

William H. Hayt, Jr. • Jack E. Kemmerly • Steven M. Durbin



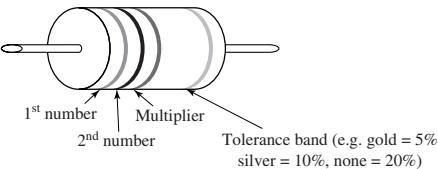
# Engineering Circuit Analysis

Eighth Edition

*This page intentionally left blank*

## The Resistor Color Code

Band color	Black	Brown	Red	Orange	Yellow	Green	Blue	Violet	Gray	White
Numeric value	0	1	2	3	4	5	6	7	8	9



1. Write down the numeric value corresponding to the first band on the left.
2. Write down the numeric value corresponding to the second band from the left.
3. Write down the number of zeros indicated by the multiplier band, which represents a power of 10 (black = no extra zeros, brown = 1 zero, etc.). A gold multiplier band indicates that the decimal is shifted one place to the left; a silver multiplier band indicates that the decimal is shifted two places to the left.
4. The tolerance band represents the precision. So, for example, we would not be surprised to find a 100  $\Omega$  5 percent tolerance resistor that measures anywhere in the range of 95 to 105  $\Omega$ .

### Example

$$\begin{array}{lll} \text{Red Red Orange Gold} & = 22,000 & \text{or } 22 \times 10^3 = 22 \text{ k}\Omega, 5\% \text{ tolerance} \\ \text{Blue Gray Gold} & = 6.8 & \text{or } 68 \times 10^{-1} = 6.8 \Omega, 20\% \text{ tolerance} \end{array}$$

### Standard 5 Percent Tolerance Resistor Values

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

TABLE 14.1 Laplace Transform Pairs

$f(t) = \mathcal{L}^{-1}\{\mathbf{F}(s)\}$	$\mathbf{F}(s) = \mathcal{L}\{f(t)\}$	$f(t) = \mathcal{L}^{-1}\{\mathbf{F}(s)\}$	$\mathbf{F}(s) = \mathcal{L}\{f(t)\}$
$\delta(t)$	1	$\frac{1}{\beta - \alpha} (e^{-\alpha t} - e^{-\beta t}) u(t)$	$\frac{1}{(s + \alpha)(s + \beta)}$
$u(t)$	$\frac{1}{s}$	$\sin \omega t u(t)$	$\frac{\omega}{s^2 + \omega^2}$
$tu(t)$	$\frac{1}{s^2}$	$\cos \omega t u(t)$	$\frac{s}{s^2 + \omega^2}$
$\frac{t^{n-1}}{(n-1)!} u(t), n = 1, 2, \dots$	$\frac{1}{s^n}$	$\sin(\omega t + \theta) u(t)$	$\frac{s \sin \theta + \omega \cos \theta}{s^2 + \omega^2}$
$e^{-\alpha t} u(t)$	$\frac{1}{s + \alpha}$	$\cos(\omega t + \theta) u(t)$	$\frac{s \cos \theta - \omega \sin \theta}{s^2 + \omega^2}$
$te^{-\alpha t} u(t)$	$\frac{1}{(s + \alpha)^2}$	$e^{-\alpha t} \sin \omega t u(t)$	$\frac{\omega}{(s + \alpha)^2 + \omega^2}$
$\frac{t^{n-1}}{(n-1)!} e^{-\alpha t} u(t), n = 1, 2, \dots$	$\frac{1}{(s + \alpha)^n}$	$e^{-\alpha t} \cos \omega t u(t)$	$\frac{s + \alpha}{(s + \alpha)^2 + \omega^2}$

TABLE 6.1 Summary of Basic Op Amp Circuits

Name	Circuit Schematic	Input-Output Relation
Inverting Amplifier		$v_{\text{out}} = -\frac{R_f}{R_1} v_{\text{in}}$
Noninverting Amplifier		$v_{\text{out}} = \left(1 + \frac{R_f}{R_1}\right) v_{\text{in}}$
Voltage Follower (also known as a Unity Gain Amplifier)		$v_{\text{out}} = v_{\text{in}}$
Summing Amplifier		$v_{\text{out}} = -\frac{R_f}{R} (v_1 + v_2 + v_3)$
Difference Amplifier		$v_{\text{out}} = v_2 - v_1$

# **ENGINEERING CIRCUIT ANALYSIS**

*This page intentionally left blank*



# ENGINEERING CIRCUIT ANALYSIS

EIGHTH EDITION

**William H. Hayt, Jr. (deceased)**

*Purdue University*

**Jack E. Kemmerly (deceased)**

*California State University*

**Steven M. Durbin**

*University at Buffalo*

*The State University of New York*





## ENGINEERING CIRCUIT ANALYSIS, EIGHTH EDITION

Published by McGraw-Hill, a business unit of The McGraw-Hill Companies, Inc., 1221 Avenue of the Americas, New York, NY 10020. Copyright © 2012 by The McGraw-Hill Companies, Inc. All rights reserved. Previous editions © 2007, 2002, and 1993. Printed in the United States of America.

No part of this publication may be reproduced or distributed in any form or by any means, or stored in a database or retrieval system, without the prior written consent of The McGraw-Hill Companies, Inc., including, but not limited to, in any network or other electronic storage or transmission, or broadcast for distance learning.

Some ancillaries, including electronic and print components, may not be available to customers outside the United States.

This book is printed on acid-free paper.

1 2 3 4 5 6 7 8 9 0 DOW/DOW 1 0 9 8 7 6 5 4 3 2 1

ISBN 978-0-07-352957-8

MHID 0-07-352957-5

Vice President & Editor-in-Chief: *Marty Lange*

Vice President & Director of Specialized Publishing: *Janice M. Roerig-Blong*

Editorial Director: *Michael Lange*

Global Publisher: *Raghothaman Srinivasan*

Senior Marketing Manager: *Curt Reynolds*

Developmental Editor: *Darlene M. Schueler*

Lead Project Manager: *Jane Mohr*

Buyer: *Kara Kudronowicz*

Design Coordinator: *Brenda A. Rolwes*

Senior Photo Research Coordinator: *John C. Leland*

Senior Media Project Manager: *Tammy Juran*

Compositor: *MPS Limited, a Macmillan Company*

Typeface: *10/12 Times Roman*

Printer: *R. R. Donnelley*

Cover Image: © *Getty Images*

Cover Designer: *Studio Montage, St. Louis, Missouri*

MATLAB is a registered trademark of The MathWorks, Inc.

PSpice is a registered trademark of Cadence Design Systems, Inc.

The following photos are courtesy of Steve Durbin: Page 5, Fig. 2.22a, 2.24a–c, 5.34, 6.1a, 7.2a–c, 7.11a–b, 13.15, 17.29

### Library of Congress Cataloging-in-Publication Data

Hayt, William Hart, 1920–1999

Engineering circuit analysis / William H. Hayt, Jr., Jack E. Kemmerly, Steven M. Durbin. — 8th ed.

p. cm.

Includes index.

ISBN 978-0-07-352957-8

1. Electric circuit analysis. 2. Electric network analysis. I. Kemmerly, Jack E. (Jack Ellsworth), 1924–1998

II. Durbin, Steven M. III. Title.

TK454.H4 2012

621.319'2—dc22

2011009912

*To Sean and Kristi.  
The best part of every day.*

*This page intentionally left blank*

# ABOUT THE AUTHORS

**WILLIAM H. HAYT, Jr.**, received his B.S. and M.S. at Purdue University and his Ph.D. from the University of Illinois. After spending four years in industry, Professor Hayt joined the faculty of Purdue University, where he served as Professor and Head of the School of Electrical Engineering, and as Professor Emeritus after retiring in 1986. Besides *Engineering Circuit Analysis*, Professor Hayt authored three other texts, including *Engineering Electromagnetics*, now in its eighth edition with McGraw-Hill. Professor Hayt's professional society memberships included Eta Kappa Nu, Tau Beta Pi, Sigma Xi, Sigma Delta Chi, Fellow of IEEE, ASEE, and NAEB. While at Purdue, he received numerous teaching awards, including the university's Best Teacher Award. He is also listed in Purdue's Book of Great Teachers, a permanent wall display in the Purdue Memorial Union, dedicated on April 23, 1999. The book bears the names of the inaugural group of 225 faculty members, past and present, who have devoted their lives to excellence in teaching and scholarship. They were chosen by their students and their peers as Purdue's finest educators.

**JACK E. KEMMERLY** received his B.S. magna cum laude from The Catholic University of America, M.S. from University of Denver, and Ph.D. from Purdue University. Professor Kemmerly first taught at Purdue University and later worked as principal engineer at the Aeronutronic Division of Ford Motor Company. He then joined California State University, Fullerton, where he served as Professor, Chairman of the Faculty of Electrical Engineering, Chairman of the Engineering Division, and Professor Emeritus. Professor Kemmerly's professional society memberships included Eta Kappa Nu, Tau Beta Pi, Sigma Xi, ASEE, and IEEE (Senior Member). His pursuits outside of academe included being an officer in the Little League and a scoutmaster in the Boy Scouts.

**STEVEN M. DURBIN** received the B.S., M.S. and Ph.D. degrees in Electrical Engineering from Purdue University, West Lafayette, Indiana. Subsequently, he was with the Department of Electrical Engineering at Florida State University and Florida A&M University before joining the University of Canterbury, New Zealand, in 2000. Since August 2010, he has been with the University at Buffalo, The State University of New York, where he holds a joint appointment between the Departments of Electrical Engineering and Physics. His teaching interests include circuits, electronics, electromagnetics, solid-state electronics and nanotechnology. His research interests are primarily concerned with the development of new semiconductor materials—in particular those based on oxide and nitride compounds—as well as novel optoelectronic device structures. He is a founding principal investigator of the MacDiarmid Institute for Advanced Materials and Nanotechnology, a New Zealand National Centre of Research Excellence, and coauthor of over 100 technical publications. He is a senior member of the IEEE, and a member of Eta Kappa Nu, the Electron Devices Society, the Materials Research Society, the AVS (formerly the American Vacuum Society), the American Physical Society, and the Royal Society of New Zealand.

*This page intentionally left blank*

# BRIEF CONTENTS

PREFACE xv

1	INTRODUCTION	1
2	BASIC COMPONENTS AND ELECTRIC CIRCUITS	9
3	VOLTAGE AND CURRENT LAWS	39
4	BASIC NODAL AND MESH ANALYSIS	79
5	HANDY CIRCUIT ANALYSIS TECHNIQUES	123
6	THE OPERATIONAL AMPLIFIER	175
7	CAPACITORS AND INDUCTORS	217
8	BASIC <i>RL</i> AND <i>RC</i> CIRCUITS	261
9	THE <i>RLC</i> CIRCUIT	321
10	SINUSOIDAL STEADY-STATE ANALYSIS	371
11	AC CIRCUIT POWER ANALYSIS	421
12	POLYPHASE CIRCUITS	457
13	MAGNETICALLY COUPLED CIRCUITS	493
14	COMPLEX FREQUENCY AND THE LAPLACE TRANSFORM	533
15	CIRCUIT ANALYSIS IN THE <i>s</i> -DOMAIN	571
16	FREQUENCY RESPONSE	619
17	TWO-PORT NETWORKS	687
18	FOURIER CIRCUIT ANALYSIS	733

Appendix 1 AN INTRODUCTION TO NETWORK TOPOLOGY 791

Appendix 2 SOLUTION OF SIMULTANEOUS EQUATIONS 803

Appendix 3 A PROOF OF THÉVENIN'S THEOREM 811

Appendix 4 A PSPICE® TUTORIAL 813

Appendix 5 COMPLEX NUMBERS 817

Appendix 6 A BRIEF MATLAB® TUTORIAL 827

Appendix 7 ADDITIONAL LAPLACE TRANSFORM THEOREMS 833

INDEX 839

*This page intentionally left blank*

# CONTENTS

## CHAPTER 1

### INTRODUCTION 1

- 1.1** Overview of Text 2
  - 1.2** Relationship of Circuit Analysis to Engineering 4
  - 1.3** Analysis and Design 5
  - 1.4** Computer-Aided Analysis 6
  - 1.5** Successful Problem-Solving Strategies 7
- READING FURTHER 8

## CHAPTER 2

### BASIC COMPONENTS AND ELECTRIC CIRCUITS 9

- 2.1** Units and Scales 9
  - 2.2** Charge, Current, Voltage, and Power 11
  - 2.3** Voltage and Current Sources 17
  - 2.4** Ohm's Law 22
- SUMMARY AND REVIEW 28
- READING FURTHER 29
- EXERCISES 29

## CHAPTER 3

### VOLTAGE AND CURRENT LAWS 39

- 3.1** Nodes, Paths, Loops, and Branches 39
  - 3.2** Kirchhoff's Current Law 40
  - 3.3** Kirchhoff's Voltage Law 42
  - 3.4** The Single-Loop Circuit 46
  - 3.5** The Single-Node-Pair Circuit 49
  - 3.6** Series and Parallel Connected Sources 51
  - 3.7** Resistors in Series and Parallel 55
  - 3.8** Voltage and Current Division 61
- SUMMARY AND REVIEW 66
- READING FURTHER 67
- EXERCISES 67

## CHAPTER 4

### BASIC NODAL AND MESH ANALYSIS 79

- 4.1** Nodal Analysis 80
- 4.2** The Supernode 89
- 4.3** Mesh Analysis 92
- 4.4** The Supermesh 98

- 4.5** Nodal vs. Mesh Analysis: A Comparison 101
  - 4.6** Computer-Aided Circuit Analysis 103
- SUMMARY AND REVIEW 107
- READING FURTHER 109
- EXERCISES 109

## CHAPTER 5

### HANDY CIRCUIT ANALYSIS TECHNIQUES 123

- 5.1** Linearity and Superposition 123
  - 5.2** Source Transformations 133
  - 5.3** Thévenin and Norton Equivalent Circuits 141
  - 5.4** Maximum Power Transfer 152
  - 5.5** Delta-Wye Conversion 154
  - 5.6** Selecting an Approach: A Summary of Various Techniques 157
- SUMMARY AND REVIEW 158
- READING FURTHER 159
- EXERCISES 159

## CHAPTER 6

### THE OPERATIONAL AMPLIFIER 175

- 6.1** Background 175
  - 6.2** The Ideal Op Amp: A Cordial Introduction 176
  - 6.3** Cascaded Stages 184
  - 6.4** Circuits for Voltage and Current Sources 188
  - 6.5** Practical Considerations 192
  - 6.6** Comparators and the Instrumentation Amplifier 203
- SUMMARY AND REVIEW 206
- READING FURTHER 207
- EXERCISES 208

## CHAPTER 7

### CAPACITORS AND INDUCTORS 217

- 7.1** The Capacitor 217
- 7.2** The Inductor 225
- 7.3** Inductance and Capacitance Combinations 235
- 7.4** Consequences of Linearity 238
- 7.5** Simple Op Amp Circuits with Capacitors 240
- 7.6** Duality 242

- 7.7** Modeling Capacitors and Inductors with PSpice 245  
**SUMMARY AND REVIEW** 247  
**READING FURTHER** 249  
**EXERCISES** 249

**CHAPTER 8****BASIC RL AND RC CIRCUITS 261**

- 8.1** The Source-Free *RL* Circuit 261  
**8.2** Properties of the Exponential Response 268  
**8.3** The Source-Free *RC* Circuit 272  
**8.4** A More General Perspective 275  
**8.5** The Unit-Step Function 282  
**8.6** Driven *RL* Circuits 286  
**8.7** Natural and Forced Response 289  
**8.8** Driven *RC* Circuits 295  
**8.9** Predicting the Response of Sequentially Switched Circuits 300  
**SUMMARY AND REVIEW** 306  
**READING FURTHER** 308  
**EXERCISES** 309

**CHAPTER 9****THE RLC CIRCUIT 321**

- 9.1** The Source-Free Parallel Circuit 321  
**9.2** The Overdamped Parallel *RLC* Circuit 326  
**9.3** Critical Damping 334  
**9.4** The Underdamped Parallel *RLC* Circuit 338  
**9.5** The Source-Free Series *RLC* Circuit 345  
**9.6** The Complete Response of the *RLC* Circuit 351  
**9.7** The Lossless *LC* Circuit 359  
**SUMMARY AND REVIEW** 361  
**READING FURTHER** 363  
**EXERCISES** 363

**CHAPTER 10****SINUSOIDAL STEADY-STATE ANALYSIS 371**

- 10.1** Characteristics of Sinusoids 371  
**10.2** Forced Response to Sinusoidal Functions 374  
**10.3** The Complex Forcing Function 378  
**10.4** The Phasor 383  
**10.5** Impedance and Admittance 389  
**10.6** Nodal and Mesh Analysis 394  
**10.7** Superposition, Source Transformations and Thévenin's Theorem 397  
**10.8** Phasor Diagrams 406

- SUMMARY AND REVIEW** 409  
**READING FURTHER** 410  
**EXERCISES** 410

**CHAPTER 11****AC CIRCUIT POWER ANALYSIS 421**

- 11.1** Instantaneous Power 422  
**11.2** Average Power 424  
**11.3** Effective Values of Current and Voltage 433  
**11.4** Apparent Power and Power Factor 438  
**11.5** Complex Power 441  
**SUMMARY AND REVIEW** 447  
**READING FURTHER** 449  
**EXERCISES** 449

**CHAPTER 12****POLYPHASE CIRCUITS 457**

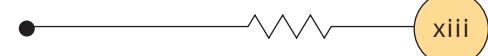
- 12.1** Polyphase Systems 458  
**12.2** Single-Phase Three-Wire Systems 460  
**12.3** Three-Phase Y-Y Connection 464  
**12.4** The Delta ( $\Delta$ ) Connection 470  
**12.5** Power Measurement in Three-Phase Systems 476  
**SUMMARY AND REVIEW** 484  
**READING FURTHER** 486  
**EXERCISES** 486

**CHAPTER 13****MAGNETICALLY COUPLED CIRCUITS 493**

- 13.1** Mutual Inductance 493  
**13.2** Energy Considerations 501  
**13.3** The Linear Transformer 505  
**13.4** The Ideal Transformer 512  
**SUMMARY AND REVIEW** 522  
**READING FURTHER** 523  
**EXERCISES** 523

**CHAPTER 14****COMPLEX FREQUENCY AND THE LAPLACE TRANSFORM 533**

- 14.1** Complex Frequency 533  
**14.2** The Damped Sinusoidal Forcing Function 537  
**14.3** Definition of the Laplace Transform 540  
**14.4** Laplace Transforms of Simple Time Functions 543  
**14.5** Inverse Transform Techniques 546  
**14.6** Basic Theorems for the Laplace Transform 553



- 14.7** The Initial-Value and Final-Value Theorems 561  
 SUMMARY AND REVIEW 564  
 READING FURTHER 565  
 EXERCISES 565

## CHAPTER 15

### CIRCUIT ANALYSIS IN THE s-DOMAIN 571

- 15.1**  $Z(s)$  and  $Y(s)$  571  
**15.2** Nodal and Mesh Analysis in the s-Domain 578  
**15.3** Additional Circuit Analysis Techniques 585  
**15.4** Poles, Zeros, and Transfer Functions 588  
**15.5** Convolution 589  
**15.6** The Complex-Frequency Plane 598  
**15.7** Natural Response and the s Plane 602  
**15.8** A Technique for Synthesizing the Voltage Ratio  $H(s) = V_{\text{out}}/V_{\text{in}}$  606  
 SUMMARY AND REVIEW 610  
 READING FURTHER 612  
 EXERCISES 612

## CHAPTER 16

### FREQUENCY RESPONSE 619

- 16.1** Parallel Resonance 619  
**16.2** Bandwidth and High- $Q$  Circuits 627  
**16.3** Series Resonance 633  
**16.4** Other Resonant Forms 637  
**16.5** Scaling 644  
**16.6** Bode Diagrams 648  
**16.7** Basic Filter Design 664  
**16.8** Advanced Filter Design 672  
 SUMMARY AND REVIEW 677  
 READING FURTHER 679  
 EXERCISES 679

## CHAPTER 17

### TWO-PORT NETWORKS 687

- 17.1** One-Port Networks 687  
**17.2** Admittance Parameters 692  
**17.3** Some Equivalent Networks 699  
**17.4** Impedance Parameters 708  
**17.5** Hybrid Parameters 713  
**17.6** Transmission Parameters 716  
 SUMMARY AND REVIEW 720  
 READING FURTHER 721  
 EXERCISES 722

## CHAPTER 18

### FOURIER CIRCUIT ANALYSIS 733

- 18.1** Trigonometric Form of the Fourier Series 733  
**18.2** The Use of Symmetry 743  
**18.3** Complete Response to Periodic Forcing Functions 748  
**18.4** Complex Form of the Fourier Series 750  
**18.5** Definition of the Fourier Transform 757  
**18.6** Some Properties of the Fourier Transform 761  
**18.7** Fourier Transform Pairs for Some Simple Time Functions 764  
**18.8** The Fourier Transform of a General Periodic Time Function 769  
**18.9** The System Function and Response in the Frequency Domain 770  
**18.10** The Physical Significance of the System Function 777  
 SUMMARY AND REVIEW 782  
 READING FURTHER 783  
 EXERCISES 783

## APPENDIX 1 AN INTRODUCTION TO NETWORK TOPOLOGY 791

## APPENDIX 2 SOLUTION OF SIMULTANEOUS EQUATIONS 803

## APPENDIX 3 A PROOF OF THÉVENIN'S THEOREM 811

## APPENDIX 4 A PSPICE® TUTORIAL 813

## APPENDIX 5 COMPLEX NUMBERS 817

## APPENDIX 6 A BRIEF MATLAB® TUTORIAL 827

## APPENDIX 7 ADDITIONAL LAPLACE TRANSFORM THEOREMS 833

## INDEX 839

*This page intentionally left blank*

The target audience colors everything about a book, being a major factor in decisions big and small, particularly both the pace and the overall writing style. Consequently it is important to note that the authors have made the conscious decision to write this book to the **student**, and not to the instructor. Our underlying philosophy is that reading the book should be enjoyable, despite the level of technical detail that it must incorporate. When we look back to the very first edition of *Engineering Circuit Analysis*, it's clear that it was developed specifically to be more of a conversation than a dry, dull discourse on a prescribed set of fundamental topics. To keep it conversational, we've had to work hard at updating the book so that it continues to speak to the increasingly diverse group of students using it all over the world.

Although in many engineering programs the introductory circuits course is preceded or accompanied by an introductory physics course in which electricity and magnetism are introduced (typically from a fields perspective), this is not required to use this book. After finishing the course, many students find themselves truly amazed that such a broad set of analytical tools have been derived from **only three simple scientific laws**—Ohm's law and Kirchhoff's voltage and current laws. The first six chapters assume only a familiarity with algebra and simultaneous equations; subsequent chapters assume a first course in calculus (derivatives and integrals) is being taken in tandem. Beyond that, we have tried to incorporate sufficient details to allow the book to be read on its own.

*So, what key features have been designed into this book with the student in mind?* First, individual chapters are organized into relatively short subsections, each having a single primary topic. The language has been updated to remain informal and to flow smoothly. Color is used to highlight important information as opposed to merely improve the aesthetics of the page layout, and white space is provided for jotting down short notes and questions. New terms are defined as they are introduced, and examples are placed strategically to demonstrate not only basic concepts, but problem-solving approaches as well. Practice problems relevant to the examples are placed in proximity so that students can try out the techniques for themselves before attempting the end-of-chapter exercises. The exercises represent a broad range of difficulties, generally ordered from simpler to more complex, and grouped according to the relevant section of each chapter. Answers to selected odd-numbered end-of-chapter exercises are posted on the book's website at [www.mhhe.com/haytdurbin8e](http://www.mhhe.com/haytdurbin8e).

Engineering is an intensive subject to study, and students often find themselves faced with deadlines and serious workloads. This does not mean that textbooks have to be dry and pompous, however, or that coursework should never contain any element of fun. In fact, successfully solving a problem often *is* fun, and learning how to do that can be fun as well. Determining how

to best accomplish this within the context of a textbook is an ongoing process. The authors have always relied on the often very candid feedback received from our own students at Purdue University; the California State University, Fullerton; Fort Lewis College in Durango, the joint engineering program at Florida A&M University and Florida State University, the University of Canterbury (New Zealand) and the University at Buffalo. We also rely on comments, corrections, and suggestions from instructors and students worldwide, and for this edition, consideration has been given to a new source of comments, namely, semianonymous postings on various websites.

The first edition of *Engineering Circuit Analysis* was written by Bill Hayt and Jack Kemmerly, two engineering professors who very much enjoyed teaching, interacting with their students, and training generations of future engineers. It was well received due to its compact structure, “to the point” informal writing style, and logical organization. There is no timidity when it comes to presenting the theory underlying a specific topic, or pulling punches when developing mathematical expressions. Everything, however, was carefully designed to assist students in their learning, present things in a straightforward fashion, and leave theory for theory’s sake to other books. They clearly put a great deal of thought into writing the book, and their enthusiasm for the subject comes across to the reader.

## KEY FEATURES OF THE EIGHTH EDITION

We have taken great care to retain key features from the seventh edition which were clearly working well. These include the general layout and sequence of chapters, the basic style of both the text and line drawings, the use of four-color printing where appropriate, numerous worked examples and related practice problems, and grouping of end-of-chapter exercises according to section. Transformers continue to merit their own chapter, and complex frequency is briefly introduced through a student-friendly extension of the phasor technique, instead of indirectly by merely stating the Laplace transform integral. We also have retained the use of icons, an idea first introduced in the sixth edition:



Provides a heads-up to common mistakes;



Indicates a point that's worth noting;



Denotes a design problem to which there is no unique answer;



Indicates a problem which requires computer-aided analysis.

The introduction of engineering-oriented analysis and design software in the book has been done with the mind-set that it should assist, not replace, the learning process. Consequently, the computer icon denotes problems that are typically phrased such that the software is used to *verify* answers, and not simply provide them. Both MATLAB® and PSpice® are used in this context.

## SPECIFIC CHANGES FOR THE EIGHTH EDITION

### INCLUDE:

- A new section in Chapter 16 on the analysis and design of multistage Butterworth filters
- Over 1000 new and revised end-of-chapter exercises
- A new overarching philosophy on end-of-chapter exercises, with each section containing problems similar to those solved in worked examples and practice problems, before proceeding to more complex problems to test the reader's skills
- Introduction of Chapter-Integrating Exercises at the end of each chapter. For the convenience of instructors and students, end-of-chapter exercises are grouped by section. To provide the opportunity for assigning exercises with less emphasis on an explicit solution method (for example, mesh or nodal analysis), as well as to give a broader perspective on key topics within each chapter, a select number of Chapter-Integrating Exercises appear at the end of each chapter.
- New photos, many in full color, to provide connection to the real world
- Updated screen captures and text descriptions of computer-aided analysis software
- New worked examples and practice problems
- Updates to the Practical Application feature, introduced to help students connect material in each chapter to broader concepts in engineering. Topics include distortion in amplifiers, modeling automotive suspension systems, practical aspects of grounding, the relationship of poles to stability, resistivity, and the memristor, sometimes called "the missing element"
- Streamlining of text, especially in the worked examples, to get to the point faster
- Answers to selected odd-numbered end-of-chapter exercises are posted on the book's website at [www.mhhe.com/haytdurbin8e](http://www.mhhe.com/haytdurbin8e).

I joined the book in 1999, and sadly never had the opportunity to speak to either Bill or Jack about the revision process, although I count myself lucky to have taken a circuits course from Bill Hayt while I was a student at Purdue. It is a distinct privilege to serve as a coauthor to *Engineering Circuit Analysis*, and in working on this book I give its fundamental philosophy and target audience the highest priority. I greatly appreciate the many people who have already provided feedback—both positive and negative—on aspects of previous editions, and welcome others to do so as well, either through the publishers (McGraw-Hill Higher Education) or to me directly ([durbin@ieee.org](mailto:durbin@ieee.org)).

Of course, this project has been a team effort, as is the case with every modern textbook. In particular I would like to thank Raghu Srinivasan (Global Publisher), Peter Massar (Sponsoring Editor), Curt Reynolds (Marketing Manager), Jane Mohr (Project Manager), Brittney-Corrigan-McElroy (Project Manager), Brenda Rolwes (Designer), Tammy Juran (Media Project Manager), and most importantly, Developmental Editor Darlene Schueller, who helped me with many, many details, issues, deadlines,



and questions. She is absolutely the best, and I'm very grateful for all the support from the team at McGraw-Hill. I would also like to thank various McGraw-Hill representatives, especially Nazier Hassan, who dropped by whenever on campus to just say hello and ask how things were going. Special thanks are also due to Catherine Shultz and Michael Hackett, former editors who continue to keep in contact. Cadence® and The MathWorks kindly provided assistance with software-aided analysis software, which was much appreciated. Several colleagues have generously supplied or helped with photographs and technical details, for which I'm very grateful: Prof. Masakazu Kobayashi of Waseda University; Dr. Wade Enright, Prof. Pat Bodger, Prof. Rick Millane, Mr. Gary Turner, and Prof. Richard Blaikie of the University of Canterbury; and Prof. Reginald Perry and Prof. Jim Zheng of Florida A&M University and the Florida State University. For the eighth edition, the following individuals deserve acknowledgment and a debt of gratitude for taking the time to review various versions of the manuscript:

Chong Koo An, *The University of Ulsan*  
Mark S. Andersland, *The University of Iowa*  
Marc Cahay, *University of Cincinnati*  
Claudio Canizares, *University of Waterloo*  
Teerapon Dachokiatawan, *King Mongkut's University of Technology North Bangkok*  
John Durkin, *The University of Akron*  
Lauren M. Fuentes, *Durham College*  
Lalit Goel, *Nanyang Technological University*  
Rudy Hofer, *Conestoga College ITAL*  
Mark Jerabek, *West Virginia University*  
Michael Kelley, *Cornell University*  
Hua Lee, *University of California, Santa Barbara*  
Georges Livanos, *Humber College Institute of Technology*  
Ahmad Nafisi, *Cal Poly State University*  
Arnost Neugroschel, *University of Florida*  
Pravin Patel, *Durham College*  
Jamie Phillips, *The University of Michigan*  
Daryl Reynolds, *West Virginia University*  
G.V.K.R. Sastry, *Andhra University*  
Michael Scordilis, *University of Miami*  
Yu Sun, *University of Toronto, Canada*  
Chanchana Tangwongsan, *Chulalongkorn University*  
Edward Wheeler, *Rose-Hulman Institute of Technology*  
Xiao-Bang Xu, *Clemson University*  
Tianyu Yang, *Embry-Riddle Aeronautical University*  
Zivan Zabar, *Polytechnic Institute of NYU*

I would also like to thank Susan Lord, *University of San Diego*, Archie L. Holmes, Jr., *University of Virginia*, Arnost Neugroschel, *University of Florida*, and Michael Scordilis, *University of Miami*, for their assistance in accuracy checking answers to selected end-of-chapter exercises.

Finally, I would like to briefly thank a number of other people who have contributed both directly and indirectly to the eighth edition. First and foremost, my wife, Kristi, and our son, Sean, for their patience, understanding, support, welcome distractions, and helpful advice. Throughout the day it has always been a pleasure to talk to friends and colleagues about what should be taught, how it should be taught, and how to measure learning. In particular, Martin Allen, Richard Blaikie, Alex Cartwright, Peter Cottrell, Wade Enright, Jeff Gray, Mike Hayes, Bill Kennedy, Susan Lord, Philippa Martin, Theresa Mayer, Chris McConville, Reginald Perry, Joan Redwing, Roger Reeves, Dick Schwartz, Leonard Tung, Jim Zheng, and many others have provided me with many useful insights, as has my father, Jesse Durbin, an electrical engineering graduate of the Indiana Institute of Technology.

**Steven M. Durbin**  
*Buffalo, New York*

## McGRAW-HILL DIGITAL OFFERINGS INCLUDE:

### McGraw-Hill Connect™ Engineering

McGraw-Hill Connect Engineering is a web-based assignment and assessment platform that gives students the means to better connect with their coursework, with their instructors, and with the important concepts that they will need to know for success now and in the future. With Connect Engineering, instructors can deliver assignments, quizzes, and tests easily online. Students can practice important skills at their own pace and on their own schedule. Ask your McGraw-Hill representative for more details and check it out at [www.mcgrawhillconnect.com/engineering](http://www.mcgrawhillconnect.com/engineering).

Instructor support materials are available from the book's website at [www.mhhe.com/haytdurbin8e](http://www.mhhe.com/haytdurbin8e). The materials include a password-protected solutions manual and image library. Instructors can also benefit from McGraw-Hill's Complete Online Solutions Manual Organization System (COSMOS). COSMOS enables instructors to generate a limitless supply of problem material for assignments, as well as transfer and integrate their own problems into the software. Contact your McGraw-Hill sales representative for additional information.



### McGraw-Hill Create™

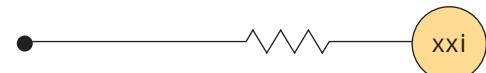
Craft your teaching resources to match the way you teach! With McGraw-Hill Create, [www.mcgrawhillcreate.com](http://www.mcgrawhillcreate.com), you can easily rearrange chapters, combine material from other content sources, and quickly upload content you have written such as your course syllabus or teaching notes. Find the content you need in Create by searching through thousands of leading McGraw-Hill textbooks. Arrange your book to fit your teaching style. Create even allows you to personalize your book's appearance by selecting the cover and adding your name, school, and course information. Order a Create book and you'll receive a complimentary print review copy in 3–5 business days or a complimentary electronic review copy (eComp) via email in minutes. Go to [www.mcgrawhillcreate.com](http://www.mcgrawhillcreate.com) today and register to experience how McGraw-Hill Create empowers you to teach *your* students *your* way.



Do More

### McGraw-Hill Higher Education and Blackboard® Have Teamed Up.

Blackboard, the web-based course management system, has partnered with McGraw-Hill to better allow students and faculty to use online materials and activities to complement face-to-face teaching. Blackboard features exciting social learning and teaching tools that foster more logical, visually



impactful, and active learning opportunities for students. You'll transform your closed-door classrooms into communities where students remain connected to their educational experience 24 hours a day.

This partnership allows you and your students access to McGraw-Hill's Connect™ and Create™ right from within your Blackboard course—all with one single sign-on.

Not only do you get single sign-on with Connect and Create, you also get deep integration of McGraw-Hill content and content engines right in Blackboard. Whether you're choosing a book for your course or building Connect assignments, all the tools you need are right where you want them—inside of Blackboard.

Gradebooks are now seamless. When a student completes an integrated Connect assignment, the grade for that assignment automatically (and instantly) feeds your Blackboard grade center.

McGraw-Hill and Blackboard can now offer you easy access to industry leading technology and content, whether your campus hosts it or we do. Be sure to ask your local McGraw-Hill representative for details.

## Electronic Textbook Option

This text is offered through CourseSmart for both instructors and students. CourseSmart is an online resource where students can purchase the complete text online at almost half the cost of a traditional text. Purchasing the eTextbook allows students to take advantage of CourseSmart's web tools for learning, which include full text search, notes and highlighting, and email tools for sharing notes between classmates. To learn more about CourseSmart options, contact your sales representative or visit [www.CourseSmart.com](http://www.CourseSmart.com).

*This page intentionally left blank*

# Introduction

## PREAMBLE

Although there are clear specialties within the field of engineering, all engineers share a considerable amount of common ground, particularly when it comes to problem solving. In fact, many practicing engineers find it is possible to work in a large variety of settings and even outside their traditional specialty, as their skill set is often transferrable to other environments. Today's engineering graduates are employed in a broad range of jobs, from design of individual components and systems, to assisting in solving socio-economic problems such as air and water pollution, urban planning, communication, mass transportation, power generation and distribution, and efficient use and conservation of natural resources.

Circuit analysis has long been a traditional introduction to **the art of problem solving** from an engineering perspective, even for those whose interests lie outside electrical engineering. There are many reasons for this, but one of the best is that in today's world it's extremely unlikely for any engineer to encounter a system that does not in some way include electrical circuitry. As circuits become smaller and require less power, and power sources become smaller and cheaper, embedded circuits are seemingly everywhere. Since most engineering situations require a team effort at some stage, having a working knowledge of circuit analysis therefore helps to provide everyone on a project with the background needed for effective communication.

Consequently, this book is not just about "circuit analysis" from an engineering perspective, but is also about developing basic problem-solving skills as they apply to situations an engineer is likely to encounter. As part of this, we also find that we're developing an intuitive understanding at a general level, and often we can

## KEY CONCEPTS

Linear versus Nonlinear Circuits

Four Main Categories of Circuit Analysis:

- DC Analysis
- Transient Analysis
- Sinusoidal Analysis
- Frequency Response

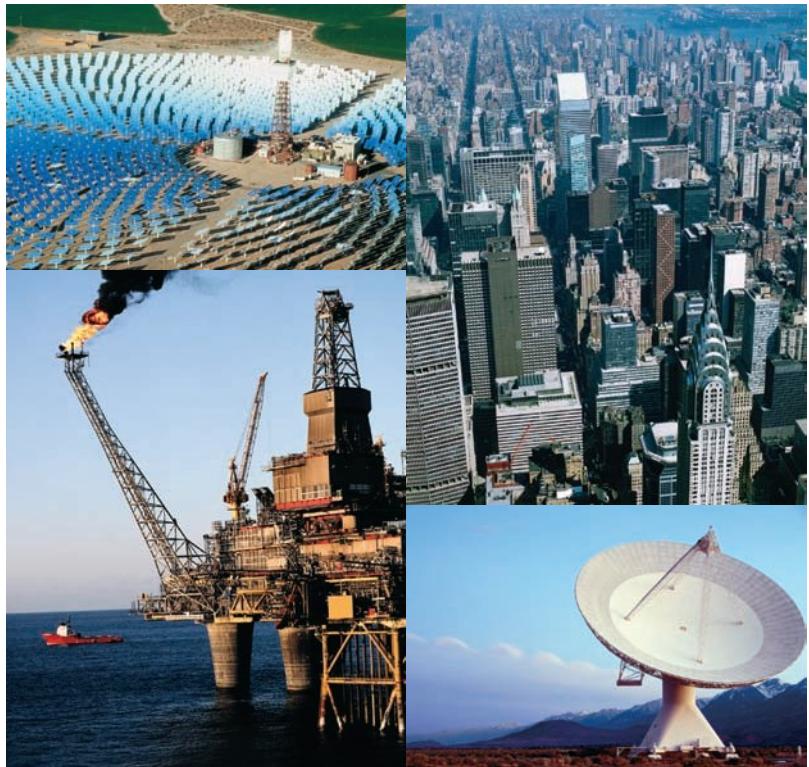
Circuit Analysis Beyond Circuits

Analysis and Design

Use of Engineering Software

A Problem-Solving Strategy





Not all electrical engineers routinely make use of circuit analysis, but they often bring to bear analytical and problem-solving skills learned early on in their careers. A circuit analysis course is one of the first exposures to such concepts. (*Solar Mirrors*: © Corbis; *Skyline*: © Getty Images/PhotoLink; *Oil Rig*: © Getty Images; *Dish*: © Getty Images/J. Luke/PhotoLink)

understand a complex system by its analogy to an electrical circuit. Before launching into all this, however, we'll begin with a quick preview of the topics found in the remainder of the book, pausing briefly to ponder the difference between analysis and design, and the evolving role computer tools play in modern engineering.

## 1.1 OVERVIEW OF TEXT

The fundamental subject of this text is *linear circuit analysis*, which sometimes prompts a few readers to ask,

“Is there ever any *nonlinear* circuit analysis?”

Sure! We encounter nonlinear circuits every day: they capture and decode signals for our TVs and radios, perform calculations millions of times a second inside microprocessors, convert speech into electrical signals for transmission over phone lines, and execute many other functions outside our field of view. In designing, testing, and implementing such nonlinear circuits, detailed analysis is unavoidable.

“Then why study *linear* circuit analysis?”

you might ask. An excellent question. The simple fact of the matter is that no physical system (including electrical circuits) is ever perfectly linear. Fortunately for us, however, a great many systems behave in a reasonably



Television sets include many nonlinear circuits. A great deal of them, however, can be understood and analyzed with the assistance of linear models. (© Sony Electronics, Inc.)

linear fashion over a limited range—allowing us to model them as linear systems if we keep the range limitations in mind.

For example, consider the common function

$$f(x) = e^x$$

A linear approximation to this function is

$$f(x) \approx 1 + x$$

Let's test this out. Table 1.1 shows both the exact value and the approximate value of  $f(x)$  for a range of  $x$ . Interestingly, the linear approximation is exceptionally accurate up to about  $x = 0.1$ , when the relative error is still less than 1%. Although many engineers are rather quick on a calculator, it's hard to argue that any approach is faster than just adding 1.

**TABLE 1.1 Comparison of a Linear Model for  $e^x$  to Exact Value**

<b>x</b>	<b>f(x)*</b>	<b>1 + x</b>	<b>Relative error**</b>
0.0001	1.0001	1.0001	0.0000005%
0.001	1.0010	1.001	0.00005%
0.01	1.0101	1.01	0.005%
0.1	1.1052	1.1	0.5%
1.0	2.7183	2.0	26%

\*Quoted to four significant figures.  
\*\*Relative error  $\triangleq \left| 100 \times \frac{e^x - (1 + x)}{e^x} \right|$

Linear problems are inherently more easily solved than their nonlinear counterparts. For this reason, we often seek reasonably accurate linear approximations (or *models*) to physical situations. Furthermore, the linear models are more easily manipulated and understood—making design a more straightforward process.

The circuits we will encounter in subsequent chapters all represent linear approximations to physical electric circuits. Where appropriate, brief discussions of potential inaccuracies or limitations to these models are provided, but generally speaking we find them to be suitably accurate for most applications. When greater accuracy is required in practice, nonlinear models are employed, but with a considerable increase in solution complexity. A detailed discussion of what constitutes a *linear electric circuit* can be found in Chap. 2.

Linear circuit analysis can be separated into four broad categories: (1) *dc analysis*, where the energy sources do not change with time; (2) *transient analysis*, where things often change quickly; (3) *sinusoidal analysis*, which applies to both ac power and signals; and (4) *frequency response*, which is the most general of the four categories, but typically assumes something is changing with time. We begin our journey with the topic of resistive circuits, which may include simple examples such as a flashlight or a toaster. This provides us with a perfect opportunity to learn a number of very powerful engineering circuit analysis techniques, such as *nodal analysis*, *mesh analysis*, *superposition*, *source transformation*, *Thévenin's theorem*, *Norton's*



Modern trains are powered by electric motors. Their electrical systems are best analyzed using ac or phasor analysis techniques. (Used with permission. Image copyright © 2010 M. Kobayashi. All rights reserved.)



Frequency-dependent circuits lie at the heart of many electronic devices, and they can be a great deal of fun to design. (© The McGraw-Hill Companies, Inc.)

*theorem*, and several methods for simplifying networks of components connected in series or parallel. The single most redeeming feature of resistive circuits is that the time dependence of any quantity of interest does not affect our analysis procedure. In other words, if asked for an electrical quantity of a resistive circuit at several specific instants in time, we do not need to analyze the circuit more than once. As a result, we will spend most of our effort early on considering only dc circuits—those circuits whose electrical parameters do not vary with time.

Although dc circuits such as flashlights or automotive rear window defoggers are undeniably important in everyday life, things are often much more interesting when something happens suddenly. In circuit analysis parlance, we refer to *transient analysis* as the suite of techniques used to study circuits which are suddenly energized or de-energized. To make such circuits interesting, we need to add elements that respond to the rate of change of electrical quantities, leading to circuit equations which include derivatives and integrals. Fortunately, we can obtain such equations using the simple techniques learned in the first part of our study.

Still, not all time-varying circuits are turned on and off suddenly. Air conditioners, fans, and fluorescent lights are only a few of the many examples we may see daily. In such situations, a calculus-based approach for every analysis can become tedious and time-consuming. Fortunately, there is a better alternative for situations where equipment has been allowed to run long enough for transient effects to die out, and this is commonly referred to as ac or sinusoidal analysis, or sometimes *phasor analysis*.

The final leg of our journey deals with a subject known as *frequency response*. Working directly with the differential equations obtained in time-domain analysis helps us develop an intuitive understanding of the operation of circuits containing energy storage elements (e.g., capacitors and inductors). As we shall see, however, circuits with even a relatively small number of components can be somewhat onerous to analyze, and so much more straightforward methods have been developed. These methods, which include Laplace and Fourier analysis, allow us to transform differential equations into algebraic equations. Such methods also enable us to design circuits to respond in specific ways to particular frequencies. We make use of frequency-dependent circuits every day when we dial a telephone, select our favorite radio station, or connect to the Internet.

## 1.2 RELATIONSHIP OF CIRCUIT ANALYSIS TO ENGINEERING

Whether we intend to pursue further circuit analysis at the completion of this course or not, it is worth noting that there are several layers to the concepts under study. Beyond the nuts and bolts of circuit analysis techniques lies the opportunity to develop a methodical approach to problem solving, the ability to determine the goal or goals of a particular problem, skill at collecting the information needed to effect a solution, and, perhaps equally importantly, opportunities for practice at verifying solution accuracy.

Students familiar with the study of other engineering topics such as fluid flow, automotive suspension systems, bridge design, supply chain management, or process control will recognize the general form of many of the



A molecular beam epitaxy crystal growth facility. The equations governing its operation closely resemble those used to describe simple linear circuits.

equations we develop to describe the behavior of various circuits. We simply need to learn how to “translate” the relevant variables (for example, replacing *voltage* with *force*, *charge* with *distance*, *resistance* with *friction coefficient*, etc.) to find that we already know how to work a new type of problem. Very often, if we have previous experience in solving a similar or related problem, our intuition can guide us through the solution of a totally new problem.

What we are about to learn regarding linear circuit analysis forms the basis for many subsequent electrical engineering courses. The study of electronics relies on the analysis of circuits with devices known as diodes and transistors, which are used to construct power supplies, amplifiers, and digital circuits. The skills which we will develop are typically applied in a rapid, methodical fashion by electronics engineers, who sometimes can analyze a complicated circuit without even reaching for a pencil! The time-domain and frequency-domain chapters of this text lead directly into discussions of signal processing, power transmission, control theory, and communications. We find that frequency-domain analysis in particular is an extremely powerful technique, easily applied to any physical system subjected to time-varying excitation, and particularly helpful in the design of filters.

### 1.3 ANALYSIS AND DESIGN

Engineers take a fundamental understanding of scientific principles, combine this with practical knowledge often expressed in mathematical terms, and (frequently with considerable creativity) arrive at a solution to a given problem. **Analysis** is the process through which we determine the scope of a problem, obtain the information required to understand it, and compute the parameters of interest. **Design** is the process by which we synthesize something new as part of the solution to a problem. Generally speaking, there is an expectation that a problem requiring design will have no unique solution, whereas the analysis phase typically will. Thus, the last step in designing is always analyzing the result to see if it meets specifications.



An example of a robotic manipulator. The feedback control system can be modeled using linear circuit elements to determine situations in which the operation may become unstable. (NASA Marshall Space Flight Center.)



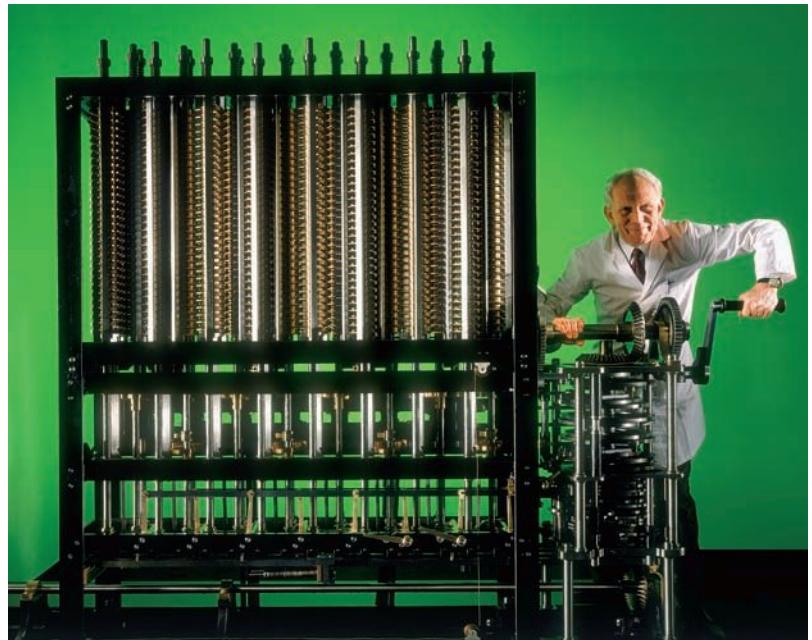
Two proposed designs for a next-generation space shuttle. Although both contain similar elements, each is unique.  
*(NASA Dryden Flight Research Center.)*

This text is focused on developing our ability to analyze and solve problems because it is the starting point in every engineering situation. The philosophy of this book is that we need clear explanations, well-placed examples, and plenty of practice to develop such an ability. Therefore, elements of design are integrated into end-of-chapter problems and later chapters so as to be enjoyable rather than distracting.

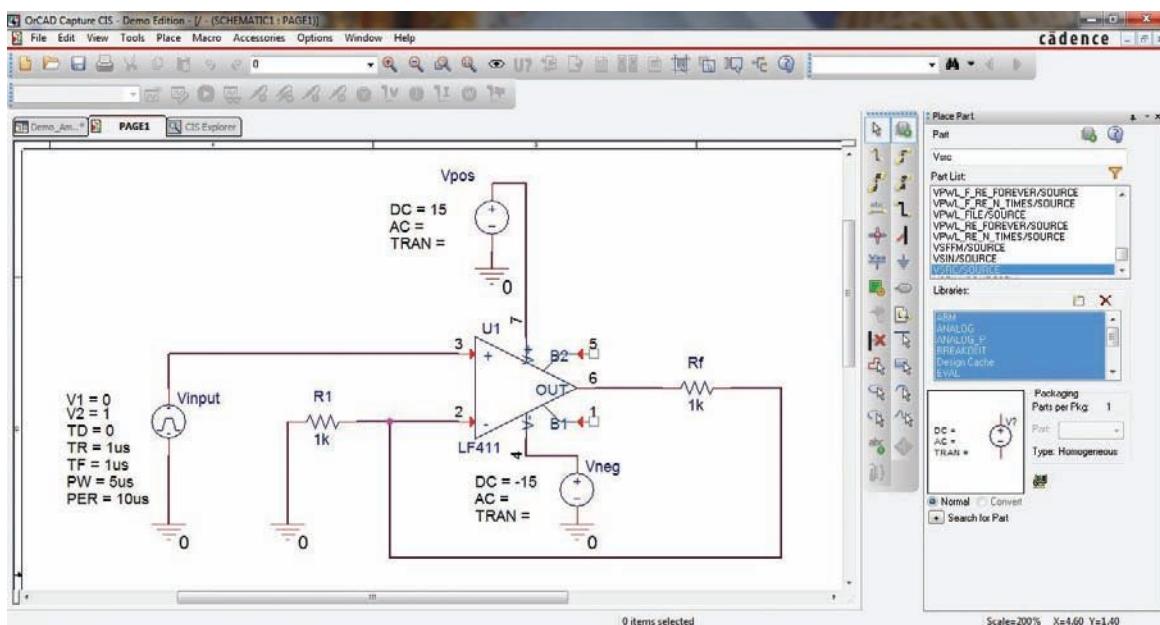
## 1.4 COMPUTER-AIDED ANALYSIS

Solving the types of equations that result from circuit analysis can often become notably cumbersome for even moderately complex circuits. This of course introduces an increased probability that errors will be made, in addition to considerable time in performing the calculations. The desire to find a tool to help with this process actually predates electronic computers, with purely mechanical computers such as the Analytical Engine designed by Charles Babbage in the 1880s proposed as possible solutions. Perhaps the earliest successful electronic computer designed for solution of differential equations was the 1940s-era ENIAC, whose vacuum tubes filled a large room. With the advent of low-cost desktop computers, however, computer-aided circuit analysis has developed into an invaluable everyday tool which has become an integral part of not only analysis but design as well.

One of the most powerful aspects of computer-aided design is the relatively recent integration of multiple programs in a fashion transparent to the user. This allows the circuit to be drawn schematically on the screen, reduced automatically to the format required by an analysis program (such as SPICE, introduced in Chap. 4), and the resulting output smoothly transferred to a third program capable of plotting various electrical quantities of



Charles Babbage's "Difference Engine Number 2," as completed by the Science Museum (London) in 1991.  
*(© Science Museum/Science & Society Picture Library.)*



An amplifier circuit drawn using a commercial schematic capture software package.

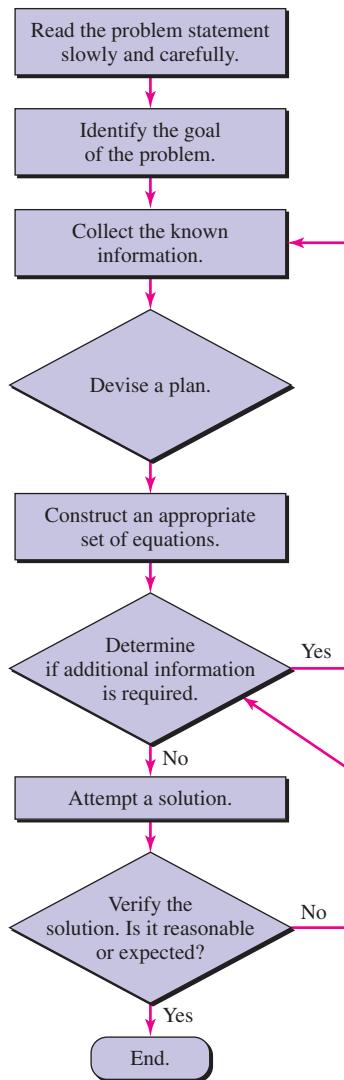
interest that describe the operation of the circuit. Once the engineer is satisfied with the simulated performance of the design, the same software can generate the printed circuit board layout using geometrical parameters in the components library. This level of integration is continually increasing, to the point where soon an engineer will be able to draw a schematic, click a few buttons, and walk to the other side of the table to pick up a manufactured version of the circuit, ready to test!

The reader should be wary, however, of one thing. Circuit analysis software, although fun to use, is by no means a replacement for good old-fashioned paper-and-pencil analysis. We need to have a solid understanding of how circuits work in order to develop an ability to design them. Simply going through the motions of running a particular software package is a little like playing the lottery: with user-generated entry errors, hidden default parameters in the myriad of menu choices, and the occasional shortcoming of human-written code, there is no substitute for having at least an approximate idea of the expected behavior of a circuit. Then, if the simulation result does not agree with expectations, we can find the error early, rather than after it's too late.

Still, computer-aided analysis is a powerful tool. It allows us to vary parameter values and evaluate the change in circuit performance, and to consider several variations to a design in a straightforward manner. The result is a reduction of repetitive tasks, and more time to concentrate on engineering details.

## 1.5 SUCCESSFUL PROBLEM-SOLVING STRATEGIES

As the reader might have picked up, this book is just as much about problem solving as it is about circuit analysis. As a result, the expectation is that during your time as an engineering student, you are learning how to solve problems—so just at this moment, those skills are not yet fully developed. As you proceed



through your course of study, you will pick up techniques that work for you, and likely continue to do so as a practicing engineer. At this stage, then, we should spend a few moments discussing some basic points.

The first point is that by far, the most common difficulty encountered by engineering students is *not knowing how to start* a problem. This improves with experience, but early on that's of no help. The best advice we can give is to adopt a methodical approach, beginning with reading the problem statement slowly and carefully (and more than once, if needed). Since experience usually gives us some type of insight into how to deal with a specific problem, worked examples appear throughout the book. Rather than just read them, however, it might be helpful to work through them with a pencil and a piece of paper.

Once we've read through the problem, and feel we might have some useful experience, the next step is to identify the goal of the problem—perhaps to calculate a voltage or a power, or to select a component value. Knowing where we're going is a big help. The next step is to collect as much information as we can, and to organize it somehow.

At this point *we're still not ready to reach for the calculator*. It's best first to devise a plan, perhaps based on experience, perhaps based simply on our intuition. Sometimes plans work, and sometimes they don't. Starting with our initial plan, it's time to construct an initial set of equations. If they appear complete, we can solve them. If not, we need to either locate more information, modify our plan, or both.

Once we have what appears to be a working solution, we should not stop, even if exhausted and ready for a break. **No engineering problem is solved unless the solution is tested somehow.** We might do this by performing a computer simulation, or solving the problem a different way, or perhaps even just estimating what answer might be reasonable.

Since not everyone likes to read to learn, these steps are summarized in the adjacent flowchart. This is just one particular problem-solving strategy, and the reader of course should feel free to modify it as necessary. The real key, however, is to try and learn in a relaxed, low-stress environment free of distractions. Experience is the best teacher, and learning from our own mistakes will always be part of the process of becoming a skilled engineer.

## READING FURTHER

This relatively inexpensive, best-selling book teaches the reader how to develop winning strategies in the face of seemingly impossible problems:

G. Polya, *How to Solve It*. Princeton, N.J.: Princeton University Press, 1971.

# Basic Components and Electric Circuits

## INTRODUCTION

In conducting circuit analysis, we often find ourselves seeking specific *currents*, *voltages*, or *powers*, so here we begin with a brief description of these quantities. In terms of components that can be used to build electrical circuits, we have quite a few from which to choose. We initially focus on the *resistor*, a simple passive component, and a range of idealized active sources of voltage and current. As we move forward, new components will be added to the inventory to allow more complex (and useful) circuits to be considered.

A quick word of advice before we begin: Pay close attention to the role of “+” and “−” signs when labeling voltages, and the significance of the arrow in defining current; they often make the difference between wrong and right answers.

### 2.1 UNITS AND SCALES

In order to state the value of some measurable quantity, we must give both a *number* and a *unit*, such as “3 meters.” Fortunately, we all use the same number system. This is not true for units, and a little time must be spent in becoming familiar with a suitable system. We must agree on a standard unit and be assured of its permanence and its general acceptability. The standard unit of length, for example, should not be defined in terms of the distance between two marks on a certain rubber band; this is not permanent, and furthermore everybody else is using another standard.

The most frequently used system of units is the one adopted by the National Bureau of Standards in 1964; it is used by all major professional engineering societies and is the language in which today’s textbooks are written. This is the International System of Units (abbreviated *SI* in all languages), adopted by the General

### KEY CONCEPTS

Basic Electrical Quantities and Associated Units:  
Charge, Current, Voltage, and Power

Current Direction and Voltage Polarity

The Passive Sign Convention for Calculating Power

Ideal Voltage and Current Sources

Dependent Sources

Resistance and Ohm’s Law



Conference on Weights and Measures in 1960. Modified several times since, the SI is built upon seven basic units: the *meter*, *kilogram*, *second*, *ampere*, *kelvin*, *mole*, and *candela* (see Table 2.1). This is a “metric system,” some form of which is now in common use in most countries of the world, although it is not yet widely used in the United States. Units for other quantities such as volume, force, energy, etc., are derived from these seven base units.

**TABLE 2.1** SI Base Units

Base Quantity	Name	Symbol
length	meter	m
mass	kilogram	kg
time	second	s
electric current	ampere	A
thermodynamic temperature	kelvin	K
amount of substance	mole	mol
luminous intensity	candela	cd

There is some inconsistency regarding whether units named after a person should be capitalized. Here, we will adopt the most contemporary convention,<sup>1,2</sup> where such units are written out in lowercase (e.g., watt, joule), but abbreviated with an uppercase symbol (e.g., W, J).

(1) H. Barrell, *Nature* 220, 1968, p. 651.

(2) V. N. Krutikov, T. K. Kanishcheva, S. A. Kononogov, L. K. Isaev, and N. I. Khanov, *Measurement Techniques* 51, 2008, p. 1045.

The “calorie” used with food, drink, and exercise is really a kilocalorie, 4.187 J.

The fundamental unit of work or energy is the *joule* (J). One joule (a  $\text{kg m}^2 \text{s}^{-2}$  in SI base units) is equivalent to 0.7376 foot pound-force (ft · lbf). Other energy units include the calorie (cal), equal to 4.187 J; the British thermal unit (Btu), which is 1055 J; and the kilowatthour (kWh), equal to  $3.6 \times 10^6$  J. Power is defined as the *rate* at which work is done or energy is expended. The fundamental unit of power is the *watt* (W), defined as 1 J/s. One watt is equivalent to 0.7376 ft · lbf/s or, equivalently, 1/745.7 horsepower (hp).

The SI uses the decimal system to relate larger and smaller units to the basic unit, and employs prefixes to signify the various powers of 10. A list of prefixes and their symbols is given in Table 2.2; the ones most commonly encountered in engineering are highlighted.

**TABLE 2.2** SI Prefixes

Factor	Name	Symbol	Factor	Name	Symbol
$10^{-24}$	yocto	y	$10^{24}$	yotta	Y
$10^{-21}$	zepto	z	$10^{21}$	zetta	Z
$10^{-18}$	atto	a	$10^{18}$	exa	E
$10^{-15}$	femto	f	$10^{15}$	peta	P
$10^{-12}$	pico	p	$10^{12}$	tera	T
$10^{-9}$	nano	n	$10^9$	giga	G
$10^{-6}$	micro	$\mu$	$10^6$	mega	M
$10^{-3}$	milli	m	$10^3$	kilo	k
$10^{-2}$	centi	c	$10^2$	hecto	h
$10^{-1}$	deci	d	$10^1$	deka	da

These prefixes are worth memorizing, for they will appear often both in this text and in other technical work. Combinations of several prefixes, such as the millimicrosecond, are unacceptable. It is worth noting that in terms of distance, it is common to see “micron ( $\mu\text{m}$ )” as opposed to “micrometer,” and often the angstrom ( $\text{\AA}$ ) is used for  $10^{-10}$  meter. Also, in circuit analysis and engineering in general, it is fairly common to see numbers expressed in what are frequently termed “engineering units.” In engineering notation, a quantity is represented by a number between 1 and 999 and an appropriate metric unit using a power divisible by 3. So, for example, it is preferable to express the quantity 0.048 W as 48 mW, instead of 4.8 cW,  $4.8 \times 10^{-2}$  W, or 48,000  $\mu\text{W}$ .

### PRACTICE

2.1 A krypton fluoride laser emits light at a wavelength of 248 nm.

This is the same as: (a) 0.0248 mm; (b) 2.48  $\mu\text{m}$ ; (c) 0.248  $\mu\text{m}$ ; (d) 24,800  $\text{\AA}$ .

2.2 A single logic gate in a prototype integrated circuit is found to be capable of switching from the “on” state to the “off” state in 12 ps. This corresponds to: (a) 1.2 ns; (b) 120 ns; (c) 1200 ns; (d) 12,000 ns.

2.3 A typical incandescent reading lamp runs at 60 W. If it is left on constantly, how much energy (J) is consumed per day, and what is the weekly cost if energy is charged at a rate of 12.5 cents per kilowatthour?

Ans: 2.1 (c); 2.2 (d); 2.3 5.18 MJ, \$1.26.

## 2.2 CHARGE, CURRENT, VOLTAGE, AND POWER

### Charge

One of the most fundamental concepts in electric circuit analysis is that of charge conservation. We know from basic physics that there are two types of charge: positive (corresponding to a proton) and negative (corresponding to an electron). For the most part, this text is concerned with circuits in which only electron flow is relevant. There are many devices (such as batteries, diodes, and transistors) in which positive charge motion is important to understanding internal operation, but external to the device we typically concentrate on the electrons which flow through the connecting wires. Although we continuously transfer charges between different parts of a circuit, we do nothing to change the total amount of charge. In other words, we neither create nor destroy electrons (or protons) when running electric circuits.<sup>1</sup> Charge in motion represents a *current*.

In the SI system, the fundamental unit of charge is the **coulomb** (C). It is defined in terms of the **ampere** by counting the total charge that passes through an arbitrary cross section of a wire during an interval of one second; one coulomb is measured each second for a wire carrying a current of 1 ampere (Fig. 2.1). In this system of units, a single electron has a charge of  $-1.602 \times 10^{-19}$  C and a single proton has a charge of  $+1.602 \times 10^{-19}$  C.

As seen in Table 2.1, the base units of the SI are not derived from fundamental physical quantities. Instead, they represent historically agreed upon measurements, leading to definitions which occasionally seem backward. For example, it would make more sense physically to define the ampere based on electronic charge.

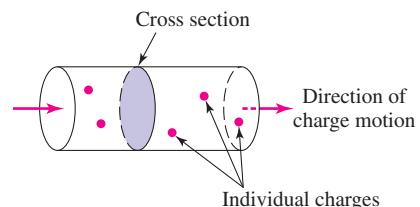


FIGURE 2.1 The definition of current illustrated using current flowing through a wire; 1 ampere corresponds to 1 coulomb of charge passing through the arbitrarily chosen cross section in 1 second.

(1) Although the occasional appearance of smoke may seem to suggest otherwise...

A quantity of charge that does not change with time is typically represented by  $Q$ . The instantaneous amount of charge (which may or may not be time-invariant) is commonly represented by  $q(t)$ , or simply  $q$ . This convention is used throughout the remainder of the text: capital letters are reserved for constant (time-invariant) quantities, whereas lowercase letters represent the more general case. Thus, a constant charge may be represented by either  $Q$  or  $q$ , but an amount of charge that changes over time *must* be represented by the lowercase letter  $q$ .

## Current

The idea of “transfer of charge” or “charge in motion” is of vital importance to us in studying electric circuits because, in moving a charge from place to place, we may also transfer energy from one point to another. The familiar cross-country power-transmission line is a practical example of a device that transfers energy. Of equal importance is the possibility of varying the rate at which the charge is transferred in order to communicate or transfer information. This process is the basis of communication systems such as radio, television, and telemetry.

The current present in a discrete path, such as a metallic wire, has both a *numerical value* and a *direction* associated with it; it is a measure of the rate at which charge is moving past a given reference point in a specified direction.

Once we have specified a reference direction, we may then let  $q(t)$  be the total charge that has passed the reference point since an arbitrary time  $t = 0$ , moving in the defined direction. A contribution to this total charge will be negative if negative charge is moving in the reference direction, or if positive charge is moving in the opposite direction. As an example, Fig. 2.2 shows a history of the total charge  $q(t)$  that has passed a given reference point in a wire (such as the one shown in Fig. 2.1).

We define the current at a specific point and flowing in a specified direction as the instantaneous rate at which net positive charge is moving past that point in the specified direction. This, unfortunately, is the historical definition, which came into popular use before it was appreciated that current in wires is actually due to negative, not positive, charge motion. Current is symbolized by  $I$  or  $i$ , and so

$$i = \frac{dq}{dt} \quad [1]$$

The unit of current is the ampere (A), named after A. M. Ampère, a French physicist. It is commonly abbreviated as an “amp,” although this is unofficial and somewhat informal. One ampere equals 1 coulomb per second.

Using Eq. [1], we compute the instantaneous current and obtain Fig. 2.3. The use of the lowercase letter  $i$  is again to be associated with an instantaneous value; an uppercase  $I$  would denote a constant (i.e., time-invariant) quantity.

The charge transferred between time  $t_0$  and  $t$  may be expressed as a definite integral:

$$\int_{q(t_0)}^{q(t)} dq = \int_{t_0}^t i dt'$$

The total charge transferred over all time is thus given by

$$q(t) = \int_{t_0}^t i dt' + q(t_0) \quad [2]$$

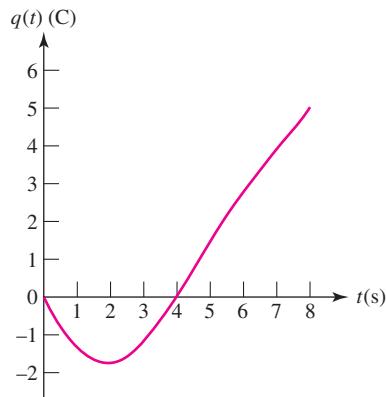


FIGURE 2.2 A graph of the instantaneous value of the total charge  $q(t)$  that has passed a given reference point since  $t = 0$ .

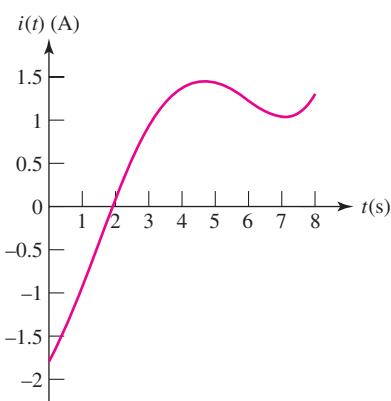


FIGURE 2.3 The instantaneous current  $i = dq/dt$ , where  $q$  is given in Fig. 2.2.

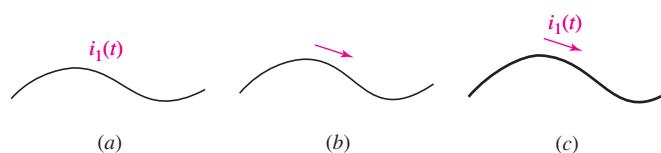
## SECTION 2.2 CHARGE, CURRENT, VOLTAGE, AND POWER

Several different types of current are illustrated in Fig. 2.4. A current that is constant in time is termed a direct current, or simply dc, and is shown by Fig. 2.4a. We will find many practical examples of currents that vary sinusoidally with time (Fig. 2.4b); currents of this form are present in normal household circuits. Such a current is often referred to as alternating current, or ac. Exponential currents and damped sinusoidal currents (Fig. 2.4c and d) will also be encountered later.

We create a graphical symbol for current by placing an arrow next to the conductor. Thus, in Fig. 2.5a the direction of the arrow and the value 3 A indicate either that a net positive charge of 3 C/s is moving to the right or that a net negative charge of  $-3$  C/s is moving to the left each second. In Fig. 2.5b there are again two possibilities: either  $-3$  A is flowing to the left or  $+3$  A is flowing to the right. All four statements and both figures represent currents that are equivalent in their electrical effects, and we say that they are equal. A nonelectrical analogy that may be easier to visualize is to think in terms of a personal savings account: e.g., a deposit can be viewed as either a *negative* cash flow *out of* your account or a *positive* flow *into* your account.

It is convenient to think of current as the motion of positive charge, even though it is known that current flow in metallic conductors results from electron motion. In ionized gases, in electrolytic solutions, and in some semiconductor materials, however, positive charges in motion constitute part or all of the current. Thus, any definition of current can agree with the physical nature of conduction only part of the time. The definition and symbolism we have adopted are standard.

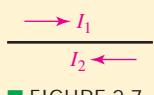
It is essential that we realize that the current arrow does not indicate the “actual” direction of current flow but is simply part of a convention that allows us to talk about “the current in the wire” in an unambiguous manner. The arrow is a fundamental part of the definition of a current! Thus, to talk about the value of a current  $i_1(t)$  without specifying the arrow is to discuss an undefined entity. For example, Fig. 2.6a and b are meaningless representations of  $i_1(t)$ , whereas Fig. 2.6c is complete.



■ FIGURE 2.6 (a, b) Incomplete, improper, and incorrect definitions of a current.  
(c) The correct definition of  $i_1(t)$ .

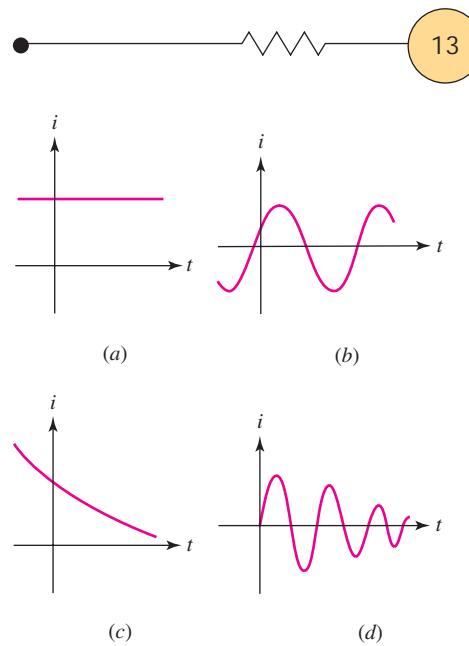
**PRACTICE**

- 2.4 In the wire of Fig. 2.7, electrons are moving *left to right* to create a current of 1 mA. Determine  $I_1$  and  $I_2$ .



■ FIGURE 2.7

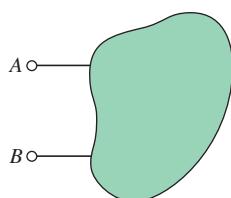
Ans:  $I_1 = -1$  mA;  $I_2 = +1$  mA.



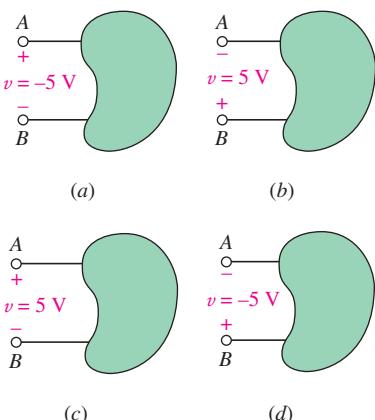
■ FIGURE 2.4 Several types of current: (a) Direct current (dc). (b) Sinusoidal current (ac). (c) Exponential current. (d) Damped sinusoidal current.



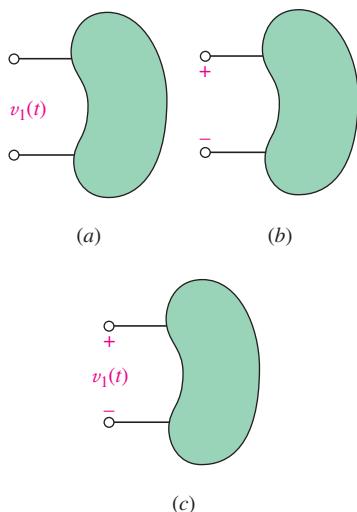
■ FIGURE 2.5 Two methods of representation for the exact same current.



■ FIGURE 2.8 A general two-terminal circuit element.



■ FIGURE 2.9 (a,b) Terminal B is 5 V positive with respect to terminal A; (c,d) terminal A is 5 V positive with respect to terminal B.



■ FIGURE 2.10 (a,b) These are inadequate definitions of a voltage. (c) A correct definition includes both a symbol for the variable and a plus-minus symbol pair.

## Voltage

We must now begin to refer to a circuit element, something best defined in general terms to begin with. Such electrical devices as fuses, light bulbs, resistors, batteries, capacitors, generators, and spark coils can be represented by combinations of simple circuit elements. We begin by showing a very general circuit element as a shapeless object possessing two terminals at which connections to other elements may be made (Fig. 2.8).

There are two paths by which current may enter or leave the element. In subsequent discussions we will define particular circuit elements by describing the electrical characteristics that may be observed at their terminals.

In Fig. 2.8, let us suppose that a dc current is sent into terminal A, through the general element, and back out of terminal B. Let us also assume that pushing charge through the element requires an expenditure of energy. We then say that an electrical voltage (or a *potential difference*) exists between the two terminals, or that there is a voltage “across” the element. Thus, the voltage across a terminal pair is a measure of the work required to move charge through the element. The unit of voltage is the volt,<sup>2</sup> and 1 volt is the same as 1 J/C. Voltage is represented by V or  $v$ .

A voltage can exist between a pair of electrical terminals whether a current is flowing or not. An automobile battery, for example, has a voltage of 12 V across its terminals even if nothing whatsoever is connected to the terminals.

According to the principle of conservation of energy, the energy that is expended in forcing charge through the element must appear somewhere else. When we later meet specific circuit elements, we will note whether that energy is stored in some form that is readily available as electric energy or whether it changes irreversibly into heat, acoustic energy, or some other nonelectrical form.

We must now establish a convention by which we can distinguish between energy supplied *to* an element and energy that is supplied *by* the element itself. We do this by our choice of sign for the voltage of terminal A with respect to terminal B. If a positive current is entering terminal A of the element and an external source must expend energy to establish this current, then terminal A is positive with respect to terminal B. (Alternatively, we may say that terminal B is negative with respect to terminal A.)

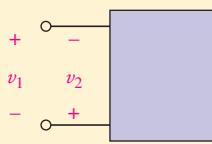
The sense of the voltage is indicated by a plus-minus pair of algebraic signs. In Fig. 2.9a, for example, the placement of the + sign at terminal A indicates that terminal A is  $v$  volts positive with respect to terminal B. If we later find that  $v$  happens to have a numerical value of  $-5$  V, then we may say either that A is  $-5$  V positive with respect to B or that B is 5 V positive with respect to A. Other cases are shown in Fig. 2.9b, c, and d.

Just as we noted in our definition of current, it is essential to realize that the plus-minus pair of algebraic signs does not indicate the “actual” polarity of the voltage but is simply part of a convention that enables us to talk unambiguously about “the voltage across the terminal pair.” *The definition of any voltage must include a plus-minus sign pair!* Using a quantity  $v_1(t)$  without specifying the location of the plus-minus sign pair is using an undefined term. Figure 2.10a and b do not serve as definitions of  $v_1(t)$ ; Fig. 2.10c does.

(2) We are probably fortunate that the full name of the 18th century Italian physicist, Alessandro Giuseppe Antonio Anastasio Volta, is not used for our unit of potential difference!

**PRACTICE**

2.5 For the element in Fig. 2.11,  $v_1 = 17$  V. Determine  $v_2$ .



■ FIGURE 2.11

Ans:  $v_2 = -17$  V.

**Power**

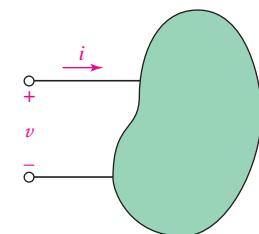
We have already defined power, and we will represent it by  $P$  or  $p$ . If one joule of energy is expended in transferring one coulomb of charge through the device in one second, then the rate of energy transfer is one watt. The absorbed power must be proportional both to the number of coulombs transferred per second (current) and to the energy needed to transfer one coulomb through the element (voltage). Thus,

$$p = vi \quad [3]$$

Dimensionally, the right side of this equation is the product of joules per coulomb and coulombs per second, which produces the expected dimension of joules per second, or watts. The conventions for current, voltage, and power are shown in Fig. 2.12.

We now have an expression for the power being absorbed by a circuit element in terms of a voltage across it and current through it. Voltage was defined in terms of an energy expenditure, and power is the rate at which energy is expended. However, no statement can be made concerning energy transfer in any of the four cases shown in Fig. 2.9, for example, until the direction of the current is specified. Let us imagine that a current arrow is placed alongside each upper lead, directed to the right, and labeled “+2 A.” First, consider the case shown in Fig. 2.9c. Terminal A is 5 V positive with respect to terminal B, which means that 5 J of energy is required to move each coulomb of positive charge into terminal A, through the object, and out terminal B. Since we are injecting +2 A (a current of 2 coulombs of positive charge per second) into terminal A, we are doing  $(5 \text{ J/C}) \times (2 \text{ C/s}) = 10 \text{ J}$  of work per second on the object. In other words, the object is absorbing 10 W of power from whatever is injecting the current.

We know from an earlier discussion that there is no difference between Fig. 2.9c and Fig. 2.9d, so we expect the object depicted in Fig. 2.9d to also be absorbing 10 W. We can check this easily enough: we are injecting +2 A into terminal A of the object, so +2 A flows out of terminal B. Another way of saying this is that we are injecting -2 A of current into terminal B. It takes -5 J/C to move charge from terminal B to terminal A, so the object is absorbing  $(-5 \text{ J/C}) \times (-2 \text{ C/s}) = +10 \text{ W}$  as expected. The only difficulty in describing this particular case is keeping the minus signs straight, but with a bit of care we see the correct answer can be obtained regardless of our choice of positive reference terminal (terminal A in Fig. 2.9c, and terminal B in Fig. 2.9d).



■ FIGURE 2.12 The power absorbed by the element is given by the product  $p = vi$ . Alternatively, we can say that the element generates or supplies a power  $-vi$ .

Now let's look at the situation depicted in Fig. 2.9a, again with  $+2\text{ A}$  injected into terminal  $A$ . Since it takes  $-5\text{ J/C}$  to move charge from terminal  $A$  to terminal  $B$ , the object is absorbing  $(-5\text{ J/C}) \times (2\text{ C/s}) = -10\text{ W}$ . What does this mean? How can anything absorb ***negative*** power? If we think about this in terms of energy transfer,  $-10\text{ J}$  is transferred to the object each second through the  $2\text{ A}$  current flowing into terminal  $A$ . The object is actually losing energy—at a rate of  $10\text{ J/s}$ . In other words, it is supplying  $10\text{ J/s}$  (i.e.,  $10\text{ W}$ ) to some other object not shown in the figure. Negative *absorbed* power, then, is equivalent to positive *supplied* power.

Let's recap. Figure 2.12 shows that if one terminal of the element is  $v$  volts positive with respect to the other terminal, and if a current  $i$  is entering the element through that terminal, then a power  $p = vi$  is being *absorbed* by the element; it is also correct to say that a power  $p = vi$  is being *delivered* to the element. When the current arrow is directed into the element at the plus-marked terminal, we satisfy the ***passive sign convention***. This convention should be studied carefully, understood, and memorized. In other words, it says that if the current arrow and the voltage polarity signs are placed such that the current enters that end of the element marked with the positive sign, then the power *absorbed* by the element can be expressed by the product of the specified current and voltage variables. If the numerical value of the product is negative, then we say that the element is absorbing negative power, or that it is actually generating power and delivering it to some external element. For example, in Fig. 2.12 with  $v = 5\text{ V}$  and  $i = -4\text{ A}$ , the element may be described as either absorbing  $-20\text{ W}$  or generating  $20\text{ W}$ .

Conventions are only required when there is more than one way to do something, and confusion may result when two different groups try to communicate. For example, it is rather arbitrary to always place "North" at the top of a map; compass needles don't point "up," anyway. Still, if we were talking to people who had secretly chosen the opposite convention of placing "South" at the top of their maps, imagine the confusion that could result! In the same fashion, there is a general convention that always draws the current arrows pointing into the positive voltage terminal, regardless of whether the element supplies or absorbs power. This convention is not incorrect but sometimes results in counterintuitive currents labeled on circuit schematics. The reason for this is that it simply seems more natural to refer to positive current flowing out of a voltage or current source that is supplying positive power to one or more circuit elements.

## EXAMPLE 2.1

Compute the power absorbed by each part in Fig. 2.13.

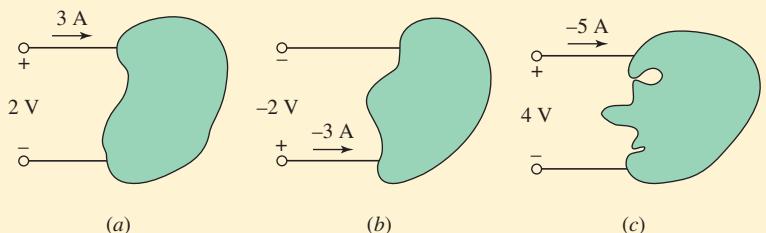


FIGURE 2.13 (a, b, c) Three examples of two-terminal elements.

In Fig. 2.13a, we see that the reference current is defined consistent with the passive sign convention, which assumes that the element is absorbing power. With +3 A flowing into the positive reference terminal, we compute

$$P = (2 \text{ V})(3 \text{ A}) = 6 \text{ W}$$

of power absorbed by the element.

Figure 2.13b shows a slightly different picture. Now, we have a current of -3 A flowing into the positive reference terminal. This gives us an absorbed power

$$P = (-2 \text{ V})(-3 \text{ A}) = 6 \text{ W}$$

Thus, we see that the two cases are actually equivalent: A current of +3 A flowing into the top terminal is the same as a current of +3 A flowing out of the bottom terminal, or, equivalently, a current of -3 A flowing into the bottom terminal.

Referring to Fig. 2.13c, we again apply the passive sign convention rules and compute an absorbed power

$$P = (4 \text{ V})(-5 \text{ A}) = -20 \text{ W}$$

Since we computed a negative *absorbed* power, this tells us that the element in Fig. 2.13c is actually *supplying* +20 W (i.e., it's a source of energy).

### PRACTICE

2.6 Determine the power being absorbed by the circuit element in Fig. 2.14a.

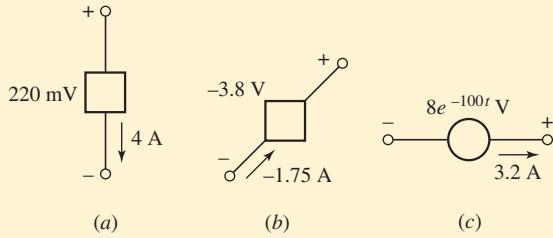


FIGURE 2.14

2.7 Determine the power being generated by the circuit element in Fig. 2.14b.

2.8 Determine the power being delivered to the circuit element in Fig. 2.14c at  $t = 5 \text{ ms}$ .

Ans: 880 mW; 6.65 W; -15.53 W.

## 2.3 VOLTAGE AND CURRENT SOURCES

Using the concepts of current and voltage, it is now possible to be more specific in defining a *circuit element*.

In so doing, it is important to differentiate between the physical device itself and the mathematical model which we will use to analyze its behavior in a circuit. The model is only an approximation.

By definition, a simple circuit element is the mathematical model of a two-terminal electrical device, and it can be completely characterized by its voltage-current relationship; it cannot be subdivided into other two-terminal devices.

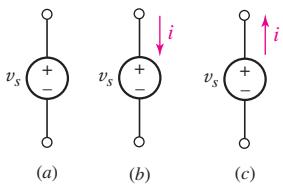


FIGURE 2.15 Circuit symbol of the independent voltage source.

If you've ever noticed the room lights dim when an air conditioner kicks on, it's because the sudden large current demand temporarily led to a voltage drop. After the motor starts moving, it takes less current to keep it in motion. At that point, the current demand is reduced, the voltage returns to its original value, and the wall outlet again provides a reasonable approximation of an ideal voltage source.

Let us agree that we will use the expression *circuit element* to refer to the mathematical model. The choice of a particular model for any real device must be made on the basis of experimental data or experience; we will usually assume that this choice has already been made. For simplicity, we initially consider circuits with idealized components represented by simple models.

All the simple circuit elements that we will consider can be classified according to the relationship of the current through the element to the voltage across the element. For example, if the voltage across the element is linearly proportional to the current through it, we will call the element a resistor. Other types of simple circuit elements have terminal voltages which are proportional to the *derivative* of the current with respect to time (an inductor), or to the *integral* of the current with respect to time (a capacitor). There are also elements in which the voltage is completely independent of the current, or the current is completely independent of the voltage; these are termed *independent sources*. Furthermore, we will need to define special kinds of sources for which either the source voltage or current depends upon a current or voltage elsewhere in the circuit; such sources are referred to as *dependent sources*. Dependent sources are used a great deal in electronics to model both dc and ac behavior of transistors, especially in amplifier circuits.

## Independent Voltage Sources

The first element we will consider is the *independent voltage source*. The circuit symbol is shown in Fig. 2.15a; the subscript *s* merely identifies the voltage as a "source" voltage, and is common but not required. An *independent voltage source* is characterized by a terminal voltage which is completely independent of the current through it. Thus, if we are given an independent voltage source and are notified that the terminal voltage is 12 V, then we always assume this voltage, regardless of the current flowing.

The independent voltage source is an *ideal* source and does not represent exactly any real physical device, because the ideal source could theoretically deliver an infinite amount of energy from its terminals. This idealized voltage source does, however, furnish a reasonable approximation to several practical voltage sources. An automobile storage battery, for example, has a 12 V terminal voltage that remains essentially constant as long as the current through it does not exceed a few amperes. A small current may flow in either direction through the battery. If it is positive and flowing out of the positively marked terminal, then the battery is furnishing power to the headlights, for example; if the current is positive and flowing into the positive terminal, then the battery is charging by absorbing energy from the alternator.<sup>3</sup> An ordinary household electrical outlet also approximates an independent voltage source, providing a voltage  $v_s = 115\sqrt{2} \cos 2\pi 60t$  V; this representation is valid for currents less than 20 A or so.

A point worth repeating here is that the presence of the plus sign at the upper end of the symbol for the independent voltage source in Fig. 2.15a does not necessarily mean that the upper terminal is numerically positive with respect to the lower terminal. Instead, it means that the upper terminal is  $v_s$  volts positive with respect to the lower. If at some instant  $v_s$  happens to be negative, then the upper terminal is actually negative with respect to the lower at that instant.

(3) Or the battery of a friend's car, if you accidentally left your headlights on...

Consider a current arrow labeled “ $i$ ” placed adjacent to the upper conductor of the source as in Fig. 2.15b. The current  $i$  is entering the terminal at which the positive sign is located, the passive sign convention is satisfied, and the source thus *absorbs* power  $p = v_s i$ . More often than not, a source is expected to deliver power to a network and not to absorb it. Consequently, we might choose to direct the arrow as in Fig. 2.15c so that  $v_s i$  will represent the power *delivered* by the source. Technically, either arrow direction may be chosen; whenever possible, we will adopt the convention of Fig. 2.15c in this text for voltage and current sources, which are not usually considered passive devices.

An independent voltage source with a constant terminal voltage is often termed an independent dc voltage source and can be represented by either of the symbols shown in Fig. 2.16a and b. Note in Fig. 2.16b that when the physical plate structure of the battery is suggested, the longer plate is placed at the positive terminal; the plus and minus signs then represent redundant notation, but they are usually included anyway. For the sake of completeness, the symbol for an independent ac voltage source is shown in Fig. 2.16c.

## Independent Current Sources

Another ideal source which we will need is the *independent current source*. Here, the current through the element is completely independent of the voltage across it. The symbol for an independent current source is shown in Fig. 2.17. If  $i_s$  is constant, we call the source an independent dc current source. An ac current source is often drawn with a tilde through the arrow, similar to the ac voltage source shown in Fig. 2.16c.

Like the independent voltage source, the independent current source is at best a reasonable approximation for a physical element. In theory it can deliver infinite power from its terminals because it produces the same finite current for any voltage across it, no matter how large that voltage may be. It is, however, a good approximation for many practical sources, particularly in electronic circuits.

Although most students seem happy enough with an independent voltage source providing a fixed voltage but essentially any current, *it is a common mistake* to view an independent current source as having zero voltage across its terminals while providing a fixed current. In fact, we do not know *a priori* what the voltage across a current source will be—it depends entirely on the circuit to which it is connected.

## Dependent Sources

The two types of ideal sources that we have discussed up to now are called *independent* sources because the value of the source quantity is not affected in any way by activities in the remainder of the circuit. This is in contrast with yet another kind of ideal source, the *dependent*, or *controlled*, source, in which the source quantity is determined by a voltage or current existing at some other location in the system being analyzed. Sources such as these appear in the equivalent electrical models for many electronic devices, such as transistors, operational amplifiers, and integrated circuits. To distinguish between dependent and independent sources, we introduce the diamond symbols shown in Fig. 2.18. In Fig. 2.18a and c,  $K$  is a dimensionless scaling constant. In Fig. 2.18b,  $g$  is a scaling factor with units of A/V; in Fig. 2.18d,  $r$  is a scaling factor with units of V/A. The controlling current  $i_x$  and the controlling voltage  $v_x$  must be defined in the circuit.

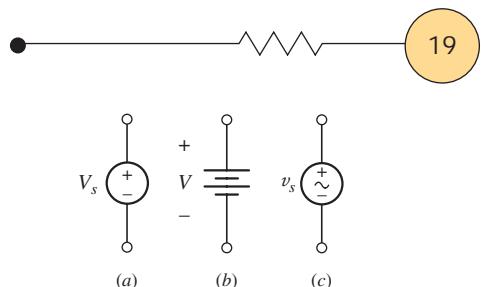


FIGURE 2.16 (a) DC voltage source symbol; (b) battery symbol; (c) ac voltage source symbol.

Terms like dc voltage source and dc current source are commonly used. Literally, they mean “direct-current voltage source” and “direct-current current source,” respectively. Although these terms may seem a little odd or even redundant, the terminology is so widely used there’s no point in fighting it.

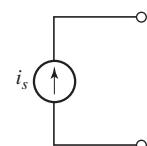


FIGURE 2.17 Circuit symbol for the independent current source.

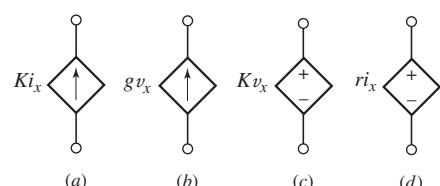


FIGURE 2.18 The four different types of dependent sources: (a) current-controlled current source; (b) voltage-controlled current source; (c) voltage-controlled voltage source; (d) current-controlled voltage source.

It does seem odd at first to have a current source whose value depends on a voltage, or a voltage source which is controlled by a current flowing through some other element. Even a voltage source depending on a remote voltage can appear strange. Such sources are invaluable for modeling complex systems, however, making the analysis algebraically straightforward. Examples include the drain current of a field effect transistor as a function of the gate voltage, or the output voltage of an analog integrated circuit as a function of differential input voltage. When encountered during circuit analysis, we write down the entire controlling expression for the dependent source just as we would if it was a numerical value attached to an independent source. This often results in the need for an additional equation to complete the analysis, unless the controlling voltage or current is already one of the specified unknowns in our system of equations.

## EXAMPLE 2.2

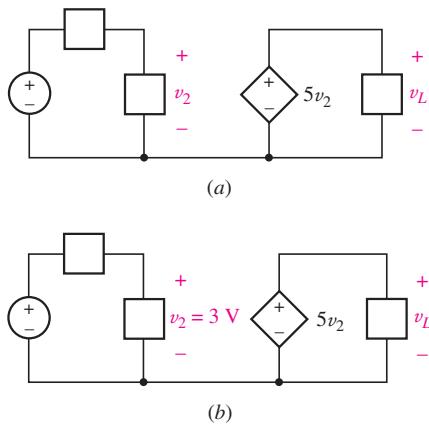


FIGURE 2.19 (a) An example circuit containing a voltage-controlled voltage source. (b) The additional information provided is included on the diagram.

In the circuit of Fig. 2.19a, if  $v_2$  is known to be 3 V, find  $v_L$ .

We have been provided with a partially labeled circuit diagram and the additional information that  $v_2 = 3$  V. This is probably worth adding to our diagram, as shown in Fig. 2.19b.

Next we step back and look at the information collected. In examining the circuit diagram, we notice that the desired voltage  $v_L$  is the same as the voltage across the dependent source. Thus,

$$v_L = 5v_2$$

At this point, we would be done with the problem if only we knew  $v_2$ !

Returning to our diagram, we see that we actually do know  $v_2$ —it was specified as 3 V. We therefore write

$$v_2 = 3$$

We now have two (simple) equations in two unknowns, and solve to find  $v_L = 15$  V.

An important lesson at this early stage of the game is that *the time it takes to completely label a circuit diagram is always a good investment*. As a final step, we should go back and check over our work to ensure that the result is correct.

## PRACTICE

2.9 Find the power *absorbed* by each element in the circuit in Fig. 2.20.

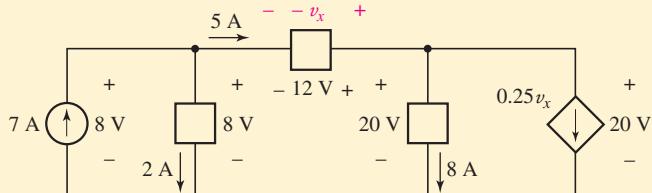


FIGURE 2.20

Ans: (left to right)  $-56$  W;  $16$  W;  $-60$  W;  $160$  W;  $-60$  W.

Dependent and independent voltage and current sources are *active* elements; they are capable of delivering power to some external device. For the present we will think of a *passive* element as one which is capable only of receiving power. However, we will later see that several passive elements are able to store finite amounts of energy and then return that energy later to various external devices; since we still wish to call such elements passive, it will be necessary to improve upon our two definitions a little later.

## Networks and Circuits

The interconnection of two or more simple circuit elements forms an electrical ***network***. If the network contains at least one closed path, it is also an electric ***circuit***. Note: Every circuit is a network, but not all networks are circuits (see Fig. 2.21)!

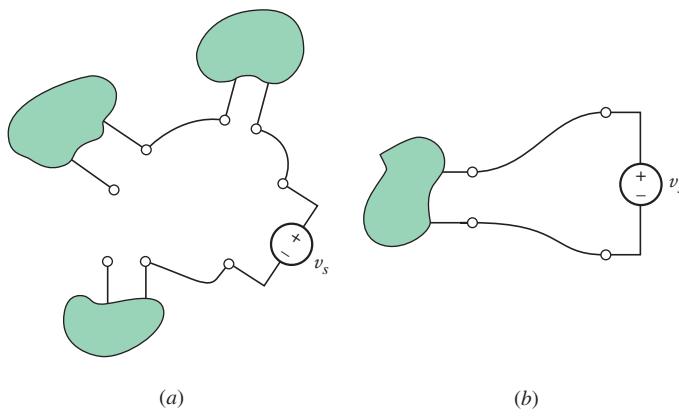
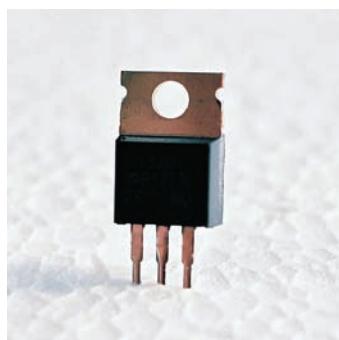


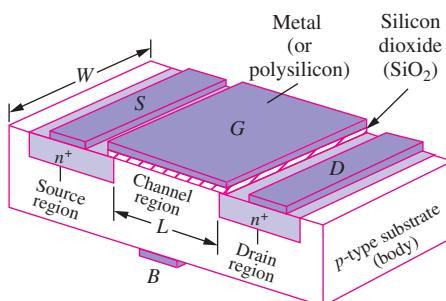
FIGURE 2.21 (a) A network that is not a circuit. (b) A network that is a circuit.

A network that contains at least one active element, such as an independent voltage or current source, is an active network. A network that does not contain any active elements is a passive network.

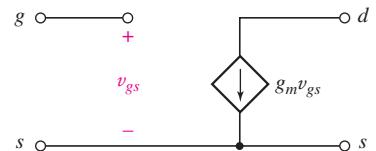
We have now defined what we mean by the term ***circuit element***, and we have presented the definitions of several specific circuit elements, the independent and dependent voltage and current sources. Throughout the remainder of the book we will define only five additional circuit elements: the resistor, inductor, capacitor, transformer, and the ideal operational amplifier (“op amp,” for short). These are all ideal elements. They are important because we may combine them into networks and circuits that represent real devices as accurately as we require. Thus, the transistor shown in Fig. 2.22a and b may be modeled by the voltage terminals designated  $v_{gs}$  and the single dependent current source of Fig. 2.22c. Note that the dependent current source produces a current that depends on a voltage elsewhere in the circuit. The parameter  $g_m$ , commonly referred to as the transconductance, is calculated using transistor-specific details as well as the operating point determined by the circuit connected to the transistor. It is generally a small number, on the order of  $10^{-2}$  to perhaps  $10 \text{ A/V}$ . This model works pretty well as long as the frequency of any sinusoidal source is neither very large nor very small; the model can be modified to account for frequency-dependent



(a)



(b)



(c)

FIGURE 2.22 The Metal Oxide Semiconductor Field Effect Transistor (MOSFET). (a) An IRF540 N-channel power MOSFET in a TO-220 package, rated at 100 V and 22 A; (b) cross-sectional view of a basic MOSFET (*R. Jaeger, Microelectronic Circuit Design, McGraw-Hill, 1997*); (c) equivalent circuit model for use in ac circuit analysis.

effects by including additional ideal circuit elements such as resistors and capacitors.

Similar (but much smaller) transistors typically constitute only one small part of an integrated circuit that may be less than 2 mm × 2 mm square and 200  $\mu\text{m}$  thick and yet contains several thousand transistors plus various resistors and capacitors. Thus, we may have a physical device that is about the size of one letter on this page but requires a model composed of ten thousand ideal simple circuit elements. We use this concept of “circuit modeling” in a number of electrical engineering topics covered in other courses, including electronics, energy conversion, and antennas.

## 2.4 OHM'S LAW

So far, we have been introduced to both dependent and independent voltage and current sources and were cautioned that they were *idealized* active elements that could only be approximated in a real circuit. We are now ready to meet another idealized element, the linear resistor. The resistor is the simplest passive element, and we begin our discussion by considering the work of an obscure German physicist, Georg Simon Ohm, who published a pamphlet in 1827 that described the results of one of the first efforts to measure currents and voltages, and to describe and relate them mathematically. One result was a statement of the fundamental relationship we now call **Ohm's law**, even though it has since been shown that this result was discovered 46 years earlier in England by Henry Cavendish, a brilliant semirecluse.

Ohm's law states that the voltage across conducting materials is directly proportional to the current flowing through the material, or

$$v = Ri \quad [4]$$

where the constant of proportionality  $R$  is called the **resistance**. The unit of resistance is the **ohm**, which is 1 V/A and customarily abbreviated by a capital omega,  $\Omega$ .

When this equation is plotted on  $i$ -versus- $v$  axes, the graph is a straight line passing through the origin (Fig. 2.23). Equation [4] is a linear equation, and we will consider it as the definition of a *linear resistor*. Resistance is normally considered to be a positive quantity, although negative resistances may be simulated with special circuitry.

Again, it must be emphasized that the linear resistor is an idealized circuit element; it is only a mathematical model of a real, physical device. "Resistors" may be easily purchased or manufactured, but it is soon found that the voltage-current ratios of these physical devices are reasonably constant only within certain ranges of current, voltage, or power, and depend also on temperature and other environmental factors. We usually refer to a linear resistor as simply a resistor; any resistor that is nonlinear will always be described as such. Nonlinear resistors should not necessarily be considered undesirable elements. Although it is true that their presence complicates an analysis, the performance of the device may depend on or be greatly improved by the nonlinearity. For example, fuses for overcurrent protection and Zener diodes for voltage regulation are very nonlinear in nature, a fact that is exploited when using them in circuit design.

