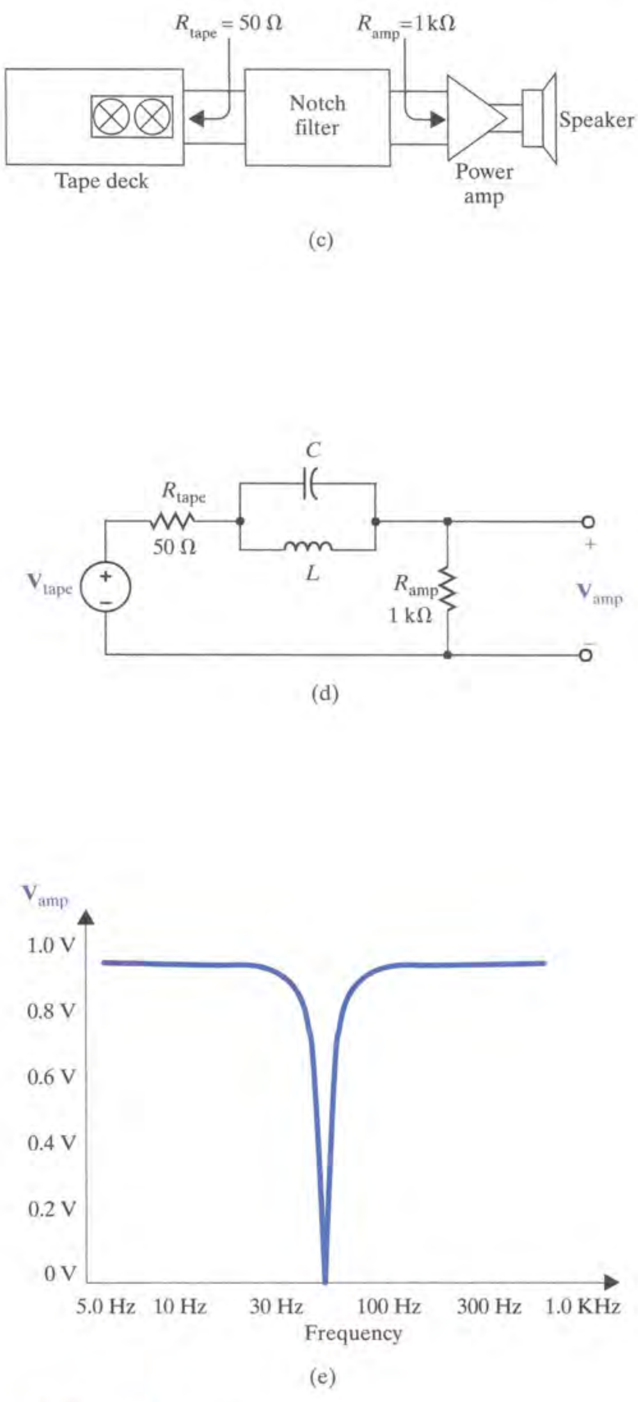


Figure 8.66 *Continued*

LEARNING Check

Summary

- There are four types of network or transfer functions:

  1. $\mathbf{Z}(j\omega)$: the ratio of the output voltage to the input current
  2. $\mathbf{Y}(j\omega)$: the ratio of the output current to the input voltage
  3. $\mathbf{G}_v(j\omega)$: the ratio of the output voltage to the input voltage
  4. $\mathbf{G}_{\mathrm{I}}(j\omega)$: the ratio of the output current to the input current

- Driving-point functions are impedances or admittances defined at a single pair of terminals, such as the input impedance of a network.

- When the network function is expressed in the form

$$\mathbf{H}(s) = \frac{N(s)}{D(s)}$$

  the roots of $N(s)$ cause $\mathbf{H}(s)$ to become zero and are called zeros of the function, and the roots of $D(s)$ cause $\mathbf{H}(s)$ to become infinite and are called poles of the function.

- Bode plots are semilog plots of the magnitude and phase of a transfer function as a function of frequency. Straight-line approximations can be used to sketch quickly the magnitude characteristic. The error between the actual characteristic and the straight-line approximation can be calculated when necessary.

- The resonant frequency, given by the expression

$$\omega_0 = \frac{1}{\sqrt{LC}}$$

  is the frequency at which the impedance of a series RLC circuit or the admittance of a parallel RLC circuit is purely real.

- The quality factor is a measure of the sharpness of the resonant peak. The higher the Q, the sharper the peak.
  For series RLC circuits, $Q = (1/R)\sqrt{L/C}$. For parallel RLC circuits, $Q = R\sqrt{C/L}$.

- The half-power, cutoff, or break frequencies are the frequencies at which the magnitude characteristic of the Bode plot is $1/\sqrt{2}$ of its maximum value.

- The parameter values for passive circuit elements can be both magnitude and frequency scaled.

- The four common types of filters are low pass, high pass, band pass, and band rejection.

- The bandwidth of a band-pass or band-rejection filter is the difference in frequency between the half-power points; that is,

$$\mathrm{BW} = \omega_{\mathrm{HI}} - \omega_{\mathrm{LO}}$$

  For a series RLC circuit, $\mathrm{BW} = R/L$. For a parallel RLC circuit, $\mathrm{BW} = 1/RC$.

Problems

For solutions and additional help on problems marked with ► go to www.wiley.com/college/irwin

SECTION 8.1

8.1 ► **Determine the driving point impedance at the input terminals of the network shown in Fig. P8.1 as a function of s.**

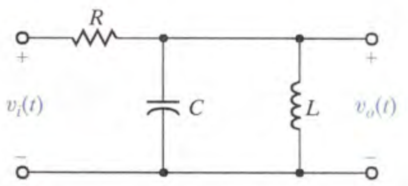


Figure P8.1

8.2 Determine the voltage transfer function $\mathbf{V}_o(s)/\mathbf{V}_i(s)$ as a function of s for the network shown in Fig. P8.1.

8.3 Determine the voltage transfer function $\mathbf{V}_o(s)/\mathbf{V}_i(s)$ as a function of s for the network shown in Fig. P8.3.

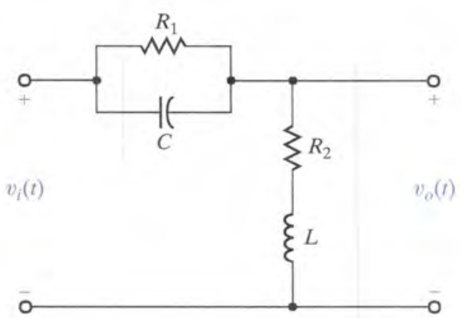


Figure P8.3

8.4 Find the transfer impedance $\mathbf{V}_o(s)/\mathbf{I}_S(s)$ for the network shown in Fig. P8.4.

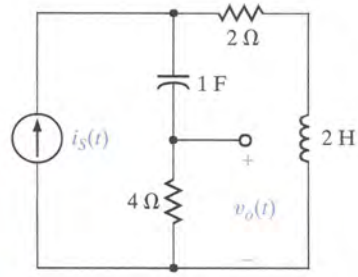


Figure P8.4

8.5 ► **Find the driving point impedance at the input terminals of the circuit in Fig. P8.5 as a function of s.**

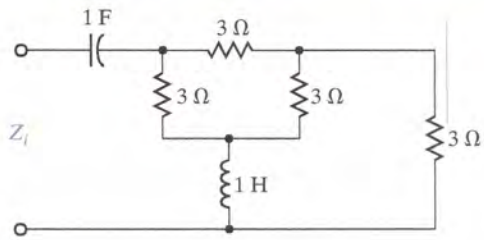


Figure P8.5

SECTION 8.2

8.6 Draw the Bode plot for the network function

$$\mathbf{H}(j\omega) = \frac{j\omega 5 + 1}{j\omega 20 + 1}$$

8.7 Draw the Bode plot for the network function

$$\mathbf{H}(j\omega) = \frac{j\omega 2 + 1}{j\omega 10 + 1}$$

8.8 Draw the Bode plot for the network function

$$\mathbf{H}(j\omega) = \frac{10(10j\omega + 1)}{(100j\omega + 1)(j\omega + 1)}$$

8.9 ► **Draw the Bode plot for the network function**

$$\mathbf{H}(j\omega) = \frac{j\omega}{(j\omega + 1)(0.1j\omega + 1)}$$

8.10 Draw the Bode plot for the network function

$$\mathbf{H}(j\omega) = \frac{10j\omega + 1}{j\omega(0.1j\omega + 1)}$$

8.11 Sketch the magnitude characteristic of the Bode plot for the transfer function

$$\mathbf{H}(j\omega) = \frac{10}{j\omega(0.1j\omega + 1)}$$

8.12 Sketch the magnitude characteristic of the Bode plot for the transfer function

$$\mathbf{H}(j\omega) = \frac{20(j\omega + 1)}{j\omega(0.1j\omega + 1)(0.01j\omega + 1)}$$

8.13 ► **Sketch the magnitude characteristic of the Bode plot for the transfer function**

$$\mathbf{H}(j\omega) = \frac{100(j\omega)}{(j\omega + 1)(j\omega + 10)(j\omega + 50)}$$

8.14 Draw the Bode plot for the network function

$$\mathbf{H}(j\omega) = \frac{16}{(j\omega)^2(j\omega 2 + 1)}$$

8.15 Sketch the magnitude characteristic of the Bode plot for the transfer function

$$\mathbf{H}(j\omega) = \frac{640(j\omega + 1)(0.01j\omega + 1)}{(j\omega)^2(j\omega + 10)}$$

8.16 Sketch the magnitude characteristic of the Bode plot for the transfer function

$$\mathbf{H}(j\omega) = \frac{10^5(5j\omega + 1)^2}{(j\omega)^2(j\omega + 10)(j\omega + 100)^2}$$

8.17 ► **Sketch the magnitude characteristic of the Bode plot for the transfer function**

$$\mathbf{G}(j\omega) = \frac{10j\omega}{(j\omega + 1)(j\omega + 10)^2}$$

8.18 Sketch the magnitude characteristic of the Bode plot for the transfer function

$$\mathbf{H}(j\omega) = \frac{10^2(j\omega)^2}{(j\omega + 1)(j\omega + 10)^2(j\omega + 50)}$$

8.19 Sketch the magnitude characteristic of the Bode plot for the transfer function

$$\mathbf{G}(j\omega) = \frac{-\omega^2 10^4}{(j\omega + 1)^2(j\omega + 10)(j\omega + 100)^2}$$

8.20 Sketch the magnitude characteristic of the Bode plot for the transfer function

$$\mathbf{G}(j\omega) = \frac{64(j\omega + 1)^2}{-j\omega^3(0.1j\omega + 1)}$$

8.21 ► **Sketch the magnitude characteristic of the Bode plot for the transfer function**

$$\mathbf{G}(j\omega) = \frac{-\omega^2}{(j\omega + 1)^3}$$

8.22 Draw the Bode plot for the network function

$$\mathbf{H}(j\omega) = \frac{72(j\omega + 2)}{j\omega\left[(j\omega)^2 + 2.4j\omega + 144\right]}$$

8.23 Sketch the magnitude characteristic of the Bode plot for the transfer function

$$\mathbf{G}(j\omega) = \frac{10^4(j\omega + 1)(-\omega^2 + 6j\omega + 225)}{j\omega(j\omega + 50)^2(j\omega + 450)}$$

8.24 Sketch the magnitude characteristic of the Bode plot for the transfer function

$$\mathbf{H}(j\omega) = \frac{+81(j\omega + 0.1)}{(j\omega)(-\omega^2 + 3.6j\omega + 81)}$$

8.25 ► **Sketch the magnitude characteristic of the Bode plot for the transfer function**

$$\mathbf{H}(j\omega) = \frac{+6.4(j\omega)}{(j\omega + 1)(-\omega^2 + 8j\omega + 64)}$$