## Power Absorption

Figure 2.24 shows several different resistor packages, as well as the most common circuit symbol used for a resistor. In accordance with the voltage, current, and power conventions already adopted, the product of  $v$  and  $i$  gives the power absorbed by the resistor. That is,  $v$  and  $i$  are selected to satisfy the passive sign convention. The absorbed power appears physically

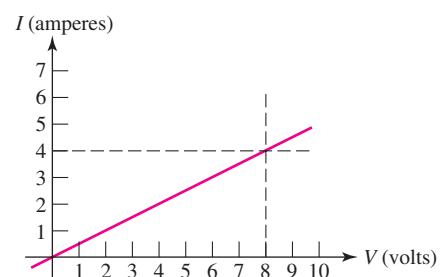
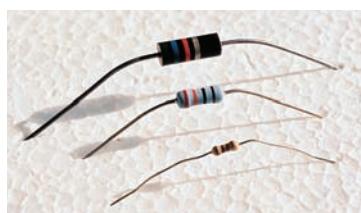


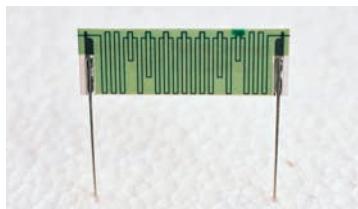
FIGURE 2.23 Current-voltage relationship for an example  $2\ \Omega$  linear resistor. Note the slope of the line is  $0.5\ A/V$ , or  $500\ m\Omega^{-1}$ .



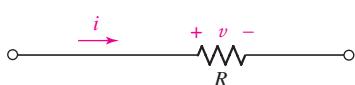
(a)



(b)



(c)



(d)

FIGURE 2.24 (a) Several common resistor packages. (b) A  $560\ \Omega$  power resistor rated at up to  $50\ W$ . (c) A  $5\%$  tolerance  $10\text{-teraohm}$  ( $10,000,000,000,000\ \Omega$ ) resistor manufactured by Ohmcraft. (d) Circuit symbol for the resistor, applicable to all of the devices in (a) through (c).

as heat and/or light and is always positive; a (positive) resistor is a passive element that cannot deliver power or store energy. Alternative expressions for the absorbed power are

$$p = vi = i^2R = v^2/R \quad [5]$$

One of the authors (who shall remain anonymous) had the unfortunate experience of inadvertently connecting a  $100\ \Omega$ , 2 W carbon resistor across a 110 V source. The ensuing flame, smoke, and fragmentation were rather disconcerting, demonstrating clearly that a practical resistor has definite limits to its ability to behave like the ideal linear model. In this case, the unfortunate resistor was called upon to absorb 121 W; since it was designed to handle only 2 W, its reaction was understandably violent.

### EXAMPLE 2.3

The  $560\ \Omega$  resistor shown in Fig. 2.24b is connected to a circuit which causes a current of 42.4 mA to flow through it. Calculate the voltage across the resistor and the power it is dissipating.

The voltage across the resistor is given by Ohm's law:

$$v = Ri = (560)(0.0424) = 23.7\text{ V}$$

The dissipated power can be calculated in several different ways. For instance,

$$p = vi = (23.7)(0.0424) = 1.005\text{ W}$$

Alternatively,

$$p = v^2/R = (23.7)^2/560 = 1.003\text{ W}$$

or

$$p = i^2R = (0.0424)^2(560) = 1.007\text{ W}$$

We note several things.

First, we calculated the power in three different ways, and we seem to have obtained *three different answers!*

In reality, however, we rounded our voltage to three significant digits, which will impact the accuracy of any subsequent quantity we calculate with it. With this in mind, we see that the answers show reasonable agreement (within 1%).

The other point worth noting is that the resistor is rated to 50 W—since we are only dissipating approximately 2% of this value, the resistor is in no danger of overheating.

### PRACTICE

With reference to Fig. 2.25, compute the following:

2.10  $R$  if  $i = -2\ \mu\text{A}$  and  $v = -44\text{ V}$ .

2.11 The power absorbed by the resistor if  $v = 1\text{ V}$  and  $R = 2\text{ k}\Omega$ .

2.12 The power absorbed by the resistor if  $i = 3\text{ nA}$  and  $R = 4.7\text{ M}\Omega$ .

Ans:  $22\text{ M}\Omega$ ;  $500\ \mu\text{W}$ ;  $42.3\text{ pW}$ .

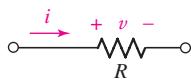


FIGURE 2.25

## PRACTICAL APPLICATION

### Wire Gauge

Technically speaking, any material (except for a superconductor) will provide resistance to current flow. As in all introductory circuits texts, however, we tacitly assume that wires appearing in circuit diagrams have zero resistance. This implies that there is no potential difference between the ends of a wire, and hence no power absorbed or heat generated. Although usually not an unreasonable assumption, it does neglect practical considerations when choosing the appropriate wire diameter for a specific application.

Resistance is determined by (1) the inherent resistivity of a material and (2) the device geometry. **Resistivity**, represented by the symbol  $\rho$ , is a measure of the ease with which electrons can travel through a certain material. Since it is the ratio of the electric field ( $V/m$ ) to the areal density of current flowing in the material ( $A/m^2$ ), the general unit of  $\rho$  is an  $\Omega \cdot m$ , although metric prefixes are often employed. Every material has a different inherent resistivity, which depends on temperature. Some examples are shown in Table 2.3; as can be seen, there is a small variation between different types of copper (less than 1%) but a very large difference between different metals. In particular, although physically stronger than copper, steel wire is several times more resistive. In some technical discussions, it is more common to see the conductivity (symbolized by  $\sigma$ ) of a

material quoted, which is simply the reciprocal of the resistivity.

The resistance of a particular object is obtained by multiplying the resistivity by the length  $\ell$  of the resistor and dividing by the cross-sectional area ( $A$ ) as in Eq. [6]; these parameters are illustrated in Fig. 2.26.

$$R = \rho \frac{\ell}{A} \quad [6]$$

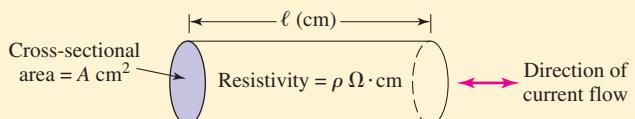


FIGURE 2.26 Definition of geometrical parameters used to compute the resistance of a wire. The resistivity of the material is assumed to be spatially uniform.

We determine the resistivity when we select the material from which to fabricate a wire and measure the temperature of the application environment. Since a finite amount of power is absorbed by the wire due to its resistance, current flow leads to the production of heat. Thicker wires have lower resistance and also dissipate heat more easily but are heavier, take up a larger volume, and are more expensive. Thus, we are motivated by practical considerations to choose the smallest wire that

TABLE 2.3 Common Electrical Wire Materials and Resistivities\*

ASTM Specification**	Temper and Shape	Resistivity at 20°C ( $\mu\Omega \cdot \text{cm}$ )
B33	Copper, tinned soft, round	1.7654
B75	Copper, tube, soft, OF copper	1.7241
B188	Copper, hard bus tube, rectangular or square	1.7521
B189	Copper, lead-coated soft, round	1.7654
B230	Aluminum, hard, round	2.8625
B227	Copper-clad steel, hard, round, grade 40 HS	4.3971
B355	Copper, nickel-coated soft, round Class 10	1.9592
B415	Aluminum-clad steel, hard, round	8.4805

\* C. B. Rawlins, "Conductor materials," *Standard Handbook for Electrical Engineering*, 13th ed., D. G. Fink and H. W. Beaty, eds. New York: McGraw-Hill, 1993, pp. 4-4 to 4-8.

\*\* American Society of Testing and Materials.

(Continued on next page)

can safely do the job, rather than simply choosing the largest-diameter wire available in an effort to minimize resistive losses. The American Wire Gauge (AWG) is a standard system of specifying wire size. In selecting a wire gauge, smaller AWG corresponds to a larger wire

diameter; an abbreviated table of common gauges is given in Table 2.4. Local fire and electrical safety codes typically dictate the required gauge for specific wiring applications, based on the maximum current expected as well as where the wires will be located.

**TABLE 2.4 Some Common Wire Gauges and the Resistance of (Soft) Solid Copper Wire\***

Conductor Size (AWG)	Cross-Sectional Area (mm <sup>2</sup> )	Ohms per 1000 ft at 20°C
28	0.0804	65.3
24	0.205	25.7
22	0.324	16.2
18	0.823	6.39
14	2.08	2.52
12	3.31	1.59
6	13.3	0.3952
4	21.1	0.2485
2	33.6	0.1563

\* C. B. Rawlins et al., *Standard Handbook for Electrical Engineering*, 13th ed., D. G. Fink and H. W. Beaty, eds. New York: McGraw-Hill, 1993, p. 4-47.

## EXAMPLE 2.4

A dc power link is to be made between two islands separated by a distance of 24 miles. The operating voltage is 500 kV and the system capacity is 600 MW. Calculate the maximum dc current flow, and estimate the resistivity of the cable, assuming a diameter of 2.5 cm and a solid (not stranded) wire.

Dividing the maximum power (600 MW, or  $600 \times 10^6$  W) by the operating voltage (500 kV, or  $500 \times 10^3$  V) yields a maximum current of

$$\frac{600 \times 10^6}{500 \times 10^3} = 1200 \text{ A}$$

The cable resistance is simply the ratio of the voltage to the current, or

$$R_{\text{cable}} = \frac{500 \times 10^3}{1200} = 417 \Omega$$

We know the length:

$$\ell = (24 \text{ miles}) \left( \frac{5280 \text{ ft}}{1 \text{ mile}} \right) \left( \frac{12 \text{ in}}{1 \text{ ft}} \right) \left( \frac{2.54 \text{ cm}}{1 \text{ in}} \right) = 3,862,426 \text{ cm}$$

Given that most of our information appears to be valid to only two significant figures, we round this to  $3.9 \times 10^6 \text{ cm}$ .

With the cable diameter specified as 2.5 cm, we know its cross-sectional area is  $4.9 \text{ cm}^2$ .

$$\text{Thus, } \rho_{\text{cable}} = R_{\text{cable}} \frac{A}{\ell} = 417 \left( \frac{4.9}{3.9 \times 10^6} \right) = 520 \mu\Omega \cdot \text{cm}$$

### PRACTICE

2.13 A 500 ft long 24 AWG soft copper wire is carrying a current of 100 mA. What is the voltage dropped across the wire?

Ans: 3.26 V.

## Conductance

For a linear resistor the ratio of current to voltage is also a constant

$$\frac{i}{v} = \frac{1}{R} = G \quad [7]$$

where  $G$  is called the *conductance*. The SI unit of conductance is the siemens (S), 1 A/V. An older, unofficial unit for conductance is the mho, which was often abbreviated as  $\mathfrak{S}$  and is still occasionally written as  $\Omega^{-1}$ . You will occasionally see it used on some circuit diagrams, as well as in catalogs and texts. The same circuit symbol (Fig. 2.24d) is used to represent both resistance and conductance. The absorbed power is again necessarily positive and may be expressed in terms of the conductance by

$$p = vi = v^2 G = \frac{i^2}{G} \quad [8]$$

Thus a  $2 \Omega$  resistor has a conductance of  $\frac{1}{2} \text{ S}$ , and if a current of 5 A is flowing through it, then a voltage of 10 V is present across the terminals and a power of 50 W is being absorbed.

All the expressions given so far in this section were written in terms of instantaneous current, voltage, and power, such as  $v = iR$  and  $p = vi$ . We should recall that this is a shorthand notation for  $v(t) = Ri(t)$  and  $p(t) = v(t)i(t)$ . The current through and voltage across a resistor must both vary with time in the same manner. Thus, if  $R = 10 \Omega$  and  $v = 2 \sin 100t \text{ V}$ , then  $i = 0.2 \sin 100t \text{ A}$ . Note that the power is given by  $0.4 \sin^2 100t \text{ W}$ , and a simple sketch will illustrate the different nature of its variation with time. Although the current and voltage are each negative during certain time intervals, the absorbed power is *never* negative!

Resistance may be used as the basis for defining two commonly used terms, *short circuit* and *open circuit*. We define a short circuit as a resistance of zero ohms; then, since  $v = iR$ , the voltage across a short circuit must be zero, although the current may have any value. In an analogous manner,



we define an open circuit as an infinite resistance. It follows from Ohm's law that the current must be zero, regardless of the voltage across the open circuit. Although real wires have a small resistance associated with them, we always assume them to have zero resistance unless otherwise specified. Thus, in all of our circuit schematics, wires are taken to be perfect short circuits.

## SUMMARY AND REVIEW

In this chapter, we introduced the topic of units – specifically those relevant to electrical circuits—and their relationship to fundamental (SI) units. We also discussed current and current sources, voltage and voltage sources, and the fact that the product of voltage and current yields power (the rate of energy consumption or generation). Since power can be either positive or negative depending on the current direction and voltage polarity, the passive sign convention was described to ensure we always know if an element is *absorbing* or *supplying* energy to the rest of the circuit. Four additional sources were introduced, forming a general class known as dependent sources. They are often used to model complex systems and electrical components, but the actual value of voltage or current supplied is typically unknown until the entire circuit is analyzed. We concluded the chapter with the resistor—by far the most common circuit element—whose voltage and current are linearly related (described by Ohm's law). Whereas the *resistivity* of a material is one of its fundamental properties (measured in  $\Omega \cdot \text{cm}$ ), *resistance* describes a device property (measured in  $\Omega$ ) and hence depends not only on resistivity but on the device geometry (i.e., length and area) as well.

We conclude with key points of this chapter to review, along with appropriate examples.

- ❑ The system of units most commonly used in electrical engineering is the SI.
- ❑ The direction in which positive charges are moving is the direction of positive current flow; alternatively, positive current flow is in the direction opposite that of moving electrons.
- ❑ To define a current, both a value and a direction must be given. Currents are typically denoted by the uppercase letter "I" for constant (dc) values, and either  $i(t)$  or simply  $i$  otherwise.
- ❑ To define a voltage across an element, it is necessary to label the terminals with "+" and "-" signs as well as to provide a value (either an algebraic symbol or a numerical value).
- ❑ Any element is said to supply positive power if positive current flows out of the positive voltage terminal. Any element absorbs positive power if positive current flows into the positive voltage terminal. (Example 2.1)
- ❑ There are six sources: the independent voltage source, the independent current source, the current-controlled dependent current source, the voltage-controlled dependent current source, the voltage-controlled dependent voltage source, and the current-controlled dependent voltage source. (Example 2.2)

Note that a current represented by  $i$  or  $i(t)$  can be constant (dc) or time-varying, but currents represented by the symbol  $I$  must be non-time-varying.

- Ohm's law states that the voltage across a linear resistor is directly proportional to the current flowing through it; i.e.,  $v = Ri$ . (Example 2.3)
- The power dissipated by a resistor (which leads to the production of heat) is given by  $p = vi = i^2R = v^2/R$ . (Example 2.3)
- Wires are typically assumed to have zero resistance in circuit analysis. When selecting a wire gauge for a specific application, however, local electrical and fire codes must be consulted. (Example 2.4)

## READING FURTHER

A good book that discusses the properties and manufacture of resistors in considerable depth:

Felix Zandman, Paul-René Simon, and Joseph Szwarc, *Resistor Theory and Technology*. Raleigh, N.C.: SciTech Publishing, 2002.

A good all-purpose electrical engineering handbook:

Donald G. Fink and H. Wayne Beaty, *Standard Handbook for Electrical Engineers*, 13th ed., New York: McGraw-Hill, 1993.

In particular, pp. 1-1 to 1-51, 2-8 to 2-10, and 4-2 to 4-207 provide an in-depth treatment of topics related to those discussed in this chapter.

A detailed reference for the SI is available on the Web from the National Institute of Standards:

Ambler Thompson and Barry N. Taylor, *Guide for the Use of the International System of Units (SI)*, NIST Special Publication 811, 2008 edition, [www.nist.gov](http://www.nist.gov).

## EXERCISES

### 2.1 Units and Scales

1. Convert the following to engineering notation:

- |                            |                                     |
|----------------------------|-------------------------------------|
| (a) 0.045 W                | (b) 2000 pJ                         |
| (c) 0.1 ns                 | (d) 39,212 as                       |
| (e) 3 Ω                    | (f) 18,000 m                        |
| (g) 2,500,000,000,000 bits | (h) $10^{15}$ atoms/cm <sup>3</sup> |

2. Convert the following to engineering notation:

- |                   |                      |
|-------------------|----------------------|
| (a) 1230 fs       | (b) 0.0001 decimeter |
| (c) 1400 mK       | (d) 32 nm            |
| (e) 13,560 kHz    | (f) 2021 micromoles  |
| (g) 13 deciliters | (h) 1 hectometer     |

3. Express the following in engineering units:

- |                        |                            |
|------------------------|----------------------------|
| (a) 1212 mV            | (b) $10^{11}$ pA           |
| (c) 1000 yoctoseconds  | (d) 33.9997 zeptoseconds   |
| (e) 13,100 attoseconds | (f) $10^{-14}$ zettasecond |
| (g) $10^{-5}$ second   | (h) $10^{-9}$ Gs           |

4. Expand the following distances in simple meters:

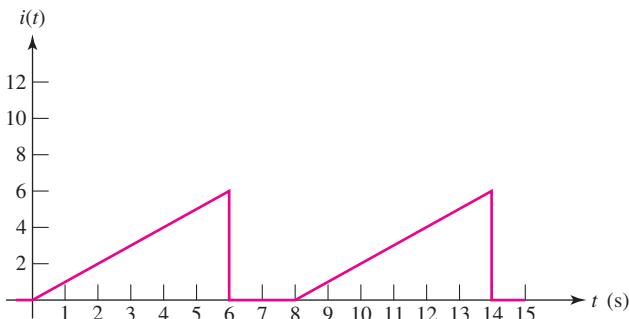
- |          |          |          |
|----------|----------|----------|
| (a) 1 Zm | (b) 1 Em | (c) 1 Pm |
| (d) 1 Tm | (e) 1 Gm | (f) 1 Mm |

5. Convert the following to SI units, taking care to employ proper engineering notation:
  - (a) 212°F
  - (b) 0°F
  - (c) 0 K
  - (d) 200 hp
  - (e) 1 yard
  - (f) 1 mile
6. Convert the following to SI units, taking care to employ proper engineering notation:
  - (a) 100°C
  - (b) 0°C
  - (c) 4.2 K
  - (d) 150 hp
  - (e) 500 Btu
  - (f) 100 J/s
7. A certain krypton fluoride laser generates 15 ns long pulses, each of which contains 550 mJ of energy. (a) Calculate the peak instantaneous output power of the laser. (b) If up to 100 pulses can be generated per second, calculate the maximum average power output of the laser.
8. When operated at a wavelength of 750 nm, a certain Ti:sapphire laser is capable of producing pulses as short as 50 fs, each with an energy content of 500 μJ. (a) Calculate the instantaneous output power of the laser. (b) If the laser is capable of a pulse repetition rate of 80 MHz, calculate the maximum average output power that can be achieved.
9. An electric vehicle is driven by a single motor rated at 40 hp. If the motor is run continuously for 3 h at maximum output, calculate the electrical energy consumed. Express your answer in SI units using engineering notation.
10. Under insolation conditions of 500 W/m<sup>2</sup> (direct sunlight), and 10% solar cell efficiency (defined as the ratio of electrical output power to incident solar power), calculate the area required for a photovoltaic (solar cell) array capable of running the vehicle in Exer. 9 at half power.
11. A certain metal oxide nanowire piezoelectricity generator is capable of producing 100 pW of usable electricity from the type of motion obtained from a person jogging at a moderate pace. (a) How many nanowire devices are required to operate a personal MP3 player which draws 1 W of power? (b) If the nanowires can be produced with a density of 5 devices per square micron directly onto a piece of fabric, what area is required, and would it be practical?
12. A particular electric utility charges customers different rates depending on their daily rate of energy consumption: \$0.05/kWh up to 20 kWh, and \$0.10/kWh for all energy usage above 20 kWh in any 24 hour period. (a) Calculate how many 100 W light bulbs can be run continuously for less than \$10 per week. (b) Calculate the daily energy cost if 2000 kW of power is used continuously.
13. The Tilting Windmill Electrical Cooperative LLC Inc. has instituted a differential pricing scheme aimed at encouraging customers to conserve electricity use during daylight hours, when local business demand is at its highest. If the price per kilowatthour is \$0.033 between the hours of 9 p.m. and 6 a.m., and \$0.057 for all other times, how much does it cost to run a 2.5 kW portable heater continuously for 30 days?
14. Assuming a global population of 9 billion people, each using approximately 100 W of power continuously throughout the day, calculate the total land area that would have to be set aside for photovoltaic power generation, assuming 800 W/m<sup>2</sup> of incident solar power and a conversion efficiency (sunlight to electricity) of 10%.

## 2.2 Charge, Current, Voltage, and Power

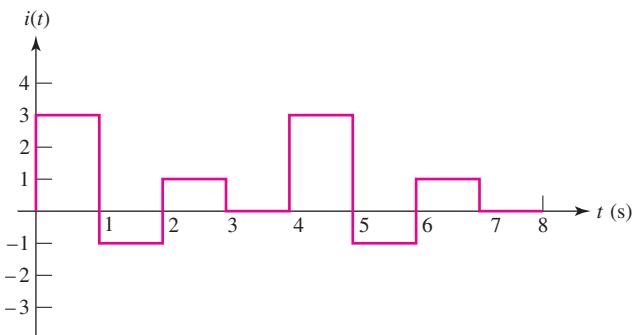
15. The total charge flowing out of one end of a small copper wire and into an unknown device is determined to follow the relationship  $q(t) = 5e^{-t/2}$  C, where  $t$  is expressed in seconds. Calculate the current flowing into the device, taking note of the sign.
16. The current flowing into the collector lead of a certain bipolar junction transistor (BJT) is measured to be 1 nA. If no charge was transferred in or out of the collector lead prior to  $t = 0$ , and the current flows for 1 min, calculate the total charge which crosses into the collector.

17. The total charge stored on a 1 cm diameter insulating plate is  $-10^{13}$  C.  
 (a) How many electrons are on the plate? (b) What is the areal density of electrons (number of electrons per square meter)? (c) If additional electrons are added to the plate from an external source at the rate of  $10^6$  electrons per second, what is the magnitude of the current flowing between the source and the plate?
18. A mysterious device found in a forgotten laboratory accumulates charge at a rate specified by the expression  $q(t) = 9 - 10t$  C from the moment it is switched on. (a) Calculate the total charge contained in the device at  $t = 0$ . (b) Calculate the total charge contained at  $t = 1$  s. (c) Determine the current flowing into the device at  $t = 1$  s, 3 s, and 10 s.
19. A new type of device appears to accumulate charge according to the expression  $q(t) = 10t^2 - 22t$  mC ( $t$  in s). (a) In the interval  $0 \leq t < 5$  s, at what time does the current flowing into the device equal zero? (b) Sketch  $q(t)$  and  $i(t)$  over the interval  $0 \leq t < 5$  s.
20. The current flowing through a tungsten-filament light bulb is determined to follow  $i(t) = 114 \sin(100\pi t)$  A. (a) Over the interval defined by  $t = 0$  and  $t = 2$  s, how many times does the current equal zero amperes? (b) How much charge is transported through the light bulb in the first second?
21. The current waveform depicted in Fig. 2.27 is characterized by a period of 8 s.  
 (a) What is the average value of the current over a single period? (b) If  $q(0) = 0$ , sketch  $q(t)$ ,  $0 < t < 20$  s.



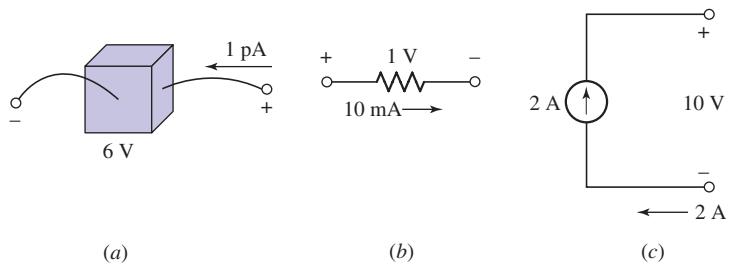
■ FIGURE 2.27 An example of a time-varying current.

22. The current waveform depicted in Fig. 2.28 is characterized by a period of 4 s.  
 (a) What is the average value of the current over a single period? (b) Compute the average current over the interval  $1 < t < 3$  s. (c) If  $q(0) = 1$  C, sketch  $q(t)$ ,  $0 < t < 4$  s.



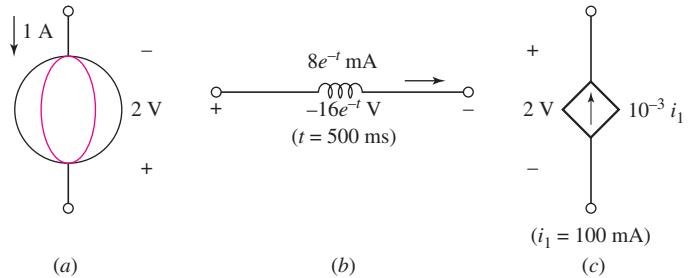
■ FIGURE 2.28 An example of a time-varying current.

23. A path around a certain electric circuit has discrete points labeled *A*, *B*, *C*, and *D*. To move an electron from points *A* to *C* requires 5 pJ. To move an electron from *B* to *C* requires 3 pJ. To move an electron from *A* to *D* requires 8 pJ.  
 (a) What is the potential difference (in volts) between points *B* and *C*, assuming a “+” reference at *C*? (b) What is the potential difference (in volts) between points *B* and *D*, assuming a “+” reference at *D*? (c) What is the potential difference (in volts) between points *A* and *B* (again, in volts), assuming a “+” reference at *B*?
24. Two metallic terminals protrude from a device. The terminal on the left is the positive reference for a voltage called  $v_x$  (the other terminal is the negative reference). The terminal on the right is the positive reference for a voltage called  $v_y$  (the other terminal being the negative reference). If it takes 1 mJ of energy to push a single electron into the left terminal, determine the voltages  $v_x$  and  $v_y$ .
25. The convention for voltmeters is to use a black wire for the negative reference terminal and a red wire for the positive reference terminal. (a) Explain why two wires are required to measure a voltage. (b) If it is dark and the wires into the voltmeter are swapped by accident, what will happen during the next measurement?
26. Determine the power absorbed by each of the elements in Fig. 2.29.



■ FIGURE 2.29 Elements for Exer. 26.

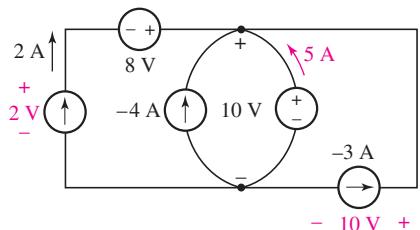
27. Determine the power absorbed by each of the elements in Fig. 2.30.



■ FIGURE 2.30 Elements for Exer. 27.

28. A constant current of 1 ampere is measured flowing into the positive reference terminal of a pair of leads whose voltage we'll call  $v_p$ . Calculate the absorbed power at  $t = 1$  s if  $v_p(t)$  equals (a) +1 V; (b) -1 V; (c)  $2 + 5 \cos(5t)$  V; (d)  $4e^{-2t}$  V, (e) Explain the significance of a negative value for absorbed power.

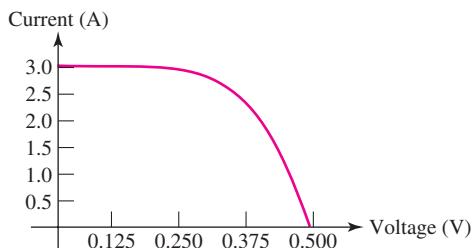
29. Determine the power supplied by the leftmost element in the circuit of Fig. 2.31.



■ FIGURE 2.31

30. The current-voltage characteristic of a silicon solar cell exposed to direct sunlight at noon in Florida during midsummer is given in Fig. 2.32. It is obtained by placing different-sized resistors across the two terminals of the device and measuring the resulting currents and voltages.

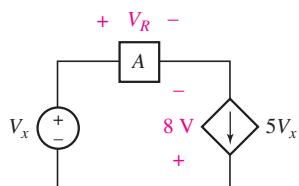
- What is the value of the short-circuit current?
- What is the value of the voltage at open circuit?
- Estimate the maximum power that can be obtained from the device.



■ FIGURE 2.32

### 2.3 Voltage and Current Sources

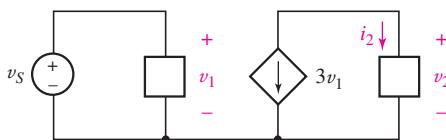
- Some of the ideal sources in the circuit of Fig. 2.31 are supplying positive power, and others are absorbing positive power. Determine which are which, and show that the algebraic sum of the power absorbed by each element (taking care to preserve signs) is equal to zero.
- By careful measurements it is determined that a benchtop argon ion laser is consuming (absorbing) 1.5 kW of electric power from the wall outlet, but only producing 5 W of optical power. Where is the remaining power going? Doesn't conservation of energy require the two quantities to be equal?
- Refer to the circuit represented in Fig. 2.33, while noting that the same current flows through each element. The voltage-controlled dependent source provides a current which is 5 times as large as the voltage  $V_x$ . (a) For  $V_R = 10 \text{ V}$  and  $V_x = 2 \text{ V}$ , determine the power absorbed by each element. (b) Is element A likely a passive or active source? Explain.



■ FIGURE 2.33

34. Refer to the circuit represented in Fig. 2.33, while noting that the same current flows through each element. The voltage-controlled dependent source provides a current which is 5 times as large as the voltage  $V_x$ . (a) For  $V_R = 100$  V and  $V_x = 92$  V, determine the power supplied by each element. (b) Verify that the algebraic sum of the supplied powers is equal to zero.

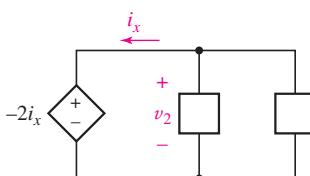
35. The circuit depicted in Fig. 2.34 contains a dependent current source; the magnitude and direction of the current it supplies are directly determined by the voltage labeled  $v_1$ . Note that therefore  $i_2 = -3v_1$ . Determine the voltage  $v_1$  if  $v_2 = 33i_2$  and  $i_2 = 100$  mA.



■ FIGURE 2.34

36. To protect an expensive circuit component from being delivered too much power, you decide to incorporate a fast-blowing fuse into the design. Knowing that the circuit component is connected to 12 V, its minimum power consumption is 12 W, and the maximum power it can safely dissipate is 100 W, which of the three available fuse ratings should you select: 1 A, 4 A, or 10 A? Explain your answer.

37. The dependent source in the circuit of Fig. 2.35 provides a voltage whose value depends on the current  $i_x$ . What value of  $i_x$  is required for the dependent source to be supplying 1 W?



■ FIGURE 2.35

## 2.4 Ohm's Law

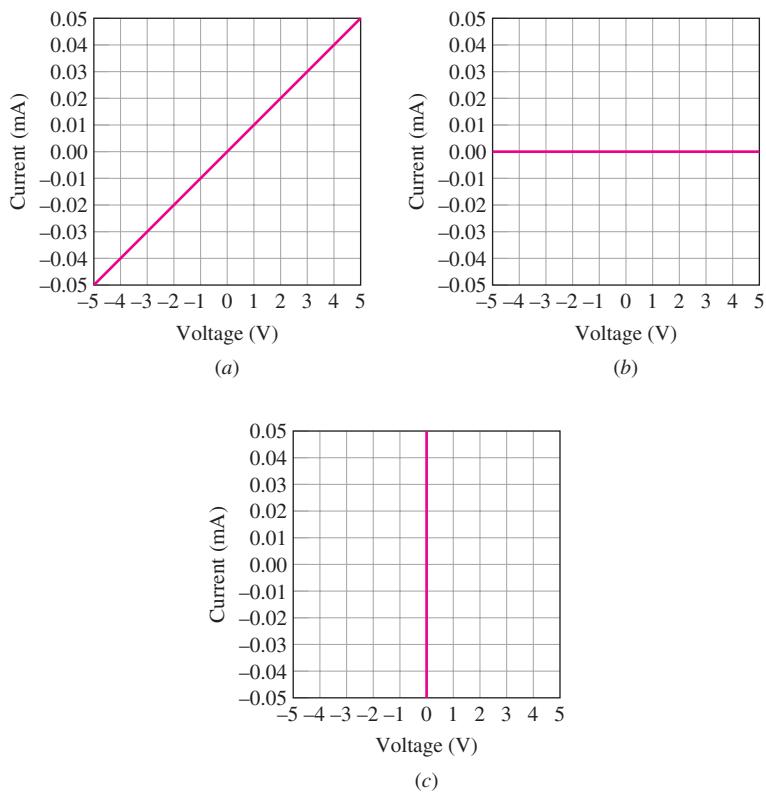
38. Determine the magnitude of the current flowing through a  $4.7\text{ k}\Omega$  resistor if the voltage across it is (a) 1 mV; (b) 10 V; (c)  $4e^{-t}$  V; (d)  $100 \cos(5t)$  V; (e) -7 V.

39. Real resistors can only be manufactured to a specific tolerance, so that in effect the value of the resistance is uncertain. For example, a  $1\text{ }\Omega$  resistor specified as 5% tolerance could in practice be found to have a value anywhere in the range of 0.95 to 1.05  $\Omega$ . Calculate the voltage across a  $2.2\text{ k}\Omega$  10% tolerance resistor if the current flowing through the element is (a) 1 mA; (b)  $4 \sin 44t$  mA.

40. (a) Sketch the current-voltage relationship (current on the y-axis) of a  $2\text{ k}\Omega$  resistor over the voltage range of  $-10\text{ V} \leq V_{\text{resistor}} \leq +10\text{ V}$ . Be sure to label both axes appropriately. (b) What is the numerical value of the slope (express your answer in siemens)?

41. Sketch the voltage across a  $33\text{ }\Omega$  resistor over the range  $0 < t < 2\pi$  s, if the current is given by  $2.8 \cos(t)$  A. Assume both the current and voltage are defined according to the passive sign convention.

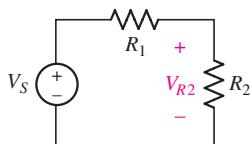
42. Figure 2.36 depicts the current-voltage characteristic of three different resistive elements. Determine the resistance of each, assuming the voltage and current are defined in accordance with the passive sign convention.

**FIGURE 2.36**

43. Determine the conductance (in siemens) of the following: (a)  $0 \Omega$ ; (b)  $100 \text{ M}\Omega$ ; (c)  $200 \text{ m}\Omega$ .
44. Determine the magnitude of the current flowing through a  $10 \text{ mS}$  conductance if the voltage across it is (a)  $2 \text{ mV}$ ; (b)  $-1 \text{ V}$ ; (c)  $100e^{-2t} \text{ V}$ ; (d)  $5 \sin(5t) \text{ V}$ ; (e)  $0 \text{ V}$ .
45. A 1% tolerance  $1 \text{ k}\Omega$  resistor may in reality have a value anywhere in the range of  $990$  to  $1010 \Omega$ . Assuming a voltage of  $9 \text{ V}$  is applied across it, determine (a) the corresponding range of current and (b) the corresponding range of absorbed power. (c) If the resistor is replaced with a 10% tolerance  $1 \text{ k}\Omega$  resistor, repeat parts (a) and (b).
46. The following experimental data is acquired for an unmarked resistor, using a variable-voltage power supply and a current meter. The current meter readout is somewhat unstable, unfortunately, which introduces error into the measurement.

Voltage (V)	Current (mA)
-2.0	-0.89
-1.2	-0.47
0.0	0.01
1.0	0.44
1.5	0.70

- (a) Plot the measured current-versus-voltage characteristic.
- (b) Using a best-fit line, estimate the value of the resistance.



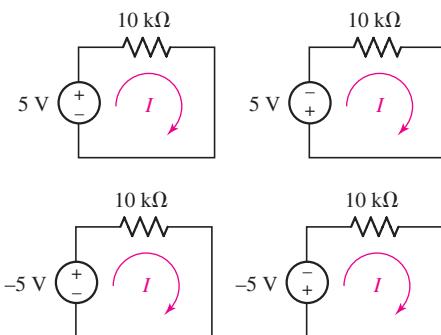
■ FIGURE 2.37

47. Utilize the fact that in the circuit of Fig. 2.37, the total power supplied by the voltage source must equal the total power absorbed by the two resistors to show that

$$V_{R2} = V_S \frac{R_2}{R_1 + R_2}$$

You may assume the same current flows through each element (a requirement of charge conservation).

48. For each of the circuits in Fig. 2.38, find the current  $I$  and compute the power absorbed by the resistor.



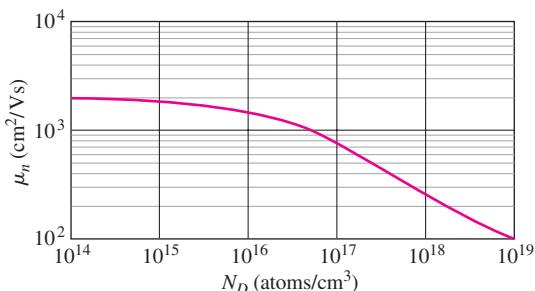
■ FIGURE 2.38

49. Sketch the power absorbed by a  $100\ \Omega$  resistor as a function of voltage over the range  $-2\text{ V} \leq V_{\text{resistor}} \leq +2\text{ V}$ .

### Chapter-Integrating Exercises



50. So-called “n-type” silicon has a resistivity given by  $\rho = (-qN_D\mu_n)^{-1}$ , where  $N_D$  is the volume density of phosphorus atoms ( $\text{atoms}/\text{cm}^3$ ),  $\mu_n$  is the electron mobility ( $\text{cm}^2/\text{V} \cdot \text{s}$ ), and  $q = -1.602 \times 10^{-19}\ \text{C}$  is the charge of each electron. Conveniently, a relationship exists between mobility and  $N_D$ , as shown in Fig. 2.39. Assume an 8 inch diameter silicon wafer (disk) having a thickness of  $300\ \mu\text{m}$ . Design a  $10\ \Omega$  resistor by specifying a phosphorus concentration in the range of  $2 \times 10^{15}\ \text{cm}^{-3} \leq N_D \leq 2 \times 10^{17}\ \text{cm}^{-3}$ , along with a suitable geometry (the wafer may be cut, but not thinned).



■ FIGURE 2.39

51. Figure 2.39 depicts the relationship between electron mobility  $\mu_n$  and dopant density  $N_D$  for n-type silicon. With the knowledge that resistivity in this material is given by  $\rho = N_D\mu_n/q$ , plot resistivity as a function of dopant density over the range  $10^{14}\ \text{cm}^{-3} \leq N_D \leq 10^{19}\ \text{cm}^{-3}$ .

- DP** 52. Referring to the data of Table 2.4, design a resistor whose value can be varied mechanically in the range of 100 to 500  $\Omega$  (assume operation at 20°C).
53. A 250 ft long span separates a dc power supply from a lamp which draws 25 A of current. If 14 AWG wire is used (note that two wires are needed for a total of 500 ft), calculate the amount of power wasted in the wire.
54. The resistance values in Table 2.4 are calibrated for operation at 20°C. They may be corrected for operation at other temperatures using the relationship<sup>4</sup>

$$\frac{R_2}{R_1} = \frac{234.5 + T_2}{234.5 + T_1}$$

where  $T_1$  = reference temperature (20°C in present case)

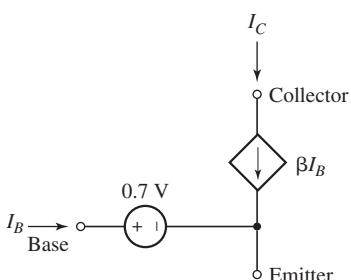
$T_2$  = desired operating temperature

$R_1$  = resistance at  $T_1$

$R_2$  = resistance at  $T_2$

A piece of equipment relies on an external wire made of 28 AWG soft copper, which has a resistance of 50.0  $\Omega$  at 20°C. Unfortunately, the operating environment has changed, and it is now 110.5°F. (a) Calculate the length of the original wire. (b) Determine by how much the wire should be shortened so that it is once again 50.0  $\Omega$ .

- DP** 55. Your favorite meter contains a precision (1% tolerance) 10  $\Omega$  resistor. Unfortunately, the last person who borrowed this meter somehow blew the resistor, and it needs to be replaced. Design a suitable replacement, assuming at least 1000 ft of each of the wire gauges listed in Table 2.4 is readily available to you.
56. At a new installation, you specified that all wiring should conform to the ASTM B33 specification (see Table 2.3). Unfortunately the subcontractor misread your instructions and installed B415 wiring instead (but the same gauge). Assuming the operating voltage is unchanged, (a) by how much will the current be reduced, and (b) how much additional power will be wasted in the lines? (Express both answers in terms of percentage.)
57. If 1 mA of current is forced through a 1 mm diameter, 2.3 meter long piece of hard, round, aluminum-clad steel (B415) wire, how much power is wasted as a result of resistive losses? If instead wire of the same dimensions but conforming to B75 specifications is used, by how much will the power wasted due to resistive losses be reduced?
58. The network shown in Fig. 2.40 can be used to accurately model the behavior of a bipolar junction transistor provided that it is operating in the forward active mode. The parameter  $\beta$  is known as the current gain. If for this device



■ FIGURE 2.40 DC model for a bipolar junction transistor operating in forward active mode.

(4) D. G. Fink and H. W. Beaty, *Standard Handbook for Electrical Engineers*, 13th ed. New York: McGraw-Hill, 1993, p. 2-9.

$\beta = 100$ , and  $I_B$  is determined to be  $100 \mu\text{A}$ , calculate (a)  $I_C$ , the current flowing into the collector terminal; and (b) the power dissipated by the base-emitter region.

59. A 100 W tungsten filament light bulb functions by taking advantage of resistive losses in the filament, absorbing 100 joules each second of energy from the wall socket. How much *optical* energy per second do you expect it to produce, and does this violate the principle of energy conservation?
60. Batteries come in a wide variety of types and sizes. Two of the most common are called “AA” and “AAA.” A single battery of either type is rated to produce a terminal voltage of 1.5 V when fully charged. So what are the differences between the two, other than size? (*Hint:* Think about energy.)

# Voltage and Current Laws

## INTRODUCTION

In Chap. 2 we were introduced to independent voltage and current sources, dependent sources, and resistors. We discovered that *dependent* sources come in four varieties, and are controlled by a voltage or current which exists elsewhere. Once we know the voltage across a resistor, we know its current (and vice versa); this is not the case for sources, however. In general, circuits must be analyzed to determine a complete set of voltages and currents. This turns out to be reasonably straightforward, and only two simple laws are needed in addition to Ohm's law. These new laws are Kirchhoff's current law (KCL) and Kirchhoff's voltage law (KVL), and they are simply restatements of charge and energy conservation, respectively. They apply to any circuit we will ever encounter, although in later chapters we will learn more efficient techniques for specific types of situations.

### 3.1 NODES, PATHS, LOOPS, AND BRANCHES

We now focus our attention on the current-voltage relationships in simple networks of two or more circuit elements. The elements will be connected by wires (sometimes referred to as "leads"), which have zero resistance. Since the network then appears as a number of simple elements and a set of connecting leads, it is called a ***lumped-parameter network***. A more difficult analysis problem arises when we are faced with a ***distributed-parameter network***, which contains an essentially infinite number of vanishingly small elements. We will concentrate on lumped-parameter networks in this text.

## KEY CONCEPTS

- New Circuit Terms: *Node, Path, Loop, and Branch*
- Kirchhoff's Current Law (KCL)
- Kirchhoff's Voltage Law (KVL)
- Analysis of Basic Series and Parallel Circuits
- Combination of Series and Parallel Sources
- Reduction of Series and Parallel Resistor Combinations
- Voltage and Current Division
- Ground Connections





In circuits assembled in the real world, the wires will always have finite resistance. However, this resistance is typically so small compared to other resistances in the circuit that we can neglect it without introducing significant error. In our idealized circuits, we will therefore refer to "zero resistance" wires from now on.

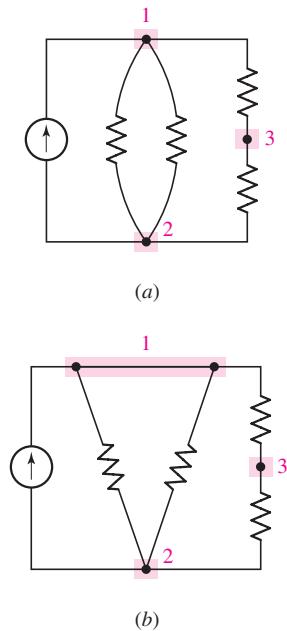


FIGURE 3.1 (a) A circuit containing three nodes and five branches. (b) Node 1 is redrawn to look like two nodes; it is still one node.

A point at which two or more elements have a common connection is called a **node**. For example, Fig. 3.1a shows a circuit containing three nodes. Sometimes networks are drawn so as to trap an unwary student into believing that there are more nodes present than is actually the case. This occurs when a node, such as node 1 in Fig. 3.1a, is shown as two separate junctions connected by a (zero-resistance) conductor, as in Fig. 3.1b. However, all that has been done is to spread the common point out into a common zero-resistance line. Thus, we must necessarily consider all of the perfectly conducting leads or portions of leads attached to the node as part of the node. Note also that every element has a node at each of its ends.

Suppose that we start at one node in a network and move through a simple element to the node at the other end. We then continue from that node through a different element to the next node, and continue this movement until we have gone through as many elements as we wish. If no node was encountered more than once, then the set of nodes and elements that we have passed through is defined as a **path**. If the node at which we started is the same as the node on which we ended, then the path is, by definition, a closed path or a **loop**.

For example, in Fig. 3.1a, if we move from node 2 through the current source to node 1, and then through the upper right resistor to node 3, we have established a path; since we have not continued on to node 2 again, we have not made a loop. If we proceeded from node 2 through the current source to node 1, down through the left resistor to node 2, and then up through the central resistor to node 1 again, we do not have a path, since a node (actually two nodes) was encountered more than once; we also do not have a loop, because a loop must be a path.

Another term whose use will prove convenient is **branch**. We define a branch as a single path in a network, composed of one simple element and the node at each end of that element. Thus, a path is a particular collection of branches. The circuit shown in Fig. 3.1a and b contains five branches.

## 3.2 KIRCHHOFF'S CURRENT LAW

We are now ready to consider the first of the two laws named for Gustav Robert Kirchhoff (two *h*'s and two *f*'s), a German university professor who was born about the time Ohm was doing his experimental work. This axiomatic law is called Kirchhoff's current law (abbreviated KCL), and it simply states that

The algebraic sum of the currents entering any node is zero.

This law represents a mathematical statement of the fact that charge cannot accumulate at a node. A *node is not a circuit element*, and it certainly cannot store, destroy, or generate charge. Hence, the currents must sum to zero. A hydraulic analogy is sometimes useful here: for example, consider three water pipes joined in the shape of a Y. We define three "currents" as flowing *into* each of the three pipes. If we insist that water is always flowing, then obviously we cannot have three positive water currents, or the pipes would burst. This is a result of our defining currents independent of

the direction that water is actually flowing. Therefore, the value of either one or two of the currents as defined must be negative.

Consider the node shown in Fig. 3.2. The algebraic sum of the four currents entering the node must be zero:

$$i_A + i_B + (-i_C) + (-i_D) = 0$$

However, the law could be equally well applied to the algebraic sum of the currents *leaving* the node:

$$(-i_A) + (-i_B) + i_C + i_D = 0$$

We might also wish to equate the sum of the currents having reference arrows directed into the node to the sum of those directed out of the node:

$$i_A + i_B = i_C + i_D$$

which simply states that the sum of the currents going in must equal the sum of the currents going out.

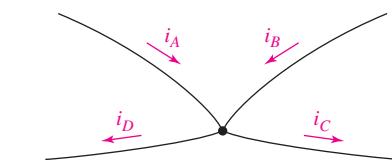
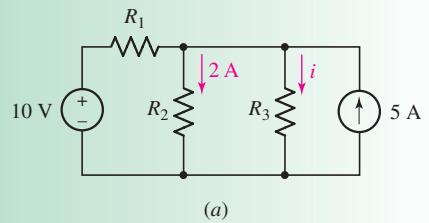


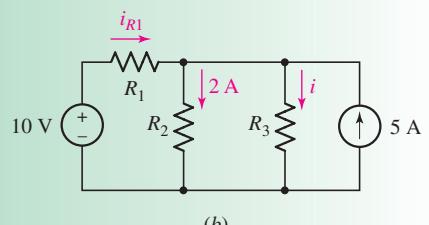
FIGURE 3.2 Example node to illustrate the application of Kirchhoff's current law.

**For the circuit in Fig. 3.3a, compute the current through resistor  $R_3$  if it is known that the voltage source supplies a current of 3 A.**

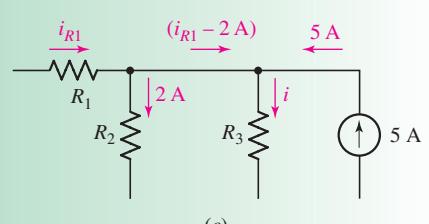
### EXAMPLE 3.1



(a)



(b)



(c)

FIGURE 3.3 (a) Simple circuit for which the current through resistor  $R_3$  is desired. (b) The current through resistor  $R_1$  is labeled so that a KCL equation can be written. (c) The currents into the top node of  $R_3$  are redrawn for clarity.

#### ► Identify the goal of the problem.

The current through resistor  $R_3$ , labeled as  $i$  on the circuit diagram.

#### ► Collect the known information.

The node at the top of  $R_3$  is connected to four branches.

Two of these currents are clearly labeled: 2 A flows out of the node into  $R_2$ , and 5 A flows into the node from the current source. We are told the current out of the 10 V source is 3 A.

#### ► Devise a plan.

If we label the current through  $R_1$  (Fig. 3.3b), we may write a KCL equation at the top node of resistors  $R_2$  and  $R_3$ .

#### ► Construct an appropriate set of equations.

Summing the currents flowing into the node:

$$i_{R_1} - 2 - i + 5 = 0$$

The currents flowing into this node are shown in the expanded diagram of Fig. 3.3c for clarity.

#### ► Determine if additional information is required.

We have one equation but two unknowns, which means we need to obtain an additional equation. At this point, the fact that we know the 10 V source is supplying 3 A comes in handy: KCL shows us that this is also the current  $i_{R_1}$ .

#### ► Attempt a solution.

Substituting, we find that  $i = 3 - 2 + 5 = 6$  A.

#### ► Verify the solution. Is it reasonable or expected?

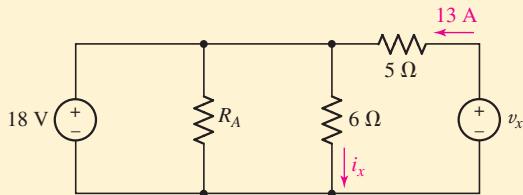
It is always worth the effort to recheck our work. Also, we can attempt to evaluate whether at least the magnitude of the solution is

(Continued on next page)

reasonable. In this case, we have two sources—one supplies 5 A, and the other supplies 3 A. There are no other sources, independent or dependent. Thus, we would not expect to find any current in the circuit in excess of 8 A.

### PRACTICE

- 3.1 Count the number of branches and nodes in the circuit in Fig. 3.4. If  $i_x = 3 \text{ A}$  and the 18 V source delivers 8 A of current, what is the value of  $R_A$ ? (*Hint:* You need Ohm's law as well as KCL.)



■ FIGURE 3.4

Ans: 5 branches, 3 nodes,  $1\Omega$ .

A compact expression for Kirchhoff's current law is

$$\sum_{n=1}^N i_n = 0 \quad [1]$$

which is just a shorthand statement for

$$i_1 + i_2 + i_3 + \dots + i_N = 0 \quad [2]$$

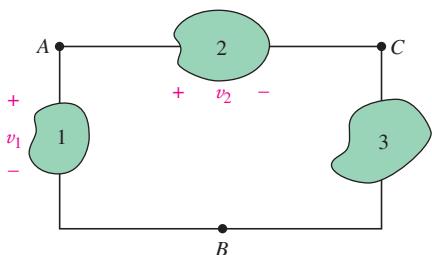
 When Eq. [1] or Eq. [2] is used, it is understood that the  $N$  current arrows are either all directed toward the node in question, or are all directed away from it.

### 3.3 KIRCHHOFF'S VOLTAGE LAW

Current is related to the charge flowing *through* a circuit element, whereas voltage is a measure of potential energy difference *across* the element. There is a single unique value for any voltage in circuit theory. Thus, the energy required to move a unit charge from point  $A$  to point  $B$  in a circuit must have a value independent of the path chosen to get from  $A$  to  $B$  (there is often more than one such path). We may assert this fact through Kirchhoff's voltage law (abbreviated **KVL**):

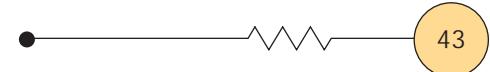
The algebraic sum of the voltages around any closed path is zero.

In Fig. 3.5, if we carry a charge of 1 C from  $A$  to  $B$  through element 1, the reference polarity signs for  $v_1$  show that we do  $v_1$  joules of work.<sup>1</sup> Now



■ FIGURE 3.5 The potential difference between points  $A$  and  $B$  is independent of the path selected.

(1) Note that we chose a 1 C charge for the sake of numerical convenience: therefore, we did  $(1 \text{ C})(v_1 \text{ J/C}) = v_1 \text{ joules of work.}$



if, instead, we choose to proceed from  $A$  to  $B$  via node  $C$ , then we expend  $(v_2 - v_3)$  joules of energy. The work done, however, is independent of the path in a circuit, and so any route must lead to the same value for the voltage. In other words,

$$v_1 = v_2 - v_3 \quad [3]$$

It follows that if we trace out a closed path, the algebraic sum of the voltages across the individual elements around it must be zero. Thus, we may write

$$v_1 + v_2 + v_3 + \cdots + v_N = 0$$

or, more compactly,

$$\sum_{n=1}^N v_n = 0 \quad [4]$$

We can apply KVL to a circuit in several different ways. One method that leads to fewer equation-writing errors than others consists of moving mentally around the closed path in a clockwise direction and writing down directly the voltage of each element whose (+) terminal is entered, and writing down the negative of every voltage first met at the (-) sign. Applying this to the single loop of Fig. 3.5, we have

$$-v_1 + v_2 - v_3 = 0$$

which agrees with our previous result, Eq. [3].



## EXAMPLE 3.2

In the circuit of Fig. 3.6, find  $v_x$  and  $i_x$ .

We know the voltage across two of the three elements in the circuit. Thus, KVL can be applied immediately to obtain  $v_x$ .

Beginning with the bottom node of the 5 V source, we apply KVL clockwise around the loop:

$$-5 - 7 + v_x = 0$$

so  $v_x = 12$  V.

KCL applies to this circuit, but only tells us that the same current ( $i_x$ ) flows through all three elements. We now know the voltage across the  $100\ \Omega$  resistor, however.

Invoking Ohm's law,

$$i_x = \frac{v_x}{100} = \frac{12}{100} \text{ A} = 120 \text{ mA}$$

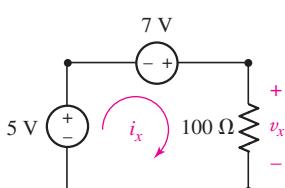


FIGURE 3.6 A simple circuit with two voltage sources and a single resistor.

## PRACTICE

3.2 Determine  $i_x$  and  $v_x$  in the circuit of Fig. 3.7.

Ans:  $v_x = -4$  V;  $i_x = -400$  mA.

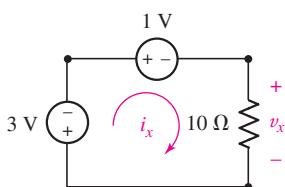


FIGURE 3.7

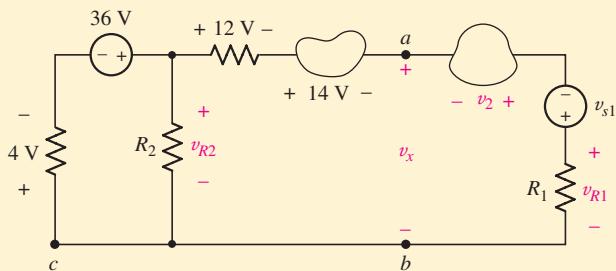
## EXAMPLE 3.3

In the circuit of Fig. 3.8 there are eight circuit elements. Find  $v_{R2}$  (the voltage across  $R_2$ ) and the voltage labeled  $v_x$ .

The best approach for finding  $v_{R2}$  is to look for a loop to which we can apply KVL. There are several options, but the leftmost loop offers a straightforward route, as two of the voltages are clearly specified. Thus, we find  $v_{R2}$  by writing a KVL equation around the loop on the left, starting at point  $c$ :

$$4 - 36 + v_{R2} = 0$$

which leads to  $v_{R2} = 32$  V.



■ FIGURE 3.8 A circuit with eight elements for which we desire  $v_{R2}$  and  $v_x$ .

To find  $v_x$ , we might think of this as the (algebraic) sum of the voltages across the three elements on the right. However, since we do not have values for these quantities, such an approach would not lead to a numerical answer. Instead, we apply KVL beginning at point  $c$ , moving up and across the top to  $a$ , through  $v_x$  to  $b$ , and through the conducting lead to the starting point:

$$+4 - 36 + 12 + 14 + v_x = 0$$

so that

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

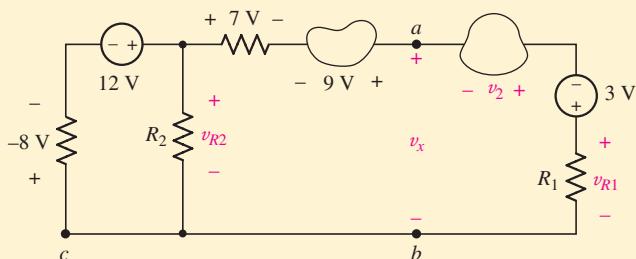
**An alternative approach:** Knowing  $v_{R2}$ , we might have taken the shortcut through  $R_2$ :

$$-32 + 12 + 14 + v_x = 0$$

yielding  $v_x = 6$  V once again.

## PRACTICE

3.3 For the circuit of Fig. 3.9, determine (a)  $v_{R2}$  and (b)  $v_2$ , if  $v_{R1} = 1$  V.



■ FIGURE 3.9

Ans: (a) 4 V; (b) -8 V.

As we have just seen, the key to correctly analyzing a circuit is to first methodically label all voltages and currents on the diagram. This way, carefully written KCL or KVL equations will yield correct relationships, and Ohm's law can be applied as necessary if more unknowns than equations are obtained initially. We illustrate these principles with a more detailed example.

## EXAMPLE 3.4

Determine  $v_x$  in the circuit of Fig. 3.10a.

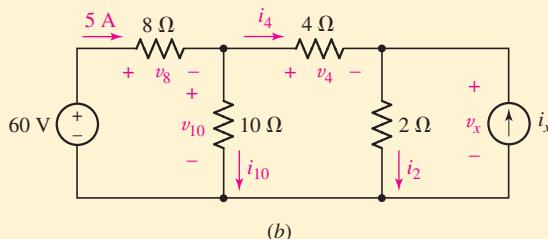
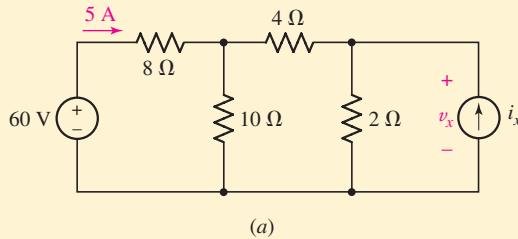


FIGURE 3.10 (a) A circuit for which  $v_x$  is to be determined using KVL. (b) Circuit with voltages and currents labeled.

We begin by labeling voltages and currents on the rest of the elements in the circuit (Fig. 3.10b). Note that  $v_x$  appears across the  $2\ \Omega$  resistor and the source  $i_x$  as well.

If we can obtain the current through the  $2\ \Omega$  resistor, Ohm's law will yield  $v_x$ . Writing the appropriate KCL equation, we see that

$$i_2 = i_4 + i_x$$

Unfortunately, we do not have values for any of these three quantities. Our solution has (temporarily) stalled.

Since we were given the current flowing from the  $60\text{ V}$  source, perhaps we should consider starting from that side of the circuit. Instead of finding  $v_x$  using  $i_2$ , it might be possible to find  $v_x$  directly using KVL. We can write the following KVL equations:

$$-60 + v_8 + v_{10} = 0$$

and

$$-v_{10} + v_4 + v_x = 0 \quad [5]$$

This is progress: we now have two equations in four unknowns, an improvement over one equation in which *all* terms were unknown. In fact, we know that  $v_8 = 40\text{ V}$  through Ohm's law, as we were told that  $5\text{ A}$  flows through the  $8\ \Omega$  resistor. Thus,  $v_{10} = 0 + 60 - 40 = 20\text{ V}$ ,

(Continued on next page)

so Eq. [5] reduces to

$$v_x = 20 - v_4$$

If we can determine  $v_4$ , the problem is solved.

The best route to finding a numerical value for the voltage  $v_4$  in this case is to employ Ohm's law, which requires a value for  $i_4$ . From KCL, we see that

$$i_4 = 5 - i_{10} = 5 - \frac{v_{10}}{10} = 5 - \frac{20}{10} = 3$$

so that  $v_4 = (4)(3) = 12$  V and hence  $v_x = 20 - 12 = 8$  V.

### PRACTICE

3.4 Determine  $v_x$  in the circuit of Fig. 3.11.

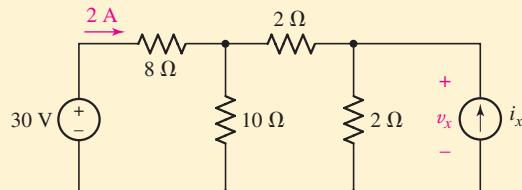


FIGURE 3.11

Ans:  $v_x = 12.8$  V.

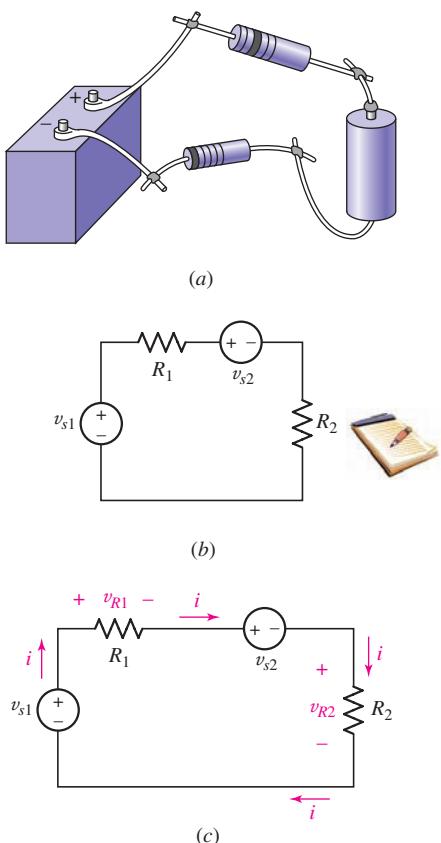


FIGURE 3.12 (a) A single-loop circuit with four elements. (b) The circuit model with source voltages and resistance values given. (c) Current and voltage reference signs have been added to the circuit.

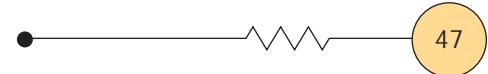
### 3.4 THE SINGLE-LOOP CIRCUIT

We have seen that repeated use of KCL and KVL in conjunction with Ohm's law can be applied to nontrivial circuits containing several loops and a number of different elements. Before proceeding further, this is a good time to focus on the concept of series (and, in the next section, parallel) circuits, as they form the basis of any network we will encounter in the future.

All of the elements in a circuit that carry the same current are said to be connected in **series**. As an example, consider the circuit of Fig. 3.10. The 60 V source is in series with the  $8\ \Omega$  resistor; they carry the same 5 A current. However, the  $8\ \Omega$  resistor is not in series with the  $4\ \Omega$  resistor; they carry different currents. Note that elements may carry equal currents and not be in series; two 100 W light bulbs in neighboring houses may very well carry equal currents, but they certainly do not carry the same current and are *not* connected in series.

Figure 3.12a shows a simple circuit consisting of two batteries and two resistors. Each terminal, connecting lead, and solder glob is assumed to have zero resistance; together they constitute an individual node of the circuit diagram in Fig. 3.12b. Both batteries are modeled by ideal voltage sources; any internal resistances they may have are assumed to be small enough to neglect. The two resistors are assumed to be ideal (linear) resistors.

We seek the current *through* each element, the voltage *across* each element, and the power *absorbed* by each element. Our first step in the analysis is the assumption of reference directions for the unknown currents. Arbitrarily, let us select a clockwise current  $i$  which flows out of the upper terminal of the voltage source on the left. This choice is indicated by an arrow labeled  $i$  at that point in the circuit, as shown in Fig. 3.12c. A trivial



application of Kirchhoff's current law assures us that this same current must also flow through every other element in the circuit; we emphasize this fact this one time by placing several other current symbols about the circuit.

Our second step in the analysis is a choice of the voltage reference for each of the two resistors. The passive sign convention requires that the resistor current and voltage variables be defined so that the current enters the terminal at which the positive voltage reference is located. Since we already (arbitrarily) selected the current direction,  $v_{R1}$  and  $v_{R2}$  are defined as in Fig. 3.12c.

The third step is the application of Kirchhoff's voltage law to the only closed path. Let us decide to move around the circuit in the clockwise direction, beginning at the lower left corner, and to write down directly every voltage first met at its positive reference, and to write down the negative of every voltage encountered at the negative terminal. Thus,

$$-v_{s1} + v_{R1} + v_{s2} + v_{R2} = 0 \quad [6]$$

We then apply Ohm's law to the resistive elements:

$$v_{R1} = R_1 i \quad \text{and} \quad v_{R2} = R_2 i$$

Substituting into Eq. [6] yields

$$-v_{s1} + R_1 i + v_{s2} + R_2 i = 0$$

Since  $i$  is the only unknown, we find that

$$i = \frac{v_{s1} - v_{s2}}{R_1 + R_2}$$

The voltage or power associated with any element may now be obtained by applying  $v = Ri$ ,  $p = vi$ , or  $p = i^2R$ .

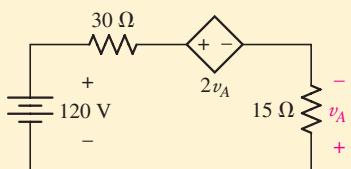
### PRACTICE

3.5 In the circuit of Fig. 3.12b,  $v_{s1} = 120$  V,  $v_{s2} = 30$  V,  $R_1 = 30$   $\Omega$ , and  $R_2 = 15$   $\Omega$ . Compute the power absorbed by each element.

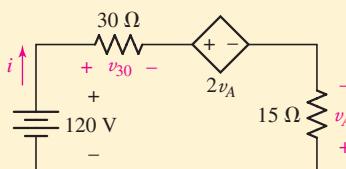
Ans:  $p_{120V} = -240$  W;  $p_{30V} = +60$  W;  $p_{30\Omega} = 120$  W;  $p_{15\Omega} = 60$  W.

### EXAMPLE 3.5

Compute the power absorbed in each element for the circuit shown in Fig. 3.13a.



(a)



(b)

FIGURE 3.13 (a) A single-loop circuit containing a dependent source. (b) The current  $i$  and voltage  $v_{30}$  are assigned.

(Continued on next page)

We first assign a reference direction for the current  $i$  and a reference polarity for the voltage  $v_{30}$  as shown in Fig. 3.13b. There is no need to assign a voltage to the  $15\ \Omega$  resistor, since the controlling voltage  $v_A$  for the dependent source is already available. (It is worth noting, however, that the reference signs for  $v_A$  are reversed from those we would have assigned based on the passive sign convention.)

This circuit contains a dependent voltage source, the value of which remains unknown until we determine  $v_A$ . However, its algebraic value  $2v_A$  can be used in the same fashion as if a numerical value were available. Thus, applying KVL around the loop:

$$-120 + v_{30} + 2v_A - v_A = 0 \quad [7]$$

Using Ohm's law to introduce the known resistor values:

$$v_{30} = 30i \quad \text{and} \quad v_A = -15i$$

Note that the negative sign is required since  $i$  flows into the negative terminal of  $v_A$ .

Substituting into Eq. [7] yields

$$-120 + 30i - 30i + 15i = 0$$

and so we find that

$$i = 8 \text{ A}$$

Computing the power *absorbed* by each element:

$$\begin{aligned} p_{120V} &= (120)(-8) = -960 \text{ W} \\ p_{30\Omega} &= (8)^2(30) = 1920 \text{ W} \\ p_{\text{dep}} &= (2v_A)(8) = 2[(-15)(8)](8) \\ &= -1920 \text{ W} \\ p_{15\Omega} &= (8)^2(15) = 960 \text{ W} \end{aligned}$$

### PRACTICE

3.6 In the circuit of Fig. 3.14, find the power absorbed by each of the five elements in the circuit.

Ans: (CW from left) 0.768 W, 1.92 W, 0.2048 W, 0.1792 W, -3.072 W.

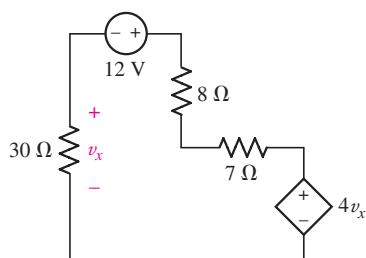
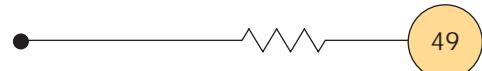


FIGURE 3.14 A simple loop circuit.

In the preceding example and practice problem, we were asked to compute the power absorbed by each element of a circuit. It is difficult to think of a situation, however, in which *all* of the absorbed power quantities of a circuit would be positive, for the simple reason that the energy must come from somewhere. Thus, from simple conservation of energy, we expect that ***the sum of the absorbed power for each element of a circuit should be zero***. In





other words, at least one of the quantities should be negative (neglecting the trivial case where the circuit is not operating). Stated another way, the sum of the supplied power for each element should be zero. More pragmatically, ***the sum of the absorbed power equals the sum of the supplied power***, which seems reasonable enough at face value.

Let's test this with the circuit of Fig. 3.13 from Example 3.5, which consists of two sources (one dependent and one independent) and two resistors. Adding the power absorbed by each element, we find

$$\sum_{\text{all elements}} p_{\text{absorbed}} = -960 + 1920 - 1920 + 960 = 0$$

In reality (our indication is the sign associated with the absorbed power) the 120 V source *supplies* +960 W, and the dependent source supplies +1920 W. Thus, the sources supply a total of  $960 + 1920 = 2880$  W. The resistors are expected to absorb positive power, which in this case sums to a total of  $1920 + 960 = 2880$  W. Thus, if we take into account each element of the circuit,

$$\sum p_{\text{absorbed}} = \sum p_{\text{supplied}}$$

as we expect.

Turning our attention to Practice Problem 3.6, the solution to which the reader might want to verify, we see that the absorbed powers sum to  $0.768 + 1.92 + 0.2048 + 0.1792 - 3.072 = 0$ . Interestingly enough, the 12 V independent voltage source is absorbing +1.92 W, which means it is *dissipating* power, not supplying it. Instead, the dependent voltage source appears to be supplying all the power in this particular circuit. Is such a thing possible? We usually expect a source to supply positive power, but since we are employing idealized sources in our circuits, it is in fact possible to have a net power flow into any source. If the circuit is changed in some way, the same source might then be found to supply positive power. The result is not known until a circuit analysis has been completed.

### 3.5 THE SINGLE-NODE-PAIR CIRCUIT

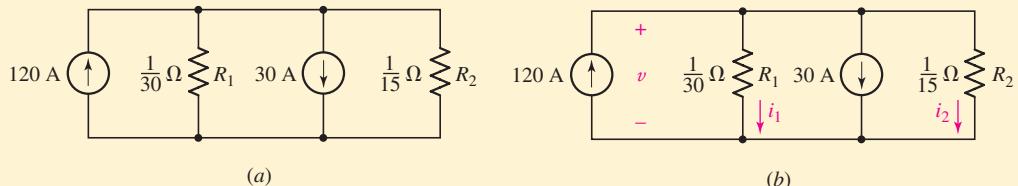
The companion of the single-loop circuit discussed in Sec. 3.4 is the single-node-pair circuit, in which any number of simple elements are connected between the same pair of nodes. An example of such a circuit is shown in Fig. 3.15a. KVL forces us to recognize that the voltage across each branch is the same as that across any other branch. *Elements in a circuit having a common voltage across them are said to be connected in parallel*.

#### EXAMPLE 3.6

**Find the voltage, current, and power associated with each element in the circuit of Fig. 3.15a.**

We first define a voltage  $v$  and arbitrarily select its polarity as shown in Fig. 3.15b. Two currents, flowing in the resistors, are selected in conformance with the passive sign convention, as shown in Fig. 3.15b.

(Continued on next page)



■ FIGURE 3.15 (a) A single-node-pair circuit. (b) A voltage and two currents are assigned.

Determining either current  $i_1$  or  $i_2$  will enable us to obtain a value for  $v$ . Thus, our next step is to apply KCL to either of the two nodes in the circuit. Equating the algebraic sum of the currents leaving the upper node to zero:

$$-120 + i_1 + 30 + i_2 = 0$$

Writing both currents in terms of the voltage  $v$  using Ohm's law

$$i_1 = 30v \quad \text{and} \quad i_2 = 15v$$

we obtain

$$-120 + 30v + 30 + 15v = 0$$

Solving this equation for  $v$  results in

$$v = 2 \text{ V}$$

and invoking Ohm's law then gives

$$i_1 = 60 \text{ A} \quad \text{and} \quad i_2 = 30 \text{ A}$$

The absorbed power in each element can now be computed. In the two resistors,

$$p_{R1} = 30(2)^2 = 120 \text{ W} \quad \text{and} \quad p_{R2} = 15(2)^2 = 60 \text{ W}$$

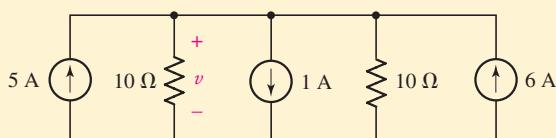
and for the two sources,

$$p_{120\text{A}} = 120(-2) = -240 \text{ W} \quad \text{and} \quad p_{30\text{A}} = 30(2) = 60 \text{ W}$$

Since the 120 A source absorbs negative 240 W, it is actually *supplying* power to the other elements in the circuit. In a similar fashion, we find that the 30 A source is actually *absorbing* power rather than *supplying* it.

### PRACTICE

3.7 Determine  $v$  in the circuit of Fig. 3.16.



■ FIGURE 3.16

Ans: 50 V.

## EXAMPLE 3.7

Determine the value of  $v$  and the power supplied by the independent current source in Fig. 3.17.

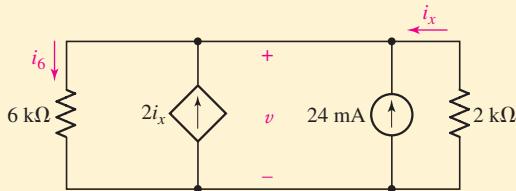


FIGURE 3.17 A voltage  $v$  and a current  $i_6$  are assigned in a single-node-pair circuit containing a dependent source.

By KCL, the sum of the currents leaving the upper node must be zero, so that

$$i_6 - 2i_x - 0.024 - i_x = 0$$

Again, note that the value of the dependent source ( $2i_x$ ) is treated the same as any other current would be, even though its exact value is not known until the circuit has been analyzed.

We next apply Ohm's law to each resistor:

$$i_6 = \frac{v}{6000} \quad \text{and} \quad i_x = \frac{-v}{2000}$$

Therefore,

$$\frac{v}{6000} - 2\left(\frac{-v}{2000}\right) - 0.024 - \left(\frac{-v}{2000}\right) = 0$$

and so  $v = (600)(0.024) = 14.4$  V.

Any other information we may want to find for this circuit is now easily obtained, usually in a single step. For example, the power supplied by the independent source is  $p_{24} = 14.4(0.024) = 0.3456$  W (345.6 mW).

## PRACTICE

3.8 For the single-node-pair circuit of Fig. 3.18, find  $i_A$ ,  $i_B$ , and  $i_C$ .

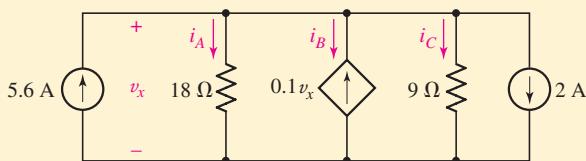
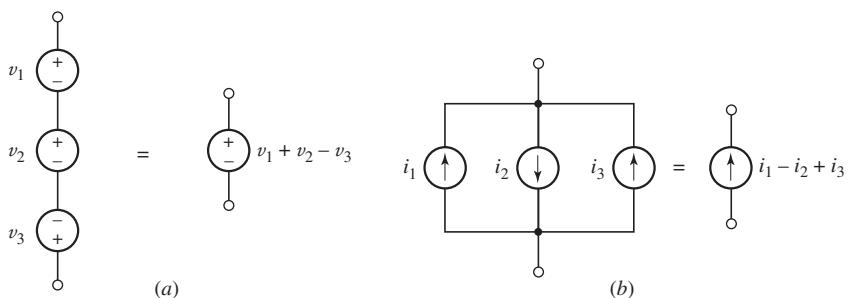


FIGURE 3.18

Ans: 3 A; -5.4 A; 6 A.

## 3.6 SERIES AND PARALLEL CONNECTED SOURCES

It turns out that some of the equation writing that we have been doing for series and parallel circuits can be avoided by combining sources. Note, however, that all the current, voltage, and power relationships in the remainder of the circuit will be unchanged. For example, several voltage



■ FIGURE 3.19 (a) Series-connected voltage sources can be replaced by a single source. (b) Parallel current sources can be replaced by a single source.

sources in series may be replaced by an equivalent voltage source having a voltage equal to the algebraic sum of the individual sources (Fig. 3.19a). Parallel current sources may also be combined by algebraically adding the individual currents, and the order of the parallel elements may be rearranged as desired (Fig. 3.19b).

### EXAMPLE 3.8

Determine the current  $i$  in the circuit of Fig. 3.20a by first combining the sources into a single equivalent voltage source.

To be able to combine the voltage sources, they must be in series. Since the same current ( $i$ ) flows through each, this condition is satisfied.

Starting from the bottom left-hand corner and proceeding clockwise,

$$-3 - 9 - 5 + 1 = -16 \text{ V}$$

so we may replace the four voltage sources with a single 16 V source having its negative reference as shown in Fig. 3.20b.

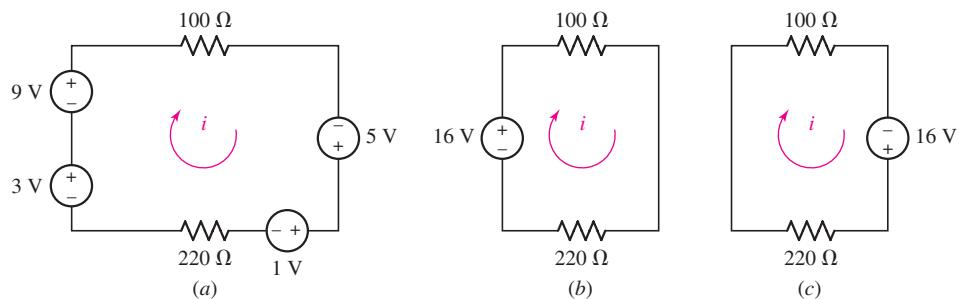
KVL combined with Ohm's law then yields

$$-16 + 100i + 220i = 0$$

or

$$i = \frac{16}{320} = 50 \text{ mA}$$

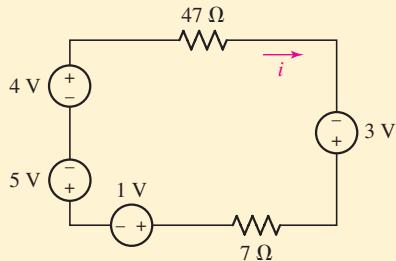
We should note that the circuit in Fig. 3.20c is also equivalent, a fact easily verified by computing  $i$ .



■ FIGURE 3.20

**PRACTICE**

3.9 Determine the current  $i$  in the circuit of Fig. 3.21 after first replacing the four sources with a single equivalent source.



■ FIGURE 3.21

Ans:  $-54\text{ A}$ .

**EXAMPLE 3.9**

Determine the voltage  $v$  in the circuit of Fig. 3.22a by first combining the sources into a single equivalent current source.

The sources may be combined if the same voltage appears across each one, which we can easily verify is the case. Thus, we create a new source, arrow pointing upward into the top node, by adding the currents that flow into that node:

$$2.5 - 2.5 - 3 = -3\text{ A}$$

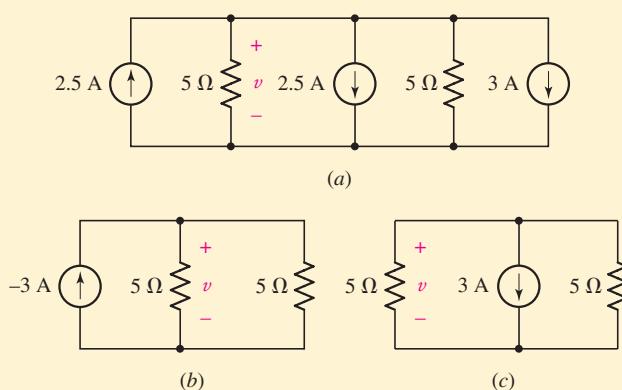
One equivalent circuit is shown in Fig. 3.22b.

KCL then allows us to write

$$-3 + \frac{v}{5} + \frac{v}{5} = 0$$

Solving, we find  $v = 7.5\text{ V}$ .

Another equivalent circuit is shown in Fig. 3.22c.

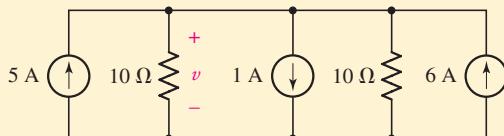


■ FIGURE 3.22

(Continued on next page)

**PRACTICE**

3.10 Determine the voltage  $v$  in the circuit of Fig. 3.23 after first replacing the three sources with a single equivalent source.



■ FIGURE 3.23

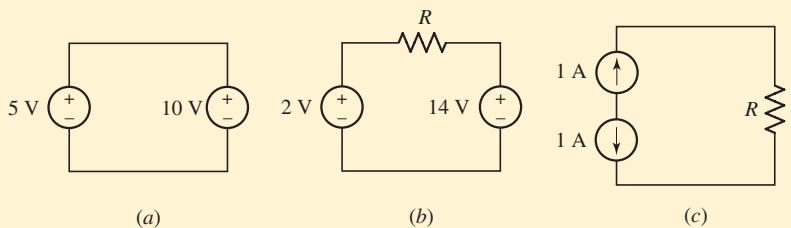
Ans: 50 V.

To conclude the discussion of parallel and series source combinations, we should consider the parallel combination of two voltage sources and the series combination of two current sources. For instance, what is the equivalent of a 5 V source in parallel with a 10 V source? By the definition of a voltage source, the voltage across the source cannot change; by Kirchhoff's voltage law, then, 5 equals 10 and we have hypothesized a physical impossibility. Thus, *ideal* voltage sources in parallel are permissible only when each has the same terminal voltage at every instant. In a similar way, two current sources may not be placed in series unless each has the same current, including sign, for every instant of time.

**EXAMPLE 3.10**

Determine which of the circuits of Fig. 3.24 are valid.

The circuit of Fig. 3.24a consists of two voltage sources in parallel. The value of each source is different, so this circuit violates KVL. For example, if a resistor is placed in parallel with the 5 V source, it is also in parallel with the 10 V source. The actual voltage across it is therefore ambiguous, and clearly the circuit cannot be constructed as indicated. If we attempt to build such a circuit in real life, we will find it impossible to locate “ideal” voltage sources—all real-world sources have an internal resistance. The presence of such resistance allows a voltage difference between the two *real* sources. Along these lines, the circuit of Fig. 3.24b is perfectly valid.

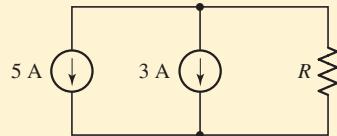


■ FIGURE 3.24 (a) to (c) Examples of circuits with multiple sources, some of which violate Kirchhoff's laws.

The circuit of Fig. 3.24c violates KCL: it is unclear what current actually flows through the resistor  $R$ .

### PRACTICE

- 3.11 Determine whether the circuit of Fig. 3.25 violates either of Kirchhoff's laws.



■ FIGURE 3.25

Ans: No. If the resistor were removed, however, the resulting circuit would.

## 3.7 RESISTORS IN SERIES AND PARALLEL

It is often possible to replace relatively complicated resistor combinations with a single equivalent resistor. This is useful when we are not specifically interested in the current, voltage, or power associated with any of the individual resistors in the combinations. *All the current, voltage, and power relationships in the remainder of the circuit will be unchanged.*

Consider the series combination of  $N$  resistors shown in Fig. 3.26a. We want to simplify the circuit with replacing the  $N$  resistors with a single resistor  $R_{\text{eq}}$  so that the remainder of the circuit, in this case only the voltage source, does not realize that any change has been made. The current, voltage, and power of the source must be the same before and after the replacement.

First, apply KVL:

$$v_s = v_1 + v_2 + \cdots + v_N$$

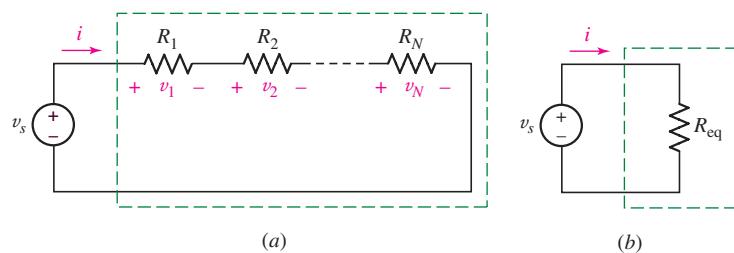
and then Ohm's law:

$$v_s = R_1 i + R_2 i + \cdots + R_N i = (R_1 + R_2 + \cdots + R_N) i$$

Now compare this result with the simple equation applying to the equivalent circuit shown in Fig. 3.26b:

$$v_s = R_{\text{eq}} i$$

**Helpful Tip:** Inspection of the KVL equation for any series circuit will show that the order in which elements are placed in such a circuit makes no difference.



■ FIGURE 3.26 (a) Series combination of  $N$  resistors. (b) Electrically equivalent circuit.

Thus, the value of the equivalent resistance for  $N$  series resistors is

$$R_{\text{eq}} = R_1 + R_2 + \cdots + R_N \quad [8]$$

We are therefore able to replace a two-terminal network consisting of  $N$  series resistors with a single two-terminal element  $R_{\text{eq}}$  that has the same  $v-i$  relationship.

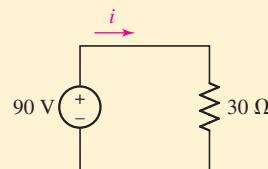
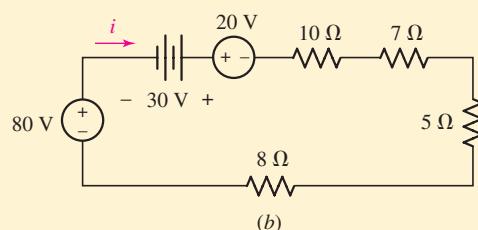
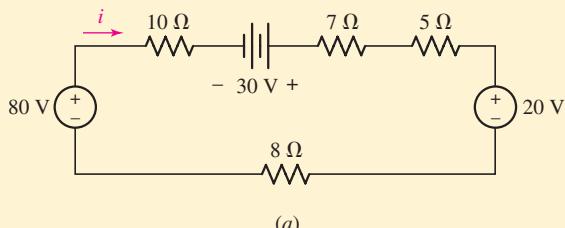


It should be emphasized again that we might be interested in the current, voltage, or power of one of the original elements. For example, the voltage of a dependent voltage source may depend upon the voltage across  $R_3$ . Once  $R_3$  is combined with several series resistors to form an equivalent resistance, then it is gone and the voltage across it cannot be determined until  $R_3$  is identified by removing it from the combination. In that case, it would have been better to look ahead and not make  $R_3$  a part of the combination initially.

### EXAMPLE 3.11

**Use resistance and source combinations to determine the current  $i$  in Fig. 3.27a and the power delivered by the 80 V source.**

We first interchange the element positions in the circuit, being careful to preserve the proper sense of the sources, as shown in Fig. 3.27b. The



(c)

■ FIGURE 3.27 (a) A series circuit with several sources and resistors.  
(b) The elements are rearranged for the sake of clarity. (c) A simpler equivalent.

next step is to then combine the three voltage sources into an equivalent 90 V source, and the four resistors into an equivalent 30  $\Omega$  resistance, as in Fig. 3.27c. Thus, instead of writing

$$-80 + 10i - 30 + 7i + 5i + 20 + 8i = 0$$

we have simply

$$-90 + 30i = 0$$

and so we find that

$$i = 3 \text{ A}$$

In order to calculate the power delivered to the circuit by the 80 V source appearing in the given circuit, it is necessary to return to Fig. 3.27a with the knowledge that the current is 3 A. The desired power is then  $80 \text{ V} \times 3 \text{ A} = 240 \text{ W}$ .

It is interesting to note that no element of the original circuit remains in the equivalent circuit.

### PRACTICE

3.12 Determine  $i$  in the circuit of Fig. 3.28.

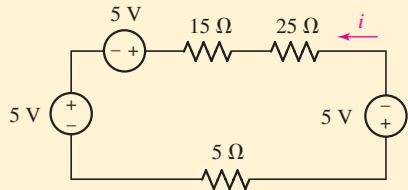


FIGURE 3.28

Ans:  $-333 \text{ mA}$ .

Similar simplifications can be applied to parallel circuits. A circuit containing  $N$  resistors in parallel, as in Fig. 3.29a, leads to the KCL equation

$$i_s = i_1 + i_2 + \dots + i_N$$

or

$$\begin{aligned} i_s &= \frac{v}{R_1} + \frac{v}{R_2} + \dots + \frac{v}{R_N} \\ &= \frac{v}{R_{\text{eq}}} \end{aligned}$$

Thus,

$$\frac{1}{R_{\text{eq}}} = \frac{1}{R_1} + \frac{1}{R_2} + \dots + \frac{1}{R_N} \quad [9]$$

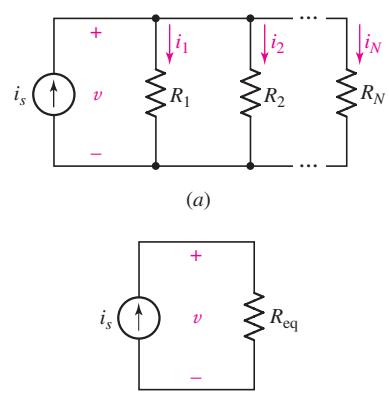


FIGURE 3.29 (a) A circuit with  $N$  resistors in parallel. (b) Equivalent circuit.

which can be written as

$$R_{\text{eq}}^{-1} = R_1^{-1} + R_2^{-1} + \cdots + R_N^{-1}$$

or, in terms of conductances, as

$$G_{\text{eq}} = G_1 + G_2 + \cdots + G_N$$

The simplified (equivalent) circuit is shown in Fig. 3.29b.

A parallel combination is routinely indicated by the following shorthand notation:

$$R_{\text{eq}} = R_1 \parallel R_2 \parallel R_3$$

The special case of only two parallel resistors is encountered fairly often, and is given by

$$\begin{aligned} R_{\text{eq}} &= R_1 \parallel R_2 \\ &= \frac{1}{\frac{1}{R_1} + \frac{1}{R_2}} \end{aligned}$$

Or, more simply,

$$R_{\text{eq}} = \frac{R_1 R_2}{R_1 + R_2}$$

[10]



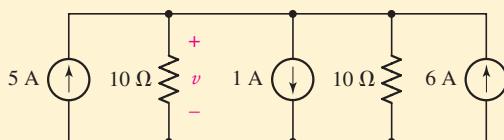
The last form is worth memorizing, although it is a common error to attempt to generalize Eq. [10] to more than two resistors, e.g.,

$$R_{\text{eq}} \not\propto \frac{R_1 R_2 R_3}{R_1 + R_2 + R_3}$$

A quick look at the units of this equation will immediately show that the expression cannot possibly be correct.

### PRACTICE

3.13 Determine  $v$  in the circuit of Fig. 3.30 by first combining the three current sources, and then the two  $10 \Omega$  resistors.

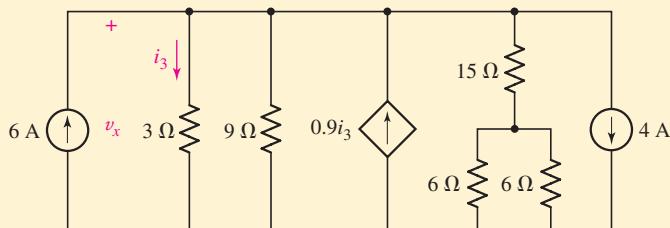


■ FIGURE 3.30

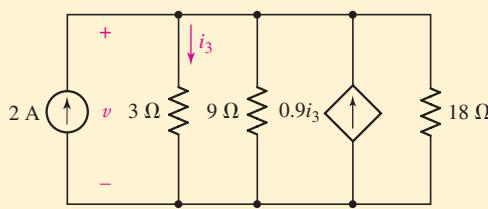
Ans: 50 V.

## EXAMPLE 3.12

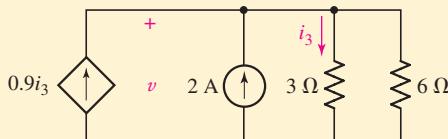
Calculate the power and voltage of the dependent source in Fig. 3.31a.



(a)



(b)



(c)

FIGURE 3.31 (a) A multinode circuit. (b) The two independent current sources are combined into a 2 A source, and the  $15 \Omega$  resistor in series with the two parallel  $6 \Omega$  resistors are replaced with a single  $18 \Omega$  resistor. (c) A simplified equivalent circuit.

We will seek to simplify the circuit before analyzing it, but take care not to include the dependent source since its voltage and power characteristics are of interest.

Despite not being drawn adjacent to one another, the two independent current sources are in fact in parallel, so we replace them with a 2 A source.

The two  $6 \Omega$  resistors are in parallel and can be replaced with a single  $3 \Omega$  resistor in series with the  $15 \Omega$  resistor. Thus, the two  $6 \Omega$  resistors and the  $15 \Omega$  resistor are replaced by an  $18 \Omega$  resistor (Fig. 3.31b).

No matter how tempting, *we should not combine the remaining three resistors*; the controlling variable  $i_3$  depends on the  $3 \Omega$  resistor and so that resistor must remain untouched. The only further simplification, then, is  $9 \Omega \parallel 18 \Omega = 6 \Omega$ , as shown in Fig. 3.31c.



(Continued on next page)

Applying KCL at the top node of Fig. 3.31c, we have

$$-0.9i_3 - 2 + i_3 + \frac{v}{6} = 0$$

Employing Ohm's law,

$$v = 3i_3$$

which allows us to compute

$$i_3 = \frac{10}{3} \text{ A}$$

Thus, the voltage across the dependent source (which is the same as the voltage across the  $3\ \Omega$  resistor) is

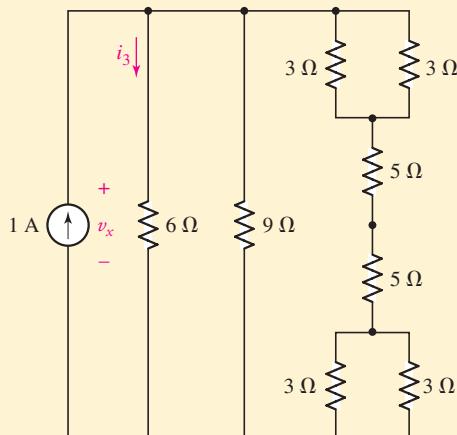
$$v = 3i_3 = 10 \text{ V}$$

The dependent source therefore furnishes  $v \times 0.9i_3 = 10(0.9)(10/3) = 30 \text{ W}$  to the remainder of the circuit.

Now if we are later asked for the power dissipated in the  $15\ \Omega$  resistor, we must return to the original circuit. This resistor is in series with an equivalent  $3\ \Omega$  resistor; a voltage of  $10 \text{ V}$  is across the  $18\ \Omega$  total; therefore, a current of  $5/9 \text{ A}$  flows through the  $15\ \Omega$  resistor and the power absorbed by this element is  $(5/9)^2(15)$  or  $4.63 \text{ W}$ .

### PRACTICE

3.14 For the circuit of Fig. 3.32, calculate the voltage  $v_x$ .



■ FIGURE 3.32

Ans: 2.819 V.

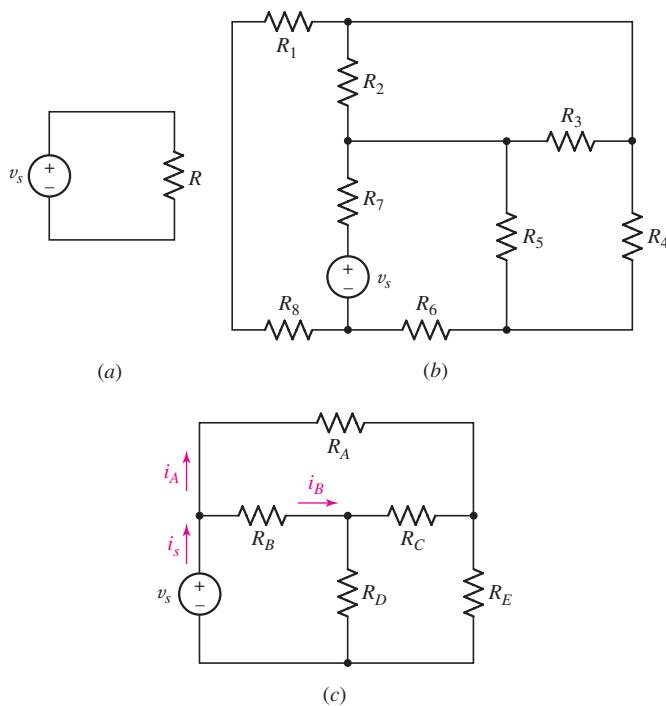


FIGURE 3.33 These two circuit elements are both in series and in parallel. (b)  $R_2$  and  $R_3$  are in parallel, and  $R_1$  and  $R_8$  are in series. (c) There are no circuit elements either in series or in parallel with one another.

Three final comments on series and parallel combinations might be helpful. The first is illustrated by referring to Fig. 3.33a and asking, “Are  $v_s$  and  $R$  in series or in parallel?” The answer is “Both.” The two elements carry the same current and are therefore in series; they also enjoy the same voltage and consequently are in parallel.

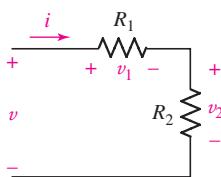
The second comment is a word of caution. Circuits can be drawn in such a way as to make series or parallel combinations difficult to spot. In Fig. 3.33b, for example, the only two resistors in parallel are  $R_2$  and  $R_3$ , while the only two in series are  $R_1$  and  $R_8$ .

The final comment is simply that a simple circuit element need not be in series or parallel with any other simple circuit element in a circuit. For example,  $R_4$  and  $R_5$  in Fig. 3.33b are not in series or parallel with any other simple circuit element, and there are no simple circuit elements in Fig. 3.33c that are in series or parallel with any other simple circuit element. In other words, we cannot simplify that circuit further using any of the techniques discussed in this chapter.



### 3.8 VOLTAGE AND CURRENT DIVISION

By combining resistances and sources, we have found one method of shortening the work of analyzing a circuit. Another useful shortcut is the application of the ideas of voltage and current division. Voltage division is used to express the voltage across one of several series resistors in terms of the



■ FIGURE 3.34 An illustration of voltage division.

voltage across the combination. In Fig. 3.34, the voltage across  $R_2$  is found via KVL and Ohm's law:

$$v = v_1 + v_2 = iR_1 + iR_2 = i(R_1 + R_2)$$

so

$$i = \frac{v}{R_1 + R_2}$$

Thus,

$$v_2 = iR_2 = \left( \frac{v}{R_1 + R_2} \right) R_2$$

or

$$v_2 = \frac{R_2}{R_1 + R_2} v$$

and the voltage across  $R_1$  is, similarly,

$$v_1 = \frac{R_1}{R_1 + R_2} v$$

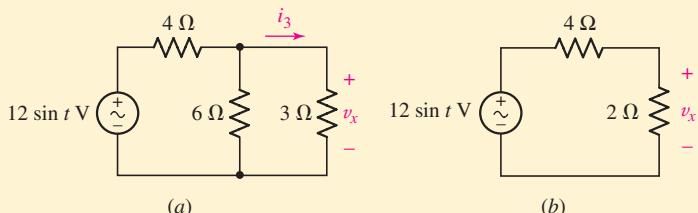
If the network of Fig. 3.34 is generalized by removing  $R_2$  and replacing it with the series combination of  $R_2, R_3, \dots, R_N$ , then we have the general result for voltage division across a string of  $N$  series resistors

$$v_k = \frac{R_k}{R_1 + R_2 + \dots + R_N} v \quad [11]$$

which allows us to compute the voltage  $v_k$  that appears across an arbitrary resistor  $R_k$  of the series.

### EXAMPLE 3.13

Determine  $v_x$  in the circuit of Fig. 3.35a.



■ FIGURE 3.35 A numerical example illustrating resistance combination and voltage division. (a) Original circuit. (b) Simplified circuit.

We first combine the  $6\ \Omega$  and  $3\ \Omega$  resistors, replacing them with  $(6)(3)/(6 + 3) = 2\ \Omega$ .

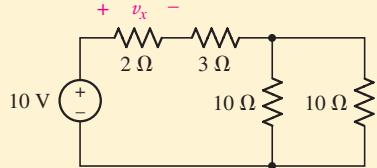
Since  $v_x$  appears across the parallel combination, our simplification has not lost this quantity. However, further simplification of the circuit by replacing the series combination of the  $4\ \Omega$  resistor with our new  $2\ \Omega$  resistor would.

Thus, we proceed by simply applying voltage division to the circuit in Fig. 3.35b:

$$v_x = (12 \sin t) \frac{2}{4+2} = 4 \sin t \quad \text{volts}$$

### PRACTICE

3.15 Use voltage division to determine  $v_x$  in the circuit of Fig. 3.36.



■ FIGURE 3.36

Ans: 2 V.

The dual<sup>2</sup> of voltage division is current division. We are now given a total current supplied to several parallel resistors, as shown in the circuit of Fig. 3.37.

The current flowing through  $R_2$  is

$$i_2 = \frac{v}{R_2} = \frac{i(R_1 \parallel R_2)}{R_2} = \frac{i}{R_2} \frac{R_1 R_2}{R_1 + R_2}$$

or

$$i_2 = i \frac{R_1}{R_1 + R_2} \quad [12]$$

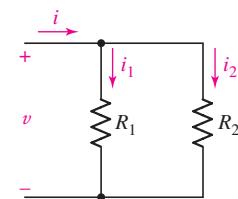
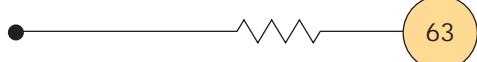
and, similarly,

$$i_1 = i \frac{R_2}{R_1 + R_2} \quad [13]$$

*Nature has not smiled on us here, for these last two equations have a factor which differs subtly from the factor used with voltage division, and some effort is going to be needed to avoid errors.* Many students look on the expression for voltage division as “obvious” and that for current division as being “different.” It helps to realize that the larger of two parallel resistors always carries the smaller current.

For a parallel combination of  $N$  resistors, the current through resistor  $R_k$  is

$$i_k = i \frac{\frac{1}{R_k}}{\frac{1}{R_1} + \frac{1}{R_2} + \cdots + \frac{1}{R_N}} \quad [14]$$



■ FIGURE 3.37 An illustration of current division.



(2) The principle of duality is encountered often in engineering. We will consider the topic briefly in Chap. 7 when we compare inductors and capacitors.

Written in terms of conductances,

$$i_k = i \cdot \frac{G_k}{G_1 + G_2 + \dots + G_N}$$

which strongly resembles Eq. [11] for voltage division.

### EXAMPLE 3.14

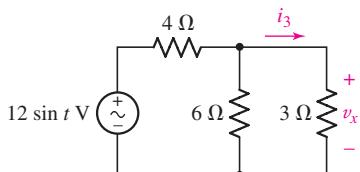


FIGURE 3.38 A circuit used as an example of current division. The wavy line in the voltage source symbol indicates a sinusoidal variation with time.

**Write an expression for the current through the  $3\ \Omega$  resistor in the circuit of Fig. 3.38.**

The total current flowing into the  $3\ \Omega$ - $6\ \Omega$  combination is

$$i(t) = \frac{12 \sin t}{4 + 3\parallel 6} = \frac{12 \sin t}{4 + 2} = 2 \sin t \quad \text{A}$$

and thus the desired current is given by current division:

$$i_3(t) = (2 \sin t) \left( \frac{6}{6+3} \right) = \frac{4}{3} \sin t \quad \text{A}$$

Unfortunately, current division is sometimes applied when it is not applicable. As one example, let us consider again the circuit shown in Fig. 3.33c, a circuit that we have already agreed contains no circuit elements that are in series or in parallel. Without parallel resistors, there is no way that current division can be applied. Even so, there are too many students who take a quick look at resistors  $R_A$  and  $R_B$  and try to apply current division, writing an incorrect equation such as

$$i_A \not\propto i_S \frac{R_B}{R_A + R_B}$$

Remember, *parallel resistors must be branches between the same pair of nodes.*

### PRACTICE

3.16 In the circuit of Fig. 3.39, use resistance combination methods and current division to find  $i_1$ ,  $i_2$ , and  $v_3$ .

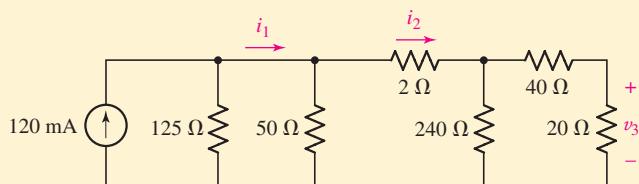


FIGURE 3.39

Ans: 100 mA; 50 mA; 0.8 V.

## PRACTICAL APPLICATION

### Not the Earth Ground from Geology

Up to now, we have been drawing circuit schematics in a fashion similar to that of the one shown in Fig. 3.40, where voltages are defined across two clearly marked terminals. Special care was taken to emphasize the fact that voltage cannot be defined at a single point—it is by definition the *difference* in potential between *two* points. However, many schematics make use of the convention of taking the earth as defining zero volts, so that all other voltages are implicitly referenced to this potential. The concept is often referred to as ***earth ground***, and is fundamentally tied to safety regulations designed to prevent fires, fatal electrical shocks, and related mayhem. The symbol for earth ground is shown in Fig. 3.41a.

Since earth ground is defined as zero volts, it is often convenient to use this as a common terminal in schematics. The circuit of Fig. 3.40 is shown redrawn in this fashion in Fig. 3.42, where the earth ground symbol represents a common node. It is important to note that the two circuits are equivalent in terms of our value for  $v_a$  (4.5 V in either case), but are no longer exactly the same. The circuit in Fig. 3.40 is said to be “floating” in that it could for all practical purposes be installed on a circuit board of a satellite in geosynchronous orbit (or on its way to Pluto). The circuit in Fig. 3.42, however, is somehow physically connected to the ground through a conducting path. For this reason, there are two other symbols that are occasionally used to denote a common terminal. Figure 3.41b shows what is commonly referred to as ***signal ground***; there can be (and often is) a large voltage between earth ground and any terminal tied to signal ground.

The fact that the common terminal of a circuit may or may not be connected by some low-resistance pathway to earth ground can lead to potentially dangerous situations. Consider the diagram of Fig. 3.43a, which depicts an innocent bystander about to touch a piece of equipment powered by an ac outlet. Only two terminals have been used from the wall socket; the round ground pin

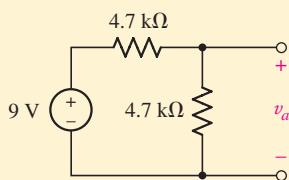


FIGURE 3.40 A simple circuit with a voltage  $v_a$  defined between two terminals.

of the receptacle was left unconnected. The common terminal of every circuit in the equipment has been tied together and electrically connected to the conducting equipment chassis; this terminal is often denoted using the ***chassis ground*** symbol of Fig. 3.41c. Unfortunately, a wiring fault exists, due to either poor manufacturing or perhaps just wear and tear. At any rate, the chassis is not “grounded,” so there is a very large resistance between chassis ground and earth ground. A pseudo-schematic (some liberty was taken with the person’s equivalent resistance symbol) of the situation is shown in Fig. 3.43b. The electrical path between the conducting chassis and ground may in fact be the table, which could represent a resistance of hundreds of megaohms or more. The resistance of the person, however, is many orders of magnitude lower. Once the person taps on the equipment to see why it isn’t working properly . . . well, let’s just say not all stories have happy endings.

The fact that “ground” is not always “earth ground” can cause a wide range of safety and electrical noise problems. One example is occasionally encountered in older buildings, where plumbing originally consisted of electrically conducting copper pipes. In such buildings, any water pipe was often treated as a low-resistance path to earth ground, and therefore used in many electrical connections. However, when corroded pipes are replaced with more modern and cost-effective

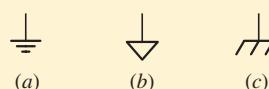


FIGURE 3.41 Three different symbols used to represent a ground or common terminal: (a) earth ground; (b) signal ground; (c) chassis ground.

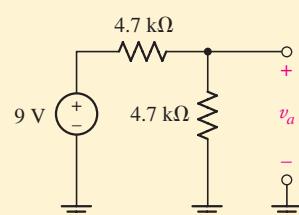


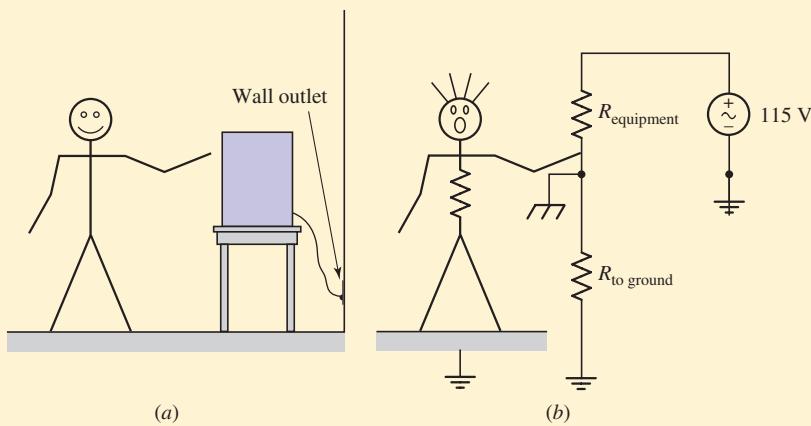
FIGURE 3.42 The circuit of Fig. 3.40, redrawn using the earth ground symbol. The rightmost ground symbol is redundant; it is only necessary to label the positive terminal of  $v_a$ ; the negative reference is then implicitly ground, or zero volts.

(Continued on next page)

nonconducting PVC piping, the low-resistance path to earth ground no longer exists. A related problem occurs when the composition of the earth varies greatly over a particular region. In such situations, it is possible to actually have two separated buildings in which the two

“earth grounds” are not equal, and current can flow as a result.

Within this text, the earth ground symbol will be used exclusively. It is worth remembering, however, that not all grounds are created equal in practice.

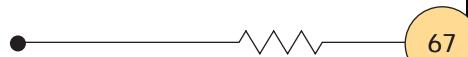


■ FIGURE 3.43 (a) A sketch of an innocent person about to touch an improperly grounded piece of equipment. It's not going to be pretty. (b) A schematic of an equivalent circuit for the situation as it is about to unfold; the person has been represented by an equivalent resistance, as has the equipment. A resistor has been used to represent the nonhuman path to ground.

## SUMMARY AND REVIEW

We began this chapter by discussing connections of circuit elements, and introducing the terms *node*, *path*, *loop*, and *branch*. The next two topics could be considered the two most important in the entire textbook, namely, Kirchhoff's current law (KCL) and Kirchhoff's voltage law. The first is derived from conservation of charge, and can be thought of in terms of “what goes in (current) must come out.” The second is based on conservation of energy, and can be viewed as “what goes up (potential) must come down.” These two laws allow us to analyze *any* circuit, linear or otherwise, provided we have a way of relating the voltage and current associated with passive elements (e.g., Ohm's law for the resistor). In the case of a single-loop circuit, the elements are connected in *series* and hence each carries the same current. The single-node-pair circuit, in which elements are connected in *parallel* with one another, is characterized by a single voltage common to each element. Extending these concepts allowed us to develop a means of simplifying voltage sources connected in series, or current sources in parallel; subsequently we obtained classic expressions for series and parallel connected resistors. The final topic, that of voltage and current division, finds considerable use in the design of circuits where a specific voltage or current is required but our choice of source is limited.

Let's conclude with key points of this chapter to review, highlighting appropriate examples.



- Kirchhoff's current law (KCL) states that the algebraic sum of the currents entering any node is zero. (Examples 3.1, 3.4)
- Kirchhoff's voltage law (KVL) states that the algebraic sum of the voltages around any closed path in a circuit is zero. (Examples 3.2, 3.3)
- All elements in a circuit that carry the same current are said to be connected in series. (Example 3.5)
- Elements in a circuit having a common voltage across them are said to be connected in parallel. (Examples 3.6, 3.7)
- Voltage sources in series can be replaced by a single source, provided care is taken to note the individual polarity of each source. (Examples 3.8, 3.10)
- Current sources in parallel can be replaced by a single source, provided care is taken to note the direction of each current arrow. (Examples 3.9, 3.10)
- A series combination of  $N$  resistors can be replaced by a single resistor having the value  $R_{\text{eq}} = R_1 + R_2 + \dots + R_N$ . (Example 3.11)
- A parallel combination of  $N$  resistors can be replaced by a single resistor having the value

$$\frac{1}{R_{\text{eq}}} = \frac{1}{R_1} + \frac{1}{R_2} + \dots + \frac{1}{R_N}$$

(Example 3.12)

- Voltage division allows us to calculate what fraction of the total voltage across a series string of resistors is dropped across any one resistor (or group of resistors). (Example 3.13)
- Current division allows us to calculate what fraction of the total current into a parallel string of resistors flows through any one of the resistors. (Example 3.14)

## READING FURTHER

A discussion of the principles of conservation of energy and conservation of charge, as well as Kirchhoff's laws, can be found in

R. Feynman, R. B. Leighton, and M. L. Sands, *The Feynman Lectures on Physics*. Reading, Mass.: Addison-Wesley, 1989, pp. 4-1, 4-7, and 25-9.

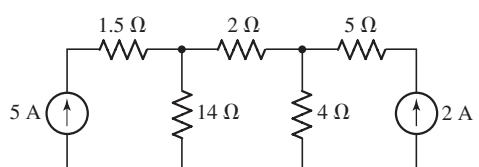
Detailed discussions of numerous aspects of grounding practices consistent with the 2008 National Electrical Code® can be found throughout

J. E. McPartland, B. J. McPartland, and F. P. Hartwell, *McGraw-Hill's National Electrical Code® 2008 Handbook*, 26th ed. New York, McGraw-Hill, 2008.

## EXERCISES

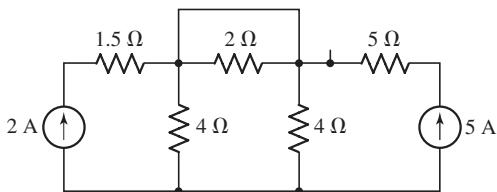
### 3.1 Nodes, Paths, Loops, and Branches

1. Referring to the circuit depicted in Fig. 3.44, count the number of (a) nodes; (b) elements; (c) branches.

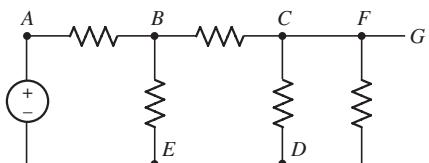


■ FIGURE 3.44

2. Referring to the circuit depicted in Fig. 3.45, count the number of (a) nodes; (b) elements; (c) branches.



■ FIGURE 3.45



■ FIGURE 3.46

3. For the circuit of Fig. 3.46:

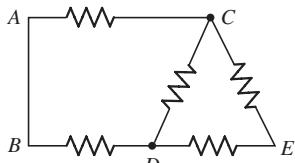
- (a) Count the number of nodes.
- (b) In moving from A to B, have we formed a path? Have we formed a loop?
- (c) In moving from C to F to G, have we formed a path? Have we formed a loop?

4. For the circuit of Fig. 3.46:

- (a) Count the number of circuit elements.
- (b) If we move from B to C to D, have we formed a path? Have we formed a loop?
- (c) If we move from E to D to C to B, have we formed a path? Have we formed a loop?

5. Refer to the circuit of Fig. 3.47, and answer the following:

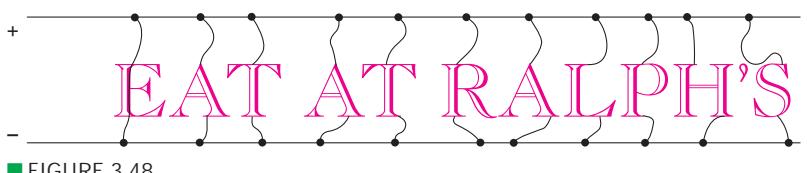
- (a) How many distinct nodes are contained in the circuit?
- (b) How many elements are contained in the circuit?
- (c) How many branches does the circuit have?
- (d) Determine if each of the following represents a path, a loop, both, or neither:
  - (i) A to B
  - (ii) B to D to C to E
  - (iii) C to E to D to B to A to C
  - (iv) C to D to B to A to C to E



■ FIGURE 3.47

### 3.2 Kirchhoff's Current Law

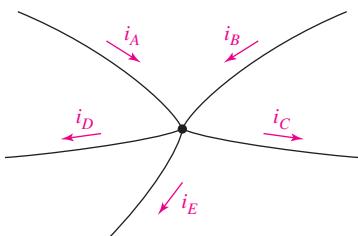
6. A local restaurant has a neon sign constructed from 12 separate bulbs; when a bulb fails, it appears as an infinite resistance and cannot conduct current. In wiring the sign, the manufacturer offers two options (Fig. 3.48). From what you've learned about KCL, which one should the restaurant owner select? Explain.



■ FIGURE 3.48

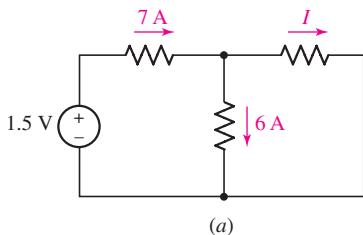
7. Referring to the single node diagram of Fig. 3.49, compute:

- $i_B$ , if  $i_A = 1 \text{ A}$ ,  $i_D = -2 \text{ A}$ ,  $i_C = 3 \text{ A}$ , and  $i_E = 0$ ;
- $i_E$ , if  $i_A = -1 \text{ A}$ ,  $i_B = -1 \text{ A}$ ,  $i_C = -1 \text{ A}$ , and  $i_D = -1 \text{ A}$ .

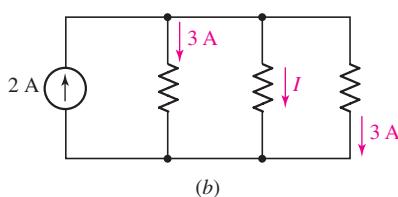


■ FIGURE 3.49

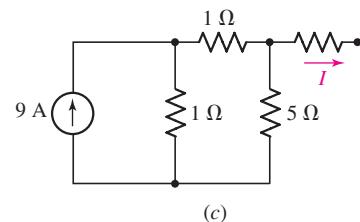
8. Determine the current labeled  $I$  in each of the circuits of Fig. 3.50.



(a)



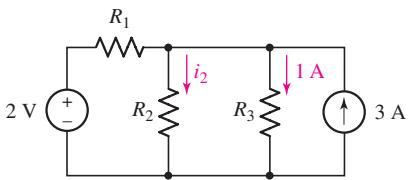
(b)



(c)

■ FIGURE 3.50

9. In the circuit shown in Fig. 3.51, the resistor values are unknown, but the 2 V source is known to be supplying a current of 7 A to the rest of the circuit. Calculate the current labeled  $i_2$ .

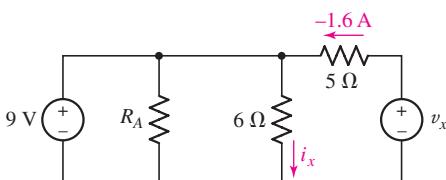


■ FIGURE 3.51

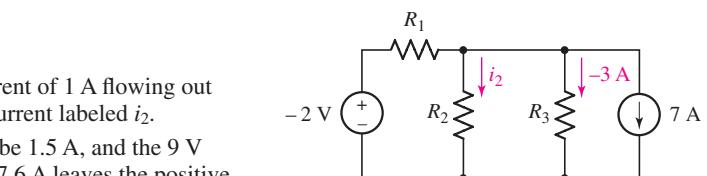
10. The voltage source in the circuit of Fig. 3.52 has a current of 1 A flowing out of its positive terminal into resistor  $R_1$ . Calculate the current labeled  $i_2$ .

11. In the circuit depicted in Fig. 3.53,  $i_x$  is determined to be 1.5 A, and the 9 V source supplies a current of 7.6 A (that is, a current of 7.6 A leaves the positive reference terminal of the 9 V source). Determine the value of resistor  $R_A$ .

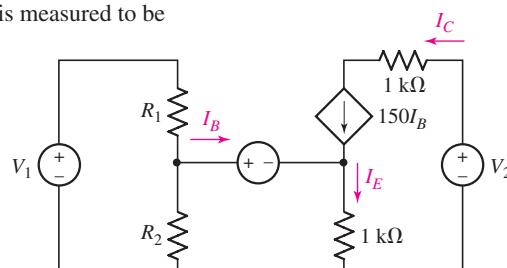
12. For the circuit of Fig. 3.54 (which is a model for the dc operation of a bipolar junction transistor biased in forward active region),  $I_B$  is measured to be 100  $\mu\text{A}$ . Determine  $I_C$  and  $I_E$ .



■ FIGURE 3.53

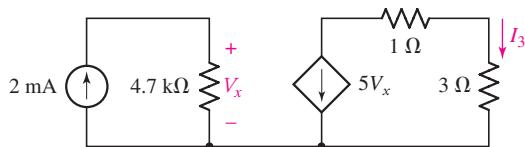


■ FIGURE 3.52



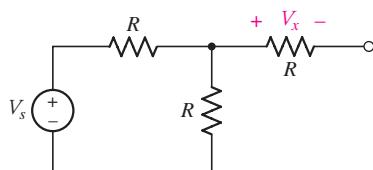
■ FIGURE 3.54

13. Determine the current labeled  $I_3$  in the circuit of Fig. 3.55.



■ FIGURE 3.55

14. Study the circuit depicted in Fig. 3.56, and explain (in terms of KCL) why the voltage labeled  $V_x$  must be zero.



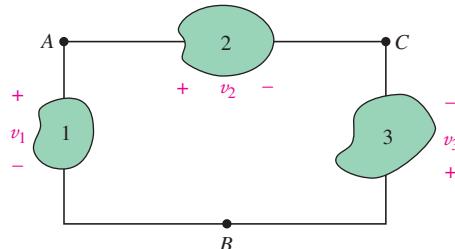
■ FIGURE 3.56

15. In many households, multiple electrical outlets within a given room are often all part of the same circuit. Draw the circuit for a four-walled room which has a single electrical outlet per wall, with a lamp (represented by a  $1 \Omega$  resistor) connected to each outlet.

### 3.3 Kirchoff's Voltage Law

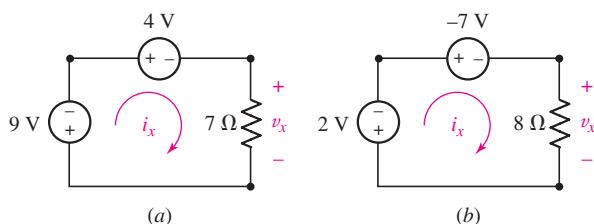
16. For the circuit of Fig. 3.57:

- Determine the voltage  $v_1$  if  $v_2 = 0$  and  $v_3 = -17 \text{ V}$ .
- Determine the voltage  $v_1$  if  $v_2 = -2 \text{ V}$  and  $v_3 = +2 \text{ V}$ .
- Determine the voltage  $v_2$  if  $v_1 = 7 \text{ V}$  and  $v_3 = 9 \text{ V}$ .
- Determine the voltage  $v_3$  if  $v_1 = -2.33 \text{ V}$  and  $v_2 = -1.70 \text{ V}$ .



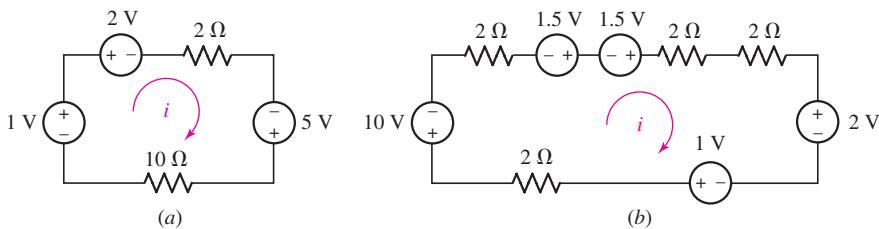
■ FIGURE 3.57

17. For each of the circuits in Fig. 3.58, determine the voltage  $v_x$  and the current  $i_x$ .



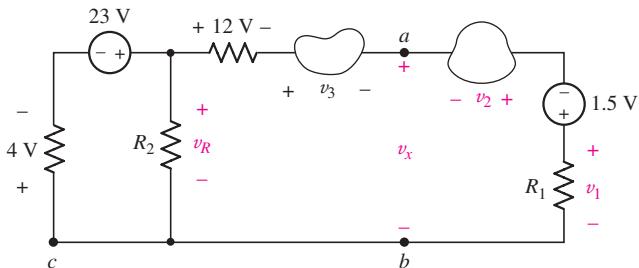
■ FIGURE 3.58

18. Use KVL to obtain a numerical value for the current labeled  $i$  in each circuit depicted in Fig. 3.59.



■ FIGURE 3.59

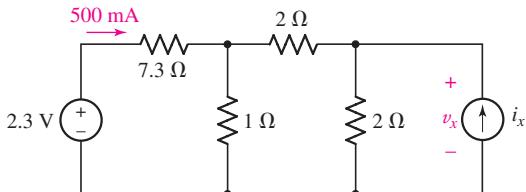
19. In the circuit of Fig. 3.60, it is determined that  $v_1 = 3 \text{ V}$  and  $v_3 = 1.5 \text{ V}$ . Calculate  $v_R$  and  $v_2$ .



■ FIGURE 3.60

20. In the circuit of Fig. 3.60, a voltmeter is used to measure the following:  $v_1 = 2 \text{ V}$  and  $v_3 = -1.5 \text{ V}$ . Calculate  $v_x$ .

21. Determine the value of  $v_x$  as labeled in the circuit of Fig. 3.61.

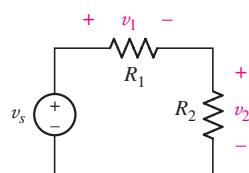


■ FIGURE 3.61

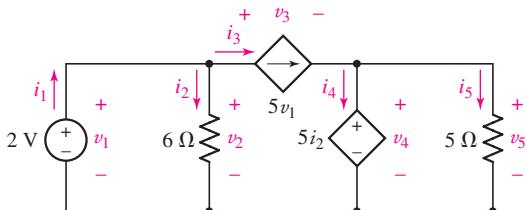
22. Consider the simple circuit shown in Fig. 3.62. Using KVL, derive the expressions

$$v_1 = v_s \frac{R_1}{R_1 + R_2} \quad \text{and} \quad v_2 = v_s \frac{R_2}{R_1 + R_2}$$

23. (a) Determine a numerical value for each current and voltage ( $i_1$ ,  $v_1$ , etc.) in the circuit of Fig. 3.63. (b) Calculate the power absorbed by each element and verify that they sum to zero.



■ FIGURE 3.62



■ FIGURE 3.63

24. The circuit shown in Fig. 3.64 includes a device known as an op amp. This device has two unusual properties in the circuit shown: (1)  $V_d = 0$  V, and (2) no current can flow into either input terminal (marked “ $-$ ” and “ $+$ ” inside the symbol), but it *can* flow through the output terminal (marked “OUT”). This seemingly impossible situation—in direct conflict with KCL—is a result of power leads to the device that are not included in the symbol. Based on this information, calculate  $V_{out}$ . (*Hint:* two KVL equations are required, both involving the 5 V source.)

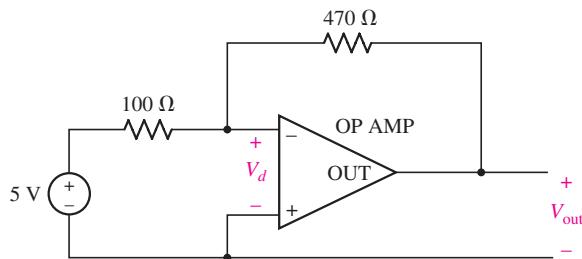


FIGURE 3.64

### 3.4 The Single-Loop Circuit

25. The circuit of Fig. 3.12b is constructed with the following:  $v_{s1} = -8$  V,  $R_1 = 1 \Omega$ ,  $v_{s2} = 16$  V, and  $R_2 = 4.7 \Omega$ . Calculate the power absorbed by each element. Verify that the absorbed powers sum to zero.
26. Obtain a numerical value for the power absorbed by each element in the circuit shown in Fig. 3.65.

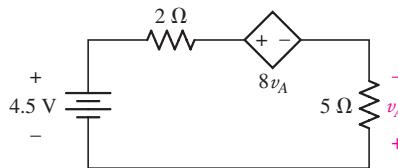


FIGURE 3.65

27. Compute the power absorbed by each element of the circuit of Fig. 3.66.
28. Compute the power absorbed by each element in the circuit of Fig. 3.67 if the mysterious element  $X$  is (a) a  $13 \Omega$  resistor; (b) a dependent voltage source labeled  $4v_1$ , “ $+$ ” reference on top; (c) a dependent voltage source labeled  $4i_x$ , “ $+$ ” reference on top.

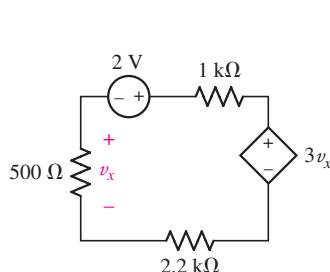


FIGURE 3.66

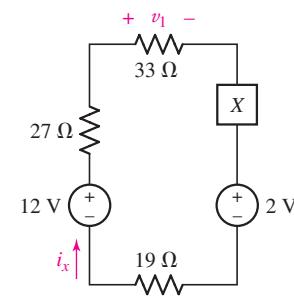


FIGURE 3.67

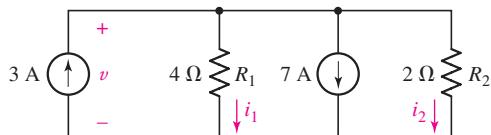
29. Kirchhoff's laws apply whether or not Ohm's law applies to a particular element. The  $I$ - $V$  characteristic of a diode, for example, is given by

$$I_D = I_S (e^{V_D/V_T} - 1)$$

where  $V_T = 27 \text{ mV}$  at room temperature and  $I_S$  can vary from  $10^{-12}$  to  $10^{-3} \text{ A}$ . In the circuit of Fig. 3.68, use KVL/KCL to obtain  $V_D$  if  $I_S = 29 \text{ pA}$ . (Note: This problem results in a transcendental equation, requiring an iterative approach to obtaining a numerical solution. Most scientific calculators will perform such a function.)

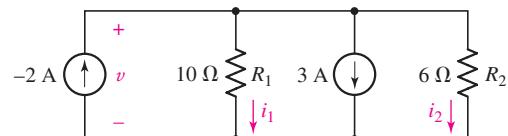
### 3.5 The Single-Node-Pair Circuit

30. Referring to the circuit of Fig. 3.69, (a) determine the two currents  $i_1$  and  $i_2$ ; (b) compute the power absorbed by each element.



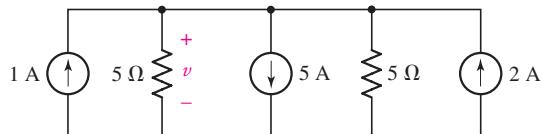
■ FIGURE 3.69

31. Determine a value for the voltage  $v$  as labeled in the circuit of Fig. 3.70, and compute the power supplied by the two current sources.



■ FIGURE 3.70

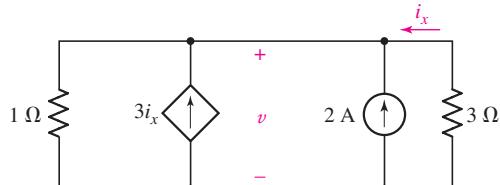
32. Referring to the circuit depicted in Fig. 3.71, determine the value of the voltage  $v$ .



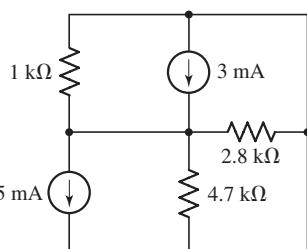
■ FIGURE 3.71

33. Determine the voltage  $v$  as labeled in Fig. 3.72, and calculate the power supplied by each current source.

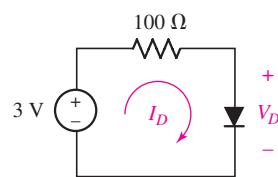
34. Although drawn so that it may not appear obvious at first glance, the circuit of Fig. 3.73 is in fact a single-node-pair circuit. (a) Determine the power absorbed by each resistor. (b) Determine the power supplied by each current source. (c) Show that the sum of the absorbed power calculated in (a) is equal to the sum of the supplied power calculated in (c).



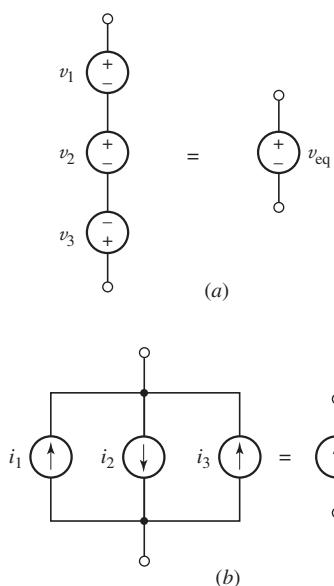
■ FIGURE 3.72



■ FIGURE 3.73



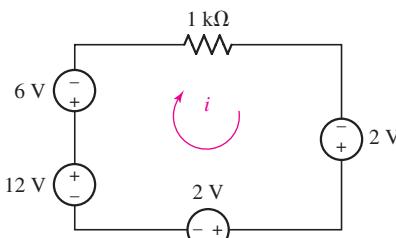
■ FIGURE 3.68



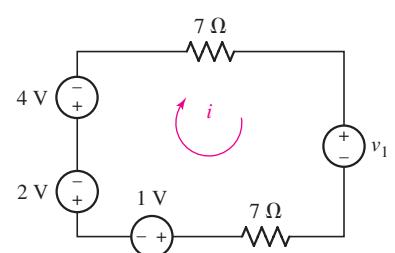
■ FIGURE 3.74

### 3.6 Series and Parallel Connected Sources

35. Determine the numerical value for  $v_{eq}$  in Fig. 3.74a, if (a)  $v_1 = 0$ ,  $v_2 = -3$  V, and  $v_3 = +3$  V; (b)  $v_1 = v_2 = v_3 = 1$  V; (c)  $v_1 = -9$  V,  $v_2 = 4.5$  V,  $v_3 = 1$  V.
36. Determine the numerical value for  $i_{eq}$  in Fig. 3.74b, if (a)  $i_1 = 0$ ,  $i_2 = -3$  A, and  $i_3 = +3$  A; (b)  $i_1 = i_2 = i_3 = 1$  A; (c)  $i_1 = -9$  A,  $i_2 = 4.5$  A,  $i_3 = 1$  A.
37. For the circuit presented in Fig. 3.75, determine the current labeled  $i$  by first combining the four sources into a single equivalent source.
38. Determine the value of  $v_1$  required to obtain a zero value for the current labeled  $i$  in the circuit of Fig. 3.76.

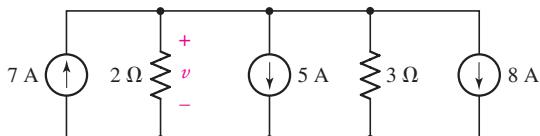


■ FIGURE 3.75



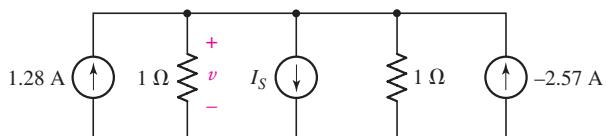
■ FIGURE 3.76

39. (a) For the circuit of Fig. 3.77, determine the value for the voltage labeled  $v$ , after first simplifying the circuit to a single current source in parallel with two resistors. (b) Verify that the power supplied by your equivalent source is equal to the sum of the supplied powers of the individual sources in the original circuit.



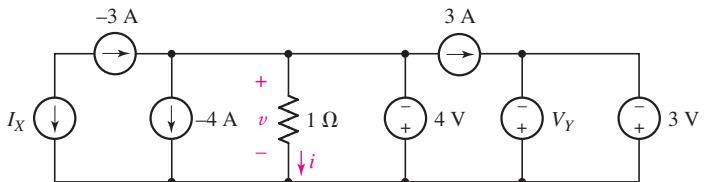
■ FIGURE 3.77

40. What value of  $I_S$  in the circuit of Fig. 3.78 will result in a zero voltage  $v$ ?



■ FIGURE 3.78

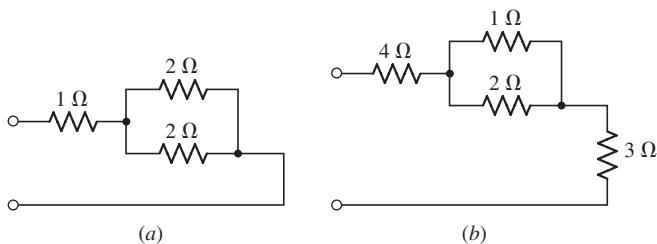
41. (a) Determine the values for  $I_X$  and  $V_Y$  in the circuit shown in Fig. 3.79.  
(b) Are those values necessarily unique for that circuit? Explain. (c) Simplify the circuit of Fig. 3.79 as much as possible and still maintain the values for  $v$  and  $i$ . (Your circuit must contain the 1 Ω resistor.)



■ FIGURE 3.79

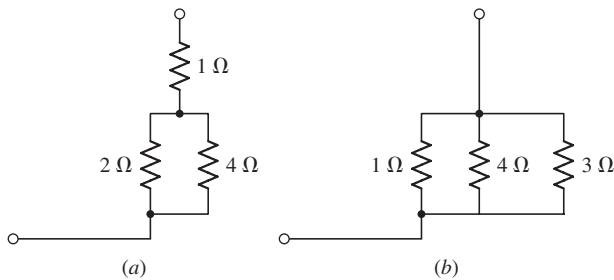
### 3.7 Resistors in Series and Parallel

42. Determine the equivalent resistance of each of the networks shown in Fig. 3.80.



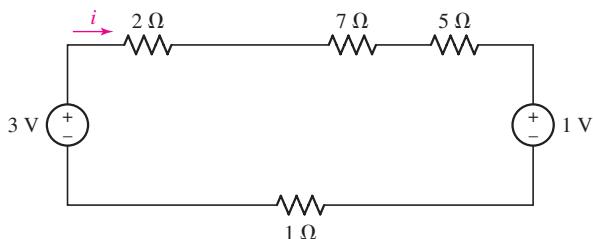
■ FIGURE 3.80

43. For each network depicted in Fig. 3.81, determine a single equivalent resistance.



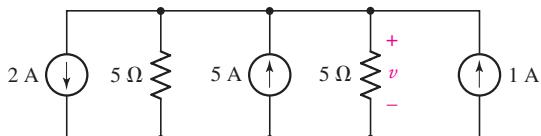
■ FIGURE 3.81

44. (a) Simplify the circuit of Fig. 3.82 as much as possible by using source and resistor combinations. (b) Calculate  $i$ , using your simplified circuit. (c) To what voltage should the 1 V source be changed to reduce  $i$  to zero? (d) Calculate the power absorbed by the 5 Ω resistor.



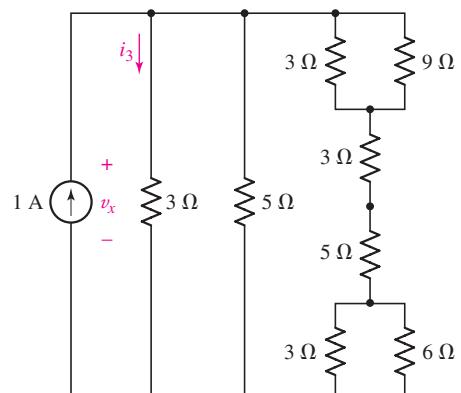
■ FIGURE 3.82

45. (a) Simplify the circuit of Fig. 3.83, using appropriate source and resistor combinations. (b) Determine the voltage labeled  $v$ , using your simplified circuit. (c) Calculate the power provided by the 2 A source to the rest of the circuit.



■ FIGURE 3.83

46. Making appropriate use of resistor combination techniques, calculate  $i_3$  in the circuit of Fig. 3.84 and the power provided to the circuit by the single current source.



■ FIGURE 3.84

47. Calculate the voltage labeled  $v_x$  in the circuit of Fig. 3.85 after first simplifying, using appropriate source and resistor combinations.

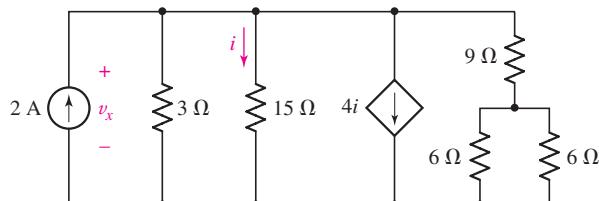


FIGURE 3.85

48. Determine the power absorbed by the  $15 \Omega$  resistor in the circuit of Fig. 3.86.

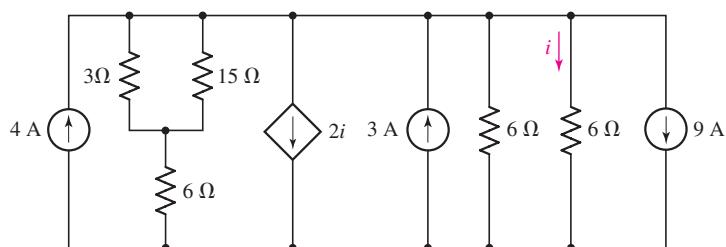


FIGURE 3.86

49. Calculate the equivalent resistance  $R_{eq}$  of the network shown in Fig. 3.87 if  $R_1 = 2R_2 = 3R_3 = 4R_4$  etc. and  $R_{11} = 3 \Omega$ .

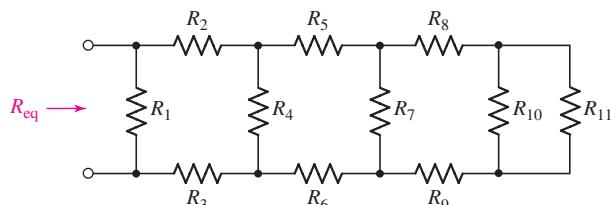


FIGURE 3.87

50. Show how to combine four  $100 \Omega$  resistors to obtain an equivalent resistance of (a)  $25 \Omega$ ; (b)  $60 \Omega$ ; (c)  $40 \Omega$ .

### 3.8 Voltage and Current Division

51. In the voltage divider network of Fig. 3.88, calculate (a)  $v_2$  if  $v = 9.2 \text{ V}$  and  $v_1 = 3 \text{ V}$ ; (b)  $v_1$  if  $v_2 = 1 \text{ V}$  and  $v = 2 \text{ V}$ ; (c)  $v$  if  $v_1 = 3 \text{ V}$  and  $v_2 = 6 \text{ V}$ ; (d)  $R_1/R_2$  if  $v_1 = v_2$ ; (e)  $v_2$  if  $v = 3.5 \text{ V}$  and  $R_1 = 2R_2$ ; (f)  $v_1$  if  $v = 1.8 \text{ V}$ ,  $R_1 = 1 \text{ k}\Omega$ , and  $R_2 = 4.7 \text{ k}\Omega$ .

52. In the current divider network represented in Fig. 3.89, calculate (a)  $i_1$  if  $i = 8 \text{ A}$  and  $i_2 = 1 \text{ A}$ ; (b)  $v$  if  $R_1 = 100 \text{ k}\Omega$ ,  $R_2 = 100 \text{ k}\Omega$ , and  $i = 1 \text{ mA}$ ; (c)  $i_2$  if  $i = 20 \text{ mA}$ ,  $R_1 = 1 \Omega$ , and  $R_2 = 4 \Omega$ ; (d)  $i_1$  if  $i = 10 \text{ A}$ ,  $R_1 = R_2 = 9 \Omega$ ; (e)  $i_2$  if  $i = 10 \text{ A}$ ,  $R_1 = 100 \text{ M}\Omega$ , and  $R_2 = 1\Omega$ .

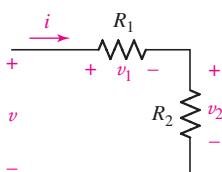


FIGURE 3.88

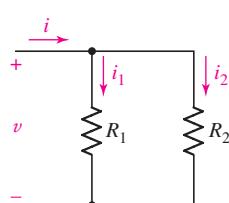
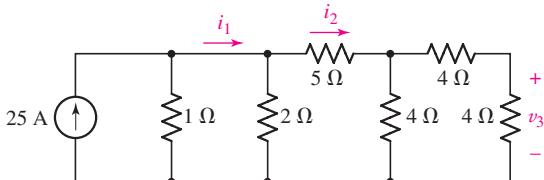
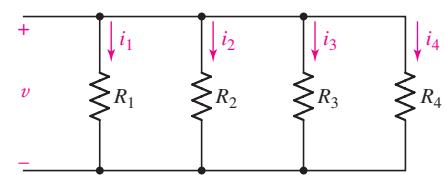


FIGURE 3.89

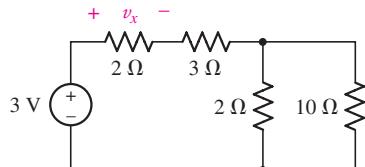
53. Choose a voltage  $v < 2.5$  V and values for the resistors  $R_1$ ,  $R_2$ ,  $R_3$ , and  $R_4$  in the circuit of Fig. 3.90 so that  $i_1 = 1$  A,  $i_2 = 1.2$  A,  $i_3 = 8$  A, and  $i_4 = 3.1$  A.
54. Employ voltage division to assist in the calculation of the voltage labeled  $v_x$  in the circuit of Fig. 3.91.
55. A network is constructed from a series connection of five resistors having values  $1\ \Omega$ ,  $3\ \Omega$ ,  $5\ \Omega$ ,  $7\ \Omega$ , and  $9\ \Omega$ . If 9 V is connected across the terminals of the network, employ voltage division to calculate the voltage across the  $3\ \Omega$  resistor, and the voltage across the  $7\ \Omega$  resistor.
56. Employing resistance combination and current division as appropriate, determine values for  $i_1$ ,  $i_2$ , and  $v_3$  in the circuit of Fig. 3.92.



■ FIGURE 3.92

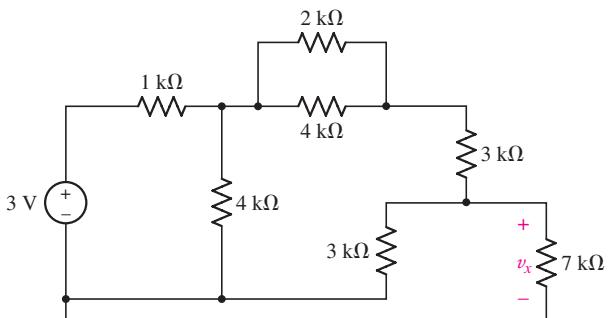


■ FIGURE 3.90



■ FIGURE 3.91

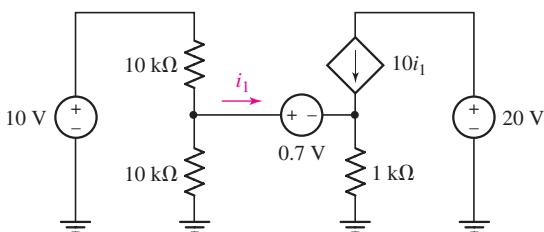
57. In the circuit of Fig. 3.93, only the voltage  $v_x$  is of interest. Simplify the circuit using appropriate resistor combinations and iteratively employ voltage division to determine  $v_x$ .



■ FIGURE 3.93

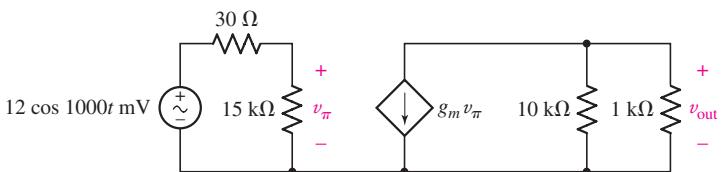
### Chapter-Integrating Exercises

58. The circuit shown in Fig. 3.94 is a linear model of a bipolar junction transistor biased in the forward active region of operation. Explain why voltage division is not a valid approach for determining the voltage across either  $10\text{ k}\Omega$  resistor.



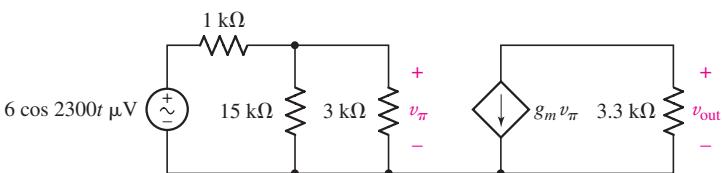
■ FIGURE 3.94

59. A common midfrequency model for a field effect-based amplifier circuit is shown in Fig. 3.95. If the controlling parameter  $g_m$  (known as the *transconductance*) is equal to  $1.2 \text{ mS}$ , employ current division to obtain the current through the  $1 \text{ k}\Omega$  resistor, and then calculate the amplifier output voltage  $v_{\text{out}}$ .



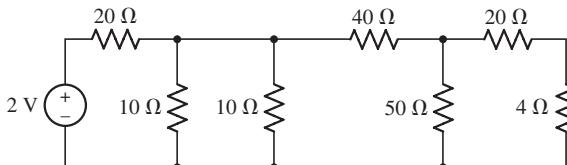
■ FIGURE 3.95

60. The circuit depicted in Fig. 3.96 is routinely employed to model the midfrequency operation of a bipolar junction transistor-based amplifier. Calculate the amplifier output  $v_{\text{out}}$  if the transconductance  $g_m$  is equal to  $322 \text{ mS}$ .



■ FIGURE 3.96

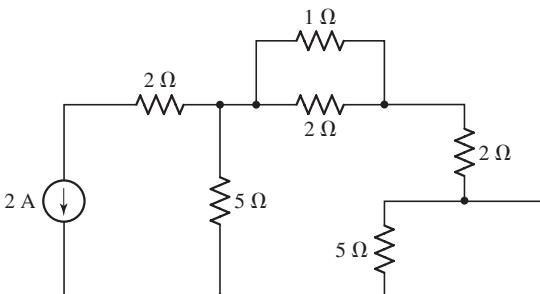
61. With regard to the circuit shown in Fig. 3.97, compute (a) the voltage across the two  $10 \Omega$  resistors, assuming the top terminal is the positive reference; (b) the power dissipated by the  $4 \Omega$  resistor.



■ FIGURE 3.97

62. Delete the leftmost  $10 \Omega$  resistor in the circuit of Fig. 3.97, and compute (a) the current flowing into the left-hand terminal of the  $40 \Omega$  resistor; (b) the power supplied by the  $2 \text{ V}$  source; (c) the power dissipated by the  $4 \Omega$  resistor.

63. Consider the seven-element circuit depicted in Fig. 3.98. (a) How many nodes, loops, and branches does it contain? (b) Calculate the current flowing through each resistor. (c) Determine the voltage across the current source, assuming the top terminal is the positive reference terminal.



■ FIGURE 3.98

# Basic Nodal and Mesh Analysis

## INTRODUCTION

Armed with the trio of Ohm's and Kirchhoff's laws, analyzing a simple linear circuit to obtain useful information such as the current, voltage, or power associated with a particular element is perhaps starting to seem a straightforward enough venture. Still, for the moment at least, every circuit seems unique, requiring (to some degree) a measure of creativity in approaching the analysis. In this chapter, we learn two basic circuit analysis techniques—***nodal analysis*** and ***mesh analysis***—both of which allow us to investigate many different circuits with a consistent, methodical approach. The result is a streamlined analysis, a more uniform level of complexity in our equations, fewer errors and, perhaps most importantly, a reduced occurrence of “*I don't know how to even start!*”

Most of the circuits we have seen up to now have been rather simple and (to be honest) of questionable practical use. Such circuits are valuable, however, in helping us to learn to apply fundamental techniques. Although the more complex circuits appearing in this chapter may represent a variety of electrical systems including control circuits, communication networks, motors, or integrated circuits, as well as electric circuit models of nonelectrical systems, we believe it best not to dwell on such specifics at this early stage. Rather, it is important to initially focus on the *methodology of problem solving* that we will continue to develop throughout the book.

## KEY CONCEPTS

Nodal Analysis

The Supernode Technique

Mesh Analysis

The Supermesh Technique

Choosing Between Nodal and Mesh Analysis

Computer-Aided Analysis, Including PSpice and MATLAB



## 4.1 NODAL ANALYSIS

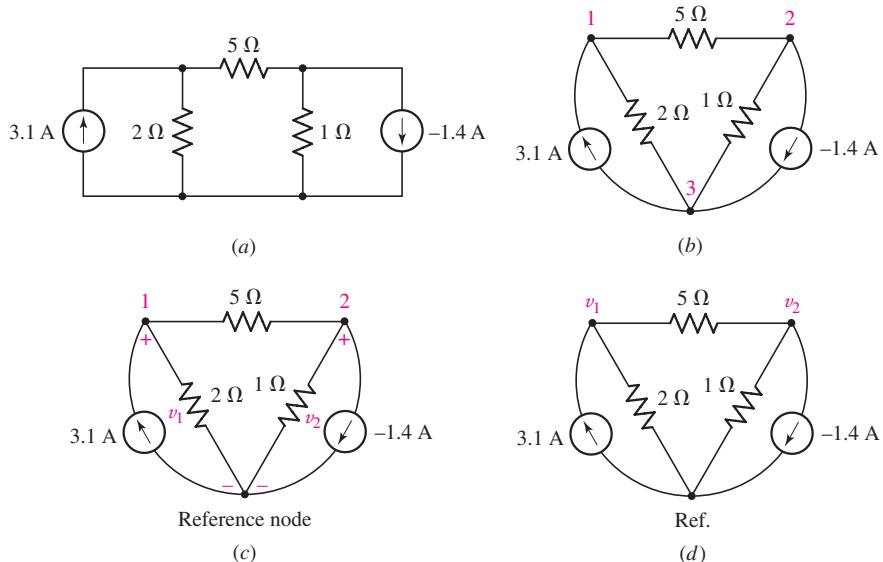
We begin our study of general methods for methodical circuit analysis by considering a powerful method based on KCL, namely ***nodal analysis***. In Chap. 3 we considered the analysis of a simple circuit containing only two nodes. We found that the major step of the analysis was obtaining a single equation in terms of a single unknown quantity—the voltage between the pair of nodes.

We will now let the number of nodes increase and correspondingly provide one additional unknown quantity and one additional equation for each added node. Thus, a three-node circuit should have two unknown voltages and two equations; a 10-node circuit will have nine unknown voltages and nine equations; an  $N$ -node circuit will need  $(N - 1)$  voltages and  $(N - 1)$  equations. Each equation is a simple KCL equation.

To illustrate the basic technique, consider the three-node circuit shown in Fig. 4.1a, redrawn in Fig. 4.1b to emphasize the fact that there are only three nodes, numbered accordingly. Our goal will be to determine the voltage across each element, and the next step in the analysis is critical. We designate one node as a ***reference node***; it will be the negative terminal of our  $N - 1 = 2$  nodal voltages, as shown in Fig. 4.1c.

A little simplification in the resultant equations is obtained if the node connected to the greatest number of branches is identified as the reference node. If there is a ground node, it is usually most convenient to select it as the reference node, although many people seem to prefer selecting the bottom node of a circuit as the reference, especially if no explicit ground is noted.

The voltage of node 1 *relative to the reference node* is named  $v_1$ , and  $v_2$  is defined as the voltage of node 2 with respect to the reference node. These



■ FIGURE 4.1 (a) A simple three-node circuit. (b) Circuit redrawn to emphasize nodes. (c) Reference node selected and voltages assigned. (d) Shorthand voltage references. If desired, an appropriate ground symbol may be substituted for "Ref."

two voltages are all we need, as the voltage between any other pair of nodes may be found in terms of them. For example, the voltage of node 1 with respect to node 2 is  $v_1 - v_2$ . The voltages  $v_1$  and  $v_2$  and their reference signs are shown in Fig. 4.1c. It is common practice once a reference node has been labeled to omit the reference signs for the sake of clarity; the node labeled with the voltage is taken to be the positive terminal (Fig. 4.1d). This is understood to be a type of shorthand voltage notation.

We now apply KCL to nodes 1 and 2. We do this by equating the total current leaving the node through the several resistors to the total source current entering the node. Thus,

$$\frac{v_1}{2} + \frac{v_1 - v_2}{5} = 3.1 \quad [1]$$

or

$$0.7v_1 - 0.2v_2 = 3.1 \quad [2]$$

At node 2 we obtain

$$\frac{v_2}{1} + \frac{v_2 - v_1}{5} = -(-1.4) \quad [3]$$

or

$$-0.2v_1 + 1.2v_2 = 1.4 \quad [4]$$

Equations [2] and [4] are the desired two equations in two unknowns, and they may be solved easily. The results are  $v_1 = 5$  V and  $v_2 = 2$  V.

From this, it is straightforward to determine the voltage across the  $5\ \Omega$  resistor:  $v_{5\Omega} = v_1 - v_2 = 3$  V. The currents and absorbed powers may also be computed in one step.

We should note at this point that there is more than one way to write the KCL equations for nodal analysis. For example, the reader may prefer to sum all the currents entering a given node and set this quantity to zero. Thus, for node 1 we might have written

$$3.1 - \frac{v_1}{2} - \frac{v_1 - v_2}{5} = 0$$

or

$$3.1 + \frac{-v_1}{2} + \frac{v_2 - v_1}{5} = 0$$

either of which is equivalent to Eq. [1].

*Is one way better than any other?* Every instructor and every student develop a personal preference, and at the end of the day the most important thing is to be consistent. The authors prefer constructing KCL equations for nodal analysis in such a way as to end up with all current source terms on one side and all resistor terms on the other. Specifically,

$$\begin{aligned} \sum \text{currents entering the node from current sources} \\ = \sum \text{currents leaving the node through resistors} \end{aligned}$$

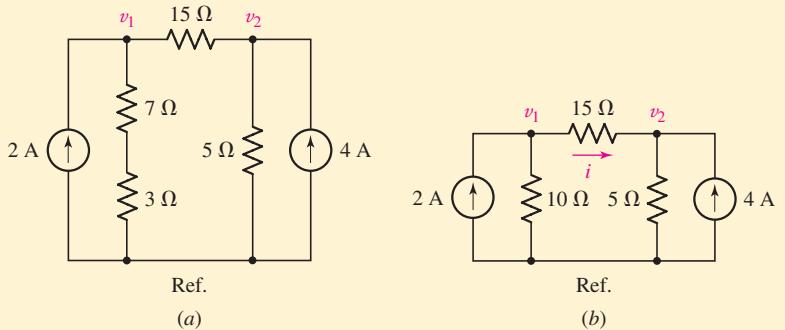
There are several advantages to such an approach. First, there is never any confusion regarding whether a term should be " $v_1 - v_2$ " or " $v_2 - v_1$ "; the

The reference node in a schematic is implicitly defined as zero volts. However, it is important to remember that any terminal can be designated as the reference terminal. Thus, the reference node is at zero volts with respect to the other defined nodal voltages, and not necessarily with respect to *earth* ground.

first voltage in every resistor current expression corresponds to the node for which a KCL equation is being written, as seen in Eqs. [1] and [3]. Second, it allows a quick check that a term has not been accidentally omitted. Simply count the current sources connected to a node and then the resistors; grouping them in the stated fashion makes the comparison a little easier.

### EXAMPLE 4.1

Determine the current flowing left to right through the  $15\ \Omega$  resistor of Fig. 4.2a.



■ FIGURE 4.2 (a) A four-node circuit containing two independent current sources. (b) The two resistors in series are replaced with a single  $10\ \Omega$  resistor, reducing the circuit to three nodes.

Nodal analysis will directly yield numerical values for the nodal voltages  $v_1$  and  $v_2$ , and the desired current is given by  $i = (v_1 - v_2)/15$ .

Before launching into nodal analysis, however, we first note that no details regarding either the  $7\ \Omega$  resistor or the  $3\ \Omega$  resistor are of interest. Thus, we may replace their series combination with a  $10\ \Omega$  resistor as in Fig. 4.2b. The result is a reduction in the number of equations to solve.

Writing an appropriate KCL equation for node 1,

$$2 = \frac{v_1}{10} + \frac{v_1 - v_2}{15} \quad [5]$$

and for node 2,

$$4 = \frac{v_2}{5} + \frac{v_2 - v_1}{15} \quad [6]$$

Rearranging, we obtain

$$5v_1 - 2v_2 = 60$$

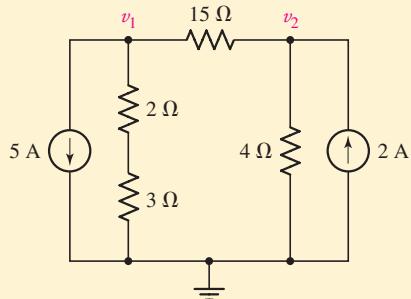
and

$$-v_1 + 4v_2 = 60$$

Solving, we find that  $v_1 = 20\text{ V}$  and  $v_2 = 20\text{ V}$  so that  $v_1 - v_2 = 0$ . In other words, **zero current** is flowing through the  $15\ \Omega$  resistor in this circuit!

**PRACTICE**

4.1 For the circuit of Fig. 4.3, determine the nodal voltages  $v_1$  and  $v_2$ .



■ FIGURE 4.3

Ans:  $v_1 = -145/8$  V,  $v_2 = 5/2$  V.

Now let us increase the number of nodes so that we may use this technique to work a slightly more difficult problem.

**EXAMPLE 4.2**

Determine the nodal voltages for the circuit of Fig. 4.4a, as referenced to the bottom node.

► **Identify the goal of the problem.**

There are four nodes in this circuit. With the bottom node as our reference, we label the other three nodes as shown in Fig. 4.4b. The circuit has been redrawn for clarity, taking care to identify the two relevant nodes for the  $4 \Omega$  resistor.

► **Collect the known information.**

We have three unknown voltages,  $v_1$ ,  $v_2$ , and  $v_3$ . All current sources and resistors have designated values, which are marked on the schematic.

► **Devise a plan.**

This problem is well suited to nodal analysis, as three independent KCL equations may be written in terms of the current sources and the current through each resistor.

► **Construct an appropriate set of equations.**

We begin by writing a KCL equation for node 1:

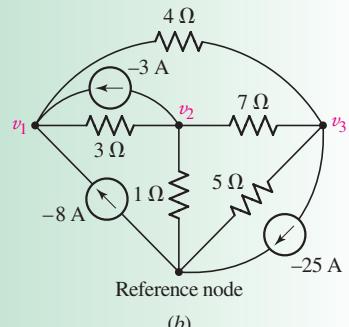
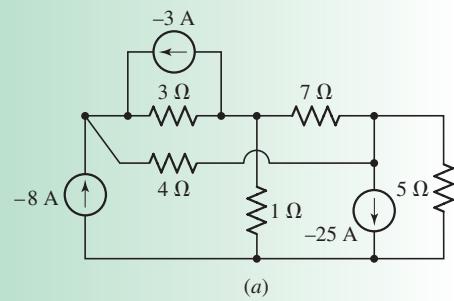
$$-8 - 3 = \frac{v_1 - v_2}{3} + \frac{v_1 - v_3}{4}$$

or

$$0.5833v_1 - 0.3333v_2 - 0.25v_3 = -11 \quad [7]$$

At node 2:

$$-(-3) = \frac{v_2 - v_1}{3} + \frac{v_2}{1} + \frac{v_2 - v_3}{7}$$



■ FIGURE 4.4 (a) A four-node circuit. (b) Redrawn circuit with reference node chosen and voltages labeled.

(Continued on next page)

or

$$-0.3333v_1 + 1.4762v_2 - 0.1429v_3 = 3 \quad [8]$$

And, at node 3:

$$-(-25) = \frac{v_3}{5} + \frac{v_3 - v_2}{7} + \frac{v_3 - v_1}{4}$$

or, more simply,

$$-0.25v_1 - 0.1429v_2 + 0.5929v_3 = 25 \quad [9]$$

► **Determine if additional information is required.**

We have three equations in three unknowns. Provided that they are independent, this is sufficient to determine the three voltages.

► **Attempt a solution.**

Equations [7] through [9] can be solved using a scientific calculator (Appendix 5), software packages such as MATLAB, or more traditional “plug-and-chug” techniques such as elimination of variables, matrix methods, or Cramer’s rule. Using the latter method, described in Appendix 2, we have

$$v_1 = \frac{\begin{vmatrix} -11 & -0.3333 & -0.2500 \\ 3 & 1.4762 & -0.1429 \\ 25 & -0.1429 & 0.5929 \end{vmatrix}}{\begin{vmatrix} 0.5833 & -0.3333 & -0.2500 \\ -0.3333 & 3 & -0.1429 \\ -0.2500 & 25 & 0.5929 \end{vmatrix}} = \frac{1.714}{0.3167} = 5.412 \text{ V}$$

Similarly,

$$v_2 = \frac{\begin{vmatrix} 0.5833 & -11 & -0.2500 \\ -0.3333 & 3 & -0.1429 \\ -0.2500 & 25 & 0.5929 \end{vmatrix}}{0.3167} = \frac{2.450}{0.3167} = 7.736 \text{ V}$$

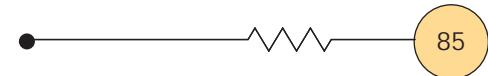
and

$$v_3 = \frac{\begin{vmatrix} 0.5833 & -0.3333 & -11 \\ -0.3333 & 1.4762 & 3 \\ -0.2500 & -0.1429 & 25 \end{vmatrix}}{0.3167} = \frac{14.67}{0.3167} = 46.32 \text{ V}$$

► **Verify the solution. Is it reasonable or expected?**

Substituting the nodal voltages into any of our three nodal equations is sufficient to ensure we made no computational errors. Beyond that, is it possible to determine whether these voltages are “reasonable” values? We have a maximum possible current of  $3 + 8 + 25 = 36$  amperes anywhere in the circuit. The largest resistor is  $7 \Omega$ , so we do not expect any voltage magnitude greater than  $7 \times 36 = 252 \text{ V}$ .

There are, of course, numerous methods available for the solution of linear systems of equations, and we describe several in Appendix 2 in detail. Prior to the advent of the scientific calculator, Cramer’s rule as seen in Example 4.2 was very common in circuit analysis, although occasionally tedious to implement. It is, however, straightforward to use on a simple



four-function calculator, and so an awareness of the technique can be valuable. MATLAB, on the other hand, although not likely to be available during an examination, is a powerful software package that can greatly simplify the solution process; a brief tutorial on getting started is provided in Appendix 6.

For the situation encountered in Example 4.2, there are several options available through MATLAB. First, we can represent Eqs. [7] to [9] in ***matrix form***:

$$\begin{bmatrix} 0.5833 & -0.3333 & -0.25 \\ -0.3333 & 1.4762 & -0.1429 \\ -0.25 & -0.1429 & 0.5929 \end{bmatrix} \begin{bmatrix} v_1 \\ v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} -11 \\ 3 \\ 25 \end{bmatrix}$$

so that

$$\begin{bmatrix} v_1 \\ v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} 0.5833 & -0.3333 & -0.25 \\ -0.3333 & 1.4762 & -0.1429 \\ -0.25 & -0.1429 & 0.5929 \end{bmatrix}^{-1} \begin{bmatrix} -11 \\ 3 \\ 25 \end{bmatrix}$$

In MATLAB, we write

```
>> a = [0.5833 -0.3333 -0.25; -0.3333 1.4762 -0.1429; -0.25 -0.1429 0.5929];
>> c = [-11; 3; 25];
>> b = a^-1 * c
b =
    5.4124
    7.7375
   46.3127
>>
```

where spaces separate elements along rows, and a semicolon separates rows. The matrix named **b**, which can also be referred to as a ***vector*** as it has only one column, is our solution. Thus,  $v_1 = 5.412$  V,  $v_2 = 7.738$  V, and  $v_3 = 46.31$  V (some rounding error has been incurred).

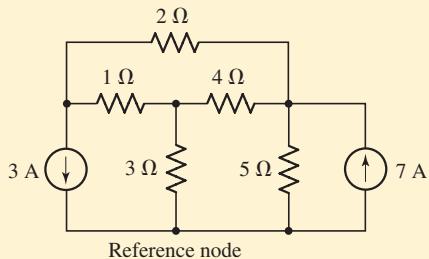
We could also use the KCL equations as we wrote them initially if we employ the symbolic processor of MATLAB.

```
>> eqn1 = '-8 -3 = (v1 - v2)/ 3 + (v1 - v3)/ 4';
>> eqn2 = '-(-3) = (v2 - v1)/ 3 + v2/ 1 + (v2 - v3)/ 7';
>> eqn3 = '-(-25) = v3/ 5 + (v3 - v2)/ 7 + (v3 - v1)/ 4';
>> answer = solve(eqn1, eqn2, eqn3, 'v1', 'v2', 'v3');
>> answer.v1
ans =
720/133
>> answer.v2
ans =
147/19
>> answer.v3
ans =
880/19
>>
```

which results in exact answers, with no rounding errors. The `solve()` routine is invoked with the list of symbolic equations we named `eqn1`, `eqn2`, and `eqn3`, but the variables `v1`, `v2` and `v3` must also be specified. If `solve()` is called with fewer variables than equations, an algebraic solution is returned. The form of the solution is worth a quick comment; it is returned in what is referred to in programming parlance as a *structure*; in this case, we called our structure “`answer`.<sup>1</sup>” Each component of the structure is accessed separately by name as shown.

### PRACTICE

4.2 For the circuit of Fig. 4.5, compute the voltage across each current source.

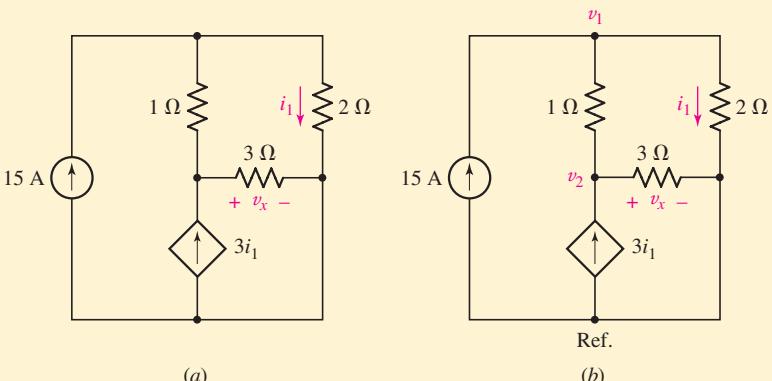


■ FIGURE 4.5

Ans:  $v_{3A} = 5.235$  V;  $v_{7A} = 11.47$  V.

### EXAMPLE 4.3

Determine the power supplied by the dependent source of Fig. 4.6a.



■ FIGURE 4.6 (a) A four-node circuit containing a dependent current source. (b) Circuit labeled for nodal analysis.

We choose the bottom node as our reference, since it has a large number of branch connections, and proceed to label the nodal voltages  $v_1$  and  $v_2$  as shown in Fig. 4.6b. The quantity labeled  $v_x$  is actually equal to  $v_2$ .

At node 1, we write

$$15 = \frac{v_1 - v_2}{1} + \frac{v_1}{2} \quad [10]$$

and at node 2

$$3i_1 = \frac{v_2 - v_1}{1} + \frac{v_2}{3} \quad [11]$$

Unfortunately, we have only two equations but three unknowns; *this is a direct result of the presence of the dependent current source, since it is not controlled by a nodal voltage*. Thus, we need an additional equation that relates  $i_1$  to one or more nodal voltages.

In this case, we find that

$$i_1 = \frac{v_1}{2} \quad [12]$$

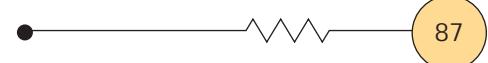
which upon substitution into Eq. [11] yields (with a little rearranging)

$$3v_1 - 2v_2 = 30 \quad [13]$$

and Eq. [10] simplifies to

$$-15v_1 + 8v_2 = 0 \quad [14]$$

Solving, we find that  $v_1 = -40$  V,  $v_2 = -75$  V, and  $i_1 = 0.5v_1 = -20$  A. Thus, the power supplied by the dependent source is equal to  $(3i_1)(v_2) = (-60)(-75) = 4.5$  kW.



We see that the presence of a dependent source will create the need for an additional equation in our analysis if the controlling quantity is not a nodal voltage. Now let's look at the same circuit, but with the controlling variable of the dependent current source changed to a different quantity—the voltage across the  $3\ \Omega$  resistor, which is in fact a nodal voltage. We will find that only *two* equations are required to complete the analysis.



## EXAMPLE 4.4

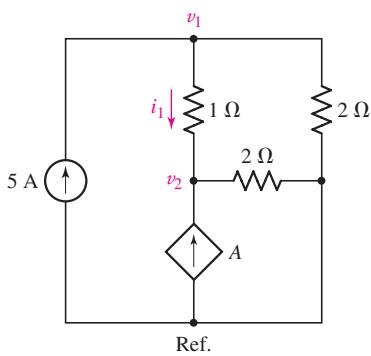
**Determine the power supplied by the dependent source of Fig. 4.7a.**

We select the bottom node as our reference and label the nodal voltages as shown in Fig. 4.7b. We have labeled the nodal voltage  $v_x$  explicitly for clarity. Note that our choice of reference node is important in this case; it led to the quantity  $v_x$  being a nodal voltage.

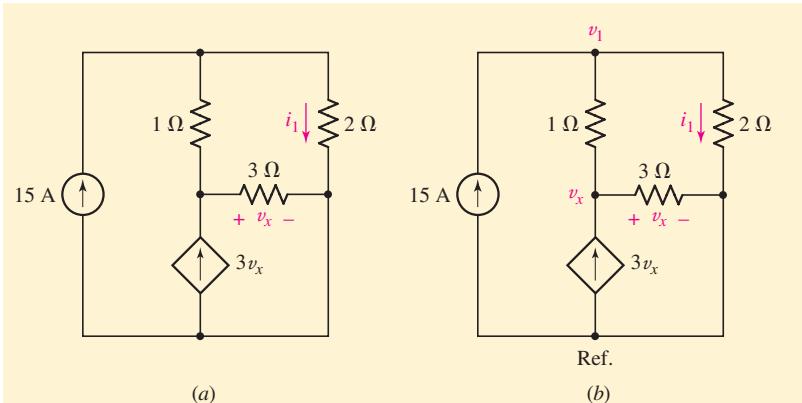
Our KCL equation for node 1 is

$$15 = \frac{v_1 - v_x}{1} + \frac{v_1}{2} \quad [15]$$

*(Continued on next page)*



■ FIGURE 4.8



■ FIGURE 4.7 (a) A four-node circuit containing a dependent current source. (b) Circuit labeled for nodal analysis.

and for node  $x$  is

$$3v_x = \frac{v_x - v_1}{1} + \frac{v_2}{3} \quad [16]$$

Grouping terms and solving, we find that  $v_1 = \frac{50}{7}$  V and  $v_x = -\frac{30}{7}$  V. Thus, the dependent source in this circuit generates  $(3v_x)(v_x) = 55.1$  W.

### PRACTICE

- 4.3 For the circuit of Fig. 4.8, determine the nodal voltage  $v_1$  if  $A$  is  
 (a)  $2i_1$ ; (b)  $2v_1$ .

Ans: (a)  $\frac{70}{9}$  V; (b)  $-10$  V.

### Summary of Basic Nodal Analysis Procedure

- Count the number of nodes ( $N$ ).**
- Designate a reference node.** The number of terms in your nodal equations can be minimized by selecting the node with the greatest number of branches connected to it.
- Label the nodal voltages** (there are  $N - 1$  of them).
- Write a KCL equation for each of the nonreference nodes.** Sum the currents flowing *into* a node from sources on one side of the equation. On the other side, sum the currents flowing *out of* the node through resistors. Pay close attention to “ $-$ ” signs.
- Express any additional unknowns such as currents or voltages other than nodal voltages in terms of appropriate nodal voltages.** This situation can occur if voltage sources or dependent sources appear in our circuit.
- Organize the equations.** Group terms according to nodal voltages.
- Solve the system of equations for the nodal voltages** (there will be  $N - 1$  of them).

These seven basic steps will work on any circuit we ever encounter, although the presence of voltage sources will require extra care. Such situations are discussed next.

## 4.2 THE SUPERNODE

As an example of how voltage sources are best handled when performing nodal analysis, consider the circuit shown in Fig. 4.9a. The original four-node circuit of Fig. 4.4 has been changed by replacing the  $7\ \Omega$  resistor between nodes 2 and 3 with a 22 V voltage source. We still assign the same node-to-reference voltages  $v_1$ ,  $v_2$ , and  $v_3$ . Previously, the next step was the application of KCL at each of the three nonreference nodes. If we try to do that once again, we see that we will run into some difficulty at both nodes 2 and 3, for we do not know what the current is in the branch with the voltage source. There is no way by which we can express the current as a function of the voltage, for the definition of a voltage source is exactly that the voltage is independent of the current.

There are two ways out of this dilemma. The more difficult approach is to assign an unknown current to the branch which contains the voltage source, proceed to apply KCL three times, and then apply KVL ( $v_3 - v_2 = 22$ ) once between nodes 2 and 3; the result is then four equations in four unknowns.

The easier method is to treat node 2, node 3, and the voltage source together as a sort of **supernode** and apply KCL to both nodes at the same time; the supernode is indicated by the region enclosed by the broken line in Fig. 4.9a. This is okay because if the total current leaving node 2 is zero and the total current leaving node 3 is zero, then the total current leaving the combination of the two nodes is zero. This concept is represented graphically in the expanded view of Fig. 4.9b.

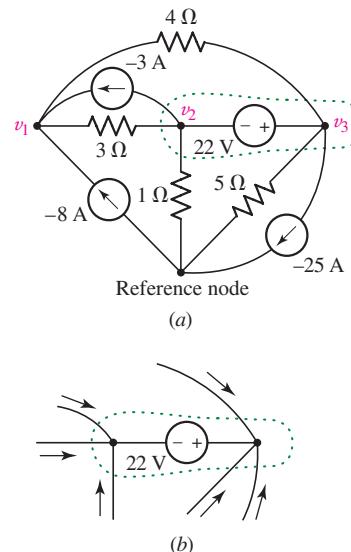


FIGURE 4.9 (a) The circuit of Example 4.2 with a 22 V source in place of the  $7\ \Omega$  resistor. (b) Expanded view of the region defined as a supernode; KCL requires that all currents flowing into the region sum to zero, or we would pile up or run out of electrons.

## EXAMPLE 4.5

Determine the value of the unknown node voltage  $v_1$  in the circuit of Fig. 4.9a.

The KCL equation at node 1 is unchanged from Example 4.2:

$$-8 - 3 = \frac{v_1 - v_2}{3} + \frac{v_1 - v_3}{4}$$

or

$$0.5833v_1 - 0.3333v_2 - 0.2500v_3 = -11 \quad [17]$$

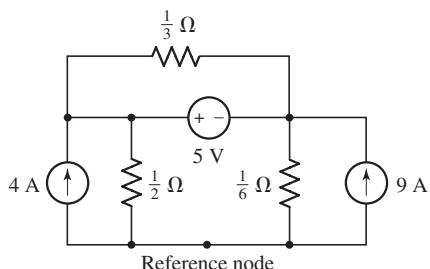
Next we consider the 2-3 supernode. Two current sources are connected, and four resistors. Thus,

$$3 + 25 = \frac{v_2 - v_1}{3} + \frac{v_3 - v_1}{4} + \frac{v_3}{5} + \frac{v_2}{1}$$

or

$$-0.5833v_1 + 1.3333v_2 + 0.45v_3 = 28 \quad [18]$$

(Continued on next page)



■ FIGURE 4.10

Since we have three unknowns, we need one additional equation, and it must utilize the fact that there is a 22 V voltage source between nodes 2 and 3:

$$v_2 - v_3 = -22 \quad [19]$$

Solving Eqs. [17] to [19], the solution for  $v_1$  is 1.071 V.

### PRACTICE

- 4.4 For the circuit of Fig. 4.10, compute the voltage across each current source.

Ans: 5.375 V, 375 mV.

The presence of a voltage source thus reduces by 1 the number of nonreference nodes at which we must apply KCL, regardless of whether the voltage source extends between two nonreference nodes or is connected between a node and the reference. We should be careful in analyzing circuits such as that of Practice Problem 4.4. Since both ends of the resistor are part of the supernode, there must technically be *two* corresponding current terms in the KCL equation, but they cancel each other out. We can summarize the supernode method as follows:

#### Summary of Supernode Analysis Procedure

- Count the number of nodes ( $N$ ).**
- Designate a reference node.** The number of terms in your nodal equations can be minimized by selecting the node with the greatest number of branches connected to it.
- Label the nodal voltages** (there are  $N - 1$  of them).
- If the circuit contains voltage sources, form a supernode about each one.** This is done by enclosing the source, its two terminals, and any other elements connected between the two terminals within a broken-line enclosure.
- Write a KCL equation for each nonreference node and for each supernode that does not contain the reference node.** Sum the currents flowing *into* a node/supernode from current sources on one side of the equation. On the other side, sum the currents flowing *out of* the node/supernode through resistors. Pay close attention to “-” signs.
- Relate the voltage across each voltage source to nodal voltages.** This is accomplished by simple application of KVL; one such equation is needed for each supernode defined.
- Express any additional unknowns (i.e., currents or voltages other than nodal voltages) in terms of appropriate nodal voltages.** This situation can occur if dependent sources appear in our circuit.
- Organize the equations.** Group terms according to nodal voltages.
- Solve the system of equations for the nodal voltages** (there will be  $N - 1$  of them).

We see that we have added two additional steps from our general nodal analysis procedure. In reality, however, application of the supernode technique to a circuit containing voltage sources not connected to the reference node will result in a reduction in the number of KCL equations required. With this in mind, let's consider the circuit of Fig. 4.11, which contains all four types of sources and has five nodes.

### Determine the node-to-reference voltages in the circuit of Fig. 4.11.

After establishing a supernode about each *voltage* source, we see that we need to write KCL equations only at node 2 and at the supernode containing the dependent voltage source. By inspection, it is clear that  $v_1 = -12$  V.

At node 2,

$$\frac{v_2 - v_1}{0.5} + \frac{v_2 - v_3}{2} = 14 \quad [20]$$

while at the 3-4 supernode,

$$0.5v_x = \frac{v_3 - v_2}{2} + \frac{v_4}{1} + \frac{v_4 - v_1}{2.5} \quad [21]$$

We next relate the source voltages to the node voltages:

$$v_3 - v_4 = 0.2v_y \quad [22]$$

and

$$0.2v_y = 0.2(v_4 - v_1) \quad [23]$$

Finally, we express the dependent current source in terms of the assigned variables:

$$0.5v_x = 0.5(v_2 - v_1) \quad [24]$$

Five nodes requires four KCL equations in general nodal analysis, but we have reduced this requirement to *only two*, as we formed two separate supernodes. Each supernode required a KVL equation (Eq. [22] and  $v_1 = -12$ , the latter written by inspection). Neither dependent source was controlled by a nodal voltage, so two additional equations were needed as a result.

With this done, we can now eliminate  $v_x$  and  $v_y$  to obtain a set of four equations in the four node voltages:

$$\begin{aligned} -2v_1 + 2.5v_2 - 0.5v_3 &= 14 \\ 0.1v_1 - v_2 + 0.5v_3 + 1.4v_4 &= 0 \\ v_1 &= -12 \\ 0.2v_1 + v_3 - 1.2v_4 &= 0 \end{aligned}$$

Solving,  $v_1 = -12$  V,  $v_2 = -4$  V,  $v_3 = 0$  V, and  $v_4 = -2$  V.

### PRACTICE

4.5 Determine the nodal voltages in the circuit of Fig. 4.12.

Ans:  $v_1 = 3$  V,  $v_2 = -2.33$  V,  $v_3 = -1.91$  V,  $v_4 = 0.945$  V.

### EXAMPLE 4.6

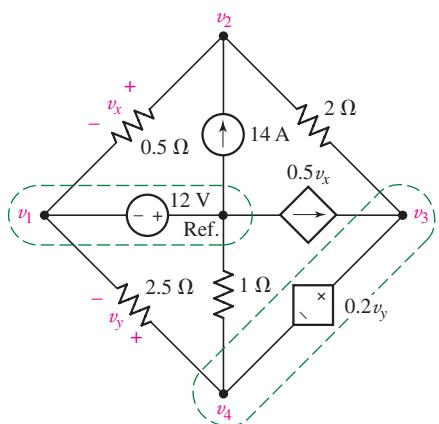


FIGURE 4.11 A five-node circuit with four different types of sources.

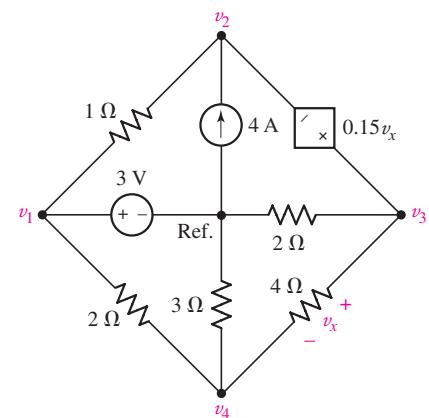
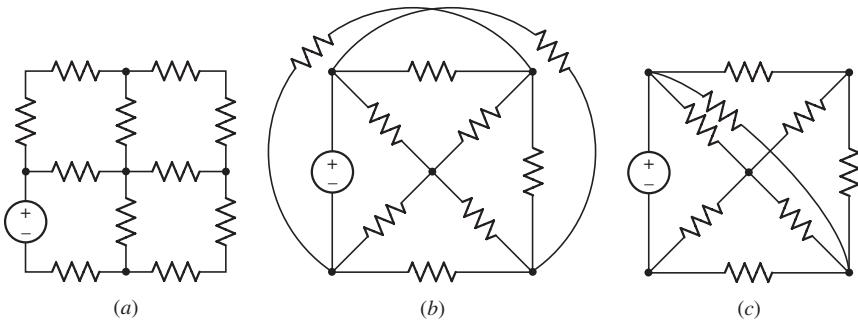


FIGURE 4.12

### 4.3 MESH ANALYSIS

As we have seen, nodal analysis is a straightforward analysis technique when only current sources are present, and voltage sources are easily accommodated with the supernode concept. Still, nodal analysis is based on KCL, and the reader might at some point wonder if there isn't a similar approach based on KVL. There is—it's known as ***mesh analysis***—and although only strictly speaking applicable to what we will shortly define as a planar circuit, it can in many cases prove simpler to apply than nodal analysis.

If it is possible to draw the diagram of a circuit on a plane surface in such a way that no branch passes over or under any other branch, then that circuit is said to be a ***planar circuit***. Thus, Fig. 4.13a shows a planar network, Fig. 4.13b shows a nonplanar network, and Fig. 4.13c also shows a planar network, although it is drawn in such a way as to make it appear nonplanar at first glance.



■ FIGURE 4.13 Examples of planar and nonplanar networks; crossed wires without a solid dot are not in physical contact with each other.

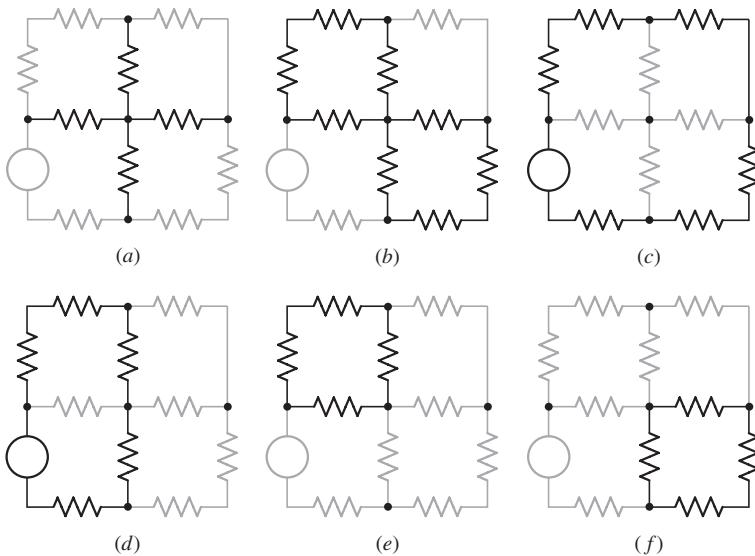
In Sec. 3.1, the terms ***path***, ***closed path***, and ***loop*** were defined. Before we define a mesh, let us consider the sets of branches drawn with heavy lines in Fig. 4.14. The first set of branches is not a path, since four branches are connected to the center node, and it is of course also not a loop. The second set of branches does not constitute a path, since it is traversed only by passing through the central node twice. The remaining four paths are all loops. The circuit contains 11 branches.

The mesh is a property of a planar circuit and is undefined for a nonplanar circuit. We define a ***mesh*** as a loop that does not contain any other loops within it. Thus, the loops indicated in Fig. 4.14c and d are not meshes, whereas those of parts e and f are meshes. Once a circuit has been drawn neatly in planar form, it often has the appearance of a multipaned window; the boundary of each pane in the window may be considered to be a mesh.

If a network is planar, mesh analysis can be used to accomplish the analysis. This technique involves the concept of a ***mesh current***, which we introduce by considering the analysis of the two-mesh circuit of Fig. 4.15a.

As we did in the single-loop circuit, we will begin by defining a current through one of the branches. Let us call the current flowing to the right through the  $6\ \Omega$  resistor  $i_1$ . We will apply KVL around each of the two meshes, and the two resulting equations are sufficient to determine two unknown currents. We next define a second current  $i_2$  flowing to the right in

We should mention that mesh-type analysis can be applied to nonplanar circuits, but since it is not possible to define a complete set of unique meshes for such circuits, assignment of unique mesh currents is not possible.



**FIGURE 4.14** (a) The set of branches identified by the heavy lines is neither a path nor a loop. (b) The set of branches here is not a path, since it can be traversed only by passing through the central node twice. (c) This path is a loop but not a mesh, since it encloses other loops. (d) This path is also a loop but not a mesh. (e, f) Each of these paths is both a loop and a mesh.

the  $4\ \Omega$  resistor. We might also choose to call the current flowing downward through the central branch  $i_3$ , but it is evident from KCL that  $i_3$  may be expressed in terms of the two previously assumed currents as  $(i_1 - i_2)$ . The assumed currents are shown in Fig. 4.15b.

Following the method of solution for the single-loop circuit, we now apply KVL to the left-hand mesh,

$$-42 + 6i_1 + 3(i_1 - i_2) = 0$$

or

$$9i_1 - 3i_2 = 42 \quad [25]$$

Applying KVL to the right-hand mesh,

$$-3(i_1 - i_2) + 4i_2 - 10 = 0$$

or

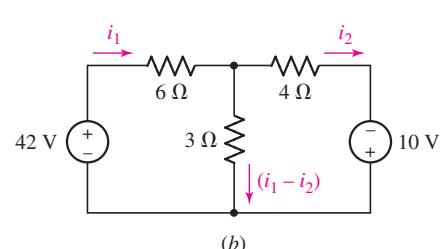
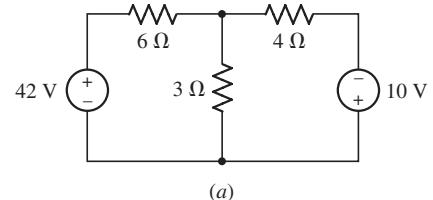
$$-3i_1 + 7i_2 = 10 \quad [26]$$

Equations [25] and [26] are independent equations; one cannot be derived from the other. With two equations and two unknowns, the solution is easily obtained:

$$i_1 = 6\text{ A} \quad i_2 = 4\text{ A} \quad \text{and} \quad (i_1 - i_2) = 2\text{ A}$$

If our circuit contains  $M$  meshes, then we expect to have  $M$  mesh currents and therefore will be required to write  $M$  independent equations.

Now let us consider this same problem in a slightly different manner by using mesh currents. We define a **mesh current** as a current that flows only around the perimeter of a mesh. One of the greatest advantages in the use of mesh currents is the fact that Kirchhoff's current law is automatically satisfied. If a mesh current flows *into* a given node, it flows *out of* it also.



**FIGURE 4.15** (a, b) A simple circuit for which currents are required.



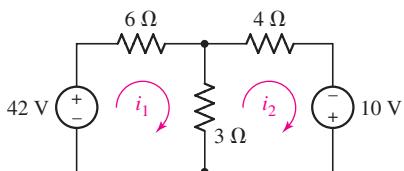


FIGURE 4.16 The same circuit considered in Fig. 4.15b, but viewed a slightly different way.

A mesh current may often be identified as a branch current, as  $i_1$  and  $i_2$  have been identified in this example. This is not always true, however, for consideration of a square nine-mesh network soon shows that the central mesh current cannot be identified as the current in any branch.

If we call the left-hand mesh of our problem mesh 1, then we may establish a mesh current  $i_1$  flowing in a clockwise direction about this mesh. A mesh current is indicated by a curved arrow that almost closes on itself and is drawn inside the appropriate mesh, as shown in Fig. 4.16. The mesh current  $i_2$  is established in the remaining mesh, again in a clockwise direction. Although the directions are arbitrary, we will always choose clockwise mesh currents because a certain error-minimizing symmetry then results in the equations.

We no longer have a current or current arrow shown directly on each branch in the circuit. The current through any branch must be determined by considering the mesh currents flowing in every mesh in which that branch appears. This is not difficult, because no branch can appear in more than two meshes. For example, the  $3\ \Omega$  resistor appears in both meshes, and the current flowing downward through it is  $i_1 - i_2$ . The  $6\ \Omega$  resistor appears only in mesh 1, and the current flowing to the right in that branch is equal to the mesh current  $i_1$ .

For the left-hand mesh,

$$-42 + 6i_1 + 3(i_1 - i_2) = 0$$

while for the right-hand mesh,

$$3(i_2 - i_1) + 4i_2 - 10 = 0$$

and these two equations are equivalent to Eqs. [25] and [26].

## EXAMPLE 4.7

Determine the power supplied by the 2 V source of Fig. 4.17a.

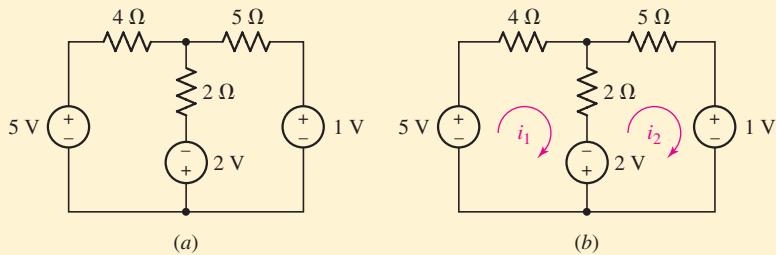


FIGURE 4.17 (a) A two-mesh circuit containing three sources. (b) Circuit labeled for mesh analysis.

We first define two clockwise mesh currents as shown in Fig. 4.17b.

Beginning at the bottom left node of mesh 1, we write the following KVL equation as we proceed clockwise through the branches:

$$-5 + 4i_1 + 2(i_1 - i_2) - 2 = 0$$

Doing the same for mesh 2, we write

$$+2 + 2(i_2 - i_1) + 5i_2 + 1 = 0$$

Rearranging and grouping terms,

$$6i_1 - 2i_2 = 7$$

and

$$-2i_1 + 7i_2 = -3$$

Solving,  $i_1 = \frac{43}{38} = 1.132$  A and  $i_2 = -\frac{2}{19} = -0.1053$  A.

The current flowing out of the positive reference terminal of the 2 V source is  $i_1 - i_2$ . Thus, the 2 V source supplies  $(2)(1.237) = 2.474$  W.

### PRACTICE

4.6 Determine  $i_1$  and  $i_2$  in the circuit in Fig. 4.18.

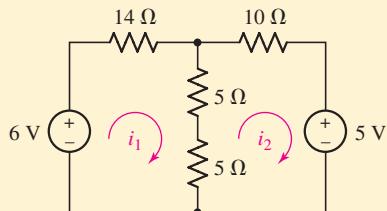


FIGURE 4.18

Ans: +184.2 mA; -157.9 mA.

Let us next consider the five-node, seven-branch, three-mesh circuit shown in Fig. 4.19. This is a slightly more complicated problem because of the additional mesh.

### EXAMPLE 4.8

Use mesh analysis to determine the three mesh currents in the circuit of Fig. 4.19.

The three required mesh currents are assigned as indicated in Fig. 4.19, and we methodically apply KVL about each mesh:

$$-7 + 1(i_1 - i_2) + 6 + 2(i_1 - i_3) = 0$$

$$1(i_2 - i_1) + 2i_2 + 3(i_2 - i_3) = 0$$

$$2(i_3 - i_1) - 6 + 3(i_3 - i_2) + 1i_3 = 0$$

Simplifying,

$$3i_1 - i_2 - 2i_3 = 1$$

$$-i_1 + 6i_2 - 3i_3 = 0$$

$$-2i_1 - 3i_2 + 6i_3 = 6$$

and solving, we obtain  $i_1 = 3$  A,  $i_2 = 2$  A, and  $i_3 = 3$  A.

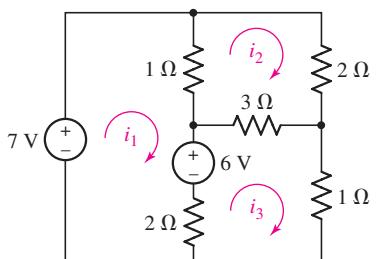
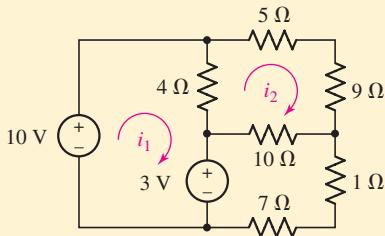


FIGURE 4.19 A five-node, seven-branch, three-mesh circuit.

**PRACTICE**

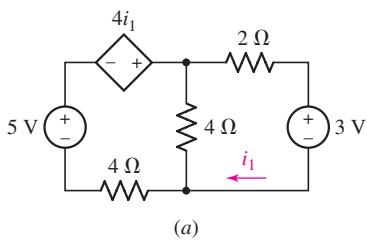
4.7 Determine  $i_1$  and  $i_2$  in the circuit of Fig 4.20.



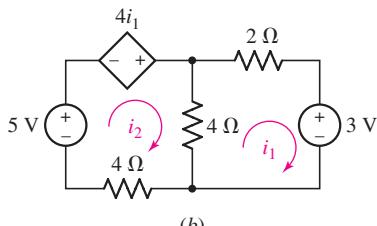
■ FIGURE 4.20

Ans: 2.220 A, 470.0 mA.

The previous examples dealt with circuits powered exclusively by independent voltage sources. If a current source is included in the circuit, it may either simplify or complicate the analysis, as discussed in Sec. 4.4. As seen in our study of the nodal analysis technique, dependent sources generally require an additional equation besides the  $M$  mesh equations, unless the controlling variable is a mesh current (or sum of mesh currents). We explore this in the following example.

**EXAMPLE 4.9**

(a)



(b)

■ FIGURE 4.21 (a) A two-mesh circuit containing a dependent source. (b) Circuit labeled for mesh analysis.

**Determine the current  $i_1$  in the circuit of Fig. 4.21a.**

The current  $i_1$  is actually a mesh current, so rather than redefine it we label the rightmost mesh current  $i_1$  and define a clockwise mesh current  $i_2$  for the left mesh, as shown in Fig. 4.21b.

For the left mesh, KVL yields

$$-5 - 4i_1 + 4(i_2 - i_1) + 4i_2 = 0 \quad [27]$$

and for the right mesh we find

$$4(i_1 - i_2) + 2i_1 + 3 = 0 \quad [28]$$

Grouping terms, these equations may be written more compactly as

$$-8i_1 + 8i_2 = 5$$

and

$$6i_1 - 4i_2 = -3$$

Solving,  $i_2 = 375$  mA, so  $i_1 = -250$  mA.

Since the dependent source of Fig. 4.21 is controlled by a mesh current ( $i_1$ ), only two equations—Eqs. [27] and [28]—were required to analyze the two-mesh circuit. In the following example, we explore the situation that arises if the controlling variable is *not* a mesh current.

## EXAMPLE 4.10

Determine the current  $i_1$  in the circuit of Fig. 4.22a.

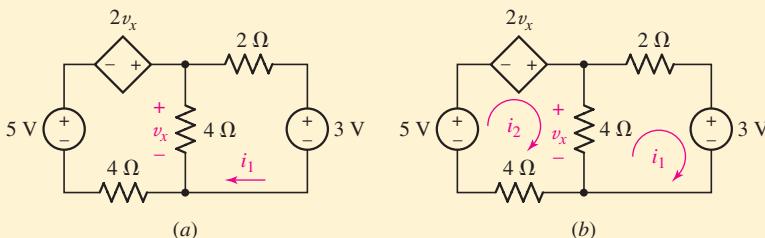


FIGURE 4.22 (a) A circuit with a dependent source controlled by a voltage. (b) Circuit labeled for mesh analysis.

In order to draw comparisons to Example 4.9 we use the same mesh current definitions, as shown in Fig. 4.22b.

For the left mesh, KVL now yields

$$-5 - 2v_x + 4(i_2 - i_1) + 4i_2 = 0 \quad [29]$$

and for the right mesh we find the same as before, namely,

$$4(i_1 - i_2) + 2i_1 + 3 = 0 \quad [30]$$

Since the dependent source is controlled by the unknown voltage  $v_x$ , we are faced with *two* equations in *three* unknowns. The way out of our dilemma is to construct an equation for  $v_x$  in terms of mesh currents, such as

$$v_x = 4(i_2 - i_1) \quad [31]$$

We simplify our system of equations by substituting Eq. [31] into Eq. [29], resulting in

$$4i_1 = 5$$

Solving, we find that  $i_1 = 1.25$  A. In this particular instance, Eq. [30] is not needed unless a value for  $i_2$  is desired.