8.26 Determine $\mathbf{H}(j\omega)$ if the amplitude characteristic for $\mathbf{H}(j\omega)$ is shown in Fig. P8.26.

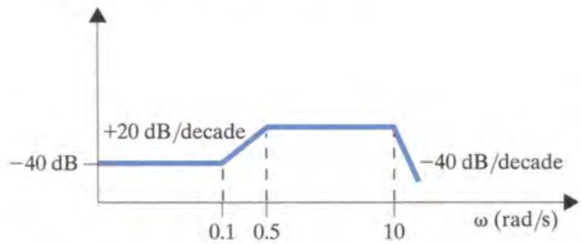


Figure P8.26

8.27 Find $\mathbf{H}(j\omega)$ if its magnitude characteristic is shown in Fig. P8.27.

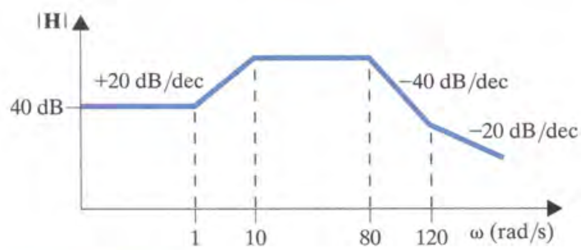


Figure P8.27

8.28 Find $\mathbf{H}(j\omega)$ if its magnitude characteristic is shown in Fig. P8.28.

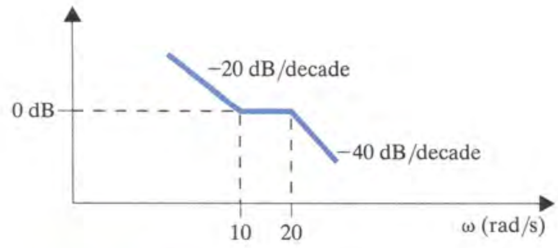


Figure P8.28

8.29 ► **Find H(jω) if its amplitude characteristic is shown in Fig. P8.29.**

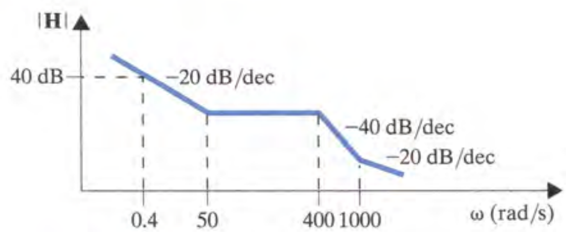


Figure P8.29

8.30 Find $\mathbf{H}(j\omega)$ if its magnitude characteristic is shown in Fig. P8.30.

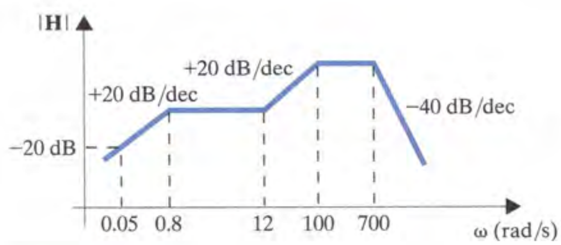


Figure P8.30

8.31 Find $\mathbf{H}(j\omega)$ if its amplitude characteristic is shown in Fig. P8.31.

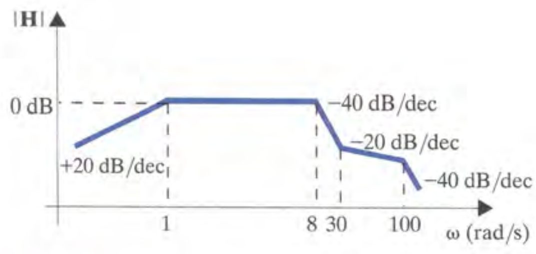


Figure P8.31

8.32 Given the magnitude characteristic in Fig. P8.32, find $\mathbf{G}(j\omega)$.

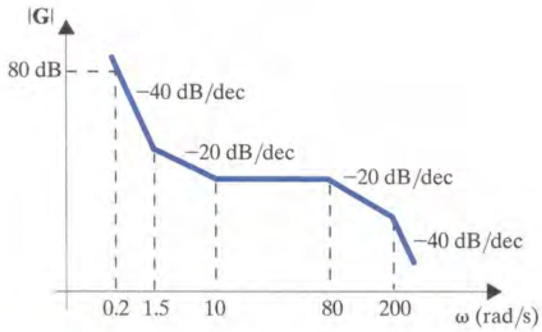


Figure P8.32

8.33 ► **Find $\mathbf{G}(j\omega)$ if the amplitude characteristic for this function is shown in Fig. P8.33.**

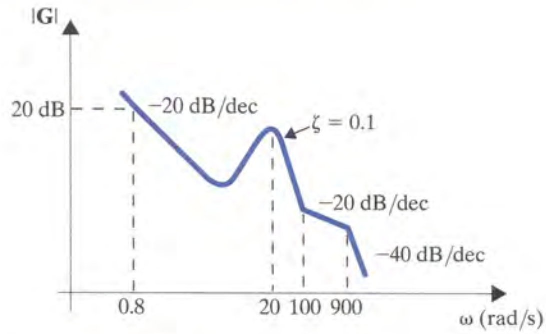


Figure P8.33

SECTION 8.3

8.34 A series *RLC* circuit resonates at 2000 rad/s. If $C = 20\ \mu\text{F}$ and it is known that the impedance at resonance is 2.4 Ω, compute the value of L, the Q of the circuit, and the bandwidth.

8.35 A series resonant circuit has a Q of 120 and a resonant frequency of 60,000 rad/s. Determine the half-power frequencies and the bandwidth of the circuit.

8.36 Given the series *RLC* circuit in Fig. P8.36, if $R = 10\ \Omega$, find the values of L and C such that the network will have a resonant frequency of 100 kHz and a bandwidth of 1 kHz.

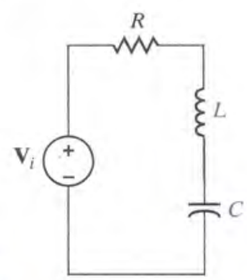


Figure P8.36

8.37 ► **Given the network in Fig. P8.37, find ω_0, Q, ω_{max}, and $|\mathbf{V}_o|_{max}$.**

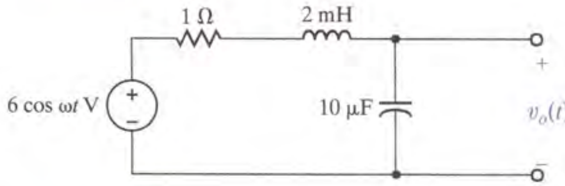


Figure P8.37

8.38 Repeat Problem 8.37 if the value of R is changed to 0.1 Ω.

8.39 A series RLC circuit is driven by a signal generator. The resonant frequency of the network is known to be 1600 rad/s, and at that frequency the impedance seen by the signal generator is 5 Ω. If $C = 20$ μF, find L, Q, and the bandwidth.

8.40 A variable-frequency voltage source drives the network in Fig. P8.40. Determine the resonant frequency, Q, BW, and the average power dissipated by the network at resonance.

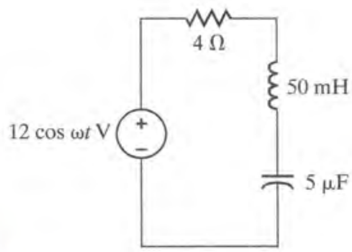


Figure P8.40

8.41 ► **In the network in Fig. P8.41, the inductor value is 10 mH, and the circuit is driven by a variable-frequency source. If the magnitude of the current at resonance is 12 A, $\omega_0 = 1000$ rad/sec, and $L = 10$ mH, find C, Q, and the bandwidth of the circuit.**

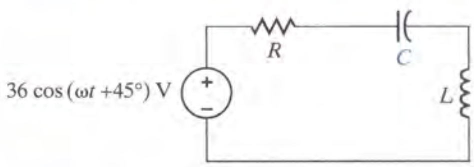


Figure P8.41

8.42 Given the network in Fig. P8.42, find $|\mathbf{V}_o|_{max}$.

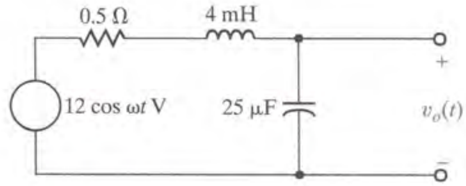


Figure P8.42

8.43 A parallel RLC resonant circuit with a resonant frequency of 20,000 rad/s has an admittance at resonance of 1 mS. If the capacitance of the network is 5 μF, find the values of R and L.

8.44 A parallel RLC resonant circuit has a resistance of 200 Ω. If it is known that the bandwidth is 80 rad/s and the lower half-power frequency is 800 rad/s, find the values of the parameters L and C.

8.45 ► **A parallel RLC circuit, which is driven by a variable-frequency 2-A current source, has the following values: $R = 1$ kΩ, $L = 100$ mH, and $C = 10$ μF. Find the bandwidth of the network, the half-power frequencies, and the voltage across the network at the half-power frequencies.**

8.46 A parallel RLC circuit, which is driven by a variable-frequency 10-A source, has the following parameters: $R = 500$ Ω, $L = 0.5$ mH, and $C = 20$ μF. Find the resonant frequency, the Q, the average power dissipated at the resonant frequency, the BW, and the average power dissipated at the half-power frequencies.

8.47 Consider the network in Fig. P8.47. If $R = 2$ kΩ, $L = 20$ mH, $C = 50$ μF, and $R_S = \infty$, determine the resonant frequency ω_0, the Q of the network, and the bandwidth of the network. What impact does an R_S of 10 kΩ have on the quantities determined?

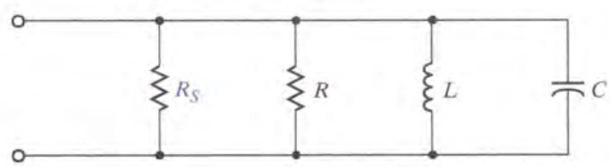


Figure P8.47

8.48 The source in the network in Fig. P8.48 is $i_S(t) = \cos 1000t + \cos 1500t$ A. $R = 200\ \Omega$ and $C = 500\ \mu F$. If $\omega_0 = 1000$ rad/sec, find L, Q, and the BW. Compute the output voltage $v_o(t)$ and discuss the magnitude of the output voltage at the two input frequencies.

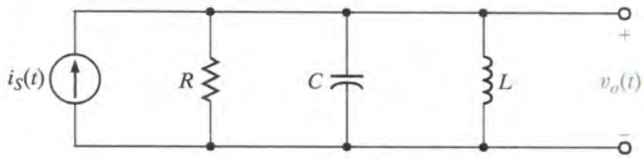


Figure P8.48

8.49 ► Determine the parameters of a parallel resonant circuit that has the following properties: $\omega_0 = 2$ Mrad/s, BW = 20 krad/s, and an impedance at resonance of 2000 Ω.

8.50 Determine the value of C in the network shown in Fig. P8.50 for the circuit to be in resonance.

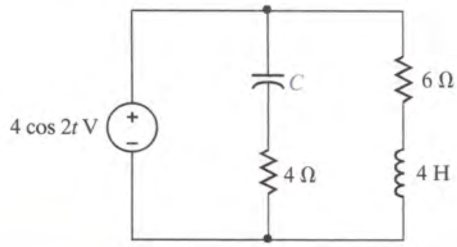


Figure P8.50

8.51 Determine the equation for the nonzero resonant frequency of the impedance shown in Fig. P8.51.

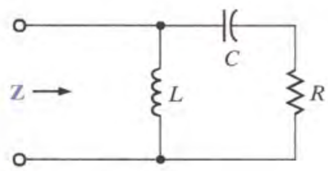


Figure P8.51

SECTION 8.4

8.52 Determine the new parameters of the network shown in Fig. P8.52 if $\mathbf{Z}_{new} = 10^4\mathbf{Z}_{old}$.

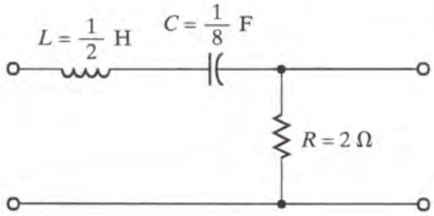


Figure P8.52

8.53 ► Determine the new parameters of the network in Problem 8.52 if $\omega_{new} = 10^4\omega_{old}$.

SECTION 8.5

8.54 Given the network in Fig. P8.54, sketch the magnitude characteristic of the transfer function

$$\mathbf{G}_v(j\omega) = \frac{\mathbf{V}_o}{\mathbf{V}_1}(j\omega)$$

Identify the type of filter.