## PRACTICE

4.8 Determine  $i_1$  in the circuit of Fig. 4.23 if the controlling quantity  $A$  is equal to (a)  $2i_2$ ; (b)  $2v_x$ .

Ans: (a) 1.35 A; (b) 546 mA.

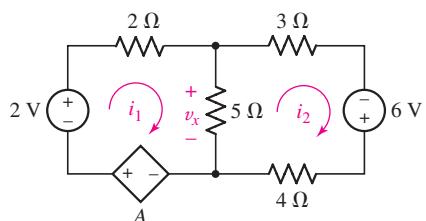


FIGURE 4.23

The mesh analysis procedure can be summarized by the seven basic steps that follow. It will work on any *planar* circuit we ever encounter, although the presence of current sources will require extra care. Such situations are discussed in Sec. 4.4.

**Summary of Basic Mesh Analysis Procedure**

1. **Determine if the circuit is a planar circuit.** If not, perform nodal analysis instead.
2. **Count the number of meshes ( $M$ ).** Redraw the circuit if necessary.
3. **Label each of the  $M$  mesh currents.** Generally, defining all mesh currents to flow clockwise results in a simpler analysis.
4. **Write a KVL equation around each mesh.** Begin with a convenient node and proceed in the direction of the mesh current. Pay close attention to “ $-$ ” signs. If a current source lies on the periphery of a mesh, no KVL equation is needed and the mesh current is determined by inspection.
5. **Express any additional unknowns such as voltages or currents other than mesh currents in terms of appropriate mesh currents.** This situation can occur if current sources or dependent sources appear in our circuit.
6. **Organize the equations.** Group terms according to mesh currents.
7. **Solve the system of equations for the mesh currents** (there will be  $M$  of them).

**4.4 THE SUPERMESH**

How must we modify this straightforward procedure when a current source is present in the network? Taking our lead from nodal analysis, we should feel that there are two possible methods. First, we could assign an unknown voltage across the current source, apply KVL around each mesh as before, and then relate the source current to the assigned mesh currents. This is generally the more difficult approach.

A better technique is one that is quite similar to the supernode approach in nodal analysis. There we formed a supernode, completely enclosing the voltage source inside the supernode and reducing the number of non-reference nodes by 1 for each voltage source. Now we create a kind of “*supermesh*” from two meshes that have a current source as a common element; the current source is in the interior of the supermesh. We thus reduce the number of meshes by 1 for each current source present. If the current source lies on the *perimeter* of the circuit, then the single mesh in which it is found is ignored. Kirchhoff’s voltage law is thus applied only to those meshes or supermeshes in the reinterpreted network.

**EXAMPLE 4.11**

Determine the three mesh currents in Fig. 4.24a.

We note that a 7 A independent current source is in the common boundary of two meshes, which leads us to create a supermesh whose interior

is that of meshes 1 and 3 as shown in Fig. 4.24b. Applying KVL about this loop,

$$-7 + 1(i_1 - i_2) + 3(i_3 - i_2) + 1i_3 = 0$$

or

$$i_1 - 4i_2 + 4i_3 = 7 \quad [32]$$

and around mesh 2,

$$1(i_2 - i_1) + 2i_2 + 3(i_2 - i_3) = 0$$

or

$$-i_1 + 6i_2 - 3i_3 = 0 \quad [33]$$

Finally, the independent source current is related to the mesh currents,

$$i_1 - i_3 = 7 \quad [34]$$

Solving Eqs. [32] through [34], we find  $i_1 = 9$  A,  $i_2 = 2.5$  A, and  $i_3 = 2$  A.

### PRACTICE

4.9 Determine the current  $i_1$  in the circuit of Fig. 4.25.

Ans:  $-1.93$  A.

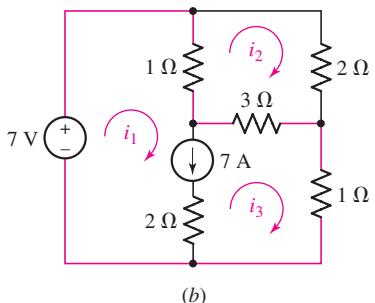
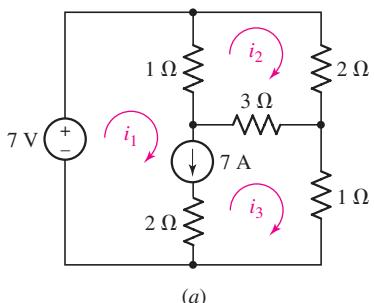
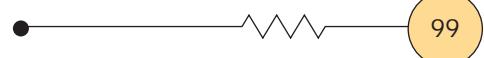


FIGURE 4.24 (a) A three-mesh circuit with an independent current source. (b) A supermesh is defined by the colored line.

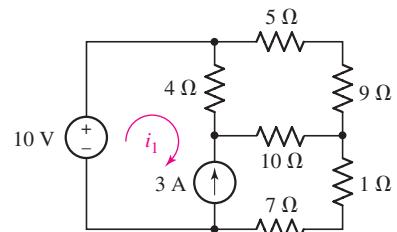


FIGURE 4.25

The presence of one or more dependent sources merely requires each of these source quantities and the variable on which it depends to be expressed in terms of the assigned mesh currents. In Fig. 4.26, for example, we note that both a dependent and an independent current source are included in the network. Let's see how their presence affects the analysis of the circuit and actually simplifies it.

### Evaluate the three unknown currents in the circuit of Fig. 4.26.

The current sources appear in meshes 1 and 3. Since the 15 A source is located on the perimeter of the circuit, we may eliminate mesh 1 from consideration—it is clear that  $i_1 = 15$  A.

We find that because we now know one of the two mesh currents relevant to the dependent current source, there is no need to write a supermesh equation about meshes 1 and 3. Instead, we simply relate  $i_1$  and  $i_3$  to the current from the dependent source using KCL:

$$\frac{v_x}{9} = i_3 - i_1 = \frac{3(i_3 - i_2)}{9}$$

(Continued on next page)

### EXAMPLE 4.12

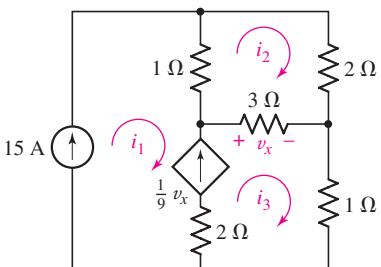


FIGURE 4.26 A three-mesh circuit with one dependent and one independent current source.

which can be written more compactly as

$$-i_1 + \frac{1}{3}i_2 + \frac{2}{3}i_3 = 0 \quad \text{or} \quad \frac{1}{3}i_2 + \frac{2}{3}i_3 = 15 \quad [35]$$

With one equation in two unknowns, all that remains is to write a KVL equation about mesh 2:

$$1(i_2 - i_1) + 2i_2 + 3(i_2 - i_3) = 0$$

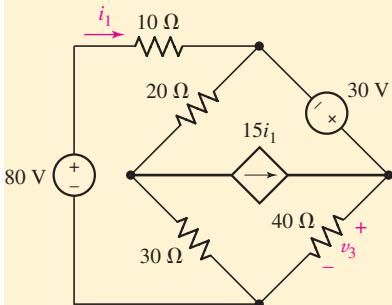
or

$$6i_2 - 3i_3 = 15 \quad [36]$$

Solving Eqs. [35] and [36], we find that  $i_2 = 11$  A and  $i_3 = 17$  A; we already determined that  $i_1 = 15$  A by inspection.

### PRACTICE

4.10 Determine  $v_3$  in the circuit of Fig. 4.27.



■ FIGURE 4.27

Ans: 104.2 V

We can now summarize the general approach to writing mesh equations, whether or not dependent sources, voltage sources, and/or current sources are present, provided that the circuit can be drawn as a planar circuit:

#### Summary of Supermesh Analysis Procedure

- Determine if the circuit is a planar circuit.** If not, perform nodal analysis instead.
- Count the number of meshes ( $M$ ).** Redraw the circuit if necessary.
- Label each of the  $M$  mesh currents.** Generally, defining all mesh currents to flow clockwise results in a simpler analysis.
- If the circuit contains current sources shared by two meshes, form a supermesh to enclose both meshes.** A highlighted enclosure helps when writing KVL equations.
- Write a KVL equation around each mesh/supermesh.** Begin with a convenient node and proceed in the direction of the mesh current. Pay close attention to “-” signs. If a current source lies

on the periphery of a mesh, no KVL equation is needed and the mesh current is determined by inspection.

6. **Relate the current flowing from each current source to mesh currents.** This is accomplished by simple application of KCL; one such equation is needed for each supermesh defined.
7. **Express any additional unknowns such as voltages or currents other than mesh currents in terms of appropriate mesh currents.** This situation can occur if dependent sources appear in our circuit.
8. **Organize the equations.** Group terms according to nodal voltages.
9. **Solve the system of equations for the mesh currents** (there will be  $M$  of them).

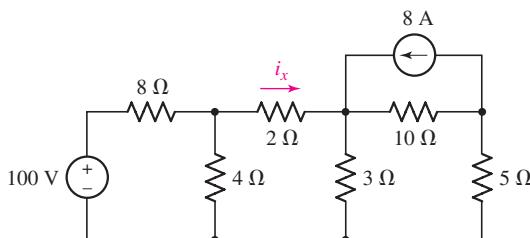
## 4.5 NODAL VS. MESH ANALYSIS: A COMPARISON

Now that we have examined two distinctly different approaches to circuit analysis, it seems logical to ask if there is ever any advantage to using one over the other. If the circuit is nonplanar, then there is no choice: only nodal analysis may be applied.

Provided that we are indeed considering the analysis of a *planar* circuit, however, there are situations where one technique has a small advantage over the other. If we plan to use nodal analysis, then a circuit with  $N$  nodes will lead to at most  $(N - 1)$  KCL equations. Each supernode defined will further reduce this number by 1. If the same circuit has  $M$  distinct meshes, then we will obtain at most  $M$  KVL equations; each supermesh will reduce this number by 1. Based on these facts, we should select the approach that will result in the smaller number of simultaneous equations.

If one or more dependent sources are included in the circuit, then each controlling quantity may influence our choice of nodal or mesh analysis. For example, a dependent voltage source controlled by a nodal voltage does not require an additional equation when we perform nodal analysis. Likewise, a dependent current source controlled by a mesh current does not require an additional equation when we perform mesh analysis. *What about the situation where a dependent voltage source is controlled by a current? Or the converse, where a dependent current source is controlled by a voltage?* Provided that the controlling quantity can be easily related to mesh currents, we might expect mesh analysis to be the more straightforward option. Likewise, if the controlling quantity can be easily related to nodal voltages, nodal analysis may be preferable. One final point in this regard is to keep in mind the *location* of the source; current sources which lie on the periphery of a mesh, whether dependent or independent, are easily treated in mesh analysis; voltage sources connected to the reference terminal are easily treated in nodal analysis.

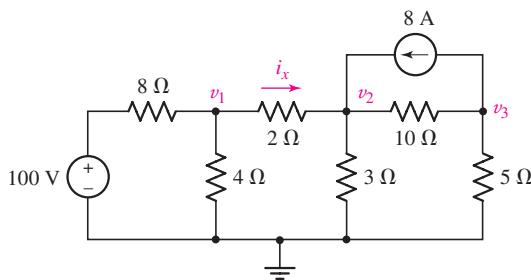
When either method results in essentially the same number of equations, it may be worthwhile to also consider what quantities are being sought. Nodal analysis results in direct calculation of nodal voltages, whereas mesh analysis provides currents. If we are asked to find currents through a set of resistors, for example, after performing nodal analysis, we must still invoke Ohm's law at each resistor to determine the current.



■ FIGURE 4.28 A planar circuit with five nodes and four meshes.

As an example, consider the circuit in Fig. 4.28. We wish to determine the current  $i_x$ .

We choose the bottom node as the reference node, and note that there are four nonreference nodes. Although this means that we can write four distinct equations, there is no need to label the node between the 100 V source and the 8 Ω resistor, since that node voltage is clearly 100 V. Thus, we label the remaining node voltages  $v_1$ ,  $v_2$ , and  $v_3$  as in Fig. 4.29.

■ FIGURE 4.29 The circuit of Fig. 4.28 with node voltages labeled.  
Note that an earth ground symbol was chosen to designate the reference terminal.

We write the following three equations:

$$\frac{v_1 - 100}{8} + \frac{v_1}{4} + \frac{v_1 - v_2}{2} = 0 \quad \text{or} \quad 0.875v_1 - 0.5v_2 = 12.5 \quad [37]$$

$$\frac{v_2 - v_1}{2} + \frac{v_2}{3} + \frac{v_2 - v_3}{10} - 8 = 0 \quad \text{or} \quad -0.5v_1 - 0.9333v_2 - 0.1v_3 = 8 \quad [38]$$

$$\frac{v_3 - v_2}{10} + \frac{v_3}{5} + 8 = 0 \quad \text{or} \quad -0.1v_2 + 0.3v_3 = -8 \quad [39]$$

Solving, we find that  $v_1 = 25.89$  V and  $v_2 = 20.31$  V. We determine the current  $i_x$  by application of Ohm's law:

$$i_x = \frac{v_1 - v_2}{2} = 2.79 \text{ A} \quad [40]$$

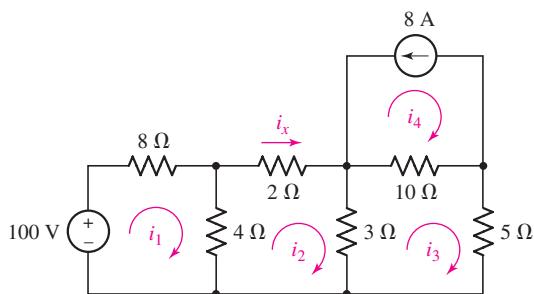


FIGURE 4.30 The circuit of Fig. 4.28 with mesh currents labeled.

Next, we consider the same circuit using mesh analysis. We see in Fig. 4.30 that we have four distinct meshes, although it is obvious that  $i_4 = -8$  A; we therefore need to write three distinct equations.

Writing a KVL equation for meshes 1, 2, and 3:

$$-100 + 8i_1 + 4(i_1 - i_2) = 0 \quad \text{or} \quad 12i_1 - 4i_2 = 100 \quad [41]$$

$$4(i_2 - i_1) + 2i_2 + 3(i_2 - i_3) = 0 \quad \text{or} \quad -4i_1 + 9i_2 - 3i_3 = 0 \quad [42]$$

$$3(i_3 - i_2) + 10(i_3 + 8) + 5i_3 = 0 \quad \text{or} \quad -3i_2 + 18i_3 = -80 \quad [43]$$

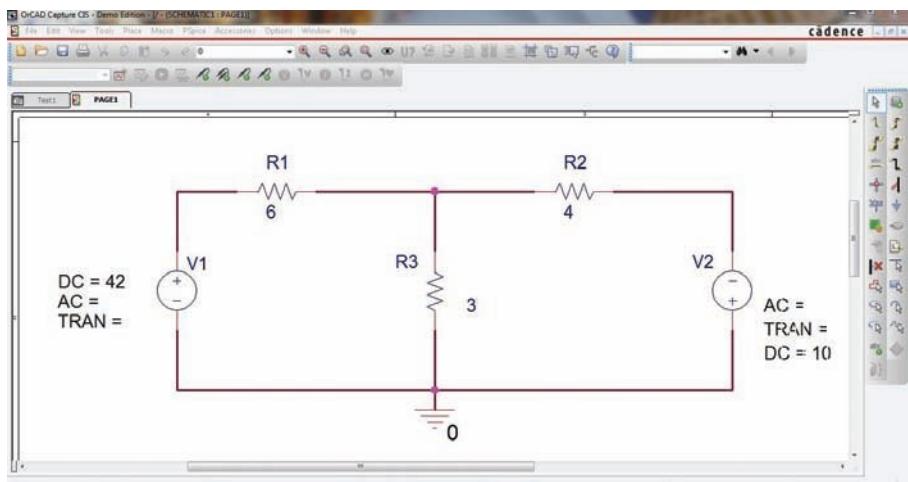
Solving, we find that  $i_2 (= i_x) = 2.79$  A. For this particular problem, mesh analysis proved to be simpler. Since either method is valid, however, working the same problem both ways can also serve as a means to check our answers.

## 4.6 COMPUTER-AIDED CIRCUIT ANALYSIS

We have seen that it does not take many components at all to create a circuit of respectable complexity. As we continue to examine even more complex circuits, it will become obvious rather quickly that it is easy to make errors during the analysis, and verifying solutions by hand can be time-consuming. A powerful computer software package known as PSpice is commonly employed for rapid analysis of circuits, and the schematic capture tools are typically integrated with either a printed circuit board or integrated circuit layout tool. Originally developed in the early 1970s at the University of California at Berkeley, SPICE (*Simulation Program with Integrated Circuit Emphasis*) is now an industry standard. MicroSim Corporation introduced PSpice in 1984, which built intuitive graphical interfaces around the core SPICE program. Depending on the type of circuit application being considered, there are now several companies offering variations of the basic SPICE package.

Although computer-aided analysis is a relatively quick means of determining voltages and currents in a circuit, we should be careful not to allow simulation packages to completely replace traditional “paper and pencil” analysis. There are several reasons for this. First, in order to design we must be able to analyze. Overreliance on software tools can inhibit the development of necessary analytical skills, similar to introducing calculators too early in grade school. Second, it is virtually impossible to use a complicated software package over a long period of time without making some type of data-entry error. If we have no basic intuition as to what type of answer to

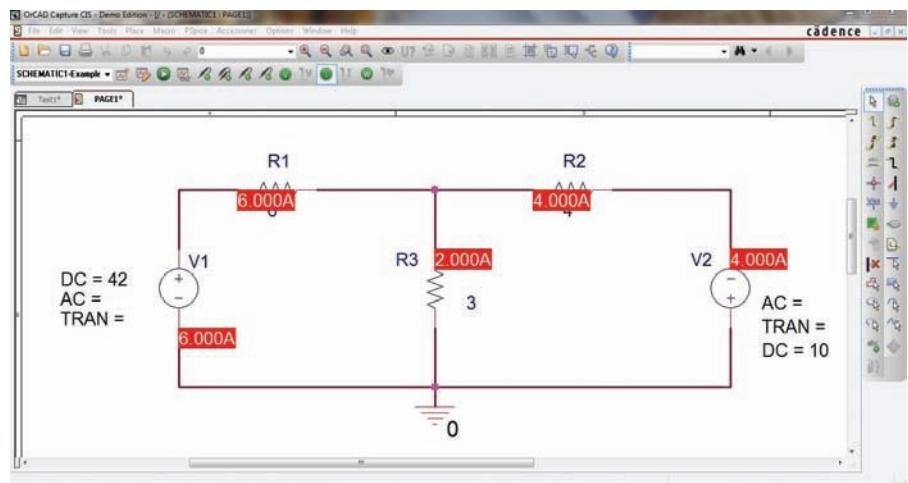




(a)



(b)



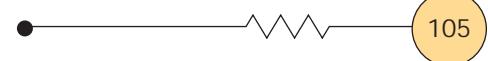
(c)

■ FIGURE 4.31 (a) Circuit of Fig. 4.15a drawn using Orcad schematic capture software. (b) Current, voltage, and power display buttons. (c) Circuit after simulation run, with current display enabled.

expect from a simulation, then there is no way to determine whether or not it is valid. Thus, the generic name really is a fairly accurate description: computer-aided analysis. Human brains are not obsolete. Not yet, anyway.

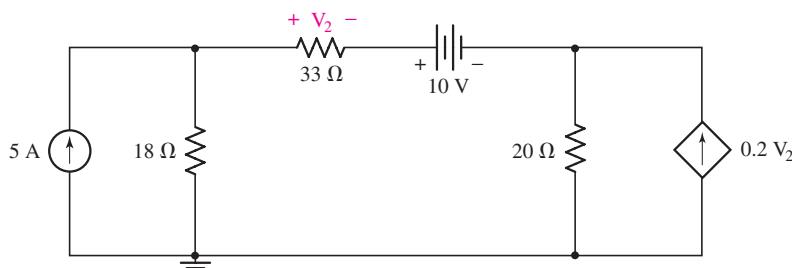
As an example, consider the circuit of Fig. 4.15b, which includes two dc voltage sources and three resistors. We wish to simulate this circuit using PSpice so that we may determine the currents  $i_1$  and  $i_2$ . Figure 4.31a shows the circuit as drawn using a schematic capture program.<sup>1</sup>

(1) Refer to Appendix 4 for a brief tutorial on PSpice and schematic capture.

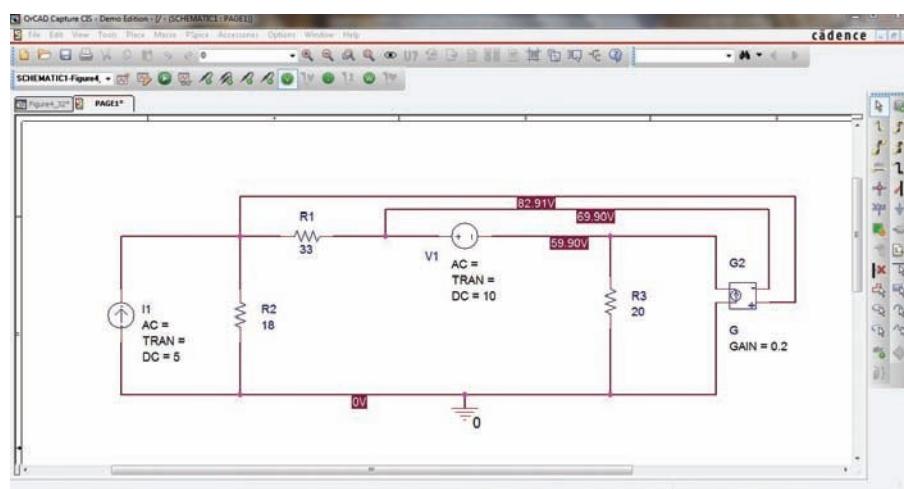


In order to determine the mesh currents, we need only run a bias point simulation. Under **PSpice**, select **New Simulation Profile**, type in a name (such as Example), and click on **Create**. Under the **Analysis type**: pull-down menu, select **Bias Point**, then click on **OK**. Returning to the original schematic window, under **PSpice** select **Run** (or use either of the two shortcuts: pressing the F11 key or clicking on the blue “Play” symbol). To see the currents calculated by PSpice, make sure the current button is selected (Fig. 4.31b). The results of our simulation are shown in Fig. 4.31c. We see that the two currents  $i_1$  and  $i_2$  are 6 A and 4 A, respectively, as we found previously.

As a further example, consider the circuit shown in Fig. 4.32a. It contains a dc voltage source, a dc current source, and a voltage-controlled current source. We are interested in the three nodal voltages, which from either nodal or mesh analysis are found to be 82.91 V, 69.9 V, and 59.9 V, respectively, as we move from left to right across the top of the circuit. Figure 4.32b shows this circuit after the simulation was performed. The three nodal voltages are indicated directly on the schematic. Note that in drawing a dependent source using the schematic capture tool, we must *explicitly* link two terminals of the source to the controlling voltage or current.



(a)



(b)

**FIGURE 4.32** (a) Circuit with dependent current source. (b) Circuit drawn using a schematic capture tool, with simulation results presented directly on the schematic.

# PRACTICAL APPLICATION

## Node-Based PSpice Schematic Creation

The most common method of describing a circuit in conjunction with computer-aided circuit analysis is with some type of graphical schematic drawing package, an example output of which was shown in Fig. 4.32. SPICE, however, was written before the advent of such software, and as such requires circuits to be described in a specific text-based format. The format has its roots in the syntax used for punch cards, which gives it a somewhat distinct appearance. The basis for circuit description is the definition of elements, each terminal of which is assigned a node number. So, although we have just studied two different generalized circuit analysis methods—the nodal and mesh techniques—it is interesting that SPICE and PSpice were written using a clearly defined nodal analysis approach.

```
* Example SPICE input deck for simple voltage divider circuit.  
.OP  
R1 1 2 1k  
R2 2 0 1k  
V1 1 0 DC 5  
  
* End of input deck.
```

.OP  
(Requests dc operating point)

R1 1 2 1k  
(Locates R1 between nodes 1 and 2; value is 1 kΩ)

R2 2 0 1k  
(Locates R2 between nodes 2 and 0; also 1 kΩ)

V1 1 0 DC 5  
(Locates 5 V source between nodes 1 and 0)

We can create the input deck by using the Notepad program from Windows or our favorite text editor. Saving the file under the name example.cir, we next invoke PSpice A/D (see Appendix 4). Under **File**, we choose **Open**, locate the directory in which we saved our file example.cir, and for **Files of type:** select **Circuit Files (\*.cir)**. After selecting our file and clicking **Open**, we see the PSpice A/D window with our circuit file loaded (Fig. 4.33a). A netlist such as this, containing instructions for the simulation to be performed, can be created by schematic capture software or created manually as in this example.

We run the simulation by either clicking the green “play” symbol at the top right, or selecting **Run** under **Simulation**.

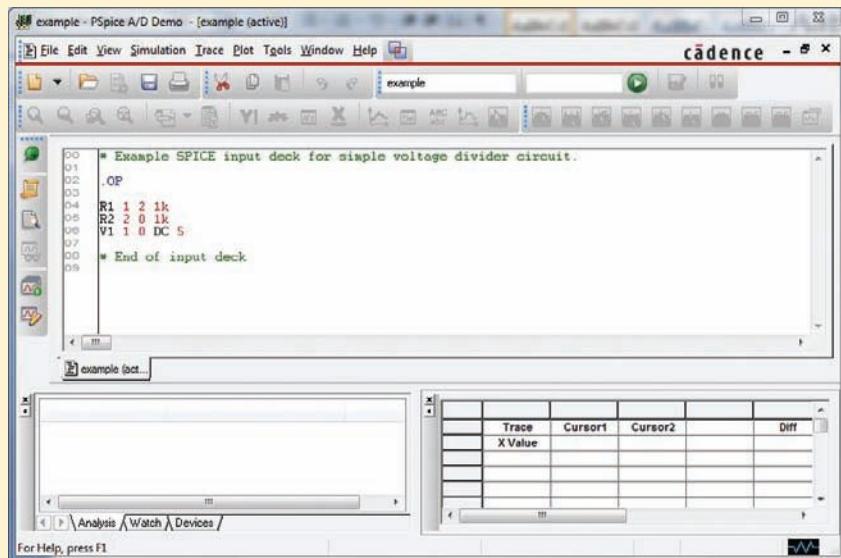
Even though modern circuit analysis is largely done using graphics-oriented interactive software, when errors are generated (usually due to a mistake in drawing the schematic or in selecting a combination of analysis options), the ability to read the text-based “input deck” generated by the schematic capture tool can be invaluable in tracking down the specific problem. The easiest way to develop such an ability is to learn how to run PSpice directly from a user-written input deck.

Consider, for example, the sample input deck below (lines beginning with an asterisk are comments, and are skipped by SPICE).

To view the results, we select **Output File** from under the **View** menu, which provides the window shown in Fig. 4.33b. Here it is worth noting that the output provides the expected nodal voltages (5 V at node 1, 2.5 V across resistor R2), but the current is quoted using the passive sign convention (i.e., -2.5 mA).

Text-based schematic entry is reasonably straightforward, but for complex (large number of elements) circuits, it can quickly become cumbersome. It is also easy to misnumber nodes, an error that can be difficult to isolate. However, reading the input and output files is often helpful when running simulations, so some experience with this format is useful.

At this point, the real power of computer-aided analysis begins to be apparent: Once you have the circuit drawn in the schematic capture program, it is easy to experiment by simply changing component values and observing the effect on currents and voltages. To gain a little experience at this point, try simulating any of the circuits shown in previous examples and practice problems.



(a)

The screenshot shows the PSpice A/D Demo window titled "example - PSpice A/D Demo - [example.out.1]". The main window displays the simulation output:

```

01 **** 07/27/10 14:51:28 ***** PSpice Lite (June 2009) ***** ID# 10013 ****
02
03 * Example SPICE input deck for simple voltage divider circuit
04
05 **** SMALL SIGNAL BIAS SOLUTION TEMPERATURE = 27.000 DEG C
06 ****
07 ****
08 ****
09 ****
10 ****
11 ****
12 ****
13 ****
14 ****
15 ****
16 ****
17 ****
18 ****
19 ****
20 ****
21 ****
22 ****
23 ****
24 ****
25 ****
26 ****
27 ****
28 ****
29 ****
30 ****
31 ****
32 ****
33 ****
34 ****
35 ****
36 ****
37 ****
38 ****
39 ****
40 ****
41 ****
42 ****
43 ****
44 ****
45 ****
46 ****
47 ****
48 ****
49 ****
50 ****
51 ****
52 ****
53 ****
54 ****
55 ****
56 ****
57 ****
58 ****
59 ****
60 ****
61 ****
62 ****
63 ****
64 ****
65 ****
66 ****
67 ****
68 ****
69 ****
70 ****
71 ****
72 ****
73 ****
74 ****
75 ****
76 ****
77 ****
78 ****
79 ****
80 ****
81 ****
82 ****
83 ****
84 ****
85 ****
86 ****
87 ****
88 ****
89 ****
90 ****
91 ****
92 ****
93 ****
94 ****
95 ****
96 ****
97 ****
98 ****
99 ****

```

The output includes nodal voltages, voltage source currents, and total power dissipation. Below the output window, there is a status bar showing the file path "C:\Users\Owner\Desktop\HKD8E\_Files\CH4\MISC\example.out.1" and a zoom level of "100%". To the right of the output window, there is a small window titled "example.out.1" showing a table with columns "Trace", "Cursor1", "Cursor2", "Diff", and "Max".

(b)

**FIGURE 4.33** (a) PSpice A/D window after the input deck describing our voltage divider is loaded.  
(b) Output window, showing nodal voltages and current from the source (but quoted using the passive sign convention). Note that the voltage across R1 requires post-simulation subtraction.

## SUMMARY AND REVIEW

Although Chap. 3 introduced KCL and KVL, both of which are sufficient to enable us to analyze any circuit, a more methodical approach proves helpful in everyday situations. Thus, in this chapter we developed the nodal analysis technique based on KCL, which results in a voltage at each node

(with respect to some designated “reference” node). We generally need to solve a system of simultaneous equations, unless voltage sources are connected so that they automatically provide nodal voltages. The controlling quantity of a *dependent* source is written down just as we would write down the numerical value of an “*independent*” source. Typically an additional equation is then required, unless the dependent source is controlled by a nodal voltage. When a voltage source bridges two nodes, the basic technique can be extended by creating a *supernode*; KCL dictates that the sum of the currents flowing into a group of connections so defined is equal to the sum of the currents flowing out.

As an alternative to nodal analysis, the mesh analysis technique was developed through application of KVL; it yields the complete set of *mesh* currents, which do not always represent the *net* current flowing through any particular element (for example, if an element is shared by two meshes). The presence of a current source will simplify the analysis if it lies on the periphery of a mesh; if the source is shared, then the *supermesh* technique is best. In that case, we write a KVL equation around a path that avoids the shared current source, then algebraically link the two corresponding mesh currents using the source.

A common question is: “*Which analysis technique should I use?*” We discussed some of the issues that might go into choosing a technique for a given circuit. These included whether or not the circuit is planar, what types of sources are present and how they are connected, and also what specific information is required (i.e., a voltage, current, or power). For complex circuits, it may take a greater effort than it is worth to determine the “optimum” approach, in which case most people will opt for the method with which they feel most comfortable. We concluded the chapter by introducing PSpice, a common circuit simulation tool, which is very useful for checking our results.

At this point we wrap up by identifying key points of this chapter to review, along with relevant example(s).

- ❑ Start each analysis with a neat, simple circuit diagram. Indicate all element and source values. (Example 4.1)
- ❑ For nodal analysis,
  - ❑ Choose one node as the reference node. Then label the node voltages  $v_1, v_2, \dots, v_{N-1}$ . Each is understood to be measured with respect to the reference node. (Examples 4.1, 4.2)
  - ❑ If the circuit contains only current sources, apply KCL at each nonreference node. (Examples 4.1, 4.2)
  - ❑ If the circuit contains voltage sources, form a supernode about each one, and then apply KCL at all nonreference nodes and supernodes. (Examples 4.5, 4.6)
- ❑ For mesh analysis, first make certain that the network is a planar network.
  - ❑ Assign a clockwise mesh current in each mesh:  $i_1, i_2, \dots, i_M$ . (Example 4.7)
  - ❑ If the circuit contains only voltage sources, apply KVL around each mesh. (Examples 4.7, 4.8, 4.9)
  - ❑ If the circuit contains current sources, create a supermesh for each one that is common to two meshes, and then apply KVL around each mesh and supermesh. (Examples 4.11, 4.12)

- Dependent sources will add an additional equation to nodal analysis if the controlling variable is a current, but not if the controlling variable is a nodal voltage. (Conversely, a dependent source will add an additional equation to mesh analysis if the controlling variable is a voltage, but not if the controlling variable is a mesh current). (Examples 4.3, 4.4, 4.6, 4.9, 4.10, 4.12)
- In deciding whether to use nodal or mesh analysis for a planar circuit, a circuit with fewer nodes/supernodes than meshes/supermeshes will result in fewer equations using nodal analysis.
- Computer-aided analysis is useful for checking results and analyzing circuits with large numbers of elements. However, common sense must be used to check simulation results.

## READING FURTHER

A detailed treatment of nodal and mesh analysis can be found in:

R. A. DeCarlo and P. M. Lin, *Linear Circuit Analysis*, 2nd ed. New York: Oxford University Press, 2001.

A solid guide to SPICE is

P. Tuinenga, *SPICE: A Guide to Circuit Simulation and Analysis Using PSPICE*, 3rd ed. Upper Saddle River, N.J.: Prentice-Hall, 1995.

## EXERCISES

### 4.1 Nodal Analysis

1. Solve the following systems of equations:

$$(a) 2v_2 - 4v_1 = 9 \text{ and } v_1 - 5v_2 = -4;$$

$$(b) -v_1 + 2v_3 = 8; 2v_1 + v_2 - 5v_3 = -7; 4v_1 + 5v_2 + 8v_3 = 6.$$

2. Evaluate the following determinants:

$$(a) \begin{vmatrix} 2 & 1 \\ -4 & 3 \end{vmatrix} \quad (b) \begin{vmatrix} 0 & 2 & 11 \\ 6 & 4 & 1 \\ 3 & -1 & 5 \end{vmatrix}.$$

3. Employ Cramer's rule to solve for  $v_2$  in each part of Exercise 1.

4. (a) Solve the following system of equations:

$$3 = \frac{v_1}{5} - \frac{v_2 - v_1}{22} + \frac{v_1 - v_3}{3}$$

$$2 - 1 = \frac{v_2 - v_1}{22} + \frac{v_2 - v_3}{14}$$

$$0 = \frac{v_3}{10} + \frac{v_3 - v_1}{3} + \frac{v_3 - v_2}{14}$$

(b) Verify your solution using MATLAB.

5. (a) Solve the following system of equations:

$$7 = \frac{v_1}{2} - \frac{v_2 - v_1}{12} + \frac{v_1 - v_3}{19}$$

$$15 = \frac{v_2 - v_1}{12} + \frac{v_2 - v_3}{2}$$

$$4 = \frac{v_3}{7} + \frac{v_3 - v_1}{19} + \frac{v_3 - v_2}{2}$$

(b) Verify your solution using MATLAB.





6. Correct (and verify by running) the following MATLAB code:

```
>> e1 = '3 = v/7 - (v2 - v1)/2 + (v1 - v3)/3';
>> e2 = '2 = (v2 - v1)/2 + (v2 - v3)/14';
>> e '0 = v3/10 + (v3 - v1)/3 + (v3 - v2)/14';
>>
>> a = solve(e e2 e3, 'v1', v2, 'v3')
```

7. Identify the obvious errors in the following complete set of nodal equations if the last equation is known to be correct:

$$7 = \frac{v_1}{4} - \frac{v_2 - v}{1} + \frac{v_1 - v_3}{9}$$

$$0 = \frac{v_2 - v_1}{2} + \frac{v_2 - v_3}{2}$$

$$4 = \frac{v_3}{7} + \frac{v_3 - v_1}{19} + \frac{v_3 - v_2}{2}$$

8. In the circuit of Fig. 4.34, determine the current labeled  $i$  with the assistance of nodal analysis techniques.

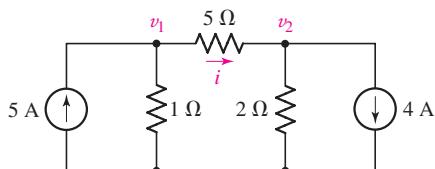


FIGURE 4.34

9. Calculate the power dissipated in the  $1 \Omega$  resistor of Fig. 4.35.

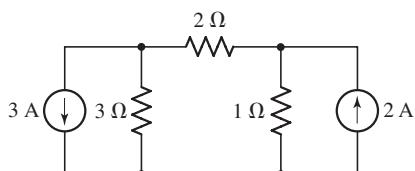


FIGURE 4.35

10. With the assistance of nodal analysis, determine  $v_1 - v_2$  in the circuit shown in Fig. 4.36.

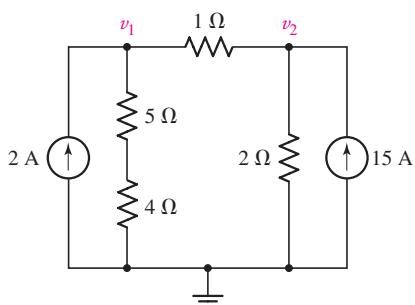
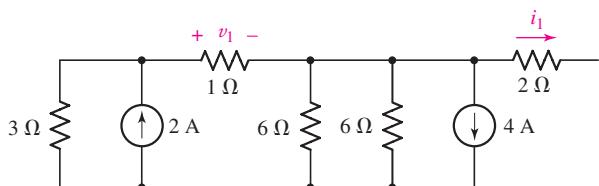


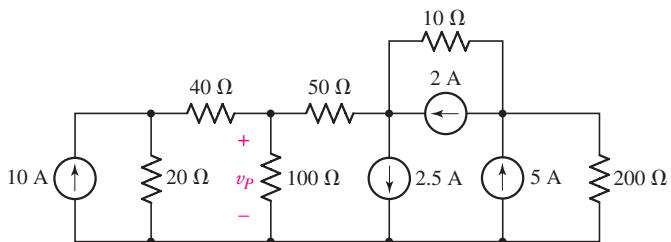
FIGURE 4.36

11. For the circuit of Fig. 4.37, determine the value of the voltage labeled  $v_1$  and the current labeled  $i_1$ .



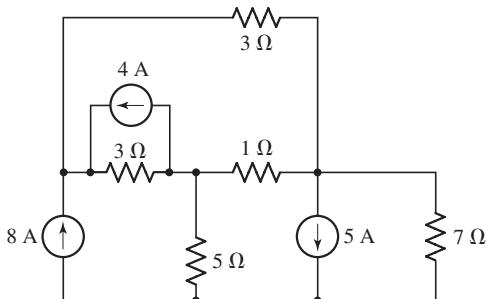
■ FIGURE 4.37

12. Use nodal analysis to find  $v_P$  in the circuit shown in Fig. 4.38.



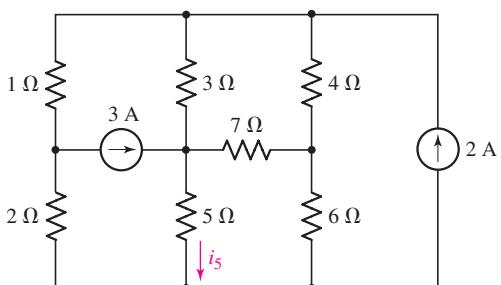
■ FIGURE 4.38

13. Using the bottom node as reference, determine the voltage across the  $5 \Omega$  resistor in the circuit of Fig. 4.39, and calculate the power dissipated by the  $7 \Omega$  resistor.



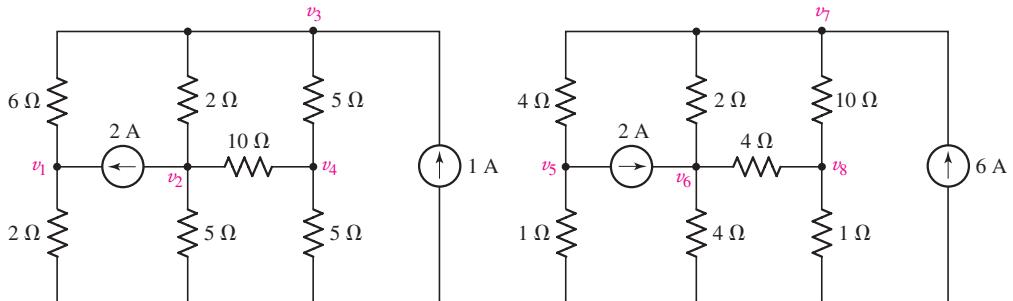
■ FIGURE 4.39

14. For the circuit of Fig. 4.40, use nodal analysis to determine the current  $i_5$ .



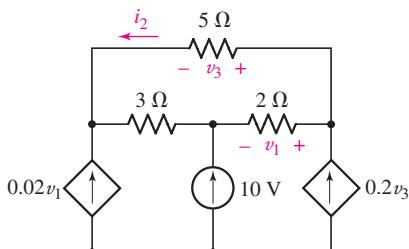
■ FIGURE 4.40

15. Determine a numerical value for each nodal voltage in the circuit of Fig. 4.41.

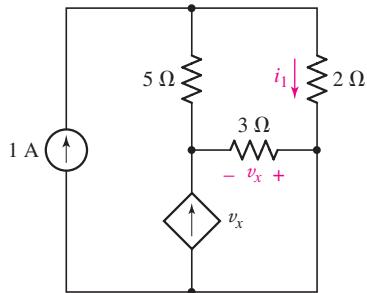


■ FIGURE 4.41

16. Determine the current  $i_2$  as labeled in the circuit of Fig. 4.42, with the assistance of nodal analysis.



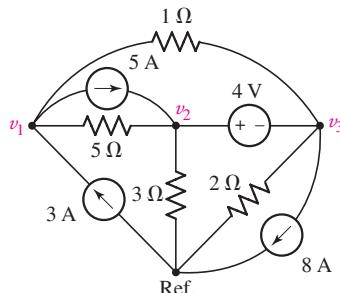
■ FIGURE 4.42



■ FIGURE 4.43

17. Using nodal analysis as appropriate, determine the current labeled  $i_1$  in the circuit of Fig. 4.43.

## 4.2 The Supernode

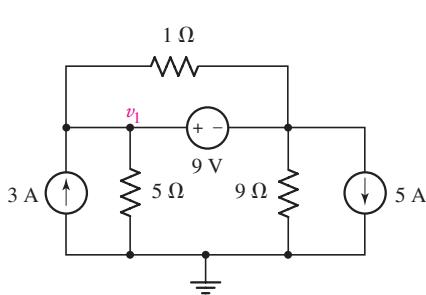


■ FIGURE 4.44

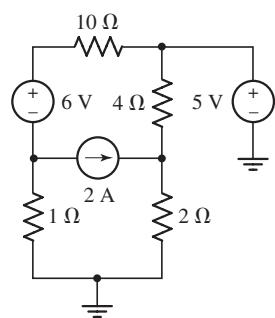
18. Determine the nodal voltages as labeled in Fig. 4.44, making use of the supernode technique as appropriate.

19. For the circuit shown in Fig. 4.45, determine a numerical value for the voltage labeled  $v_1$ .

20. For the circuit of Fig. 4.46, determine all four nodal voltages.

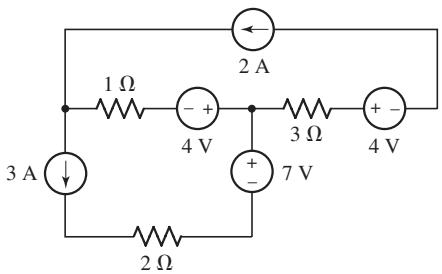


■ FIGURE 4.45



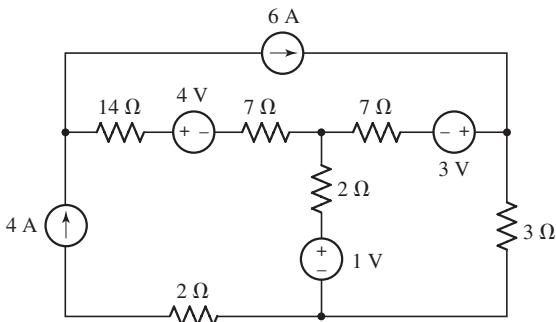
■ FIGURE 4.46

21. Employing supernode/nodal analysis techniques as appropriate, determine the power dissipated by the  $1\ \Omega$  resistor in the circuit of Fig. 4.47.



■ FIGURE 4.47

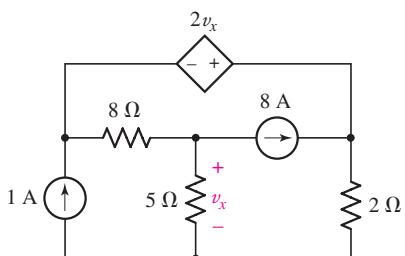
22. Referring to the circuit of Fig. 4.48, obtain a numerical value for the power supplied by the  $1\text{ V}$  source.



■ FIGURE 4.48

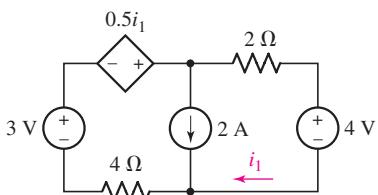
23. Determine the voltage labeled  $v$  in the circuit of Fig. 4.49.

24. Determine the voltage  $v_x$  in the circuit of Fig. 4.50, and the power supplied by the  $1\text{ A}$  source.

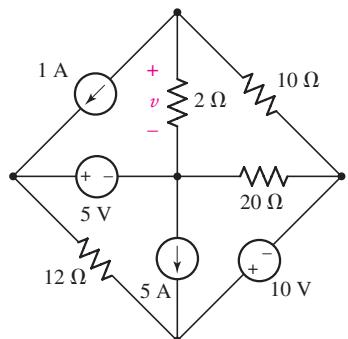


■ FIGURE 4.50

25. Consider the circuit of Fig. 4.51. Determine the current labeled  $i_1$ .

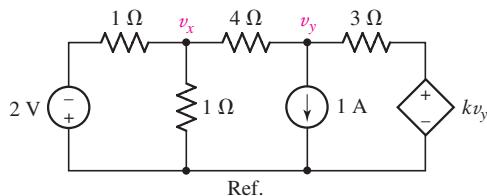


■ FIGURE 4.51



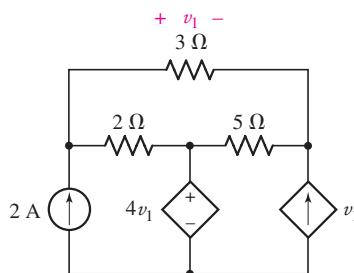
■ FIGURE 4.49

26. Determine the value of  $k$  that will result in  $v_x$  being equal to zero in the circuit of Fig. 4.52.



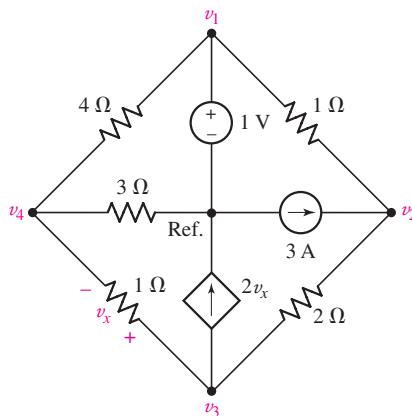
■ FIGURE 4.52

27. For the circuit depicted in Fig. 4.53, determine the voltage labeled  $v_1$  across the  $3 \Omega$  resistor.



■ FIGURE 4.53

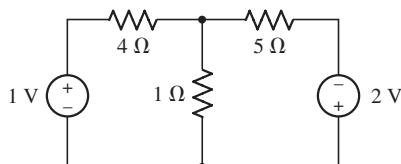
28. For the circuit of Fig. 4.54, determine all four nodal voltages.



■ FIGURE 4.54

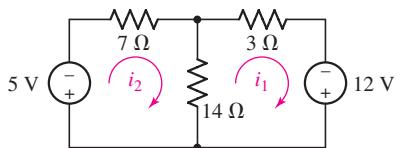
### 4.3 Mesh Analysis

29. Determine the currents flowing out of the positive terminal of each voltage source in the circuit of Fig. 4.55.



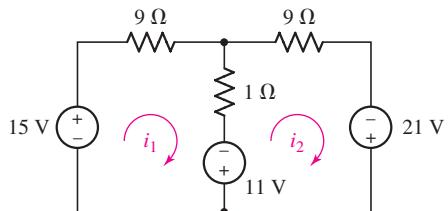
■ FIGURE 4.55

30. Obtain numerical values for the two mesh currents  $i_1$  and  $i_2$  in the circuit shown in Fig. 4.56.



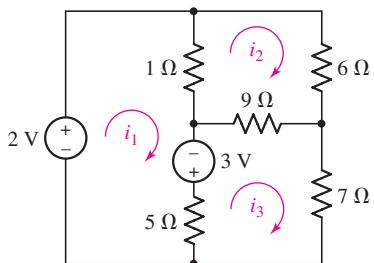
■ FIGURE 4.56

31. Use mesh analysis as appropriate to determine the two mesh currents labeled in Fig. 4.57.



■ FIGURE 4.57

32. Determine numerical values for each of the three mesh currents as labeled in the circuit diagram of Fig. 4.58.

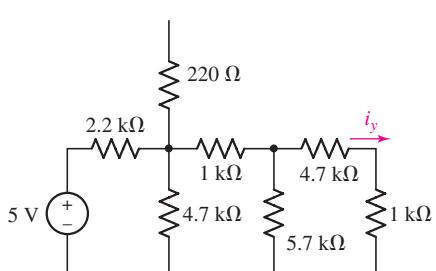


■ FIGURE 4.58

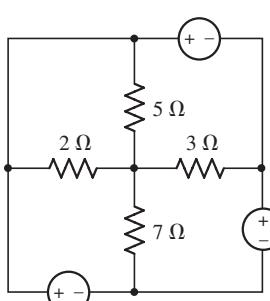
33. Calculate the power dissipated by each resistor in the circuit of Fig. 4.58.

34. Employing mesh analysis as appropriate, obtain (a) a value for the current  $i_y$  and (b) the power dissipated by the  $220 \Omega$  resistor in the circuit of Fig. 4.59.

35. Choose nonzero values for the three voltage sources of Fig. 4.60 so that no current flows through any resistor in the circuit.

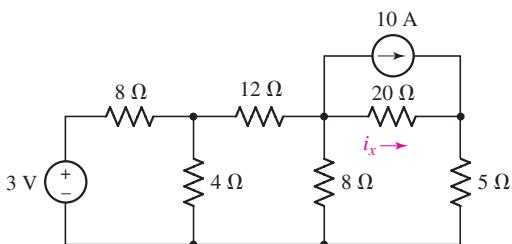


■ FIGURE 4.59



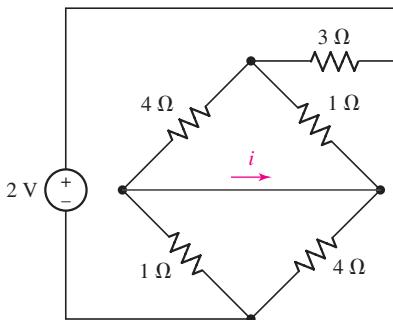
■ FIGURE 4.60

36. Calculate the current  $i_x$  in the circuit of Fig. 4.61.



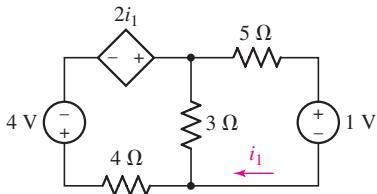
■ FIGURE 4.61

37. Employing mesh analysis procedures, obtain a value for the current labeled  $i$  in the circuit represented by Fig. 4.62.



■ FIGURE 4.62

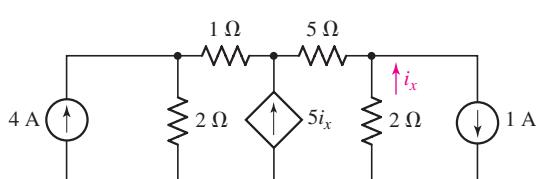
38. Determine the power dissipated in the  $4 \Omega$  resistor of the circuit shown in Fig. 4.63.



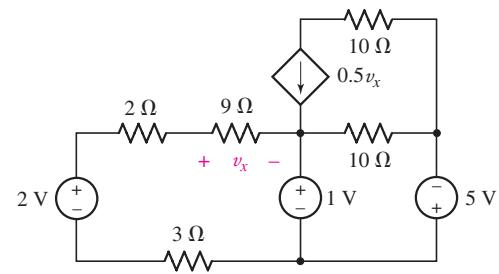
■ FIGURE 4.63

39. (a) Employ mesh analysis to determine the power dissipated by the  $1 \Omega$  resistor in the circuit represented schematically by Fig. 4.64. (b) Check your answer using nodal analysis.

40. Define three clockwise mesh currents for the circuit of Fig. 4.65, and employ mesh analysis to obtain a value for each.

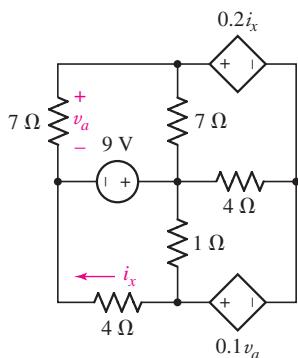


■ FIGURE 4.64



■ FIGURE 4.65

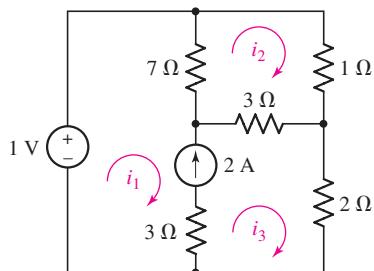
41. Employ mesh analysis to obtain values for  $i_x$  and  $v_a$  in the circuit of Fig. 4.66.



■ FIGURE 4.66

#### 4.4 The Supermesh

42. Determine values for the three mesh currents of Fig. 4.67.

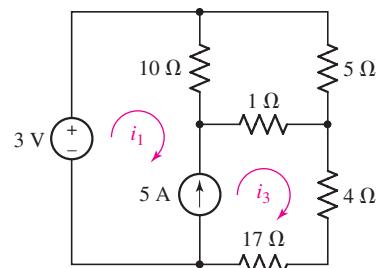


■ FIGURE 4.67

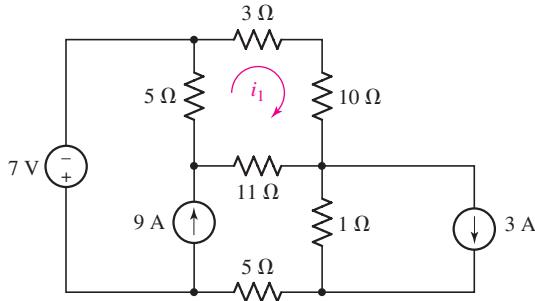
43. Through appropriate application of the supermesh technique, obtain a numerical value for the mesh current  $i_3$  in the circuit of Fig. 4.68, and calculate the power dissipated by the  $1 \Omega$  resistor.

44. For the circuit of Fig. 4.69, determine the mesh current  $i_1$  and the power dissipated by the  $1 \Omega$  resistor.

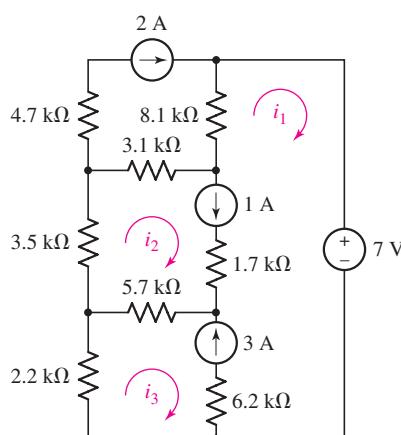
45. Calculate the three mesh currents labeled in the circuit diagram of Fig. 4.70.



■ FIGURE 4.68

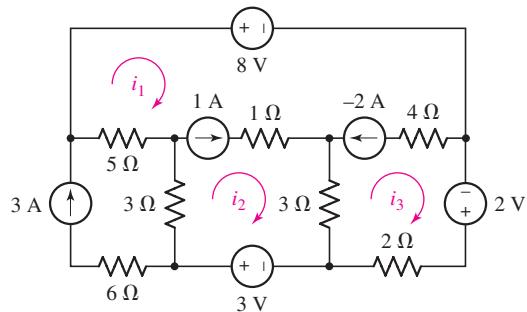


■ FIGURE 4.69



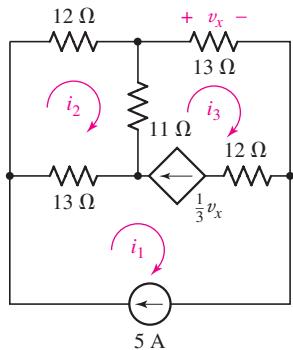
■ FIGURE 4.70

46. Employing the supermesh technique to best advantage, obtain numerical values for each of the mesh currents identified in the circuit depicted in Fig. 4.71.

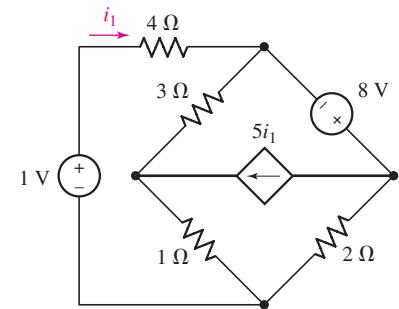


■ FIGURE 4.71

47. Through careful application of the supermesh technique, obtain values for all three mesh currents as labeled in Fig. 4.72.

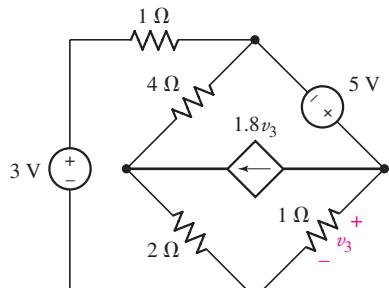


■ FIGURE 4.72

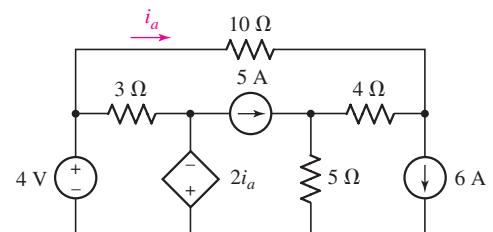


■ FIGURE 4.73

48. Determine the power supplied by the 1 V source in Fig. 4.73.  
 49. Define three clockwise mesh currents for the circuit of Fig. 4.74, and employ the supermesh technique to obtain a numerical value for each.  
 50. Determine the power absorbed by the  $10 \Omega$  resistor in Fig. 4.75.



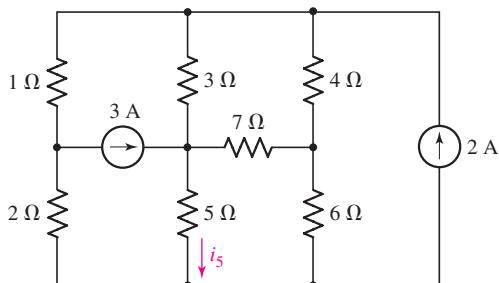
■ FIGURE 4.74



■ FIGURE 4.75

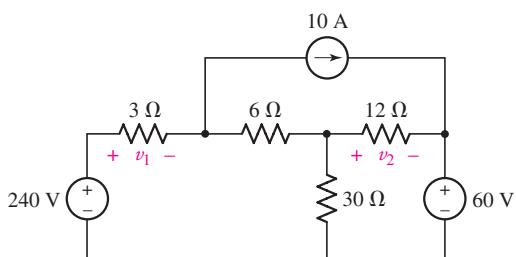
#### 4.5 Nodal vs. Mesh Analysis: A Comparison

51. For the circuit represented schematically in Fig. 4.76: (a) How many nodal equations would be required to determine  $i_5$ ? (b) Alternatively, how many mesh equations would be required? (c) Would your preferred analysis method change if only the voltage across the  $7\ \Omega$  resistor were needed? Explain.



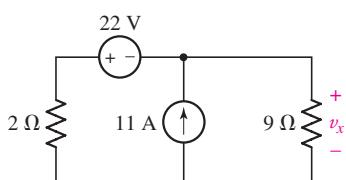
■ FIGURE 4.76

52. The circuit of Fig. 4.76 is modified such that the 3 A source is replaced by a 3 V source whose positive reference terminal is connected to the  $7\ \Omega$  resistor. (a) Determine the number of nodal equations required to determine  $i_5$ . (b) Alternatively, how many mesh equations would be required? (c) Would your preferred analysis method change if only the voltage across the  $7\ \Omega$  resistor were needed? Explain.
53. The circuit of Fig. 4.77 contains three sources. (a) As presently drawn, would nodal or mesh analysis result in fewer equations to determine the voltages  $v_1$  and  $v_2$ ? Explain. (b) If the voltage source were replaced with current sources, and the current source replaced with a voltage source, would your answer to part (a) change? Explain?



■ FIGURE 4.77

54. Solve for the voltage  $v_x$  as labeled in the circuit of Fig. 4.78 using (a) mesh analysis. (b) Repeat using nodal analysis. (c) Which approach was easier, and why?



■ FIGURE 4.78

55. Consider the five-source circuit of Fig. 4.79. Determine the total number of simultaneous equations that must be solved in order to determine  $v_1$  using (a) nodal analysis; (b) mesh analysis. (c) Which method is preferred, and does it depend on which side of the  $40\ \Omega$  resistor is chosen as the reference node? Explain your answer.

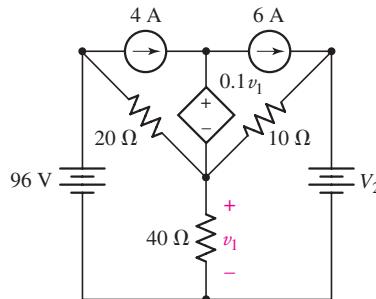


FIGURE 4.79

56. Replace the dependent voltage source in the circuit of Fig. 4.79 with a dependent current source oriented such that the arrow points upward. The controlling expression  $0.1 v_1$  remains unchanged. The value  $V_2$  is zero. (a) Determine the total number of simultaneous equations required to obtain the power dissipated by the  $40\ \Omega$  resistor if nodal analysis is employed. (b) Is mesh analysis preferred instead? Explain.
57. After studying the circuit of Fig. 4.80, determine the total number of simultaneous equations that must be solved to determine voltages  $v_1$  and  $v_3$  using (a) nodal analysis; (b) mesh analysis.

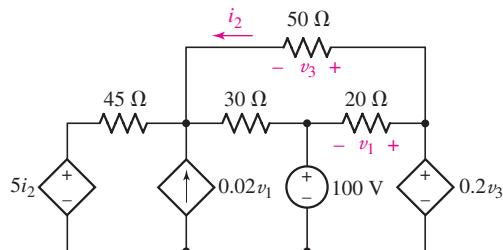


FIGURE 4.80

58. From the perspective of determining voltages and currents associated with all components, (a) design a five-node, four-mesh circuit that is analyzed more easily using nodal techniques. (b) Modify your circuit by replacing only one component such that it is now more easily analyzed using mesh techniques.

#### 4.6 Computer-Aided Circuit Analysis

59. Employ PSpice (or similar CAD tool) to verify the solution of Exercise 8. Submit a printout of a properly labeled schematic with the answer highlighted, along with your hand calculations.
60. Employ PSpice (or similar CAD tool) to verify the solution of Exercise 10. Submit a printout of a properly labeled schematic with the two nodal voltages highlighted, along with your hand calculations solving for the same quantities.
61. Employ PSpice (or similar CAD tool) to verify the voltage across the  $5\ \Omega$  resistor in the circuit of Exercise 13. Submit a printout of a properly labeled schematic with the answer highlighted, along with your hand calculations.

-  62. Verify numerical values for each nodal voltage in Exercise 15 by employing PSpice or a similar CAD tool. Submit a printout of an appropriately labeled schematic with the nodal voltages highlighted, along with your hand calculations.
-  63. Verify the numerical values for  $i_1$  and  $v_x$  as indicated in the circuit accompanying Exercise 17, using PSpice or a similar CAD tool. Submit a printout of a properly labeled schematic with the answers highlighted, along with hand calculations.
-  64. (a) Generate an input deck for SPICE to determine the voltage  $v_9$  as labeled in Fig. 4.81. Submit a printout of the output file with the solution highlighted.  
(b) Verify your answer by hand.

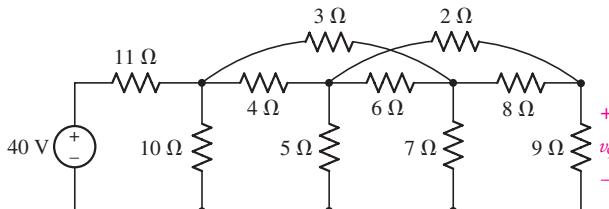


FIGURE 4.81

### Chapter-Integrating Exercises

-  65. (a) Design a circuit employing only 9 V batteries and standard 5% tolerance value resistors that provide voltages of 1.5 V, 4.5 V, and 5 V and at least one mesh current of 1 mA. (b) Verify your design using PSpice or similar CAD tool.
-  66. A decorative string of multicolored outdoor lights is installed on a home in a quiet residential area. After plugging the 12 V ac adapter into the electrical socket, the homeowner immediately notes that two bulbs are burned out.  
(a) Are the individual lights connected in series or parallel? Explain. (b) Simulate the string by writing a SPICE input deck, assuming 44 lights, 12 V dc power supply, 24 AWG soft solid copper wire, and individual bulbs rated at 10 mW each. Submit a printout of the output file, with the power supplied by the 12 V supply highlighted. (c) Verify your simulation with hand calculations.
-  67. Consider the circuit depicted in Fig. 4.82. Employ either nodal or mesh analysis as a design tool to obtain a value of 200 mA for  $i_1$ , if elements A, B, C, D, E, and F must be either current or voltage sources with nonzero values.

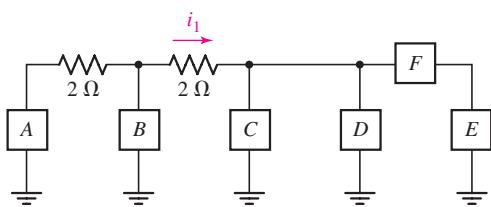
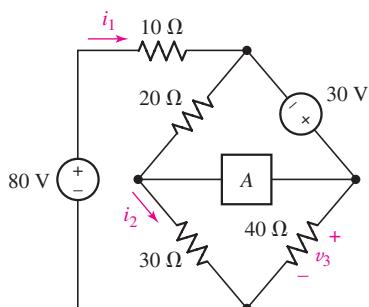


FIGURE 4.82

68. (a) Under what circumstances does the presence of an independent voltage source greatly simplify nodal analysis? Explain. (b) Under what circumstances does the presence of an independent current source significantly simplify mesh analysis? Explain. (c) On which fundamental physical principle do we base nodal analysis? (d) On which fundamental physical principle do we base mesh analysis?

69. Referring to Fig. 4.83, (a) determine whether nodal or mesh analysis is more appropriate in determining  $i_2$  if element A is replaced with a short circuit, then carry out the analysis. (b) Verify your answer with an appropriate PSpice simulation. Submit a properly labeled schematic along with the answer highlighted.



■ FIGURE 4.83

70. The element marked A in the circuit of Fig. 4.83 is replaced by a 2.5 V independent voltage source with the positive reference terminal connected to the common node of the 20 Ω and 30 Ω resistors. (a) Determine whether mesh or nodal analysis is more straightforward for determining the voltage marked  $v_3$ . (b) Verify your answer using PSpice. (c) Would your conclusion for part (a) change if the current  $i_2$  were required as well? Explain.

# Handy Circuit Analysis Techniques

## INTRODUCTION

The techniques of nodal and mesh analysis described in Chap. 4 are reliable and extremely powerful methods. However, both require that we develop a complete set of equations to describe a particular circuit as a general rule, even if only one current, voltage, or power quantity is of interest. In this chapter, we investigate several different techniques for isolating specific parts of a circuit in order to simplify the analysis. After examining each of these techniques, we focus on how one might go about selecting one method over another.

### 5.1 LINEARITY AND SUPERPOSITION

All of the circuits which we plan to analyze can be classified as *linear circuits*, so this is a good time to be more specific in defining exactly what we mean by that. Having done this, we can then consider the most important consequence of linearity, the principle of *superposition*. This principle is very basic and will appear repeatedly in our study of linear circuit analysis. As a matter of fact, the nonapplicability of superposition to nonlinear circuits is the very reason they are so difficult to analyze!

The principle of superposition states that the *response* (a desired current or voltage) in a linear circuit having more than one independent source can be obtained by adding the responses caused by the separate independent sources *acting alone*.

### Linear Elements and Linear Circuits

We define a *linear element* as a passive element that has a linear voltage-current relationship. By a “linear voltage-current relationship”

### KEY CONCEPTS

Superposition: Determining the *Individual Contributions* of Different Sources to Any Current or Voltage

Source Transformation as a Means of Simplifying Circuits

Thévenin's Theorem

Norton's Theorem

Thévenin and Norton Equivalent Networks

Maximum Power Transfer

$\Delta \leftrightarrow Y$  Transformations for Resistive Networks

Selecting a Particular Combination of Analysis Techniques

Performing dc Sweep Simulations Using PSpice



we simply mean that multiplication of the current through the element by a constant  $K$  results in the multiplication of the voltage across the element by the same constant  $K$ . At this time, only one passive element has been defined (the resistor), and its voltage-current relationship

$$v(t) = Ri(t)$$

is clearly linear. As a matter of fact, if  $v(t)$  is plotted as a function of  $i(t)$ , the result is a straight line.

We define a **linear dependent source** as a dependent current or voltage source whose output current or voltage is proportional only to the first power of a specified current or voltage variable in the circuit (or to the *sum* of such quantities).

We now define a **linear circuit** as a circuit composed entirely of independent sources, linear dependent sources, and linear elements. From this definition, it is possible to show<sup>1</sup> that “the response is proportional to the source,” or that multiplication of all independent source voltages and currents by a constant  $K$  increases all the current and voltage responses by the same factor  $K$  (including the dependent source voltage or current outputs).

## The Superposition Principle

The most important consequence of linearity is **superposition**.

Let us explore the superposition principle by considering first the circuit of Fig. 5.1, which contains two independent sources, the current generators that force the currents  $i_a$  and  $i_b$  into the circuit. Sources are often called *forcing functions* for this reason, and the nodal voltages that they produce can be termed *response functions*, or simply *responses*. Both the forcing functions and the responses may be functions of time. The two nodal equations for this circuit are

$$0.7v_1 - 0.2v_2 = i_a \quad [1]$$

$$-0.2v_1 + 1.2v_2 = i_b \quad [2]$$

Now let us perform experiment  $x$ . We change the two forcing functions to  $i_{ax}$  and  $i_{bx}$ ; the two unknown voltages will now be different, so we will call them  $v_{1x}$  and  $v_{2x}$ . Thus,

$$0.7v_{1x} - 0.2v_{2x} = i_{ax} \quad [3]$$

$$-0.2v_{1x} + 1.2v_{2x} = i_{bx} \quad [4]$$

We next perform experiment  $y$  by changing the source currents to  $i_{ay}$  and  $i_{by}$  and measure the responses  $v_{1y}$  and  $v_{2y}$ :

$$0.7v_{1y} - 0.2v_{2y} = i_{ay} \quad [5]$$

$$-0.2v_{1y} + 1.2v_{2y} = i_{by} \quad [6]$$

(1) The proof involves first showing that the use of nodal analysis on the linear circuit can produce only linear equations of the form

$$a_1v_1 + a_2v_2 + \dots + a_Nv_N = b$$

where the  $a_i$  are constants (combinations of resistance or conductance values, constants appearing in dependent source expressions, 0, or  $\pm 1$ ), the  $v_i$  are the unknown node voltages (responses), and  $b$  is an independent source value or a sum of independent source values. Given a set of such equations, if we multiply all the  $b$ 's by  $K$ , then it is evident that the solution of this new set of equations will be the node voltages  $Kv_1, Kv_2, \dots, Kv_N$ .

The dependent voltage source  $v_s = 0.6i_1 - 14v_2$  is linear, but  $v_s = 0.6i_1^2$  and  $v_s = 0.6i_1v_2$  are not.

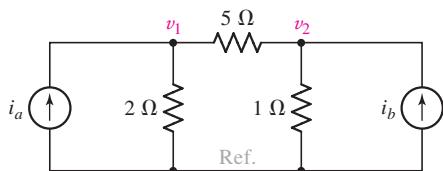


FIGURE 5.1 A circuit with two independent current sources.

These three sets of equations describe the same circuit with three different sets of source currents. Let us *add* or “*superpose*” the last two sets of equations. Adding Eqs. [3] and [5],

$$(0.7v_{1x} + 0.7v_{1y}) - (0.2v_{2x} + 0.2v_{2y}) = i_{ax} + i_{ay} \quad [7]$$

$$0.7v_1 - 0.2v_2 = i_a \quad [1]$$

and adding Eqs. [4] and [6],

$$-(0.2v_{1x} + 0.2v_{1y}) + (1.2v_{2x} + 1.2v_{2y}) = i_{bx} + i_{by} \quad [8]$$

$$-0.2v_1 + 1.2v_2 = i_b \quad [2]$$

where Eq. [1] has been written immediately below Eq. [7] and Eq. [2] below Eq. [8] for easy comparison.

The linearity of all these equations allows us to compare Eq. [7] with Eq. [1] and Eq. [8] with Eq. [2] and draw an interesting conclusion. If we select  $i_{ax}$  and  $i_{ay}$  such that their sum is  $i_a$  and select  $i_{bx}$  and  $i_{by}$  such that their sum is  $i_b$ , then the desired responses  $v_1$  and  $v_2$  may be found by adding  $v_{1x}$  to  $v_{1y}$  and  $v_{2x}$  to  $v_{2y}$ , respectively. In other words, we can perform experiment  $x$  and note the responses, perform experiment  $y$  and note the responses, and finally add the two sets of responses. This leads to the fundamental concept involved in the superposition principle: to look at each independent source (and the response it generates) one at a time with the other independent sources “turned off” or “zeroed out.”

If we reduce a voltage source to zero volts, we have effectively created a short circuit (Fig. 5.2a). If we reduce a current source to zero amps, we have effectively created an open circuit (Fig. 5.2b). Thus, the ***superposition theorem*** can be stated as:

In any linear resistive network, the voltage across or the current through any resistor or source may be calculated by adding algebraically all the individual voltages or currents caused by the separate independent sources acting alone, with all other independent voltage sources replaced by short circuits and all other independent current sources replaced by open circuits.

Thus, if there are  $N$  independent sources, we must perform  $N$  experiments, each having only one of the independent sources active and the others inactive/turned off/zeroed out. Note that *dependent* sources are in general active in every experiment.

There is also no reason that an independent source must assume only its given value or a zero value in the several experiments; it is necessary only for the sum of the several values to be equal to the original value. An inactive source almost always leads to the simplest circuit, however.

The circuit we have just used as an example should indicate that a much stronger theorem might be written; a *group* of independent sources may be made active and inactive collectively, if we wish. For example, suppose there are three independent sources. The theorem states that we may find a given response by considering each of the three sources acting alone and adding the three results. Alternatively, we may find the response due to the first and second sources operating with the third inactive, and then add to this the response caused by the third source acting alone. This amounts to treating several sources collectively as a sort of “supersource.”

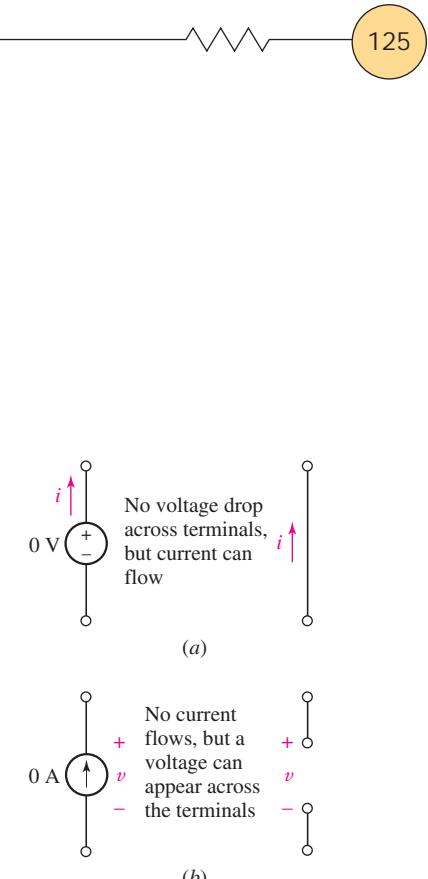


FIGURE 5.2 (a) A voltage source set to zero acts like a short circuit. (b) A current source set to zero acts like an open circuit.



## EXAMPLE 5.1

For the circuit of Fig. 5.3a, use superposition to determine the unknown branch current  $i_x$ .

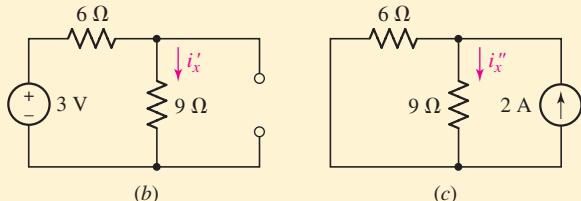
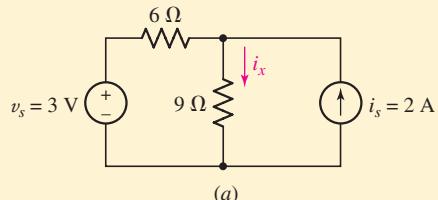


FIGURE 5.3 (a) An example circuit with two independent sources for which the branch current  $i_x$  is desired; (b) same circuit with current source open-circuited; (c) original circuit with voltage source short-circuited.

First set the current source equal to zero and redraw the circuit as shown in Fig. 5.3b. The portion of  $i_x$  due to the voltage source has been designated  $i'_x$  to avoid confusion and is easily found to be 0.2 A.

Next set the voltage source in Fig. 5.3a to zero and again redraw the circuit, as shown in Fig. 5.3c. Current division lets us determine that  $i''_x$  (the portion of  $i_x$  due to the 2 A current source) is 0.8 A.

Now compute the total current  $i_x$  by adding the two individual components:

$$i_x = i_{x|3V} + i_{x|2A} = i'_x + i''_x$$

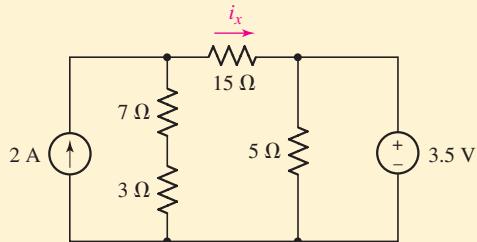
or

$$i_x = \frac{3}{6+9} + 2 \left( \frac{6}{6+9} \right) = 0.2 + 0.8 = 1.0 \text{ A}$$

Another way of looking at Example 5.1 is that the 3 V source and the 2 A source are each performing work on the circuit, resulting in a total current  $i_x$  flowing through the 9 Ω resistor. However, the contribution of the 3 V source to  $i_x$  does not depend on the contribution of the 2 A source, and vice versa. For example, if we double the output of the 2 A source to 4 A, it will now contribute 1.6 A to the total current  $i_x$  flowing through the 9 Ω resistor. However, the 3 V source will still contribute only 0.2 A to  $i_x$ , for a new total current of  $0.2 + 1.6 = 1.8$  A.

**PRACTICE**

5.1 For the circuit of Fig. 5.4, use superposition to compute the current  $i_x$ .



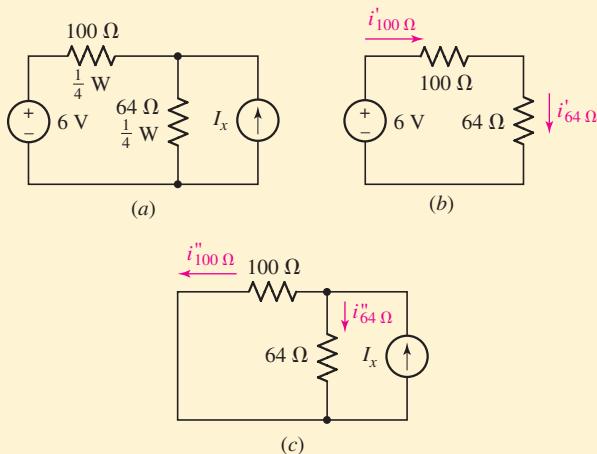
■ FIGURE 5.4

Ans: 660 mA.

As we will see, superposition does not generally reduce our workload when considering a particular circuit, since it leads to the analysis of several new circuits to obtain the desired response. However, it is particularly useful in identifying the significance of various parts of a more complex circuit. It also forms the basis of phasor analysis, which is introduced in Chap. 10.

**EXAMPLE 5.2**

Referring to the circuit of Fig. 5.5a, determine the maximum positive current to which the source  $I_x$  can be set before any resistor exceeds its power rating and overheats.



■ FIGURE 5.5 (a) A circuit with two resistors each rated at  $\frac{1}{4}$  W. (b) Circuit with only the 6 V source active. (c) Circuit with the source  $I_x$  active.

► **Identify the goal of the problem.**

Each resistor is rated to a maximum of 250 mW. If the circuit allows this value to be exceeded (by forcing too much current through either resistor), excessive heating will occur—possibly leading to

(Continued on next page)

an accident. The 6 V source cannot be changed, so we are looking for an equation involving  $I_x$  and the maximum current through each resistor.

► **Collect the known information.**

Based on its 250 mW power rating, the maximum current the 100  $\Omega$  resistor can tolerate is

$$\sqrt{\frac{P_{\max}}{R}} = \sqrt{\frac{0.250}{100}} = 50 \text{ mA}$$

and, similarly, the current through the 64  $\Omega$  resistor must be less than 62.5 mA.

► **Devise a plan.**

Either nodal or mesh analysis may be applied to the solution of this problem, but superposition may give us a slight edge, since we are primarily interested in the effect of the current source.

► **Construct an appropriate set of equations.**

Using superposition, we redraw the circuit as in Fig. 5.5b and find that the 6 V source contributes a current

$$i'_{100 \Omega} = \frac{6}{100 + 64} = 36.59 \text{ mA}$$

to the 100  $\Omega$  resistor and, since the 64  $\Omega$  resistor is in series,  $i'_{64 \Omega} = 36.59$  mA as well.

Recognizing the current divider in Fig. 5.5c, we note that  $i''_{64 \Omega}$  will add to  $i'_{64 \Omega}$ , but  $i''_{100 \Omega}$  is opposite in direction to  $i'_{100 \Omega}$ . Therefore,  $I_x$  can safely contribute  $62.5 - 36.59 = 25.91$  mA to the 64  $\Omega$  resistor current, and  $50 - (-36.59) = 86.59$  mA to the 100  $\Omega$  resistor current.

The 100  $\Omega$  resistor therefore places the following constraint on  $I_x$ :

$$I_x < (86.59 \times 10^{-3}) \left( \frac{100 + 64}{64} \right)$$

and the 64  $\Omega$  resistor requires that

$$I_x < (25.91 \times 10^{-3}) \left( \frac{100 + 64}{100} \right)$$

► **Attempt a solution.**

Considering the 100  $\Omega$  resistor first, we see that  $I_x$  is limited to  $I_x < 221.9$  mA. The 64  $\Omega$  resistor limits  $I_x$  such that  $I_x < 42.49$  mA. In order to satisfy both constraints,  $I_x$  must be less than 42.49 mA. If the value is increased, the 64  $\Omega$  resistor will overheat long before the 100  $\Omega$  resistor does.

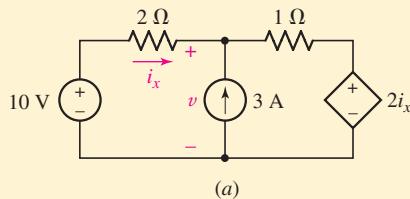
► **Verify the solution. Is it reasonable or expected?**

One particularly useful way to evaluate our solution is to perform a dc sweep analysis in PSpice as described after the next example. An interesting question, however, is whether we would have expected the 64  $\Omega$  resistor to overheat first.

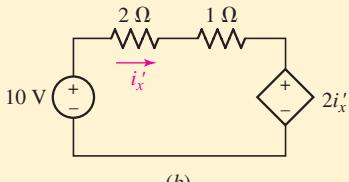
Originally we found that the 100  $\Omega$  resistor has a smaller maximum current, so it might be reasonable to expect it to limit  $I_x$ . However, because  $I_x$  opposes the current sent by the 6 V source through the 100  $\Omega$  resistor but adds to the 6 V source's contribution to the current through the 64  $\Omega$  resistor, it turns out to work the other way—it's the 64  $\Omega$  resistor that sets the limit on  $I_x$ .

## EXAMPLE 5.3

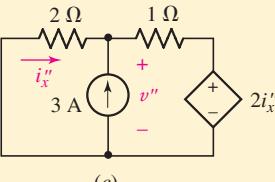
In the circuit of Fig. 5.6a, use the superposition principle to determine the value of  $i_x$ .



(a)



(b)



(c)

FIGURE 5.6 (a) An example circuit with two independent sources and one dependent source for which the branch current  $i_x$  is desired. (b) Circuit with the 3 A source open-circuited. (c) Original circuit with the 10 V source short-circuited.

First open-circuit the 3 A source (Fig. 5.6b). The single mesh equation is

$$-10 + 2i'_x + i'_x + 2i'_x = 0$$

so that

$$i'_x = 2 \text{ A}$$

Next, short-circuit the 10 V source (Fig. 5.6c) and write the single-node equation

$$\frac{v''}{2} + \frac{v'' - 2i''_x}{1} = 3$$

and relate the dependent-source-controlling quantity to  $v''$ :

$$v'' = 2(-i''_x)$$

Solving, we find

$$i''_x = -0.6 \text{ A}$$

and, thus,

$$i_x = i'_x + i''_x = 2 + (-0.6) = 1.4 \text{ A}$$

Note that in redrawing each subcircuit, we are always careful to use some type of notation to indicate that we are not working with the original variables. This prevents the possibility of rather disastrous errors when we add the individual results.

## PRACTICE

5.2 For the circuit of Fig. 5.7, use superposition to obtain the voltage across each current source.

Ans:  $v_{1|2A} = 9.180 \text{ V}$ ,  $v_{2|2A} = -1.148 \text{ V}$ ,  $v_{1|3V} = 1.967 \text{ V}$ ,  $v_{2|3V} = -0.246 \text{ V}$ ;  $v_1 = 11.147 \text{ V}$ ,  $v_2 = -1.394 \text{ V}$ .

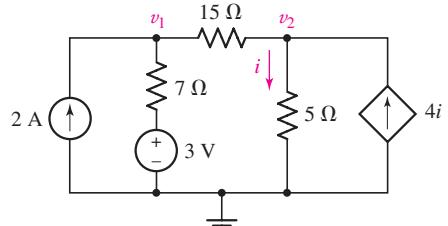


FIGURE 5.7

### Summary of Basic Superposition Procedure

1. **Select one of the independent sources. Set all other independent sources to zero.** This means voltage sources are replaced with short circuits and current sources are replaced with open circuits. Leave dependent sources in the circuit.
2. **Relabel voltages and currents using suitable notation** (e.g.,  $v'$ ,  $i_2''$ ). Be sure to relabel controlling variables of dependent sources to avoid confusion.
3. **Analyze the simplified circuit to find the desired currents and/or voltages.**
4. **Repeat steps 1 through 3 until each independent source has been considered.**
5. **Add the partial currents and/or voltages obtained from the separate analyses.** Pay careful attention to voltage signs and current directions when summing.
6. **Do not add power quantities.** If power quantities are required, calculate only after partial voltages and/or currents have been summed.

Note that step 1 may be altered in several ways. First, independent sources can be considered in groups as opposed to individually if it simplifies the analysis, as long as no independent source is included in more than one subcircuit. Second, it is technically not necessary to set sources to zero, although this is almost always the best route. For example, a 3 V source may appear in two subcircuits as a 1.5 V source, since  $1.5 + 1.5 = 3$  V just as  $0 + 3 = 3$  V. Because it is unlikely to simplify our analysis, however, there is little point to such an exercise.

### COMPUTER-AIDED ANALYSIS

Although PSpice is extremely useful in verifying that we have analyzed a complete circuit correctly, it can also assist us in determining the contribution of each source to a particular response. To do this, we employ what is known as a *dc parameter sweep*.

Consider the circuit presented in Example 5.2, when we were asked to determine the maximum positive current that could be obtained from the current source without exceeding the power rating of either resistor in the circuit. The circuit is shown redrawn using the Orcad Capture CIS schematic tool in Fig. 5.8. Note that no value has been assigned to the current source.

After the schematic has been entered and saved, the next step is to specify the dc sweep parameters. This option allows us to specify a range of values for a voltage or current source (in the present case, the current source  $I_x$ ), rather than a specific value. Selecting **New Simulation Profile** under **PSpice**, we provide a name for our profile and are then provided with the dialog box shown in Fig. 5.9.

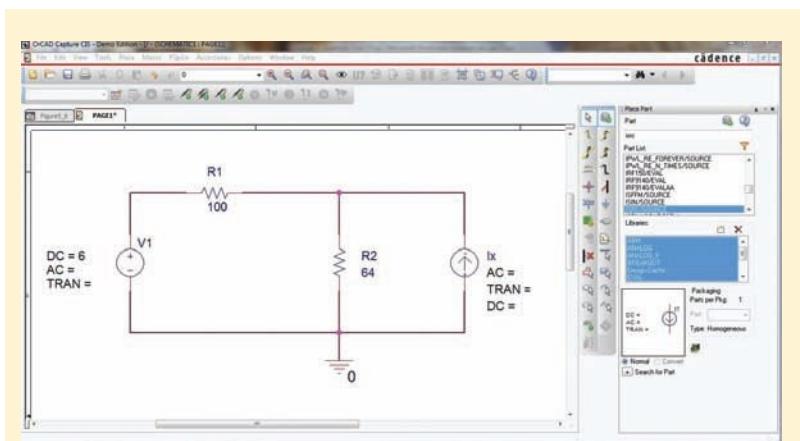
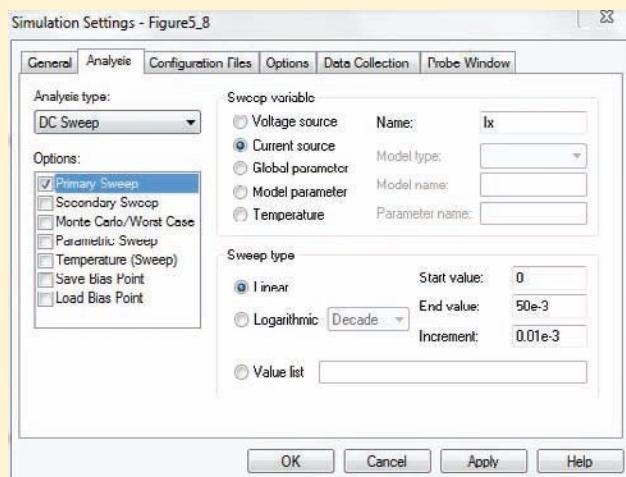


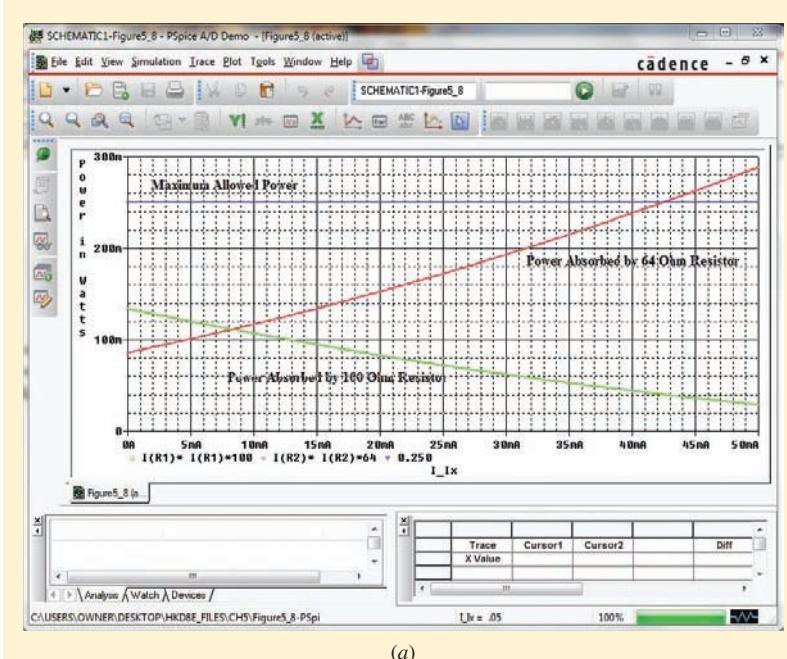
FIGURE 5.8 The circuit from Example 5.2.

FIGURE 5.9 DC Sweep dialog box shown with  $I_x$  selected as the sweep variable.

Under **Analysis Type**, we pull down the **DC Sweep** option, specify the “sweep variable” as **Current Source**, and then type in  $I_x$  in the **Name** box. There are several options under **Sweep Type**: **Linear**, **Logarithmic**, and **Value List**. The last option allows us to specify each value to assign to  $I_x$ . In order to generate a smooth plot, however, we choose to perform a **Linear** sweep, with a **Start Value** of 0 mA, an **End Value** of 50 mA, and a value of 0.01 mA for the **Increment**.

After we perform the simulation, the graphical output package Probe is automatically launched. When the window appears, the horizontal axis (corresponding to our variable,  $I_x$ ) is displayed, but the vertical axis variable must be chosen. Selecting **Add Trace** from the **Trace** menu, we click on **I(R1)**, then type an asterisk in the **Trace Expression** box, click on **I(R1)** once again, insert yet another asterisk, and finally type in 100. This asks Probe to plot the power absorbed by the 100  $\Omega$  resistor. In a similar fashion, we repeat the process to add the power

(Continued on next page)



(a)

Trace	Cursor1	Cursor2	Diff	Max	Min
X Value	42.520m	0.000	42.520m	42.520m	0.000
$I(R1)*I(R1)*100$	39.988m	133.820m	93.851m	133.820m	39.988m
$I(R2)*I(R2)*64$	250.146m	95.692m	164.454m	250.146m	95.692m
0.250	250.000m	250.000m	0.000	250.000m	250.000m

(b)

FIGURE 5.10 (a) Probe output with text labels identifying the power absorbed by the two resistors individually. A horizontal line indicating 250 mW has also been included, as well as text labels to improve clarity. (b) Cursor dialog box.

absorbed by the  $64 \Omega$  resistor, resulting in a plot similar to that shown in Fig. 5.10a. A horizontal reference line at 250 mW was also added to the plot by typing 0.250 in the **Trace Expression** box after selecting **Add Trace** from the **Trace** menu a third time.

We see from the plot that the  $64 \Omega$  resistor *does* exceed its 250 mW power rating in the vicinity of  $I_x = 43$  mA. In contrast, however, we also see that regardless of the value of the current source  $I_x$  (provided that it is between 0 and 50 mA), the  $100 \Omega$  resistor will never dissipate 250 mW; in fact, the absorbed power *decreases* with increasing current from the current source. If we desire a more precise answer, we can make use of the cursor tool, which is invoked by selecting **Trace**, **Cursor**, **Display** from the menu bar. Figure 5.10b shows the result of dragging cursor 1 to 42.52 A, where the  $64 \Omega$  resistor is dissipating just over its maximum rated power of 250 mW. Increased precision can be obtained by decreasing the increment value used in the dc sweep.

This technique is very useful in analyzing electronic circuits, where we might need, for example, to determine what input voltage is required

to a complicated amplifier circuit in order to obtain a zero output voltage. We also notice that there are several other types of parameter sweeps that we can perform, including a dc voltage sweep. The ability to vary temperature is useful only when dealing with component models that have a temperature parameter built in, such as diodes and transistors.

Unfortunately, it usually turns out that little if any time is saved in analyzing a circuit containing one or more dependent sources by use of the superposition principle, for there must always be at least two sources in operation: one independent source and all the dependent sources.

*We must constantly be aware of the limitations of superposition.* It is applicable only to linear responses, and thus the most common nonlinear response—power—is not subject to superposition. For example, consider two 1 V batteries in series with a 1  $\Omega$  resistor. The power delivered to the resistor is 4 W, but if we mistakenly try to apply superposition, we might say that each battery alone furnished 1 W and thus the calculated power is only 2 W. This is incorrect, but a surprisingly easy mistake to make.



## 5.2 SOURCE TRANSFORMATIONS

### Practical Voltage Sources

So far, we've only worked with *ideal* sources—elements whose terminal voltage is independent of the current flowing through them. To see the relevance of this fact, consider a simple independent (“ideal”) 9 V source connected to a 1  $\Omega$  resistor. The 9 volt source will force a current of 9 amperes through the 1  $\Omega$  resistor (perhaps this seems reasonable enough), but the same source would apparently force 9,000,000 amperes through a 1 m $\Omega$  resistor (which hopefully does not seem reasonable). On paper, there's nothing to stop us from reducing the resistor value all the way to 0  $\Omega$  ... but that would lead to a contradiction, as the source would be “trying” to maintain 9 V across a dead short, which Ohm's law tells us can't happen ( $V = 9 = RI = 0?$ ).

**What happens in the real world when we do this type of experiment?** For example, if we try to start a car with the headlights already on, we most likely notice the headlights dim as the battery is asked to supply a large ( $\sim 100$  A or more) starter current in parallel with the current running to the headlights. If we model the 12 V battery with an ideal 12 V source as in Fig. 5.11a, our observation cannot be explained. Another way of saying this is that our model breaks down when the load draws a large current from the source.

To better approximate the behavior of a real device, the ideal voltage source must be modified to account for the lowering of its terminal voltage when large currents are drawn from it. Let us suppose that we observe experimentally that our car battery has a terminal voltage of 12 V when no current is flowing through it, and a reduced voltage of 11 V when 100 A is flowing. How could we model this behavior? Well, a more accurate model might be an ideal voltage source of 12 V in series with a resistor across which 1 V appears when 100 A flows through it. A quick calculation shows that the resistor must be  $1 \text{ V}/100 \text{ A} = 0.01 \Omega$ , and the ideal voltage source and this series resistor constitute a *practical voltage source* (Fig. 5.11b).

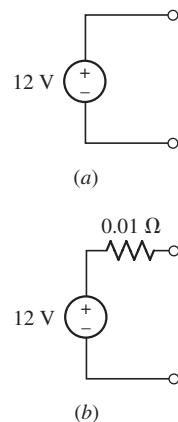


FIGURE 5.11 (a) An ideal 12 V dc voltage source used to model a car battery. (b) A more accurate model that accounts for the observed reduction in terminal voltage at large currents.

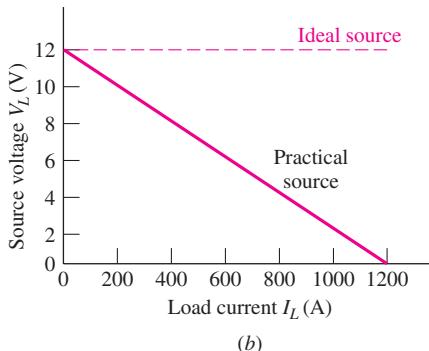
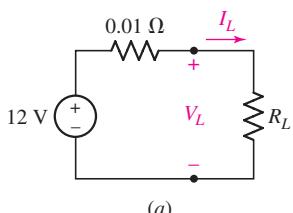


FIGURE 5.12 (a) A practical source, which approximates the behavior of a certain 12 V automobile battery, is shown connected to a load resistor  $R_L$ . (b) The relationship between  $I_L$  and  $V_L$  is linear.

Thus, we are using the series combination of two ideal circuit elements, an independent voltage source and a resistor, to model a real device.

We do not expect to find such an arrangement of ideal elements inside our car battery, of course. Any real device is characterized by a certain current-voltage relationship at its terminals, and our problem is to develop some combination of ideal elements that can furnish a similar current-voltage characteristic, at least over some useful range of current, voltage, or power.

In Fig. 5.12a, we show our two-piece practical model of the car battery now connected to some load resistor  $R_L$ . The terminal voltage of the practical source is the same as the voltage across  $R_L$  and is marked<sup>2</sup>  $V_L$ . Figure 5.12b shows a plot of load voltage  $V_L$  as a function of the load current  $I_L$  for this practical source. The KVL equation for the circuit of Fig. 5.12a may be written in terms of  $I_L$  and  $V_L$ :

$$12 = 0.01I_L + V_L$$

and thus

$$V_L = -0.01I_L + 12$$

This is a linear equation in  $I_L$  and  $V_L$ , and the plot in Fig. 5.12b is a straight line. Each point on the line corresponds to a different value of  $R_L$ . For example, the midpoint of the straight line is obtained when the load resistance is equal to the internal resistance of the practical source, or  $R_L = 0.01 \Omega$ . Here, the load voltage is exactly one-half the ideal source voltage.

When  $R_L = \infty$  and no current whatsoever is being drawn by the load, the practical source is open-circuited and the terminal voltage, or open-circuit voltage, is  $V_{Loc} = 12$  V. If, on the other hand,  $R_L = 0$ , thereby short-circuiting the load terminals, then a load current or short-circuit current,  $I_{Lsc} = 1200$  A, would flow. (*In practice, such an experiment would probably result in the destruction of the short circuit, the battery, and any measuring instruments incorporated in the circuit!*)

Since the plot of  $V_L$  versus  $I_L$  is a straight line for this practical voltage source, we should note that the values of  $V_{Loc}$  and  $I_{Lsc}$  uniquely determine the entire  $V_L$ - $I_L$  curve.

The horizontal broken line of Fig. 5.12b represents the  $V_L$ - $I_L$  plot for an *ideal* voltage source; the terminal voltage remains constant for any value of load current. For the practical voltage source, the terminal voltage has a value near that of the ideal source only when the load current is relatively small.

Let us now consider a *general* practical voltage source, as shown in Fig. 5.13a. The voltage of the ideal source is  $v_s$ , and a resistance  $R_s$ , called an *internal resistance* or *output resistance*, is placed in series with it. Again, we must note that the resistor is not really present as a separate component but merely serves to account for a terminal voltage that decreases as the load current increases. Its presence enables us to model the behavior of a physical voltage source more closely.

The linear relationship between  $v_L$  and  $i_L$  is

$$v_L = v_s - R_s i_L \quad [9]$$

(2) From this point on we will endeavor to adhere to the standard convention of referring to strictly dc quantities using capital letters, whereas lowercase letters denote a quantity that we know to possess some time-varying component. However, in describing general theorems which apply to either dc or ac, we will continue to use lowercase to emphasize the general nature of the concept.

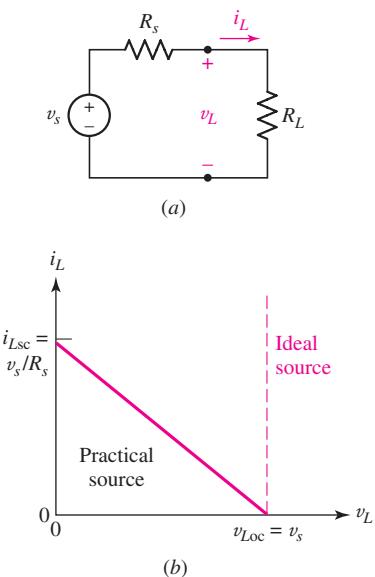


FIGURE 5.13 (a) A general practical voltage source connected to a load resistor  $R_L$ . (b) The terminal voltage of a practical voltage source decreases as  $i_L$  increases and  $R_L = v_L/i_L$  decreases. The terminal voltage of an ideal voltage source (also plotted) remains the same for any current delivered to a load.

and this is plotted in Fig. 5.13b. The open-circuit voltage ( $R_L = \infty$ , so  $i_L = 0$ ) is

$$v_{Loc} = v_s \quad [10]$$

and the short-circuit current ( $R_L = 0$ , so  $v_L = 0$ ) is

$$i_{Lsc} = \frac{v_s}{R_s} \quad [11]$$

Once again, these values are the intercepts for the straight line in Fig. 5.13b, and they serve to define it completely.

## Practical Current Sources

An ideal current source is also nonexistent in the real world; there is no physical device that will deliver a constant current regardless of the load resistance to which it is connected or the voltage across its terminals. Certain transistor circuits will deliver a constant current to a wide range of load resistances, but the load resistance can always be made sufficiently large that the current through it becomes very small. Infinite power is simply never available (unfortunately).

A practical current source is defined as an ideal current source in parallel with an internal resistance  $R_p$ . Such a source is shown in Fig. 5.14a, and the current  $i_L$  and voltage  $v_L$  associated with a load resistance  $R_L$  are indicated. Application of KCL yields

$$i_L = i_s - \frac{v_L}{R_p} \quad [12]$$

which is again a linear relationship. The open-circuit voltage and the short-circuit current are

$$v_{Loc} = R_p i_s \quad [13]$$

and

$$i_{Lsc} = i_s \quad [14]$$

The variation of load current with changing load voltage may be investigated by changing the value of  $R_L$  as shown in Fig. 5.14b. The straight line is traversed from the short-circuit, or “northwest,” end to the open-circuit termination at the “southeast” end by increasing  $R_L$  from zero to infinite ohms. The midpoint occurs for  $R_L = R_p$ . The load current  $i_L$  and the ideal source current are approximately equal only for small values of load voltage, which are obtained with values of  $R_L$  that are small compared to  $R_p$ .

## Equivalent Practical Sources

It may be no surprise that we can improve upon models to increase their accuracy; at this point we now have a practical voltage source model and also a practical current source model. Before we proceed, however, let’s take a moment to compare Fig. 5.13b and Fig. 5.14b. One is for a circuit with a voltage source and the other, with a current source, *but the graphs are indistinguishable!*

It turns out that this is no coincidence. In fact, we are about to show that a practical voltage source *can be* electrically equivalent to a practical current source—meaning that a load resistor  $R_L$  connected to either will have

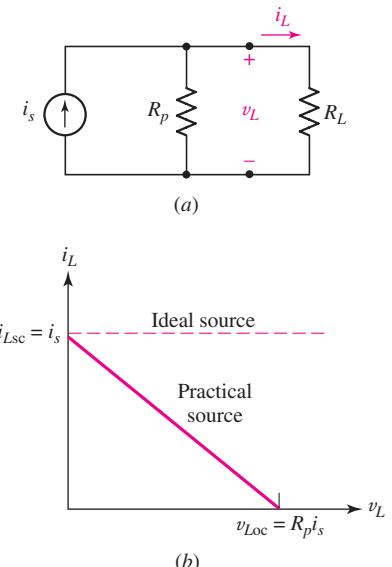
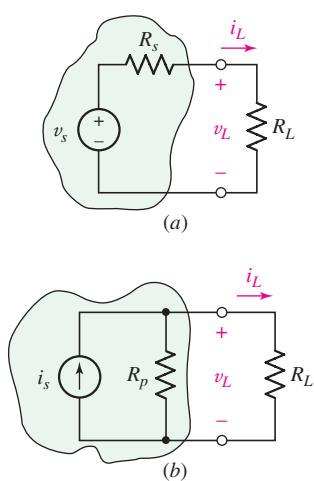
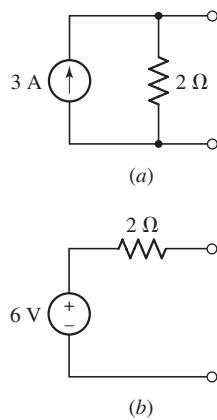


FIGURE 5.14 (a) A general practical current source connected to a load resistor  $R_L$ . (b) The load current provided by the practical current source is shown as a function of the load voltage.



**FIGURE 5.15** (a) A given practical voltage source connected to a load  $R_L$ . (b) The equivalent practical current source connected to the same load.



**FIGURE 5.16** (a) A given practical current source. (b) The equivalent practical voltage source.

### EXAMPLE 5.4

the same  $v_L$  and  $i_L$ . This means we can replace one practical source with the other and the rest of the circuit will not know the difference.

Consider the practical voltage source and resistor  $R_L$  shown in Fig. 5.15a, and the circuit composed of a practical current source and resistor  $R_L$  shown in Fig. 5.15b. A simple calculation shows that the voltage across the load  $R_L$  of Fig. 5.15a is

$$v_L = v_s \frac{R_L}{R_s + R_L} \quad [15]$$

A similar calculation shows that the voltage across the load  $R_L$  in Fig. 5.15b is

$$v_L = \left( i_s \frac{R_p}{R_p + R_L} \right) \cdot R_L$$

The two practical sources are electrically equivalent, then, if

$$R_s = R_p \quad [16]$$

and

$$v_s = R_p i_s = R_s i_s \quad [17]$$

where we now let  $R_s$  represent the internal resistance of either practical source, which is the conventional notation.

Let's try this with the practical current source shown in Fig. 5.16a. Since its internal resistance is 2 Ω, the internal resistance of the equivalent practical voltage source is also 2 Ω; the voltage of the ideal voltage source contained within the practical voltage source is  $(2)(3) = 6$  V. The equivalent practical voltage source is shown in Fig. 5.16b.

To check the equivalence, let us visualize a 4 Ω resistor connected to each source. In both cases a current of 1 A, a voltage of 4 V, and a power of 4 W are associated with the 4 Ω load. However, we should note very carefully that the ideal current source is delivering a total power of 12 W, while the ideal voltage source is delivering only 6 W. Furthermore, the internal resistance of the practical current source is absorbing 8 W, whereas the internal resistance of the practical voltage source is absorbing only 2 W. Thus we see that the two practical sources are equivalent only with respect to what transpires at the load terminals; they are *not* equivalent internally!



**Compute the current through the 4.7 kΩ resistor in Fig. 5.17a after transforming the 9 mA source into an equivalent voltage source.**

It's not just the 9 mA source at issue, but also the resistance in parallel with it (5 kΩ). We remove these components, leaving two terminals "dangling." We then replace them with a voltage source in series with a 5 kΩ resistor. The value of the voltage source must be  $(0.009)(5000) = 45$  V.

Redrawing the circuit as in Fig. 5.17b, we can write a simple KVL equation

$$-45 + 5000I + 4700I + 3000I + 3 = 0$$

which is easily solved to yield  $I = 3.307$  mA.

We can check our answer of course by analyzing the circuit of Fig. 5.17a using either nodal or mesh techniques.

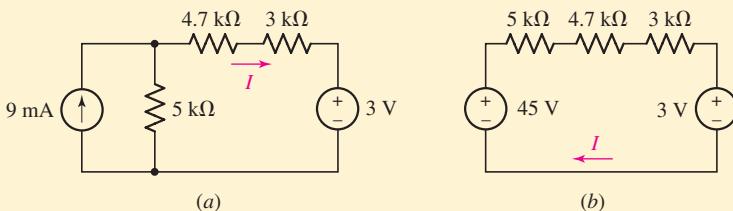


FIGURE 5.17 (a) A circuit with both a voltage source and a current source. (b) The circuit after the 9 mA source is transformed into an equivalent voltage source.

### PRACTICE

5.3 For the circuit of Fig. 5.18, compute the current  $I_X$  through the  $47 \text{ k}\Omega$  resistor after performing a source transformation on the voltage source.

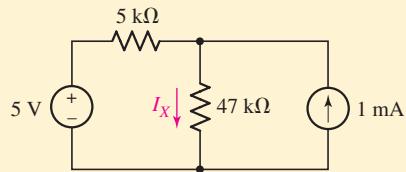


FIGURE 5.18

Ans:  $192 \mu\text{A}$ .

### EXAMPLE 5.5

Calculate the current through the  $2 \Omega$  resistor in Fig. 5.19a by making use of source transformations to first simplify the circuit.

We begin by transforming each current source into a voltage source (Fig. 5.19b), the strategy being to convert the circuit into a simple loop.

We must be careful to retain the  $2 \Omega$  resistor for two reasons: first, the dependent source controlling variable appears across it, and second, we desire the current flowing through it. However, we can combine the  $17 \Omega$  and  $9 \Omega$  resistors, since they appear in series. We also see that the  $3 \Omega$  and  $4 \Omega$  resistors may be combined into a single  $7 \Omega$  resistor, which can then be used to transform the  $15 \text{ V}$  source into a  $15/7 \text{ A}$  source as in Fig. 5.19c.

Finally, we note that the two  $7 \Omega$  resistors can be combined into a single  $3.5 \Omega$  resistor, which may be used to transform the  $15/7 \text{ A}$  current source into a  $7.5 \text{ V}$  voltage source. The result is a simple loop circuit, shown in Fig. 5.19d.

The current  $I$  can now be found using KVL:

$$-7.5 + 3.5I - 51V_x + 28I + 9 = 0$$

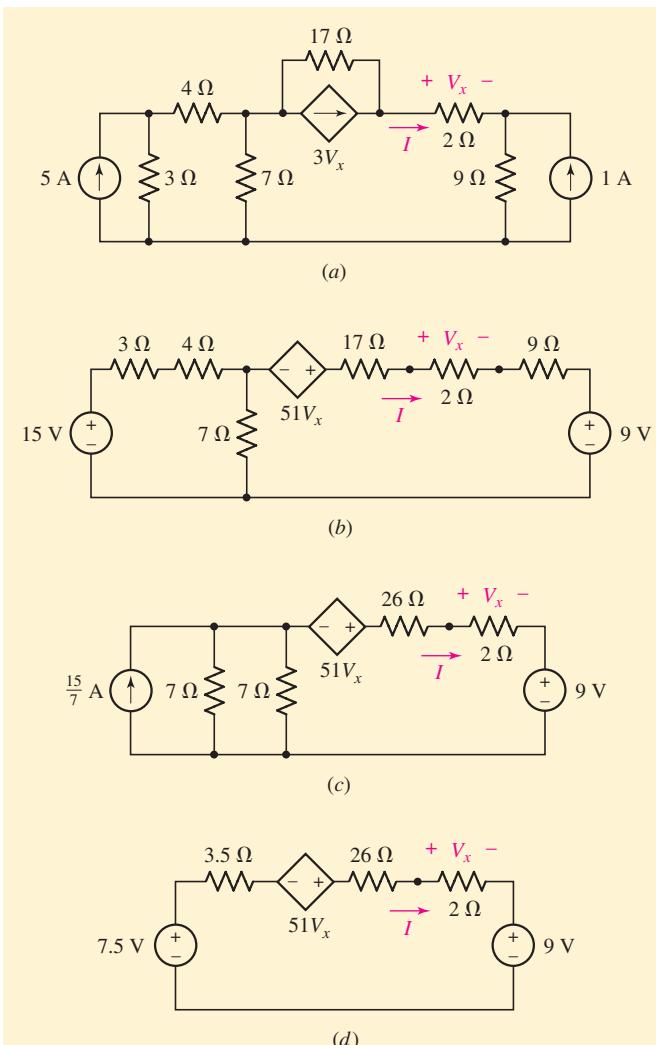
where

$$V_x = 2I$$

Thus,

$$I = 21.28 \text{ mA}$$

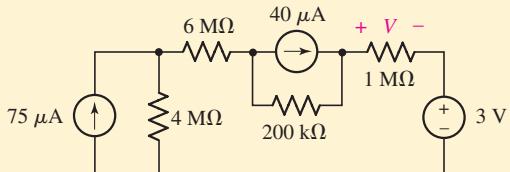
(Continued on next page)



■ FIGURE 5.19 (a) A circuit with two independent current sources and one dependent source. (b) The circuit after each source is transformed into a voltage source. (c) The circuit after further combinations. (d) The final circuit.

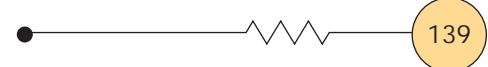
### PRACTICE

5.4 For the circuit of Fig. 5.20, compute the voltage  $V$  across the  $1 \text{ M}\Omega$  resistor using repeated source transformations.



■ FIGURE 5.20

Ans: 27.2 V.



## Several Key Points

We conclude our discussion of practical sources and source transformations with a few observations. First, when we transform a voltage source, we must be sure that the source is in fact *in series* with the resistor under consideration. For example, in the circuit of Fig. 5.21, it is perfectly valid to perform a source transformation on the voltage source using the  $10\ \Omega$  resistor, as they are in series. However, it would be incorrect to attempt a source transformation using the  $60\text{ V}$  source and the  $30\ \Omega$  resistor—a very common type of error.

In a similar fashion, when we transform a current source and resistor combination, we must be sure that they are in fact *in parallel*. Consider the current source shown in Fig. 5.22a. We may perform a source transformation including the  $3\ \Omega$  resistor, as they are in parallel, but after the transformation there may be some ambiguity as to where to place the resistor. In such circumstances, it is helpful to first redraw the components to be transformed as in Fig. 5.22b. Then the transformation to a voltage source in series with a resistor may be drawn correctly as shown in Fig. 5.22c; the resistor may in fact be drawn above or below the voltage source.

It is also worthwhile to consider the unusual case of a current source in series with a resistor, and its dual, the case of a voltage source in parallel

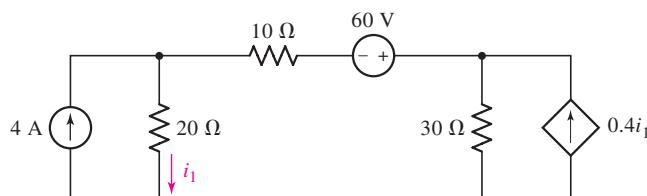


FIGURE 5.21 An example circuit to illustrate how to determine if a source transformation can be performed.

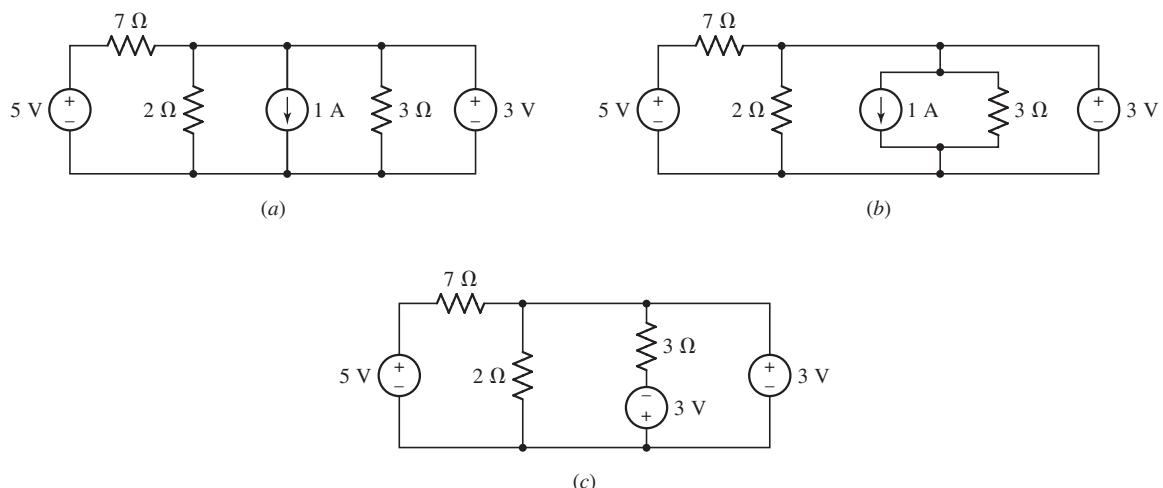
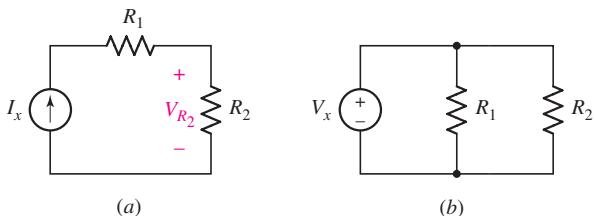


FIGURE 5.22 (a) A circuit with a current source to be transformed to a voltage source. (b) Circuit redrawn so as to avoid errors. (c) Transformed source/resistor combination.



with a resistor. Let's start with the simple circuit of Fig. 5.23a, where we are interested only in the voltage across the resistor marked  $R_2$ . We note that regardless of the value of resistor  $R_1$ ,  $V_{R_2} = I_x R_2$ . Although we might be tempted to perform an inappropriate source transformation on such a circuit, in fact *we may simply omit resistor  $R_1$*  (provided that it is of no interest to us itself). A similar situation arises with a voltage source in parallel with a resistor, as depicted in Fig. 5.23b. Again, if we are only interested in some quantity regarding resistor  $R_2$ , we may find ourselves tempted to perform some strange (and incorrect) source transformation on the voltage source and resistor  $R_1$ . In reality, we may omit resistor  $R_1$  from our circuit as far as resistor  $R_2$  is concerned—its presence does not alter the voltage across, the current through, or the power dissipated by resistor  $R_2$ .



■ FIGURE 5.23 (a) Circuit with a resistor  $R_1$  in series with a current source. (b) A voltage source in parallel with two resistors.

### Summary of Source Transformation

1. **A common goal in source transformation is to end up with either all current sources or all voltage sources in the circuit.** This is especially true if it makes nodal or mesh analysis easier.
2. **Repeated source transformations can be used to simplify a circuit by allowing resistors and sources to eventually be combined.**
3. **The resistor value does not change during a source transformation, but it is not the same resistor.** This means that currents or voltages associated with the original resistor are irretrievably lost when we perform a source transformation.
4. **If the voltage or current associated with a particular resistor is used as a controlling variable for a dependent source, it should not be included in any source transformation.** The original resistor must be retained in the final circuit, untouched.
5. **If the voltage or current associated with a particular element is of interest, that element should not be included in any source transformation.** The original element must be retained in the final circuit, untouched.
6. **In a source transformation, the head of the current source arrow corresponds to the “+” terminal of the voltage source.**
7. **A source transformation on a current source and resistor requires that the two elements be in parallel.**
8. **A source transformation on a voltage source and resistor requires that the two elements be in series.**

## 5.3 THÉVENIN AND NORTON EQUIVALENT CIRCUITS

Now that we have been introduced to source transformations and the superposition principle, it is possible to develop two more techniques that will greatly simplify the analysis of many linear circuits. The first of these theorems is named after L. C. Thévenin, a French engineer working in telegraphy who published the theorem in 1883; the second may be considered a corollary of the first and is credited to E. L. Norton, a scientist with the Bell Telephone Laboratories.

Let us suppose that we need to make only a partial analysis of a circuit. For example, perhaps we need to determine the current, voltage, and power delivered to a single “load” resistor by the remainder of the circuit, which may consist of a sizable number of sources and resistors (Fig. 5.24a). Or, perhaps we wish to find the response for different values of the load resistance. Thévenin’s theorem tells us that it is possible to replace everything except the load resistor with an independent voltage source in series with a resistor (Fig. 5.24b); the response measured *at the load resistor* will be unchanged. Using Norton’s theorem, we obtain an equivalent composed of an independent current source in parallel with a resistor (Fig. 5.24c).

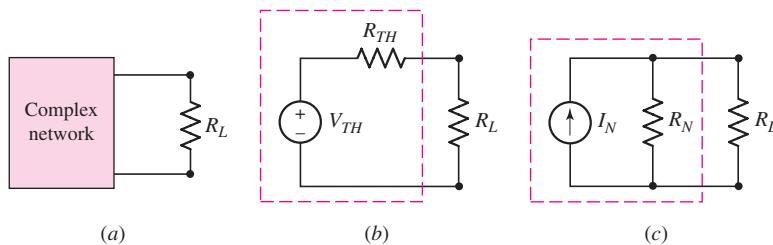


FIGURE 5.24 (a) A complex network including a load resistor  $R_L$ . (b) A Thévenin equivalent network connected to the load resistor  $R_L$ . (c) A Norton equivalent network connected to the load resistor  $R_L$ .

Thus, one of the main uses of Thévenin’s and Norton’s theorems is the replacement of a large part of a circuit, often a complicated and uninteresting part, with a very simple equivalent. The new, simpler circuit enables us to make rapid calculations of the voltage, current, and power which the original circuit is able to deliver to a load. It also helps us to choose the best value of this load resistance. In a transistor power amplifier, for example, the Thévenin or Norton equivalent enables us to determine the maximum power that can be taken from the amplifier and delivered to the speakers.

### EXAMPLE 5.6

Consider the circuit shown in Fig. 5.25a. Determine the Thévenin equivalent of network A, and compute the power delivered to the load resistor  $R_L$ .

The dashed regions separate the circuit into networks A and B; our main interest is in network B, which consists only of the load resistor  $R_L$ . Network A may be simplified by making repeated source transformations.

(Continued on next page)

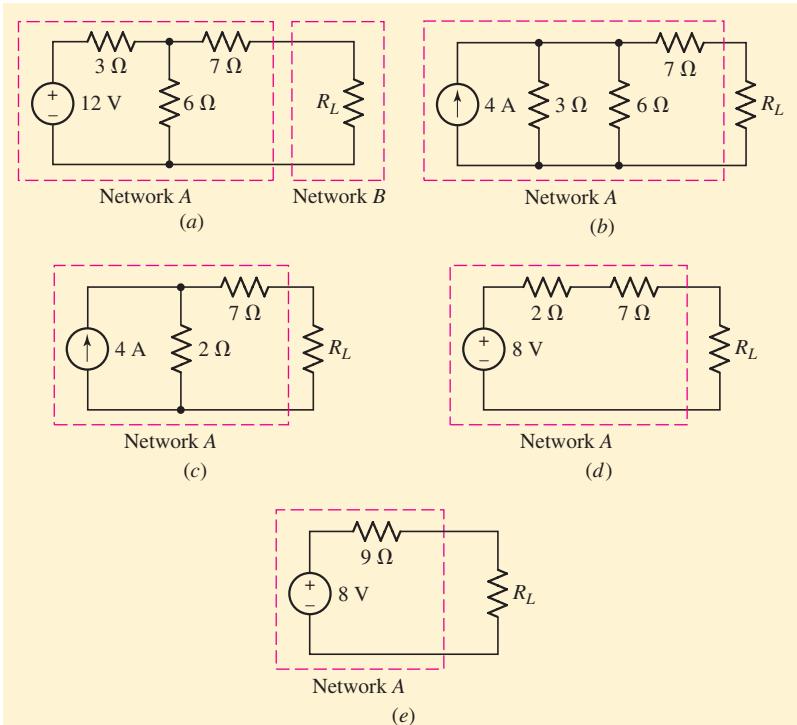


FIGURE 5.25 (a) A circuit separated into two networks. (b)–(d) Intermediate steps to simplifying network A. (e) The Thévenin equivalent circuit.

We first treat the 12 V source and the  $3 \Omega$  resistor as a practical voltage source and replace it with a practical current source consisting of a 4 A source in parallel with  $3 \Omega$  (Fig. 5.25b). The parallel resistances are then combined into  $2 \Omega$  (Fig. 5.25c), and the practical current source that results is transformed back into a practical voltage source (Fig. 5.25d). The final result is shown in Fig. 5.25e.

*From the viewpoint of the load resistor  $R_L$ , this network A (the Thévenin equivalent) is equivalent to the original network A; from our viewpoint, the circuit is much simpler, and we can now easily compute the power delivered to the load:*

$$P_L = \left( \frac{8}{9 + R_L} \right)^2 R_L$$

Furthermore, we can see from the equivalent circuit that the maximum voltage that can be obtained across  $R_L$  is 8 V and corresponds to  $R_L = \infty$ . A quick transformation of network A to a practical current source (the Norton equivalent) indicates that the maximum current that may be delivered to the load is  $8/9$  A, which occurs when  $R_L = 0$ . Neither of these facts is readily apparent from the original circuit.

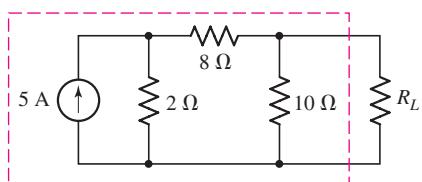
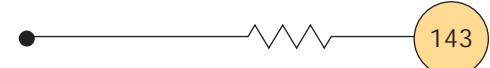


FIGURE 5.26

### PRACTICE

5.5 Using repeated source transformations, determine the Norton equivalent of the highlighted network in the circuit of Fig. 5.26.

Ans: 1 A, 5 Ω.



## Thévenin's Theorem

Using the technique of source transformation to find a Thévenin or Norton equivalent network worked well enough in Example 5.6, but it can rapidly become impractical in situations where dependent sources are present or the circuit is composed of a large number of elements. An alternative is to employ Thévenin's theorem (or Norton's theorem) instead. We will state the theorem<sup>3</sup> as a somewhat formal procedure and then consider various ways to make the approach more practical depending on the situation we face.

### A Statement of Thévenin's Theorem

1. **Given any linear circuit, rearrange it in the form of two networks, A and B, connected by two wires.** Network A is the network to be simplified; B will be left untouched.
2. **Disconnect network B.** Define a voltage  $v_{oc}$  as the voltage now appearing across the terminals of network A.
3. **Turn off or “zero out” every independent source in network A to form an inactive network.** Leave dependent sources unchanged.
4. **Connect an independent voltage source with value  $v_{oc}$  in series with the inactive network.** Do not complete the circuit; leave the two terminals disconnected.
5. **Connect network B to the terminals of the new network A.** All currents and voltages in B will remain unchanged.

Note that if either network contains a dependent source, *its control variable must be in the same network*.

Let us see if we can apply Thévenin's theorem successfully to the circuit we considered in Fig. 5.25. We have already found the Thévenin equivalent of the circuit to the left of  $R_L$  in Example 5.6, but we want to see if there is an easier way to obtain the same result.



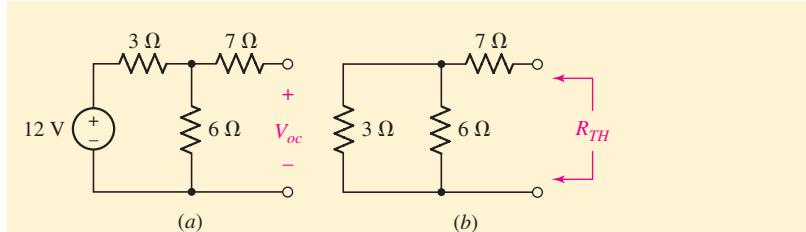
### EXAMPLE 5.7

Use Thévenin's theorem to determine the Thévenin equivalent for that part of the circuit in Fig. 5.25a to the left of  $R_L$ .

We begin by disconnecting  $R_L$ , and note that no current flows through the  $7\ \Omega$  resistor in the resulting partial circuit shown in Fig. 5.27a. Thus,  $V_{oc}$  appears across the  $6\ \Omega$  resistor (with no current through the  $7\ \Omega$  resistor there is no voltage drop across it), and voltage division enables us to determine that

$$V_{oc} = 12 \left( \frac{6}{3+6} \right) = 8\text{ V}$$

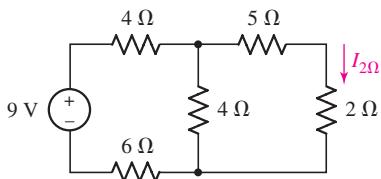
(3) A proof of Thévenin's theorem in the form in which we have stated it is rather lengthy, and therefore it has been placed in Appendix 3, where the curious may peruse it.



■ FIGURE 5.27 (a) The circuit of Fig. 5.25a with network  $B$  (the resistor  $R_L$ ) disconnected and the voltage across the connecting terminals labeled as  $V_{oc}$ . (b) The independent source in Fig. 5.25a has been killed, and we look into the terminals where network  $B$  was connected to determine the effective resistance of network  $A$ .

Turning off network  $A$  (i.e., replacing the 12 V source with a short circuit) and looking back into the dead network, we see a 7  $\Omega$  resistor connected in series with the parallel combination of 6  $\Omega$  and 3  $\Omega$  (Fig. 5.27b).

Thus, the inactive network can be represented here by a 9  $\Omega$  resistor, referred to as the **Thévenin equivalent resistance** of network  $A$ . The Thévenin equivalent then is  $V_{oc}$  in series with a 9  $\Omega$  resistor, which agrees with our previous result.



■ FIGURE 5.28

### PRACTICE

5.6 Use Thévenin's theorem to find the current through the 2  $\Omega$  resistor in the circuit of Fig. 5.28. (Hint: Designate the 2  $\Omega$  resistor as network  $B$ .)

Ans:  $V_{TH} = 2.571$  V,  $R_{TH} = 7.857$   $\Omega$ ,  $I_{2\Omega} = 260.8$  mA.

## A Few Key Points

The equivalent circuit we have learned how to obtain is completely independent of network  $B$ : we have been instructed to first remove network  $B$  and then measure the open-circuit voltage produced by network  $A$ , an operation that certainly does not depend on network  $B$  in any way. The  $B$  network is mentioned only to indicate that an equivalent for  $A$  may be obtained no matter what arrangement of elements is connected to the  $A$  network; the  $B$  network represents this general network.

There are several points about the theorem which deserve emphasis.

- The only restriction that we must impose on  $A$  or  $B$  is that all *dependent* sources in  $A$  have their control variables in  $A$ , and similarly for  $B$ .
- No restrictions are imposed on the complexity of  $A$  or  $B$ ; either one may contain any combination of independent voltage or current sources, linear dependent voltage or current sources, resistors, or any other circuit elements which are linear.
- The dead network  $A$  can be represented by a single equivalent resistance  $R_{TH}$ , which we will call the Thévenin equivalent resistance.

This holds true whether or not dependent sources exist in the inactive *A* network, an idea we will explore shortly.

- A Thévenin equivalent consists of two components: a voltage source in series with a resistance. Either may be zero, although this is not usually the case.

## Norton's Theorem

Norton's theorem bears a close resemblance to Thévenin's theorem and may be stated as follows:

### A Statement of Norton's Theorem

1. **Given any linear circuit, rearrange it in the form of two networks, *A* and *B*, connected by two wires.** Network *A* is the network to be simplified; *B* will be left untouched. As before, if either network contains a dependent source, *its controlling variable must be in the same network*.
2. **Disconnect network *B*, and short the terminals of *A*.** Define a current  $i_{sc}$  as the current now flowing through the shorted terminals of network *A*.
3. **Turn off or “zero out” every independent source in network *A* to form an inactive network.** Leave dependent sources unchanged.
4. **Connect an independent current source with value  $i_{sc}$  in parallel with the inactive network.** Do not complete the circuit; leave the two terminals disconnected.
5. **Connect network *B* to the terminals of the new network *A*.** All currents and voltages in *B* will remain unchanged.

The Norton equivalent of a linear network is the Norton current source  $i_{sc}$  in parallel with the Thévenin resistance  $R_{TH}$ . Thus, we see that in fact it is possible to obtain the Norton equivalent of a network by performing a source transformation on the Thévenin equivalent. This results in a direct relationship between  $v_{oc}$ ,  $i_{sc}$ , and  $R_{TH}$ :

$$v_{oc} = R_{TH}i_{sc} \quad [18]$$

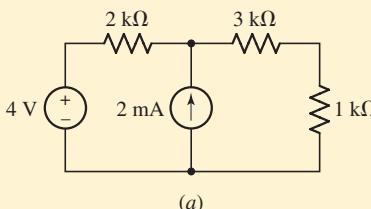


In circuits containing dependent sources, we will often find it more convenient to determine either the Thévenin or Norton equivalent by finding both the open-circuit voltage and the short-circuit current and then determining the value of  $R_{TH}$  as their quotient. It is therefore advisable to become adept at finding both open-circuit voltages and short-circuit currents, even in the simple problems that follow. If the Thévenin and Norton equivalents are determined independently, Eq. [18] can serve as a useful check.

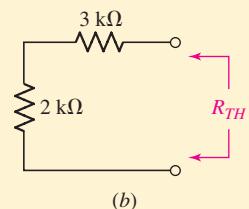
Let's consider three different examples of the determination of a Thévenin or Norton equivalent circuit.

## EXAMPLE 5.8

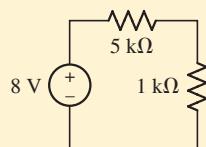
Find the Thévenin and Norton equivalent circuits for the network faced by the  $1\text{k}\Omega$  resistor in Fig. 5.29a.



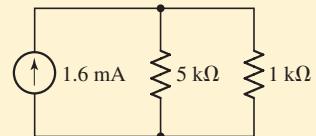
(a)



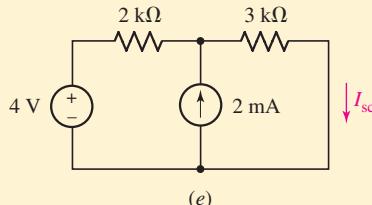
(b)



(c)



(d)



(e)

■ FIGURE 5.29 (a) A given circuit in which the  $1\text{k}\Omega$  resistor is identified as network *B*.  
 (b) Network *A* with all independent sources killed. (c) The Thévenin equivalent is shown for network *A*. (d) The Norton equivalent is shown for network *A*. (e) Circuit for determining  $I_{sc}$ .

From the wording of the problem statement, network *B* is the  $1\text{k}\Omega$  resistor, so network *A* is everything else.

Choosing to find the Thévenin equivalent of network *A* first, we apply superposition, noting that no current flows through the  $3\text{k}\Omega$  resistor once network *B* is disconnected. With the current source set to zero,  $V_{oc|_{4\text{V}}}=4\text{V}$ . With the voltage source set to zero,

$$V_{oc|_{2\text{mA}}} = (0.002)(2000) = 4\text{V}. \text{ Thus, } V_{oc} = 4 + 4 = 8\text{V}.$$

To find  $R_{TH}$ , set both sources to zero as in Fig. 5.29b. By inspection,  $R_{TH} = 2\text{k}\Omega + 3\text{k}\Omega = 5\text{k}\Omega$ . The complete Thévenin equivalent, with network *B* reconnected, is shown in Fig. 5.29c.

The Norton equivalent is found by a simple source transformation of the Thévenin equivalent, resulting in a current source of  $8/5000 = 1.6\text{mA}$  in parallel with a  $5\text{k}\Omega$  resistor (Fig. 5.29d).

**Check:** Find the Norton equivalent directly from Fig. 5.29a. Removing the  $1\text{k}\Omega$  resistor and shorting the terminals of network *A*, we

find  $I_{sc}$  as shown in Fig. 5.29e by superposition and current division:

$$\begin{aligned} I_{sc} &= I_{sc|4\text{V}} + I_{sc|2\text{mA}} = \frac{4}{2+3} + (2)\frac{2}{2+3} \\ &= 0.8 + 0.8 = 1.6 \text{ mA} \end{aligned}$$

which completes the check.

### PRACTICE

5.7 Determine the Thévenin and Norton equivalents of the circuit of Fig. 5.30.

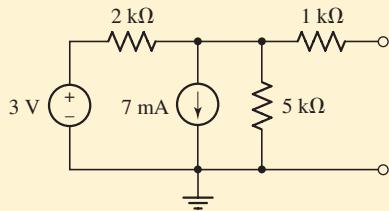


FIGURE 5.30

Ans:  $-7.857 \text{ V}$ ,  $-3.235 \text{ mA}$ ,  $2.429 \text{ k}\Omega$ .

## When Dependent Sources Are Present

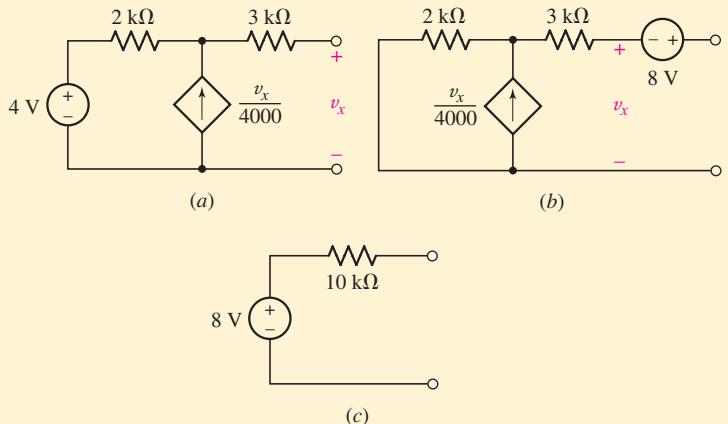
Technically speaking, there does not always have to be a “network  $B$ ” for us to invoke either Thévenin’s theorem or Norton’s theorem; we could instead be asked to find the equivalent of a network with two terminals not yet connected to another network. If there *is* a network  $B$  that we do not want to involve in the simplification procedure, however, we must use a little caution if it contains dependent sources. In such situations, the controlling variable and the associated element(s) must be included in network  $B$  and excluded from network  $A$ . Otherwise, there will be no way to analyze the final circuit because the controlling quantity will be lost.

If network  $A$  contains a dependent source, then again we must ensure that the controlling variable and its associated element(s) cannot be in network  $B$ . Up to now, we have only considered circuits with resistors and independent sources. Although technically speaking it is correct to leave a dependent source in the “inactive” network when creating a Thévenin or Norton equivalent, in practice this does not result in any kind of simplification. What we really want is an independent voltage source in series with a single resistor, or an independent current source in parallel with a single resistor—in other words, a two-component equivalent. In the following examples, we consider various means of reducing networks with dependent sources and resistors into a single resistance.



## EXAMPLE 5.9

Determine the Thévenin equivalent of the circuit in Fig. 5.31a.



■ FIGURE 5.31 (a) A given network whose Thévenin equivalent is desired. (b) A possible, but rather useless, form of the Thévenin equivalent. (c) The best form of the Thévenin equivalent for this linear resistive network.

To find  $V_{oc}$  we note that  $v_x = V_{oc}$  and that the dependent source current must pass through the  $2\text{k}\Omega$  resistor, since no current can flow through the  $3\text{k}\Omega$  resistor. Using KVL around the outer loop:

$$-4 + 2 \times 10^3 \left( -\frac{v_x}{4000} \right) + 3 \times 10^3(0) + v_x = 0$$

and

$$v_x = 8 \text{ V} = V_{oc}$$

By Thévenin's theorem, then, the equivalent circuit could be formed with the inactive  $A$  network in series with an 8 V source, as shown in Fig. 5.31b. This is correct, but not very simple and not very helpful; in the case of linear resistive networks, we really want a simpler equivalent for the inactive  $A$  network, namely,  $R_{TH}$ .

The dependent source prevents us from determining  $R_{TH}$  directly for the inactive network through resistance combination; we therefore seek  $I_{sc}$ . Upon short-circuiting the output terminals in Fig. 5.31a, it is apparent that  $V_x = 0$  and the dependent current source is not active. Hence,  $I_{sc} = 4/(5 \times 10^3) = 0.8 \text{ mA}$ . Thus,

$$R_{TH} = \frac{V_{oc}}{I_{sc}} = \frac{8}{0.8 \times 10^{-3}} = 10 \text{ k}\Omega$$

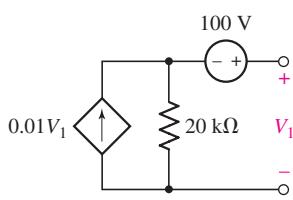
and the acceptable Thévenin equivalent of Fig. 5.31c is obtained.

### PRACTICE

5.8 Find the Thévenin equivalent for the network of Fig. 5.32. (Hint: a quick source transformation on the dependent source might help.)

Ans:  $-502.5 \text{ mV}$ ,  $-100.5 \Omega$ .

Note: a negative resistance might seem strange—and it is! Such a thing is physically possible only if, for example, we do a bit of clever electronic circuit design to create something that behaves like the dependent current source we represented in Fig. 5.32.



■ FIGURE 5.32

As another example, let us consider a network having a dependent source but no independent source.

## EXAMPLE 5.10

Find the Thévenin equivalent of the circuit shown in Fig. 5.33a.

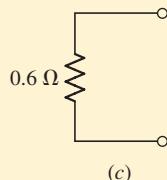
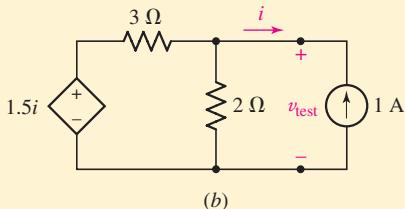
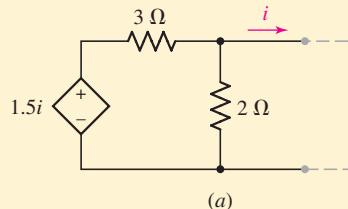


FIGURE 5.33 (a) A network with no independent sources. (b) A hypothetical measurement to obtain  $R_{TH}$ . (c) The Thévenin equivalent to the original circuit.

The rightmost terminals are already open-circuited, hence  $i = 0$ .

Consequently, the dependent source is inactive, so  $v_{oc} = 0$ .

We next seek the value of  $R_{TH}$  represented by this two-terminal network. However, we cannot find  $v_{oc}$  and  $i_{sc}$  and take their quotient, for there is no independent source in the network and both  $v_{oc}$  and  $i_{sc}$  are zero. Let us, therefore, be a little tricky.

We apply a 1 A source externally, measure the voltage  $v_{test}$  that results, and then set  $R_{TH} = v_{test}/1$ . Referring to Fig. 5.33b, we see that  $i = -1$  A. Applying nodal analysis,

$$\frac{v_{test} - 1.5(-1)}{3} + \frac{v_{test}}{2} = 1$$

so that

$$v_{test} = 0.6 \text{ V}$$

and thus

$$R_{TH} = 0.6 \Omega$$

The Thévenin equivalent is shown in Fig. 5.33c.

## A Quick Recap of Procedures

We have now looked at three examples in which we determined a Thévenin or Norton equivalent circuit. The first example (Fig. 5.29) contained only independent sources and resistors, and several different methods could have been applied to it. One would involve calculating  $R_{TH}$  for the inactive network and then  $V_{oc}$  for the live network. We could also have found  $R_{TH}$  and  $I_{sc}$ , or  $V_{oc}$  and  $I_{sc}$ .

## PRACTICAL APPLICATION

### The Digital Multimeter

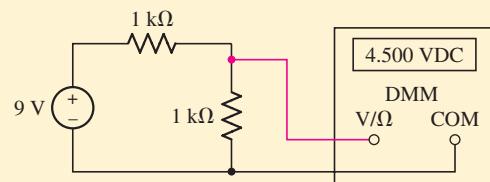
One of the most common pieces of electrical test equipment is the DMM, or digital multimeter (Fig. 5.34), which is designed to measure voltage, current, and resistance values.



■ FIGURE 5.34 A handheld digital multimeter.

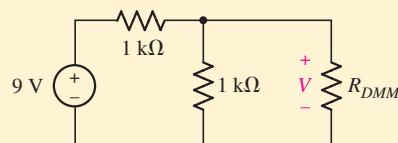
In a voltage measurement, two leads from the DMM are connected across the appropriate circuit element, as depicted in Fig. 5.35. The positive reference terminal of the meter is typically marked “V/Ω,” and the negative reference terminal—often referred to as the *common terminal*—is typically designated by “COM.” The convention is to use a red lead for the positive reference terminal and a black lead for the common terminal.

From our discussion of Thévenin and Norton equivalents, it may now be apparent that the DMM has its own



■ FIGURE 5.35 A DMM connected to measure voltage.

Thévenin equivalent resistance. This Thévenin equivalent resistance will appear in parallel with our circuit, and its value can affect the measurement (Fig. 5.36). The DMM does not supply power to the circuit to measure voltage, so its Thévenin equivalent consists of only a resistance, which we will name  $R_{DMM}$ .



■ FIGURE 5.36 DMM in Fig. 5.35 shown as its Thévenin equivalent resistance,  $R_{DMM}$ .

The input resistance of a good DMM is typically  $10\text{ M}\Omega$  or more. The measured voltage  $V$  thus appears across  $1\text{ k}\Omega \parallel 10\text{ M}\Omega = 999.9\ \Omega$ . Using voltage division, we find that  $V = 4.4998$  volts, slightly less than the expected value of 4.5 volts. Thus, the finite input resistance of the voltmeter introduces a small error in the measured value.

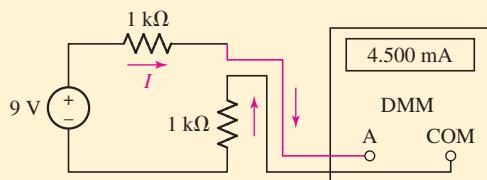
In the second example (Fig. 5.31), both independent and dependent sources were present, and the method we used required us to find  $V_{oc}$  and  $I_{sc}$ . We could not easily find  $R_{TH}$  for the inactive network because the dependent source could not be made inactive.

The last example did not contain any independent sources, and therefore the Thévenin and Norton equivalents do not contain an independent source. We found  $R_{TH}$  by applying 1 A and finding  $v_{test} = 1 \times R_{TH}$ . We could also apply 1 V and determine  $i = 1/R_{TH}$ . These two related techniques can be applied to any circuit with dependent sources, *as long as all independent sources are set to zero first*.

Two other methods have a certain appeal because they can be used for any of the three types of networks considered. In the first, simply replace network  $B$  with a voltage source  $v_s$ , define the current leaving its positive terminal as  $i$ , analyze network  $A$  to obtain  $i$ , and put the equation in the form  $v_s = ai + b$ . Then,  $a = R_{TH}$  and  $b = v_{oc}$ .



To measure current, the DMM must be placed in series with a circuit element, generally requiring that we cut a wire (Fig. 5.37). One DMM lead is connected to the common terminal of the meter, and the other lead is placed in a connector usually marked “A” to signify current measurement. Again, the DMM does not supply power to the circuit in this type of measurement.



■ FIGURE 5.37 A DMM connected to measure current.

We see that the Thévenin equivalent resistance ( $R_{DMM}$ ) of the DMM is in series with our circuit, so its value can affect the measurement. Writing a simple KVL equation around the loop,

$$-9 + 1000I + R_{DMM}I + 1000I = 0$$

Note that since we have reconfigured the meter to perform a current measurement, the Thévenin equivalent resistance is not the same as when the meter is configured to measure voltages. In fact, we would ideally like  $R_{DMM}$  to be  $0\ \Omega$  for current measurements, and  $\infty$  for voltage measurements. If  $R_{DMM}$  is now  $0.1\ \Omega$ , we see that the measured current  $I$  is  $4.4998\text{ mA}$ , which is only slightly different from the expected value of  $4.5\text{ mA}$ . Depending on the number of digits that can be displayed by

We could also apply a current source  $i_s$ , let its voltage be  $v$ , and then determine  $i_s = cv - d$ , where  $c = 1/R_{TH}$  and  $d = i_{sc}$  (the minus sign arises from assuming both current source arrows are directed into the same node). Both of these last two procedures are universally applicable, but some other method can usually be found that is easier and more rapid.

Although we are devoting our attention almost entirely to the analysis of linear circuits, it is good to know that Thévenin’s and Norton’s theorems are both valid if network  $B$  is nonlinear; only network  $A$  must be linear.

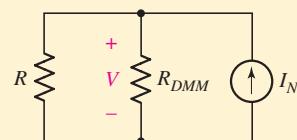
### PRACTICE

5.9 Find the Thévenin equivalent for the network of Fig. 5.39. (Hint: Try a  $1\text{ V}$  test source.)

Ans:  $I_{test} = 50\text{ mA}$  so  $R_{TH} = 20\ \Omega$ .

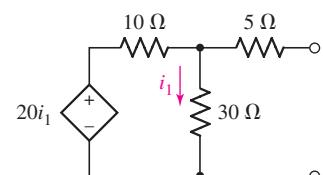
the meter, we may not even notice the effect of nonzero DMM resistance on our measurement.

The same meter can be used to determine resistance, provided no independent sources are active during the measurement. Internally, a known current is passed through the resistor being measured, and the voltmeter circuitry is used to measure the resulting voltage. Replacing the DMM with its Norton equivalent (which now includes an active independent current source to generate the predetermined current), we see that  $R_{DMM}$  appears in parallel with our unknown resistor  $R$  (Fig. 5.38).



■ FIGURE 5.38 DMM in resistance measurement configuration replaced by its Norton equivalent, showing  $R_{DMM}$  in parallel with the unknown resistor  $R$  to be measured.

As a result, the DMM actually measures  $R \parallel R_{DMM}$ . If  $R_{DMM} = 10\text{ M}\Omega$  and  $R = 10\ \Omega$ ,  $R_{measured} = 9.9999\ \Omega$ , which is more than accurate enough for most purposes. However, if  $R = 10\text{ M}\Omega$ ,  $R_{measured} = 5\text{ M}\Omega$ . The input resistance of a DMM therefore places a practical upper limit on the values of resistance that can be measured, and special techniques must be used to measure larger resistances. We should note that if a digital multimeter is *programmed* with knowledge of  $R_{DMM}$ , it is possible to compensate and allow measurement of larger resistances.



■ FIGURE 5.39 See Practice Problem 5.9.

## 5.4 MAXIMUM POWER TRANSFER

A very useful power theorem may be developed with reference to a practical voltage or current source. For the practical voltage source (Fig. 5.40), the power delivered to the load  $R_L$  is

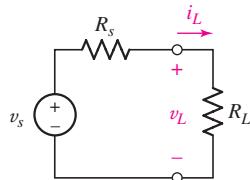


FIGURE 5.40 A practical voltage source connected to a load resistor  $R_L$ .

$$p_L = i_L^2 R_L = \frac{v_s^2 R_L}{(R_s + R_L)^2} \quad [19]$$

To find the value of  $R_L$  that absorbs maximum power from the given practical source, we differentiate with respect to  $R_L$ :

$$\frac{dp_L}{dR_L} = \frac{(R_s + R_L)^2 v_s^2 - v_s^2 R_L (2)(R_s + R_L)}{(R_s + R_L)^4}$$

and equate the derivative to zero, obtaining

$$2R_L(R_s + R_L) = (R_s + R_L)^2$$

or

$$R_s = R_L$$

Since the values  $R_L = 0$  and  $R_L = \infty$  both give a minimum ( $p_L = 0$ ), and since we have already developed the equivalence between practical voltage and current sources, we have therefore proved the following **maximum power transfer theorem**:

An independent voltage source in series with a resistance  $R_s$ , or an independent current source in parallel with a resistance  $R_s$ , delivers maximum power to a load resistance  $R_L$  such that  $R_L = R_s$ .

An alternative way to view the maximum power theorem is possible in terms of the Thévenin equivalent resistance of a network:

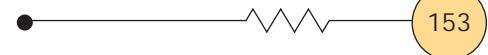
A network delivers maximum power to a load resistance  $R_L$  when  $R_L$  is equal to the Thévenin equivalent resistance of the network.

Thus, the maximum power transfer theorem tells us that a  $2\ \Omega$  resistor draws the greatest power (4.5 W) from either practical source of Fig. 5.16, whereas a resistance of  $0.01\ \Omega$  receives the maximum power (3.6 kW) in Fig. 5.11.

 There is a distinct difference between *drawing* maximum power from a *source* and *delivering* maximum power to a *load*. If the load is sized such that its Thévenin resistance is equal to the Thévenin resistance of the network to which it is connected, it will receive maximum power from that network. Any change to the load resistance will reduce the power delivered to the load. However, consider just the Thévenin equivalent of the network itself. We draw the maximum possible power from the voltage source by drawing the maximum possible current—which is achieved by shorting the network terminals! However, in this extreme example we *deliver zero power* to the “load”—a short circuit in this case—as  $p = i^2 R$ , and we just set  $R = 0$  by shorting the network terminals.

A minor amount of algebra applied to Eq. [19] coupled with the maximum power transfer requirement that  $R_L = R_s = R_{TH}$  will provide

$$p_{\max} \mid_{\text{delivered to load}} = \frac{v_s^2}{4R_s} = \frac{v_{TH}^2}{4R_{TH}}$$



where  $v_{TH}$  and  $R_{TH}$  recognize that the practical voltage source of Fig. 5.40 can also be viewed as a Thévenin equivalent of some specific source.

It is also not uncommon for the maximum power theorem to be misinterpreted. It is designed to help us select an optimum load in order to maximize power absorption. If the load resistance is already specified, however, the maximum power theorem is of no assistance. If for some reason we can affect the size of the Thévenin equivalent resistance of the network connected to our load, setting it equal to the load does not guarantee maximum power transfer to our predetermined load. A quick consideration of the power lost in the Thévenin resistance will clarify this point.



### EXAMPLE 5.11

The circuit shown in Fig. 5.41 is a model for the common-emitter bipolar junction transistor amplifier. Choose a load resistance so that maximum power is transferred to it from the amplifier, and calculate the actual power absorbed.

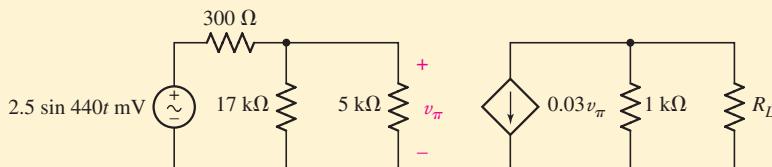
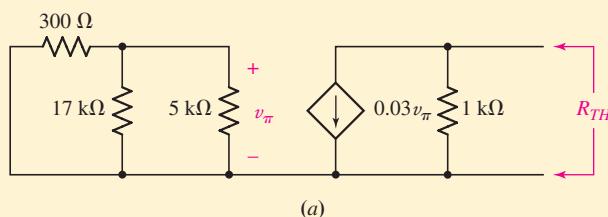


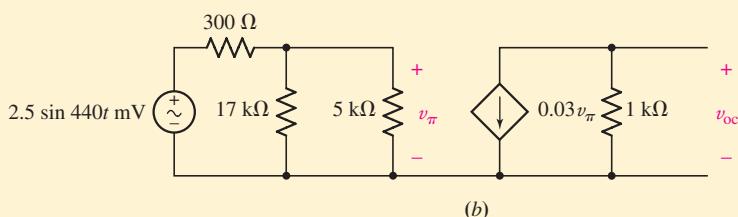
FIGURE 5.41 A small-signal model of the common-emitter amplifier, with the load resistance unspecified.

Since it is the load resistance we are asked to determine, the maximum power theorem applies. The first step is to find the Thévenin equivalent of the rest of the circuit.

We first determine the Thévenin equivalent resistance, which requires that we remove  $R_L$  and short-circuit the independent source as in Fig. 5.42a.



(a)



(b)

FIGURE 5.42 (a) Circuit with  $R_L$  removed and independent source short-circuited. (b) Circuit for determining  $v_{TH}$ .

(Continued on next page)

Since  $v_\pi = 0$ , the dependent current source is an open circuit, and  $R_{TH} = 1 \text{ k}\Omega$ . This can be verified by connecting an independent 1 A current source across the  $1 \text{ k}\Omega$  resistor;  $v_\pi$  will still be zero, so the dependent source remains inactive and hence contributes nothing to  $R_{TH}$ .

In order to obtain maximum power delivered into the load,  $R_L$  should be set to  $R_{TH} = 1 \text{ k}\Omega$ .

To find  $v_{TH}$  we consider the circuit shown in Fig. 5.42b, which is Fig. 5.41 with  $R_L$  removed. We may write

$$v_{oc} = -0.03v_\pi(1000) = -30v_\pi$$

where the voltage  $v_\pi$  may be found from simple voltage division:

$$v_\pi = (2.5 \times 10^{-3} \sin 440t) \left( \frac{3864}{300 + 3864} \right)$$

so that our Thévenin equivalent is a voltage  $-69.6 \sin 440t \text{ mV}$  in series with  $1 \text{ k}\Omega$ .

The maximum power is given by

$$P_{max} = \frac{v_{TH}^2}{4R_{TH}} = 1.211 \sin^2 440t \mu\text{W}$$

### PRACTICE

5.10 Consider the circuit of Fig. 5.43.

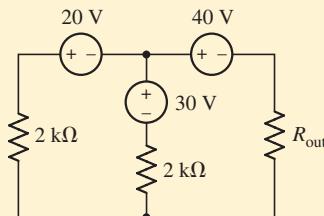


FIGURE 5.43

- (a) If  $R_{out} = 3 \text{ k}\Omega$ , find the power delivered to it.
- (b) What is the maximum power that can be delivered to any  $R_{out}$ ?
- (c) What two different values of  $R_{out}$  will have exactly 20 mW delivered to them?

Ans: 230 mW; 306 mW;  $59.2 \text{ k}\Omega$  and  $16.88 \Omega$ .

## 5.5 DELTA-WYE CONVERSION

We saw previously that identifying parallel and series combinations of resistors can often lead to a significant reduction in the complexity of a circuit. In situations where such combinations do not exist, we can often make use of source transformations to enable such simplifications. There is another useful technique, called  $\Delta$ - $Y$  (*delta-wye*) conversion, that arises out of network theory.

Consider the circuits in Fig. 5.44. There are no series or parallel combinations that can be made to further simplify any of the circuits (note that 5.44a and 5.44b are identical, as are 5.44c and 5.44d), and without any

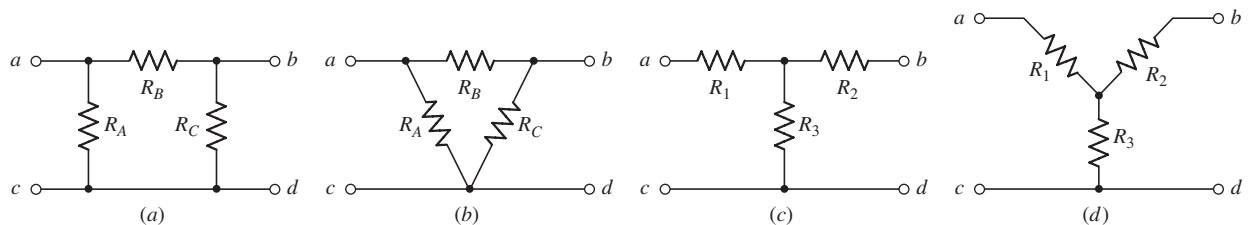


FIGURE 5.44 (a)  $\Pi$  network consisting of three resistors and three unique connections. (b) Same network drawn as a  $\Delta$  network. (c) A T network consisting of three resistors. (d) Same network drawn as a Y network.

sources present, no source transformations can be performed. However, it is possible to convert between these two types of networks.

We first define two voltages  $v_{ac}$  and  $v_{bc}$ , and three currents  $i_1$ ,  $i_2$ , and  $i_3$  as depicted in Fig. 5.45. If the two networks are equivalent, then the terminal voltages and currents must be equal (there is no current  $i_2$  in the T-connected network). A set of relationships between  $R_A$ ,  $R_B$ ,  $R_C$  and  $R_1$ ,  $R_2$ , and  $R_3$  can now be defined simply by performing mesh analysis. For example, for the network of Fig. 5.45a we may write

$$R_A i_1 - R_A i_2 = v_{ac} \quad [20]$$

$$-R_A i_1 + (R_A + R_B + R_C) i_2 - R_C i_3 = 0 \quad [21]$$

$$-R_C i_2 + R_C i_3 = -v_{bc} \quad [22]$$

and for the network of Fig. 5.45b we have

$$(R_1 + R_3) i_1 - R_3 i_3 = v_{ac} \quad [23]$$

$$-R_3 i_1 + (R_2 + R_3) i_3 = -v_{bc} \quad [24]$$

We next remove  $i_2$  from Eqs. [20] and [22] using Eq. [21], resulting in

$$\left( R_A - \frac{R_A^2}{R_A + R_B + R_C} \right) i_1 - \frac{R_A R_C}{R_A + R_B + R_C} i_3 = v_{ac} \quad [25]$$

and

$$-\frac{R_A R_C}{R_A + R_B + R_C} i_1 + \left( R_C - \frac{R_C^2}{R_A + R_B + R_C} \right) i_3 = -v_{bc} \quad [26]$$

Comparing terms between Eq. [25] and Eq. [23], we see that

$$R_3 = \frac{R_A R_C}{R_A + R_B + R_C}$$

In a similar fashion, we may find expressions for  $R_1$  and  $R_2$  in terms of  $R_A$ ,  $R_B$ , and  $R_C$ , as well as expressions for  $R_A$ ,  $R_B$ , and  $R_C$  in terms of  $R_1$ ,  $R_2$ , and  $R_3$ ; we leave the remainder of the derivations as an exercise for the reader. Thus, to convert from a Y network to a  $\Delta$  network, the new resistor values are calculated using

$$\boxed{\begin{aligned} R_A &= \frac{R_1 R_2 + R_2 R_3 + R_3 R_1}{R_2} \\ R_B &= \frac{R_1 R_2 + R_2 R_3 + R_3 R_1}{R_3} \\ R_C &= \frac{R_1 R_2 + R_2 R_3 + R_3 R_1}{R_1} \end{aligned}}$$

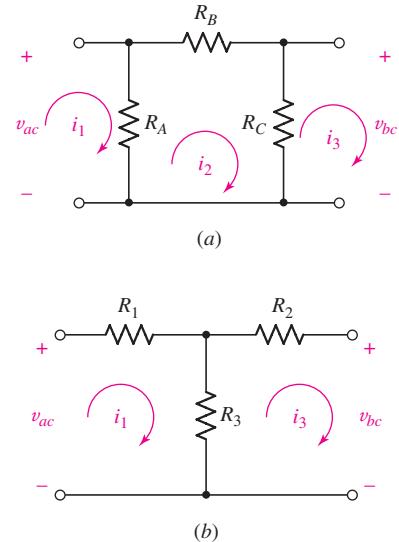


FIGURE 5.45 (a) Labeled  $\Pi$  network; (b) labeled T network.

and to convert from a  $\Delta$  network to a Y network,

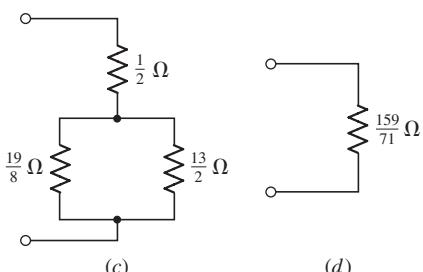
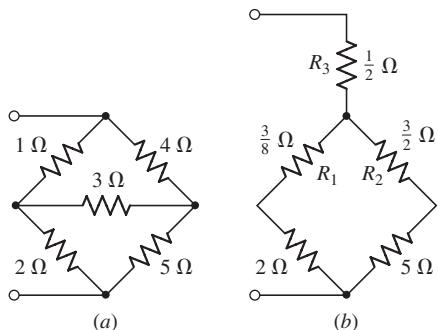
$$R_1 = \frac{R_A R_B}{R_A + R_B + R_C}$$

$$R_2 = \frac{R_B R_C}{R_A + R_B + R_C}$$

$$R_3 = \frac{R_C R_A}{R_A + R_B + R_C}$$

Application of these equations is straightforward, although identifying the actual networks sometimes requires a little concentration.

### EXAMPLE 5.12



**FIGURE 5.46** (a) A given resistive network whose input resistance is desired. (b) The upper  $\Delta$  network is replaced by an equivalent Y network. (c, d) Series and parallel combinations result in a single resistance value.

Use the technique of  $\Delta$ -Y conversion to find the Thévenin equivalent resistance of the circuit in Fig. 5.46a.

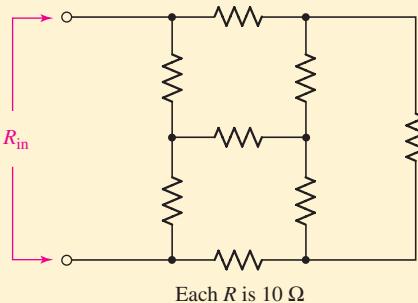
We see that the network in Fig. 5.46a is composed of two  $\Delta$ -connected networks that share the  $3\ \Omega$  resistor. We must be careful at this point not to be too eager, attempting to convert both  $\Delta$ -connected networks to two Y-connected networks. The reason for this may be more obvious after we convert the top network consisting of the  $1\ \Omega$ ,  $4\ \Omega$ , and  $3\ \Omega$  resistors into a Y-connected network (Fig. 5.46b).

Note that in converting the upper network to a Y-connected network, we have removed the  $3\ \Omega$  resistor. As a result, there is no way to convert the original  $\Delta$ -connected network consisting of the  $2\ \Omega$ ,  $5\ \Omega$ , and  $3\ \Omega$  resistors into a Y-connected network.

We proceed by combining the  $\frac{3}{8}\ \Omega$  and  $2\ \Omega$  resistors and the  $\frac{3}{2}\ \Omega$  and  $5\ \Omega$  resistors (Fig. 5.46c). We now have a  $\frac{19}{8}\ \Omega$  resistor in parallel with a  $\frac{13}{2}\ \Omega$  resistor, and this parallel combination is in series with the  $\frac{1}{2}\ \Omega$  resistor. Thus, we can replace the original network of Fig. 5.46a with a single  $\frac{159}{71}\ \Omega$  resistor (Fig. 5.46d).

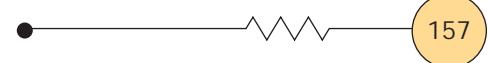
### PRACTICE

5.11 Use the technique of Y- $\Delta$  conversion to find the Thévenin equivalent resistance of the circuit of Fig. 5.47.



**FIGURE 5.47**

Ans:  $11.43\ \Omega$ .



## 5.6 SELECTING AN APPROACH: A SUMMARY OF VARIOUS TECHNIQUES

In Chap. 3, we were introduced to Kirchhoff's current law (KCL) and Kirchhoff's voltage law (KVL). These two laws apply to any circuit we will ever encounter, provided that we take care to consider the entire system that the circuits represent. The reason for this is that KCL and KVL enforce charge and energy conservation, respectively, which are fundamental principles. Based on KCL, we developed the very powerful method of nodal analysis. A similar technique based on KVL (unfortunately only applicable to planar circuits) is known as mesh analysis and is also a useful circuit analysis approach.

For the most part, this text is concerned with developing analytical skills that apply to *linear* circuits. If we know a circuit is constructed of only linear components (in other words, all voltages and currents are related by linear functions), then we can often simplify circuits prior to employing either mesh or nodal analysis. Perhaps the most important result that comes from the knowledge that we are dealing with a completely linear system is that the principle of superposition applies: given a number of independent sources acting on our circuit, we can add the contribution of each source independently of the other sources. This technique is extremely pervasive throughout the field of engineering, and we will encounter it often. In many real situations, we will find that although several "sources" are acting simultaneously on our "system," typically one of them dominates the system response. Superposition allows us to quickly identify that source, provided that we have a reasonably accurate linear model of the system.

However, from a circuit analysis standpoint, unless we are asked to find which independent source contributes the most to a particular response, we find that rolling up our sleeves and launching straight into either nodal or mesh analysis is often a more straightforward tactic. The reason for this is that applying superposition to a circuit with 12 independent sources will require us to redraw the original circuit 12 times, and often we will have to apply nodal or mesh analysis to each partial circuit, anyway.

The technique of source transformations, on the other hand, is often a very useful tool in circuit analysis. Performing source transformations can allow us to consolidate resistors or sources that are not in series or parallel in the original circuit. Source transformations may also allow us to convert all or at least most of the sources in the original circuit to the same type (either all voltage sources or all current sources), so nodal or mesh analysis is more straightforward.

Thévenin's theorem is extremely important for a number of reasons. In working with electronic circuits, we are always aware of the Thévenin equivalent resistance of different parts of our circuit, especially the input and output resistances of amplifier stages. The reason for this is that matching of resistances is frequently the best route to optimizing the performance of a given circuit. We have seen a small preview of this in our discussion of maximum power transfer, where the load resistance should be chosen to match the Thévenin equivalent resistance of the network to which the load is connected. In terms of day-to-day circuit analysis, however, we find that converting part of a circuit to its Thévenin or Norton equivalent is almost as much work as analyzing the complete circuit. Therefore, as in the case of

superposition, Thévenin's and Norton's theorems are typically applied only when we require specialized information about part of our circuit.

## SUMMARY AND REVIEW

Although we asserted in Chap. 4 that nodal analysis and mesh analysis are sufficient to analyze any circuit we might encounter (provided we have the means to relate voltage and current for any passive element, such as Ohm's law for resistors), the simple truth is that often we do not really need *all* voltages, or *all* currents. Sometimes, it is simply *one* element, or a *small portion* of a larger circuit, that has our attention. Perhaps there is some uncertainty in the final value of a particular element, and we'd like to see how the circuit performs over the range of expected values. In such instances, we can exploit the fact that at the moment we have confined ourselves to *linear* circuits. This allows the development of other tools: *superposition*, where individual contributions of sources can be identified; *source transformations*, where a voltage source in series with a resistor can be replaced with a current source in parallel with a resistor; and the most powerful of all—*Thévenin* (and *Norton*) *equivalents*.

An interesting offshoot of these topics is the idea of *maximum power transfer*. Assuming we can represent our (arbitrarily complex) circuit by two networks, one passive and one active, maximum power transfer to the passive network is achieved when its Thévenin resistance is equal to the Thévenin resistance of the active network. Finally, we introduced the concept of delta-wye conversion, a process that allows us to simplify some resistive networks which at face value are not reducible using standard series-parallel combination techniques.

We are still faced with the perpetual question, “*Which tool should I use to analyze this circuit?*” The answer typically lies in the type of information required about our circuit. Experience will eventually guide us a bit, but it is not always true that there is one “best” approach. Certainly one issue to focus on is whether one or more components might be changed—this can suggest whether superposition, a Thévenin equivalent, or a partial simplification such as can be achieved with source or delta-wye transformation is the most practical route.

We conclude this chapter by reviewing key points, along with identifying relevant example(s).

- ❑ The principle of superposition states that the *response* in a linear circuit can be obtained by adding the individual responses caused by the separate *independent* sources *acting alone*. (Examples 5.1, 5.2, 5.3)
- ❑ Superposition is most often used when it is necessary to determine the individual contribution of each source to a particular response. (Examples 5.2, 5.3)
- ❑ A practical model for a real voltage source is a resistor in series with an independent voltage source. A practical model for a real current source is a resistor in parallel with an independent current source.
- ❑ Source transformations allow us to convert a practical voltage source into a practical current source, and vice versa. (Example 5.4)

- Repeated source transformations can greatly simplify analysis of a circuit by providing the means to combine resistors and sources. (Example 5.5)
- The Thévenin equivalent of a network is a resistor in series with an independent voltage source. The Norton equivalent is the same resistor in parallel with an independent current source. (Example 5.6)
- There are several ways to obtain the Thévenin equivalent resistance, depending on whether or not dependent sources are present in the network. (Examples 5.7, 5.8, 5.9, 5.10)
- Maximum power transfer occurs when the load resistor matches the Thévenin equivalent resistance of the network to which it is connected. (Example 5.11)
- When faced with a  $\Delta$ -connected resistor network, it is straightforward to convert it to a Y-connected network. This can be useful in simplifying the network prior to analysis. Conversely, a Y-connected resistor network can be converted to a  $\Delta$ -connected network to assist in simplification of the network. (Example 5.12)

## READING FURTHER

A book about battery technology, including characteristics of built-in resistance:

D. Linden, *Handbook of Batteries*, 2nd ed. New York: McGraw-Hill, 1995.

An excellent discussion of pathological cases and various circuit analysis theorems can be found in:

R. A. DeCarlo and P. M. Lin, *Linear Circuit Analysis*, 2nd ed. New York: Oxford University Press, 2001.

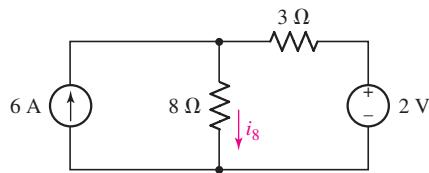
## EXERCISES

### 5.1 Linearity and Superposition

1. Linear systems are so easy to work with that engineers often construct linear models of real (nonlinear) systems to assist in analysis and design. Such models are often surprisingly accurate over a limited range. For example, consider the simple exponential function  $e^x$ . The Taylor series representation of this function is

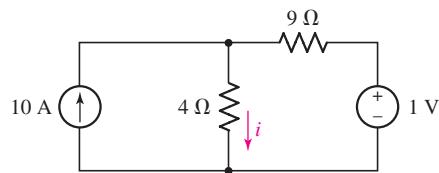
$$e^x \approx 1 + x + \frac{x^2}{2} + \frac{x^3}{6} + \dots$$

- (a) Construct a linear model for this function by truncating the Taylor series expansion after the linear term. (b) Evaluate your model function at  $x = 0.000001, 0.0001, 0.01, 0.1$ , and  $1.0$ . (c) For which values of  $x$  does your model yield a “reasonable” approximation to  $e^x$ ? Explain your reasoning.
2. Construct a linear approximation to the function  $y(t) = 4 \sin 2t$ . (a) Evaluate your approximation at  $t = 0, 0.001, 0.01, 0.1$ , and  $1.0$ . (b) For which values of  $t$  does your model provide a “reasonable” approximation to the actual (nonlinear) function  $y(t)$ ? Explain your reasoning.
3. Considering the circuit of Fig. 5.48, employ superposition to determine the two components of  $i_8$  arising from the action of the two independent sources, respectively.



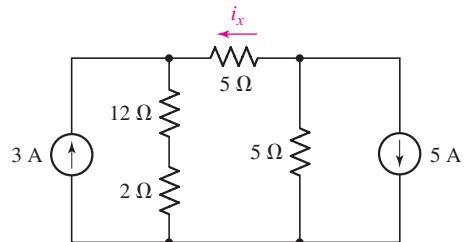
■ FIGURE 5.48

4. (a) Employ superposition to determine the current labeled  $i$  in the circuit of Fig. 5.49. (b) Express the contribution the 1 V source makes to the total current  $i$  in terms of a percentage. (c) Changing only the value of the 10 A source, adjust the circuit of Fig. 5.49 so that the two sources contribute equally to the current  $i$ .



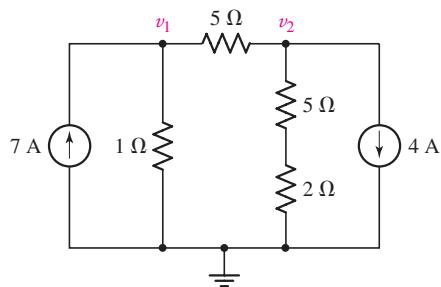
■ FIGURE 5.49

5. (a) Employ superposition to obtain the individual contributions each of the two sources in Fig. 5.50 makes to the current labeled  $i_x$ . (b) Adjusting only the value of the rightmost current source, alter the circuit so that the two sources contribute equally to  $i_x$ .



■ FIGURE 5.50

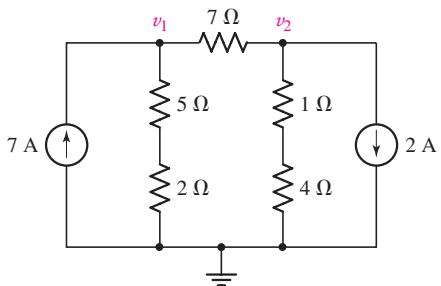
6. (a) Determine the individual contributions of each of the two current sources in the circuit of Fig. 5.51 to the nodal voltage  $v_1$ . (b) Determine the percentage contribution of each of the two sources to the power dissipated by the 2 Ω resistor.



■ FIGURE 5.51

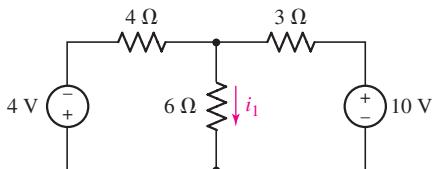


7. (a) Determine the individual contributions of each of the two current sources shown in Fig. 5.52 to the nodal voltage labeled  $v_2$ . (b) Instead of performing two separate PSpice simulations, verify your answer by using a single dc sweep. Submit a labeled schematic, relevant Probe output, and a short description of the results.



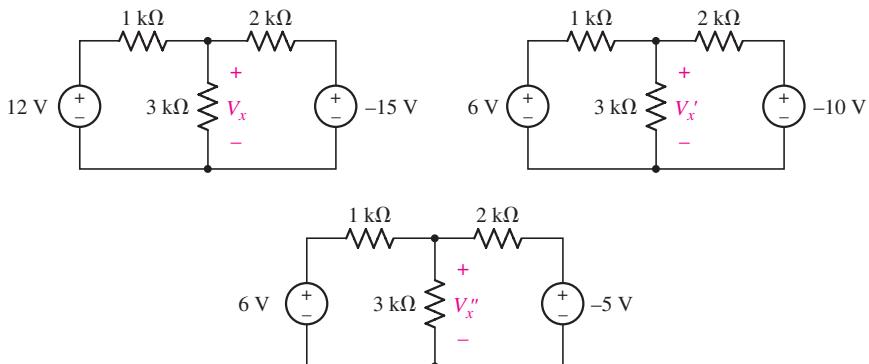
■ FIGURE 5.52

8. After studying the circuit of Fig. 5.53, change both voltage source values such that (a)  $i_1$  doubles; (b) the direction of  $i_1$  reverses, but its magnitude is unchanged; (c) both sources contribute equally to the power dissipated by the  $6\ \Omega$  resistor.



■ FIGURE 5.53

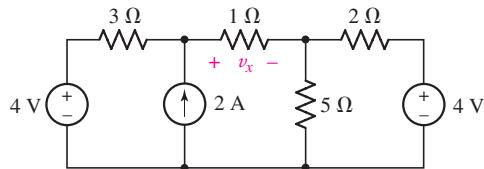
9. Consider the three circuits shown in Fig. 5.54. Analyze each circuit, and demonstrate that  $V_x = V'_x + V''_x$  (i.e., superposition is most useful when sources are set to zero, but the principle is in fact much more general than that).



■ FIGURE 5.54

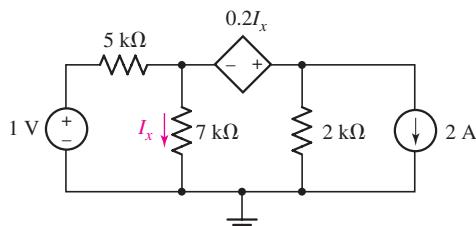


10. (a) Using superposition, determine the voltage labeled  $v_x$  in the circuit represented in Fig. 5.55. (b) To what value should the 2 A source be changed to reduce  $v_x$  by 10%? (c) Verify your answers by performing three dc sweeps in PSpice (one for each source). Submit a labeled schematic, relevant Probe output, and a short description of the results.



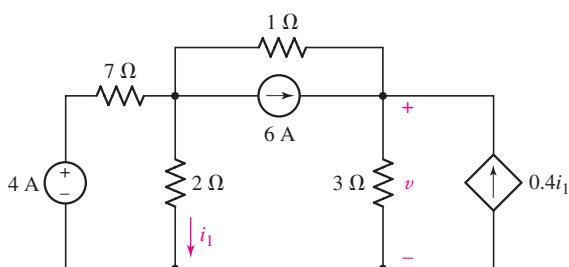
■ FIGURE 5.55

11. Employ superposition principles to obtain a value for the current  $I_x$  as labeled in Fig. 5.56.



■ FIGURE 5.56

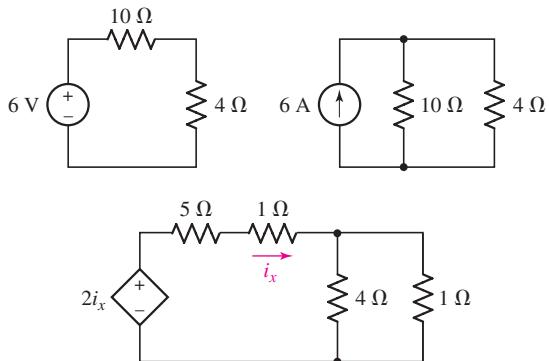
12. (a) Employ superposition to determine the individual contribution from each independent source to the voltage  $v$  as labeled in the circuit shown in Fig. 5.57.  
 (b) Compute the power absorbed by the 2 Ω resistor.



■ FIGURE 5.57

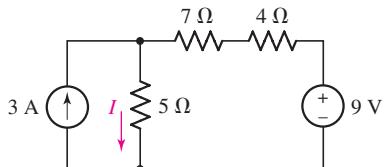
## 5.2 Source Transformations

13. Perform an appropriate source transformation on each of the circuits depicted in Fig. 5.58, taking care to retain the 4 Ω resistor in each final circuit.



■ FIGURE 5.58

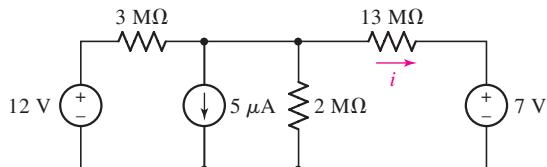
14. For the circuit of Fig. 5.59, plot  $i_L$  versus  $v_L$  corresponding to the range of  $0 \leq R \leq \infty$ .
15. Determine the current labeled  $I$  in the circuit of Fig. 5.60 by first performing source transformations and parallel-series combinations as required to reduce the circuit to only two elements.



■ FIGURE 5.60

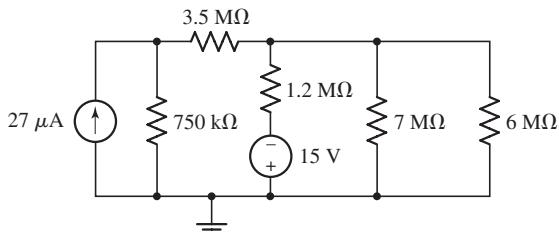


16. Verify that the power absorbed by the  $7 \Omega$  resistor in Fig. 5.22a remains the same after the source transformation illustrated in Fig. 5.22c.
17. (a) Determine the current labeled  $i$  in the circuit of Fig. 5.61 after first transforming the circuit such that it contains only resistors and voltage sources.  
 (b) Simulate each circuit to verify the same current flows in both cases.



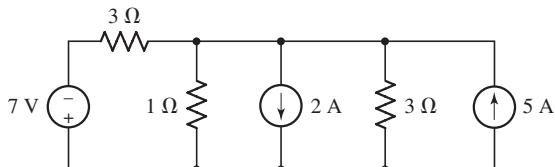
■ FIGURE 5.61

18. (a) Using repeated source transformations, reduce the circuit of Fig. 5.62 to a voltage source in series with a resistor, both of which are in series with the  $6 \text{ M}\Omega$  resistor. (b) Calculate the power dissipated by the  $6 \text{ M}\Omega$  resistor using your simplified circuit.

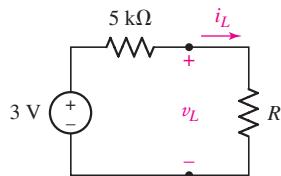


■ FIGURE 5.62

19. (a) Using as many source transformations and element combination techniques as required, simplify the circuit of Fig. 5.63 so that it contains only the  $7 \text{ V}$  source, a single resistor, and one other voltage source. (b) Verify that the  $7 \text{ V}$  source delivers the same amount of power in both circuits.



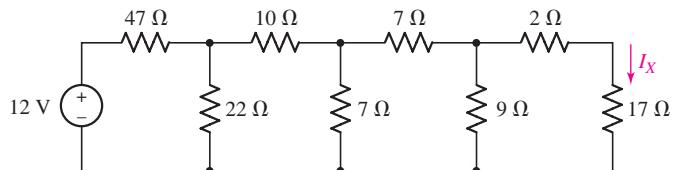
■ FIGURE 5.63



■ FIGURE 5.59

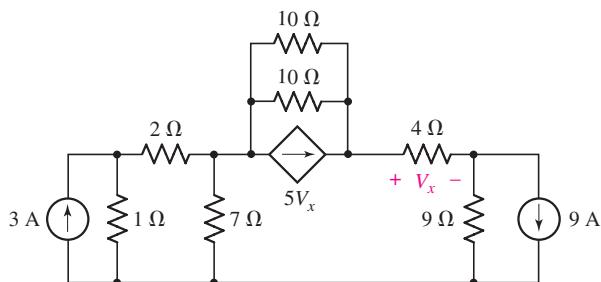


20. (a) Making use of repeated source transformations, reduce the circuit of Fig. 5.64 such that it contains a single voltage source, the  $17\ \Omega$  resistor, and one other resistor. (b) Calculate the power dissipated by the  $17\ \Omega$  resistor. (c) Verify your results by simulating both circuits with PSpice or another suitable CAD tool.



■ FIGURE 5.64

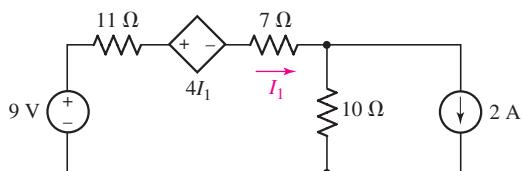
21. Make use of source transformations to first convert all three sources in Fig. 5.65 to voltage sources, then simplify the circuit as much as possible and calculate the voltage  $V_x$  which appears across the  $4\ \Omega$  resistor. Be sure to draw and label your simplified circuit.



■ FIGURE 5.65

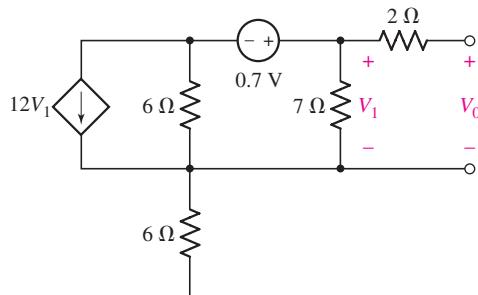


22. (a) With the assistance of source transformations, alter the circuit of Fig. 5.66 such that it contains only current sources. (b) Simplify your new circuit as much as possible, and calculate the power dissipated in the  $7\ \Omega$  resistor. (c) Verify your solution by simulating both circuits with PSpice or another appropriate CAD tool.



■ FIGURE 5.66

23. Transform the dependent source in Fig. 5.67 to a voltage source, then calculate  $V_0$ .



■ FIGURE 5.67

24. With regard to the circuit represented in Fig. 5.68, first transform both voltage sources to current sources, reduce the number of elements as much as possible, and determine the voltage  $v_3$ .

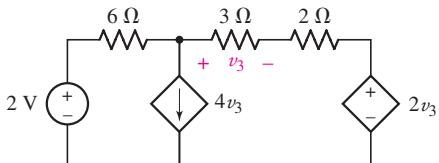


FIGURE 5.68

### 5.3 Thévenin and Norton Equivalent Circuits

25. Referring to Fig. 5.69, determine the Thévenin equivalent of the network connected to  $R_L$ . (b) Determine  $v_L$  for  $R_L = 1 \Omega$ ,  $3.5 \Omega$ ,  $6.257 \Omega$ , and  $9.8 \Omega$ .
26. (a) With respect to the circuit depicted in Fig. 5.69, obtain the Norton equivalent of the network connected to  $R_L$ . (b) Plot the power dissipated in resistor  $R_L$  as a function of  $i_L$  corresponding to the range of  $0 < R_L < 5 \Omega$ . (c) Using your graph, estimate at what value of  $R_L$  does the dissipated power reach its maximum value.
27. (a) Obtain the Norton equivalent of the network connected to  $R_L$  in Fig. 5.70. (b) Obtain the Thévenin equivalent of the same network. (c) Use either to calculate  $i_L$  for  $R_L = 0 \Omega$ ,  $1 \Omega$ ,  $4.923 \Omega$ , and  $8.107 \Omega$ .
28. (a) Determine the Thévenin equivalent of the circuit depicted in Fig. 5.71 by first finding  $V_{oc}$  and  $I_{sc}$  (defined as flowing into the positive reference terminal of  $V_{oc}$ ). (b) Connect a  $4.7 \text{ k}\Omega$  resistor to the open terminals of your new network and calculate the power it dissipates.
29. Referring to the circuit of Fig. 5.71: (a) Determine the Norton equivalent of the circuit by first finding  $V_{oc}$  and  $I_{sc}$  (defined as flowing into the positive reference terminal of  $V_{oc}$ ). (b) Connect a  $1.7 \text{ k}\Omega$  resistor to the open terminals of your new network and calculate the power supplied to that resistor.
30. (a) Employ Thévenin's theorem to obtain a simple two-component equivalent of the circuit shown in Fig. 5.72. (b) Use your equivalent circuit to determine the power delivered to a  $100 \Omega$  resistor connected to the open terminals. (c) Verify your solution by analyzing the original circuit with the same  $100 \Omega$  resistor connected across the open terminals.

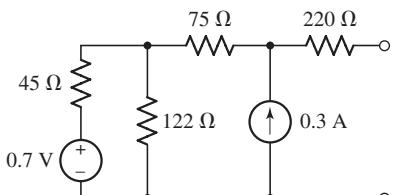


FIGURE 5.72

31. (a) Employ Thévenin's theorem to obtain a two-component equivalent for the network shown in Fig. 5.73. (b) Determine the power supplied to a  $1 \text{ M}\Omega$  resistor connected to the network if  $i_1 = 19 \mu\text{A}$ ,  $R_1 = R_2 = 1.6 \text{ M}\Omega$ ,  $R_3 = 3 \text{ M}\Omega$ , and  $R_4 = R_5 = 1.2 \text{ M}\Omega$ . (c) Verify your solution by simulating both circuits with PSpice or another appropriate CAD tool.

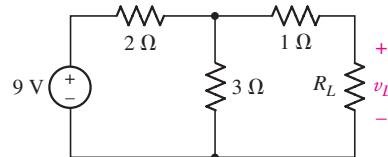


FIGURE 5.69

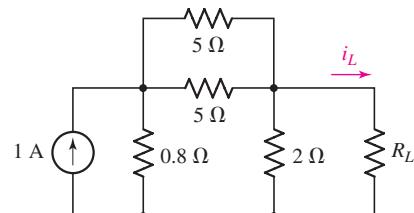


FIGURE 5.70

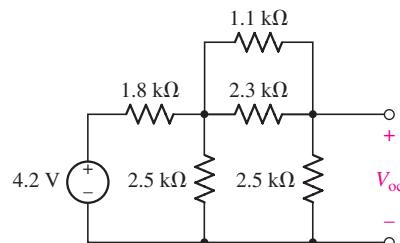


FIGURE 5.71

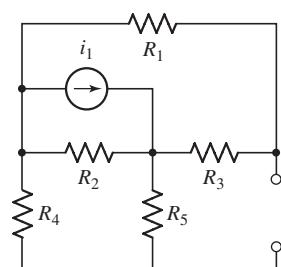
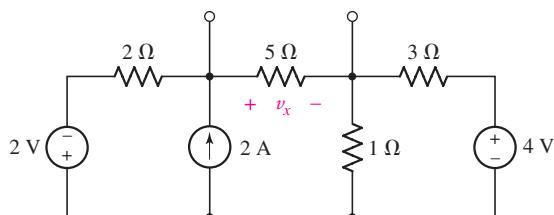


FIGURE 5.73

32. Determine the Thévenin equivalent of the network shown in Fig. 5.74 as seen looking into the two open terminals.

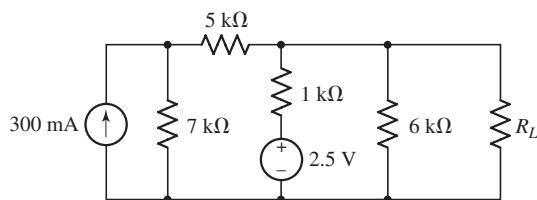


■ FIGURE 5.74

33. (a) Determine the Norton equivalent of the circuit depicted in Fig. 5.74 as seen looking into the two open terminals. (b) Compute power dissipated in a  $5\ \Omega$  resistor connected in parallel with the existing  $5\ \Omega$  resistor. (c) Compute the current flowing through a short circuit connecting the two terminals.

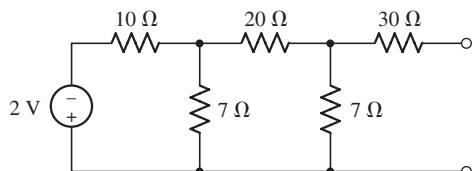


34. For the circuit of Fig. 5.75: (a) Employ Norton's theorem to reduce the network connected to  $R_L$  to only two components. (b) Calculate the downward-directed current flowing through  $R_L$  if it is a  $3.3\text{ k}\Omega$  resistor. (c) Verify your answer by simulating both circuits with PSpice or a comparable CAD tool.



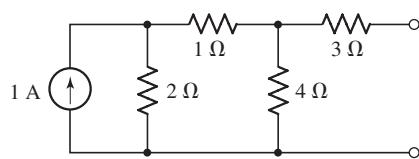
■ FIGURE 5.75

35. (a) Obtain a value for the Thévenin equivalent resistance seen looking into the open terminals of the circuit in Fig. 5.76 by first finding  $V_{oc}$  and  $I_{sc}$ . (b) Connect a 1 A test source to the open terminals of the original circuit after shorting the voltage source, and use this to obtain  $R_{TH}$ . (c) Connect a 1 V test source to the open terminals of the original circuit after again zeroing the 2 V source, and use this now to obtain  $R_{TH}$ .



■ FIGURE 5.76

36. Refer to the circuit depicted in Fig. 5.77. (a) Obtain a value for the Thévenin equivalent resistance seen looking into the open terminals by first finding  $V_{oc}$  and  $I_{sc}$ . (b) Connect a 1 A test source to the open terminals of the original

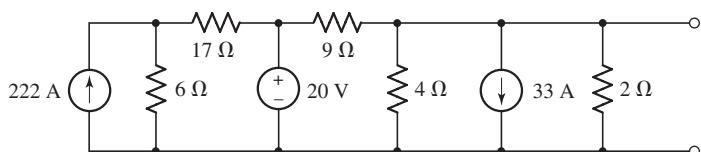


■ FIGURE 5.77

circuit after deactivating the other current source, and use this to obtain  $R_{TH}$ .

(c) Connect a 1 V test source to the open terminals of the original circuit, once again zeroing out the original source, and use this now to obtain  $R_{TH}$ .

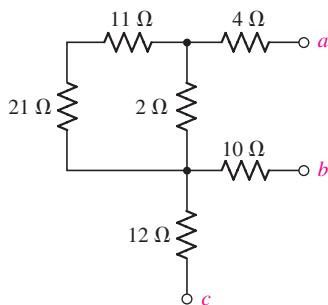
37. Obtain a value for the Thévenin equivalent resistance seen looking into the open terminals of the circuit in Fig. 5.78 by (a) finding  $V_{oc}$  and  $I_{sc}$ , and then taking their ratio; (b) setting all independent sources to zero and using resistor combination techniques; (c) connecting an unknown current source to the terminals, deactivating (zero out) all other sources, finding an algebraic expression for the voltage that develops across the source, and taking the ratio of the two quantities.



■ FIGURE 5.78



38. With regard to the network depicted in Fig. 5.79, determine the Thévenin equivalent as seen by an element connected to terminals (a) *a* and *b*; (b) *a* and *c*; (c) *b* and *c*. (d) Verify your answers using PSpice or other suitable CAD tool. (Hint: Connect a test source to the terminals of interest.)

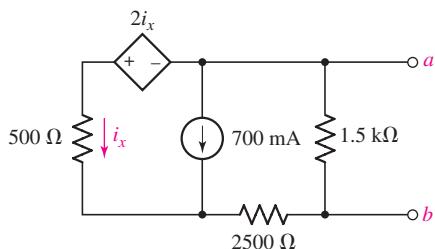


■ FIGURE 5.79

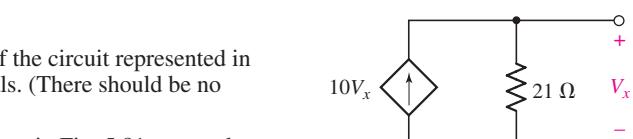
39. Determine the Thévenin and Norton equivalents of the circuit represented in Fig. 5.80 from the perspective of the open terminals. (There should be no dependent sources in your answer.)

40. Determine the Norton equivalent of the circuit drawn in Fig. 5.81 as seen by terminals *a* and *b*. (There should be no dependent sources in your answer.)

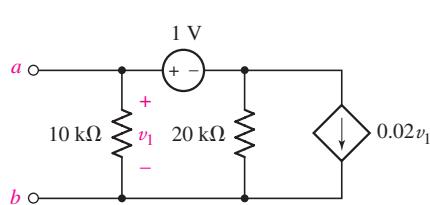
41. With regard to the circuit of Fig. 5.82, determine the power dissipated by (a) a 1 kΩ resistor connected between *a* and *b*; (b) a 4.7 kΩ resistor connected between *a* and *b*; (c) a 10.54 kΩ resistor connected between *a* and *b*.



■ FIGURE 5.81



■ FIGURE 5.80



■ FIGURE 5.82

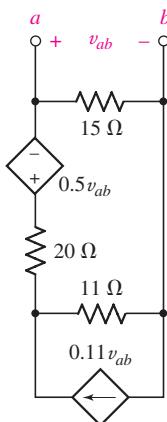


FIGURE 5.83

42. Determine the Thévenin and Norton equivalents of the circuit shown in Fig. 5.83, as seen by an unspecified element connected between terminals *a* and *b*.

43. Referring to the circuit of Fig. 5.84, determine the Thévenin equivalent resistance of the circuit to the right of the dashed line. This circuit is a common-source transistor amplifier, and you are calculating its input resistance.

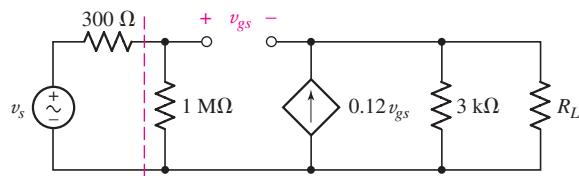


FIGURE 5.84

44. Referring to the circuit of Fig. 5.85, determine the Thévenin equivalent resistance of the circuit to the right of the dashed line. This circuit is a common-collector transistor amplifier, and you are calculating its input resistance.

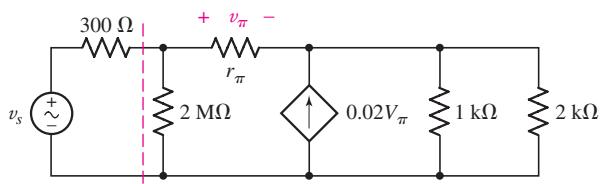


FIGURE 5.85

45. The circuit shown in Fig. 5.86 is a reasonably accurate model of an operational amplifier. In cases where  $R_i$  and  $A$  are very large and  $R_o \sim 0$ , a resistive load (such as a speaker) connected between ground and the terminal labeled  $v_{out}$  will see a voltage  $-R_f/R_1$  times larger than the input signal  $v_{in}$ . Find the Thévenin equivalent of the circuit, taking care to label  $v_{out}$ .

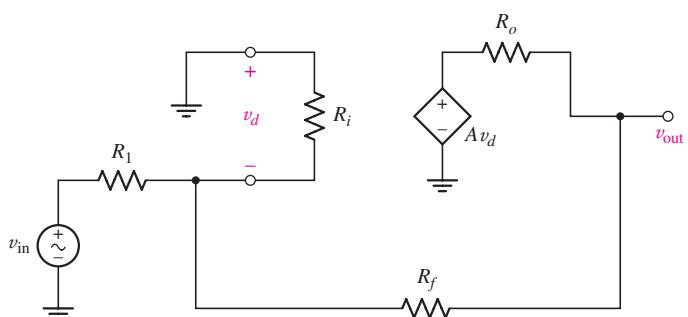


FIGURE 5.86

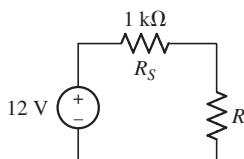
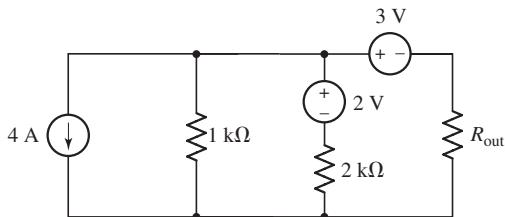


FIGURE 5.87

## 5.4 Maximum Power Transfer

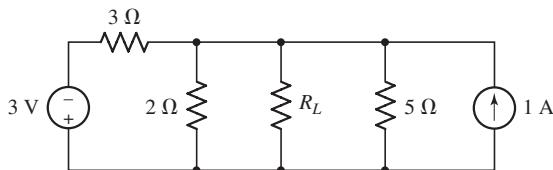
46. (a) For the simple circuit of Fig. 5.87, graph the power dissipated by the resistor  $R$  as a function of  $R/R_S$ , if  $0 \leq R \leq 3000 \Omega$ . (b) Graph the first derivative of the power versus  $R/R_S$ , and verify that maximum power is transferred to  $R$  when it is equal to  $R_S$ .