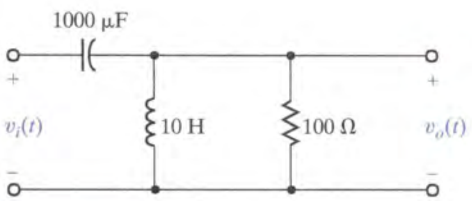


Figure P8.54

8.55 Given the network in Fig. P8.55, sketch the magnitude characteristic of the transfer function

$$\mathbf{G}_v(j\omega) = \frac{\mathbf{V}_o}{\mathbf{V}_1}(j\omega)$$

Identify the type of filter.

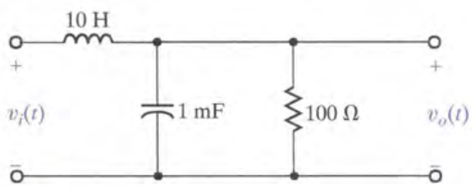


Figure P8.55

8.56 Determine what type of filter the network shown in Fig. P8.56 represents by determining the voltage transfer function.

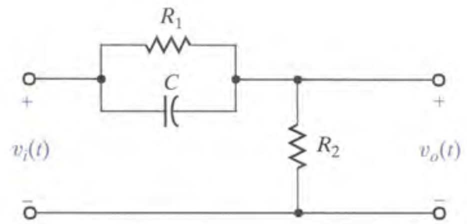


Figure P8.56

8.57 ► **Determine what type of filter the network shown in Fig. P8.57 represents by determining the voltage transfer function.**

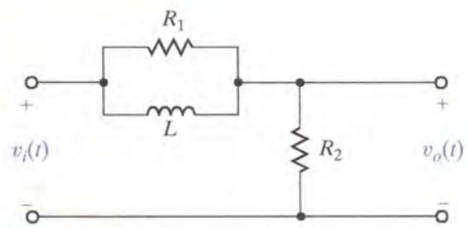


Figure P8.57

8.58 Given the lattice network shown in Fig. P8.58, determine what type of filter this network represents by determining the voltage transfer function.

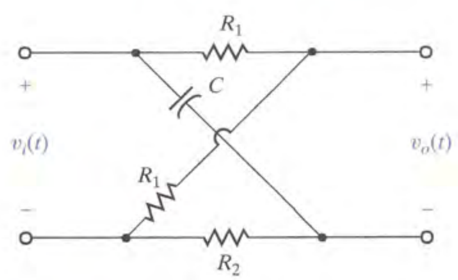


Figure P8.58

8.59 Given the network in Fig. P8.59, and employing the voltage follower analyzed in Chapter 3, determine the voltage transfer function and its magnitude characteristic. What type of filter does the network represent?

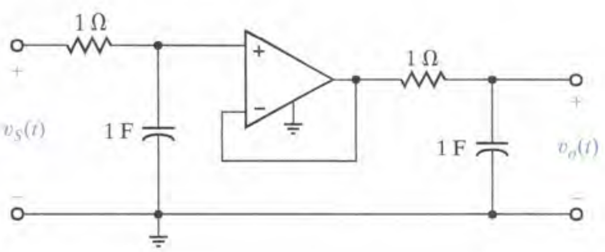


Figure P8.59

8.60 Determine the voltage transfer function and its magnitude characteristic for the network shown in Fig. P8.60 and identify the filter properties.

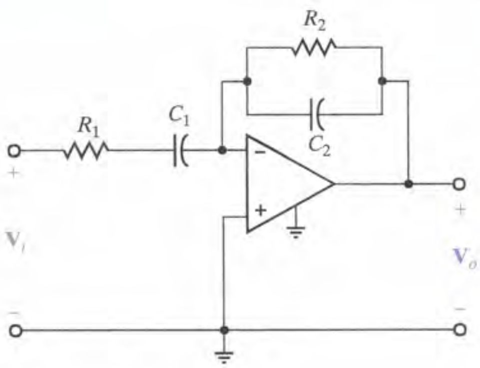


Figure P8.60

8.61 Given the network in Fig. P8.61, find the transfer function

$$\frac{\mathbf{V}_o}{\mathbf{V}_1}(j\omega)$$

and determine what type of filter the network represents.

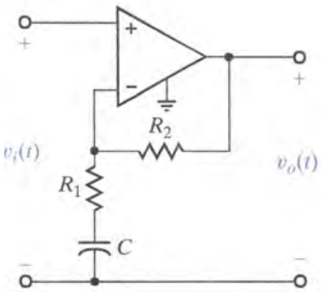


Figure P8.61

8.62 Repeat Problem 8.54 for the network shown in Fig. P8.62.

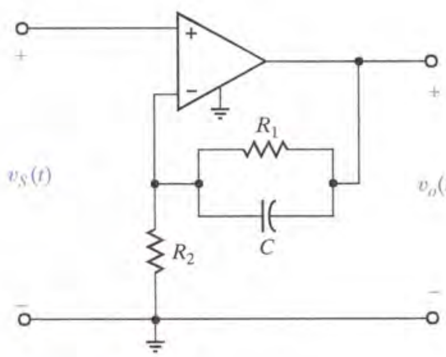


Figure P8.62

In all OTA problems, the specifications are: $g_m - I_{ABC}$ sensitivity = 20, maximum g_m = 1 mS with range of 4 decades.

8.63 For the circuit in Fig. 8.50, find the g_m and I_{ABC} values required for a simulated resistance of 10 kΩ.

8.64 For the circuit in Fig. 8.53, find the g_m and I_{ABC} values required for a simulated resistance of 10 kΩ. Repeat for 750 kΩ.

8.65 Use the summing circuit in Fig. 8.51 to design a circuit that realizes the following function.

$$v_o = 7v_1 + 3v_2$$

8.66 Prove that the circuit in Fig. P8.66 is a simulated inductor. Find the inductance in terms of C, g_{m1} and g_{m2}.

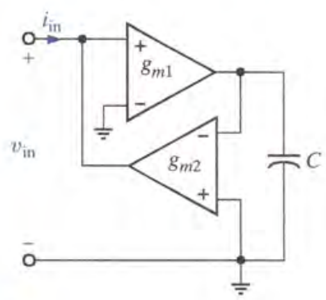


Figure P8.66

8.67 In the Tow–Thomas biquad in Fig. 8.57, $C_1 = 20$ pF, $C_2 = 10$ pF, $g_{m1} = 10$ μS, $g_{m2} = 80$ μS, and $g_{m3} = 10$ μS. Find the low-pass filter transfer function for the $v_{i1} - v_{o2}$ input-output pair. Plot the corresponding Bode plot.

8.68 Find the transfer function of the OTA filter in Fig. P8.68. Express ω_0 and Q in terms of the capacitances and transconductances. What kind of filter is it?

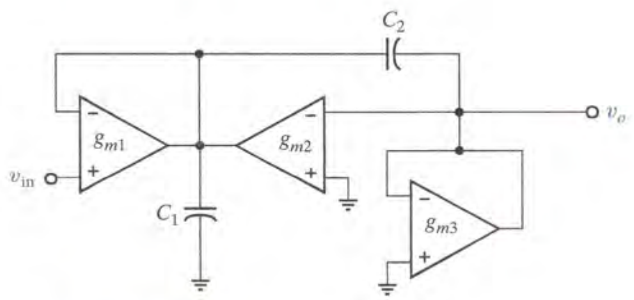


Figure P8.68

8.69 Find the transfer function of the OTA filter in Fig. P8.69. Express ω_0 and Q in terms of the capacitances and transconductances. What kind of filter is it?

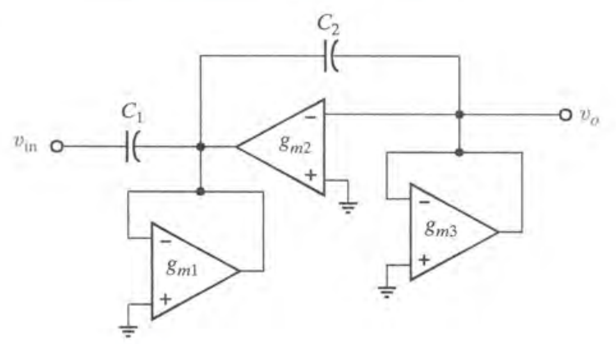


Figure P8.69

8.70 Refer to the ac/dc converter low-pass filter application of Example 8.25. If we put the converter to use powering a calculator, the load current can be modeled by a resistor as shown in Fig. P8.70. The load resistor will affect both the magnitude of the dc component of V_{OF} and the pole frequency. Plot both the pole frequency and the ratio of the 60-Hz component of the output voltage to the dc component of V_{OF} versus R_L for $100\ \Omega \le R_L \le 100$ kΩ. Comment on the advisable limitations on R_L if **(a)** the dc component of V_{OF} is to remain within 20% of its 9-V ideal value; **(b)** the 60-Hz component of V_{OF} remains less than 15% of the dc component.

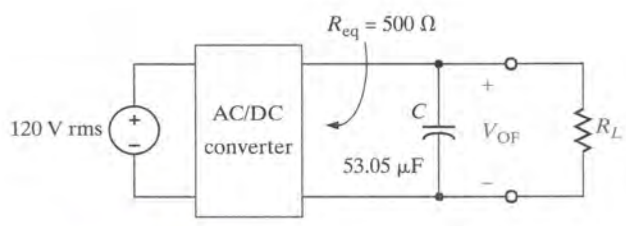


Figure P8.70

8.71 Referring to Example 8.28, design a notch filter for the tape deck for use in Europe, where power utilities generate at 50 Hz.

Typical Problems Found on the FE Exam

8FE-1 ► **Determine the resonant frequency of the circuit in Fig. 8PFE-1, and find the voltage V_o at resonance.**

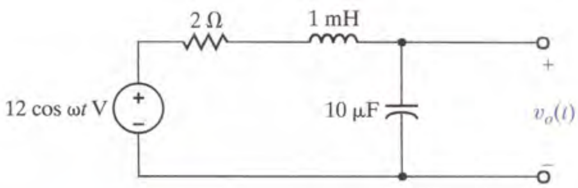


Figure 8PFE-1

8FE-2 Given the series circuit in Fig. 8PFE-2, determine the resonant frequency, and find the value of R so that the BW of the network about the resonant frequency is 200 r/s.

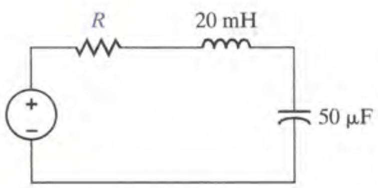


Figure 8PFE-2

8FE-3 ► **Given the low-pass filter circuit shown in Fig. 8PFE-3, find the frequency in Hz at which the output is down 3 dB from the dc, or very low frequency, output.**

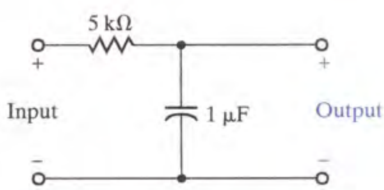


Figure 8PFE-3

8FE-4 Given the band-pass filter shown in Fig. 8PFE-4, find the components L and R necessary to provide a resonant frequency of 1000 r/s and a BW of 100 r/s.

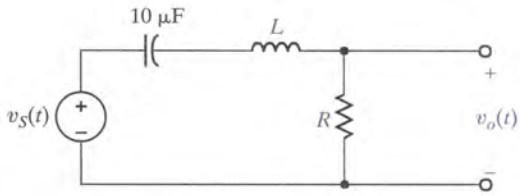


Figure 8PFE-4

8FE-5 ► **Given the low-pass filter shown in Fig. 8PFE-5, find the half-power frequency and the gain of this circuit, if the source frequency is 8 Hz.**

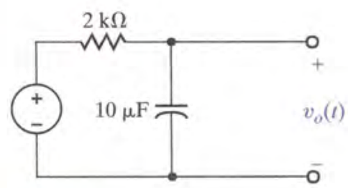


Figure 8PFE-5

A Short Mathematics Primer for Basic Circuit Analysis

Appendix

This primer is designed to help students with the mathematics that is fundamental to their study of basic circuit analysis. Although students taking courses in circuit analysis have typically already encountered the mathematics covered in this document, the concise presentation contained in this primer provides a quick and often much needed review. The material is replete with examples and organized to follow the outline of the book, and thus can be used in a just-in-time basis.

Table of Contents

Techniques for Solving Linearly Independent Simultaneous Equations

INTRODUCTION In our study of both dc and ac circuit analysis we encounter a system of simultaneous equations of the form

$$\begin{aligned} a_{11}x_1 + a_{12}x_2 + \cdots + a_{1n}x_n &= b_1 \\ a_{21}x_1 + a_{22}x_2 + \cdots + a_{2n}x_n &= b_2 \\ \vdots \qquad \vdots \qquad\qquad \vdots \quad &\quad \vdots \\ a_{n1}x_1 + a_{n2}x_2 + \cdots + a_{nn}x_n &= b_n \end{aligned} \qquad \mathbf{1}$$

where the x's and b's are typically currents and voltages or voltages and currents, respectively.

As the title of this section implies, we will assume that the equations are *linearly independent*. Before we outline the solution techniques for the equations of this form, it is perhaps worthwhile to pause long enough to review the meaning of linear independence. Consider the following equations.

$$\tfrac{3}{2}V_1 - \tfrac{1}{2}V_2 - 4 = 0 \qquad \mathbf{2}$$

$$-\tfrac{1}{2}V_1 + \tfrac{5}{6}V_2 + 5 = 0 \qquad \mathbf{3}$$

$$-V_1 - \tfrac{1}{3}V_2 - 1 = 0 \qquad \mathbf{4}$$

Linear independence implies that we cannot find constants a_1, a_2, and a_3 such that

$$a_1\left(\tfrac{3}{2}V_1 - \tfrac{1}{2}V_2 - 4\right) + a_2\left(-\tfrac{1}{2}V_1 + \tfrac{5}{6}V_2 + 5\right) + a_3\left(-V_1 - \tfrac{1}{3}V_2 - 1\right) = 0 \qquad \mathbf{5}$$

However, in this case if we select $a_1 = a_2 = a_3 = 1$, we obtain

$$+\tfrac{3}{2}V_1 - \tfrac{1}{2}V_2 - 4 - \tfrac{1}{2}V_1 + \tfrac{5}{6}V_2 + 5 - V_1 - \tfrac{1}{3}V_2 - 1 = 0$$

$$0 = 0$$

Said another way, this means, for example, that Eqs. (2) and (3) can be used to obtain Eq. (4), and therefore, Eq. (4) is linearly *dependent* on Eqs. (2) and (3). Furthermore, any two of the equations could be used to obtain the third equation. Therefore, only two of the three equations are linearly *independent*.

We will now describe three techniques for solving linearly independent simultaneous equations. The techniques are Gaussian elimination, determinants, and matrices.

GAUSSIAN ELIMINATION The following example will serve to demonstrate the steps involved in applying this technique.

LEARNING Example 1

Let us find the solution to the following set of equations:

$$7X_1 - 4X_2 - X_3 = 4 \qquad \mathbf{6}$$

$$-4X_1 + 7X_2 - 2X_3 = 0 \qquad \mathbf{7}$$

$$-X_1 - 2X_2 + 3X_3 = -1 \qquad \mathbf{8}$$

SOLUTION The algorithm (i.e., step-by-step procedure) for applying the Gaussian elimination method proceeds in the following systematic way. First, we solve Eq. (6) for the variable X_1 in terms of the other variables in X_2 and X_3.

$$X_1 = \tfrac{4}{7} + \tfrac{4}{7}X_2 + \tfrac{1}{7}X_3 \qquad \mathbf{9}$$

We then substitute this result into Eqs. (7) and (8) to obtain

$$\tfrac{33}{7}X_2 - \tfrac{18}{7}X_3 = \tfrac{16}{7} \qquad \mathbf{10}$$

$$-\tfrac{18}{7}X_2 + \tfrac{20}{7}X_3 = -\tfrac{3}{7} \qquad \mathbf{11}$$

Continuing the reduction, we now solve Eq. (10) for X_2 in terms of X_3:

$$X_2 = \tfrac{16}{33} + \tfrac{18}{33}X_3 \qquad \mathbf{12}$$

Substituting this expression for X_2 into Eq. (11) yields

$$\tfrac{336}{231}X_3 = \tfrac{189}{231}$$

or

$$X_3 = 0.563 \tag{13}$$

Now backtracking through the equations, we can determine X_2 from Eq. (12) as

$$X_2 = 0.792$$

and X_1 from Eq. (9) as

$$X_1 = 1.104$$

In this simple example we have not addressed such issues as zero coefficients or the impact of round-off errors. We have, however, illustrated the basic procedure.

Because of the very methodical manner in which the elimination takes place, the algorithm is easily adaptable to computer analysis, and efficient computer codes that implement the technique are available in standard software packages.

DETERMINANTS A *determinant* of order n is a square array of elements a_{ij} arranged as follows:

$$\Delta = \begin{vmatrix} a_{11} & a_{12} \cdots a_{1n} \\ a_{21} & a_{22} \cdots a_{2n} \\ & \vdots & \\ a_{n1} & a_{n2} \cdots a_{nn} \end{vmatrix} \tag{14}$$

The *cofactor* c_{ij} of the element a_{ij} is given by the expression

$$c_{ij} = (-1)^{i+j} A_{ij} \tag{15}$$

where A_{ij} is the determinant that remains after the ith row and jth column are deleted.

LEARNING Example 2

Given the determinant

$$\Delta = \begin{vmatrix} a_{11} & a_{12} & a_{13} \\ a_{21} & a_{22} & a_{23} \\ a_{31} & a_{32} & a_{33} \end{vmatrix}$$

find the cofactor of the element a_{21}.

SOLUTION The cofactor c_{21} for the element a_{21} is

$$c_{21} = (-1)^{2+1} \begin{vmatrix} a_{12} & a_{13} \\ a_{32} & a_{33} \end{vmatrix}$$

The numerical value of the determinant is equal to the sum of products of the elements in any row or column and their cofactors.

LEARNING Example 3

Let us determine the value of the determinant in Example 2 using either the first row or the second column.

SOLUTION Using the first row

$$\begin{aligned} \Delta &= a_{11}c_{11} + a_{12}c_{12} + a_{13}c_{13} \\ &= a_{11}(-1)^{1+1}A_{11} + a_{12}(-1)^{1+2}A_{12} + a_{13}(-1)^{1+3}A_{13} \\ &= a_{11}\begin{vmatrix} a_{22} & a_{23} \\ a_{32} & a_{33} \end{vmatrix} - a_{12}\begin{vmatrix} a_{21} & a_{23} \\ a_{31} & a_{33} \end{vmatrix} + a_{13}\begin{vmatrix} a_{21} & a_{22} \\ a_{31} & a_{32} \end{vmatrix} \end{aligned}$$

Although the 2-by-2 determinants can be evaluated in the same manner, as illustrated above, the result is simply

$$\begin{vmatrix} a & b \\ c & d \end{vmatrix} = ad - cb \tag{16}$$

Therefore, Δ is

$$\begin{aligned} \Delta = a_{11}(a_{22}a_{33} - a_{32}a_{23}) \\ - a_{12}(a_{21}a_{33} - a_{31}a_{23}) + a_{13}(a_{21}a_{32} - a_{31}a_{22}) \end{aligned}$$

If instead of using the first row we had used the second column, we would obtain

$$\begin{aligned} \Delta &= a_{12}(-1)^{1+2}A_{12} + a_{22}(-1)^{2+2}A_{22} + a_{32}(-1)^{3+2}A_{32} \\ &= -a_{12}A_{12} + a_{22}A_{22} - a_{32}A_{32} \\ &= -a_{12}\begin{vmatrix} a_{21} & a_{23} \\ a_{31} & a_{33} \end{vmatrix} + a_{22}\begin{vmatrix} a_{11} & a_{13} \\ a_{31} & a_{33} \end{vmatrix} - a_{32}\begin{vmatrix} a_{11} & a_{13} \\ a_{21} & a_{23} \end{vmatrix} \\ &= -a_{12}(a_{21}a_{33} - a_{31}a_{23}) \\ &\quad + a_{22}(a_{11}a_{33} - a_{31}a_{13}) - a_{32}(a_{11}a_{23} - a_{21}a_{13}) \end{aligned}$$

We could evaluate the determinant using any row or column.

The method of solving the set of simultaneous equations of the form shown in Eq. (1) using determinants is known as *Cramer's rule*. Cramer's rule states that if $\Delta \neq 0$ (that is, the equations are linearly independent), the value of the variable x_1 in Eq. (1) is given by the expression

$$x_1 = \frac{\Delta_1}{\Delta} = \left| \frac{\begin{matrix} b_1 & a_{12} & a_{1n} \\ b_2 & a_{22}\cdots & a_{2n} \\ \vdots & & \vdots \\ b_n & a_{2n} & a_{nn} \end{matrix}}{\Delta} \right| \qquad \mathbf{17}$$

where Δ_1 is the determinant Δ in which the first column is replaced with the column of coefficients. In the general case, x_i is given by an expression similar to Eq. (17) with the ith column replaced by the column of coefficients.

LEARNING Example 4

Let us solve the following equations using determinants.

$$2I_1 - 4I_2 = 8$$
$$-4I_1 + 6I_2 = -4$$

SOLUTION In this case, Δ defined by Eq. (16) is

$$\Delta = \begin{vmatrix} 2 & -4 \\ -4 & 6 \end{vmatrix} = (2)(6) - (-4)(-4) = -4$$

Then using Eq. (17),

$$I_1 = \left| \frac{\begin{matrix} 8 & -4 \\ -4 & 6 \end{matrix}}{-4} \right| = \frac{(8)(6) - (-4)(-4)}{-4} = -8$$

and

$$I_2 = \left| \frac{\begin{matrix} 2 & 8 \\ -4 & -4 \end{matrix}}{-4} \right| = \frac{(2)(-4) - (-4)(8)}{-4} = -6$$

LEARNING Example 5

Let us determine the solution of the following equations using determinants

$$2V_1 - V_2 = 8$$
$$-V_1 + 3V_2 - 2V_3 = 3$$
$$-2V_2 + 4V_3 = -8$$

SOLUTION The determinant for this system of equations is

$$\Delta = \begin{vmatrix} 2 & -1 & 0 \\ -1 & 3 & -2 \\ 0 & -2 & 4 \end{vmatrix}$$

Evaluating the determinant using the first column, we obtain

$$\Delta = 2(-1)^{(1+1)}\begin{vmatrix} 3 & -2 \\ -2 & 4 \end{vmatrix} - 1(-1)^{1+2}\begin{vmatrix} -1 & 0 \\ -2 & 4 \end{vmatrix}$$
$$= 2(12 - 4) + 1(-4) = 12$$

Then

$$V_1 = \frac{1}{12}\begin{vmatrix} 8 & -1 & 0 \\ 3 & 3 & -2 \\ -8 & -2 & 4 \end{vmatrix}$$

Evaluating the determinant using the first row yields

$$V_1 = \frac{1}{12}\left[8(-1)^{1+1}\begin{vmatrix} 3 & -2 \\ -2 & 4 \end{vmatrix} - 1(-1)^{1+2}\begin{vmatrix} 3 & -2 \\ -8 & 4 \end{vmatrix} \right]$$
$$= \frac{1}{12}[8(12 - 4) + 1(12 - 16)] = 5$$

Similarly,

$$V_2 = \frac{1}{12}\begin{vmatrix} 2 & 8 & 0 \\ -1 & 3 & -2 \\ 0 & -8 & 4 \end{vmatrix}$$
$$= \frac{1}{12}\left[2(-1)^{1+1}\begin{vmatrix} 3 & -2 \\ -8 & 4 \end{vmatrix} + 8(-1)^{1+2}\begin{vmatrix} -1 & -2 \\ 0 & 4 \end{vmatrix} \right] = 2$$

and

$$V_3 = \frac{1}{12}\begin{vmatrix} 2 & -1 & 8 \\ -1 & 3 & 3 \\ 0 & -2 & -8 \end{vmatrix}$$
$$= \frac{1}{12}\left[2(-1)^{1+1}\begin{vmatrix} 3 & 3 \\ -2 & -8 \end{vmatrix} - 1(-1)^{2+1}\begin{vmatrix} -1 & 8 \\ -2 & -8 \end{vmatrix} \right] = -1$$

MATRICES A *matrix* is defined to be a rectangular array of numbers arranged in rows and columns and written in the form

$$\begin{bmatrix} a_{11} & a_{12} \cdots & a_{1n} \\ a_{21} & a_{22} \cdots & a_{2n} \\ \vdots & \vdots & \vdots \\ a_{m1} & a_{m2} \cdots & a_{mn} \end{bmatrix}$$

This array is called an m-by-n $(m \times n)$ matrix because it has m rows and n columns. A matrix is a convenient way of representing arrays of numbers; however, one must remember that the matrix itself has no numerical value. In the preceding array the numbers or functions a_{ij} are called the *elements* of the matrix. Any matrix that has the same number of rows as columns is called a *square matrix*. The sum of the diagonal elements of a square matrix is called the *trace* of the matrix.