47. For the circuit drawn in Fig. 5.88, (a) determine the Thévenin equivalent connected to  $R_{out}$ . (b) Choose  $R_{out}$  such that maximum power is delivered to it.
48. Study the circuit of Fig. 5.89. (a) Determine the Norton equivalent connected to resistor  $R_{out}$ . (b) Select a value for  $R_{out}$  such that maximum power will be delivered to it.



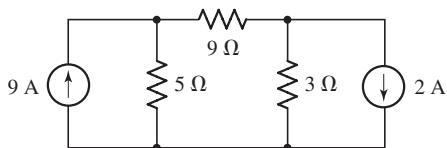
■ FIGURE 5.89

49. Assuming that we can determine the Thévenin equivalent resistance of our wall socket, why don't toaster, microwave oven, and TV manufacturers match each appliance's Thévenin equivalent resistance to this value? Wouldn't it permit maximum power transfer from the utility company to our household appliances?
50. For the circuit of Fig. 5.90, what value of  $R_L$  will ensure it absorbs the maximum possible amount of power?



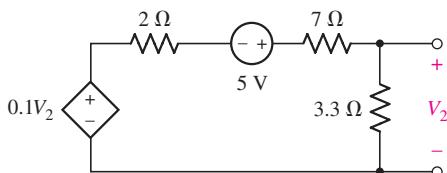
■ FIGURE 5.90

51. With reference to the circuit of Fig. 5.91, (a) calculate the power absorbed by the 9 Ω resistor; (b) adjust the size of the 5 Ω resistor so that the new network delivers maximum power to the 9 Ω resistor.

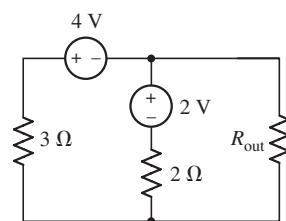


■ FIGURE 5.91

52. Referring to the circuit of Fig. 5.92, (a) determine the power absorbed by the 3.3 Ω resistor; (b) replace the 3.3 Ω resistor with another resistor such that it absorbs maximum power from the rest of the circuit.

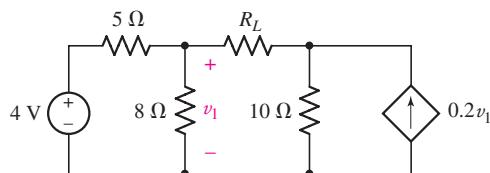


■ FIGURE 5.92



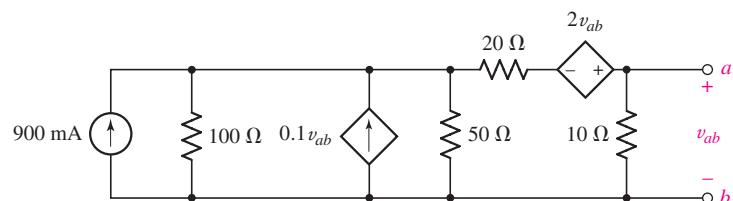
■ FIGURE 5.88

53. Select a value for  $R_L$  in Fig. 5.93 such that it is ensured to absorb maximum power from the circuit.



■ FIGURE 5.93

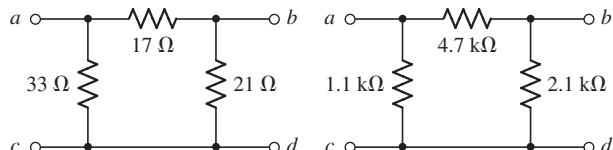
54. Determine what value of resistance would absorb maximum power from the circuit of Fig. 5.94 when connected across terminals *a* and *b*.



■ FIGURE 5.94

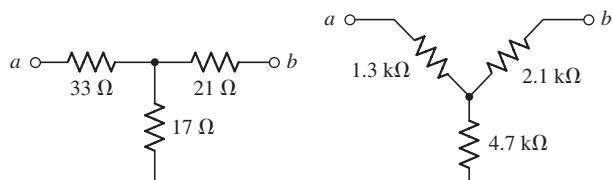
## 5.5 Delta-Wye Conversion

55. Derive the equations required to convert from a Y-connected network to a  $\Delta$ -connected network.  
 56. Convert the  $\Delta$ - (or “ $\Pi$ -”) connected networks in Fig. 5.95 to Y-connected networks.



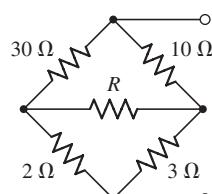
■ FIGURE 5.95

57. Convert the Y- (or “T-”) connected networks in Fig. 5.96 to  $\Delta$ -connected networks.



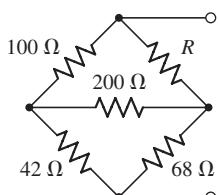
■ FIGURE 5.96

58. For the network of Fig. 5.97, select a value of  $R$  such that the network has an equivalent resistance of 9  $\Omega$ . Round your answer to two significant figures.



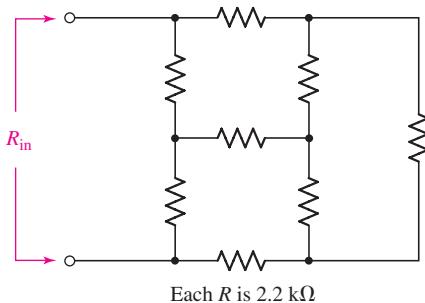
■ FIGURE 5.97

59. For the network of Fig. 5.98, select a value of  $R$  such that the network has an equivalent resistance of  $70.6 \Omega$ .



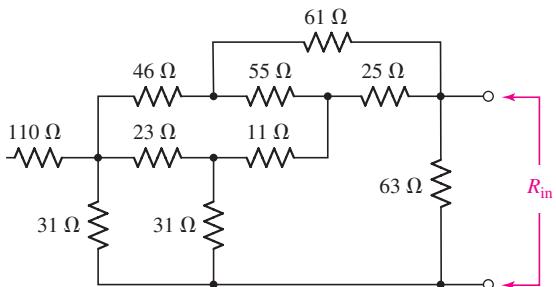
■ FIGURE 5.98

60. Determine the effective resistance  $R_{in}$  of the network exhibited in Fig. 5.99.



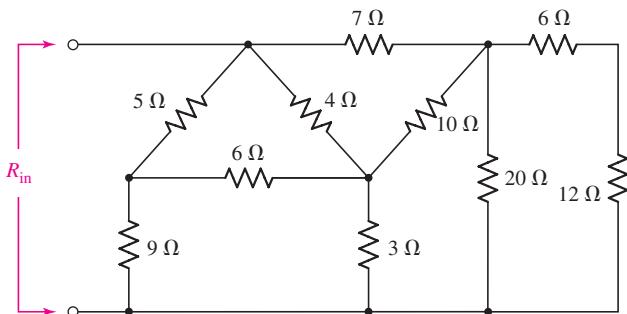
■ FIGURE 5.99

61. Calculate  $R_{in}$  as indicated in Fig. 5.100.



■ FIGURE 5.100

62. Employ  $\Delta/Y$  conversion techniques as appropriate to determine  $R_{in}$  as labeled in Fig. 5.101.



■ FIGURE 5.101

63. (a) Determine the two-component Thévenin equivalent of the network in Fig. 5.102. (b) Calculate the power dissipated by a  $1\ \Omega$  resistor connected between the open terminals.

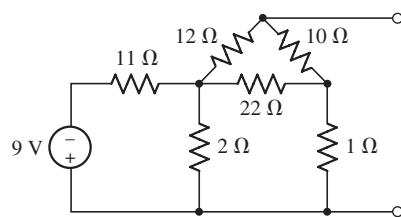


FIGURE 5.102

64. (a) Use appropriate techniques to obtain both the Thévenin and Norton equivalents of the network drawn in Fig. 5.103. (b) Verify your answers by simulating each of the three circuits connected to a  $1\ \Omega$  resistor.

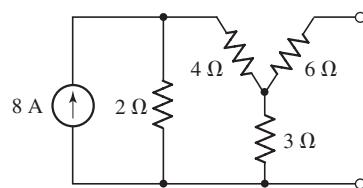


FIGURE 5.103

65. (a) Replace the network in Fig. 5.104 with an equivalent three-resistor  $\Delta$  network.  
 (b) Perform a PSpice analysis to verify that your answer is in fact equivalent. (Hint: Try adding a load resistor.)

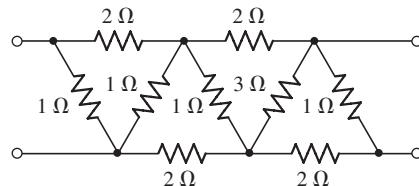


FIGURE 5.104

## 5.6 Selecting an Approach: A Summary of Various Techniques

66. Determine the power absorbed by a resistor connected between the open terminal of the circuit shown in Fig. 5.105 if it has a value of (a)  $1\ \Omega$ ; (b)  $100\ \Omega$ ; (c)  $2.65\ k\Omega$ ; (d)  $1.13\ M\Omega$ .

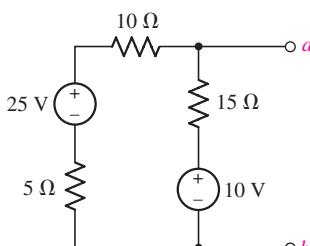


FIGURE 5.106

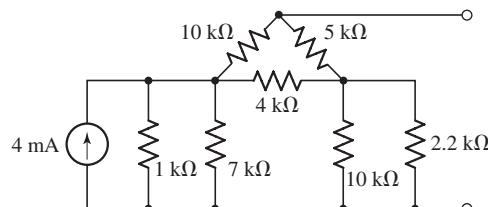


FIGURE 5.105

67. It is known that a load resistor of some type will be connected between terminals *a* and *b* of the network of Fig. 5.106. (a) Change the value of the 25 V source such that both voltage sources contribute equally to the power delivered to the load resistor, assuming its value is chosen such that it absorbs maximum power. (b) Calculate the value of the load resistor.

68. A  $2.57 \Omega$  load is connected between terminals *a* and *b* of the network drawn in Fig. 5.106. Unfortunately, the power delivered to the load is only 50% of the required amount. Altering only voltage sources, modify the circuit so that the required power is delivered and both sources contribute equally.
69. A load resistor is connected across the open terminals of the circuit shown in Fig. 5.107, and its value was chosen carefully to ensure maximum power transfer from the rest of the circuit. (a) What is the value of the resistor? (b) If the power absorbed by the load resistor is three times as large as required, modify the circuit so that it performs as desired, without losing the maximum power transfer condition already enjoyed.

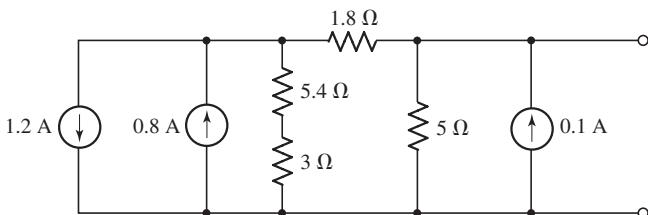


FIGURE 5.107

70. A backup is required for the circuit depicted in Fig. 5.107. It is unknown what will be connected to the open terminals, or whether it will be purely linear. If a simple battery is to be used, what no-load ("open circuit") voltage should it have, and what is the maximum tolerable internal resistance?

### Chapter-Integrating Exercises

71. Three 45 W light bulbs originally wired in a Y network configuration with a 120 V ac source connected across each port are rewired as a  $\Delta$  network. The neutral, or center, connection is not used. If the intensity of each light is proportional to the power it draws, design a new 120 V ac power circuit so that the three lights have the same intensity in the  $\Delta$  configuration as they did when connected in a Y configuration. Verify your design using PSpice by comparing the power drawn by each light in your circuit (modeled as an appropriately chosen resistor value) with the power each would draw in the original Y-connected circuit.
72. (a) Explain in general terms how source transformation can be used to simplify a circuit prior to analysis. (b) Even if source transformations can greatly simplify a particular circuit, when might it not be worth the effort? (c) Multiplying all the independent sources in a circuit by the same scaling factor results in all other voltages and currents being scaled by the same amount. Explain why we don't scale the dependent sources as well. (d) In a general circuit, if we set an independent voltage source to zero, what current can flow through it? (e) In a general circuit, if we set an independent current source to zero, what voltage can be sustained across its terminals?
73. The load resistor in Fig. 5.108 can safely dissipate up to 1 W before overheating and bursting into flame. The lamp can be treated as a  $10.6 \Omega$  resistor if less than 1 A flows through it and a  $15 \Omega$  resistor if more than 1 A flows through it. What is the maximum permissible value of  $I_s$ ? Verify your answer with PSpice.

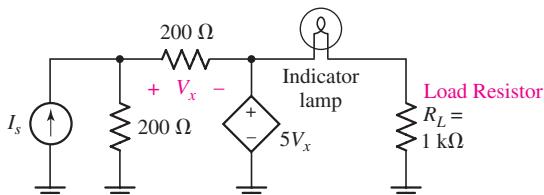


FIGURE 5.108



74. A certain red LED has a maximum current rating of 35 mA, and if this value is exceeded, overheating and catastrophic failure will result. The resistance of the LED is a nonlinear function of its current, but the manufacturer warrants a minimum resistance of  $47\ \Omega$  and a maximum resistance of  $117\ \Omega$ . Only 9 V batteries are available to power the LED. Design a suitable circuit to deliver the maximum power possible to the LED without damaging it. Use only combinations of the standard resistor values given in the inside front cover.



75. As part of a security system, a very thin  $100\ \Omega$  wire is attached to a window using nonconducting epoxy. Given only a box of 12 rechargeable 1.5 V AAA batteries, one thousand  $1\ \Omega$  resistors, and a 2900 Hz piezo buzzer that draws 15 mA at 6 V, design a circuit with no moving parts that will set off the buzzer if the window is broken (and hence the thin wire as well). Note that the buzzer requires a dc voltage of at least 6 V (maximum 28 V) to operate.

# The Operational Amplifier

## INTRODUCTION

At this point we have a good set of circuit analysis tools at our disposal, but have focused primarily on somewhat general circuits composed of only sources and resistors. In this chapter, we introduce a new component which, although technically nonlinear, can be treated effectively with linear models. This element, known as the *operational amplifier* or *op amp* for short, finds daily usage in a large variety of electronic applications. It also provides us a new element to use in building circuits, and another opportunity to test out our developing analytical skills.

## 6.1 BACKGROUND

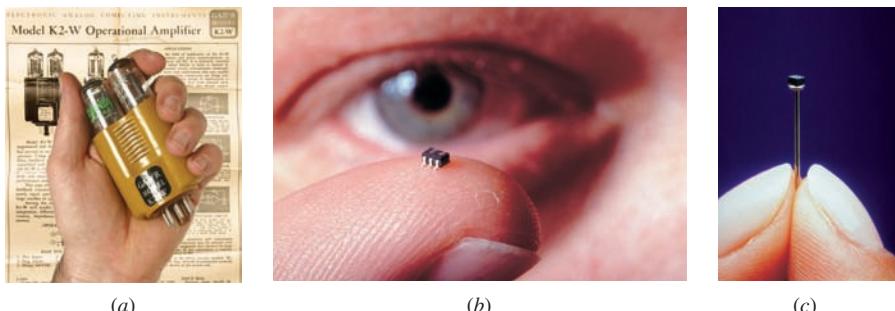
The origins of the operational amplifier date to the 1940s, when basic circuits were constructed using vacuum tubes to perform mathematical operations such as addition, subtraction, multiplication, division, differentiation, and integration. This enabled the construction of analog (as opposed to digital) computers tasked with the solution of complex differential equations. The first commercially available op amp *device* is generally considered to be the K2-W, manufactured by Philbrick Researches, Inc. of Boston from about 1952 through the early 1970s (Fig. 6.1a). These early vacuum tube devices weighed 3 oz (85 g), measured  $1\frac{3}{4}$  in  $\times$   $2\frac{1}{4}$  in  $\times$   $4\frac{1}{4}$  in (3.8 cm  $\times$  5.4 cm  $\times$  10.4 cm), and sold for about US\$22. In contrast, integrated circuit (IC) op amps such as the Fairchild KA741 weigh less than 500 mg, measure 5.7 mm  $\times$  4.9 mm  $\times$  1.8 mm, and sell for approximately US\$0.22.

Compared to op amps based on vacuum tubes, modern IC op amps are constructed using perhaps 25 or more transistors all on the same silicon “chip,” as well as resistors and capacitors needed to obtain the desired performance characteristics. As a result, they run at

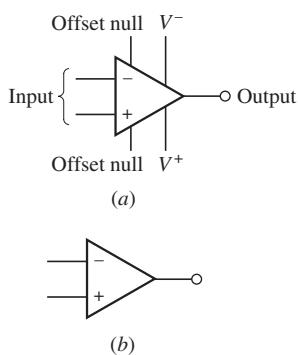
## KEY CONCEPTS

- Characteristics of Ideal Op Amps
- Inverting and Noninverting Amplifiers
- Summing and Difference Amplifier Circuits
- Cascaded Op Amp Stages
- Using Op Amps to Build Voltage and Current Sources
- Nonideal Characteristics of Op Amps
- Voltage Gain and Feedback
- Basic Comparator and Instrumentation Amplifier Circuits





**FIGURE 6.1** (a) A Philbrick K2-W op amp, based on a matched pair of 12AX7A vacuum tubes. (b) LMV321 op amp, used in a variety of phone and game applications. (c) LMC6035 operational amplifier, which packs 114 transistors into a package so small that it fits on the head of a pin.  
(b-c) Copyright © 2011 National Semiconductor Corporation ([www.national.com](http://www.national.com)). All rights reserved. Used with permission.



**FIGURE 6.2** (a) Electrical symbol for the op amp.  
(b) Minimum required connections to be shown on a circuit schematic.

much lower dc supply voltages ( $\pm 18$  V, for example, as opposed to  $\pm 300$  V for the K2-W), are more reliable, and considerably smaller (Fig. 6.1b,c). In some cases, the IC may contain several op amps. In addition to the output pin and the two inputs, other pins enable power to be supplied to run the transistors, and for external adjustments to be made to balance and compensate the op amp. The symbol commonly used for an op amp is shown in Fig. 6.2a. At this point, we are not concerned with the internal circuitry of the op amp or the IC, but only with the voltage and current relationships that exist between the input and output terminals. Thus, for the time being we will use a simpler electrical symbol, shown in Fig. 6.2b. Two input terminals are shown on the left, and a single output terminal appears at the right. The terminal marked by a “+” is referred to as the **noninverting input**, and the “-” marked terminal is called the **inverting input**.

## 6.2 THE IDEAL OP AMP: A CORDIAL INTRODUCTION

In practice, we find that most op amps perform so well that we can often make the assumption that we are dealing with an “ideal” op amp. The characteristics of an **ideal op amp** form the basis for two fundamental rules that at first may seem somewhat unusual:

### Ideal Op Amp Rules

1. No current ever flows into either input terminal.
2. There is no voltage difference between the two input terminals.

In a real op amp, a very small leakage current will flow into the input (sometimes as low as 40 femtoamperes). It is also possible to obtain a very small voltage across the two input terminals. However, compared to other voltages and currents in most circuits, such values are so small that including them in the analysis does not typically affect our calculations.

When analyzing op amp circuits, we should keep one other point in mind. As opposed to the circuits that we have studied so far, an op amp circuit always has an *output* that depends on some type of *input*. Therefore, we will analyze op amp circuits with the goal of obtaining an expression for the output in terms of the input quantities. *We will find that it is usually a good idea to begin the analysis of an op amp circuit at the input, and proceed from there.*



The circuit shown in Fig. 6.3 is known as an ***inverting amplifier***. We choose to analyze this circuit using KVL, beginning with the input voltage source. The current labeled  $i$  flows only through the two resistors  $R_1$  and  $R_f$ ; ideal op amp rule 1 states that no current flows into the inverting input terminal. Thus, we can write

$$-v_{\text{in}} + R_1 i + R_f i + v_{\text{out}} = 0$$

which can be rearranged to obtain an equation that relates the output to the input:

$$v_{\text{out}} = v_{\text{in}} - (R_1 + R_f) i \quad [1]$$

Given  $v_{\text{in}} = 5 \sin 3t$  mV,  $R_1 = 4.7$  k $\Omega$ , and  $R_f = 47$  k $\Omega$ , we require one additional equation that expresses  $i$  only in terms of  $v_{\text{out}}$ ,  $v_{\text{in}}$ ,  $R_1$ , and/or  $R_f$ .

This is a good time to mention that we have not yet made use of ideal op amp rule 2. Since the noninverting input is grounded, it is at zero volts. By ideal op amp rule 2, the inverting input is therefore also at zero volts! *This does not mean that the two inputs are physically shorted together, and we should be careful not to make such an assumption.* Rather, the two input voltages simply track each other: if we try to change the voltage at one pin, the other pin will be driven by internal circuitry to the same value. Thus, we can write one more KVL equation:

$$-v_{\text{in}} + R_1 i + 0 = 0$$

or

$$i = \frac{v_{\text{in}}}{R_1} \quad [2]$$

Combining Eq. [2] with Eq. [1], we obtain an expression for  $v_{\text{out}}$  in terms of  $v_{\text{in}}$ :

$$v_{\text{out}} = -\frac{R_f}{R_1} v_{\text{in}} \quad [3]$$

Substituting  $v_{\text{in}} = 5 \sin 3t$  mV,  $R_1 = 4.7$  k $\Omega$ , and  $R_f = 47$  k $\Omega$ ,

$$v_{\text{out}} = -50 \sin 3t \quad \text{mV}$$

Since  $R_f > R_1$ , this circuit amplifies the input voltage signal  $v_{\text{in}}$ . If we choose  $R_f < R_1$ , the signal will be attenuated instead. We also note that the output voltage has the opposite sign of the input voltage,<sup>1</sup> hence the name “inverting amplifier.” The output is sketched in Fig. 6.4, along with the input waveform for comparison.

At this point, it is worth mentioning that the ideal op amp seems to be violating KCL. Specifically, in the above circuit no current flows into or out of either input terminal, but somehow current is able to flow into the output pin! This would imply that the op amp is somehow able to either create electrons out of nowhere or store them forever (depending on the direction of current flow). Obviously, this is not possible. The conflict arises because we have been treating the op amp the same way we treated passive elements

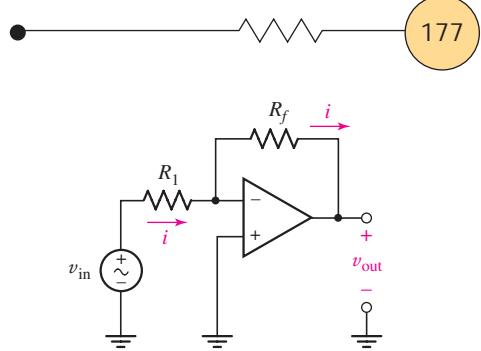


FIGURE 6.3 An op amp used to construct an inverting amplifier circuit. The current  $i$  flows to ground through the output pin of the op amp.



The fact that the inverting input terminal finds itself at zero volts in this type of circuit configuration leads to what is often referred to as a “virtual ground.” This does not mean that the pin is actually grounded, which is sometimes a source of confusion for students. The op amp makes whatever internal adjustments are necessary to prevent a voltage difference between the input terminals. The input terminals are not shorted together.

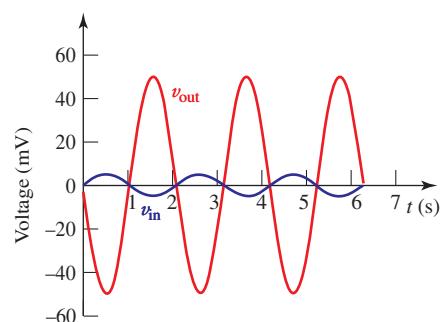


FIGURE 6.4 Input and output waveforms of the inverting amplifier circuit.

(1) Or, “the output is  $180^\circ$  out of phase with the input,” which sounds more impressive.

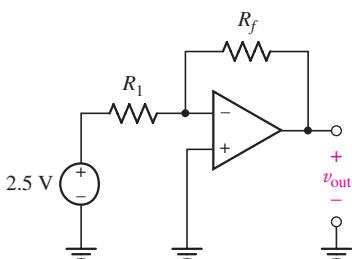


FIGURE 6.5 An inverting amplifier circuit with a 2.5 V input.

such as the resistor. In reality, however, the op amp cannot function unless it is connected to external power sources. It is through those power sources that we can direct current flow through the output terminal.

Although we have shown that the inverting amplifier circuit of Fig. 6.3 can amplify an ac signal (a sine wave in this case having a frequency of 3 rad/s and an amplitude of 5 mV), it works just as well with dc inputs. We consider this type of situation in Fig. 6.5, where values for  $R_1$  and  $R_f$  are to be selected to obtain an output voltage of  $-10$  V.

This is the same circuit as shown in Fig. 6.3, but with a 2.5 V dc input. Since no other change has been made, the expression we presented as Eq. [3] is valid for this circuit as well. To obtain the desired output, we seek a ratio of  $R_f$  to  $R_1$  of  $10/2.5$ , or 4. Since it is only the ratio that is important here, we simply need to pick a convenient value for one resistor, and the other resistor value is then fixed at the same time. For example, we could choose  $R_1 = 100 \Omega$  (so  $R_f = 400 \Omega$ ), or even  $R_f = 8 M\Omega$  (so  $R_1 = 2 M\Omega$ ). In practice, other constraints (such as bias current) may limit our choices.

This circuit configuration therefore acts as a convenient type of voltage amplifier (or **attenuator**, if the ratio of  $R_f$  to  $R_1$  is less than 1), but does have the sometimes inconvenient property of inverting the sign of the input. There is an alternative, however, which is analyzed just as easily—the non-inverting amplifier shown in Fig. 6.6. We examine such a circuit in the following example.

## EXAMPLE 6.1

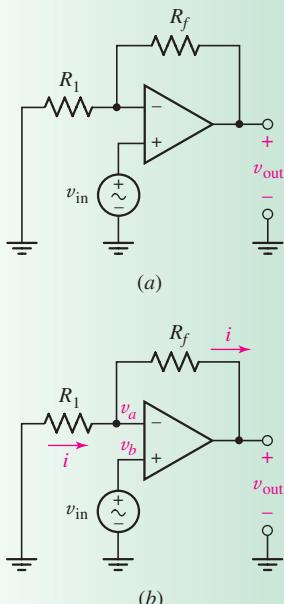


FIGURE 6.6 (a) An op amp used to construct a noninverting amplifier circuit. (b) Circuit with the current through  $R_1$  and  $R_f$  defined, as well as both input voltages labeled.

Sketch the output waveform of the noninverting amplifier circuit in Fig. 6.6a. Use  $v_{in} = 5 \sin 3t$  mV,  $R_1 = 4.7 k\Omega$ , and  $R_f = 47 k\Omega$ .

### ► Identify the goal of the problem.

We require an expression for  $v_{out}$  that only depends on the known quantities  $v_{in}$ ,  $R_1$ , and  $R_f$ .

### ► Collect the known information.

Since values have been specified for the resistors and the input waveform, we begin by labeling the current  $i$  and the two input voltages as shown in Fig. 6.6b. We will assume that the op amp is an ideal op amp.

### ► Devise a plan.

Although mesh analysis is a favorite technique of students, it turns out to be more practical in most op amp circuits to apply nodal analysis, since there is no direct way to determine the current flowing out of the op amp output.

### ► Construct an appropriate set of equations.

Note that we are using ideal op amp rule 1 implicitly by defining the same current through both resistors: no current flows into the inverting input terminal. Employing nodal analysis to obtain our expression for  $v_{out}$  in terms of  $v_{in}$ , we thus find that

At node  $a$ :

$$0 = \frac{v_a}{R_1} + \frac{v_a - v_{\text{out}}}{R_f} \quad [4]$$

At node  $b$ :

$$v_b = v_{\text{in}} \quad [5]$$

#### ► Determine if additional information is required.

Our goal is to obtain a single expression that relates the input and output voltages, although neither Eq. [4] nor Eq. [5] appears to do so. However, we have not yet employed ideal op amp rule 2, and we will find that in almost every op amp circuit *both* rules need to be invoked in order to obtain such an expression.

Thus, we recognize that  $v_a = v_b = v_{\text{in}}$ , and Eq. [4] becomes

$$0 = \frac{v_{\text{in}}}{R_1} + \frac{v_{\text{in}} - v_{\text{out}}}{R_f}$$

#### ► Attempt a solution.

Rearranging, we obtain an expression for the output voltage in terms of the input voltage  $v_{\text{in}}$ :

$$v_{\text{out}} = \left(1 + \frac{R_f}{R_1}\right) v_{\text{in}} = 11 v_{\text{in}} = 55 \sin 3t \quad \text{mV}$$

#### ► Verify the solution. Is it reasonable or expected?

The output waveform is sketched in Fig. 6.7, along with the input waveform for comparison. In contrast to the output waveform of the inverting amplifier circuit, we note that the input and output are in phase for the noninverting amplifier. This should not be entirely unexpected: it is implicit in the name “noninverting amplifier.”

### PRACTICE

6.1 Derive an expression for  $v_{\text{out}}$  in terms of  $v_{\text{in}}$  for the circuit shown in Fig. 6.8.

Ans:  $v_{\text{out}} = v_{\text{in}}$ . The circuit is known as a “*voltage follower*,” since the output voltage tracks or “*follows*” the input voltage.

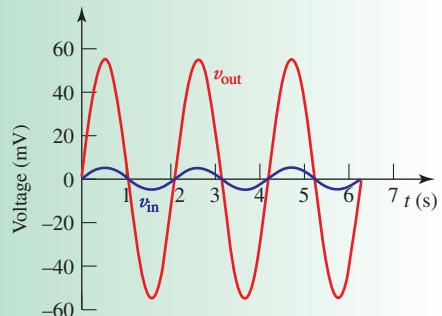


FIGURE 6.7 Input and output waveforms for the noninverting amplifier circuit.

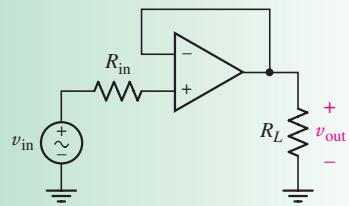


FIGURE 6.8

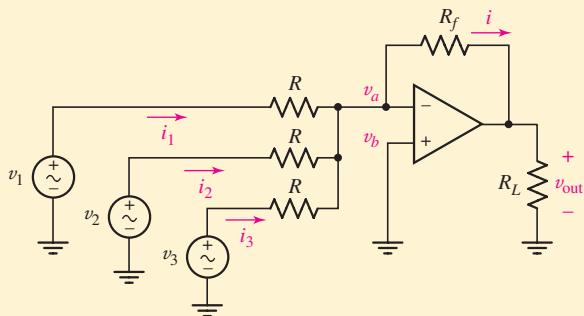
Just like the inverting amplifier, the noninverting amplifier works with dc as well as ac inputs, but has a voltage gain of  $v_{\text{out}}/v_{\text{in}} = 1 + (R_f/R_1)$ . Thus, if we set  $R_f = 9 \Omega$  and  $R_1 = 1 \Omega$ , we obtain an output  $v_{\text{out}}$  which is 10 times larger than the input voltage  $v_{\text{in}}$ . In contrast to the inverting amplifier, the output and input of the noninverting amplifier always have the same sign, and the output voltage cannot be less than the input; the minimum gain is 1. Which amplifier we choose depends on the application we are considering. In the special case of the voltage follower circuit shown in Fig. 6.8,

which represents a noninverting amplifier with  $R_1$  set to  $\infty$  and  $R_f$  set to zero, the output is identical to the input in both sign *and* magnitude. This may seem rather pointless as a general type of circuit, but we should keep in mind that *the voltage follower draws no current from the input* (in the ideal case)—it therefore can act as a **buffer** between the voltage  $v_{in}$  and some resistive load  $R_L$  connected to the output of the op amp.

We mentioned earlier that the name “operational amplifier” originates from using such devices to perform arithmetical operations on analog (i.e., nondigitized, real-time, real-world) signals. As we see in the following two circuits, this includes both addition and subtraction of input voltage signals.

### EXAMPLE 6.2

Obtain an expression for  $v_{out}$  in terms of  $v_1$ ,  $v_2$ , and  $v_3$  for the op amp circuit in Fig. 6.9, also known as a *summing amplifier*.



■ FIGURE 6.9 Basic summing amplifier circuit with three inputs.

We first note that this circuit is similar to the inverting amplifier circuit of Fig. 6.3. Again, the goal is to obtain an expression for  $v_{out}$  (which in this case appears across a load resistor  $R_L$ ) in terms of the inputs ( $v_1$ ,  $v_2$ , and  $v_3$ ).

Since no current can flow into the inverting input terminal, we can write

$$i = i_1 + i_2 + i_3$$

Therefore, we can write the following equation at the node labeled  $v_a$ :

$$0 = \frac{v_a - v_{out}}{R_f} + \frac{v_a - v_1}{R} + \frac{v_a - v_2}{R} + \frac{v_a - v_3}{R}$$

This equation contains both  $v_{out}$  and the input voltages, but unfortunately it also contains the nodal voltage  $v_a$ . To remove this unknown quantity from our expression, we need to write an additional equation that relates  $v_a$  to  $v_{out}$ , the input voltages,  $R_f$ , and/or  $R$ . At this point, we remember that we have not yet used ideal op amp rule 2, and that we will almost certainly require the use of both rules when analyzing an op amp circuit. Thus, since  $v_a = v_b = 0$ , we can write the following:

$$0 = \frac{v_{out}}{R_f} + \frac{v_1}{R} + \frac{v_2}{R} + \frac{v_3}{R}$$

Rearranging, we obtain the following expression for  $v_{\text{out}}$ :

$$v_{\text{out}} = -\frac{R_f}{R}(v_1 + v_2 + v_3) \quad [6]$$

In the special case where  $v_2 = v_3 = 0$ , we see that our result agrees with Eq. [3], which was derived for essentially the same circuit.

There are several interesting features about the result we have just derived. First, if we select  $R_f$  so that it is equal to  $R$ , then the output is the (negative of the) sum of the three input signals  $v_1$ ,  $v_2$ , and  $v_3$ . Further, we can select the ratio of  $R_f$  to  $R$  to multiply this sum by a fixed constant. So, for example, if the three voltages represented signals from three separate scales calibrated so that  $-1 \text{ V} = 1 \text{ lb}$ , we could set  $R_f = R/2.205$  to obtain a voltage signal that represented the combined weight in kilograms (to within about 1 percent accuracy due to our conversion factor).

Also, we notice that  $R_L$  did not appear in our final expression. As long as its value is not too low, the operation of the circuit will not be affected; at present, we have not considered a detailed enough model of an op amp to predict such an occurrence. This resistor represents the Thévenin equivalent of whatever we use to monitor the amplifier output. If our output device is a simple voltmeter, then  $R_L$  represents the Thévenin equivalent resistance seen looking into the voltmeter terminals (typically  $10 \text{ M}\Omega$  or more). Or, our output device might be a speaker (typically  $8 \Omega$ ), in which case we hear the sum of the three separate sources of sound;  $v_1$ ,  $v_2$ , and  $v_3$  might represent microphones in that case.

One word of caution: It is frequently tempting to assume that the current labeled  $i$  in Fig. 6.9 flows not only through  $R_f$  but through  $R_L$  also. Not true! It is very possible that current is flowing through the output terminal of the op amp as well, so that *the currents through the two resistors are not the same*. It is for this reason that we almost universally avoid writing KCL equations at the output pin of an op amp, which leads to the preference of nodal over mesh analysis when working with most op amp circuits.

For convenience, we summarize the most common op amp circuits in Table 1.



### PRACTICE

6.2 Derive an expression for  $v_{\text{out}}$  in terms of  $v_1$  and  $v_2$  for the circuit shown in Fig. 6.10, also known as a *difference amplifier*.

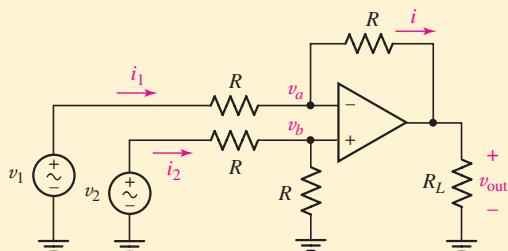


FIGURE 6.10

Ans:  $v_{\text{out}} = v_2 - v_1$ . Hint: Use voltage division to obtain  $v_b$ .

TABLE 6.1 Summary of Basic Op Amp Circuits

Name	Circuit Schematic	Input-Output Relation
Inverting Amplifier		$v_{\text{out}} = -\frac{R_f}{R_1} v_{\text{in}}$
Noninverting Amplifier		$v_{\text{out}} = \left(1 + \frac{R_f}{R_1}\right) v_{\text{in}}$
Voltage Follower (also known as a Unity Gain Amplifier)		$v_{\text{out}} = v_{\text{in}}$
Summing Amplifier		$v_{\text{out}} = -\frac{R_f}{R}(v_1 + v_2 + v_3)$
Difference Amplifier		$v_{\text{out}} = v_2 - v_1$

# PRACTICAL APPLICATION

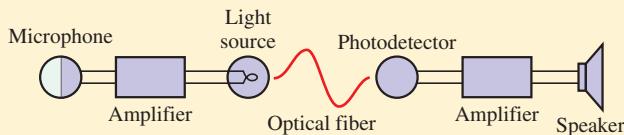
## A Fiber Optic Intercom

A point-to-point intercom system can be constructed using a number of different approaches, depending on the intended application environment. Low-power radio frequency (RF) systems work very well and are generally cost-effective, but are subject to interference from other RF sources and are also prone to eavesdropping. Use of a simple wire to connect the two intercom systems instead can eliminate a great deal of the RF interference as well as increase privacy. However, wires are subject to corrosion and short circuits when the plastic insulation wears, and their weight can be a concern in aircraft and related applications (Fig. 6.11).



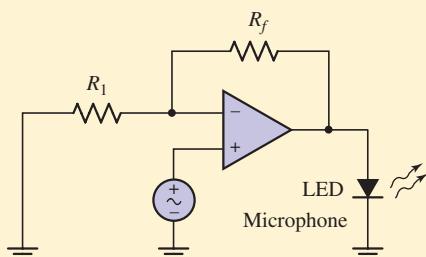
■ FIGURE 6.11 The application environment often dictates design constraints. (© Michael Melford/Riser/Getty Images.)

An alternative design would be to convert the electrical signal from the microphone to an optical signal, which could then be transmitted through a thin ( $\sim 50 \mu\text{m}$  diameter) optical fiber. The optical signal is then converted back to an electrical signal, which is amplified and delivered to a speaker. A schematic diagram of such a system is shown in Fig. 6.12; two such systems would be needed for two-way communication.



■ FIGURE 6.12 Schematic diagram of one-half of a simple fiber optic intercom.

We can consider the design of the transmission and reception circuits separately, since the two circuits are in fact electrically independent. Figure 6.13 shows a simple



■ FIGURE 6.13 Circuit used to convert the electrical microphone signal into an optical signal for transmission through a fiber.

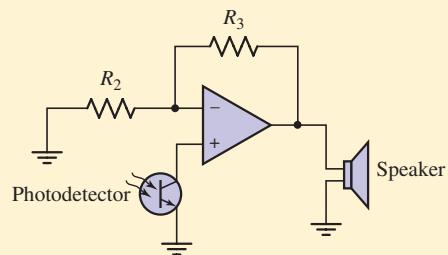
signal generation circuit consisting of a microphone, a light-emitting diode (LED), and an op amp used in a noninverting amplifier circuit to drive the LED; not shown are the power connections required for the op amp itself. The light output of the LED is roughly proportional to its current, although less so for very small and very large values of current.

We know the gain of the amplifier is given by

$$\frac{v_{\text{out}}}{v_{\text{in}}} = 1 + \frac{R_f}{R_1}$$

which is independent of the resistance of the LED. In order to select values for  $R_f$  and  $R_1$ , we need to know the input voltage from the microphone and the necessary output voltage to power the LED. A quick measurement indicates that the typical voltage output of the microphone peaks at 40 mV when someone is using a normal speaking voice. The LED manufacturer recommends operating at approximately 1.6 V, so we design for a gain of  $1.6/0.04 = 40$ . Arbitrarily choosing  $R_1 = 1 \text{ k}\Omega$  leads to a required value of  $39 \text{ k}\Omega$  for  $R_f$ .

The circuit of Fig. 6.14 is the receiver part of our one-way intercom system. It converts the optical signal from the fiber into an electrical signal, amplifying it so that an audible sound emanates from the speaker.



■ FIGURE 6.14 Receiver circuit used to convert the optical signal into an audio signal.

(Continued on next page)

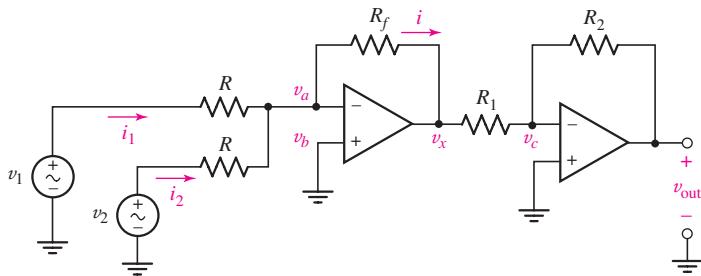
After coupling the LED output of the transmitting circuit to the optical fiber, a signal of approximately 10 mV is measured from the photodetector. The speaker is rated for a maximum of 100 mW and has an equivalent resistance of  $8 \Omega$ . This equates to a maximum speaker voltage of 894 mV, so we need to select values of  $R_2$  and  $R_3$  to obtain a gain of  $894/10 = 89.4$ . With the arbitrary

selection of  $R_2 = 10 \text{ k}\Omega$ , we find that a value of  $884 \text{ k}\Omega$  completes our design.

This circuit will work in practice, although the nonlinear characteristics of the LED lead to a noticeable distortion of the audio signal. We leave improved designs for more advanced texts.

### 6.3 CASCADED STAGES

Although the op amp is an extremely versatile device, there are numerous applications in which a single op amp will not suffice. In such instances, it is often possible to meet application requirements by cascading several individual op amps together in the same circuit. An example of this is shown in Fig. 6.15, which consists of the summing amplifier circuit of Fig. 6.9 with only two input sources, and the output fed into a simple inverting amplifier. The result is a two-stage op amp circuit.



■ FIGURE 6.15 A two-stage op amp circuit consisting of a summing amplifier cascaded with an inverting amplifier circuit.

We have already analyzed each of these op amp circuits separately. Based on our previous experience, if the two op amp circuits were disconnected, we would expect

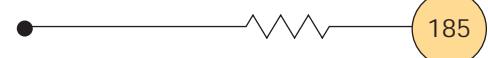
$$v_x = -\frac{R_f}{R}(v_1 + v_2) \quad [7]$$

and

$$v_{\text{out}} = -\frac{R_2}{R_1}v_x \quad [8]$$

In fact, since the two circuits are connected at a single point and the voltage  $v_x$  is not influenced by the connection, we can combine Eqs. [7] and [8] to obtain

$$v_{\text{out}} = \frac{R_2}{R_1} \frac{R_f}{R} (v_1 + v_2) \quad [9]$$



which describes the input-output characteristics of the circuit shown in Fig. 6.15. We may not always be able to reduce such a circuit to familiar stages, however, so it is worth seeing how the two-stage circuit of Fig. 6.15 can be analyzed as a whole.

When analyzing cascaded circuits, it is sometimes helpful to begin with the last stage and work backward toward the input stage. Referring to ideal op amp rule 1, the same current flows through  $R_1$  and  $R_2$ . Writing the appropriate nodal equation at the node labeled  $v_c$  yields

$$0 = \frac{v_c - v_x}{R_1} + \frac{v_c - v_{\text{out}}}{R_2} \quad [10]$$

Applying ideal op amp rule 2, we can set  $v_c = 0$  in Eq. [10], resulting in

$$0 = \frac{v_x}{R_1} + \frac{v_{\text{out}}}{R_2} \quad [11]$$

Since our goal is an expression for  $v_{\text{out}}$  in terms of  $v_1$  and  $v_2$ , we proceed to the first op amp in order to obtain an expression for  $v_x$  in terms of the two input quantities.

Applying ideal op amp rule 1 at the inverting input of the first op amp,

$$0 = \frac{v_a - v_x}{R_f} + \frac{v_a - v_1}{R} + \frac{v_a - v_2}{R} \quad [12]$$

Ideal op amp rule 2 allows us to replace  $v_a$  in Eq. [12] with zero, since  $v_a = v_b = 0$ . Thus, Eq. [12] becomes

$$0 = \frac{v_x}{R_f} + \frac{v_1}{R} + \frac{v_2}{R} \quad [13]$$

We now have an equation for  $v_{\text{out}}$  in terms of  $v_x$  (Eq. [11]), and an equation for  $v_x$  in terms of  $v_1$  and  $v_2$  (Eq. [13]). These equations are identical to Eqs. [7] and [8], respectively, which means that cascading the two separate circuits as in Fig. 6.15 did not affect the input-output relationship of either stage. Combining Eqs. [11] and [13], we find that the input-output relationship for the cascaded op amp circuit is

$$v_{\text{out}} = \frac{R_2}{R_1} \frac{R_f}{R} (v_1 + v_2) \quad [14]$$

which is identical to Eq. [9].

Thus, the cascaded circuit acts as a summing amplifier, but without a phase reversal between the input and output. By choosing the resistor values carefully, we can either amplify or attenuate the sum of the two input voltages. If we select  $R_2 = R_1$  and  $R_f = R$ , we can also obtain an amplifier circuit where  $v_{\text{out}} = v_1 + v_2$ , if desired.

## EXAMPLE 6.3

A multiple-tank gas propellant fuel system is installed in a small lunar orbit runabout. The amount of fuel in any tank is monitored by measuring the tank pressure (in psia).<sup>2</sup> Technical details for tank capacity as well as sensor pressure and voltage range are given in Table 6.2. Design a circuit which provides a positive dc voltage signal proportional to the total fuel remaining, such that 1 V = 100 percent.



© Corbis

**TABLE 6.2 Technical Data for Tank Pressure Monitoring System**

Tank 1 Capacity	10,000 psia
Tank 2 Capacity	10,000 psia
Tank 3 Capacity	2000 psia
Sensor Pressure Range	0 to 12,500 psia
Sensor Voltage Output	0 to 5 Vdc

We see from Table 6.2 that the system has three separate gas tanks, requiring three separate sensors. Each sensor is rated up to 12,500 psia, with a corresponding output of 5 V. Thus, when tank 1 is full, its sensor will provide a voltage signal of  $5 \times (10,000/12,500) = 4$  V; the same is true for the sensor monitoring tank 2. The sensor connected to tank 3, however, will only provide a maximum voltage signal of  $5 \times (2000/12,500) = 800$  mV.

One possible solution is the circuit shown in Fig. 6.16a, which employs a summing amplifier stage with  $v_1$ ,  $v_2$ , and  $v_3$  representing the three sensor outputs, followed by an inverting amplifier to adjust the voltage sign and magnitude. Since we are not told the output resistance of the sensor, we employ a buffer for each one as shown in Fig. 6.16b; the result is (in the ideal case) no current flow from the sensor.

To keep the design as simple as possible, we begin by choosing  $R_1$ ,  $R_2$ ,  $R_3$ , and  $R_4$  to be 1 k $\Omega$ ; any value will do as long as all four resistors are equal. Thus, the output of the summing stage is

$$v_x = -(v_1 + v_2 + v_3)$$

The final stage must invert this voltage and scale it such that the output voltage is 1 V when all three tanks are full. The full condition results in  $v_x = -(4 + 4 + 0.8) = -8.8$  V. Thus, the final stage needs a voltage ratio of  $R_6/R_5 = 1/8.8$ . Arbitrarily choosing  $R_6 = 1$  k $\Omega$ , we find that a value of 8.8 k $\Omega$  for  $R_5$  completes the design.

(2) Pounds per square inch, absolute. This is a differential pressure measurement relative to a vacuum reference.

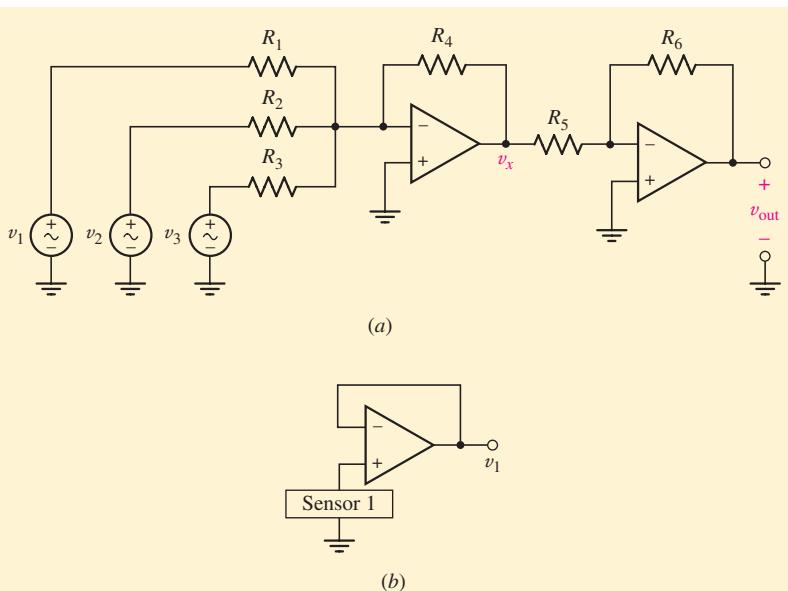


FIGURE 6.16 (a) A proposed circuit to provide a total fuel remaining readout. (b) Buffer design to avoid errors associated with the internal resistance of the sensor and limitations on its ability to provide current. One such buffer is used for each sensor, providing the inputs  $v_1$ ,  $v_2$ , and  $v_3$  to the summing amplifier stage.

### PRACTICE

6.3 An historic bridge is showing signs of deterioration. Until renovations can be performed, it is decided that only cars weighing less than 1600 kg will be allowed across. To monitor this, a four-pad weighing system is designed. There are four independent voltage signals, one from each wheel pad, with  $1 \text{ mV} = 1 \text{ kg}$ . Design a circuit to provide a positive voltage signal to be displayed on a DMM (digital multimeter) that represents the total weight of a vehicle, such that  $1 \text{ mV} = 1 \text{ kg}$ . You may assume there is no need to buffer the wheel pad voltage signals.

Ans: See Fig. 6.17.

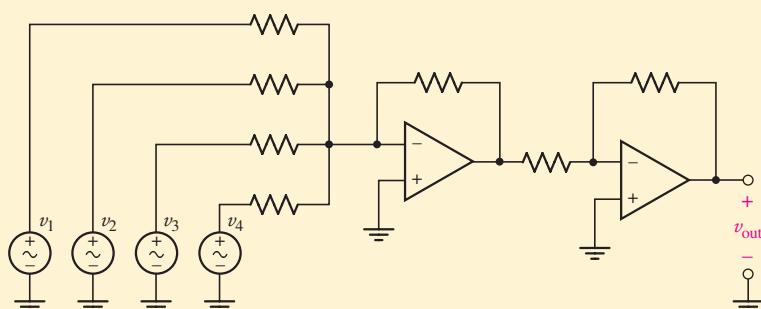


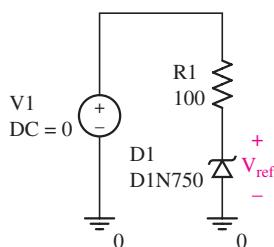
FIGURE 6.17 One possible solution to Practice Problem 6.3; all resistors are  $10 \text{ k}\Omega$  (although any value will do as long as they are all equal). Input voltages  $v_1$ ,  $v_2$ ,  $v_3$ , and  $v_4$  represent the voltage signals from the four wheel pad sensors, and  $v_{out}$  is the output signal to be connected to the positive input terminal of the DMM. All five voltages are referenced to ground, and the common terminal of the DMM should be connected to ground as well.

## 6.4 CIRCUITS FOR VOLTAGE AND CURRENT SOURCES

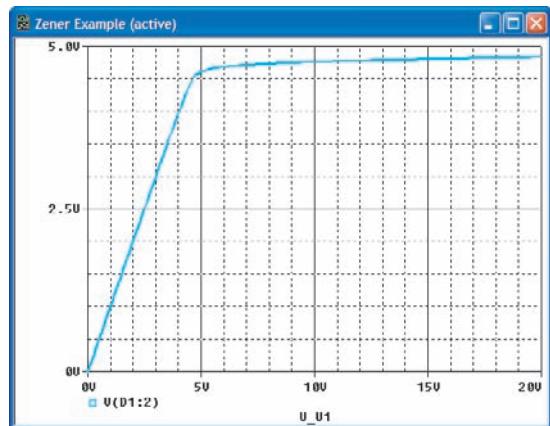
In this and previous chapters we have often made use of ideal current and voltage sources, which we assume provide the same value of current or voltage, respectively, regardless of how they are connected in a circuit. Our assumption of independence has its limits, of course, as mentioned in Sec. 5.2 when we discussed practical sources which included a “built-in” or inherent resistance. The effect of such a resistance was a reduction of the voltage output of a voltage source as more current was demanded, or a diminished current output as more voltage was required from a current source. As discussed in this section, it is possible to construct circuits with more reliable characteristics using op amps.

### A Reliable Voltage Source

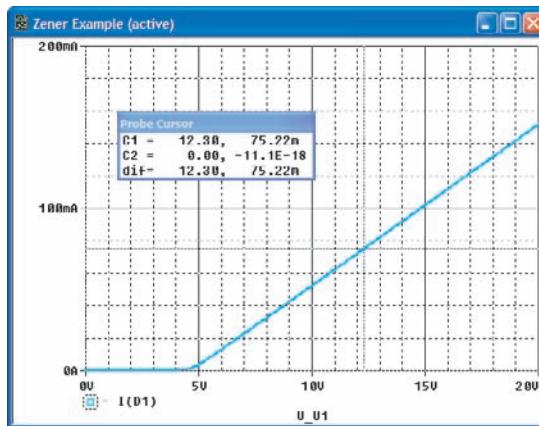
One of the most common means of providing a stable and consistent reference voltage is to make use of a nonlinear device known as a **Zener diode**. Its symbol is a triangle with a Z-like line across the top of the triangle, as shown for a 1N750 in the circuit of Fig. 6.18a. Diodes are characterized by



(a)



(b)



(c)

FIGURE 6.18 (a) PSpice schematic of a simple voltage reference circuit based on the 1N750 Zener diode. (b) Simulation of the circuit showing the diode voltage  $V_{\text{ref}}$  as a function of the driving voltage  $V_1$ . (c) Simulation of the diode current, showing that its maximum rating is exceeded when  $V_1$  exceeds 12.3 V. (Note that performing this calculation assuming an *ideal* Zener diode yields 12.2 V.)

a strongly asymmetric current-voltage relationship. For small voltages, they either conduct essentially zero current—or experience an exponentially increasing current—depending on the voltage polarity. In this way, they distinguish themselves from the simple resistor, where the magnitude of the current is the same for either voltage polarity and hence the resistor current-voltage relationship is symmetric. Consequently, the terminals of a diode are not interchangeable, and have unique names: the **anode** (the flat part of the triangle) and the **cathode** (the point of the triangle).

A Zener diode is a special type of diode designed to be used with a positive voltage at the cathode with respect to the anode; when connected this way, the diode is said to be *reverse biased*. For low voltages, the diode acts like a resistor with a small linear increase in current flow as the voltage is increased. Once a certain voltage ( $V_{BR}$ ) is reached, however—known as the *reverse breakdown voltage* or **Zener voltage** of the diode—the voltage does not significantly increase further, but essentially any current can flow up to the maximum rating of the diode (75 mA for a 1N750, whose Zener voltage is 4.7 V).

Let's consider the simulation result presented in Fig. 6.18b, which shows the voltage  $V_{ref}$  across the diode as the voltage source  $V_1$  is swept from 0 to 20 V. Provided  $V_1$  remains above 5 V, *the voltage across our diode is essentially constant*. Thus, we could replace  $V_1$  with a 9 V battery, and not be too concerned with changes in our voltage reference as the battery voltage begins to drop as it discharges. The purpose of  $R_1$  in this circuit is simply to provide the necessary voltage drop between the battery and the diode; its value should be chosen to ensure that the diode is operating at its Zener voltage but below its maximum rated current. For example, Fig. 6.18c shows that the 75 mA rating is exceeded in our circuit if the source voltage  $V_1$  is much greater than 12 V. Thus, the value of resistor  $R_1$  should be sized corresponding to the source voltage available, as we explore in Example 6.4.

## EXAMPLE 6.4

**Design a circuit based on the 1N750 Zener diode that runs on a single 9 V battery and provides a reference voltage of 4.7 V.**

The 1N750 has a maximum current rating of 75 mA, and a Zener voltage of 4.7 V. The voltage of a 9 V battery can vary slightly depending on its state of charge, but we neglect this for the present design.

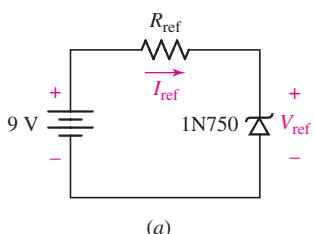
A simple circuit such as the one shown in Fig. 6.19a is adequate for our purposes; the only issue is determining a suitable value for the resistor  $R_{ref}$ .

If 4.7 V is dropped across the diode, then  $9 - 4.7 = 4.3$  V must be dropped across  $R_{ref}$ . Thus,

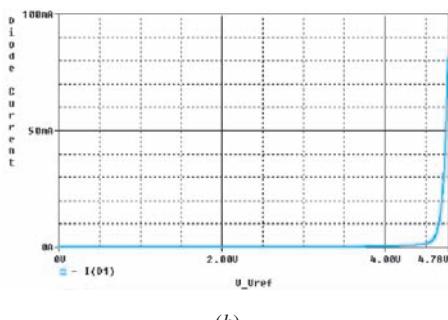
$$R_{ref} = \frac{9 - V_{ref}}{I_{ref}} = \frac{4.3}{I_{ref}}$$

We determine  $R_{ref}$  by specifying a current value. We know that  $I_{ref}$  should not be allowed to exceed 75 mA for this diode, and large currents will discharge the battery more quickly. However, as seen in Fig. 6.19b, we cannot simply select  $I_{ref}$  arbitrarily; very low currents do not allow

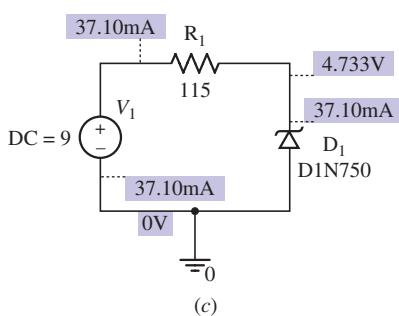
(Continued on next page)



(a)



(b)



(c)

FIGURE 6.19 (a) A voltage reference circuit based on the 1N750 Zener diode. (b) Diode  $I$ - $V$  relationship. (c) PSpice simulation of the final design.

the diode to operate in the Zener breakdown region. In the absence of a detailed equation for the diode's current-voltage relationship (which is clearly nonlinear), we design for 50 percent of the maximum rated current as a rule of thumb. Thus,

$$R_{\text{ref}} = \frac{4.3}{0.0375} = 115 \Omega$$

Detailed “tweaking” can be obtained by performing a PSpice simulation of the final circuit, although we see from Fig. 6.19c that our first pass is reasonably close (within 1 percent) to our target value.

The basic Zener diode voltage reference circuit of Fig. 6.18a works very well in many situations, but we are limited somewhat in the value of the voltage depending on which Zener diodes are available. Also, we often find that the circuit shown is not well suited to applications requiring more than a few milliamperes of current. In such instances, we may use the Zener reference circuit in conjunction with a simple amplifier stage, as shown in Fig. 6.20. The result is a stable voltage that can be controlled by adjusting the value of either  $R_1$  or  $R_f$ , without having to switch to a different Zener diode.

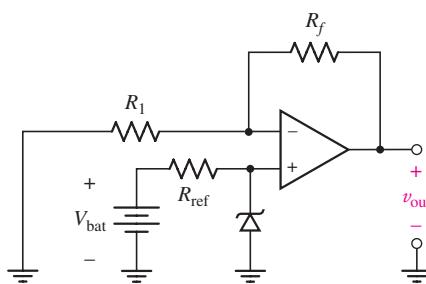


FIGURE 6.20 An op amp-based voltage source using on a Zener voltage reference.

### PRACTICE

6.4 Design a circuit to provide a reference voltage of 6 V using a 1N750 Zener diode and a noninverting amplifier.

Ans: Using the circuit topology shown in Fig. 6.20, choose  $V_{\text{bat}} = 9 \text{ V}$ ,  $R_{\text{ref}} = 115 \Omega$ ,  $R_1 = 1 \text{ k}\Omega$ , and  $R_f = 268 \Omega$ .

## A Reliable Current Source

Consider the circuit shown in Fig. 6.21a, where  $V_{\text{ref}}$  is provided by a regulated voltage source such as the one shown in Fig. 6.19a. The reader may recognize this circuit as a simple inverting amplifier configuration, assuming we tap the output pin of the op amp. We can also use this circuit as a current source, however, where  $R_L$  represents a resistive load.

The input voltage  $V_{\text{ref}}$  appears across reference resistor  $R_{\text{ref}}$ , since the noninverting input of the op amp is connected to ground. With no current

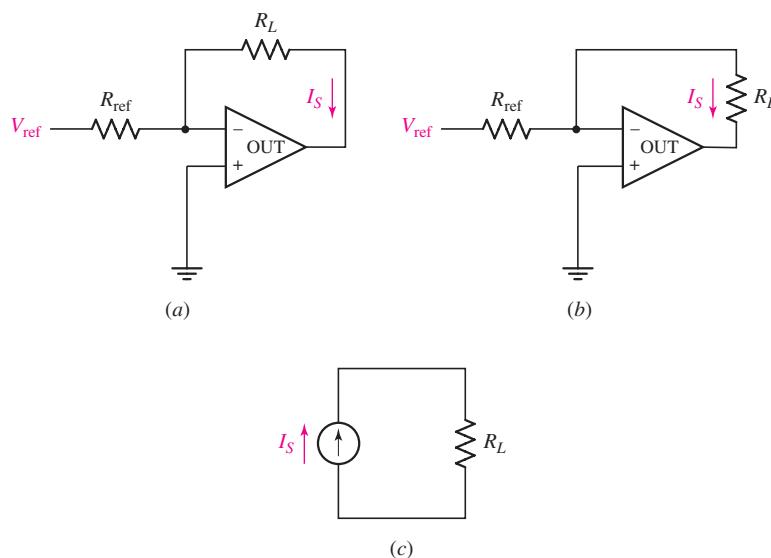


FIGURE 6.21 (a) An op amp-based current source, controlled by the reference voltage  $V_{\text{ref}}$ .  
(b) Circuit redrawn to highlight load. (c) Circuit model. Resistor  $R_L$  represents the Norton equivalent of an unknown passive load circuit.

flowing into the inverting input, the current flowing through the load resistor  $R_L$  is simply

$$I_s = \frac{V_{\text{ref}}}{R_{\text{ref}}}$$

In other words, the current supplied to  $R_L$  does not depend on its resistance—the primary attribute of an ideal current source. It is also worth noting that we are not tapping the output voltage of the op amp here as a quantity of interest. Instead, we may view the load resistor  $R_L$  as the Norton (or Thévenin) equivalent of some unknown passive load circuit, which receives power from the op amp circuit. Redrawing the circuit slightly as in Fig. 6.21b, we see that it has a great deal in common with the more familiar circuit of Fig. 6.21c. In other words, we may use this op amp circuit as an independent current source with essentially ideal characteristics, up to the maximum rated output current of the op amp selected.

### EXAMPLE 6.5

**Design a current source that will deliver 1 mA to an arbitrary resistive load.**

Basing our design on the circuits of Fig. 6.20 and Fig. 6.21a, we know that the current through our load  $R_L$  will be given by

$$I_s = \frac{V_{\text{ref}}}{R_{\text{ref}}}$$

where values for  $V_{\text{ref}}$  and  $R_{\text{ref}}$  must be selected, and a circuit to provide  $V_{\text{ref}}$  must also be designed. If we use a 1N750 Zener diode in series with a 9 V

(Continued on next page)

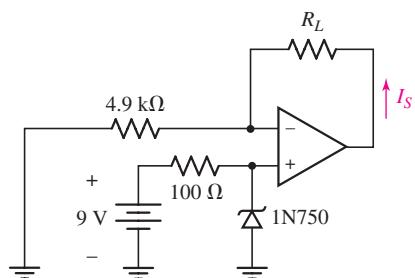


FIGURE 6.22 One possible design for the desired current source. Note the change in current direction from Fig. 6.21b.

battery and a  $100\ \Omega$  resistor, we know from Fig. 6.18b that a voltage of 4.9 V will exist across the diode. Thus,  $V_{ref} = 4.9\text{ V}$ , dictating a value of  $4.9/10^{-3} = 4.9\text{ k}\Omega$  for  $R_{ref}$ . The complete circuit is shown in Fig. 6.22.

Note that if we had assumed a diode voltage of 4.7 V instead, the error in our designed current would only be a few percent, well within the typical 5 to 10 percent tolerance in resistor values we might expect.

The only issue remaining is whether 1 mA can in fact be provided to any value of  $R_L$ . For the case of  $R_L = 0$ , the output of the op amp will be 4.9 V, which is not unreasonable. As the load resistor is increased, however, the op amp output voltage increases. Eventually we must reach some type of limit, as discussed in Sec. 6.5.

### PRACTICE

6.5 Design a current source capable of providing  $500\ \mu\text{A}$  to a resistive load.

Ans: See Fig. 6.23 for one possible solution.

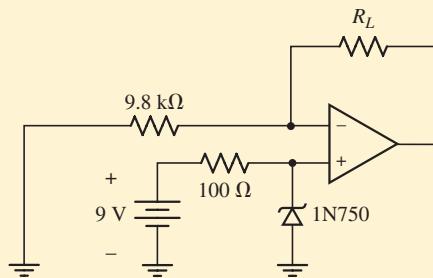


FIGURE 6.23 One possible solution to Practice Problem 6.5.

## 6.5 PRACTICAL CONSIDERATIONS

### A More Detailed Op Amp Model

Reduced to its essentials, the op amp can be thought of as a voltage-controlled dependent voltage source. The dependent voltage source provides the output of the op amp, and the voltage on which it depends is applied to the input terminals. A schematic diagram of a reasonable model for a practical op amp is shown in Fig. 6.24; it includes a dependent voltage source with voltage gain  $A$ , an output resistance  $R_o$ , and an input resistance  $R_i$ . Table 6.3 gives typical values for these parameters for several types of commercially available op amps.

The parameter  $A$  is referred to as the *open-loop voltage gain* of the op amp, and is typically in the range of  $10^5$  to  $10^6$ . We notice that all of the op amps listed in Table 6.3 have extremely large open-loop voltage gain, especially compared to the voltage gain of 11 that characterized the noninverting amplifier circuit of Example 6.1. It is important to remember the distinction

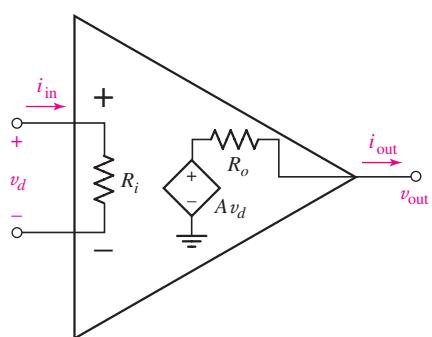


FIGURE 6.24 A more detailed model for the op amp.

**TABLE 6.3 Typical Parameter Values for Several Types of Op Amps**

Part Number	<b><math>\mu A741</math></b>	<b>LM324</b>	<b>LF411</b>	<b>AD549K</b>	<b>OPA690</b>
Description	General purpose	Low-power quad	Low-offset, low-drift JFET input	Ultralow input bias current	Wideband video frequency op amp
Open loop gain $A$	$2 \times 10^5$ V/V	$10^5$ V/V	$2 \times 10^5$ V/V	$10^6$ V/V	2800 V/V
Input resistance	$2 \text{ M}\Omega$	*	$1 \text{ T}\Omega$	$10 \text{ T}\Omega$	$190 \text{ k}\Omega$
Output resistance	$75 \Omega$	*	$\sim 1 \Omega$	$\sim 15 \Omega$	*
Input bias current	80 nA	45 nA	50 pA	75 fA	$3 \mu\text{A}$
Input offset voltage	1.0 mV	2.0 mV	0.8 mV	0.150 mV	$\pm 1.0 \text{ mV}$
CMRR	90 dB	85 dB	100 dB	100 dB	65 dB
Slew rate	$0.5 \text{ V}/\mu\text{s}$	*	$15 \text{ V}/\mu\text{s}$	$3 \text{ V}/\mu\text{s}$	$1800 \text{ V}/\mu\text{s}$
PSpice Model	✓	✓	✓		

\* Not provided by manufacturer.

✓ Indicates that a PSpice model is included in Orcad Capture CIS Lite Edition 16.3.

between the open-loop voltage gain of the op amp itself, and the ***closed-loop voltage gain*** that characterizes a particular op amp circuit. The “loop” in this case refers to an *external* path between the output pin and the inverting input pin; it can be a wire, a resistor, or another type of element, depending on the application.

The  $\mu A741$  is a very common op amp, originally produced by Fairchild Corporation in 1968. It is characterized by an open-loop voltage gain of 200,000, an input resistance of  $2 \text{ M}\Omega$ , and an output resistance of  $75 \Omega$ . In order to evaluate how well the ideal op amp model approximates the behavior of this particular device, let’s revisit the inverting amplifier circuit of Fig. 6.3.

## EXAMPLE 6.6

Using the appropriate values for the  $\mu A741$  op amp in the model of Fig. 6.24, reanalyze the inverting amplifier circuit of Fig. 6.3.

We begin by replacing the ideal op amp symbol of Fig. 6.3 with the detailed model, resulting in the circuit shown in Fig. 6.25.

Note that we can no longer invoke the ideal op amp rules, since we are not using the ideal op amp model. Thus, we write two nodal equations:

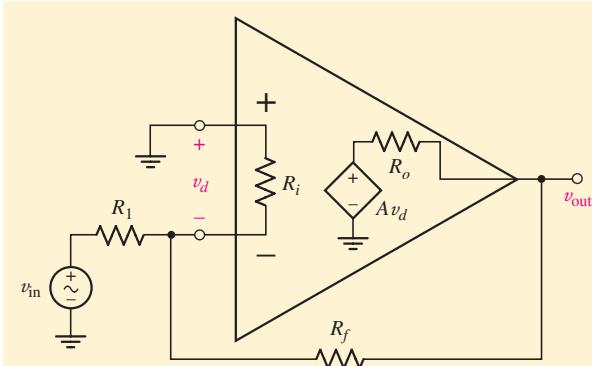
$$0 = \frac{-v_d - v_{in}}{R_1} + \frac{-v_d - v_{out}}{R_f} + \frac{-v_d}{R_i}$$

$$0 = \frac{v_{out} + v_d}{R_f} + \frac{v_{out} - Av_d}{R_o}$$



Performing some straightforward but rather lengthy algebra, we eliminate  $v_d$  and combine these two equations to obtain the following

(Continued on next page)



■ FIGURE 6.25 Inverting amplifier circuit drawn using detailed op amp model.

expression for  $v_{\text{out}}$  in terms of  $v_{\text{in}}$ :

$$v_{\text{out}} = \left[ \frac{R_o + R_f}{R_o - AR_f} \left( \frac{1}{R_1} + \frac{1}{R_f} + \frac{1}{R_i} \right) - \frac{1}{R_f} \right]^{-1} \frac{v_{\text{in}}}{R_1} \quad [15]$$

Substituting  $v_{\text{in}} = 5 \sin 3t \text{ mV}$ ,  $R_1 = 4.7 \text{ k}\Omega$ ,  $R_f = 47 \text{ k}\Omega$ ,  $R_o = 75 \Omega$ ,  $R_i = 2 \text{ M}\Omega$ , and  $A = 2 \times 10^5$ , we obtain

$$v_{\text{out}} = -9.999448v_{\text{in}} = -49.99724 \sin 3t \quad \text{mV}$$

Upon comparing this to the expression found assuming an ideal op amp ( $v_{\text{out}} = -10v_{\text{in}} = -50 \sin 3t \text{ mV}$ ), we see that the ideal op amp is indeed a reasonably accurate model. Further, assuming an ideal op amp leads to a significant reduction in the algebra required to perform the circuit analysis. Note that if we allow  $A \rightarrow \infty$ ,  $R_o \rightarrow 0$ , and  $R_i \rightarrow \infty$ , Eq. [15] reduces to

$$v_{\text{out}} = -\frac{R_f}{R_1}v_{\text{in}}$$

which is what we derived earlier for the inverting amplifier when assuming the op amp was ideal.

### PRACTICE

6.6 Assuming a finite open-loop gain ( $A$ ), a finite input resistance ( $R_i$ ), and zero output resistance ( $R_o$ ), derive an expression for  $v_{\text{out}}$  in terms of  $v_{\text{in}}$  for the op amp circuit of Fig. 6.3.

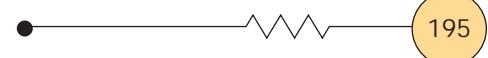
Ans:  $v_{\text{out}}/v_{\text{in}} = -AR_f R_i / [(1 + A)R_1 R_i + R_1 R_f + R_f R_i]$ .

### Derivation of the Ideal Op Amp Rules

We have seen that the ideal op amp can be a reasonably accurate model for the behavior of practical devices. However, using our more detailed model which includes a finite open-loop gain, finite input resistance, and nonzero output resistance, it is actually straightforward to derive the two ideal op amp rules.

Referring to Fig. 6.24, we see that the open circuit output voltage of a practical op amp can be expressed as

$$v_{\text{out}} = Av_d \quad [16]$$



Rearranging this equation, we find that  $v_d$ , sometimes referred to as the **differential input voltage**, can be written as

$$v_d = \frac{v_{\text{out}}}{A} \quad [17]$$

As we might expect, there are practical limits to the output voltage  $v_{\text{out}}$  that can be obtained from a real op amp. As described in the next section, we must connect our op amp to external dc voltage supplies in order to power the internal circuitry. These external voltage supplies represent the maximum value of  $v_{\text{out}}$ , and are typically in the range of 5 to 24 V. If we divide 24 V by the open-loop gain of the  $\mu\text{A741}$  ( $2 \times 10^5$ ), we obtain  $v_d = 120 \mu\text{V}$ . Although this is not the same as zero volts, such a small value compared to the output voltage of 24 V is *practically* zero. An ideal op amp would have infinite open-loop gain, resulting in  $v_d = 0$  regardless of  $v_{\text{out}}$ ; this leads to ideal op amp rule 2.

Ideal op amp rule 1 states that “*No current ever flows into either input terminal.*” Referring to Fig. 6.23, the input current of an op amp is simply

$$i_{\text{in}} = \frac{v_d}{R_i}$$

We have just determined that  $v_d$  is typically a very small voltage. As we can see from Table 6.3, the input resistance of an op amp is very large, ranging from the megaohms to the teraohms! Using the value of  $v_d = 120 \mu\text{V}$  from above and  $R_i = 2 \text{ M}\Omega$ , we compute an input current of 60 pA. This is an extremely small current, and we would require a specialized ammeter (known as a picoammeter) to measure it. We see from Table 6.3 that the typical input current (more accurately termed the **input bias current**) of a  $\mu\text{A741}$  is 80 nA, three orders of magnitude larger than our estimate. This is a shortcoming of the op amp model we are using, which is not designed to provide accurate values for input bias current. Compared to the other currents flowing in a typical op amp circuit, however, either value is essentially zero. More modern op amps (such as the AD549) have even lower input bias currents. Thus, we conclude that ideal op amp rule 1 is a fairly reasonable assumption.

From our discussion, it is clear that an ideal op amp has infinite open-loop voltage gain, and infinite input resistance. However, we have not yet considered the output resistance of the op amp and its possible effects on our circuit. Referring to Fig. 6.24, we see that

$$v_{\text{out}} = A v_d - R_o i_{\text{out}}$$

where  $i_{\text{out}}$  flows from the output pin of the op amp. Thus, a nonzero value of  $R_o$  acts to reduce the output voltage, an effect which becomes more pronounced as the output current increases. For this reason, an *ideal* op amp has an output resistance of zero ohms. The  $\mu\text{A741}$  has a maximum output resistance of  $75 \Omega$ , and more modern devices such as the AD549 have even lower output resistance.

## Common-Mode Rejection

The op amp is occasionally referred to as a *difference amplifier*, since the output is proportional to the voltage difference between the two input

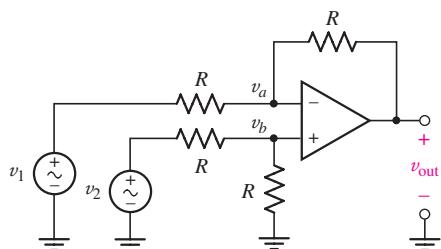


FIGURE 6.26 An op amp connected as a difference amplifier.

terminals. This means that if we apply identical voltages to both input terminals, we expect the output voltage to be zero. This ability of the op amp is one of its most attractive qualities, and is known as **common-mode rejection**. The circuit shown in Fig. 6.26 is connected to provide an output voltage

$$v_{\text{out}} = v_2 - v_1$$

If  $v_1 = 2 + 3 \sin 3t$  volts and  $v_2 = 2$  volts, we would expect the output to be  $-3 \sin 3t$  volts; the 2 V component common to  $v_1$  and  $v_2$  would not be amplified, nor does it appear in the output.

For practical op amps, we do in fact find a small contribution to the output in response to common-mode signals. In order to compare one op amp type to another, it is often helpful to express the ability of an op amp to reject common-mode signals through a parameter known as the common-mode rejection ratio, or **CMRR**. Defining  $v_{\text{OCM}}$  as the output obtained when both inputs are equal ( $v_1 = v_2 = v_{\text{CM}}$ ), we can determine  $A_{\text{CM}}$ , the common-mode gain of the op amp

$$A_{\text{CM}} = \left| \frac{v_{\text{OCM}}}{v_{\text{CM}}} \right|$$

We then define CMRR in terms of the ratio of differential-mode gain  $A$  to the common-mode gain  $A_{\text{CM}}$ , or

$$\text{CMRR} \equiv \left| \frac{A}{A_{\text{CM}}} \right| \quad [18]$$

although this is often expressed in decibels (dB), a logarithmic scale:

$$\text{CMRR}_{(\text{dB})} \equiv 20 \log_{10} \left| \frac{A}{A_{\text{CM}}} \right| \quad \text{dB} \quad [19]$$

Typical values for several different op amps are provided in Table 6.3; a value of 100 dB corresponds to an absolute ratio of  $10^5$  for  $A$  to  $A_{\text{CM}}$ .

## Negative Feedback

We have seen that the open-loop gain of an op amp is very large, ideally infinite. In practical situations, however, its exact value can vary from the value specified by the manufacturer as typical. Temperature, for example, can have a number of significant effects on the performance of an op amp, so that the operating behavior in  $-20^{\circ}\text{C}$  weather may be significantly different from the behavior observed on a warm sunny day. Also, there are typically small variations between devices fabricated at different times. If we design a circuit in which the output voltage is the open-loop gain times the voltage at one of the input terminals, the output voltage could therefore be difficult to predict with a reasonable degree of precision, and might be expected to change depending on the ambient temperature.

A solution to such potential problems is to employ the technique of **negative feedback**, which is the process of subtracting a small portion of the output from the input. If some event changes the characteristics of the amplifier such that the output tries to increase, the input is decreasing at the same time. Too much negative feedback will prevent any useful amplification, but a small amount provides stability. An example of negative

feedback is the unpleasant sensation we feel as our hand draws near a flame. The closer we move toward the flame, the larger the negative signal sent from our hand. Overdoing the proportion of negative feedback, however, might cause us to abhor heat, and eventually freeze to death. **Positive feedback** is the process where some fraction of the output signal is added back to the input. A common example is when a microphone is directed toward a speaker—a very soft sound is rapidly amplified over and over until the system “screams.” Positive feedback generally leads to an unstable system.

All of the circuits considered in this chapter incorporate negative feedback through the presence of a resistor between the output pin and the inverting input. The resulting loop between the output and the input reduces the dependency of the output voltage on the actual value of the open-loop gain (as seen in Example 6.6). This obviates the need to measure the precise open-loop gain of each op amp we use, as small variations in  $A$  will not significantly impact the operation of the circuit. Negative feedback also provides increased stability in situations where  $A$  is sensitive to the op amp’s surroundings. For example, if  $A$  suddenly increases in response to a change in the ambient temperature, a larger feedback voltage is added to the inverting input. This acts to reduce the differential input voltage  $v_d$ , and therefore the change in output voltage  $Av_d$  is smaller. We should note that the closed-loop circuit gain is always less than the open-loop device gain; this is the price we pay for stability and reduced sensitivity to parameter variations.



## Saturation

So far, we have treated the op amp as a purely linear device, assuming that its characteristics are independent of the way in which it is connected in a circuit. In reality, it is necessary to supply power to an op amp in order to run the internal circuitry, as shown in Fig. 6.27. A positive supply, typically in the range of 5 to 24 V dc, is connected to the terminal marked  $V^+$ , and a negative supply of equal magnitude is connected to the terminal marked  $V^-$ . There are also a number of applications where a single voltage supply is acceptable, as well as situations where the two voltage magnitudes may be unequal. The op amp manufacturer will usually specify a maximum power supply voltage, beyond which damage to the internal transistors will occur.

The power supply voltages are a critical choice when designing an op amp circuit, because they represent the maximum possible output voltage of the op amp.<sup>3</sup> For example, consider the op amp circuit shown in Fig. 6.26, now connected as a noninverting amplifier having a gain of 10. As shown in the PSpice simulation in Fig. 6.28, we do in fact observe linear behavior from the op amp, but only in the range of  $\pm 1.71$  V for the input voltage. Outside of this range, the output voltage is no longer proportional to the input, reaching a peak magnitude of 17.6 V. This important nonlinear effect is known as **saturation**, which refers to the fact that further increases in the input voltage do not result in a change in the output voltage. This phenomenon refers to the fact that the output of a real op amp cannot exceed its

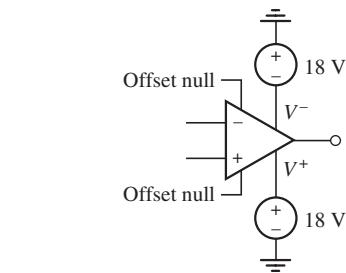
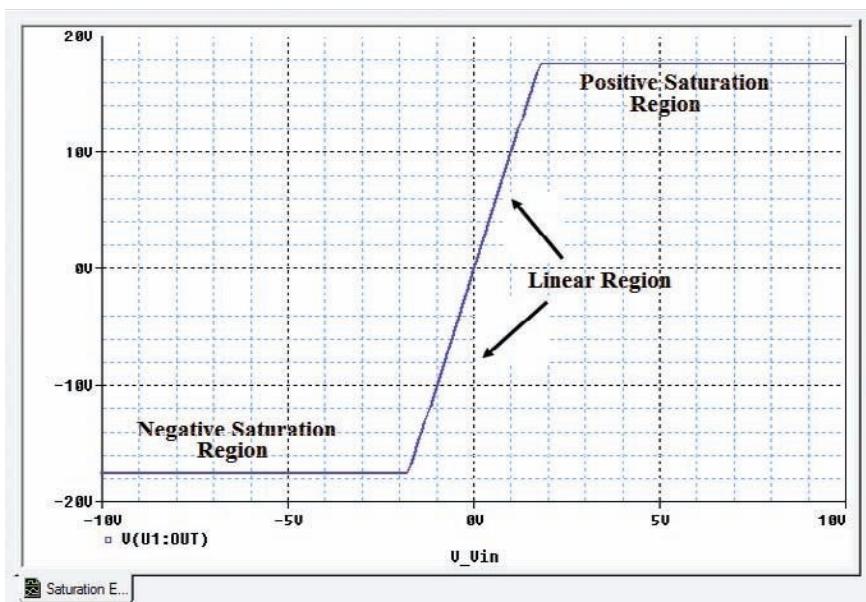


FIGURE 6.27 Op amp with positive and negative voltage supplies connected. Two 18 V supplies are used as an example; note the polarity of each source.

(3) In practice, we find the maximum output voltage is slightly less than the supply voltage by as much as a volt or so.



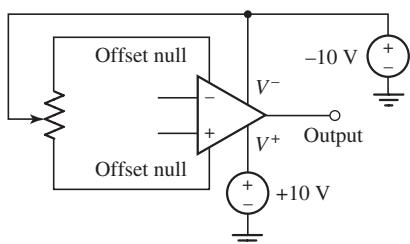
■ FIGURE 6.28 Simulated input-output characteristics of a  $\mu$ A741 connected as a noninverting amplifier with a gain of 10, and powered by  $\pm 18$  V supplies.

supply voltages. For example, if we choose to run the op amp with a +9 V supply and a -5 V supply, then our output voltage will be limited to the range of -5 to +9 V. The output of the op amp is a linear response bounded by the positive and negative saturation regions, and as a general rule, we try to design our op amp circuits so that we do not accidentally enter the saturation region. This requires us to select the operating voltage carefully based on the closed-loop gain and maximum expected input voltage.

## Input Offset Voltage

As we are discovering, there are a number of practical considerations to keep in mind when working with op amps. One particular nonideality worth mentioning is the tendency for real op amps to have a nonzero output even when the two input terminals are shorted together. The value of the output under such conditions is known as the offset voltage, and the input voltage required to reduce the output to zero is referred to as the *input offset voltage*. Referring to Table 6.3, we see that typical values for the input offset voltage are on the order of a few millivolts or less.

Most op amps are provided with two pins marked either “offset null” or “balance.” These terminals can be used to adjust the output voltage by connecting them to a variable resistor. A variable resistor is a three-terminal device commonly used for such applications as volume controls on radios. The device comes with a knob that can be rotated to select the actual value of resistance, and has three terminals. Measured between the two extreme terminals, its resistance is fixed regardless of the position of the knob. Using the middle terminal and one of the end terminals creates a resistor whose value depends on the knob position. Figure 6.29 shows a typical circuit used to adjust the output voltage of an op amp; the manufacturer’s data sheet may suggest alternative circuitry for a particular device.



■ FIGURE 6.29 Suggested external circuitry for obtaining a zero output voltage. The  $\pm 10$  V supplies are shown as an example; the actual supply voltages used in the final circuit would be chosen in practice.

## Slew Rate

Up to now, we have tacitly assumed that the op amp will respond equally well to signals of any frequency, although perhaps we would not be surprised to find that in practice there is some type of limitation in this regard. Since we know that op amp circuits work well at dc, which is essentially zero frequency, it is the performance as the signal frequency is *increased* that we must consider. One measure of the frequency performance of an op amp is its *slew rate*, which is the rate at which the output voltage can respond to changes in the input; it is most often expressed in  $V/\mu s$ . The typical slew rate specification for several commercially available devices is provided in Table 6.3, showing values on the order of a few volts per microsecond. One notable exception is the OPA690, which is designed as a high-speed op amp for video applications requiring operation at several hundred MHz. As can be seen, a respectable slew rate of  $1800 \text{ V}/\mu\text{s}$  is not unrealistic for this device, although its other parameters, particularly input bias current and CMRR, suffer somewhat as a result.

The PSpice simulations shown in Fig. 6.30 illustrate the degradation in performance of an op amp due to slew rate limitations. The circuit simulated is an LF411 configured as a noninverting amplifier with a gain of 2 and powered by  $\pm 15 \text{ V}$  supplies. The input waveform is shown in green, and has

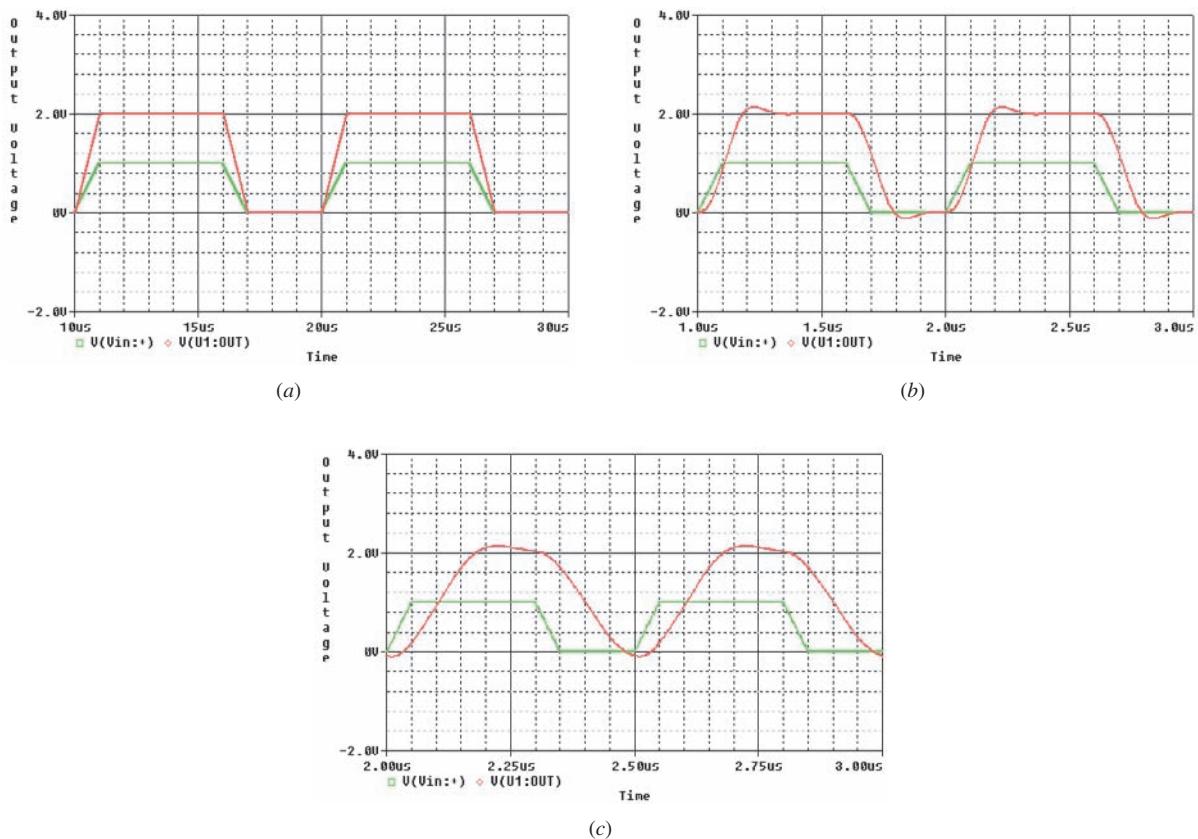


FIGURE 6.30 Simulated performance of an LF411 op amp connected as a noninverting amplifier having a gain of 2, with  $\pm 15 \text{ V}$  supplies and a pulsed input waveform. (a) Rise and fall times =  $1 \mu\text{s}$ , pulse width =  $5 \mu\text{s}$ ; (b) rise and fall times =  $100 \text{ ns}$ , pulse width =  $500 \text{ ns}$ ; (c) rise and fall times =  $50 \text{ ns}$ , pulse width =  $250 \text{ ns}$ .

a peak voltage of 1 V; the output voltage is shown in red. The simulation of Fig. 6.30a corresponds to a rise and fall time of 1  $\mu$ s which, although a short time span for humans, is easily coped with by the LF411. As the rise and fall times are decreased by a factor of 10 to 100 ns (Fig. 6.30b), we begin to see that the LF411 is having a small difficulty in tracking the input. In the case of a 50 ns rise and fall time (Fig. 6.30c), we see that not only is there a significant delay between the output and the input, but the waveform is noticeably distorted as well—not a good feature of an amplifier. This observed behavior is consistent with the typical slew rate of 15 V/ $\mu$ s specified in Table 6.3, which indicates that the output might be expected to require roughly 130 ns to change from 0 to 2 V (or 2 V to 0 V).

## Packaging

Modern op amps are available in a number of different types of packages. Some styles are better suited to high temperatures, and there are a variety of different ways to mount ICs on printed-circuit boards. Figure 6.31 shows several different styles of the LM741, manufactured by National Semiconductor. The label “NC” next to a pin means “no connection.” The package styles shown in the figure are standard configurations, and are used for a large number of different integrated circuits; occasionally there are more pins available on a package than required.

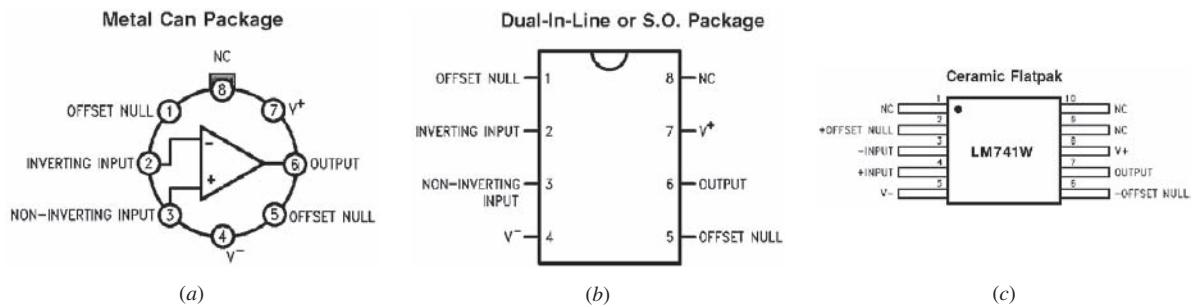


FIGURE 6.31 Several different package styles for the LM741 op amp: (a) metal can; (b) dual-in-line package; (c) ceramic flatpak.  
(Copyright © 2011 National Semiconductor Corporation ([www.national.com](http://www.national.com)). All rights reserved. Used with permission.)

## COMPUTER-AIDED ANALYSIS



As we have just seen, PSpice can be enormously helpful in predicting the output of an op amp circuit, especially in the case of time-varying inputs. We will find, however, that our ideal op amp model agrees fairly well with PSpice simulations as a general rule.

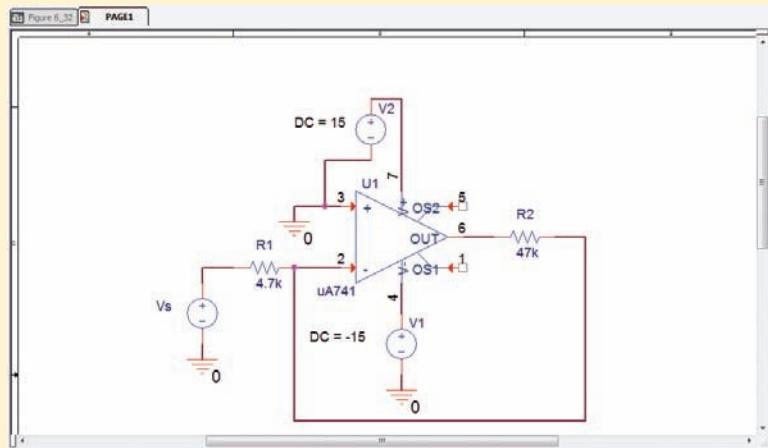
When performing a PSpice simulation of an op amp circuit, we must be careful to remember that positive and negative dc supplies must be connected to the device (with the exception of the LM324, which is designed to be a single-supply op amp). Although the model shows the offset null pins used to zero the output voltage, PSpice does not build in any offset, so these pins are typically left floating (unconnected).

Table 6.3 shows the different op amp part numbers available in the Evaluation version of PSpice; other models are available in the commercial version of the software and from some manufacturers.

## EXAMPLE 6.7

Simulate the circuit of Fig. 6.3 using PSpice. Determine the point(s) at which saturation begins if  $\pm 15$  V dc supplies are used to power the device. Compare the gain calculated by PSpice to what was predicted using the ideal op amp model.

We begin by drawing the inverting amplifier circuit of Fig. 6.3 using the schematic capture tool as shown in Fig. 6.32. Note that two separate 15 V dc supplies are required to power the op amp.



■ FIGURE 6.32 The inverting amplifier of Fig. 6.3 drawn using a  $\mu$ A741 op amp.

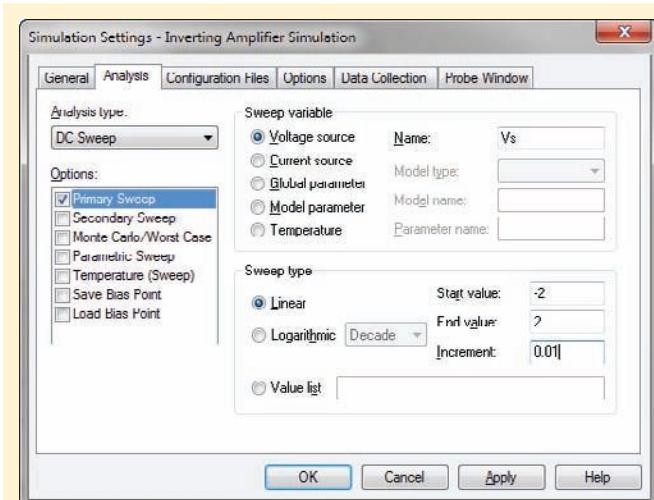
Our previous analysis using an ideal op amp model predicted a gain of  $-10$ . With an input of  $5 \sin 3t$  mV, this led to an output voltage of  $-50 \sin 3t$  mV. However, an implicit assumption in the analysis was that *any* voltage input would be amplified by a factor of  $-10$ . Based on practical considerations, we expect this to be true for *small* input voltages, but the output will eventually saturate to a value comparable to the corresponding power supply voltage.

We perform a dc sweep from  $-2$  to  $+2$  volts, as shown in Fig. 6.33; this is a slightly larger range than the supply voltage divided by the gain, so we expect our results to include the positive and negative saturation regions.

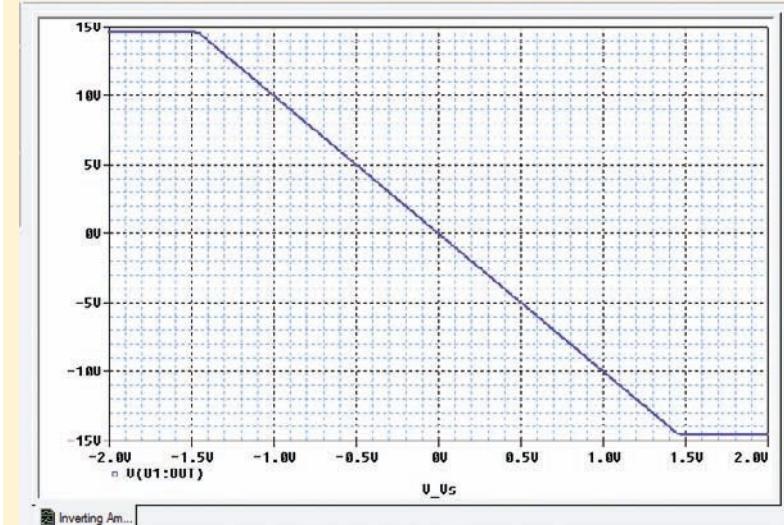
Using the cursor tool on the simulation results shown in Fig. 6.34a, the input-output characteristic of the amplifier is indeed linear over a wide input range, corresponding approximately to  $-1.45 < V_s < +1.45$  V (Fig. 6.34b): This range is slightly less than the range defined by dividing the positive and negative supply voltages by the gain. Outside this range, the output of the op amp saturates, with only a slight dependence on the input voltage. In the two saturation regions, then, the circuit does not perform as a linear amplifier.

Increasing the number of cursor digits (**Tools**, **Options**, **Number of Cursor Digits**) to 10, we find that at an input voltage of  $V_s = 1.0$  V, the

(Continued on next page)



■ FIGURE 6.33 DC sweep setup window.



(a)



(b)

■ FIGURE 6.34 (a) Output voltage of the inverting amplifier circuit, with the onset of saturation identified with the cursor tool. (b) Close-up of the cursor window.

output voltage is  $-9.99548340$ , slightly less than the value of  $-10$  predicted from the ideal op amp model, and slightly different from the value of  $-9.999448$  obtained in Example 6.6 using an analytical model. Still, the results predicted by the PSpice  $\mu$ A741 model are within a few

hundredths of a percent of either analytical model, demonstrating that the ideal op amp model is indeed a remarkably accurate approximation for modern operational amplifier integrated circuits.

### PRACTICE

- 6.7 Simulate the remaining op amp circuits described in this chapter, and compare the results to those predicted using the ideal op amp model.

## 6.6 COMPARATORS AND THE INSTRUMENTATION AMPLIFIER

### The Comparator

Every op amp circuit we have discussed up to now has featured an electrical connection between the output pin and the inverting input pin. This is known as *closed-loop* operation, and is used to provide negative feedback as discussed previously. Closed loop is the preferred method of using an op amp as an amplifier, as it serves to isolate the circuit performance from variations in the open-loop gain that arise from changes in temperature or manufacturing differences. There are a number of applications, however, where it is advantageous to use an op amp in an *open-loop* configuration. Devices intended for such applications are frequently referred to as *comparators*, as they are designed slightly differently from regular op amps in order to improve their speed in open-loop operation.