The addition and subtraction of two matrices of the same order (i.e., $m \times n$) is accomplished as follows:

$$\mathbf{C} = \mathbf{A} \pm \mathbf{B} \tag{18}$$

or

$$c_{ij} = a_{ij} \pm b_{ij} \quad \text{for all } i \text{ and } j$$

That is, the elements of $\mathbf{C}$ are the sum or difference of the corresponding elements of $\mathbf{A}$ and $\mathbf{B}$.

$$\mathbf{C} = \begin{bmatrix} c_{11} \cdots & c_{1n} \\ \vdots & \vdots \\ c_{m1} \cdots & c_{mn} \end{bmatrix} = \begin{bmatrix} a_{11} \cdots & a_{1n} \\ \vdots & \vdots \\ a_{m1} \cdots & a_{mn} \end{bmatrix} \pm \begin{bmatrix} b_{11} \cdots & b_{1n} \\ \vdots & \vdots \\ b_{m1} \cdots & b_{mn} \end{bmatrix}$$
$$= \begin{bmatrix} a_{11} \pm b_{11} \cdots & a_{1n} \pm b_{1n} \\ a_{m1} \pm b_{m1} \cdots & a_{mn} \pm b_{mn} \end{bmatrix}$$

LEARNING Example 6

If

$$\mathbf{A} = \begin{bmatrix} 1 & 3 \\ 2 & 4 \end{bmatrix}, \mathbf{B} = \begin{bmatrix} -1 & -2 \\ 3 & 4 \end{bmatrix}, \text{ and } \mathbf{C} = \begin{bmatrix} 1 & 3 \\ 4 & 2 \end{bmatrix}$$

Find $\mathbf{A} + \mathbf{B}, \mathbf{A} - \mathbf{B}$, and $\mathbf{A} - \mathbf{B} + \mathbf{C}$.

SOLUTION

$$\mathbf{A} + \mathbf{B} = \begin{bmatrix} 0 & 1 \\ 5 & 8 \end{bmatrix}, \mathbf{A} - \mathbf{B} = \begin{bmatrix} 2 & 5 \\ -1 & 0 \end{bmatrix},$$

$$\mathbf{A} - \mathbf{B} + \mathbf{C} = \begin{bmatrix} 3 & 8 \\ 3 & 2 \end{bmatrix}$$

Consider now the multiplication of two matrices. If we are given an $m \times n$ matrix $\mathbf{A}$ and an $n \times r$ matrix $\mathbf{B}$, the product $\mathbf{AB}$ is defined to be an $m \times r$ matrix $\mathbf{C}$ whose elements are given by the expression

$$c_{ij} = \sum_{k=1}^{n} a_{ik}b_{kj}, \quad i = 1, \ldots, m, \quad j = 1, \ldots, r \tag{19}$$

Note that the product $\mathbf{AB}$ is defined only when the number of columns of $\mathbf{A}$ is equal to the number of rows of $\mathbf{B}$.

LEARNING Example 7

Suppose that the matrices $\boldsymbol{A}$ and $\boldsymbol{B}$ are defined as follows:

$$\boldsymbol{A} = \begin{bmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \\ a_{31} & a_{32} \end{bmatrix} \qquad \boldsymbol{B} = \begin{bmatrix} b_{11} & b_{12} \\ b_{21} & b_{22} \end{bmatrix}$$

Find the product $\boldsymbol{C} = \boldsymbol{AB}$.

SOLUTION Note that in the preceding formula, $m = 3$, $n = 2$, $r = 2$. Using this formula, we can calculate

$$c_{11} = \sum_{k=1}^{2} a_{1k}b_{k1} = a_{11}b_{11} + a_{12}b_{21}$$

$$c_{12} = \sum_{k=1}^{2} a_{1k}b_{k2} = a_{11}b_{12} + a_{12}b_{22}$$

$$\vdots$$

These elements form the array

$$\boldsymbol{C} = \boldsymbol{AB} = \begin{bmatrix} a_{11}b_{11} + a_{12}b_{21} & a_{11}b_{12} + a_{12}b_{22} \\ a_{21}b_{11} + a_{22}b_{21} & a_{21}b_{12} + a_{22}b_{22} \\ a_{31}b_{11} + a_{32}b_{21} & a_{31}b_{12} + a_{32}b_{22} \end{bmatrix}$$

A close inspection of the product above illustrates that multiplication is a "row-by-column" operation. In other words, each element in a row of the first matrix is multiplied by the corresponding element in a column of the second matrix and then the products are summed. This operation is diagrammed as follows:

$$\begin{bmatrix} c_{11} & \cdots & c_{1p} \\ \vdots & \boxed{c_{ij}} & \vdots \\ c_{m1} & \cdots & c_{mp} \end{bmatrix} = \begin{bmatrix} a_{11} & \cdots & a_{1n} \\ \boxed{a_{i1} \; \cdots \; a_{in}} \\ a_{m1} & \cdots & a_{mn} \end{bmatrix} \begin{bmatrix} b_{11} & \cdots & \boxed{b_{1j}} & \cdots & b_{1p} \\ \vdots & & \vdots & & \vdots \\ b_{n1} & \cdots & \boxed{b_{nj}} & \cdots & b_{np} \end{bmatrix} \qquad \mathbf{20}$$

The following examples will illustrate the computational technique.

LEARNING Example 8

If

$$\boldsymbol{A} = \begin{bmatrix} 1 & 3 \\ 2 & 4 \end{bmatrix} \quad \text{and} \quad \boldsymbol{B} = \begin{bmatrix} 2 & 1 \\ 3 & 5 \end{bmatrix}$$

$$\boldsymbol{C} = \begin{bmatrix} 1 & 2 \\ 3 & 4 \end{bmatrix} \quad \text{and} \quad \boldsymbol{D} = \begin{bmatrix} 1 \\ 2 \end{bmatrix}$$

Find $\boldsymbol{AB}$ and $\boldsymbol{CD}$.

SOLUTION

$$\boldsymbol{AB} = \begin{bmatrix} (1)(2) + (3)(3) & (1)(1) + (3)(5) \\ (2)(2) + (4)(3) & (2)(1) + (4)(5) \end{bmatrix} = \begin{bmatrix} 11 & 16 \\ 16 & 22 \end{bmatrix}$$

$$\boldsymbol{CD} = \begin{bmatrix} (1)(1) + (2)(2) \\ (3)(1) + (4)(2) \end{bmatrix} = \begin{bmatrix} 5 \\ 11 \end{bmatrix}$$

As defined for determinants, the *cofactor* A_{ij} of the element a_{ij} of any square matrix $\boldsymbol{A}$ is equal to the product $(-1)^{i+j}$ and the determinant of the submatrix obtained from $\boldsymbol{A}$ by deleting row i and column j.

LEARNING Example 9

Given the matrix

$$\boldsymbol{A} = \begin{bmatrix} a_{11} & a_{12} & a_{13} \\ a_{21} & a_{22} & a_{23} \\ a_{31} & a_{32} & a_{33} \end{bmatrix}$$

Find the cofactors A_{11}, A_{12}, and A_{22}.

SOLUTION The cofactors A_{11}, A_{12}, and A_{22} are

$$A_{11} = (-1)^2 \begin{vmatrix} a_{22} & a_{23} \\ a_{32} & a_{33} \end{vmatrix} = a_{22}a_{33} - a_{32}a_{23}$$

$$A_{12} = (-1)^3 \begin{vmatrix} a_{21} & a_{23} \\ a_{31} & a_{33} \end{vmatrix} = -(a_{21}a_{33} - a_{31}a_{23})$$

$$A_{22} = (-1)^2 \begin{vmatrix} a_{11} & a_{13} \\ a_{31} & a_{33} \end{vmatrix} = a_{11}a_{33} - a_{31}a_{13}$$

The *adjoint* of the matrix $\mathbf{A}$ (adj $\mathbf{A}$) is the transpose of the matrix obtained from $\mathbf{A}$ by replacing each element a_{ij} by its cofactors A_{ij}. In others words, if

$$\mathbf{A} = \begin{bmatrix} a_{11} & a_{12} \cdots a_{1n} \\ a_{21} & a_{22} \cdots \cdot \\ \vdots & \vdots \quad \vdots \\ a_{n1} & \cdots \cdots a_{nn} \end{bmatrix}$$

then

$$\text{adj}\,\mathbf{A} = \begin{bmatrix} A_{11} & A_{21} \cdots A_{n1} \\ A_{12} & A_{22} \cdots \cdot \\ \vdots & \vdots \quad \vdots \\ A_{1n} & \cdots \cdots A_{nn} \end{bmatrix}$$

If $\mathbf{A}$ is a square matrix and if there exists a square matrix $\mathbf{A}^{-1}$ such that

$$\mathbf{A}^{-1}\mathbf{A} = \mathbf{A}\mathbf{A}^{-1} = \mathbf{I} \qquad \mathbf{21}$$

then $\mathbf{A}^{-1}$ is called the *inverse* of $\mathbf{A}$. It can be shown that the inverse of the matrix $\mathbf{A}$ is equal to the adjoint divided by the determinant (written here as $|\mathbf{A}|$); that is,

$$\mathbf{A}^{-1} = \frac{\text{adj}\,\mathbf{A}}{|\mathbf{A}|} \qquad \mathbf{22}$$

LEARNING Example 10

Given

$$\mathbf{A} = \begin{bmatrix} 2 & 3 \\ 1 & 4 \end{bmatrix}$$

and

$$\mathbf{B} = \begin{bmatrix} 2 & 3 & 1 \\ 1 & 2 & 3 \\ 3 & 1 & 2 \end{bmatrix}$$

Find $\mathbf{A}^{-1}$ and $\mathbf{B}^{-1}$.

SOLUTION

$$|\mathbf{A}| = (2)(4) - (1)(3) = 5$$

and

$$\text{adj}\,\mathbf{A} = \begin{bmatrix} 4 & -3 \\ -1 & 2 \end{bmatrix}$$

Therefore,

$$\mathbf{A}^{-1} = \tfrac{1}{5}\begin{bmatrix} 4 & -3 \\ -1 & 2 \end{bmatrix}$$

Also

$$|\mathbf{B}| = 2\begin{vmatrix} 2 & 3 \\ 1 & 2 \end{vmatrix} - 1\begin{vmatrix} 3 & 1 \\ 1 & 2 \end{vmatrix} + 3\begin{bmatrix} 3 & 1 \\ 2 & 3 \end{bmatrix}$$

$$= 2 - 5 + 21 = 18$$

and

$$\text{adj}\,\mathbf{B} = \begin{bmatrix} 1 & -5 & 7 \\ 7 & 1 & -5 \\ -5 & 7 & 1 \end{bmatrix}$$

Therefore,

$$\mathbf{B}^{-1} = \tfrac{1}{18}\begin{bmatrix} 1 & -5 & 7 \\ 7 & 1 & -5 \\ -5 & 7 & 1 \end{bmatrix}$$

We now have the tools necessary to solve Eqs. (1) using matrices. The following example illustrates the approach.

LEARNING Example 11

Consider the following set of linearly independent simultaneous equations.

$$2V_1 + 3V_2 + V_3 = 9$$
$$V_1 + 2V_2 + 3V_3 = 6$$
$$3V_1 + V_2 + 2V_3 = 8$$

Let us solve this set of equations using matrix analysis.

SOLUTION Note that this set of simultaneous equations can be written as a single matrix equation in the form

$$\begin{bmatrix} 2 & 3 & 1 \\ 1 & 2 & 3 \\ 3 & 1 & 2 \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \\ V_3 \end{bmatrix} = \begin{bmatrix} 9 \\ 6 \\ 8 \end{bmatrix}$$

or

$$\mathbf{AV} = \mathbf{I}$$

Multiplying both sides of the preceding equation through by $\mathbf{A}^{-1}$ yields

$$\mathbf{A}^{-1}\mathbf{AV} = \mathbf{A}^{-1}\mathbf{I}$$

or

$$\mathbf{V} = \mathbf{A}^{-1}\mathbf{I}$$

$\mathbf{A}^{-1}$ was calculated in Example 10. Employing that inverse here, we obtain

$$\mathbf{V} = \tfrac{1}{18}\begin{bmatrix} 1 & -5 & 7 \\ 7 & 1 & -5 \\ -5 & 7 & 1 \end{bmatrix} \begin{bmatrix} 9 \\ 6 \\ 8 \end{bmatrix}$$

or

$$\begin{bmatrix} V_1 \\ V_2 \\ V_3 \end{bmatrix} = \tfrac{1}{18}\begin{bmatrix} 35 \\ 29 \\ 5 \end{bmatrix}$$

and hence,

$$V_1 = \tfrac{35}{18}, \quad V_2 = \tfrac{29}{18}, \quad \text{and} \quad V_3 = \tfrac{5}{18}$$

Basic Calculus

FUNCTIONS The variable v is said to be a *function* of the variable t if the value of v is determined by the value of t. We use $f(\ )$ to denote the function, and the dependence of v on t is specified by the relationship

$$v = f(t) \qquad \mathbf{23}$$

Since v is dependent upon t, v is the dependent variable and t is the independent variable. In general, v could be dependent upon a number of independent variables.

Linear functions are of special interest, and a *linear function* of t is simply a polynomial of the first degree in t. Such a polynomial can be written as

$$v = mt + b \qquad \mathbf{24}$$

where m and b are constants. The term linear relates to the fact that the graph of a linear function is a straight line. Consider, for example, the two straight lines shown in Fig. 1. Recall that

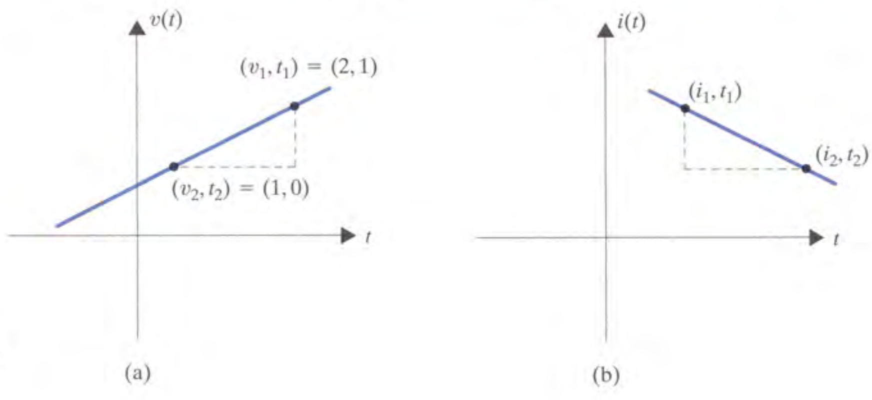


Figure 1
Examples of positive and negative slope.

only two points are required to define a straight line. The points (2, 1) and (1, 0) in Fig. 1a can be used to determine the slope of the line—that is, the vertical difference in the points divided by the corresponding horizontal difference. We call this slope m and express it in the form

$$m = \frac{v_1 - v_2}{t_1 - t_2} \tag{25}$$

Performing the same operation for the straight line in Fig. 1b yields

$$m = \frac{i_1 - i_2}{t_1 - t_2} \tag{26}$$

Note that the slope of the line in Fig. 1a is positive and the slope of the line in Fig. 1b is negative. Furthermore, the slope of a horizontal line is zero.

Equation (24) is called the slope intercept equation because m is the slope and b is the v intercept; that is if $t = 0$, then $v = b$, the point at which the straight line crosses the v axis. If the line passes through the origin of the graph, then $b = 0$.

LEARNING Example 12

Let us determine the equation of the line in Fig. 2.

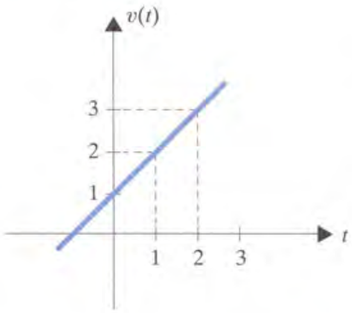


Figure 2
Graph used in Example 12.

SOLUTION The equation for the line is

$$v(t) = mt + b$$

The slope, m, can be determined from the two points (2, 1) and (1, 0). Thus,

$$m = \frac{2 - 1}{1 - 0} = 1$$

In addition, at $t = 0$, the point at which the line crosses the $v(t)$ axis, $v(t) = b = 1$. Therefore, the equation of the line is

$$v(t) = t + 1$$

LEARNING Example 13

We wish to find the equation of the line in Fig. 3.

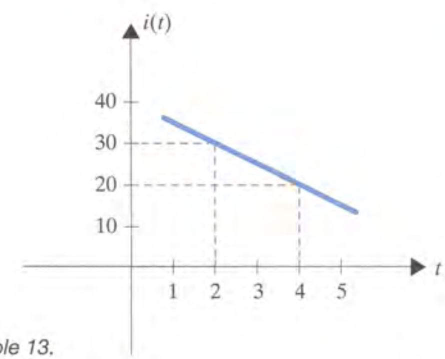


Figure 3
Graph used in Example 13.

SOLUTION Once again, the equation of the line is

$$i(t) = mt + b$$

The slope of the line can be determined from the points (30, 2) and (20, 4). Using these points,

$$m = \frac{30 - 20}{2 - 4} = -5$$

and thus

$$i(t) = -5t + b$$

Substituting the point (30, 2) into the equation for the line yields the value of b.

$$30 = -5(2) + b$$

and b is 40. Hence, the equation of the line is $i(t) = -5t + 40$.

LEARNING Example 14

Let us write the equation for the piecewise linear curve shown in Fig. 4.

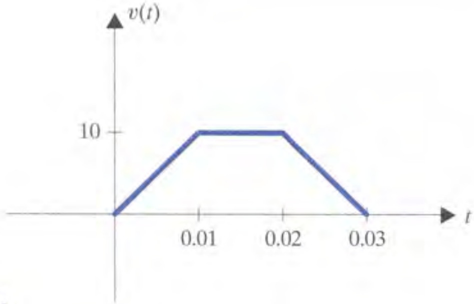


Figure 4
Graph used in Example 14.

SOLUTION As Fig. 4 illustrates, there are three intervals of interest. In each interval the equation for the straight line is $v(t) = mt + b$. In the first interval, $0 < t < 0.01$, the line passes through the origin, and therefore $b = 0$. The slope of this line is

$$m = \frac{10}{0.01} = 1000$$

and therefore

$$v(t) = 1000t, \quad 0 < t < 0.01$$

In the interval $0.01 < t < 0.02$ the curve is horizontal, and therefore the slope is zero. Clearly, $b = 10$ and hence

$$v(t) = 10, \quad 0.01 < t < 0.02$$

In the interval $0.02 < t < 0.03$ the slope of the line is

$$m = \frac{10 - 0}{0.02 - 0.03} = -1000$$

and we find that if this line is extended to the left, it moves up a value of 10 for every 0.01 interval along the t axis. Thus, if the line is extended to the left it will encounter the $v(t)$ axis at $b = 30$. Hence, the equation of the line in this interval is

$$v(t) = -1000t + 30, \quad 0.02 < t < 0.03$$

Therefore, the piecewise linear curve is described by the equations

$$\begin{aligned} v(t) &= 1000t, \quad 0 < t < 0.01 \\ &= 10, \quad 0.01 < t < 0.02 \\ &= -1000t + 30, \quad 0.02 < t < 0.03 \end{aligned}$$

Another function, although nonlinear, of special interest is the exponential function

$$v(t) = ke^{-at}$$

where k is some constant. A graph of this function is shown in Fig. 5. Note that at $t = 0$, $v(t) = k$, and as $t \to \infty$, $v(t) \to 0$. The rate at which $v(t)$ approaches zero is determined by the factor a. If a is large, the decay to 0 is fast, and if a is small, the rate of decay is slow.

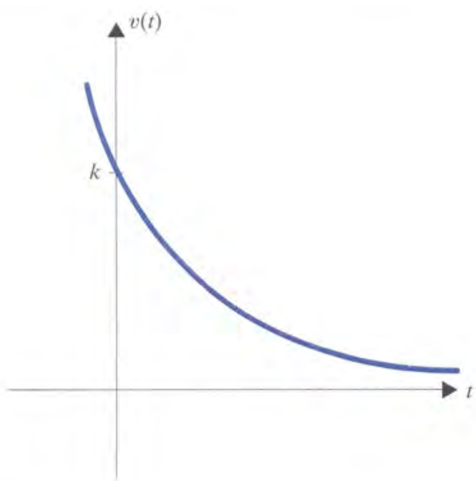


Figure 5
Graph of the exponential function ke^{-at}.

DERIVATIVES The derivative of a function $v = f(t)$ at some point t_0 is expressed as

$$\lim_{\Delta t \to 0} \frac{\Delta v}{\Delta t} = \lim_{\Delta t \to 0} \frac{f(t_0 + \Delta t) - f(t_0)}{\Delta t} \qquad 27$$

and is the limit of the average rate of change of the function between t_0 and $t_0 + \Delta t$. The derivative at some arbitrary point t is written as

$$\frac{dv(t)}{dt} = \lim_{\Delta t \to 0} \frac{\Delta v}{\Delta t} = \lim_{\Delta t \to 0} \frac{f(t + \Delta t) - f(t)}{\Delta t} \qquad 28$$

where we tacitly assume that the limit exists.

A number of derivatives are of particular interest. Assuming b is a constant and x and y are functions of t, the following derivatives are very useful.

$$\frac{d}{dt}(b) = 0$$

$$\frac{d}{dt}(t) = 1$$

$$\frac{d}{dt}(by) = b\frac{dy}{dt}$$

$$\frac{d}{dt}(y \pm x) = \frac{dy}{dt} \pm \frac{dx}{dt}$$

$$\frac{d}{dt}(xy) = x\frac{dy}{dt} + y\frac{dx}{dt}$$

$$\frac{d}{dt}(t^b) = bt^{b-1}$$

$$\frac{d}{dt}(e^{-bt}) = -be^{-bt}$$

$$\frac{d}{dt}(\sin bt) = b\cos bt$$

$$\frac{d}{dt}(\cos bt) = -b\sin bt$$

The following example will illustrate the use of these various derivatives.

LEARNING Example 15

Let us calculate the derivative of the following functions with respect to t.

(a) $v(t) = 4 + 10te^{-at}$

(b) $v(t) = 12e^{-2t}\cos 4t$

(c) $v(t) = t + 6e^{-t}\sin 2t$

SOLUTION

(a) $$\frac{d}{dt}v(t) = 0 + 10\left[e^{-at} - ate^{-at}\right] = 10e^{-at} - 10ate^{-at}$$

(b) $$\frac{d}{dt}v(t) = 12\left[e^{-2t}(-4)\sin 4t - 2e^{-2t}\cos 4t\right] = -48e^{-2t}\sin 4t - 24e^{-2t}\cos 4t$$

(c) $$\frac{d}{dt}v(t) = 1 + 6\left[e^{-t}(2)\cos 2t - e^{-t}\sin 2t\right] = 1 + 12e^{-t}\cos 2t - 6e^{-t}\sin 2t$$

INTEGRALS If $f(t)$ is some particular function and $F(t)$ is a function whose derivative is $f(t)$—that is,

$$\frac{dF(t)}{dt} = f(t) \quad \textbf{29}$$

then we say that $F(t)$ is an integral of $f(t)$.

The most general integral of a function $f(t)$ is called the *indefinite integral* and is defined as

$$\int f(t)\,dt = F(t) + C \quad \textbf{30}$$

where C is the constant of integration and is determined by additional data other than the value of the derivative. Because this function whose derivative is provided is not completely determined—that is, it contains some arbitrary constant of integration—the general integral $\int f(t)\,dt$ is commonly referred to as an indefinite integral. As an example, we note that

$$\int t^2\,dt = \frac{1}{3}t^3 + C$$

since

$$\frac{d}{dt}\left(\frac{1}{3}t^3\right) = t^2$$

Now assume that there is a continuous function $f(t) \geq 0$ in the closed interval $[a, b]$ as shown in Fig. 6a. We wish to compute the area under this curve above the t axis and between the vertical lines defined by $t = a$ and $t = b$.