Figure 6.35a shows a simple comparator circuit where a 2.5 V reference voltage is connected to the noninverting input, and the voltage being compared ( $v_{in}$ ) is connected to the inverting input. Since the op amp has a very large open-loop gain  $A$  ( $10^5$  or greater, typically, as seen in Table 6.3), it does not take a large voltage difference between the input terminals to drive it into saturation. In fact, a differential input voltage as small as the supply voltage divided by  $A$  is required—approximately  $\pm 120 \mu\text{V}$  in the case of the circuit in Fig. 6.35a and  $A = 10^5$ . The distinctive output of the comparator circuit is shown in Fig. 6.35b, where the response swings

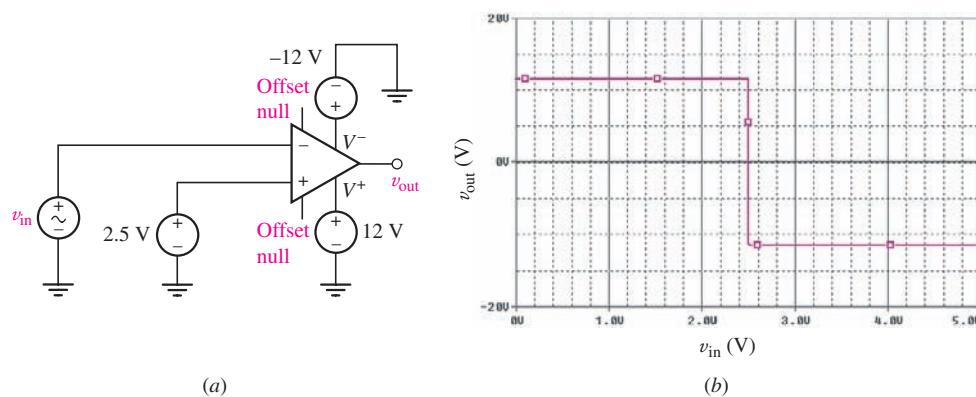


FIGURE 6.35 (a) An example comparator circuit with a 2.5 V reference voltage. (b) Graph of input-output characteristic.

between positive and negative saturation, with essentially no linear “amplification” region. Thus, a positive 12 V output from the comparator indicates that the input voltage is *less than* the reference voltage, and a negative 12 V output indicates an input voltage *greater than* the reference. Opposite behavior is obtained if we connect the reference voltage to the inverting input instead.

### EXAMPLE 6.8

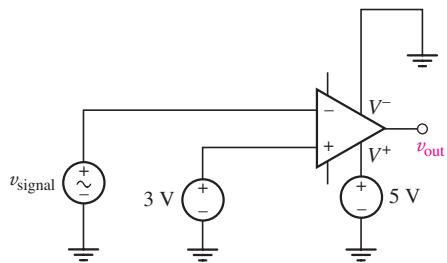


FIGURE 6.36 One possible design for the required circuit.

**Design a circuit that provides a “logic 1” 5 V output if a certain voltage signal drops below 3 V, and zero volts otherwise.**

Since we want the output of our comparator to swing between 0 and 5 V, we will use an op amp with a single-ended +5 V supply, connected as shown in Fig. 6.36. We connect a +3 V reference voltage to the noninverting input, which may be provided by two 1.5 V batteries in series, or a suitable Zener diode reference circuit. The input voltage signal (designated  $v_{\text{signal}}$ ) is then connected to the inverting input. In reality, the saturation voltage range of a comparator circuit will be slightly less than that of the supply voltages, so that some adjustment may be required in conjunction with simulation or testing.

### PRACTICE

6.8 Design a circuit that provides a 12 V output if a certain voltage ( $v_{\text{signal}}$ ) exceeds 0 V, and a -2 V output otherwise.

Ans: One possible solution is shown in Fig. 6.37.

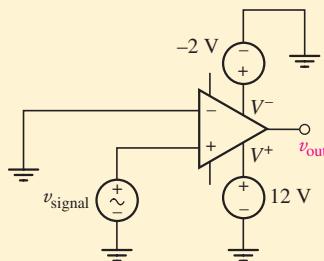


FIGURE 6.37 One possible solution to Practice Problem 6.8.

## The Instrumentation Amplifier

The basic comparator circuit acts on the voltage difference between the two input terminals to the device, although it does not technically amplify signals as the output is not proportional to the input. The difference amplifier of Fig. 6.10 also acts on the voltage difference between the inverting and noninverting inputs, and as long as care is taken to avoid saturation, *does* provide an output directly proportional to this difference. When dealing with a very small input voltage, however, a better alternative is a device

known as an **instrumentation amplifier**, which is actually three op amp devices in a single package.

An example of the common instrumentation amplifier configuration is shown in Fig. 6.38a, and its symbol is shown in Fig. 6.38b. Each input is fed directly into a voltage follower stage, and the output of both voltage followers is fed into a difference amplifier stage. It is particularly well suited to applications where the input voltage signal is very small (for example, on the order of millivolts), such as that produced by thermocouples or strain gauges, and where a significant common-mode noise signal of several volts may be present.

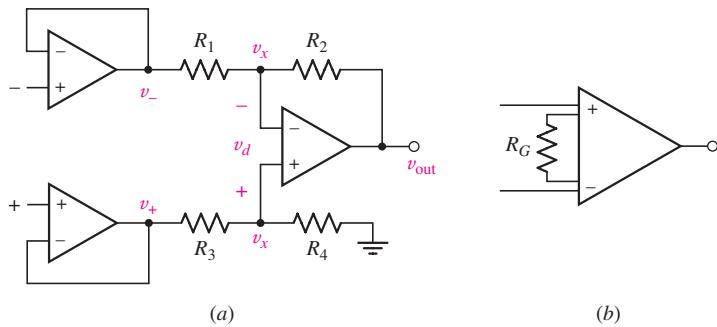


FIGURE 6.38 (a) The basic instrumentation amplifier. (b) Commonly used symbol.

If components of the instrumentation amplifier are fabricated all on the same silicon “chip,” then it is possible to obtain well-matched device characteristics and to achieve precise ratios for the two sets of resistors. In order to maximize the CMRR of the instrumentation amplifier, we expect  $R_4/R_3 = R_2/R_1$ , so that equal amplification of common-mode components of the input signals is obtained. To explore this further, we identify the voltage at the output of the top voltage follower as “ $v_-$ ,” and the voltage at the output of the bottom voltage follower as “ $v_+$ .” Assuming all three op amps are ideal and naming the voltage at either input of the difference stage  $v_x$ , we may write the following nodal equations:

$$\frac{v_x - v_-}{R_1} + \frac{v_x - v_{out}}{R_2} = 0 \quad [20]$$

and

$$\frac{v_x - v_+}{R_3} + \frac{v_x}{R_4} = 0 \quad [21]$$

Solving Eq. [21] for  $v_x$ , we find that

$$v_x = \frac{v_+}{1 + R_3/R_4} \quad [22]$$

and upon substituting into Eq. [20], obtain an expression for  $v_{out}$  in terms of the input:

$$v_{out} = \frac{R_4}{R_3} \left( \frac{1 + R_2/R_1}{1 + R_4/R_3} \right) v_+ - \frac{R_2}{R_1} v_- \quad [23]$$

From Eq. [23] it is clear that the general case allows amplification of common-mode components to the two inputs. In the specific case where

$R_4/R_3 = R_2/R_1 = K$ , however, Eq. [23] reduces to  $K(v_+ - v_-) = Kv_d$ , so that (assuming ideal op amps) only the difference is amplified and the gain is set by the resistor ratio. Since these resistors are internal to the instrumentation amplifier and not accessible to the user, devices such as the AD622 allow the gain to be set anywhere in the range of 1 to 1000 by connecting an external resistor between two pins (shown as  $R_G$  in Fig. 6.38b).

## SUMMARY AND REVIEW

In this chapter we introduced a new circuit element—a three-terminal device—called the operational amplifier (or more commonly, the *op amp*). In many circuit analysis situations it is approximated as an ideal device, which leads to two rules that are applied. We studied several op amp circuits in detail, including the *inverting amplifier* with gain  $R_f/R_1$ , the *noninverting amplifier* with gain  $1 + R_f/R_1$ , and the *summing amplifier*. We were also introduced to the *voltage follower* and the *difference amplifier*, although the analysis of these two circuits was left for the reader. The concept of cascaded stages was found to be particularly useful, as it allows a design to be broken down into distinct units, each of which has a specific function. We took a slight detour and introduced briefly a two-terminal nonlinear circuit element, the *Zener diode*, as it provides a practical and straightforward voltage reference. We then used this element to construct practical voltage and current sources using op amps, removing some of the mystery as to their origins.

Modern op amps have nearly ideal characteristics, as we found when we opted for a more detailed model based on a dependent source. Still, nonidealities are encountered occasionally, so we considered the role of *negative feedback* in reducing the effect of temperature and manufacturing-related variations in various parameters, *common-mode rejection*, and *saturation*. One of the most interesting nonideal characteristics of any op amp is *slew rate*. By simulating three different cases, we were able to see how the output voltage can struggle to follow the form of the input voltage signal once its frequency becomes high enough. We concluded the chapter with two special cases: the *comparator*, which intentionally makes use of our ability to saturate a practical (nonideal) op amp, and the instrumentation amplifier, which is routinely used to amplify very small voltages.

This is a good point to pause, take a breath, and recap some of the key points. At the same time, we will highlight relevant examples as an aid to the reader.

- ❑ There are two fundamental rules that must be applied when analyzing *ideal* op amp circuits:
  1. No current ever flows into either input terminal. (Example 6.1)
  2. No voltage ever exists between the input terminals.
- ❑ Op amp circuits are usually analyzed for an output voltage in terms of some input quantity or quantities. (Examples 6.1, 6.2)
- ❑ Nodal analysis is typically the best choice in analyzing op amp circuits, and it is usually better to begin at the input, and work toward the output. (Examples 6.1, 6.2)



- The output current of an op amp cannot be assumed; it must be found after the output voltage has been determined independently. (Example 6.2)
- The gain of an inverting op amp circuit is given by the equation

$$v_{\text{out}} = -\frac{R_f}{R_1} v_{\text{in}}$$

- The gain of a noninverting op amp circuit is given by the equation

$$v_{\text{out}} = \left(1 + \frac{R_f}{R_1}\right) v_{\text{in}}$$

(Example 6.1)

- Cascaded stages may be analyzed one stage at a time to relate the output to the input. (Example 6.3)
- Zener diodes provide a convenient voltage reference. They are not symmetric, however, meaning the two terminals are not interchangeable. (Example 6.4)
- Op amps can be used to construct current sources which are independent of the load resistance over a specific current range. (Example 6.5)
- A resistor is almost always connected from the output pin of an op amp to its inverting input pin, which incorporates negative feedback into the circuit for increased stability.
- The ideal op amp model is based on the approximation of infinite open-loop gain  $A$ , infinite input resistance  $R_i$ , and zero output resistance  $R_o$ . (Example 6.6)
- In practice, the output voltage range of an op amp is limited by the supply voltages used to power the device. (Example 6.7)
- Comparators are op amps designed to be driven into saturation. These circuits operate in open loop, and hence have no external feedback resistor. (Example 6.8)

## READING FURTHER

Two very readable books which deal with a variety of op amp applications are:

R. Mancini (ed.), *Op Amps Are For Everyone*, 2nd ed. Amsterdam: Newnes, 2003. Also available on the Texas Instruments website ([www.ti.com](http://www.ti.com)).

W. G. Jung, *Op Amp Cookbook*, 3rd ed. Upper Saddle River, N.J.: Prentice-Hall, 1997.

Characteristics of Zener and other types of diodes are covered in Chapter 1 of

W. H. Hayt, Jr., and G. W. Neudeck, *Electronic Circuit Analysis and Design*, 2nd ed. New York: Wiley, 1995.

One of the first reports of the implementation of an “operational amplifier” can be found in

J. R. Ragazzini, R. M. Randall, and F. A. Russell, “Analysis of problems in dynamics by electronic circuits,” *Proceedings of the IRE* 35(5), 1947, pp. 444–452.

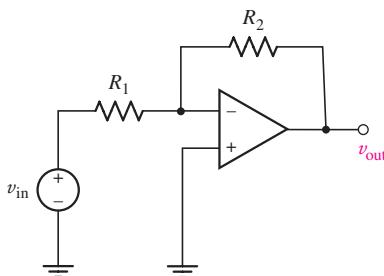
And an early applications guide for the op amp can be found on the Analog Devices, Inc. website ([www.analog.com](http://www.analog.com)):

George A. Philbrick Researches, Inc., *Applications Manual for Computing Amplifiers for Modelling, Measuring, Manipulating & Much Else.*  
Norwood, Mass.: Analog Devices, 1998.

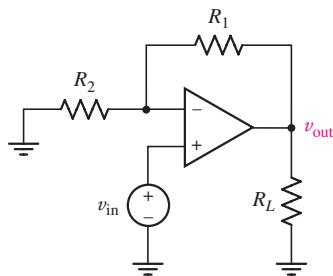
## EXERCISES

### 6.2 The Ideal Op Amp

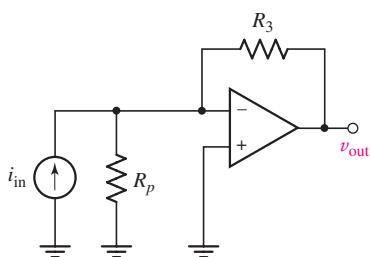
- For the op amp circuit shown in Fig. 6.39, calculate  $v_{\text{out}}$  if (a)  $R_1 = R_2 = 100 \Omega$  and  $v_{\text{in}} = 5 \text{ V}$ ; (b)  $R_2 = 200R_1$  and  $v_{\text{in}} = 1 \text{ V}$ ; (c)  $R_1 = 4.7 \text{ k}\Omega$ ,  $R_2 = 47 \text{ k}\Omega$ , and  $v_{\text{in}} = 20 \sin 5t \text{ V}$ .



■ FIGURE 6.39



■ FIGURE 6.40

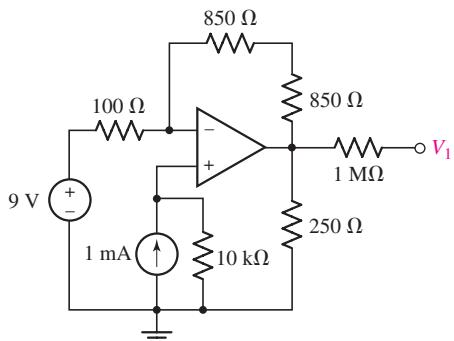


■ FIGURE 6.41

- Determine the power dissipated by a  $100 \Omega$  resistor connected between ground and the output pin of the op amp of Fig. 6.39 if  $v_{\text{in}} = 4 \text{ V}$  and (a)  $R_1 = 2R_2$ ; (b)  $R_1 = 1 \text{ k}\Omega$  and  $R_2 = 22 \text{ k}\Omega$ ; (c)  $R_1 = 100 \Omega$  and  $R_2 = 101 \Omega$ .
- Connect a  $1 \Omega$  resistor between ground and the output terminal of the op amp of Fig. 6.39, and sketch  $v_{\text{out}}(t)$  if (a)  $R_1 = R_2 = 10 \Omega$  and  $v_{\text{in}} = 5 \sin 10t \text{ V}$ ; (b)  $R_1 = 0.2R_2 = 1 \text{ k}\Omega$ , and  $v_{\text{in}} = 5 \cos 10t \text{ V}$ ; (c)  $R_1 = 10 \Omega$ ,  $R_2 = 200 \Omega$ , and  $v_{\text{in}} = 1.5 + 5e^{-t} \text{ V}$ .
- For the circuit of Fig. 6.40, calculate  $v_{\text{out}}$  if (a)  $R_1 = R_2 = 100 \text{ k}\Omega$ ,  $R_L = 100 \Omega$ , and  $v_{\text{in}} = 5 \text{ V}$ ; (b)  $R_1 = 0.1R_2$ ,  $R_L = \infty$ , and  $v_{\text{in}} = 2 \text{ V}$ ; (c)  $R_1 = 1 \text{ k}\Omega$ ,  $R_2 = 0$ ,  $R_L = 1 \Omega$ , and  $v_{\text{in}} = 43.5 \text{ V}$ .
- (a) Design a circuit which converts a voltage  $v_1(t) = 9 \cos 5t \text{ V}$  into  $9 \sin 5t \text{ V}$ .  
(b) Verify your design by analyzing the final circuit.
- A certain load resistor requires a constant  $5 \text{ V}$  dc supply. Unfortunately, its resistance value changes with temperature. Design a circuit which supplies the requisite voltage if only  $9 \text{ V}$  batteries and standard  $10\%$  tolerance resistor values are available.
- For the circuit of Fig. 6.40,  $R_1 = R_L = 50 \Omega$ . Calculate the value of  $R_2$  required to deliver  $5 \text{ W}$  to  $R_L$  if  $V_{\text{in}}$  equals (a)  $5 \text{ V}$ ; (b)  $1.5 \text{ V}$ . (c) Repeat parts (a) and (b) if  $R_L$  is reduced to  $22 \Omega$ .
- Calculate  $v_{\text{out}}$  as labeled in the schematic of Fig. 6.41 if (a)  $i_{\text{in}} = 1 \text{ mA}$ ,  $R_p = 2.2 \text{ k}\Omega$ , and  $R_3 = 1 \text{ k}\Omega$ ; (b)  $i_{\text{in}} = 2 \text{ A}$ ,  $R_p = 1.1 \Omega$ , and  $R_3 = 8.5 \Omega$ . (c) For each case, state whether the circuit is wired as a noninverting or an inverting amplifier. Explain your reasoning.
- (a) Design a circuit using only a single op amp which adds two voltages  $v_1$  and  $v_2$  and provides an output voltage twice their sum (i.e.,  $v_{\text{out}} = 2v_1 + 2v_2$ ).  
(b) Verify your design by analyzing the final circuit.
- (a) Design a circuit that provides a current  $i$  which is equal in magnitude to the sum of three input voltages  $v_1$ ,  $v_2$ , and  $v_3$ . (Compare volts to amperes.)  
(b) Verify your design by analyzing the final circuit.

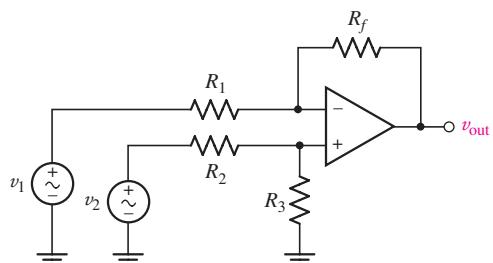
- DP** 11. (a) Design a circuit that provides a voltage  $v_{\text{out}}$  which is equal to the difference between two voltages  $v_2$  and  $v_1$  (i.e.,  $v_{\text{out}} = v_2 - v_1$ ), if you have only the following resistors from which to choose: two  $1.5 \text{ k}\Omega$  resistors, four  $6 \text{ k}\Omega$  resistors, or three  $500 \Omega$  resistors. (b) Verify your design by analyzing the final circuit.

12. Analyze the circuit of Fig. 6.42 and determine a value for  $V_1$ , which is referenced to ground.



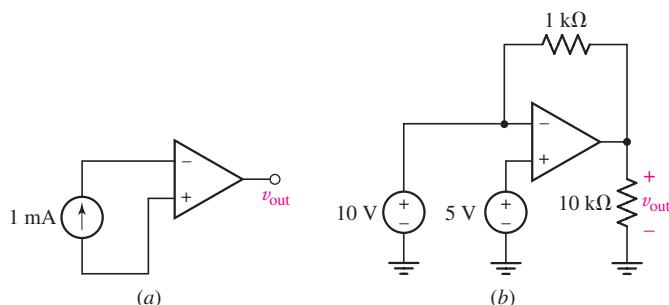
■ FIGURE 6.42

13. Derive an expression for  $v_{\text{out}}$  as a function of  $v_1$  and  $v_2$  for the circuit represented in Fig. 6.43.



■ FIGURE 6.43

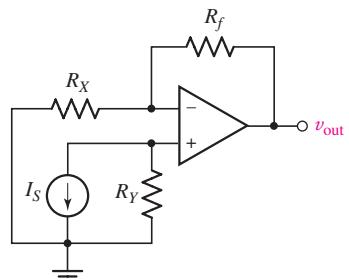
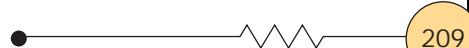
14. Explain what is wrong with each diagram in Fig. 6.44 if the two op amps are known to be *perfectly ideal*.



■ FIGURE 6.44

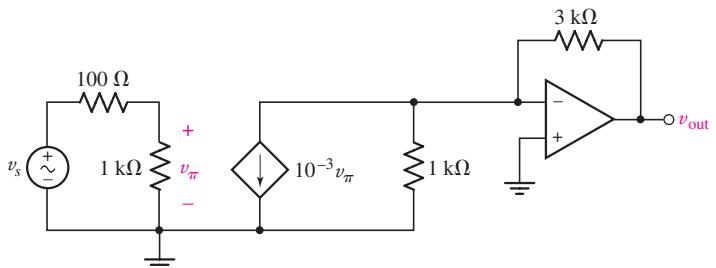
15. For the circuit depicted in Fig. 6.45, calculate  $v_{\text{out}}$  if  $I_s = 2 \text{ mA}$ ,  $R_Y = 4.7 \text{ k}\Omega$ ,  $R_X = 1 \text{ k}\Omega$ , and  $R_f = 500 \Omega$ .

16. Consider the amplifier circuit shown in Fig. 6.45. What value of  $R_f$  will yield  $v_{\text{out}} = 2 \text{ V}$  when  $I_s = 10 \text{ mA}$  and  $R_Y = 2R_X = 500 \Omega$ ?



■ FIGURE 6.45

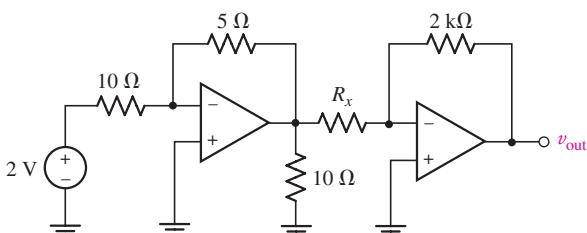
17. With respect to the circuit shown in Fig. 6.46, calculate  $v_{\text{out}}$  if  $v_s$  equals  
 (a)  $2 \cos 100t$  mV; (b)  $2 \sin(4t + 19^\circ)$  V.



■ FIGURE 6.46

### 6.3 Cascaded Stages

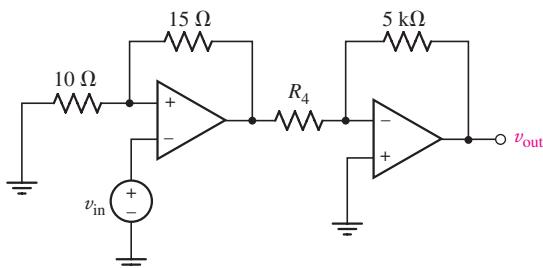
18. Calculate  $v_{\text{out}}$  as labeled in the circuit of Fig. 6.47 if  $R_x = 1 \text{ k}\Omega$ .



■ FIGURE 6.47

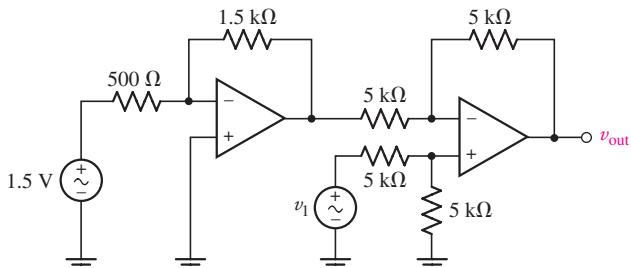
19. For the circuit of Fig. 6.47, determine the value of  $R_x$  that will result in a value of  $v_{\text{out}} = 10$  V.

20. Referring to Fig. 6.48, sketch  $v_{\text{out}}$  as a function of (a)  $v_{\text{in}}$  over the range of  $-2 \text{ V} \leq v_{\text{in}} \leq +2 \text{ V}$ , if  $R_4 = 2 \text{ k}\Omega$ ; (b)  $R_4$  over the range of  $1 \text{ k}\Omega \leq R_4 \leq 10 \text{ k}\Omega$ , if  $v_{\text{in}} = 300 \text{ mV}$ .



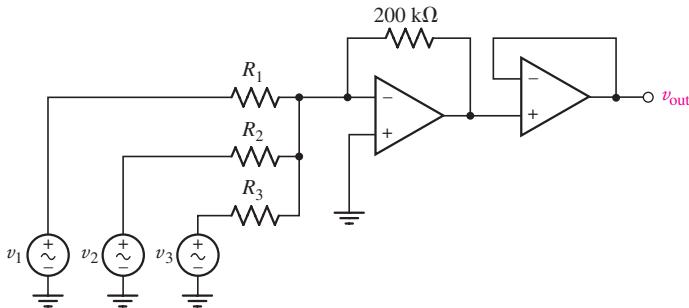
■ FIGURE 6.48

21. Obtain an expression for  $v_{\text{out}}$  as labeled in the circuit of Fig. 6.49 if  $v_1$  equals  
 (a) 0 V; (b) 1 V; (c) -5 V; (d)  $2 \sin 100t$  V.



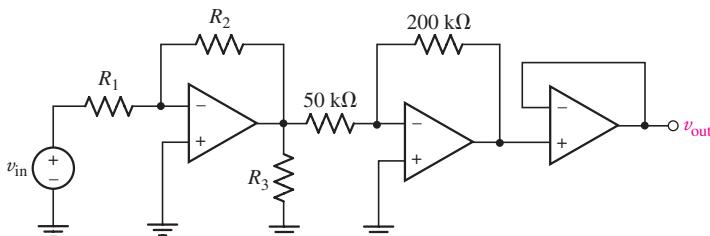
■ FIGURE 6.49

22. The 1.5 V source of Fig. 6.49 is disconnected, and the output of the circuit shown in Fig. 6.48 is connected to the left-hand terminal of the  $500\ \Omega$  resistor instead. Calculate  $v_{out}$  if  $R_4 = 2\text{ k}\Omega$  and (a)  $v_{in} = 2\text{ V}$ ,  $v_1 = 1\text{ V}$ ; (b)  $v_{in} = 1\text{ V}$ ,  $v_1 = 0$ ; (c)  $v_{in} = 1\text{ V}$ ,  $v_1 = -1\text{ V}$ .
23. For the circuit shown in Fig. 6.50, compute  $v_{out}$  if (a)  $v_1 = 2v_2 = 0.5v_3 = 2.2\text{ V}$  and  $R_1 = R_2 = R_3 = 50\text{ k}\Omega$ ; (b)  $v_1 = 0$ ,  $v_2 = -8\text{ V}$ ,  $v_3 = 9\text{ V}$ , and  $R_1 = 0.5R_2 = 0.4R_3 = 100\text{ k}\Omega$ .



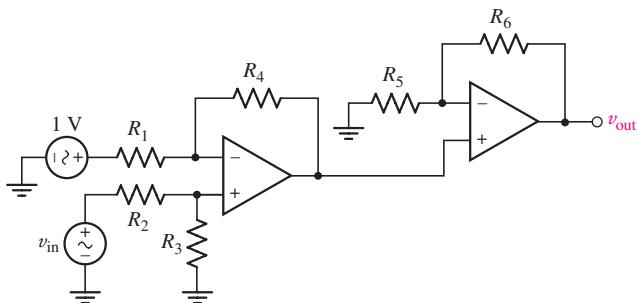
■ FIGURE 6.50

- DP** 24. (a) Design a circuit which will add the voltages produced by three separate pressure sensors, each in the range of  $0 \leq v_{\text{sensor}} \leq 5\text{ V}$ , and produce a positive voltage  $v_{out}$  linearly correlated to the voltage sum such that  $v_{out} = 0$  when all three voltages are zero, and  $v_{out} = 2\text{ V}$  when all three voltages are at their maximum. (b) Verify your design by analyzing the final circuit.
- DP** 25. (a) Design a circuit which produces an output voltage  $v_{out}$  proportional to the difference of two positive voltages  $v_1$  and  $v_2$  such that  $v_{out} = 0$  when both voltages are equal, and  $v_{out} = 10\text{ V}$  when  $v_1 - v_2 = 1\text{ V}$ . (b) Verify your design by analyzing the final circuit.
- DP** 26. (a) Three pressure-sensitive sensors are used to double-check the weight readings obtained from the suspension systems of a long-range jet airplane. Each sensor is calibrated such that  $10\ \mu\text{V}$  corresponds to 1 kg. Design a circuit which adds the three voltage signals to produce an output voltage calibrated such that  $10\text{ V}$  corresponds to 400,000 kg, the maximum takeoff weight of the aircraft. (b) Verify your design by analyzing the final circuit.
- DP** 27. (a) The oxygen supply to a particular bathysphere consists of four separate tanks, each equipped with a pressure sensor capable of measuring between 0 (corresponding to 0 V output) and 500 bar (corresponding to 5 V output). Design a circuit which produces a voltage proportional to the total pressure in all tanks, such that  $1.5\text{ V}$  corresponds to 0 bar and  $3\text{ V}$  corresponds to 2000 bar. (b) Verify your design by analyzing the final circuit.
28. For the circuit shown in Fig. 6.51, let  $v_{in} = 8\text{ V}$ , and select values for  $R_1$ ,  $R_2$ , and  $R_3$  to ensure an output voltage  $v_{out} = 4\text{ V}$ .



■ FIGURE 6.51

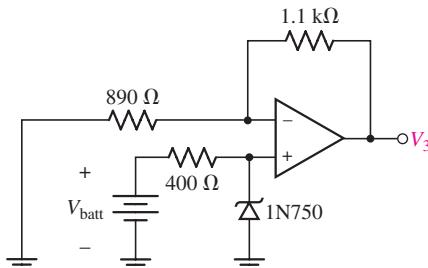
29. For the circuit of Fig. 6.52, derive an expression for  $v_{\text{out}}$  in terms of  $v_{\text{in}}$ .



■ FIGURE 6.52

#### 6.4 Circuits for Voltage and Current Sources

30. Construct a circuit based on the 1N4740 diode which provides a reference voltage of 10 V if only 9 V batteries are available. Note that the breakdown voltage of this diode is equal to 10 V at a current of 25 mA.
31. Employ a 1N4733 Zener diode to construct a circuit which provides a 4 V reference voltage to a  $1 \text{ k}\Omega$  load, if only 9 V batteries are available as sources. Note that the Zener breakdown voltage of this diode is 5.1 V at a current of 76 mA.
32. (a) Design a circuit which provides a 5 V dc reference voltage to an unknown (nonzero resistance) load, if only a 9 V battery is available as a supply.  
(b) Verify your design with an appropriate simulation. As part of this, determine the acceptable range for the load resistor.
33. A particular passive network can be represented by a Thévenin equivalent resistance between  $10 \Omega$  and  $125 \Omega$  depending on the operating temperature.  
(a) Design a circuit which provides a constant 2.2 V to this network regardless of temperature. (b) Verify your design with an appropriate simulation (resistance can be varied from within a single simulation, as described in Chap. 8).
34. Calculate the voltage  $V_1$  as labeled in the circuit of Fig. 6.53 if the battery is rated at  $V_{\text{batt}}$  equal to (a) 9 V; (b) 12 V. (c) Verify your solutions with appropriate simulations, commenting on the possible origin of any discrepancies.



■ FIGURE 6.53

35. (a) Design a current source based on the 1N750 diode which is capable of providing a dc current of  $750 \mu\text{A}$  to a load  $R_L$ , such that  $1 \text{ k}\Omega < R_L < 50 \text{ k}\Omega$ .  
(b) Verify your design with an appropriate simulation (note that resistance can be varied within a single simulation, as described in Chap. 8).
36. (a) Design a current source able to provide a dc current of 50 mA to an unspecified load. Use a 1N4733 diode ( $V_{\text{br}} = 5.1 \text{ V}$  at 76 mA). (b) Use an appropriate simulation to determine the permissible range of load resistance for your design.



DP

37. (a) Design a current source able to provide a dc current of 10 mA to an unspecified load. Use a 1N4747 diode ( $V_{br} = 20$  V at 12.5 mA). (b) Use an appropriate simulation to determine the permissible range of load resistance for your design.
38. The circuit depicted in Fig. 6.54 is known as a Howland current source. Derive expressions for  $v_{out}$  and  $I_L$ , respectively as a function of  $V_1$  and  $V_2$ .
39. For the circuit depicted in Fig. 6.54, known as a Howland current source, set  $V_2 = 0$ ,  $R_1 = R_3$ , and  $R_2 = R_4$ ; then solve for the current  $I_L$  when  $R_1 = 2R_2 = 1\text{ k}\Omega$  and  $R_L = 100\Omega$ .

## 6.5 Practical Considerations

40. (a) Employ the parameters listed in Table 6.3 for the  $\mu$ A741 op amp to analyze the circuit of Fig. 6.55 and compute a value for  $v_{out}$ . (b) Compare your result to what is predicted using the ideal op amp model.

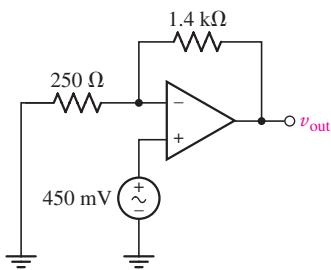


FIGURE 6.55

41. (a) Employ the parameters listed in Table 6.3 for the  $\mu$ A741 op amp to analyze the circuit of Fig. 6.10 if  $R = 1.5\text{ k}\Omega$ ,  $v_1 = 2$  V, and  $v_2 = 5$  V. (b) Compare your solution to what is predicted using the ideal op amp model.
42. Define the following terms, and explain when and how each can impact the performance of an op amp circuit: (a) common-mode rejection ratio; (b) slew rate; (c) saturation; (d) feedback.
43. For the circuit of Fig. 6.56, replace the  $470\Omega$  resistor with a short circuit, and compute  $v_{out}$  using (a) the ideal op amp model; (b) the parameters listed in Table 6.3 for the  $\mu$ A741 op amp; (c) an appropriate PSpice simulation. (d) Compare the values obtained in parts (a) to (c) and comment on the possible origin of any discrepancies.
44. If the circuit of Fig. 6.55 is analyzed using the detailed model of an op amp (as opposed to the ideal op amp model), calculate the value of open-loop gain  $A$  required to achieve a closed-loop gain within 2% of its ideal value.
45. Replace the 2 V source in Fig. 6.56 with a sinusoidal voltage source having a magnitude of 3 V and radian frequency  $\omega = 2\pi f$ . (a) Which device, a  $\mu$ A741 op amp or an LF411 op amp, will track the source frequency better over the range  $1\text{ Hz} < f < 10\text{ MHz}$ ? Explain. (b) Compare the frequency performance of the circuit over the range  $1\text{ Hz} < f < 10\text{ MHz}$  using appropriate PSpice simulations, and compare the results to your prediction in part (a).
46. (a) For the circuit of Fig. 6.56, if the op amp (assume LF411) is powered by matched 9 V supplies, estimate the maximum value to which the  $470\Omega$  resistor can be increased before saturation effects become apparent. (b) Verify your prediction with an appropriate simulation.
47. For the circuit of Fig. 6.55, calculate the differential input voltage and the input bias current if the op amp is a(n) (a)  $\mu$ A741; (b) LF411; (c) AD549K; (d) OPA690.
48. Calculate the common-mode gain for each device listed in Table 6.3. Express your answer in units of V/V, not dB.

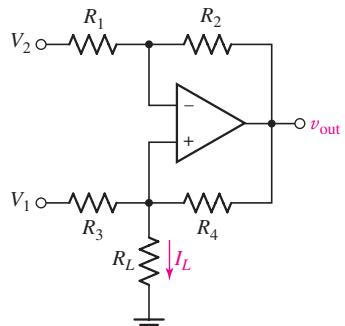


FIGURE 6.54

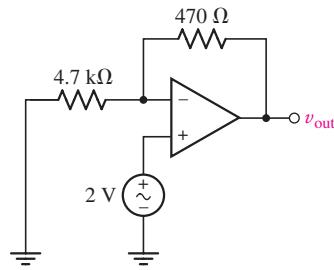
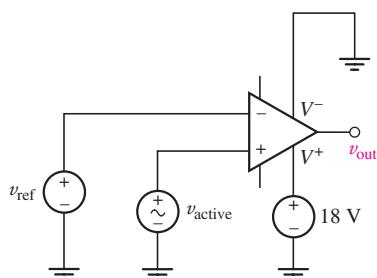
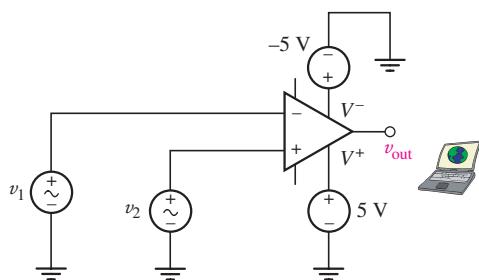


FIGURE 6.56



■ FIGURE 6.57



■ FIGURE 6.58

## 6.6 Comparators and the Instrumentation Amplifier

- DP** 49. Human skin, especially when damp, is a reasonable conductor of electricity. If we assume a resistance of less than  $10 \text{ M}\Omega$  for a fingertip pressed across two terminals, design a circuit which provides a  $+1 \text{ V}$  output if this nonmechanical switch is “closed” and  $-1 \text{ V}$  if it is “open.”

50. Design a circuit which provides an output voltage  $v_{\text{out}}$  based on the behavior of another voltage  $v_{\text{in}}$ , such that

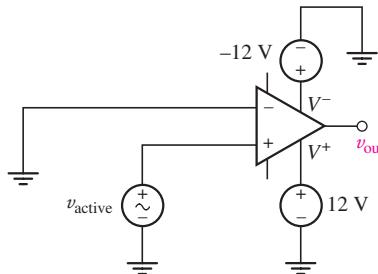
$$v_{\text{out}} = \begin{cases} +2.5 \text{ V} & v_{\text{in}} > 1 \text{ V} \\ 1.2 \text{ V} & \text{otherwise} \end{cases}$$

51. For the instrumentation amplifier shown in Fig. 6.38a, assume that the three internal op amps are ideal, and determine the CMRR of the circuit if (a)  $R_1 = R_3$  and  $R_2 = R_4$ ; (b) all four resistors have different values.

52. For the circuit depicted in Fig. 6.57, sketch the expected output voltage  $v_{\text{out}}$  as a function of  $v_{\text{active}}$  for  $-5 \text{ V} \leq v_{\text{active}} \leq +5 \text{ V}$ , if  $v_{\text{ref}}$  is equal to (a)  $-3 \text{ V}$ ; (b)  $+3 \text{ V}$ .

53. For the circuit depicted in Fig. 6.58, (a) sketch the expected output voltage  $v_{\text{out}}$  as a function of  $v_1$  for  $-5 \text{ V} \leq v_1 \leq +5 \text{ V}$ , if  $v_2 = +2 \text{ V}$ ; (b) sketch the expected output voltage  $v_{\text{out}}$  as a function of  $v_2$  for  $-5 \text{ V} \leq v_2 \leq +5 \text{ V}$ , if  $v_1 = +2 \text{ V}$ .

54. For the circuit depicted in Fig. 6.59, sketch the expected output voltage  $v_{\text{out}}$  as a function of  $v_{\text{active}}$ , if  $-2 \text{ V} \leq v_{\text{active}} \leq +2 \text{ V}$ . Verify your solution using a  $\mu\text{A741}$  (although it is slow compared to op amps designed specifically for use as comparators, its PSpice model works well, and as this is a dc application speed is not an issue). Submit a properly labeled schematic with your results.



■ FIGURE 6.59



55. In digital logic applications, a  $+5 \text{ V}$  signal represents a logic “1” state, and a  $0 \text{ V}$  signal represents a logic “0” state. In order to process real-world information using a digital computer, some type of interface is required, which typically includes an analog-to-digital (A/D) converter—a device that converts analog signals into digital signals. Design a circuit that acts as a simple 1-bit A/D, with any signal less than  $1.5 \text{ V}$  resulting in a logic “0” and any signal greater than  $1.5 \text{ V}$  resulting in a logic “1.”



56. A common application for instrumentation amplifiers is to measure voltages in resistive strain gauge circuits. These strain sensors work by exploiting the changes in resistance that result from geometric distortions, as in Eq. [6] of Chap. 2. They are often part of a bridge circuit, as shown in Fig. 6.60a, where the strain gauge is identified as  $R_G$ . (a) Show that

$$V_{\text{out}} = V_{\text{in}} \left[ \frac{R_2}{R_1 + R_2} - \frac{R_3}{R_3 + R_{\text{Gauge}}} \right].$$

(b) Verify that  $V_{\text{out}} = 0$  when the three fixed-value resistors  $R_1$ ,  $R_2$ , and  $R_3$  are all chosen to be equal to the unstrained gauge resistance  $R_{\text{Gauge}}$ . (c) For the intended application, the gauge selected has an unstrained resistance of  $5 \text{ k}\Omega$ , and a maximum resistance increase of

50 mΩ is expected. Only  $\pm 12$  V supplies are available. Using the instrumentation amplifier of Fig. 6.60b, design a circuit that will provide a voltage signal of +1 V when the strain gauge is at its maximum loading.

### AD622 Specifications

Amplifier gain  $G$  can be varied from 2 to 1000 by connecting a resistor between pins 1 and 8 with a value calculated by  $R = \frac{50.5}{G - 1}$  kΩ.

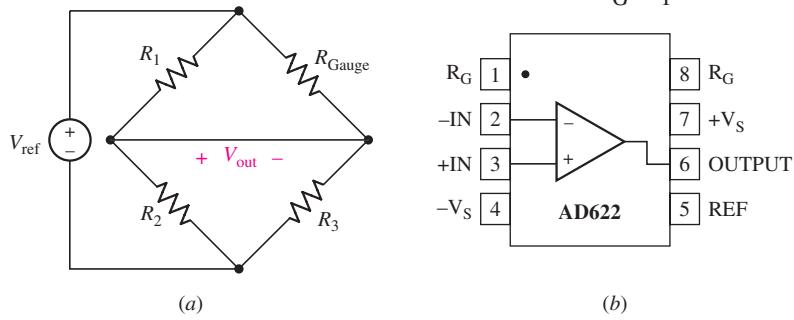


FIGURE 6.60

© Analog Devices.

### Chapter-Integrating Exercises

- DP** 57. (a) You're given an electronic switch which requires 5 V at 1 mA in order to close; it is open with no voltage present at its input. If the only microphone available produces a peak voltage of 250 mV, design a circuit which will energize the switch when someone speaks into the microphone. Note that the audio level of a general voice may not correspond to the peak voltage of the microphone. (b) Discuss any issues that may need to be addressed if your circuit were to be implemented.
- DP** 58. You've formed a band, despite advice to the contrary. Actually, the band is pretty good except for the fact that the lead singer (who owns the drum set, the microphones, and the garage where you practice) is a bit tone-deaf. Design a circuit that takes the output from each of the five microphones your band uses, and adds the voltages to create a single voltage signal which is fed to the amplifier. Except not all voltages should be equally amplified. One microphone output should be attenuated such that its peak voltage is 10% of any other microphone's peak voltage.
- DP** 59. Cadmium sulfide (CdS) is commonly used to fabricate resistors whose value depends on the intensity of light shining on the surface. In Fig. 6.61 a CdS "photocell" is used as the feedback resistor  $R_f$ . In total darkness, it has a resistance of 100 kΩ, and a resistance of 10 kΩ under a light intensity of 6 candela.  $R_L$  represents a circuit that is activated when a voltage of 1.5 V or less is applied to its terminals. Choose  $R_1$  and  $V_s$  so that the circuit represented by  $R_L$  is activated by a light of 2 candela or brighter.

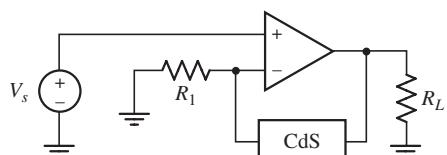


FIGURE 6.61



60. A fountain outside a certain office building is designed to reach a maximum height of 5 meters at a flow rate of 100 l/s. A variable position valve in line with the water supply to the fountain can be controlled electrically, such that 0 V applied results in the valve being fully open, and 5 V results in the valve being closed. In adverse wind conditions the maximum height of the fountain needs to be adjusted; if the wind velocity exceeds 50 km/h, the height cannot exceed 2 meters. A wind velocity sensor is available which provides a voltage calibrated such that 1 V corresponds to a wind velocity of 25 km/h. Design a circuit which uses the velocity sensor to control the fountain according to specifications.
61. For the circuit of Fig. 6.43, let all resistor values equal  $5\text{ k}\Omega$ . Sketch  $v_{\text{out}}$  as a function of time if (a)  $v_1 = 5 \sin 5t \text{ V}$  and  $v_2 = 5 \cos 5t \text{ V}$ ; (b)  $v_1 = 4e^{-t} \text{ V}$  and  $v_2 = 5e^{-2t} \text{ V}$ ; (c)  $v_1 = 2 \text{ V}$  and  $v_2 = e^{-t} \text{ V}$ .

# Capacitors and Inductors

## INTRODUCTION

In this chapter we introduce two new passive circuit elements, the *capacitor* and the *inductor*, each of which has the ability to both store and deliver *finite* amounts of energy. They differ from ideal sources in this respect, since they cannot sustain a finite average power flow over an infinite time interval. Although they are classed as linear elements, the current-voltage relationships for these new elements are time-dependent, leading to many interesting circuits. The range of capacitance and inductance values we might encounter can be huge, so that at times they may dominate circuit behavior, and at other times be essentially insignificant. Such issues continue to be relevant in modern circuit applications, particularly as computer and communication systems move to increasingly higher operating frequencies and component densities.

## 7.1 THE CAPACITOR

### Ideal Capacitor Model

Previously, we termed independent and dependent sources *active* elements, and the linear resistor a *passive* element, although our definitions of active and passive are still slightly fuzzy and need to be brought into sharper focus. We now define an **active element** as an element that is capable of furnishing an average power greater than zero to some external device, where the average is taken over an infinite time interval. Ideal sources are active elements, and the operational amplifier is also an active device. A **passive element**, however, is defined as an element that cannot supply an average power that is greater than zero over an infinite time interval. The resistor falls into this category; the energy it receives is usually transformed into heat, and it never supplies energy.

### KEY CONCEPTS

The Voltage-Current Relationship of an Ideal Capacitor

The Current-Voltage Relationship of an Ideal Inductor

Calculating Energy Stored in Capacitors and Inductors

Response of Capacitors and Inductors to Time-Varying Waveforms

Series and Parallel Combinations

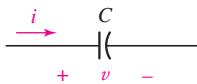
Op Amp Circuits with Capacitors

PSpice Modeling of Energy Storage Elements



We now introduce a new passive circuit element, the *capacitor*. We define capacitance  $C$  by the voltage-current relationship

$$i = C \frac{dv}{dt} \quad [1]$$



■ FIGURE 7.1 Electrical symbol and current-voltage conventions for a capacitor.

where  $v$  and  $i$  satisfy the conventions for a passive element, as shown in Fig. 7.1. We should bear in mind that  $v$  and  $i$  are functions of time; if needed, we can emphasize this fact by writing  $v(t)$  and  $i(t)$  instead. From Eq. [1], we may determine the unit of capacitance as an ampere-second per volt, or coulomb per volt. We will now define the *farad*<sup>1</sup> (F) as one coulomb per volt, and use this as our unit of capacitance.

The ideal capacitor defined by Eq. [1] is only a mathematical model of a real device. A capacitor consists of two conducting surfaces on which charge may be stored, separated by a thin insulating layer that has a very large resistance. If we assume that this resistance is sufficiently large that it may be considered infinite, then equal and opposite charges placed on the capacitor “plates” can never recombine, at least by any path within the element. The construction of the physical device is suggested by the circuit symbol shown in Fig. 7.1.

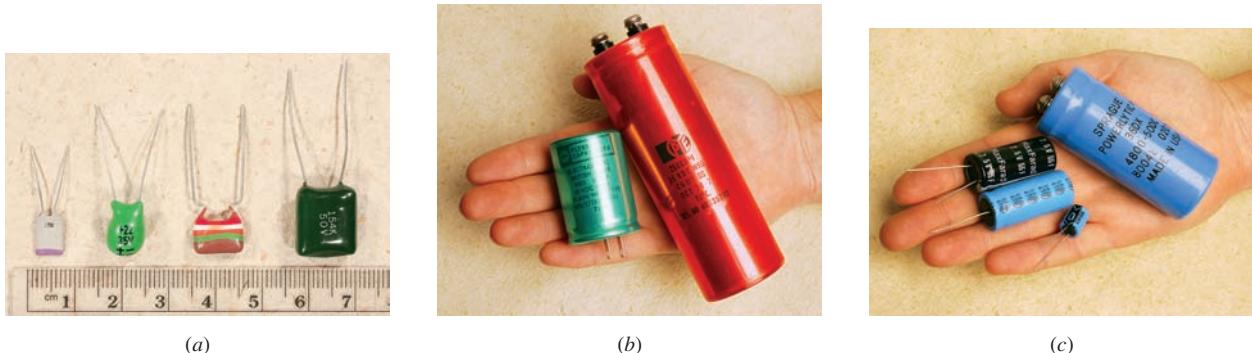
Let us visualize some external device connected to this capacitor and causing a positive current to flow into one plate of the capacitor and out of the other plate. Equal currents are entering and leaving the two terminals, and this is no more than we expect for any circuit element. Now let us examine the interior of the capacitor. The positive current entering one plate represents positive charge moving toward that plate through its terminal lead; this charge cannot pass through the interior of the capacitor, and it therefore accumulates on the plate. As a matter of fact, the current and the increasing charge are related by the familiar equation

$$i = \frac{dq}{dt}$$

Now let us consider this plate as an overgrown node and apply Kirchhoff's current law. It apparently does not hold; current is approaching the plate from the external circuit, but it is not flowing out of the plate into the “internal circuit.” This dilemma bothered a famous Scottish scientist, James Clerk Maxwell, more than a century ago. The unified electromagnetic theory that he subsequently developed hypothesizes a “displacement current” that is present wherever an electric field or a voltage is varying with time. The displacement current flowing internally between the capacitor plates is exactly equal to the conduction current flowing in the capacitor leads; Kirchhoff's current law is therefore satisfied if we include both conduction and displacement currents. However, circuit analysis is not concerned with this internal displacement current, and since it is fortunately equal to the conduction current, we may consider Maxwell's hypothesis as relating the conduction current to the changing voltage across the capacitor.

A capacitor constructed of two parallel conducting plates of area  $A$ , separated by a distance  $d$ , has a capacitance  $C = \epsilon A/d$ , where  $\epsilon$  is the permittivity, a constant of the insulating material between the plates; this assumes

(1) Named in honor of Michael Faraday.



**FIGURE 7.2** Several examples of commercially available capacitors. (a) Left to right: 270 pF ceramic, 20  $\mu\text{F}$  tantalum, 15 nF polyester, 150 nF polyester. (b) Left: 2000  $\mu\text{F}$  40 VDC rated electrolytic, 25,000  $\mu\text{F}$  35 VDC rated electrolytic. (c) Clockwise from smallest: 100  $\mu\text{F}$  63 VDC rated electrolytic, 2200  $\mu\text{F}$  50 VDC rated electrolytic, 55 F 2.5 VDC rated electrolytic, and 4800  $\mu\text{F}$  50 VDC rated electrolytic. Note that generally speaking larger capacitance values require larger packages, with one notable exception above. What was the tradeoff in that case?

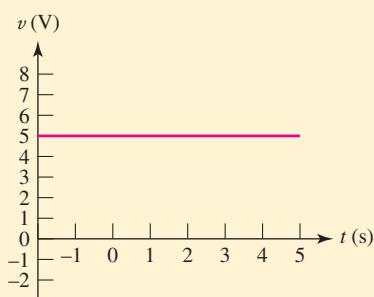
the linear dimensions of the conducting plates are all very much greater than  $d$ . For air or vacuum,  $\epsilon = \epsilon_0 = 8.854 \text{ pF/m}$ . Most capacitors employ a thin dielectric layer with a larger permittivity than air in order to minimize the device size. Examples of various types of commercially available capacitors are shown in Fig. 7.2, although we should remember that any two conducting surfaces not in direct contact with each other may be characterized by a nonzero (although probably small) capacitance. We should also note that a capacitance of several hundred *microfarads* ( $\mu\text{F}$ ) is considered “large.”

Several important characteristics of our new mathematical model can be discovered from the defining equation, Eq. [1]. A constant voltage across a capacitor results in zero current passing through it; a capacitor is thus an “*open circuit to dc*.” This fact is pictorially represented by the capacitor symbol. It is also apparent that a sudden jump in the voltage requires an infinite current. Since this is physically impossible, we will therefore prohibit the voltage across a capacitor to change in zero time.

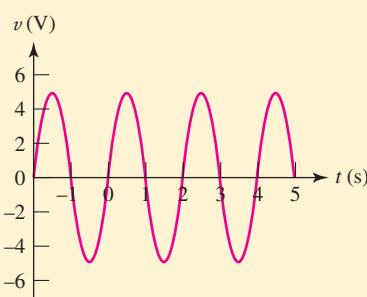


### EXAMPLE 7.1

Determine the current  $i$  flowing through the capacitor of Fig. 7.1 for the two voltage waveforms of Fig. 7.3 if  $C = 2 \text{ F}$ .



(a)



(b)

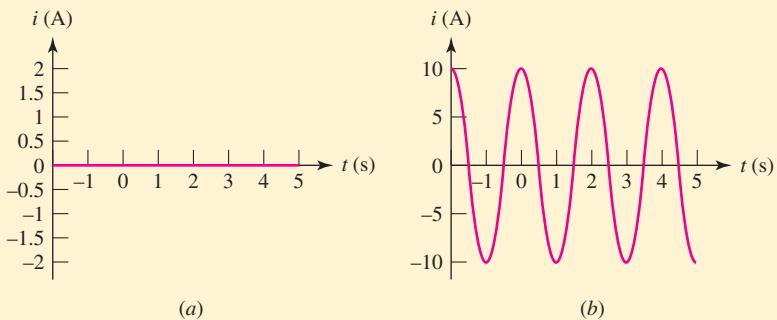
**FIGURE 7.3** (a) A dc voltage applied to the terminals of the capacitor. (b) A sinusoidal voltage waveform applied to the capacitor terminals.

(Continued on next page)

The current  $i$  is related to the voltage  $v$  across the capacitor by Eq. [1]:

$$i = C \frac{dv}{dt}$$

For the voltage waveform depicted in Fig. 7.3a,  $dv/dt = 0$ , so  $i = 0$ ; the result is plotted in Fig. 7.4a. For the case of the sinusoidal waveform of Fig. 7.3b, we expect a cosine current waveform to flow in response, having the same frequency and twice the magnitude (since  $C = 2 \text{ F}$ ). The result is plotted in Fig. 7.4b.



■ FIGURE 7.4 (a)  $i = 0$  as the voltage applied is dc. (b) The current has a cosine form in response to a sine wave voltage.

### PRACTICE

7.1 Determine the current flowing through a  $5 \text{ mF}$  capacitor in response to a voltage  $v$  equal to: (a)  $-20 \text{ V}$ ; (b)  $2e^{-5t} \text{ V}$ .

Ans: 0 A;  $-50e^{-5t} \text{ mA}$ .

## Integral Voltage-Current Relationships

The capacitor voltage may be expressed in terms of the current by integrating Eq. [1]. We first obtain

$$dv = \frac{1}{C} i(t) dt$$

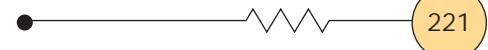
and then integrate<sup>2</sup> between the times  $t_0$  and  $t$  and between the corresponding voltages  $v(t_0)$  and  $v(t)$ :

$$v(t) = \frac{1}{C} \int_{t_0}^t i(t') dt' + v(t_0) \quad [2]$$

Equation [2] may also be written as an indefinite integral plus a constant of integration:

$$v(t) = \frac{1}{C} \int i dt + k$$

(2) Note that we are employing the mathematically correct procedure of defining a *dummy variable*  $t'$  in situations where the integration variable  $t$  is also a limit.



Finally, in many situations we will find that  $v(t_0)$ , the voltage initially across the capacitor, is not able to be discerned. In such instances it is mathematically convenient to set  $t_0 = -\infty$  and  $v(-\infty) = 0$ , so that

$$v(t) = \frac{1}{C} \int_{-\infty}^t i \, dt'$$

Since the integral of the current over any time interval is the corresponding charge accumulated on the capacitor plate into which the current is flowing, we may also define capacitance as

$$q(t) = Cv(t)$$

where  $q(t)$  and  $v(t)$  represent instantaneous values of the charge on either plate and the voltage between the plates, respectively.

### EXAMPLE 7.2

**Find the capacitor voltage that is associated with the current shown graphically in Fig. 7.5a. The value of the capacitance is  $5 \mu\text{F}$ .**

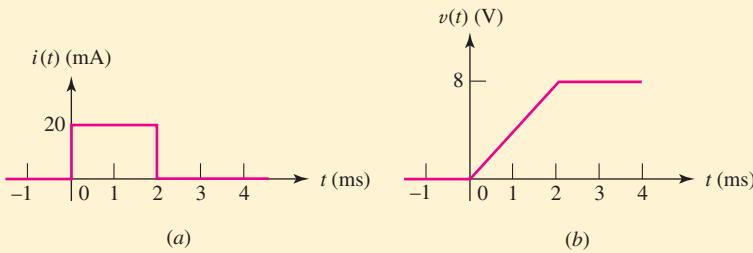


FIGURE 7.5 (a) The current waveform applied to a  $5 \mu\text{F}$  capacitor. (b) The resultant voltage waveform obtained by graphical integration.

Equation [2] is the appropriate expression here:

$$v(t) = \frac{1}{C} \int_{t_0}^t i(t') \, dt' + v(t_0)$$

but now it needs to be interpreted graphically. To do this, we note that the difference in voltage between times  $t$  and  $t_0$  is proportional to the area under the current curve defined by the same two times. The constant of proportionality is  $1/C$ .

From Fig. 7.5a, we see three separate intervals:  $t \leq 0$ ,  $0 \leq t \leq 2 \text{ ms}$ , and  $t \geq 2 \text{ ms}$ . Defining the first interval more specifically as between  $-\infty$  and  $0$ , so that  $t_0 = -\infty$ , we note two things, both a consequence of the fact that the current has always been zero up to  $t = 0$ : First,

$$v(t_0) = v(-\infty) = 0$$

Second, the integral of the current between  $t_0 = -\infty$  and  $0$  is simply zero, since  $i = 0$  in that interval. Thus,

$$v(t) = 0 + v(-\infty) \quad -\infty \leq t \leq 0$$

or

$$v(t) = 0 \quad t \leq 0$$

(Continued on next page)

If we now consider the time interval represented by the rectangular pulse, we obtain

$$v(t) = \frac{1}{5 \times 10^{-6}} \int_0^t 20 \times 10^{-3} dt' + v(0)$$

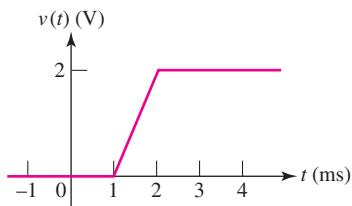
Since  $v(0) = 0$ ,

$$v(t) = 4000t \quad 0 \leq t \leq 2 \text{ ms}$$

For the semi-infinite interval following the pulse, the integral of  $i(t)$  is once again zero, so that

$$v(t) = 8 \quad t \geq 2 \text{ ms}$$

The results are expressed much more simply in a sketch than by these analytical expressions, as shown in Fig. 7.5b.



■ FIGURE 7.6

### PRACTICE

7.2 Determine the current through a 100 pF capacitor if its voltage as a function of time is given by Fig. 7.6.

Ans: 0 A,  $-\infty \leq t \leq 1$  ms; 200 nA,  $1 \text{ ms} \leq t \leq 2$  ms; 0 A,  $t \geq 2$  ms.

## Energy Storage

To determine the energy stored in a capacitor, we begin with the power delivered to it:

$$p = vi = Cv \frac{dv}{dt}$$

The change in energy stored in its electric field is simply

$$\int_{t_0}^t p dt' = C \int_{t_0}^t v \frac{dv}{dt'} dt' = C \int_{v(t_0)}^{v(t)} v' dv' = \frac{1}{2} C \{ [v(t)]^2 - [v(t_0)]^2 \}$$

and thus

$$w_C(t) - w_C(t_0) = \frac{1}{2} C \{ [v(t)]^2 - [v(t_0)]^2 \} \quad [3]$$

where the stored energy is  $w_C(t_0)$  in joules (J) and the voltage at  $t_0$  is  $v(t_0)$ . If we select a zero-energy reference at  $t_0$ , implying that the capacitor voltage is also zero at that instant, then

$$w_C(t) = \frac{1}{2} Cv^2 \quad [4]$$

Let us consider a simple numerical example. As sketched in Fig. 7.7, a sinusoidal voltage source is in parallel with a  $1 \text{ M}\Omega$  resistor and a  $20 \mu\text{F}$  capacitor. The parallel resistor may be assumed to represent the finite resistance of the dielectric between the plates of the physical capacitor (an *ideal* capacitor has infinite resistance).

**EXAMPLE 7.3**

Find the maximum energy stored in the capacitor of Fig. 7.7 and the energy dissipated in the resistor over the interval  $0 < t < 0.5$  s.

**► Identify the goal of the problem.**

The energy stored in the capacitor varies with time; we are asked for the *maximum* value over a specific time interval. We are also asked to find the *total* amount of energy dissipated by the resistor over this interval. These are actually two completely different questions.

**► Collect the known information.**

The only source of energy in the circuit is the independent voltage source, which has a value of  $100 \sin 2\pi t$  V. We are only interested in the time interval of  $0 < t < 0.5$  s. The circuit is properly labeled.

**► Devise a plan.**

Determine the energy in the capacitor by evaluating the voltage. To find the energy dissipated in the resistor during the same time interval, integrate the dissipated power,  $p_R = i_R^2 \cdot R$ .

**► Construct an appropriate set of equations.**

The energy stored in the capacitor is simply

$$w_C(t) = \frac{1}{2} Cv^2 = 0.1 \sin^2 2\pi t \quad \text{J}$$

We obtain an expression for the power dissipated by the resistor in terms of the current  $i_R$ :

$$i_R = \frac{v}{R} = 10^{-4} \sin 2\pi t \quad \text{A}$$

and so

$$p_R = i_R^2 R = (10^{-4})^2 (10^6) \sin^2 2\pi t$$

so that the energy dissipated in the resistor between 0 and 0.5 s is

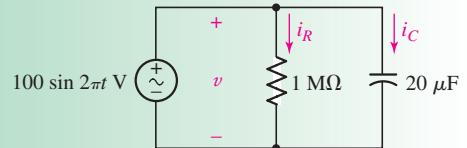
$$w_R = \int_0^{0.5} p_R dt = \int_0^{0.5} 10^{-2} \sin^2 2\pi t dt \quad \text{J}$$

**► Determine if additional information is required.**

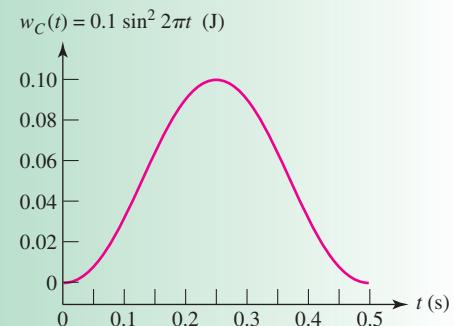
We have an expression for the energy stored in the capacitor; a sketch is shown in Fig. 7.8. The expression derived for the energy dissipated by the resistor does not involve any unknown quantities, and so may also be readily evaluated.

**► Attempt a solution.**

From our sketch of the expression for the energy stored in the capacitor, we see that it increases from zero at  $t = 0$  to a maximum of 100 mJ at  $t = \frac{1}{4}$  s, and then decreases to zero in another  $\frac{1}{4}$  s. Thus,  $w_{C_{\max}} = 100$  mJ. Evaluating our integral expression for the energy dissipated in the resistor, we find that  $w_R = 2.5$  mJ.



**FIGURE 7.7** A sinusoidal voltage source is applied to a parallel RC network. The  $1 \text{ M}\Omega$  resistor might represent the finite resistance of the “real” capacitor’s dielectric layer.



**FIGURE 7.8** A sketch of the energy stored in the capacitor as a function of time.

(Continued on next page)

► **Verify the solution. Is it reasonable or expected?**

We do not expect to calculate a *negative* stored energy, which is borne out in our sketch. Further, since the maximum value of  $\sin 2\pi t$  is 1, the maximum energy expected anywhere would be  $(1/2)(20 \times 10^{-6})(100)^2 = 100 \text{ mJ}$ .

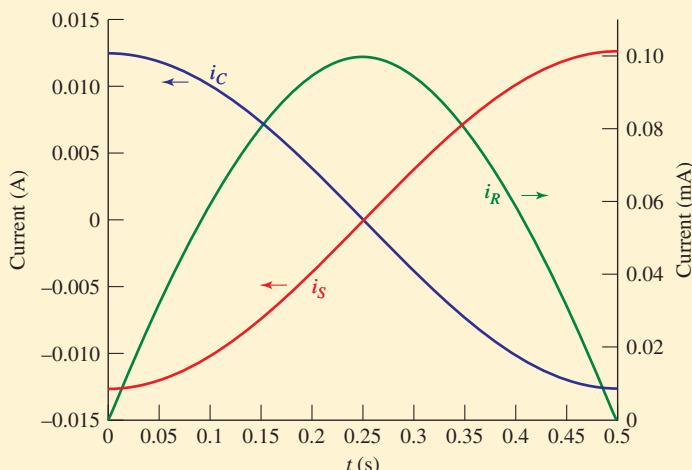
The resistor dissipated 2.5 mJ in the period of 0 to 500 ms, although the capacitor stored a maximum of 100 mJ at one point during that interval. What happened to the “other” 97.5 mJ? To answer this, we compute the capacitor current

$$i_C = 20 \times 10^{-6} \frac{dv}{dt} = 0.004\pi \cos 2\pi t$$

and the current  $i_s$  defined as flowing *into* the voltage source

$$i_s = -i_C - i_R$$

both of which are plotted in Fig. 7.9. We observe that the current flowing through the resistor is a small fraction of the source current, not entirely surprising as  $1 \text{ M}\Omega$  is a relatively large resistance value. As current flows from the source, a small amount is diverted to the resistor, with the rest flowing into the capacitor as it charges. After  $t = 250 \text{ ms}$ , the source current is seen to change sign; current is now flowing from the capacitor back into the source. Most of the energy stored in the capacitor is being returned to the ideal voltage source, except for the small fraction dissipated in the resistor.



■ FIGURE 7.9 Plot of the resistor, capacitor, and source currents during the interval of 0 to 500 ms.

**PRACTICE**

- 7.3 Calculate the energy stored in a  $1000 \mu\text{F}$  capacitor at  $t = 50 \mu\text{s}$  if the voltage across it is  $1.5 \cos 10^5 t$  volts.

Ans:  $90.52 \mu\text{J}$ .

### Important Characteristics of an Ideal Capacitor

1. There is no current through a capacitor if the voltage across it is not changing with time. A capacitor is therefore an *open circuit to dc*.
2. A finite amount of energy can be stored in a capacitor even if the current through the capacitor is zero, such as when the voltage across it is constant.
3. It is impossible to change the voltage across a capacitor by a finite amount in zero time, as this requires an infinite current through the capacitor. (A capacitor resists an abrupt change in the voltage across it in a manner analogous to the way a spring resists an abrupt change in its displacement.)
4. A capacitor never dissipates energy, but only stores it. Although this is true for the *mathematical model*, it is not true for a *physical* capacitor due to finite resistances associated with the dielectric as well as the packaging.

## 7.2 THE INDUCTOR

### Ideal Inductor Model

In the early 1800s the Danish scientist Oersted showed that a current-carrying conductor produced a magnetic field (compass needles were affected in the presence of a wire when current was flowing). Shortly thereafter, Ampère made some careful measurements which demonstrated that this magnetic field was *linearly* related to the current which produced it. The next step occurred some 20 years later when the English experimentalist Michael Faraday and the American inventor Joseph Henry discovered almost simultaneously<sup>3</sup> that a changing magnetic field could induce a voltage in a neighboring circuit. They showed that this voltage was proportional to the time rate of change of the current producing the magnetic field. The constant of proportionality is what we now call the *inductance*, symbolized by  $L$ , and therefore

$$v = L \frac{di}{dt} \quad [5]$$

where we must realize that  $v$  and  $i$  are both functions of time. When we wish to emphasize this, we may do so by using the symbols  $v(t)$  and  $i(t)$ .

The circuit symbol for the inductor is shown in Fig. 7.10, and it should be noted that the passive sign convention is used, just as it was with the resistor and the capacitor. The unit in which inductance is measured is the **henry** (H), and the defining equation shows that the henry is just a shorter expression for a volt-second per ampere.

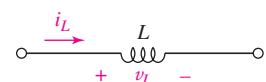


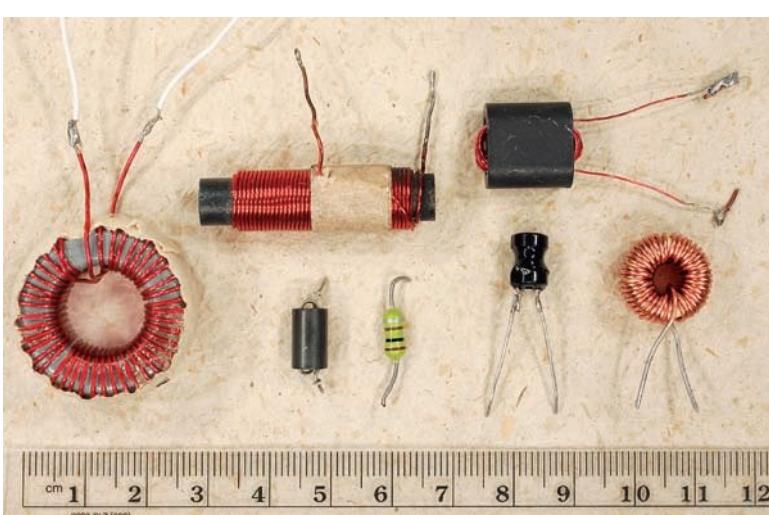
FIGURE 7.10 Electrical symbol and current-voltage conventions for an inductor.

(3) Faraday won.

The inductor whose inductance is defined by Eq. [5] is a mathematical model; it is an *ideal* element which we may use to approximate the behavior of a *real* device. A physical inductor may be constructed by winding a length of wire into a coil. This serves effectively to increase the current that is causing the magnetic field and also to increase the “number” of neighboring circuits into which Faraday’s voltage may be induced. The result of this twofold effect is that the inductance of a coil is approximately proportional to the square of the number of complete turns made by the conductor out of which it is formed. For example, an inductor or “coil” that has the form of a long helix of very small pitch is found to have an inductance of  $\mu N^2 A/s$ , where  $A$  is the cross-sectional area,  $s$  is the axial length of the helix,  $N$  is the number of complete turns of wire, and  $\mu$  (mu) is a constant of the material inside the helix, called the permeability. For free space (and very closely for air),  $\mu = \mu_0 = 4\pi \times 10^{-7} \text{ H/m} = 4\pi \text{ nH/cm}$ . Several examples of commercially available inductors are shown in Fig. 7.11.

Let us now scrutinize Eq. [5] to determine some of the electrical characteristics of the mathematical model. This equation shows that the voltage across an inductor is proportional to the time rate of change of the current through it. In particular, it shows that there is no voltage across an inductor carrying a constant current, regardless of the magnitude of this current. Accordingly, we may view an inductor as a *short circuit to dc*.

Another fact that can be obtained from Eq. [5] is that a sudden or discontinuous change in the current must be associated with an infinite voltage across the inductor. In other words, if we wish to produce an abrupt change in an inductor current, we must apply an infinite voltage. Although an infinite-voltage forcing function might be amusing theoretically, it can never be a part of the phenomena displayed by a real physical device. As we

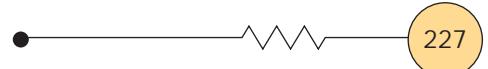


(a)



(b)

**FIGURE 7.11** (a) Several different types of commercially available inductors, sometimes also referred to as “chokes.” Clockwise, starting from far left: 287  $\mu\text{H}$  ferrite core toroidal inductor, 266  $\mu\text{H}$  ferrite core cylindrical inductor, 215  $\mu\text{H}$  ferrite core inductor designed for VHF frequencies, 85  $\mu\text{H}$  powder core toroidal inductor, 10  $\mu\text{H}$  bobbin-style inductor, 100  $\mu\text{H}$  axial lead inductor, and 7  $\mu\text{H}$  lossy-core inductor used for RF suppression. (b) An 11  $\text{H}$  inductor, measuring 10 cm (tall)  $\times$  8 cm (wide)  $\times$  8 cm (deep).



shall see shortly, an abrupt change in the inductor current also requires an abrupt change in the energy stored in the inductor, and this sudden change in energy requires infinite power at that instant; infinite power is again not a part of the real physical world. In order to avoid infinite voltage and infinite power, an inductor current must not be allowed to jump *instantaneously* from one value to another.

If an attempt is made to open-circuit a physical inductor through which a finite current is flowing, an arc may appear across the switch. This is useful in the ignition system of some automobiles, where the current through the spark coil is interrupted by the distributor and the arc appears across the spark plug. Although this does not occur instantaneously, it happens in a very short timespan, leading to the creation of a large voltage. The presence of a large voltage across a short distance equates to a very large electric field; the stored energy is dissipated in ionizing the air in the path of the arc.

Equation [5] may also be interpreted (and solved, if necessary) by graphical methods, as seen in Example 7.4.

### EXAMPLE 7.4

Given the waveform of the current in a 3 H inductor as shown in Fig. 7.12a, determine the inductor voltage and sketch it.

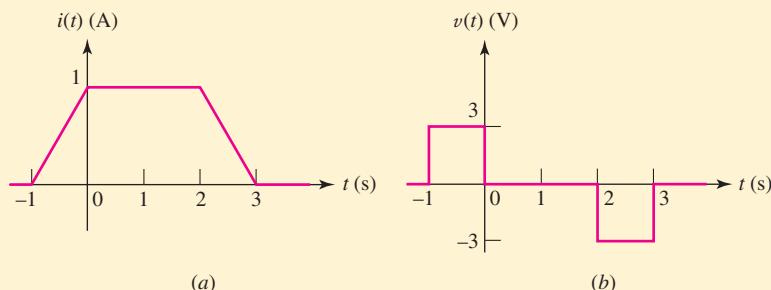


FIGURE 7.12 (a) The current waveform in a 3 H inductor. (b) The corresponding voltage waveform,  $v = 3 di/dt$ .

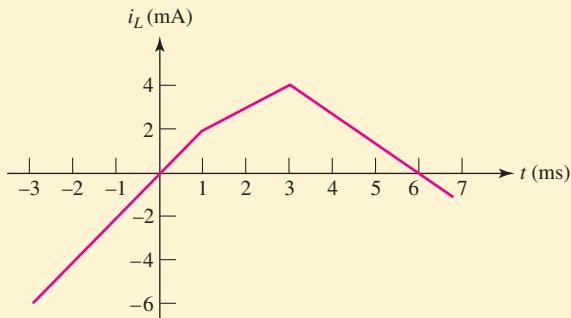
Defining the voltage  $v$  and the current  $i$  to satisfy the passive sign convention, we may obtain  $v$  from Fig. 7.12a using Eq. [5]:

$$v = 3 \frac{di}{dt}$$

Since the current is zero for  $t < -1$  s, the voltage is zero in this interval. The current then begins to increase at the linear rate of 1 A/s, and thus a constant voltage of  $L di/dt = 3$  V is produced. During the following 2 s interval, the current is constant and the voltage is therefore zero. The final decrease of the current results in  $di/dt = -1$  A/s, yielding  $v = -3$  V. For  $t > 3$  s,  $i(t)$  is a constant (zero), so that  $v(t) = 0$  for that interval. The complete voltage waveform is sketched in Fig. 7.12b.

**PRACTICE**

7.4 The current through a 200 mH inductor is shown in Fig. 7.13. Assume the passive sign convention, and find  $v_L$  at  $t$  equal to (a) 0; (b) 2 ms; (c) 6 ms.



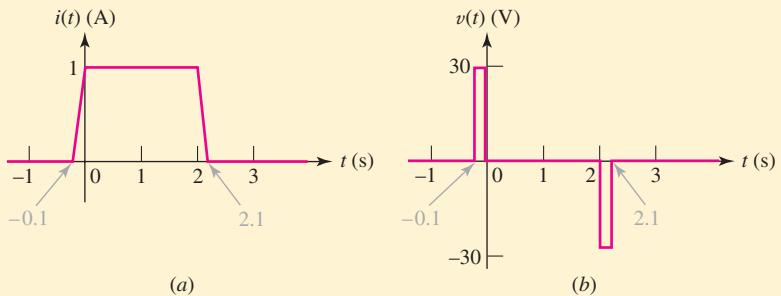
■ FIGURE 7.13

Ans: 0.4 V; 0.2 V; -0.267 V.

Let us now investigate the effect of a more rapid rise and decay of the current between the 0 and 1 A values.

**EXAMPLE 7.5**

Find the inductor voltage that results from applying the current waveform shown in Fig. 7.14a to the inductor of Example 7.4.



■ FIGURE 7.14 (a) The time required for the current of Fig. 7.12a to change from 0 to 1 and from 1 to 0 is decreased by a factor of 10. (b) The resultant voltage waveform. The pulse widths are exaggerated for clarity.

Note that the intervals for the rise and fall have decreased to 0.1 s. Thus, the magnitude of each derivative will be 10 times larger; this condition is shown in the current and voltage sketches of Fig. 7.14a and b. In the voltage waveforms of Fig. 7.13b and 7.14b, it is interesting to note that the area under each voltage pulse is 3 V · s.

Just for curiosity's sake, let's continue in the same vein for a moment. A further decrease in the rise and fall times of the current waveform will produce a proportionally larger voltage magnitude, but only within the interval

in which the current is increasing or decreasing. An abrupt change in the current will cause the infinite voltage “spikes” (each having an area of  $3 \text{ V} \cdot \text{s}$ ) that are suggested by the waveforms of Fig. 7.15a and b; or, from the equally valid but opposite point of view, these infinite voltage spikes are required to produce the abrupt changes in the current.

### PRACTICE

- 7.5 The current waveform of Fig. 7.14a has equal rise and fall times of duration 0.1 s (100 ms). Calculate the maximum positive and negative voltages across the same inductor if the rise and fall times, respectively, are changed to (a) 1 ms, 1 ms; (b) 12  $\mu\text{s}$ , 64  $\mu\text{s}$ ; (c) 1 s, 1 ns.

Ans: 3 kV, -3 kV; 250 kV, -46.88 kV; 3 V, -3 GV.

## Integral Voltage-Current Relationships

We have defined inductance by a simple differential equation,

$$v = L \frac{di}{dt}$$

and we have been able to draw several conclusions about the characteristics of an inductor from this relationship. For example, we have found that we may consider an inductor to be a short circuit to direct current, and we have agreed that we cannot permit an inductor current to change abruptly from one value to another, because this would require that an infinite voltage and power be associated with the inductor. The simple defining equation for inductance contains still more information, however. Rewritten in a slightly different form,

$$di = \frac{1}{L} v dt$$

it invites integration. Let us first consider the limits to be placed on the two integrals. We desire the current  $i$  at time  $t$ , and this pair of quantities therefore provides the upper limits on the integrals appearing on the left and right sides of the equation, respectively; the lower limits may also be kept general by merely assuming that the current is  $i(t_0)$  at time  $t_0$ . Thus,

$$\int_{i(t_0)}^{i(t)} di' = \frac{1}{L} \int_{t_0}^t v(t') dt'$$

which leads to the equation

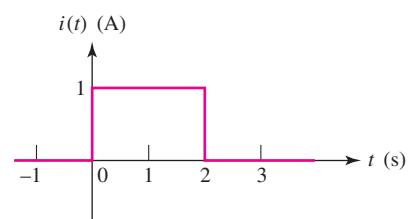
$$i(t) - i(t_0) = \frac{1}{L} \int_{t_0}^t v dt'$$

or

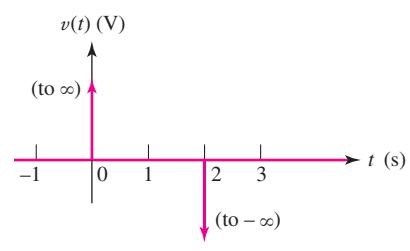
$$i(t) = \frac{1}{L} \int_{t_0}^t v dt' + i(t_0)$$

[6]

Equation [5] expresses the inductor voltage in terms of the current, whereas Eq. [6] gives the current in terms of the voltage. Other forms are



(a)



(b)

FIGURE 7.15 (a) The time required for the current of Fig. 7.14a to change from 0 to 1 and from 1 to 0 is decreased to zero; the rise and fall are abrupt. (b) The resultant voltage across the 3 H inductor consists of a positive and a negative infinite spike.

also possible for the latter equation. We may write the integral as an indefinite integral and include a constant of integration  $k$ :

$$i(t) = \frac{1}{L} \int v dt + k \quad [7]$$

We also may assume that we are solving a realistic problem in which the selection of  $t_0$  as  $-\infty$  ensures no current or energy in the inductor. Thus, if  $i(t_0) = i(-\infty) = 0$ , then

$$i(t) = \frac{1}{L} \int_{-\infty}^t v dt' \quad [8]$$

Let us investigate the use of these several integrals by working a simple example where the voltage across an inductor is specified.

### EXAMPLE 7.6

**The voltage across a 2 H inductor is known to be  $6 \cos 5t$  V. Determine the resulting inductor current if  $i(t = -\pi/2) = 1$  A.**

From Eq. [6],

$$i(t) = \frac{1}{2} \int_{t_0}^t 6 \cos 5t' dt' + i(t_0)$$

or

$$\begin{aligned} i(t) &= \frac{1}{2} \left( \frac{6}{5} \right) \sin 5t - \frac{1}{2} \left( \frac{6}{5} \right) \sin 5t_0 + i(t_0) \\ &= 0.6 \sin 5t - 0.6 \sin 5t_0 + i(t_0) \end{aligned}$$

The first term indicates that the inductor current varies sinusoidally; the second and third terms together represent a constant which becomes known when the current is numerically specified at some instant of time. Using the fact that the current is 1 A at  $t = -\pi/2$  s, we identify  $t_0$  as  $-\pi/2$  with  $i(t_0) = 1$ , and find that

$$i(t) = 0.6 \sin 5t - 0.6 \sin(-2.5\pi) + 1$$

or

$$i(t) = 0.6 \sin 5t + 1.6$$

Alternatively, from Eq. [6],

$$i(t) = 0.6 \sin 5t + k$$

and we establish the numerical value of  $k$  by forcing the current to be 1 A at  $t = -\pi/2$ :

$$1 = 0.6 \sin(-2.5\pi) + k$$

or

$$k = 1 + 0.6 = 1.6$$

and so, as before,

$$i(t) = 0.6 \sin 5t + 1.6$$

Equation [8] is going to cause trouble with this particular voltage. We based the equation on the assumption that the current was zero when  $t = -\infty$ . To be sure, this must be true in the real, physical world, but we are working in the land of the mathematical model; our elements and forcing functions are all idealized. The difficulty arises after we integrate, obtaining

$$i(t) = 0.6 \sin 5t' \Big|_{-\infty}^t$$

and attempt to evaluate the integral at the lower limit:

$$i(t) = 0.6 \sin 5t - 0.6 \sin(-\infty)$$

The sine of  $\pm\infty$  is indeterminate, and therefore we cannot evaluate our expression. Equation [8] is only useful if we are evaluating functions which approach zero as  $t \rightarrow -\infty$ .

### PRACTICE

7.6 A 100 mH inductor has voltage  $v_L = 2e^{-3t}$  V across its terminals. Determine the resulting inductor current if  $i_L(-0.5) = 1$  A.

Ans:  $-\frac{20}{3}e^{-3t} + 30.9$  A.



We should not make any snap judgments, however, as to which single form of Eqs. [6], [7], and [8] we are going to use forever after; each has its advantages, depending on the problem and the application. Equation [6] represents a long, general method, but it shows clearly that the constant of integration is a current. Equation [7] is a somewhat more concise expression of Eq. [6], but the nature of the integration constant is suppressed. Finally, Eq. [8] is an excellent expression, since no constant is necessary; however, it applies only when the current is zero at  $t = -\infty$  and when the analytical expression for the current is not indeterminate there.

### Energy Storage

Let us now turn our attention to power and energy. The absorbed power is given by the current-voltage product

$$p = vi = Li \frac{di}{dt}$$

The energy  $w_L$  accepted by the inductor is stored in the magnetic field around the coil. The change in this energy is expressed by the integral of the power over the desired time interval:

$$\begin{aligned} \int_{t_0}^t p dt' &= L \int_{t_0}^t i \frac{di}{dt'} dt' = L \int_{i(t_0)}^{i(t)} i' di' \\ &= \frac{1}{2} L \{ [i(t)]^2 - [i(t_0)]^2 \} \end{aligned}$$

Thus,

$$w_L(t) - w_L(t_0) = \frac{1}{2} L \{ [i(t)]^2 - [i(t_0)]^2 \} \quad [9]$$

where we have again assumed that the current is  $i(t_0)$  at time  $t_0$ . In using the energy expression, it is customary to assume that a value of  $t_0$  is selected at which the current is zero; it is also customary to assume that the energy is zero at this time. We then have simply

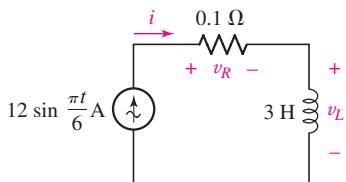
$$w_L(t) = \frac{1}{2}Li^2$$

[10]

where we now understand that our reference for zero energy is any time at which the inductor current is zero. At any subsequent time at which the current is zero, we also find no energy stored in the coil. Whenever the current is not zero, and regardless of its direction or sign, energy is stored in the inductor. It follows, therefore, that energy may be delivered to the inductor for a part of the time and recovered from the inductor later. All the stored energy may be recovered from an ideal inductor; there are no storage charges or agent's commissions in the mathematical model. A physical coil, however, must be constructed out of real wire and thus will always have a resistance associated with it. Energy can no longer be stored and recovered without loss.

These ideas may be illustrated by a simple example. In Fig. 7.16, a 3 H inductor is shown in series with a  $0.1\ \Omega$  resistor and a sinusoidal current source,  $i_s = 12 \sin \frac{\pi t}{6}\text{ A}$ . The resistor should be interpreted as the resistance of the wire which must be associated with the physical coil.

### EXAMPLE 7.7



**FIGURE 7.16** A sinusoidal current is applied as a forcing function to a series  $RL$  circuit. The  $0.1\ \Omega$  represents the inherent resistance of the wire from which the inductor is fabricated.

**Find the maximum energy stored in the inductor of Fig. 7.16, and calculate how much energy is dissipated in the resistor in the time during which the energy is being stored in, and then recovered from, the inductor.**

The energy stored in the inductor is

$$w_L = \frac{1}{2}Li^2 = 216 \sin^2 \frac{\pi t}{6}\text{ J}$$

and this energy increases from zero at  $t = 0$  to 216 J at  $t = 3$  s. Thus, the maximum energy stored in the inductor is 216 J.

After reaching its peak value at  $t = 3$  s, the energy has completely left the inductor 3 s later. Let us see what price we have paid in this coil for the privilege of storing and removing 216 J in these 6 seconds. The power dissipated in the resistor is easily found as

$$p_R = i^2R = 14.4 \sin^2 \frac{\pi t}{6}\text{ W}$$

and the energy converted into heat in the resistor within this 6 s interval is therefore

$$w_R = \int_0^6 p_R dt = \int_0^6 14.4 \sin^2 \frac{\pi}{6} t dt$$

or

$$w_R = \int_0^6 14.4 \left(\frac{1}{2}\right) \left(1 - \cos \frac{\pi}{3}t\right) dt = 43.2 \text{ J}$$

Thus, we have expended 43.2 J in the process of storing and then recovering 216 J in a 6 s interval. This represents 20 percent of the maximum stored energy, but it is a reasonable value for many coils having this large an inductance. For coils having an inductance of about 100  $\mu\text{H}$ , we might expect a figure closer to 2 or 3 percent.

### PRACTICE

- 7.7 Let  $L = 25 \text{ mH}$  for the inductor of Fig. 7.10. (a) Find  $v_L$  at  $t = 12 \text{ ms}$  if  $i_L = 10te^{-100t} \text{ A}$ . (b) Find  $i_L$  at  $t = 0.1 \text{ s}$  if  $v_L = 6e^{-12t} \text{ V}$  and  $i_L(0) = 10 \text{ A}$ . If  $i_L = 8(1 - e^{-40t}) \text{ mA}$ , find (c) the power being delivered to the inductor at  $t = 50 \text{ ms}$  and (d) the energy stored in the inductor at  $t = 40 \text{ ms}$ .

Ans:  $-15.06 \text{ mV}$ ;  $24.0 \text{ A}$ ;  $7.49 \mu\text{W}$ ;  $0.510 \mu\text{J}$ .

We summarize by listing four key characteristics of an inductor which result from its defining equation  $v = L di/dt$ :

#### Important Characteristics of an Ideal Inductor

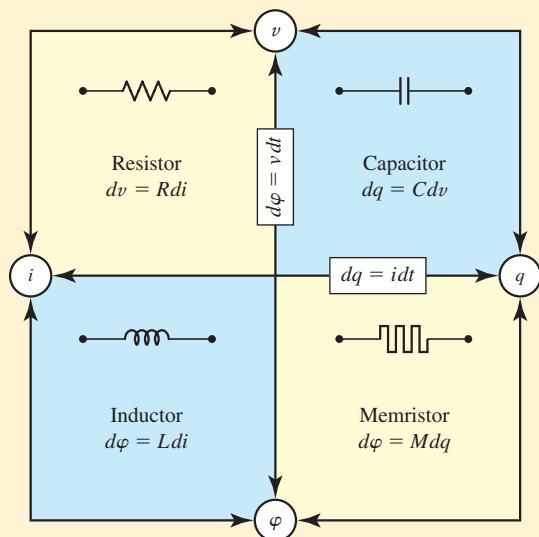
1. There is no voltage across an inductor if the current through it is not changing with time. An inductor is therefore a *short circuit to dc*.
2. A finite amount of energy can be stored in an inductor even if the voltage across the inductor is zero, such as when the current through it is constant.
3. It is impossible to change the current through an inductor by a finite amount in zero time, for this requires an infinite voltage across the inductor. (An inductor resists an abrupt change in the current through it in a manner analogous to the way a mass resists an abrupt change in its velocity.)
4. The inductor never dissipates energy, but only stores it. Although this is true for the *mathematical model*, it is not true for a *physical inductor* due to series resistances.

It is interesting to anticipate our discussion of ***duality*** in Sec. 7.6 by rereading the previous four statements with certain words replaced by their “duals.” If *capacitor* and *inductor*, *capacitance* and *inductance*, *voltage* and *current*, *across* and *through*, *open circuit* and *short circuit*, *spring* and *mass*, and *displacement* and *velocity* are interchanged (in either direction), the four statements previously given for capacitors are obtained.

## PRACTICAL APPLICATION

### In Search of the Missing Element

So far, we have been introduced to three different two-terminal passive elements: the *resistor*, the *capacitor*, and the *inductor*. Each has been defined in terms of its current-voltage relationship ( $v = Ri$ ,  $i = C dv/dt$ , and  $v = L di/dt$ , respectively). From a more fundamental perspective, however, we can view these three elements as part of a larger picture relating four basic quantities, namely, charge  $q$ , current  $i$ , voltage  $v$ , and flux linkage  $\varphi$ . Charge, current, and voltage are discussed in Chap. 2. Flux linkage is the product of magnetic flux and the number of turns of conducting wire linked by the flux, and it can be expressed in terms of the voltage  $v$  across the coil as  $\varphi = \int v dt$  or  $v = d\varphi/dt$ .



■ FIGURE 7.17 A graphical representation of the four basic two-terminal passive elements (resistor, capacitor, inductor, and memristor) and their interrelationships. Note that flux linkage is more commonly represented by the Greek letter  $\lambda$  to distinguish it from flux: then  $\lambda = N\varphi$  where  $N$  is the number of turns and  $\varphi$  is the flux. (Reprinted by permission from Macmillan Publishers Ltd. Nature Publishing Group, "Electronics: The Fourth Element," Volume 453, pg. 42, 2008.)

Figure 7.17 graphically represents how these four quantities are interrelated. First, apart from any circuit elements and their characteristics, we have  $dq = i dt$  (Chap. 2) and now  $d\varphi = v dt$ . Charge is related to voltage in the context of a capacitor, since  $C = dq/dv$  or  $dq = C dv$ . The element we call a resistor provides a

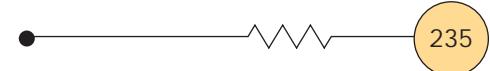
direct relationship between voltage and current, which for consistency can be expressed as  $dv = R di$ . Continuing our counterclockwise journey around the perimeter of Fig. 7.17, we note that our original expression connecting the voltage and current associated with an inductor can be written in terms of current  $i$  and flux linkage  $\varphi$ , since rearranging yields  $v dt = L di$ , and we know  $d\varphi = v dt$ . Thus, for the inductor, we can write  $d\varphi = L di$ .

So far, we have traveled from  $q$  to  $v$  with the aid of a capacitor,  $v$  to  $i$  using the resistor, and  $i$  to  $\varphi$  using the inductor. However, we have not yet used any element to connect  $\varphi$  and  $q$ , although symmetry suggests such a thing should be possible. In the early 1970s, Leon Chua found himself thinking along these lines, and postulated a new device—a “missing” two-terminal circuit element—and named it the **memristor**.<sup>1</sup> He went on to demonstrate that the electrical characteristics of a memristor should be nonlinear, and depend on its history—in other words, a memristor might be characterized by having a memory (hence its name). Independent of his work, others had proposed a similar device, not so much for its practical use in real circuits, but for its potential in device modeling and signal processing.

Not a great deal was heard of this hypothetical element afterward, at least until Dmitri Strukov and coworkers at HP Labs in Palo Alto published a short paper in 2008 claiming to have “found” the memristor.<sup>2</sup> They offer several reasons why it took almost four decades to realize the general type of device Chua hypothesized in 1971, but one of the most interesting has to do with size. In making their prototype memristor, *nanotechnology* (the art of fabricating devices with at least one dimension less than 1000 nm, which is approximately 1% of the diameter of human hair) played a key role. A 5 nm thick oxide layer sandwiched between platinum electrodes comprises the entire device. The nonlinear electrical characteristics of the prototype immediately generated considerable excitement, most notably for its potential applications in integrated circuits, where devices are already approaching their smallest realistic size; and many believe new types of devices will be required to further extend integrated circuit density and functionality. Whether the memristor is the circuit element that will allow this remains to be seen—despite the report of a prototype, there remains much work to be done before it becomes practical.

(1) L. O. Chua, “Memristor—The missing circuit element,” *IEEE Transactions on Circuit Theory* CT-18 (5), 1971, p. 507.

(2) D. B. Strukov, G. S. Snider, D. R. Stewart, and R. S. Williams, “The missing memristor found,” *Nature* 453, 2008, p. 80.



## 7.3 INDUCTANCE AND CAPACITANCE COMBINATIONS

Now that we have added the inductor and capacitor to our list of passive circuit elements, we need to decide whether or not the methods we have developed for resistive circuit analysis are still valid. It will also be convenient to learn how to replace series and parallel combinations of either of these elements with simpler equivalents, just as we did with resistors in Chap. 3.

We look first at Kirchhoff's two laws, both of which are axiomatic. However, when we hypothesized these two laws, we did so with no restrictions as to the types of elements constituting the network. Both, therefore, remain valid.

### Inductors in Series

Now we may extend the procedures we have derived for reducing various combinations of resistors into one equivalent resistor to the analogous cases of inductors and capacitors. We shall first consider an ideal voltage source applied to the series combination of  $N$  inductors, as shown in Fig. 7.18a. We desire a single equivalent inductor, with inductance  $L_{\text{eq}}$ , which may replace the series combination so that the source current  $i(t)$  is unchanged. The equivalent circuit is sketched in Fig. 7.18b. Applying KVL to the original circuit,

$$\begin{aligned} v_s &= v_1 + v_2 + \cdots + v_N \\ &= L_1 \frac{di}{dt} + L_2 \frac{di}{dt} + \cdots + L_N \frac{di}{dt} \\ &= (L_1 + L_2 + \cdots + L_N) \frac{di}{dt} \end{aligned}$$

or, written more concisely,

$$v_s = \sum_{n=1}^N v_n = \sum_{n=1}^N L_n \frac{di}{dt} = \frac{di}{dt} \sum_{n=1}^N L_n$$

But for the equivalent circuit we have

$$v_s = L_{\text{eq}} \frac{di}{dt}$$

and thus the equivalent inductance is

$$L_{\text{eq}} = L_1 + L_2 + \cdots + L_N$$

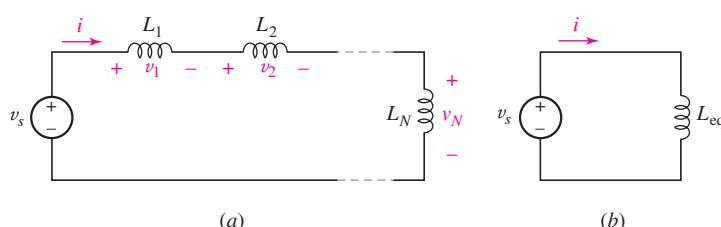
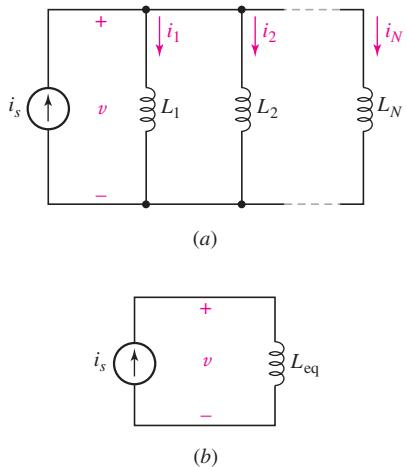


FIGURE 7.18 (a) A circuit containing  $N$  inductors in series. (b) The desired equivalent circuit, in which  $L_{\text{eq}} = L_1 + L_2 + \cdots + L_N$ .

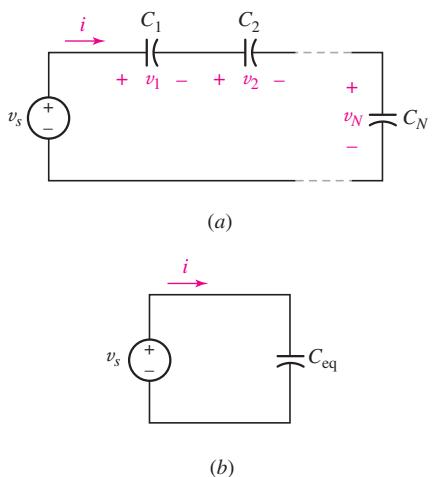
or

$$L_{\text{eq}} = \sum_{n=1}^N L_n \quad [11]$$

The inductor which is equivalent to several inductors connected in series is one whose inductance is the sum of the inductances in the original circuit. This is exactly the same result we obtained for resistors in series.



**FIGURE 7.19** (a) The parallel combination of  $N$  inductors. (b) The equivalent circuit, where  $L_{\text{eq}} = [1/L_1 + 1/L_2 + \dots + 1/L_N]^{-1}$ .



**FIGURE 7.20** (a) A circuit containing  $N$  capacitors in series. (b) The desired equivalent circuit, where  $C_{\text{eq}} = [1/C_1 + 1/C_2 + \dots + 1/C_N]^{-1}$ .

### Inductors in Parallel

The combination of a number of parallel inductors is accomplished by writing the single nodal equation for the original circuit, shown in Fig. 7.19a,

$$\begin{aligned} i_s &= \sum_{n=1}^N i_n = \sum_{n=1}^N \left[ \frac{1}{L_n} \int_{t_0}^t v dt' + i_n(t_0) \right] \\ &= \left( \sum_{n=1}^N \frac{1}{L_n} \right) \int_{t_0}^t v dt' + \sum_{n=1}^N i_n(t_0) \end{aligned}$$

and comparing it with the result for the equivalent circuit of Fig. 7.19b,

$$i_s = \frac{1}{L_{\text{eq}}} \int_{t_0}^t v dt' + i_s(t_0)$$

Since Kirchhoff's current law demands that  $i_s(t_0)$  be equal to the sum of the branch currents at  $t_0$ , the two integral terms must also be equal; hence,

$$L_{\text{eq}} = \frac{1}{1/L_1 + 1/L_2 + \dots + 1/L_N} \quad [12]$$

For the special case of two inductors in parallel,

$$L_{\text{eq}} = \frac{L_1 L_2}{L_1 + L_2} \quad [13]$$

and we note that inductors in parallel combine exactly as do resistors in parallel.