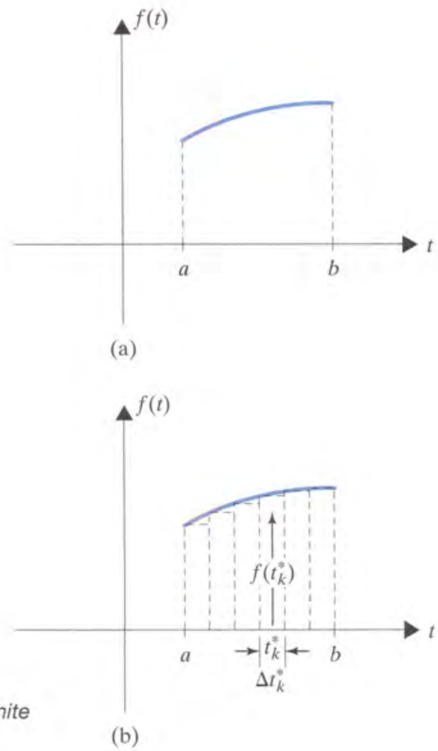


Figure 6
Curves used to define a definite integral.

Therefore, we divide the interval $[a, b]$ into n subintervals Δt_k, $k = 1, 2, 3, \ldots, n$, and in each of the subintervals select a point t_k^*. Then using each subinterval as a base we construct a rectangle with altitude $f(t_k^*)$ as shown in Fig. 6b. If we now sum all the rectangles, we obtain

$$S_n = \sum_{k=1}^{n} f(t_k^*)\Delta t_k \qquad \textbf{31}$$

Note that as we divide the area into more and more rectangles the area is more accurately covered. Finally, in the limit as $n \to \infty$ and $\Delta t_k \to 0$, S_n becomes the definite integral $\int_a^b f(t)\,dt$ and is the area under the curve. Furthermore, a fundamental theorem of integral calculus states that if $f(t)$ is continuous in the interval $[a, b]$, and if

$$F(t) = \int f(t)\,dt \qquad \textbf{32}$$

is any indefinite integral of $f(t)$, then the value of this *definite integral* is given by the expression

$$\int_a^b f(t)\,dt = F(a) - F(b) \qquad \textbf{33}$$

The following examples contain integrals that are typical of those encountered in a study of basic circuit analysis.

LEARNING Example 16

Let us evaluate the following integrals.

(a) $f(t) = \int_0^4 a\,dt$

(b) $f(t) = \int_1^2 t^3\,dt$

SOLUTION

(a) $f(t) = \int_0^4 a\,dt = at|_0^4 = 4a - a(0) = 4a$

(b) $f(t) = \int_1^2 t^3\,dt = \left.\frac{t^4}{4}\right|_1^2 = \frac{1}{4}((2)^4 - (1)^4) = \frac{15}{4}$

LEARNING Example 17

We wish to determine the value of the integral

$$f(t) = \int_0^2 te^{-2t}\,dt$$

SOLUTION We can either look up the form of the integral in a table of integrals or evaluate the integral using what is called integration by parts; that is,

$$\int_a^b u\,dv = uv|_a^b - \int_a^b v\,du$$

Using this latter technique, we let

$$u = t \qquad dv = e^{-2t}\,dt$$

$$du = dt \qquad v = -\frac{1}{2}e^{-2t}$$

Then,

$$\int_0^2 te^{-2t}\,dt = -\left.\frac{t}{2}e^{-2t}\right|_0^2 + \frac{1}{2}\int_0^2 e^{-2t}\,dt$$

$$= -e^{-4} - \left.\frac{1}{4}e^{-2t}\right|_0^2$$

$$= -e^{-4} - \frac{1}{4}e^{-4} + \frac{1}{4}$$

Differential Equations

INTRODUCTION In our study of transient analysis we find that it is necessary to solve both first- and second-order differential equations. These equations naturally arise as we describe the behavior of a network as a function of time when a sudden change in the circuit occurs as a result of switches that open or close. Energy storage elements within the network cause the circuit to pass through a transition period before settling down to some steady-state value. Differential equations are the vehicle needed to study this transition.

We will confine our discussion to first- and second-order differential equations with both constant coefficients and a constant forcing function of the form

$$\frac{dx(t)}{dt} + ax(t) = F \tag{34}$$

and

$$\frac{d^2x(t)}{dt^2} + a_1\frac{dx(t)}{dt} + a_2x(t) = F \tag{35}$$

FIRST-ORDER EQUATIONS A fundamental theorem of differential equations states that if $x(t) = x_p(t)$ is any solution to the equation

$$\frac{dx_p(t)}{dt} + ax_p(t) = F \tag{36}$$

and $x(t) = x_c(t)$ is a solution to the homogeneous equation

$$\frac{dx_c(t)}{dt} + ax_c(t) = 0 \tag{37}$$

then

$$x(t) = x_p(t) + x_c(t) \tag{38}$$

is a solution to the original equation (34). $x_p(t)$ is called the particular integral solution and $x_c(t)$ is called the complementary solution. Therefore, our task is reduced to finding solutions to the two equations (36) and (37).

A perfectly valid technique for solving these equations is simply to guess the solutions. A careful examination of these equations gives us clues that aid us in making these guesses. For example, the right side of equation (36) is a constant, and if we assume that $x_p(t)$ is a constant, then the derivative of a constant is zero and both sides of the equation can be satisfied with this guess.

An examination of the second equation indicates that the solution must be such that the function and its derivative must be of the same form in order for the two terms to add to zero. A logical choice for this solution is an exponential, since the derivative of an exponential is also an exponential. Thus, it is reasonable to assume that the solution to the original equation is of the form

$$x(t) = k_1 + k_2e^{-t/\tau} \tag{39}$$

This is indeed the general solution to Eq. (34). Hence, given a first-order differential equation and some initial condition, our task is simply to find the constants k_1, k_2, and τ. The following example will illustrate the approach.

LEARNING Example 18

Let us determine the solution of the following first-order differential equation with the given initial condition

$$\frac{dx(t)}{dt} + 4x(t) = 8, \quad x(0) = 1$$

SOLUTION The general solution to this equation must be of the form

$$x(t) = k_1 + k_2 e^{-t/\tau}$$

Therefore, substituting this general solution into the equation yields

$$\frac{-k_2}{\tau} e^{-t/\tau} + 4k_1 + 4k_2 e^{-t/\tau} = 8$$

Equating like terms, we obtain

$$\frac{-k_2}{\tau} + 4k_2 = 0$$
$$4k_1 - 8 = 0$$

From which we can determine τ and k_1 as

$$\tau = \frac{1}{4}$$
$$k_1 = 2$$

Then

$$x(t) = 2 + k_2 e^{-4t}$$

We now use the initial condition, $x(0) = 1$, to determine the constant k_2.

$$x(0) = 1 = 2 + k_2$$

Thus,

$$k_2 = -1$$

and therefore the complete solution to the differential equation is

$$x(t) = 2 - 1e^{-4t}$$

Note that this equation satisfies the initial condition.

SECOND-ORDER EQUATIONS In following our development of the solution to the first-order equation, we are once again faced with finding a solution to the two equations

$$\frac{d^2x_p(t)}{dt^2} + a_1 \frac{dx_p(t)}{dt} + a_2 x_p(t) = F \qquad \textbf{40}$$

and

$$\frac{d^2x_c(t)}{dt^2} + a_1 \frac{dx_c(t)}{dt} + a_2 x_c(t) = 0 \qquad \textbf{41}$$

The same arguments used in guessing the solution to the first-order equation are valid in this case also. Hence, we guess that the solution to Eq. (40) is a constant and the solution to Eq. (41) is an exponential.

We now examine the homogeneous equation in some detail. First, we express this equation in a different form by simply making the following substitutions:

$$a_1 = 2\zeta\omega_0, \qquad a_2 = \omega_0^2 \qquad \textbf{42}$$

The equation can now be expressed in the form

$$\frac{d^2x_c(t)}{dt^2} + 2\zeta\omega_0 \frac{dx_c(t)}{dt^2} + \omega_0^2 x_c(t) = 0 \qquad \textbf{43}$$

When we employ these results in circuit analysis, the reason for this substitution will become obvious. The assumed exponential solution is then

$$x_c(t) = ke^{st} \qquad \textbf{44}$$

Substituting this expression into Eq. (43) yields

$$s^2ke^{st} + 2\zeta\omega_0 ske^{st} + \omega_0^2 ke^{st} \qquad \textbf{45}$$

If we now divide both sides of this equation by ke^{st} we obtain what is called the *characteristic equation*

$$s^2 + 2\zeta\omega_0 s + \omega_0^2 \qquad \textbf{46}$$

Applying the quadratic formula to this equation yields the two solutions

$$s_1 = -\zeta\omega_0 + \omega_0\sqrt{\zeta^2 - 1} \qquad 47$$
$$s_2 = -\zeta\omega_0 - \omega_0\sqrt{\zeta^2 - 1}$$

Thus, the general form of the complementary solution is

$$x_c(t) = k_1 e^{s_1 t} + k_2 e^{s_2 t} \qquad 48$$

and the general form of the original second-order equation is

$$x(t) = k_0 + k_1 e^{s_1 t} + k_2 e^{s_2 t} \qquad 49$$

The roots of the characteristic equation determine the form of the complementary solution, and this form is primarily dependent upon the term ζ. If $\zeta > 1$, then the roots are real and unequal; that is, $s_1, s_2 = -a, -b$, and the solution consists of two decaying exponential terms of the form

$$x_c(t) = k_1 e^{-at} + k_2 e^{-bt}$$

If $\zeta < 1$, then the roots are complex conjugates; that is, $s_1, s_2 = -a \pm jb$, and the solution contains two exponential terms with complex exponents that, with the use of Euler's identity, can be expressed in the form

$$x_c(t) = A_1 e^{-at}\cos bt + A_2 e^{-at}\sin bt$$

If $\zeta = 1$, then the roots are real and equal; that is, $s_1, s_2 = -a$. Because the roots are identical, the form of the solution in this case is

$$x_c(t) = B_1 e^{-at} + B_2 t e^{-at}$$

LEARNING Example 19

Let us determine the solution of the differential equation

$$\frac{d^2x(t)}{dt^2} + 7\frac{dx(t)}{dt} + 12x(t) = 24$$

with the initial conditions

$$x(0) = 4, \qquad \frac{dx(0)}{dt} = -2$$

SOLUTION The particular integral solution is assumed to be a constant k_0. Substituting this term into the equation yields

$$\frac{d^2k_0}{dt^2} + 7\frac{dk_0}{dt} + 12k_0 = 24$$

and

$$k_0 = \frac{24}{12} = 2$$

The characteristic equation is

$$s^2 + 7S + 12 = 0$$

and thus $\omega_0 = \sqrt{12}$ and $\zeta = \dfrac{7}{(2\sqrt{12})} > 1$. Hence, the roots are real and unequal. The two roots of the equation can be easily obtained by asking what two numbers multiplied together yield 12 and added together yield 7. The roots are clearly 3 and 4. Then,

$$s_1 = -3$$
$$s_4 = -4$$

and hence,

$$x_c(t) = k_1 e^{-3t} + k_2 e^{-4t}$$

The general form of the solution is therefore,

$$x(t) = 2 + k_1 e^{-3t} + k_2 e^{-4t}$$

Using the initial conditions, we obtain the two equations

$$x(0) = 4 = 2 + k_1 + k_2$$
$$\frac{dx(0)}{dt} = -2 = -3k_1 - 4k_2$$

These equations produce the terms $k_1 = 6$ and $k_2 = -4$, and hence the general solution is

$$x(t) = 2 + 6e^{-3t} - 4e^{-4t}$$

Note that this solution satisfies the initial conditions.

LEARNING Example 20

We wish to find the solution to the equation

$$\frac{d^2x(t)}{dt^2} + 6\frac{dx(t)}{dt} + 13x(t) = 0$$

with the initial conditions

$$x(0) = 2, \quad \frac{dx(0)}{dt} = 1$$

SOLUTION The characteristic equation in this case is

$$s^2 + 6s + 13$$

and hence $\omega_0 = \sqrt{13}$ and $\zeta = \frac{6}{(2\sqrt{13})} < 1$. Thus, the roots are complex conjugates. The factors of the characteristic equation can be derived using the quadratic formula, or we can simply recognize that the equation can be written in the form

$$s^2 + 2(3)s + (3)^2 + (2)^2$$

and then the factors are

$$s_1 = -3 + j2$$
$$s_2 = -3 - j2$$

As a result, the solution is of the form

$$x(t) = k_1e^{-3t}\cos 2t + k_2e^{-3t}\sin 2t$$

Using the initial conditions, we can derive the equations

$$x(0) = 2 = k_1$$

$$\frac{dx(0)}{dt} = 1 = -3k_1 + 2k_2$$

Then $k_1 = 2$ and $k_2 = \frac{7}{2}$, and finally

$$x(t) = 2e^{-3t}\cos 2t + \frac{7}{2}e^{-3t}\sin 2t$$

LEARNING Example 21

Let us determine the solution of the differential equation

$$\frac{d^2x(t)}{dt^2} + 6\frac{dx(t)}{dt} + 9x(t) = 36$$

With the initial conditions

$$x(0) = 1, \quad \frac{dx(0)}{dt} = 2$$

SOLUTION The particular integral solution is assumed to be a constant. Therefore,

$$\frac{d^2k_0}{dt^2} + 6\frac{dk_0}{dt} + 9k_0 = 36$$

and hence

$$k_0 = 4$$

The characteristic equation is

$$s^2 + 6s + 9 = 0$$

and hence $\omega_0 = 3$ and $\zeta = 1$. The roots of the equation are real and equal; that is,

$$s_1 = -3$$
$$s_2 = -3$$

Therefore, the solution is of the form

$$x(t) = 4 + k_1e^{-3t} + k_2te^{-3t}$$

Applying the initial conditions, we obtain

$$x(0) = 1 = 4 + k_1$$

$$\frac{dx(0)}{dt} = 2 = -3k_1 + k_2$$

We find that $k_1 = -3$ and $k_2 = -7$, and therefore the complete solution is

$$x(t) = 4 - 3e^{-3t} - 7te^{-3t}$$

Complex Numbers

REPRESENTATIONS Complex numbers are typically represented in three forms: exponential, polar, or rectangular. In the exponential form a complex number **A** is written as

$$\mathbf{A} = ze^{j\theta} \qquad \textbf{50}$$

The real quantity z is known as the amplitude or magnitude, the real quantity θ is called the *angle* as shown in Fig. 7, and j is the imaginary operator $j = \sqrt{-1}$. θ, which is the angle between the real axis and **A**, may be expressed in either radians or degrees.

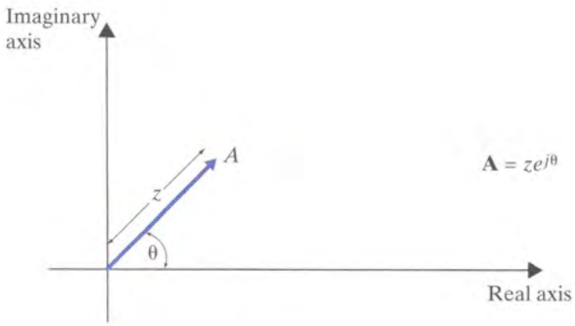


Figure 7
The exponential form of a complex number.

The polar form of a complex number **A**, which is symbolically equivalent to the exponential form, is written as

$$\mathbf{A} = z\underline{/\theta} \qquad (51)$$

and the rectangular representation of a complex number is written as

$$\mathbf{A} = x + jy \qquad (52)$$

where x is the real part of **A** and y is the imaginary part of **A**.

The connection between the various representations of **A** can be seen via Euler's identity, which is

$$e^{j\theta} = \cos\theta + j\sin\theta \qquad (53)$$

Figure 8 illustrates that this function in rectangular form is a complex number with a unit amplitude.

Using this identity, the complex number **A** can be written as

$$\mathbf{A} = ze^{j\theta} = z\cos\theta + jz\sin\theta \qquad (54)$$

which, as shown in Fig. 9, can be written as

$$\mathbf{A} = x + jy$$

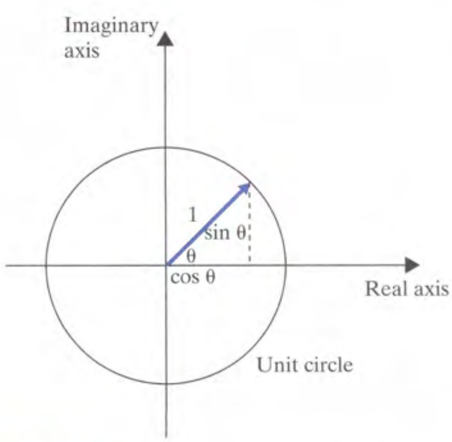


Figure 8
A graphical interpretation of Euler's identity.

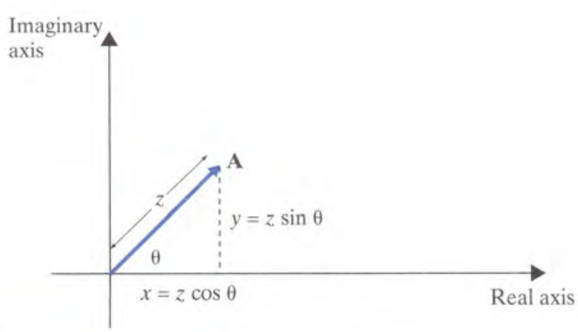


Figure 9
The relationship between the exponential and rectangular representation of a complex number.

Equating the real and imaginary parts of these two equations yields

$$x = z\cos\theta \qquad 55$$
$$y = z\sin\theta$$

From these equations we obtain

$$x^2 + y^2 = z^2\cos^2\theta + z^2\sin^2\theta = z^2 \qquad 56$$

Therefore,

$$z = \sqrt{x^2 + y^2} \qquad 57$$

Additionally,

$$\frac{z\sin\theta}{z\cos\theta} = \tan\theta = \frac{y}{x}$$

and hence

$$\theta = \tan^{-1}\frac{y}{x} \qquad 58$$

The interrelationships among the three representations of a complex number are as follows.

Exponential	**Polar**	**Rectangular**
$ze^{j\theta}$	$z\underline{/\theta}$	$x + jy$
$\theta = \tan^{-1} y/x$	$\theta = \tan^{-1} y/x$	$x = z\cos\theta$
$z = \sqrt{x^2 + y^2}$	$z = \sqrt{x^2 + y^2}$	$y = z\sin\theta$

We will now show that the operations of addition, subtraction, multiplication, and division apply to complex numbers in the same manner that they apply to real numbers.

ADDITION The *sum* of two complex numbers $\mathbf{A} = x_1 + jy_1$ and $\mathbf{B} = x_2 + jy_2$ is

$$\begin{aligned}\mathbf{A} + \mathbf{B} &= x_1 + jy_1 + x_2 + jy_2 \\ &= (x_1 + x_2) + j(y_1 + y_2)\end{aligned} \qquad 59$$

That is, we simply add the individual real parts, and we add the individual imaginary parts to obtain the components of the resultant complex number.

LEARNING Example 22

Suppose we wish to calculate the sum $\mathbf{A} + \mathbf{B}$ if $\mathbf{A} = 5\underline{/36.9°}$ and $\mathbf{B} = 5\underline{/53.1°}$.

SOLUTION We must first convert from polar to rectangular form.

$$\mathbf{A} = 5\underline{/36.9°} = 4 + j3$$
$$\mathbf{B} = 5\underline{/53.1°} = 3 + j4$$

Therefore,

$$\begin{aligned}\mathbf{A} + \mathbf{B} &= 4 + j3 + 3 + j4 = 7 + j7 \\ &= 9.9\underline{/45°}\end{aligned}$$

SUBTRACTION The *difference* of two complex numbers $\mathbf{A} = x_1 + jy_1$ and $\mathbf{B} = x_2 + jy_2$ is

$$\begin{aligned}\mathbf{A} - \mathbf{B} &= (x_1 + jy_1) - (x_2 + jy_2) \\ &= (x_1 - x_2) + j(y_1 - y_2)\end{aligned} \qquad 60$$

That is, we simply subtract the individual real parts, and we subtract the individual imaginary parts to obtain the components of the resultant complex number.

LEARNING Example 23

Let us calculate the difference **A** − **B** if $A = 5\angle 36.9^\circ$ and $\mathbf{B} = 5\angle 53.1^\circ$.

SOLUTION Converting both numbers from polar to rectangular form,

$$\mathbf{A} = 5\angle 36.9^\circ = 4 + j3$$
$$\mathbf{B} = 5\angle 53.1^\circ = 3 + j4$$

Then

$$\mathbf{A} - \mathbf{B} = (4 + j3) - (3 + j4) = 1 - j1 = \sqrt{2}\angle -45^\circ$$

MULTIPLICATION The *product* of two complex numbers $\mathbf{A} = z_1\angle\theta_1 = x_1 + jy_1$ and $\mathbf{B} = z_2\angle\theta_2 = x_2 + jy_2$ is

$$\mathbf{AB} = \left(z_1 e^{j\theta_1}\right)\left(z_2 e^{j(\theta_2)}\right) = z_1 z_2 \angle \theta_1 + \theta_2 \qquad \textbf{61}$$

LEARNING Example 24

Given $\mathbf{A} = 5\angle 36.9^\circ$ and $\mathbf{B} = 5\angle 53.1^\circ$, we wish to calculate the product in both polar and rectangular forms.

SOLUTION

$$\mathbf{AB} = (5\angle 36.9^\circ)(5\angle 53.1^\circ) = 25\angle 90^\circ = (4 + j3)(3 + j4)$$
$$= 12 + j16 + j9 + j^2 12 = 25j = 25\angle 90^\circ$$

LEARNING Example 25

Given $\mathbf{A} = 2 + j2$ and $\mathbf{B} = 3 + j4$, we wish to calculate the product **AB**.

SOLUTION

$$\mathbf{A} = 2 + j2 = 2.828\angle 45^\circ$$
$$\mathbf{B} = 3 + j4 = 5\angle 53.1^\circ$$

and

$$\mathbf{AB} = (2.828\angle 45^\circ)(5\angle 53.1^\circ) = 14.14\angle 98.1^\circ$$

DIVISION The *quotient* of two complex numbers $\mathbf{A} = z_1\angle\theta_1 = x_1 + jy_1$ and $\mathbf{B} = z_2\angle\theta_2 = x_2 + jy_2$ is

$$\frac{\mathbf{A}}{\mathbf{B}} = \frac{z_1 e^{j\theta_1}}{z_2 e^{j\theta_2}} = \frac{z_1}{z_2} e^{j(\theta_1 - \theta_2)} = \frac{z_1}{z_2} \angle \theta_1 - \theta_2 \qquad \textbf{62}$$

LEARNING Example 26

Given $\mathbf{A} = 10\angle 30^\circ$ and $\mathbf{B} = 5\angle 53.1^\circ$, we wish to determine the quotient **A**/**B** in both polar and rectangular forms.

SOLUTION

$$\frac{\mathbf{A}}{\mathbf{B}} = \frac{10\angle 30^\circ}{5\angle 53.1^\circ} = 2\angle -23.1^\circ = 1.84 - j0.79$$

LEARNING Example 27

Given $\mathbf{A} = 3 + j4$ and $\mathbf{B} = 1 + j2$, we wish to calculate the quotient $\mathbf{A}/\mathbf{B}$.

SOLUTION

$$\mathbf{A} = 3 + j4 = 5\underline{/53.1^\circ}$$

and

$$\mathbf{B} = 1 + j2 = 2.236\underline{/63^\circ}$$

$$\frac{\mathbf{A}}{\mathbf{B}} = \frac{5\underline{/53.1^\circ}}{2.236\underline{/63^\circ}} = 2.236\underline{/-9.9^\circ}$$

LEARNING Example 28

If $\mathbf{A} = 3 + j4$, let us compute $1/\mathbf{A}$.

SOLUTION

$$\mathbf{A} = 3 + j4 = 5\underline{/53.1^\circ}$$

and

$$1/\mathbf{A} = \frac{1\underline{/0^\circ}}{5\underline{/53.1^\circ}} = 0.2\underline{/-53.1^\circ}$$

or

$$\frac{1}{\mathbf{A}} = \frac{1}{3 + j4} = \frac{3 - j4}{(3 + j4)(3 - j4)}$$

$$= \frac{3 - j4}{25} = 0.12 - j0.16$$

Index