### Capacitors in Series

In order to find a capacitor that is equivalent to  $N$  capacitors in series, we use the circuit of Fig. 7.20a and its equivalent in Fig. 7.20b to write

$$\begin{aligned} v_s &= \sum_{n=1}^N v_n = \sum_{n=1}^N \left[ \frac{1}{C_n} \int_{t_0}^t i dt' + v_n(t_0) \right] \\ &= \left( \sum_{n=1}^N \frac{1}{C_n} \right) \int_{t_0}^t i dt' + \sum_{n=1}^N v_n(t_0) \end{aligned}$$

and

$$v_s = \frac{1}{C_{\text{eq}}} \int_{t_0}^t i dt' + v_s(t_0)$$

However, Kirchhoff's voltage law establishes the equality of  $v_s(t_0)$  and the sum of the capacitor voltages at  $t_0$ ; thus

$$C_{\text{eq}} = \frac{1}{1/C_1 + 1/C_2 + \dots + 1/C_N} \quad [14]$$

and capacitors in series combine as do conductances in series, or resistors in parallel. The special case of two capacitors in series, of course, yields

$$C_{\text{eq}} = \frac{C_1 C_2}{C_1 + C_2} \quad [15]$$

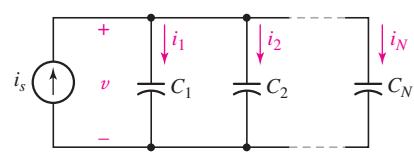
## Capacitors in Parallel

Finally, the circuits of Fig. 7.21 enable us to establish the value of the capacitor which is equivalent to  $N$  parallel capacitors as

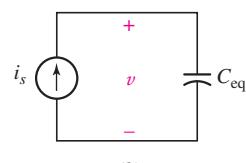
$$C_{\text{eq}} = C_1 + C_2 + \dots + C_N \quad [16]$$

and it is no great source of amazement to note that capacitors in parallel combine in the same manner in which we combine resistors in series, that is, by simply adding all the individual capacitances.

These formulas are well worth memorizing. The formulas applying to series and parallel combinations of inductors are identical to those for resistors, so they typically seem “obvious.” Care should be exercised, however, in the case of the corresponding expressions for series and parallel combinations of capacitors, as they are opposite those of resistors and inductors, frequently leading to errors when calculations are made too hastily.



(a)

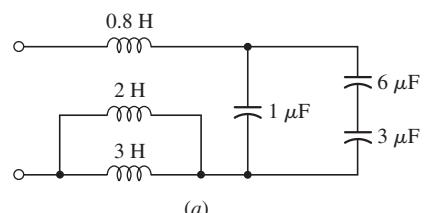


(b)

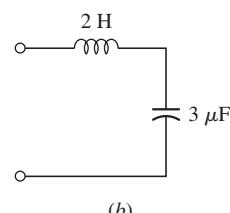
FIGURE 7.21 (a) The parallel combination of  $N$  capacitors. (b) The equivalent circuit, where  $C_{\text{eq}} = C_1 + C_2 + \dots + C_N$ .



## EXAMPLE 7.8



(a)



(b)

FIGURE 7.22 (a) A given LC network. (b) A simpler equivalent circuit.

### Simplify the network of Fig. 7.22a using series-parallel combinations.

The  $6 \mu\text{F}$  and  $3 \mu\text{F}$  series capacitors are first combined into a  $2 \mu\text{F}$  equivalent, and this capacitor is then combined with the  $1 \mu\text{F}$  element with which it is in parallel to yield an equivalent capacitance of  $3 \mu\text{F}$ . In addition, the  $3 \text{ H}$  and  $2 \text{ H}$  inductors are replaced by an equivalent  $1.2 \text{ H}$  inductor, which is then added to the  $0.8 \text{ H}$  element to give a total equivalent inductance of  $2 \text{ H}$ . The much simpler (and probably less expensive) equivalent network is shown in Fig. 7.22b.

### PRACTICE

7.8 Find  $C_{\text{eq}}$  for the network of Fig. 7.23.

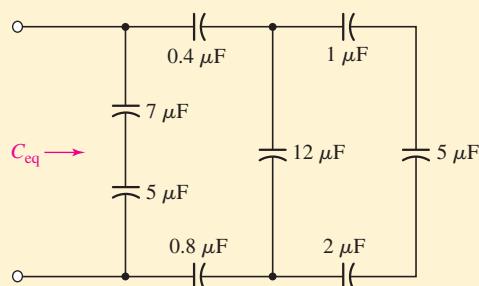
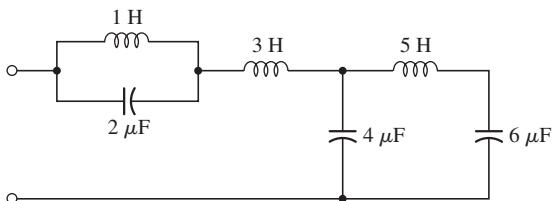


FIGURE 7.23

Ans:  $3.18 \mu\text{F}$ .

The network shown in Fig. 7.24 contains three inductors and three capacitors, but no series or parallel combinations of either the inductors or the capacitors can be achieved. Simplification of this network cannot be accomplished using the techniques presented here.

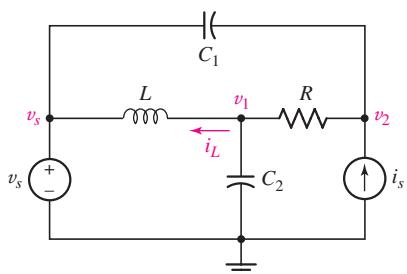


■ FIGURE 7.24 An LC network in which no series or parallel combinations of either the inductors or the capacitors can be made.

## 7.4 CONSEQUENCES OF LINEARITY

Next let us turn to nodal and mesh analysis. Since we already know that we may safely apply Kirchhoff's laws, we can apply them in writing a set of equations that are both sufficient and independent. They will be constant-coefficient linear integrodifferential equations, however, which are hard enough to pronounce, let alone solve. Consequently, we shall write them now to gain familiarity with the use of Kirchhoff's laws in RLC circuits and discuss the solution of the simpler cases in subsequent chapters.

### EXAMPLE 7.9



■ FIGURE 7.25 A four-node RLC circuit with node voltages assigned.

#### Write appropriate nodal equations for the circuit of Fig. 7.25.

Node voltages are already chosen, so we sum currents leaving the central node:

$$\frac{1}{L} \int_{t_0}^t (v_1 - v_s) dt' + i_L(t_0) + \frac{v_1 - v_2}{R} + C_2 \frac{dv_1}{dt} = 0$$

where  $i_L(t_0)$  is the value of the inductor current at the time the integration begins. At the right-hand node,

$$C_1 \frac{d(v_2 - v_s)}{dt} + \frac{v_2 - v_1}{R} - i_s = 0$$

Rewriting these two equations, we have

$$\begin{aligned} \frac{v_1}{R} + C_2 \frac{dv_1}{dt} + \frac{1}{L} \int_{t_0}^t v_1 dt' - \frac{v_2}{R} &= \frac{1}{L} \int_{t_0}^t v_s dt' - i_L(t_0) \\ -\frac{v_1}{R} + \frac{v_2}{R} + C_1 \frac{dv_2}{dt} &= C_1 \frac{dv_s}{dt} + i_s \end{aligned}$$

These are the promised integrodifferential equations, and we note several interesting points about them. First, the source voltage  $v_s$  happens to enter the equations as an integral and as a derivative, but not simply as  $v_s$ . Since both sources are specified for all time, we should be able to evaluate the derivative or integral. Second, the initial value of the inductor current,  $i_L(t_0)$ , acts as a (constant) source current at the center node.

**PRACTICE**

7.9 If  $v_C(t) = 4 \cos 10^5 t$  V in the circuit in Fig. 7.26, find  $v_s(t)$ .

Ans:  $-2.4 \cos 10^5 t$  V.

We will not attempt the solution of integrodifferential equations here. It is worthwhile pointing out, however, that when the voltage forcing functions are sinusoidal functions of time, it will be possible to define a voltage-current ratio (called **impedance**) or a current-voltage ratio (called **admittance**) for each of the three passive elements. The factors operating on the two node voltages in the preceding equations will then become simple multiplying factors, and the equations will be linear algebraic equations once again. These we may solve by determinants or a simple elimination of variables as before.

We may also show that the benefits of linearity apply to *RLC* circuits as well. In accordance with our previous definition of a linear circuit, these circuits are also linear because the voltage-current relationships for the inductor and capacitor are linear relationships. For the inductor, we have

$$v = L \frac{di}{dt}$$

and multiplication of the current by some constant  $K$  leads to a voltage that is also greater by a factor  $K$ . In the integral formulation,

$$i(t) = \frac{1}{L} \int_{t_0}^t v dt' + i(t_0)$$

it can be seen that, if each term is to increase by a factor of  $K$ , then the initial value of the current must also increase by this same factor.

A corresponding investigation of the capacitor shows that it, too, is linear. Thus, a circuit composed of independent sources, linear dependent sources, and linear resistors, inductors, and capacitors is a linear circuit.

In this linear circuit the response is again proportional to the forcing function. The proof of this statement is accomplished by first writing a general system of integrodifferential equations. Let us place all the terms having the form of  $Ri$ ,  $L di/dt$ , and  $1/C \int i dt$  on the left side of each equation, and keep the independent source voltages on the right side. As a simple example, one of the equations might have the form

$$Ri + L \frac{di}{dt} + \frac{1}{C} \int_{t_0}^t i dt' + v_C(t_0) = v_s$$

If every independent source is now increased by a factor  $K$ , then the right side of each equation is greater by the factor  $K$ . Now each term on the left side is either a linear term involving some loop current or an initial capacitor voltage. In order to cause all the responses (loop currents) to increase by a factor  $K$ , it is apparent that we must also increase the initial capacitor voltages by a factor  $K$ . That is, we must treat the initial capacitor voltage as an independent source voltage and increase it also by a factor  $K$ . In a similar manner, initial inductor currents appear as independent source currents in nodal analysis.

The principle of proportionality between source and response can thus be extended to the general *RLC* circuit, and it follows that the principle of superposition also applies. It should be emphasized that initial inductor

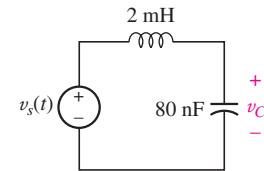


FIGURE 7.26

currents and capacitor voltages must be treated as independent sources in applying the superposition principle; each initial value must take its turn in being rendered inactive. In Chap. 5 we learned that the principle of superposition is a natural consequence of the linear nature of resistive circuits. The resistive circuits are linear because the voltage-current relationship for the resistor is linear and Kirchhoff's laws are linear.

Before we can apply the superposition principle to *RLC* circuits, however, it is first necessary to develop methods of solving the equations describing these circuits when only one independent source is present. At this time we should feel convinced that a linear circuit will possess a response whose amplitude is proportional to the amplitude of the source. We should be prepared to apply superposition later, considering an inductor current or capacitor voltage specified at  $t = t_0$  as a source that must be deactivated when its turn comes.

Thévenin's and Norton's theorems are based on the linearity of the initial circuit, the applicability of Kirchhoff's laws, and the superposition principle. The general *RLC* circuit conforms perfectly to these requirements, and it follows, therefore, that all linear circuits that contain any combinations of independent voltage and current sources, linear dependent voltage and current sources, and linear resistors, inductors, and capacitors may be analyzed with the use of these two theorems, if we wish.

## 7.5 SIMPLE OP AMP CIRCUITS WITH CAPACITORS

In Chap. 6 we were introduced to several different types of amplifier circuits based on the ideal op amp. In almost every case, we found that the output was related to the input voltage by some combination of resistance ratios. If we replace one or more of these resistors with a capacitor, it is possible to obtain some interesting circuits in which the output is proportional to either the derivative or integral of the input voltage. Such circuits find widespread use in practice. For example, a velocity sensor can be connected to an op amp circuit that provides a signal proportional to the acceleration, or an output signal can be obtained that represents the total charge incident on a metal electrode during a specific period of time by simply integrating the measured current.

To create an integrator using an ideal op amp, we ground the noninverting input, install an ideal capacitor as a feedback element from the output back to the inverting input, and connect a signal source  $v_s$  to the inverting input through an ideal resistor as shown in Fig. 7.27.

Performing nodal analysis at the inverting input,

$$0 = \frac{v_a - v_s}{R_1} + i$$

We can relate the current  $i$  to the voltage across the capacitor,

$$i = C_f \frac{dv_{C_f}}{dt}$$

resulting in

$$0 = \frac{v_a - v_s}{R_1} + C_f \frac{dv_{C_f}}{dt}$$

Invoking ideal op amp rule 2, we know that  $v_a = v_b = 0$ , so

$$0 = \frac{-v_s}{R_1} + C_f \frac{dv_{C_f}}{dt}$$

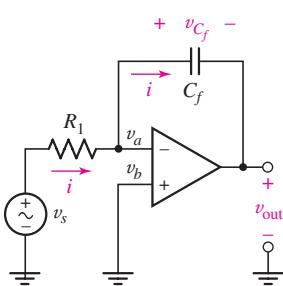


FIGURE 7.27 An ideal op amp connected as an integrator.

Integrating and solving for  $v_{\text{out}}$ , we obtain

$$v_{C_f} = v_a - v_{\text{out}} = 0 - v_{\text{out}} = \frac{1}{R_1 C_f} \int_0^t v_s dt' + v_{C_f}(0)$$

or

$$v_{\text{out}} = -\frac{1}{R_1 C_f} \int_0^t v_s dt' - v_{C_f}(0) \quad [17]$$

We therefore have combined a resistor, a capacitor, and an op amp to form an integrator. Note that the first term of the output is  $1/RC$  times the negative of the integral of the input from  $t' = 0$  to  $t$ , and the second term is the negative of the initial value of  $v_{C_f}$ . The value of  $(RC)^{-1}$  can be made equal to unity, if we wish, by choosing  $R = 1 \text{ M}\Omega$  and  $C = 1 \mu\text{F}$ , for example; other selections may be made that will increase or decrease the output voltage.

Before we leave the integrator circuit, we might anticipate a question from an inquisitive reader, “*Could we use an inductor in place of the capacitor and obtain a differentiator?*” Indeed we could, but circuit designers usually avoid the use of inductors whenever possible because of their size, weight, cost, and associated resistance and capacitance. Instead, it is possible to interchange the positions of the resistor and capacitor in Fig. 7.27 and obtain a differentiator.

## EXAMPLE 7.10

Derive an expression for the output voltage of the op amp circuit shown in Fig. 7.28.

We begin by writing a nodal equation at the inverting input pin, with  $v_{C_1} \triangleq v_a - v_s$ :

$$0 = C_1 \frac{dv_{C_1}}{dt} + \frac{v_a - v_{\text{out}}}{R_f}$$

Invoking ideal op amp rule 2,  $v_a = v_b = 0$ . Thus,

$$C_1 \frac{dv_{C_1}}{dt} = \frac{v_{\text{out}}}{R_f}$$

Solving for  $v_{\text{out}}$ ,

$$v_{\text{out}} = R_f C_1 \frac{dv_{C_1}}{dt}$$

Since  $v_{C_1} = v_a - v_s = -v_s$ ,

$$v_{\text{out}} = -R_f C_1 \frac{dv_s}{dt}$$

So, simply by swapping the resistor and capacitor in the circuit of Fig. 7.27, we obtain a differentiator instead of an integrator.

## PRACTICE

7.10 Derive an expression for  $v_{\text{out}}$  in terms of  $v_s$  for the circuit shown in Fig. 7.29.

Ans:  $v_{\text{out}} = -L_f / R_1 dv_s / dt$ .

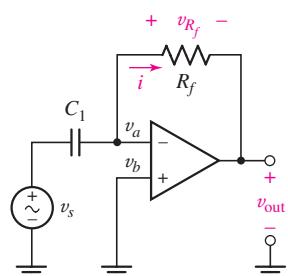


FIGURE 7.28 An ideal op amp connected as a differentiator.

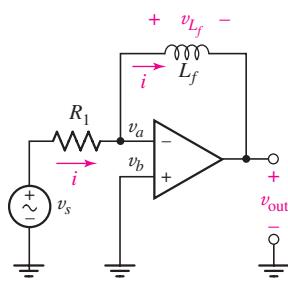


FIGURE 7.29

## 7.6 DUALITY

The concept of **duality** applies to many fundamental engineering concepts. In this section, we shall define duality in terms of the circuit equations. Two circuits are “duals” if the mesh equations that characterize one of them have the *same mathematical form* as the nodal equations that characterize the other. They are said to be exact duals if each mesh equation of one circuit is numerically identical with the corresponding nodal equation of the other; the current and voltage variables themselves cannot be identical, of course. Duality itself merely refers to any of the properties exhibited by dual circuits.

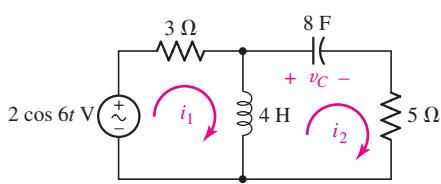


FIGURE 7.30 A given circuit to which the definition of duality may be applied to determine the dual circuit. Note that  $v_c(0) = 10$  V.

Let us use the definition to construct an exact dual circuit by writing the two mesh equations for the circuit shown in Fig. 7.30. Two mesh currents  $i_1$  and  $i_2$  are assigned, and the mesh equations are

$$3i_1 + 4 \frac{di_1}{dt} - 4 \frac{di_2}{dt} = 2 \cos 6t \quad [18]$$

$$-4 \frac{di_1}{dt} + 4 \frac{di_2}{dt} + \frac{1}{8} \int_0^t i_2 dt' + 5i_2 = -10 \quad [19]$$

We may now construct the two equations that describe the exact dual of our circuit. We wish these to be nodal equations, and thus begin by replacing the mesh currents  $i_1$  and  $i_2$  in Eqs. [18] and [19] by the two nodal voltages  $v_1$  and  $v_2$ , respectively. We obtain

$$3v_1 + 4 \frac{dv_1}{dt} - 4 \frac{dv_2}{dt} = 2 \cos 6t \quad [20]$$

$$-4 \frac{dv_1}{dt} + 4 \frac{dv_2}{dt} + \frac{1}{8} \int_0^t v_2 dt' + 5v_2 = -10 \quad [21]$$

and we now seek the circuit represented by these two nodal equations.

Let us first draw a line to represent the reference node, and then we may establish two nodes at which the positive references for  $v_1$  and  $v_2$  are located. Equation [20] indicates that a current source of  $2 \cos 6t$  A is connected between node 1 and the reference node, oriented to provide a current entering node 1. This equation also shows that a 3 S conductance appears between node 1 and the reference node. Turning to Eq. [21], we first consider the nonmutual terms, i.e., those terms which do not appear in Eq. [20], and they instruct us to connect an 8 H inductor and a 5 S conductance (in parallel) between node 2 and the reference. The two similar terms in Eqs. [20] and [21] represent a 4 F capacitor present mutually at nodes 1 and 2; the circuit is completed by connecting this capacitor between the two nodes. The constant term on the right side of Eq. [21] is the value of the inductor current at  $t = 0$ ; in other words,  $i_L(0) = 10$  A. The dual circuit is shown in Fig. 7.31; since the two sets of equations are numerically identical, the circuits are exact duals.

Dual circuits may be obtained more readily than by this method, for the equations need not be written. In order to construct the dual of a given circuit, we think of the circuit in terms of its mesh equations. With each mesh we must associate a nonreference node, and, in addition, we must supply the reference node. On a diagram of the given circuit we therefore place a

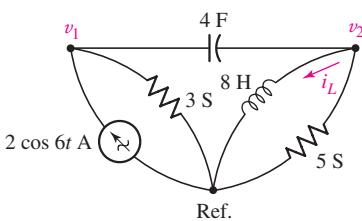


FIGURE 7.31 The exact dual of the circuit of Fig. 7.30.

node in the center of each mesh and supply the reference node as a line near the diagram or a loop enclosing the diagram. Each element that appears jointly in two meshes is a mutual element and gives rise to identical terms, except for sign, in the two corresponding mesh equations. It must be replaced by an element that supplies the dual term in the two corresponding nodal equations. This dual element must therefore be connected directly between the two nonreference nodes that are within the meshes in which the given mutual element appears.

The nature of the dual element itself is easily determined; the mathematical form of the equations will be the same only if inductance is replaced by capacitance, capacitance by inductance, conductance by resistance, and resistance by conductance. Thus, the 4 H inductor which is common to meshes 1 and 2 in the circuit of Fig. 7.30 appears as a 4 F capacitor connected directly between nodes 1 and 2 in the dual circuit.

Elements that appear only in one mesh must have duals that appear between the corresponding node and the reference node. Referring again to Fig. 7.30, the voltage source  $2 \cos 6t$  V appears only in mesh 1; its *dual* is a current source  $2 \cos 6t$  A, which is connected only to node 1 and the reference node. Since the voltage source is clockwise-sensed, the current source must be into-the-nonreference-node-sensed. Finally, provision must be made for the dual of the initial voltage present across the 8 F capacitor in the given circuit. The equations have shown us that the dual of this initial voltage across the capacitor is an initial current through the inductor in the dual circuit; the numerical values are the same, and the correct sign of the initial current may be determined most readily by considering both the initial voltage in the given circuit and the initial current in the dual circuit as sources. Thus, if  $v_C$  in the given circuit is treated as a source, it would appear as  $-v_C$  on the right side of the mesh equation; in the dual circuit, treating the current  $i_L$  as a source would yield a term  $-i_L$  on the right side of the nodal equation. Since each has the same sign when treated as a source, then, if  $v_C(0) = 10$  V,  $i_L(0)$  must be 10 A.

The circuit of Fig. 7.30 is repeated in Fig. 7.32, and its exact dual is constructed on the circuit diagram itself by merely drawing the dual of each given element between the two nodes that are inside the two meshes that are common to the given element. A reference node that surrounds the given circuit may be helpful. After the dual circuit is redrawn in more standard form, it appears as shown in Fig. 7.31.

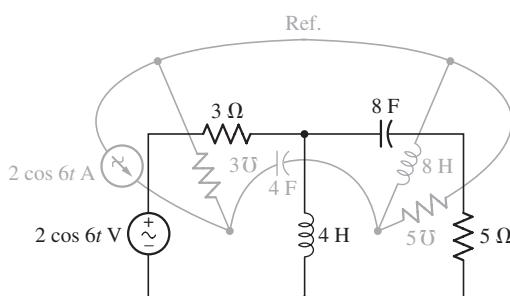


FIGURE 7.32 The dual of the circuit of Fig. 7.30 is constructed directly from the circuit diagram.

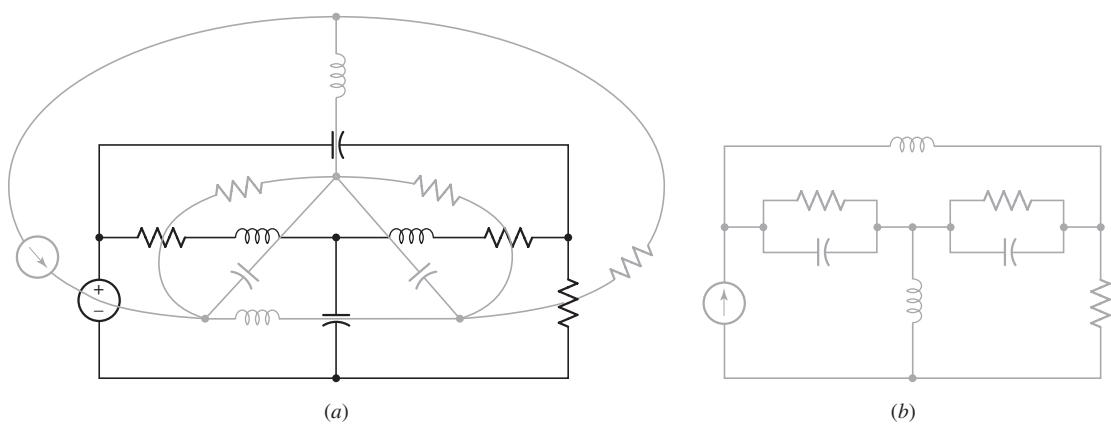


FIGURE 7.33 (a) The dual (in gray) of a given circuit (in black) is constructed on the given circuit. (b) The dual circuit is drawn in more conventional form for comparison to the original.

An additional example of the construction of a dual circuit is shown in Fig. 7.33a and b. Since no particular element values are specified, these two circuits are duals, but not necessarily exact duals. The original circuit may be recovered from the dual by placing a node in the center of each of the five meshes of Fig. 7.33b and proceeding as before.

The concept of duality may also be carried over into the language by which we describe circuit analysis or operation. For example, if we are given a voltage source in series with a capacitor, we might wish to make the important statement, “*The voltage source causes a current to flow through the capacitor.*” The dual statement is, “*The current source causes a voltage to exist across the inductor.*” The dual of a less carefully worded statement, such as “*The current goes round and round the series circuit,*” may require a little inventiveness.<sup>4</sup>

Practice in using dual language can be obtained by reading Thévenin’s theorem in this sense; Norton’s theorem should result.

We have spoken of dual elements, dual language, and dual circuits. What about a dual network? Consider a resistor  $R$  and an inductor  $L$  in series. The dual of this two-terminal network exists and is most readily obtained by connecting some ideal source to the given network. The dual circuit is then obtained as the dual source in parallel with a conductance  $G$  with the same magnitude as  $R$ , and a capacitance  $C$  having the same magnitude as  $L$ . We consider the dual network as the two-terminal network that is connected to the dual source; it is thus a pair of terminals between which  $G$  and  $C$  are connected in parallel.

Before leaving the definition of duality, we should point out that duality is defined on the basis of mesh and nodal equations. Since nonplanar circuits cannot be described by a system of mesh equations, a circuit that cannot be drawn in planar form does not possess a dual.

We shall use duality principally to reduce the work that we must do to analyze the simple standard circuits. After we have analyzed the series  $RL$  circuit, the parallel  $RC$  circuit requires less attention, not because it is less important, but because the analysis of the dual network is already known. Since the analysis of some complicated circuit is not apt to be well known, duality will usually not provide us with any quick solution.

(4) Someone suggested, “The voltage is across all over the parallel circuit.”

**PRACTICE**

- 7.11 Write the single nodal equation for the circuit of Fig. 7.34a, and show, by direct substitution, that  $v = -80e^{-10^6t}$  mV is a solution. Knowing this, find (a)  $v_1$ ; (b)  $v_2$ ; and (c)  $i$  for the circuit of Fig. 7.34b.

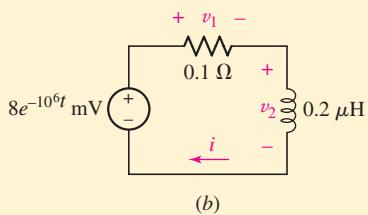
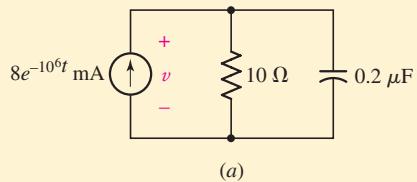


FIGURE 7.34

Ans:  $-8e^{-10^6t}$  mV;  $16e^{-10^6t}$  mV;  $-80e^{-10^6t}$  mA.

## 7.7 MODELING CAPACITORS AND INDUCTORS WITH PSPICE

When using PSpice to analyze circuits containing inductors and capacitors, it is frequently necessary to be able to specify the initial condition of each element [i.e.,  $v_C(0)$  and  $i_L(0)$ ]. This is achieved by double-clicking on the element symbol, resulting in the dialog box shown in Fig. 7.35a. At the far right (not shown), we find the value of the capacitance, which defaults to 1 nF. We can also specify the initial condition (IC), set to 2 V in Fig. 7.35a. Clicking on the right mouse button and selecting **Display** results in the dialog box shown in Fig. 7.35b, which allows the initial condition to be displayed on the schematic. The procedure for setting the initial condition of an inductor is essentially the same. We should also note that when a capacitor is first placed in the schematic, it appears horizontally; the positive reference terminal for the initial voltage is the *left* terminal.

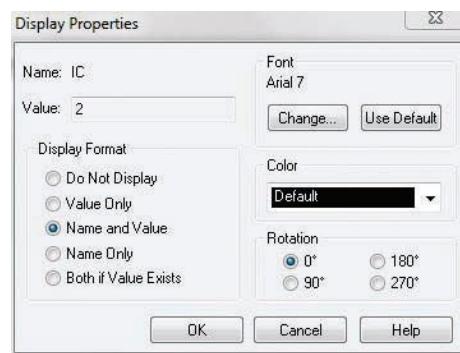
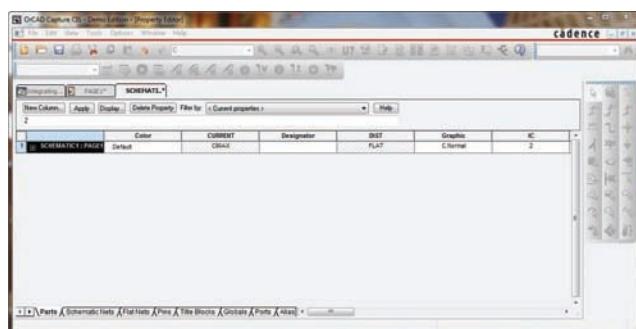


FIGURE 7.35 (a) Capacitor property editor window. (b) Display Properties dialog box, obtained by right-clicking in the IC box.

## EXAMPLE 7.11

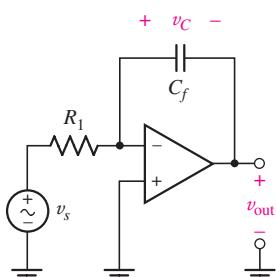


FIGURE 7.36 An integrating op amp circuit.

Simulate the output voltage waveform of the circuit in Fig. 7.36 if  $v_s = 1.5 \sin 100t$  V,  $R_1 = 10 \text{ k}\Omega$ ,  $C_f = 4.7 \mu\text{F}$ , and  $v_C(0) = 2$  V.

We begin by drawing the circuit schematic, making sure to set the initial voltage across the capacitor (Fig. 7.37). Note that we had to convert the frequency from  $100 \text{ rad/s}$  to  $100/2\pi = 15.92 \text{ Hz}$ .

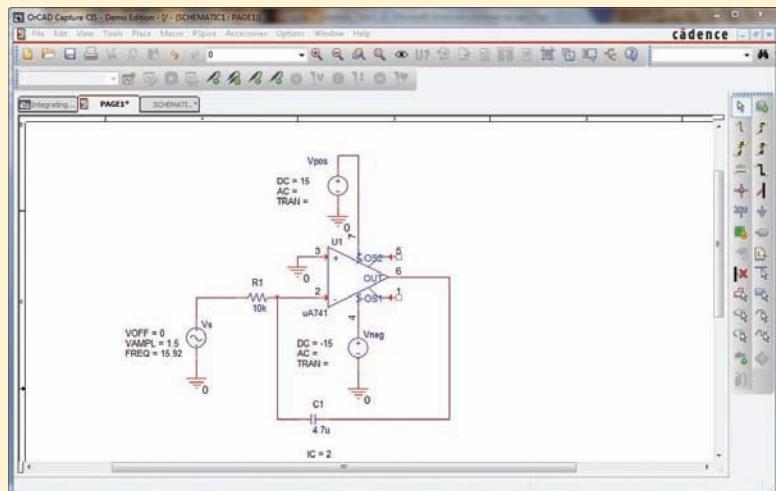
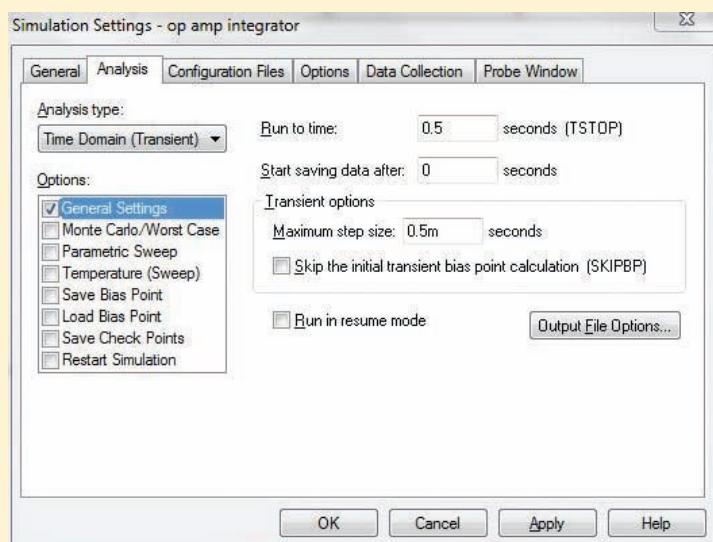


FIGURE 7.37 The schematic representation of the circuit shown in Fig. 7.36, with the initial capacitor voltage set to 2 V.

In order to obtain time-varying voltages and currents, we need to perform what is referred to as a *transient analysis*. Under the **PSpice** menu, we create a **New Simulation Profile** named **op amp integrator**, which leads to the dialog box recreated in Fig. 7.38. **Run to time** represents the

FIGURE 7.38 Dialog box for setting up a transient analysis. We choose a final time of 0.5 s to obtain several periods of the output waveform ( $1/15.92 \approx 0.06$  s).

time at which the simulation is terminated; PSpice will select its own discrete times at which to calculate the various voltages and currents. Occasionally we obtain an error message stating that the transient solution could not converge, or the output waveform does not appear as smooth as we would like. In such situations, it is useful to set a value for **Maximum step size**, which has been set to 0.5 ms in this example.

From our earlier analysis and Eq. [17], we expect the output to be proportional to the negative integral of the input waveform, i.e.,  $v_{out} = 0.319 \cos 100t - 2.319$  V, as shown in Fig. 7.39. The initial condition of 2 V across the capacitor has combined with a constant term from the integration to result in a nonzero average value for the output, unlike the input which has an average value of zero.

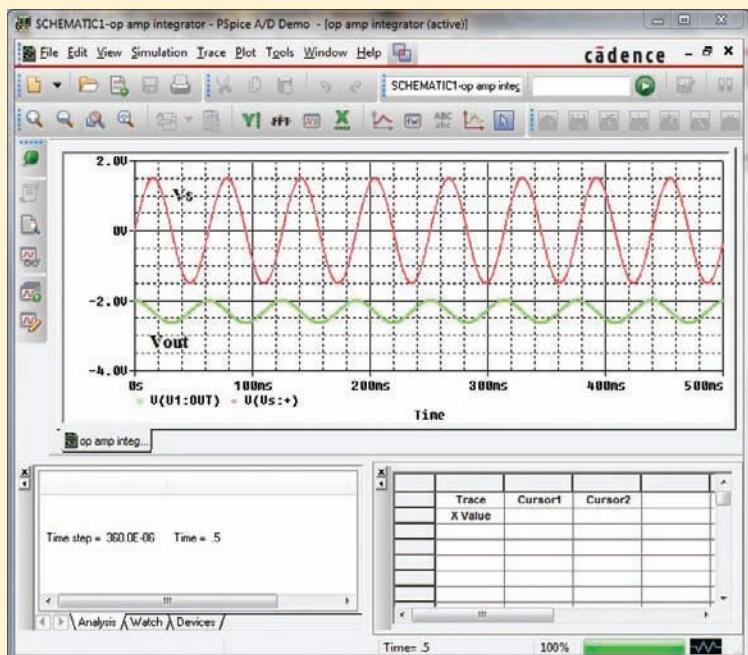


FIGURE 7.39 Probe output for the simulated integrator circuit along with the input waveform for comparison.

## SUMMARY AND REVIEW

A large number of practical circuits can be effectively modeled using only resistors and voltage/current sources. However, most interesting everyday occurrences somehow involve something changing with time, and in such cases intrinsic capacitances and/or inductances can become important. We employ such energy storage elements consciously as well, for example, in the design of frequency-selective filters, capacitor banks, and electric vehicle motors. An *ideal* capacitor is modeled as having infinite shunt resistance, and a current which depends on the time rate of change of the terminal voltage. Capacitance is measured in units of farads (F). Conversely, an *ideal* inductor is modeled as having zero series resistance, and a terminal voltage which depends on the time rate of change of the current. Inductance is measured in

units of *henrys* (H). Either element can *store energy*; the amount of energy present in a capacitor (stored in its electric field) is proportional to the *square* of the terminal voltage, and the amount of energy present in an inductor (stored in its magnetic field) is proportional to the *square* of its current.

As we found for resistors, we can simplify some connections of capacitors (or inductors) using series/parallel combinations. The validity of such equivalents arises from KCL and KVL. Once we have simplified a circuit as much as possible (taking care not to “combine away” a component which is used to define a current or voltage of interest to us), nodal and mesh analysis can be applied to circuits with capacitors and inductors. However, the resulting integrodifferential equations are often nontrivial to solve, and so we will consider some practical approaches in the next two chapters. Simple circuits, however, such as those which involve a single operational amplifier, can be analyzed easily. We found (to our surprise) that such circuits can be used as signal *integrators* or *differentiators*. Consequently, they provide an output signal that tells us how some input quantity (accumulating charge during ion implantation into a silicon wafer, for example) varies with time.

As a final note, capacitors and inductors provide a particularly strong example of the concept known as *duality*. KCL and KVL, mesh and nodal analysis are other examples. Circuits are rarely analyzed using this idea, but it is nevertheless important, since the implication is that we only need to learn roughly “half” of the complete set of concepts, and then determine how to translate the remainder. Some people find this helpful; others don’t. Regardless, capacitors and inductors are straightforward to model in PSpice and other circuit simulation tools, allowing us to check our answers. The difference between those elements and resistors in such software packages is that we must take care to *set the initial condition properly*.

As an additional review aid, here we list some key points from the chapter, and identify relevant example(s).

- The current through a capacitor is given by  $i = C dv/dt$ . (Example 7.1)
- The voltage across a capacitor is related to its current by

$$v(t) = \frac{1}{C} \int_{t_0}^t i(t') dt' + v(t_0)$$

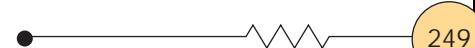
(Example 7.2)

- A capacitor is an *open circuit* to dc voltages. (Example 7.1)
- The voltage across an inductor is given by  $v = L di/dt$ . (Examples 7.4, 7.5)
- The current through an inductor is related to its voltage by

$$i(t) = \frac{1}{L} \int_{t_0}^t v dt' + i(t_0)$$

(Example 7.6)

- An inductor is a *short circuit* to dc currents. (Examples 7.4, 7.5)
- The energy presently stored in a capacitor is given by  $\frac{1}{2} Cv^2$ , whereas the energy presently stored in an inductor is given by  $\frac{1}{2} Li^2$ ; both are referenced to a time at which no energy was stored. (Examples 7.3, 7.7)
- Series and parallel combinations of inductors can be combined using the same equations as for resistors. (Example 7.8)



- Series and parallel combinations of capacitors work the *opposite* way as they do for resistors. (Example 7.8)
- Since capacitors and inductors are linear elements, KVL, KCL, superposition, Thévenin's and Norton's theorems, and nodal and mesh analysis apply to their circuits as well. (Example 7.9)
- A capacitor as the feedback element in an inverting op amp leads to an output voltage proportional to the *integral* of the input voltage. Swapping the input resistor and the feedback capacitor leads to an output voltage proportional to the *derivative* of the input voltage. (Example 7.10)
- PSpice allows us to set the initial voltage across a capacitor, and the initial current through an inductor. A transient analysis provides details of the time-dependent response of circuits containing these types of elements. (Example 7.11)

## READING FURTHER

A detailed guide to characteristics and selection of various capacitor and inductor types can be found in:

H. B. Drexler, *Passive Electronic Component Handbook*, 2nd ed., C. A. Harper, ed. New York: McGraw-Hill, 2003, pp. 69–203.

C. J. Kaiser, *The Inductor Handbook*, 2nd ed. Olathe, Kans.: C.J. Publishing, 1996.

Two books that describe capacitor-based op amp circuits are:

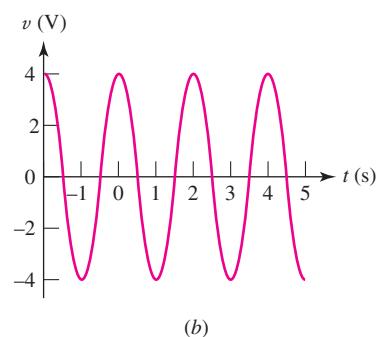
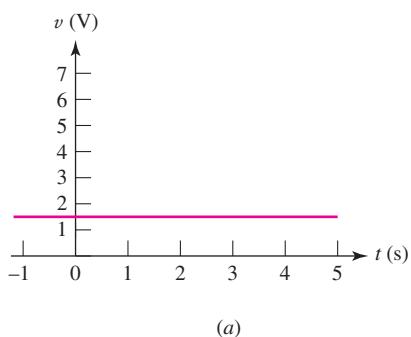
R. Mancini (ed.), *Op Amps Are For Everyone*, 2nd ed. Amsterdam: Newnes, 2003.

W. G. Jung, *Op Amp Cookbook*, 3rd ed. Upper Saddle River, N.J.: Prentice-Hall, 1997.

## EXERCISES

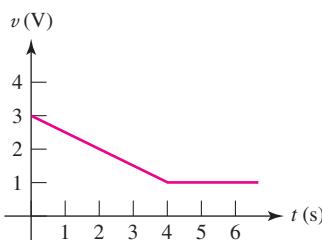
### 7.1 The Capacitor

1. Making use of the passive sign convention, determine the current flowing through a 220 nF capacitor for  $t \geq 0$  if its voltage  $v_C(t)$  is given by (a)  $-3.35 \text{ V}$ ; (b)  $16.2e^{-9t} \text{ V}$ ; (c)  $8 \cos 0.01t \text{ mV}$ ; (d)  $5 + 9 \sin 0.08t \text{ V}$ .
2. Sketch the current flowing through a 13 pF capacitor for  $t \geq 0$  as a result of the waveforms shown in Fig. 7.40. Assume the passive sign convention.



■ FIGURE 7.40

3. (a) If the voltage waveform depicted in Fig. 7.41 is applied across the terminals of a  $1 \mu\text{F}$  electrolytic capacitor, graph the resulting current, assuming the passive sign convention. (b) Repeat part (a) if the capacitor is replaced with a  $17.5 \text{ pF}$  capacitor.



■ FIGURE 7.41

4. A capacitor is constructed from two copper plates, each measuring  $1 \text{ mm} \times 2.5 \text{ mm}$  and  $155 \mu\text{m}$  thick. The two plates are placed such that they face each other and are separated by a  $1 \mu\text{m}$  gap. Calculate the resulting capacitance if (a) the intervening dielectric has a permittivity of  $1.35\epsilon_0$ ; (b) the intervening dielectric has a permittivity of  $3.5\epsilon_0$ ; (c) the plate separation is increased by  $3.5 \mu\text{m}$  and the gap is filled with air; (d) the plate area is doubled and the  $1 \mu\text{m}$  gap is filled with air.
5. Two pieces of gadolinium, each measuring  $100 \mu\text{m} \times 750 \mu\text{m}$  and  $604 \text{ nm}$  thick, are used to construct a capacitor. The two plates are arranged such that they face each other and are separated by a  $100 \text{ nm}$  gap. Calculate the resulting capacitance if (a) the intervening dielectric has a permittivity of  $13.8\epsilon_0$ ; (b) the intervening dielectric has a permittivity of  $500\epsilon_0$ ; (c) the plate separation is increased by  $100 \text{ nm}$  and the gap is filled with air; (d) the plate area is quadrupled and the  $100 \text{ nm}$  gap is filled with air.
6. Design a  $100 \text{ nF}$  capacitor constructed from  $1 \mu\text{m}$  thick gold foil, and which fits entirely within a volume equal to that of a standard AAA battery, if the only dielectric available has a permittivity of  $3.1\epsilon_0$ .
7. Design a capacitor whose capacitance can be varied mechanically with a simple vertical motion, between the values of  $100 \text{ nF}$  and  $300 \text{ nF}$ .
8. Design a capacitor whose capacitance can be varied mechanically over the range of  $50 \text{ nF}$  and  $100 \text{ nF}$  by rotating a knob  $90^\circ$ .
9. A silicon  $pn$  junction diode is characterized by a junction capacitance defined as

$$C_j = \frac{K_s \epsilon_0 A}{W}$$

where  $K_s = 11.8$  for silicon,  $\epsilon_0$  is the vacuum permittivity,  $A$  = the cross-sectional area of the junction, and  $W$  is known as the depletion width of the junction. Width  $W$  depends not only on how the diode is fabricated, but also on the voltage applied to its two terminals. It can be computed using

$$W = \sqrt{\frac{2K_s \epsilon_0}{qN} (V_{bi} - V_A)}$$

Thus, diodes are frequently used in electronic circuits, since they can be thought of as voltage-controlled capacitors. Assuming parameter values of  $N = 10^{18} \text{ cm}^{-3}$ ,  $V_{bi} = 0.57 \text{ V}$ , and using  $q = 1.6 \times 10^{-19} \text{ C}$ , calculate the capacitance of a diode with cross-sectional area  $A = 1 \mu\text{m} \times 1 \mu\text{m}$  at applied voltages of  $V_A = -1, -5$ , and  $-10$  volts.

10. Assuming the passive sign convention, sketch the voltage which develops across the terminals of a  $2.5 \text{ F}$  capacitor in response to the current waveforms shown in Fig. 7.42.

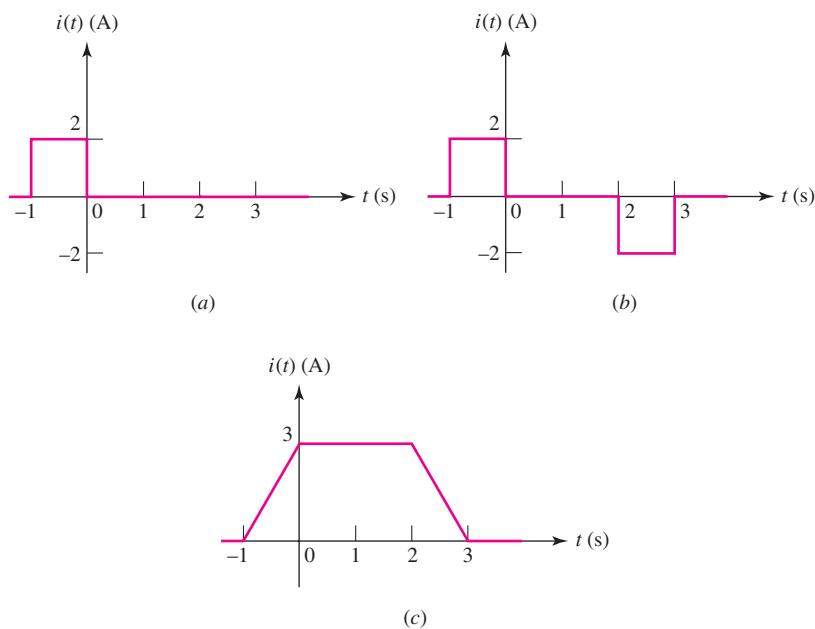


FIGURE 7.42

11. The current flowing through a 33 mF capacitor is shown graphically in Fig. 7.43.  
 (a) Assuming the passive sign convention, sketch the resulting voltage waveform across the device.  
 (b) Compute the voltage at 300 ms, 600 ms, and 1.1 s.

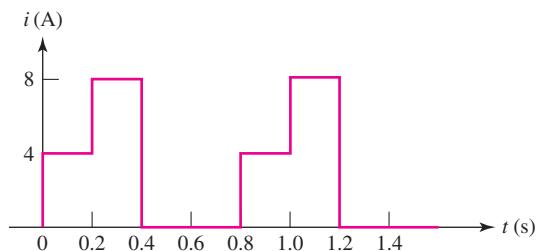


FIGURE 7.43

12. Calculate the energy stored in a capacitor at time  $t = 1$  s if (a)  $C = 1.4$  F and  $v_C = 8$  V,  $t > 0$ ; (b)  $C = 23.5$  pF and  $v_C = 0.8$  V,  $t > 0$ ; (c)  $C = 17$  nF,  $v_C(1) = 12$  V,  $v_C(0) = 2$  V, and  $w_C(0) = 295$  nJ.  
 13. A 137 pF capacitor is connected to a voltage source such that  $v_C(t) = 12e^{-2t}$  V,  $t \geq 0$  and  $v_C(t) = 12$  V,  $t < 0$ . Calculate the energy stored in the capacitor at  $t$  equal to (a) 0; (b) 200 ms; (c) 500 ms; (d) 1 s.  
 14. Calculate the power dissipated in the  $40\ \Omega$  resistor and the voltage labeled  $v_C$  in each of the circuits depicted in Fig. 7.44.

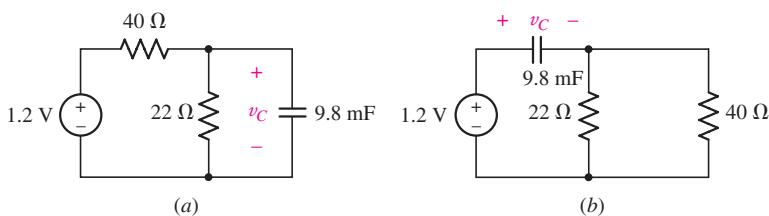
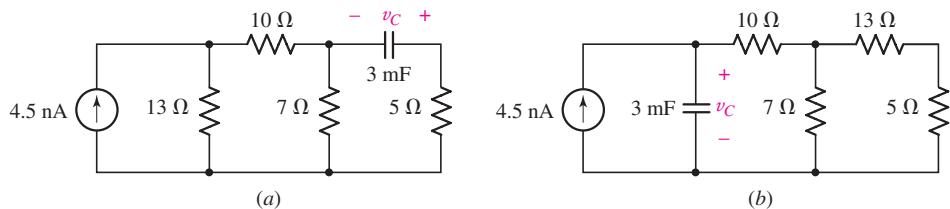


FIGURE 7.44

15. For each circuit shown in Fig. 7.45, calculate the voltage labeled  $v_C$ .

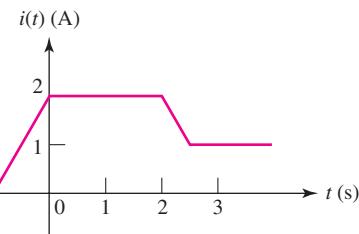


■ FIGURE 7.45

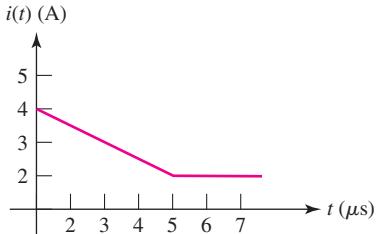
## 7.2 The Inductor



16. Design a 30 nH inductor using 29 AWG solid soft copper wire. Include a sketch of your design and label geometrical parameters as necessary for clarity. Assume the coil is filled with air only.
17. If the current flowing through a 75 mH inductor has the waveform shown in Fig. 7.46, (a) sketch the voltage which develops across the inductor terminals for  $t \geq 0$ , assuming the passive sign convention; and (b) calculate the voltage at  $t = 1$  s, 2.9 s, and 3.1 s.

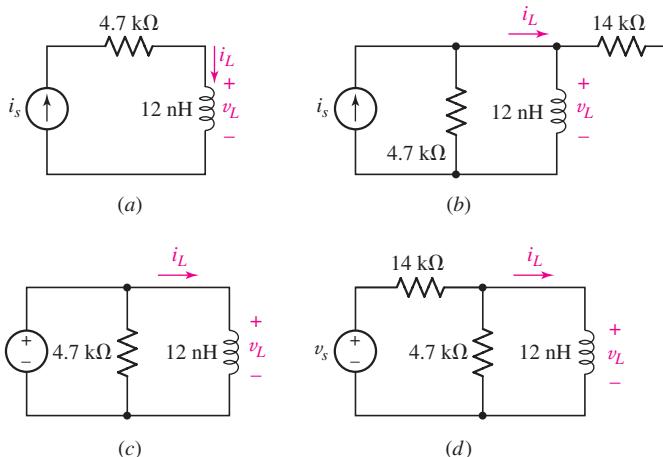


■ FIGURE 7.46



■ FIGURE 7.47

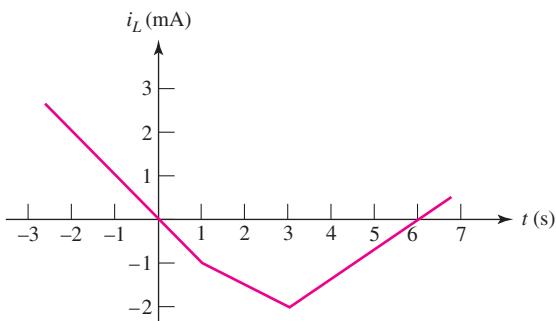
18. The current through a 17 nH aluminum inductor is shown in Fig. 7.47. Sketch the resulting voltage waveform for  $t \geq 0$ , assuming the passive sign convention.
19. Determine the voltage for  $t \geq 0$  which develops across the terminals of a 4.2 mH inductor, if the current (defined consistent with the passive sign convention) is (a)  $-10$  mA; (b)  $3 \sin 6t$  A; (c)  $11 + 115\sqrt{2} \cos(100\pi t - 9^\circ)$  A; (d)  $13e^{-t}$  nA; (e)  $3 + te^{-14t}$  A.
20. Determine the voltage for  $t \geq 0$  which develops across the terminals of an 8 pH inductor, if the current (defined consistent with the passive sign convention) is (a) 8 mA; (b) 800 mA; (c) 8 A; (d)  $4e^{-t}$  A; (e)  $-3 + te^{-t}$  A.
21. Calculate  $v_L$  and  $i_L$  for each of the circuits depicted in Fig. 7.48, if  $i_s = 1$  mA and  $v_s = 2.1$  V.



■ FIGURE 7.48

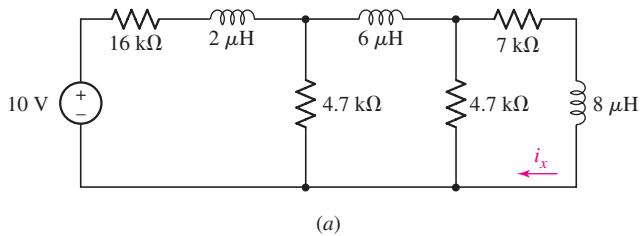
22. The current waveform shown in Fig. 7.14 has a rise time of 0.1 (100 ms) and a fall time of the same duration. If the current is applied to the “+” voltage reference terminal of a 200 nH inductor, sketch the expected voltage waveform if the rise and fall times are changed, respectively, to (a) 200 ms, 200 ms; (b) 10 ms, 50 ms; (c) 10 ns, 20 ns.

23. Determine the inductor voltage which results from the current waveform shown in Fig. 7.49 (assuming the passive sign convention) at  $t$  equal to (a) -1 s; (b) 0 s; (c) 1.5 s; (d) 2.5 s; (e) 4 s; (f) 5 s.

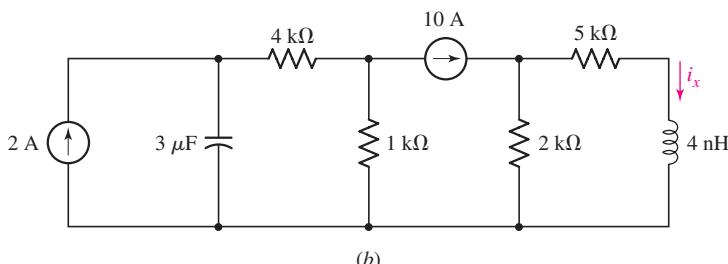


■ FIGURE 7.49

24. Determine the current flowing through a 6 mH inductor if the voltage (defined such that it is consistent with the passive sign convention) is given by (a) 5 V; (b)  $100 \sin 120\pi t$ ,  $t \geq 0$  and 0,  $t < 0$ .
25. The voltage across a 2 H inductor is given by  $v_L = 4.3t$ ,  $0 \leq t \leq 50$  ms. With the knowledge that  $i_L(-0.1) = 100 \mu\text{A}$ , calculate the current (assuming it is defined consistent with the passive sign convention) at  $t$  equal to (a) 0; (b) 1.5 ms; (c) 45 ms.
26. Calculate the energy stored in a 1 nH inductor if the current flowing through it is (a) 0 mA; (b) 1 mA; (c) 20 A; (d)  $5 \sin 6t$  mA,  $t > 0$ .
27. Determine the amount of energy stored in a 33 mH inductor at  $t = 1$  ms as a result of a current  $i_L$  given by (a) 7 A; (b)  $3 - 9e^{-10^3 t}$  mA.
28. Making the assumption that the circuits in Fig. 7.50 have been connected for a very long time, determine the value for each current labeled  $i_x$ .



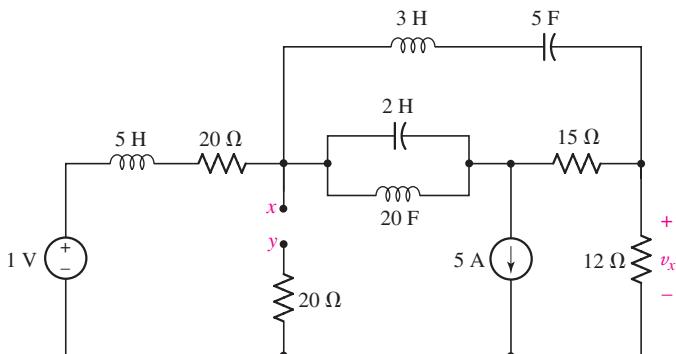
(a)



(b)

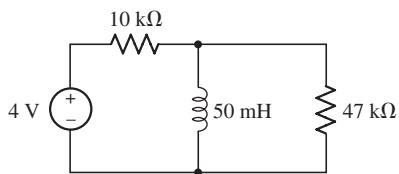
■ FIGURE 7.50

29. Calculate the voltage labeled  $v_x$  in Fig. 7.51, assuming the circuit has been running a very long time, if (a) a  $10\ \Omega$  resistor is connected between terminals  $x$  and  $y$ ; (b) a  $1\ H$  inductor is connected between terminals  $x$  and  $y$ ; (c) a  $1\ F$  capacitor is connected between terminals  $x$  and  $y$ ; (d) a  $4\ H$  inductor in parallel with a  $1\ \Omega$  resistor is connected between terminals  $x$  and  $y$ .



■ FIGURE 7.51

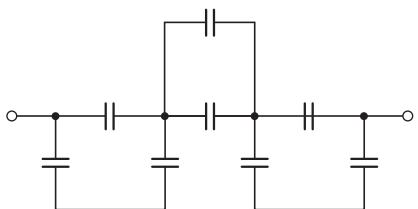
30. For the circuit shown in Fig. 7.52, (a) compute the Thévenin equivalent seen by the inductor; (b) determine the power being dissipated by both resistors; (c) calculate the energy stored in the inductor.



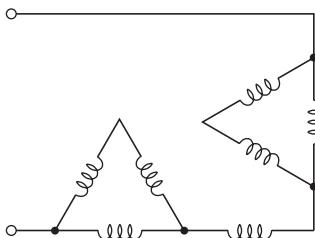
■ FIGURE 7.52

### 7.3 Inductance and Capacitance Combinations

31. If each capacitor has a value of  $1\ F$ , determine the equivalent capacitance of the network shown in Fig. 7.53.
32. Determine an equivalent inductance for the network shown in Fig. 7.54 if each inductor has value  $L$ .

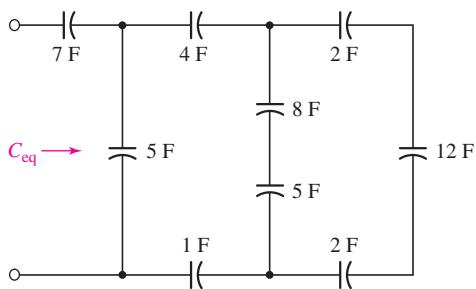


■ FIGURE 7.53



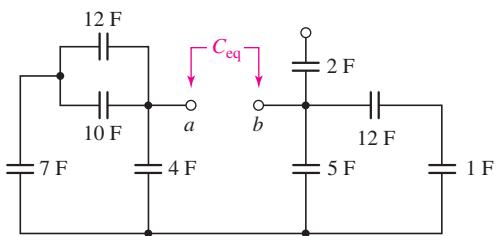
■ FIGURE 7.54

- DP** 33. Using as many  $1\ nH$  inductors as you like, design two networks, each of which has an equivalent inductance of  $1.25\ nH$ .
34. Compute the equivalent capacitance  $C_{eq}$  as labeled in Fig. 7.55.



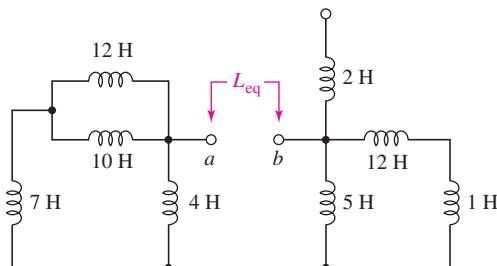
■ FIGURE 7.55

35. Determine the equivalent capacitance  $C_{eq}$  of the network shown in Fig. 7.56.



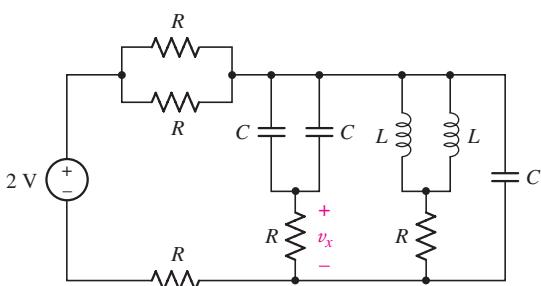
■ FIGURE 7.56

36. Apply combinatorial techniques as appropriate to obtain a value for the equivalent inductance  $L_{eq}$  as labeled on the network of Fig. 7.57.



■ FIGURE 7.57

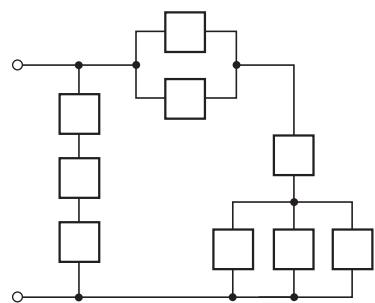
37. Reduce the circuit depicted in Fig. 7.58 to as few components as possible.



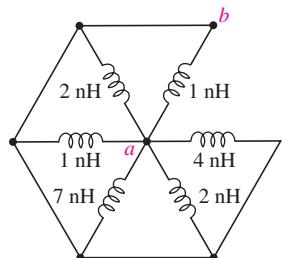
■ FIGURE 7.58

38. Refer to the network shown in Fig. 7.59 and find (a)  $R_{eq}$  if each element is a  $10 \Omega$  resistor; (b)  $L_{eq}$  if each element is a  $10 \text{ H}$  inductor; and (c)  $C_{eq}$  if each element is a  $10 \text{ F}$  capacitor.

39. Determine the equivalent inductance seen looking into the terminals marked *a* and *b* of the network represented in Fig. 7.60.

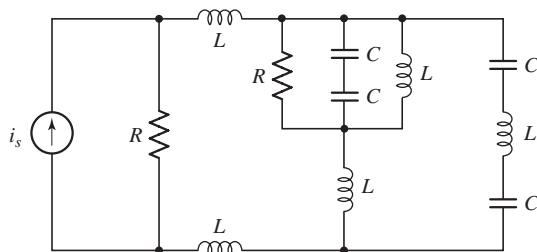


■ FIGURE 7.59



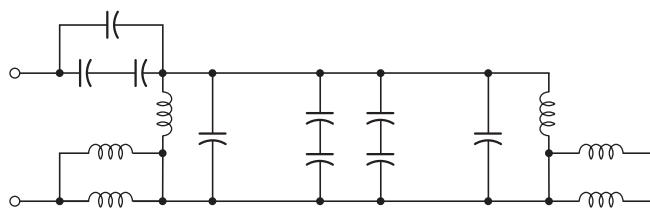
■ FIGURE 7.60

40. Reduce the circuit represented in Fig. 7.61 to the smallest possible number of components.



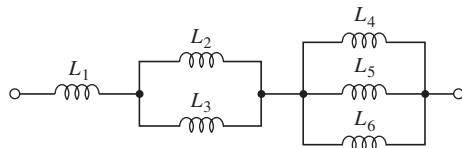
■ FIGURE 7.61

41. Reduce the network of Fig. 7.62 to the smallest possible number of components if each inductor is 1 nH and each capacitor is 1 mF.

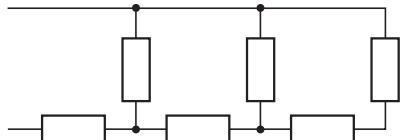


■ FIGURE 7.62

42. For the network of Fig. 7.63,  $L_1 = 1\text{ H}$ ,  $L_2 = L_3 = 2\text{ H}$ ,  $L_4 = L_5 = L_6 = 3\text{ H}$ .  
 (a) Find the equivalent inductance. (b) Derive an expression for a general network of this type having  $N$  stages, assuming stage  $N$  is composed of  $N$  inductors, each having inductance  $N$  henrys.



■ FIGURE 7.63

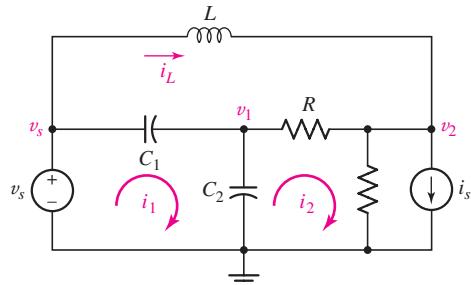


■ FIGURE 7.64

43. Extend the concept of  $\Delta$ -Y transformations to simplify the network of Fig. 7.64 if each element is a 2 pF capacitor.  
 44. Extend the concept of  $\Delta$ -Y transformations to simplify the network of Fig. 7.64 if each element is a 1 nH inductor.

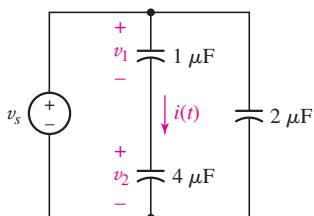
#### 7.4 Consequences of Linearity

45. With regard to the circuit represented in Fig. 7.65, (a) write a complete set of nodal equations and (b) write a complete set of mesh equations.



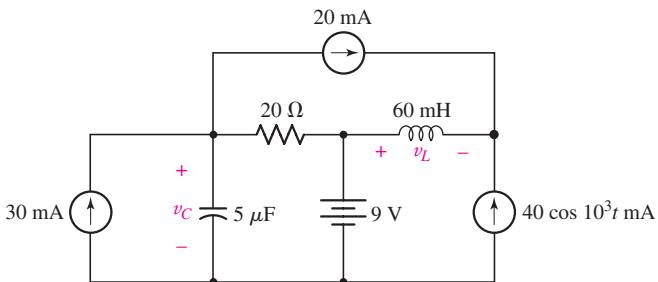
■ FIGURE 7.65

46. (a) Write nodal equations for the circuit of Fig. 7.66. (b) Write mesh equations for the same circuit.
47. In the circuit shown in Fig. 7.67, let  $i_s = 60e^{-200t}$  mA with  $i_1(0) = 20$  mA. (a) Find  $v(t)$  for all  $t$ . (b) Find  $i_1(t)$  for  $t \geq 0$ . (c) Find  $i_2(t)$  for  $t \geq 0$ .
48. Let  $v_s = 100e^{-80t}$  V and  $v_1(0) = 20$  V in the circuit of Fig. 7.68. (a) Find  $i(t)$  for all  $t$ . (b) Find  $v_1(t)$  for  $t \geq 0$ . (c) Find  $v_2(t)$  for  $t \geq 0$ .



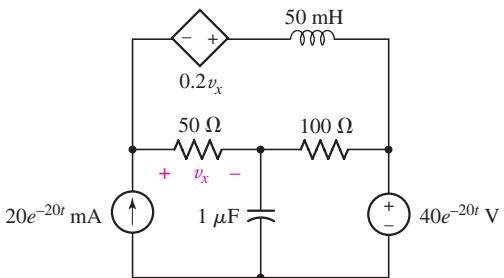
■ FIGURE 7.68

49. If it is assumed that all the sources in the circuit of Fig. 7.69 have been connected and operating for a very long time, use the superposition principle to find  $v_C(t)$  and  $v_L(t)$ .



■ FIGURE 7.69

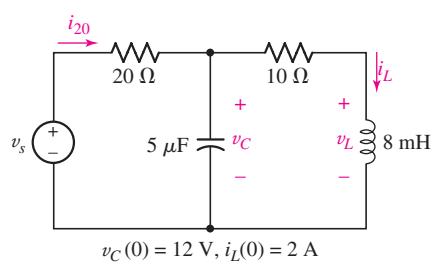
50. For the circuit of Fig. 7.70, assume no energy is stored at  $t = 0$ , and write a complete set of nodal equations.



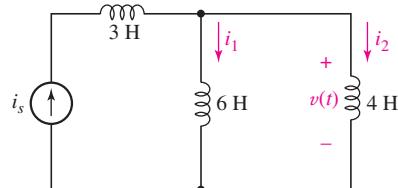
■ FIGURE 7.70

## 7.5 Simple Op Amp Circuits with Capacitors

51. Interchange the location of  $R_1$  and  $C_f$  in the circuit of Fig. 7.27, and assume that  $R_i = \infty$ ,  $R_o = 0$ , and  $A = \infty$  for the op amp. (a) Find  $v_{out}(t)$  as a function of  $v_s(t)$ . (b) Obtain an equation relating  $v_o(t)$  and  $v_s(t)$  if  $A$  is not assumed to be infinite.
52. For the integrating amplifier circuit of Fig. 7.27,  $R_1 = 100\text{ k}\Omega$ ,  $C_f = 500\ \mu F$ , and  $v_s = 20 \sin 540t$  mV. Calculate  $v_{out}$  if (a)  $A = \infty$ ,  $R_i = \infty$ , and  $R_o = 0$ ; (b)  $A = 5000$ ,  $R_i = 1\text{ M}\Omega$ , and  $R_o = 3\ \Omega$ .



■ FIGURE 7.66



■ FIGURE 7.67

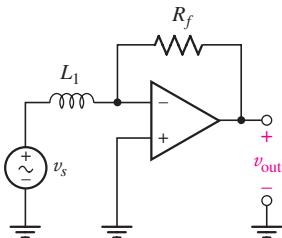


FIGURE 7.71

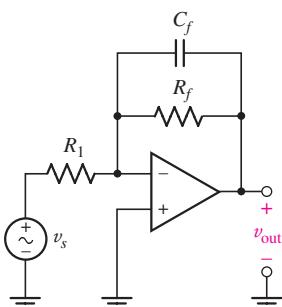


FIGURE 7.72

53. Derive an expression for  $v_{\text{out}}$  in terms of  $v_s$  for the amplifier circuit shown in Fig. 7.71.

54. In practice, circuits such as those depicted in Fig. 7.27 may not function correctly unless there is a conducting pathway between the output and input terminals of the op amp. (a) Analyze the modified integrating amplifier circuit shown in Fig. 7.72 to obtain an expression for  $v_{\text{out}}$  in terms of  $v_s$ , and (b) compare this expression to Eq. [17].

DP

55. A new piece of equipment designed to make crystals from molten constituents is experiencing too many failures (cracked products). The production manager wants to monitor the cooling rate to see if this is related to the problem. The system has two output terminals available, where the voltage across them is linearly proportional to the crucible temperature such that 30 mV corresponds to 30°C and 1 V corresponds to 1000°C. Design a circuit whose voltage output represents the cooling rate, calibrated such that 1 V = 1°C/s.

DP

56. A confectionary company has decided to increase the production rate of its milk chocolate bars to compensate for a recent increase in the cost of raw materials. However, the wrapping unit cannot accept more than 1 bar per second, or it drops bars. A 200 mV peak-to-peak sinusoidal voltage signal is available from the bar-making system which feeds into the wrapping unit, such that its frequency matches the bar production frequency (i.e., 1 Hz = 1 bar/s). Design a circuit that provides a voltage output sufficient to power a 12 V audible alarm when the production rate exceeds the capacity of the wrapping unit.

DP

57. One problem satellites face is exposure to high-energy particles, which can cause damage to sensitive electronics as well as solar arrays used to provide power. A new communications satellite is equipped with a high-energy proton detector measuring 1 cm × 1 cm. It provides a current directly equal to the number of protons impinging the surface per second. Design a circuit whose output voltage provides a running total of the number of proton hits, calibrated such that 1 V = 1 million hits.

DP

58. The output of a velocity sensor attached to a sensitive piece of mobile equipment is calibrated to provide a signal such that 10 mV corresponds to linear motion at 1 m/s. If the equipment is subjected to sudden shock, it can be damaged. Since force = mass × acceleration, monitoring of the rate of change of velocity can be used to determine if the equipment is transported improperly. (a) Design a circuit to provide a voltage proportional to the linear acceleration such that 10 mV = 1 m/s<sup>2</sup>. (b) How many sensor-circuit combinations does this application require?

DP

59. A floating sensor in a certain fuel tank is connected to a variable resistor (often called a potentiometer) such that a full tank (100 liters) corresponds to 1 Ω and an empty tank corresponds to 10 Ω. (a) Design a circuit that provides an output voltage which indicates the amount of fuel remaining, so that 1 V = empty and 5 V = full. (b) Design a circuit to indicate the rate of fuel consumption by providing a voltage output calibrated to yield 1 V = 1 l/s.

## 7.6 Duality

60. (a) Draw the exact dual of the circuit depicted in Fig. 7.73. (b) Label the new (dual) variables. (c) Write nodal equations for both circuits.

61. (a) Draw the exact dual of the simple circuit shown in Fig. 7.74. (b) Label the new (dual) variables. (c) Write mesh equations for both circuits.

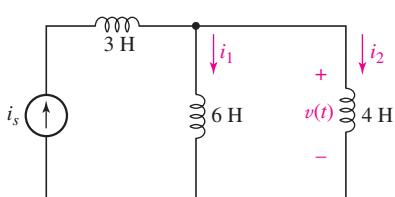


FIGURE 7.73

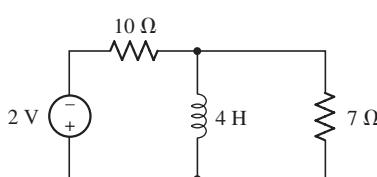
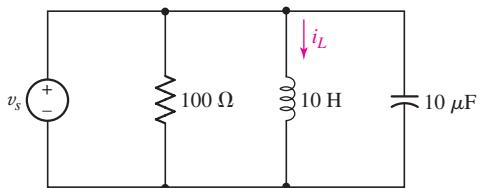


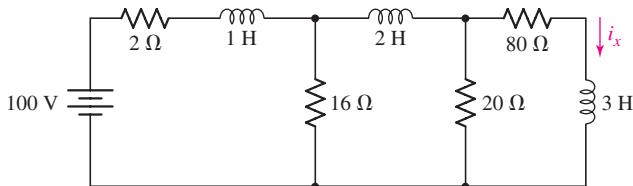
FIGURE 7.74

62. (a) Draw the exact dual of the simple circuit shown in Fig. 7.75. (b) Label the new (dual) variables. (c) Write mesh equations for both circuits.



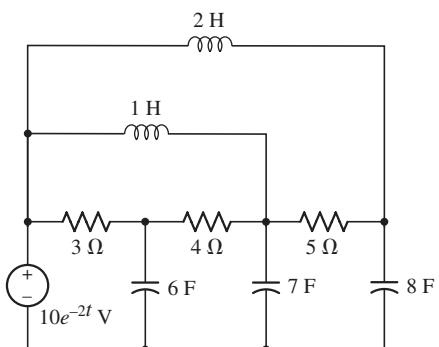
■ FIGURE 7.75

63. (a) Draw the exact dual of the simple circuit shown in Fig. 7.76. (b) Label the new (dual) variables. (c) Write nodal and mesh equations for both circuits.



■ FIGURE 7.76

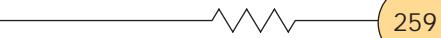
64. Draw the exact dual of the circuit shown in Fig. 7.77. Keep it neat!



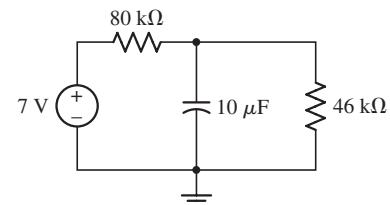
■ FIGURE 7.77

## 7.7 Modeling Capacitors and Inductors with PSpice

65. Taking the bottom node in the circuit of Fig. 7.78 as the reference terminal, calculate (a) the current through the inductor and (b) the power dissipated by the 7 Ω resistor. (c) Verify your answers with an appropriate PSpice simulation.
66. For the four-element circuit shown in Fig. 7.79, (a) calculate the power absorbed in each resistor; (b) determine the voltage across the capacitor; (c) compute the energy stored in the capacitor; and (d) verify your answers with an appropriate PSpice simulation. (Recall that calculations can be performed in Probe.)



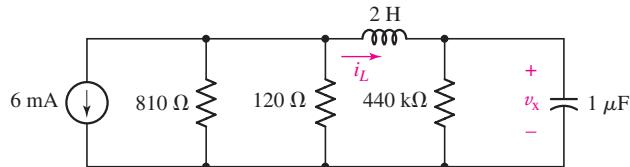
■ FIGURE 7.78



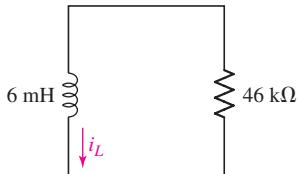
■ FIGURE 7.79



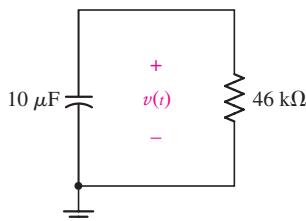
67. (a) Compute  $i_L$  and  $v_x$  as indicated in the circuit of Fig. 7.80. (b) Determine the energy stored in the inductor and in the capacitor. (c) Verify your answers with an appropriate PSpice simulation.



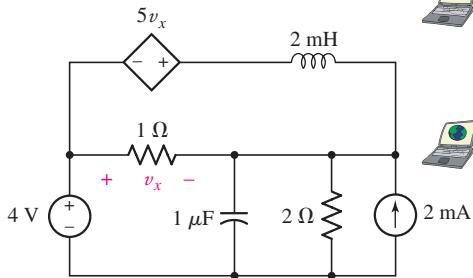
■ FIGURE 7.80



■ FIGURE 7.81



■ FIGURE 7.82



■ FIGURE 7.83



68. For the circuit depicted in Fig. 7.81, the value of  $i_L(0) = 1 \text{ mA}$ . (a) Compute the energy stored in the element at  $t = 0$ . (b) Perform a transient simulation of the circuit over the range of  $0 \leq t \leq 500 \text{ ns}$ . Determine the value of  $i_L$  at  $t = 0, 130 \text{ ns}, 260 \text{ ns}$ , and  $500 \text{ ns}$ . (c) What fraction of the initial energy remains in the inductor at  $t = 130 \text{ ns}$ ? At  $t = 500 \text{ ns}$ ?



69. Assume an initial voltage of  $9 \text{ V}$  across the  $10 \mu\text{F}$  capacitor shown in Fig. 7.82 (i.e.,  $v(0) = 9 \text{ V}$ ). (a) Compute the initial energy stored in the capacitor. (b) For  $t > 0$ , do you expect the energy to remain in the capacitor? Explain. (c) Perform a transient simulation of the circuit over the range of  $0 \leq t \leq 2.5 \text{ s}$  and determine  $v(t)$  at  $t = 460 \text{ ms}, 920 \text{ ms}$ , and  $2.3 \text{ s}$ . (c) What fraction of the initial energy remains stored in the capacitor at  $t = 460 \text{ ms}$ ? At  $t = 2.3 \text{ s}$ ?



70. Referring to the circuit of Fig. 7.83, (a) calculate the energy stored in each energy storage element; (b) verify your answers with an appropriate PSpice simulation.

### Chapter-Integrating Exercises



71. For the circuit of Fig. 7.28, (a) sketch  $v_{out}$  over the range of  $0 \leq t \leq 5 \text{ ms}$  if  $R_f = 1 \text{ k}\Omega$ ,  $C_1 = 100 \text{ mF}$ , and  $v_s$  is a  $1 \text{ kHz}$  sinusoidal source having a peak voltage of  $2 \text{ V}$ . (b) Verify your answer with an appropriate transient simulation, plotting both  $v_s$  and  $v_{out}$  in Probe. (Hint: Between plotting traces, add a second y axis using Plot, Add Y Axis. This allows both traces to be seen clearly.)



72. (a) Sketch the output function  $v_{out}$  of the amplifier circuit in Fig. 7.29 over the range of  $0 \leq t \leq 100 \text{ ms}$  if  $v_s$  is a  $60 \text{ Hz}$  sinusoidal source having a peak voltage of  $400 \text{ mV}$ ,  $R_1$  is  $1 \text{ k}\Omega$ , and  $L_f$  is  $80 \text{ nH}$ . (b) Verify your answer with an appropriate transient simulation, plotting both  $v_s$  and  $v_{out}$  in Probe. (Hint: Between plotting traces, add a second y axis using Plot, Add Y Axis. This allows both traces to be seen clearly.)



73. For the circuit of Fig. 7.71, (a) sketch  $v_{out}$  over the range of  $0 \leq t \leq 2.5 \text{ ms}$  if  $R_f = 100 \text{ k}\Omega$ ,  $L_1 = 100 \text{ mH}$ , and  $v_s$  is a  $2 \text{ kHz}$  sinusoidal source having a peak voltage of  $5 \text{ V}$ . (b) Verify your answer with an appropriate transient simulation, plotting both  $v_s$  and  $v_{out}$  in Probe. (Hint: Between plotting traces, add a second y axis using Plot, Add Y Axis. This allows both traces to be seen clearly.)



74. Consider the modified integrator depicted in Fig. 7.72. Take  $R_1 = 100 \text{ }\Omega$ ,  $R_f = 10 \text{ M}\Omega$ , and  $C_1 = 10 \text{ mF}$ . The source  $v_s$  provides a  $10 \text{ Hz}$  sinusoidal voltage having a peak amplitude of  $0.5 \text{ V}$ . (a) Sketch  $v_{out}$  over the range of  $0 \leq t \leq 500 \text{ ms}$ . (b) Verify your answer with an appropriate transient simulation, plotting both  $v_s$  and  $v_{out}$  in Probe. (Hint: Between plotting traces, add a second y axis using Plot, Add Y Axis. This allows both traces to be seen clearly.)

# Basic $RL$ and $RC$ Circuits

## INTRODUCTION

In Chap. 7 we wrote equations for the response of several circuits containing both inductance and capacitance, but we did not solve any of them. Now we are ready to proceed with the solution of the simpler circuits, namely, those which contain only resistors and inductors, or only resistors and capacitors.

Although the circuits we are about to consider have a very elementary appearance, they are also of practical importance. Networks of this form find use in electronic amplifiers, automatic control systems, operational amplifiers, communications equipment, and many other applications. Familiarity with these simple circuits will enable us to predict the accuracy with which the output of an amplifier can follow an input that is changing rapidly with time, or to predict how quickly the speed of a motor will change in response to a change in its field current. Our understanding of simple  $RL$  and  $RC$  circuits will also enable us to suggest modifications to the amplifier or motor in order to obtain a more desirable response.

### 8.1 THE SOURCE-FREE $RL$ CIRCUIT

The analysis of circuits containing inductors and/or capacitors is dependent upon the formulation and solution of the integrodifferential equations that characterize the circuits. We will call the special type of equation we obtain a *homogeneous linear differential equation*, which is simply a differential equation in which every term is of the first degree in the dependent variable or one of its derivatives. A solution is obtained when we have found an expression for the

## KEY CONCEPTS

- $RL$  and  $RC$  Time Constants
- Natural and Forced Response
- Calculating the Time-Dependent Response to DC Excitation
- How to Determine Initial Conditions and Their Effect on the Circuit Response
- Analyzing Circuits with Step Function Input and with Switches
- Construction of Pulse Waveforms Using Unit-Step Functions
- The Response of Sequentially Switched Circuits



dependent variable that satisfies both the differential equation and also the prescribed energy distribution in the inductors or capacitors at a prescribed instant of time, usually  $t = 0$ .

The solution of the differential equation represents a response of the circuit, and it is known by many names. Since this response depends upon the general “nature” of the circuit (the types of elements, their sizes, the interconnection of the elements), it is often called a **natural response**. However, any real circuit we construct cannot store energy forever; the resistances intrinsically associated with inductors and capacitors will eventually convert all stored energy into heat. The response must eventually die out, and for this reason it is frequently referred to as the **transient response**. Finally, we should also be familiar with the mathematicians’ contribution to the nomenclature; they call the solution of a homogeneous linear differential equation a **complementary function**.

When we consider independent sources acting on a circuit, part of the response will resemble the nature of the particular source (or *forcing function*) used; this part of the response, called the *particular solution*, the *steady-state response*, or the **forced response**, will be “complemented” by the complementary response produced in the source-free circuit. The complete response of the circuit will then be given by the sum of the complementary function and the particular solution. In other words, the complete response is the sum of the natural response and the forced response. The source-free response may be called the *natural response*, the *transient response*, the *free response*, or the *complementary function*, but because of its more descriptive nature, we will most often call it the *natural response*.

We will consider several different methods of solving these differential equations. The mathematical manipulation, however, is not circuit analysis. Our greatest interest lies in the solutions themselves, their meaning, and their interpretation, and we will try to become sufficiently familiar with the form of the response that we are able to write down answers for new circuits by just plain thinking. Although complicated analytical methods are needed when simpler methods fail, a well-developed intuition is an invaluable resource in such situations.

We begin our study of transient analysis by considering the simple series *RL* circuit shown in Fig. 8.1. Let us designate the time-varying current as  $i(t)$ ; we will represent the value of  $i(t)$  at  $t = 0$  as  $I_0$ ; in other words,  $i(0) = I_0$ . We therefore have

$$Ri + v_L = Ri + L \frac{di}{dt} = 0$$

or

$$\frac{di}{dt} + \frac{R}{L} i = 0 \quad [1]$$

Our goal is an expression for  $i(t)$  which satisfies this equation and also has the value  $I_0$  at  $t = 0$ . The solution may be obtained by several different methods.

### A Direct Approach

One very direct method of solving a differential equation consists of writing the equation in such a way that the variables are separated, and then

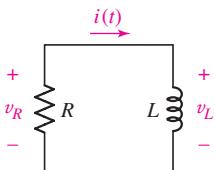
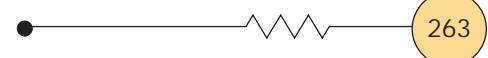


FIGURE 8.1 A series *RL* circuit for which  $i(t)$  is to be determined, subject to the initial condition that  $i(0) = I_0$ .

It may seem pretty strange to discuss a time-varying current flowing in a circuit with no sources! Keep in mind that we only know the current at the time specified as  $t = 0$ ; we don’t know the current prior to that time. In the same vein, we don’t know what the circuit looked like prior to  $t = 0$ , either. In order for a current to be flowing, a source had to have been present at some point, but we are not privy to this information. Fortunately, it is not required in order to analyze the circuit we are given.



integrating each side of the equation. The variables in Eq. [1] are  $i$  and  $t$ , and it is apparent that the equation may be multiplied by  $dt$ , divided by  $i$ , and arranged with the variables separated:

$$\frac{di}{i} = -\frac{R}{L} dt \quad [2]$$

Since the current is  $I_0$  at  $t = 0$  and  $i(t)$  at time  $t$ , we may equate the two definite integrals which are obtained by integrating each side between the corresponding limits:

$$\int_{I_0}^{i(t)} \frac{di'}{i'} = \int_0^t -\frac{R}{L} dt'$$

Performing the indicated integration,

$$\ln i'|_{I_0}^i = -\frac{R}{L} t'|_0^t$$

which results in

$$\ln i - \ln I_0 = -\frac{R}{L}(t - 0)$$

After a little manipulation, we find that the current  $i(t)$  is given by

$$i(t) = I_0 e^{-Rt/L} \quad [3]$$



We check our solution by first showing that substitution of Eq. [3] in Eq. [1] yields the identity  $0 = 0$ , and then showing that substitution of  $t = 0$  in Eq. [3] produces  $i(0) = I_0$ . Both steps are necessary; the solution must satisfy the differential equation which characterizes the circuit, and it must also satisfy the initial condition.

## EXAMPLE 8.1

If the inductor of Fig. 8.2 has a current  $i_L = 2$  A at  $t = 0$ , find an expression for  $i_L(t)$  valid for  $t > 0$ , and its value at  $t = 200 \mu s$ .

This is the identical type of circuit just considered, so we expect an inductor current of the form

$$i_L = I_0 e^{-Rt/L}$$

where  $R = 200 \Omega$ ,  $L = 50 \text{ mH}$  and  $I_0$  is the initial current flowing through the inductor at  $t = 0$ . Thus,

$$i_L(t) = 2e^{-4000t}$$

Substituting  $t = 200 \times 10^{-6}$  s, we find that  $i_L(t) = 898.7 \text{ mA}$ , less than half the initial value.

### PRACTICE

8.1 Determine the current  $i_R$  through the resistor of Fig. 8.3 at  $t = 1 \text{ ns}$  if  $i_R(0) = 6 \text{ A}$ .

Ans: 812 mA.

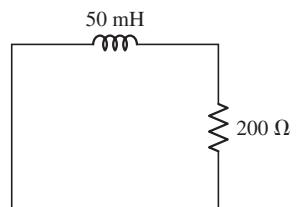


FIGURE 8.2 A simple  $RL$  circuit in which energy is stored in the inductor at  $t = 0$ .

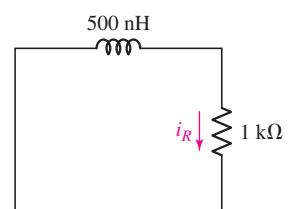


FIGURE 8.3 Circuit for Practice Problem 8.1.

## An Alternative Approach

The solution may also be obtained by a slight variation of the method we just described. After separating the variables, we now also include a constant of integration. Thus,

$$\int \frac{di}{i} = -\int \frac{R}{L} dt + K$$

and integration gives us

$$\ln i = -\frac{R}{L}t + K \quad [4]$$

The constant  $K$  cannot be evaluated by substitution of Eq. [4] in the original differential equation [1]; the identity  $0 = 0$  will result, because Eq. [4] is a solution of Eq. [1] for *any* value of  $K$  (try it out on your own). The constant of integration must be selected to satisfy the initial condition  $i(0) = I_0$ . Thus, at  $t = 0$ , Eq. [4] becomes

$$\ln I_0 = K$$

and we use this value for  $K$  in Eq. [4] to obtain the desired response

$$\ln i = -\frac{R}{L}t + \ln I_0$$

or

$$i(t) = I_0 e^{-Rt/L}$$

as before.

## A More General Solution Approach

Either of these methods can be used when the variables are separable, but this is not always the situation. In the remaining cases we will rely on a very powerful method, the success of which will depend upon our intuition or experience. We simply guess or assume a form for the solution and then test our assumptions, first by substitution in the differential equation, and then by applying the given initial conditions. Since we cannot be expected to guess the exact numerical expression for the solution, we will assume a solution containing several unknown constants and select the values for these constants in order to satisfy the differential equation and the initial conditions. Many of the differential equations encountered in circuit analysis have a solution which may be represented by the exponential function or by the sum of several exponential functions. Let us assume a solution of Eq. [1] in exponential form,

$$i(t) = A e^{s_1 t} \quad [5]$$

where  $A$  and  $s_1$  are constants to be determined. After substituting this assumed solution in Eq. [1], we have

$$A s_1 e^{s_1 t} + A \frac{R}{L} e^{s_1 t} = 0$$

or

$$\left(s_1 + \frac{R}{L}\right) A e^{s_1 t} = 0 \quad [6]$$

In order to satisfy this equation for all values of time, it is necessary that  $A = 0$ , or  $s_1 = -\infty$ , or  $s_1 = -R/L$ . But if  $A = 0$  or  $s_1 = -\infty$ , then every response is zero; neither can be a solution to our problem. Therefore, we must choose

$$s_1 = -\frac{R}{L} \quad [7]$$

and our assumed solution takes on the form

$$i(t) = A e^{-Rt/L}$$

The remaining constant must be evaluated by applying the initial condition  $i(0) = I_0$ . Thus,  $A = I_0$ , and the final form of the assumed solution is (again)

$$i(t) = I_0 e^{-Rt/L}$$

A summary of the basic approach is outlined in Fig. 8.4.

## A Direct Route: The Characteristic Equation

In fact, there is a more direct route that we can take. In obtaining Eq. [7], we solved

$$s_1 + \frac{R}{L} = 0 \quad [8]$$

which is known as the **characteristic equation**. We can obtain the characteristic equation directly from the differential equation, without the need for substitution of our trial solution. Consider the general first-order differential equation

$$a \frac{df}{dt} + bf = 0$$

where  $a$  and  $b$  are constants. We substitute  $s^1$  for  $df/dt$  and  $s^0$  for  $f$ , resulting in

$$a \frac{df}{dt} + bf = (as + b)f = 0$$

From this we may directly obtain the characteristic equation

$$as + b = 0$$

which has the single root  $s = -b/a$ . The solution to our differential equation is then

$$f = A e^{-bt/a}$$

This basic procedure is easily extended to second-order differential equations, as we will explore in Chap. 9.

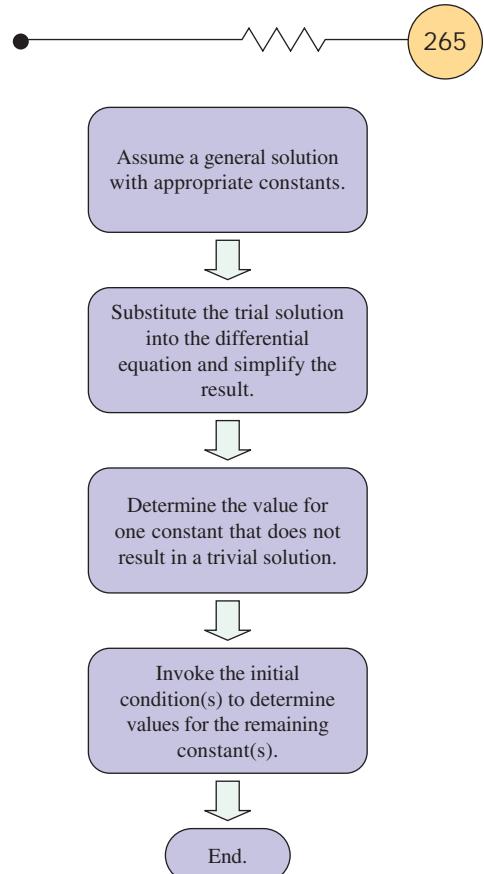


FIGURE 8.4 Flowchart for the general approach to solution of first-order differential equations where, based on experience, we can guess the form of the solution.

## EXAMPLE 8.2

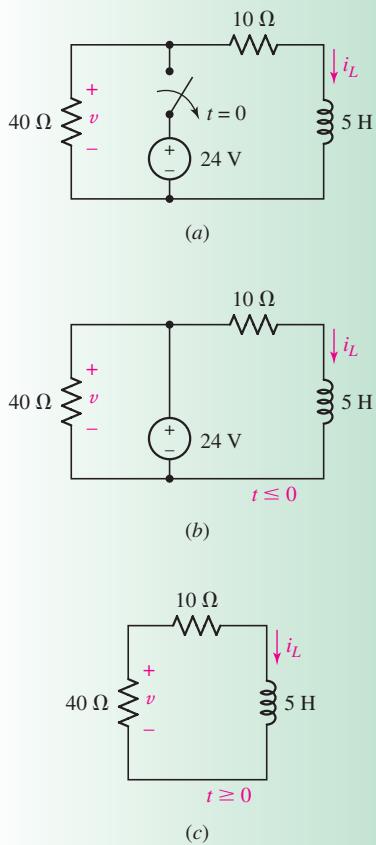


FIGURE 8.5 (a) A simple  $RL$  circuit with a switch thrown at time  $t = 0$ . (b) The circuit as it exists prior to  $t = 0$ . (c) The circuit after the switch is thrown, and the 24 V source is removed.



For the circuit of Fig. 8.5a, find the voltage labeled  $v$  at  $t = 200$  ms.

► **Identify the goal of the problem.**

The schematic of Fig. 8.5a actually represents *two different* circuits: one with the switch closed (Fig. 8.5b) and one with the switch open (Fig. 8.5c). We are asked to find  $v(0.2)$  for the circuit shown in Fig. 8.5c.

► **Collect the known information.**

Both new circuits are drawn and labeled correctly. We next make the assumption that the circuit in Fig. 8.5b has been connected for a long time, so that any transients have dissipated. We may make such an assumption as a general rule unless instructed otherwise. This circuit determines  $i_L(0)$ .

► **Devise a plan.**

The circuit of Fig. 8.5c may be analyzed by writing a KVL equation. Ultimately we want a differential equation with only  $v$  and  $t$  as variables; we will then solve the differential equation for  $v(t)$ .

► **Construct an appropriate set of equations.**

Referring to Fig. 8.5c, we write

$$-v + 10i_L + 5 \frac{di_L}{dt} = 0$$

Substituting  $i_L = -v/40$ , we find that

$$\frac{5}{40} \frac{dv}{dt} + \left( \frac{10}{40} + 1 \right)v = 0$$

or, more simply,

$$\frac{dv}{dt} + 10v = 0 \quad [9]$$

► **Determine if additional information is required.**

From previous experience, we know that a complete expression for  $v$  will require knowledge of  $v$  at a specific instant of time, with  $t = 0$  being the most convenient. We might be tempted to look at Fig. 8.5b and write  $v(0) = 24$  V, but this is only true *just before the switch opens*. The resistor voltage can change to any value in the instant that the switch is thrown; only the inductor current must remain unchanged.

In the circuit of Fig. 8.5b,  $i_L = 24/10 = 2.4$  A since the inductor acts like a short circuit to a dc current. Therefore,  $i_L(0) = 2.4$  A in the circuit of Fig. 8.5c, as well—a key point in analyzing this type of circuit. Therefore, in the circuit of Fig. 8.5c,  $v(0) = (40)(-2.4) = -96$  V.

► **Attempt a solution.**

Any of the three basic solution techniques can be brought to bear; let's start by writing the characteristic equation corresponding to Eq. [9]:

$$s + 10 = 0$$

Solving, we find that  $s = -10$ , so

$$v(t) = Ae^{-10t} \quad [10]$$

(which, upon substitution into the left-hand side of Eq. [9], results in

$$-10Ae^{-10t} + 10Ae^{-10t} = 0$$

as expected.)

We find  $A$  by setting  $t = 0$  in Eq. [10] and employing the fact that  $v(0) = -96$  V. Thus,

$$v(t) = -96e^{-10t} \quad [11]$$

and so  $v(0.2) = -12.99$  V, down from a maximum of  $-96$  V.

### ► Verify the solution. Is it reasonable or expected?

Instead of writing a differential equation in  $v$ , we could have written our differential equation in terms of  $i_L$ :

$$40i_L + 10i_L + 5 \frac{di_L}{dt} = 0$$

or

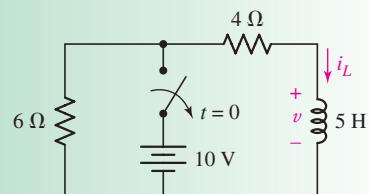
$$\frac{di_L}{dt} + 10i_L = 0$$

which has the solution  $i_L = Be^{-10t}$ . With  $i_L(0) = 2.4$ , we find that  $i_L(t) = 2.4e^{-10t}$ . Since  $v = -40i_L$ , we once again obtain Eq. [11]. We should note: it is **no coincidence** that the inductor current and the resistor voltage have the same exponential dependence!

### PRACTICE

8.2 Determine the inductor voltage  $v$  in the circuit of Fig. 8.6 for  $t > 0$ .

Ans:  $-25e^{-2t}$  V.



■ FIGURE 8.6 Circuit for Practice Problem 8.2.

## Accounting for the Energy

Before we turn our attention to the interpretation of the response, let us return to the circuit of Fig. 8.1, and check the power and energy relationships. The power being dissipated in the resistor is

$$p_R = i^2 R = I_0^2 R e^{-2Rt/L}$$

and the total energy turned into heat in the resistor is found by integrating the instantaneous power from zero time to infinite time:

$$\begin{aligned} w_R &= \int_0^\infty p_R dt = I_0^2 R \int_0^\infty e^{-2Rt/L} dt \\ &= I_0^2 R \left( \frac{-L}{2R} \right) e^{-2Rt/L} \Big|_0^\infty = \frac{1}{2} L I_0^2 \end{aligned}$$

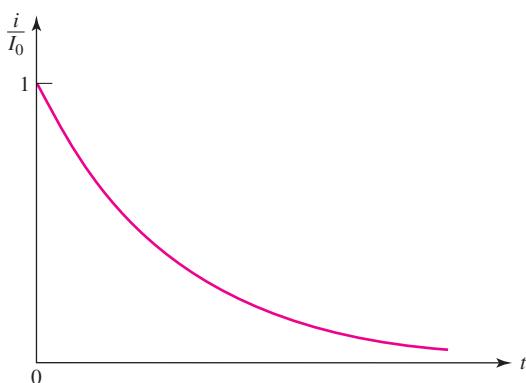
This is the result we expect, because the total energy stored initially in the inductor is  $\frac{1}{2} L I_0^2$ , and there is no longer any energy stored in the inductor at infinite time since its current eventually drops to zero. All the initial energy therefore is accounted for by dissipation in the resistor.

## 8.2 PROPERTIES OF THE EXPONENTIAL RESPONSE

Let us now consider the nature of the response in the series  $RL$  circuit. We have found that the inductor current is represented by

$$i(t) = I_0 e^{-Rt/L}$$

At  $t = 0$ , the current has value  $I_0$ , but as time increases, the current decreases and approaches zero. The shape of this decaying exponential is seen by the plot of  $i(t)/I_0$  versus  $t$  shown in Fig. 8.7. Since the function we are plotting is  $e^{-Rt/L}$ , the curve will not change if  $R/L$  remains unchanged. Thus, the same curve must be obtained for every series  $RL$  circuit having the same  $L/R$  ratio. Let us see how this ratio affects the shape of the curve.



■ FIGURE 8.7 A plot of  $e^{-Rt/L}$  versus  $t$ .

If we double the ratio of  $L$  to  $R$ , the exponent will be unchanged only if  $t$  is also doubled. In other words, the original response will occur at a later time, and the new curve is obtained by moving each point on the original curve twice as far to the right. With this larger  $L/R$  ratio, the current takes longer to decay to any given fraction of its original value. We might have a tendency to say that the “width” of the curve is doubled, or that the width is proportional to  $L/R$ . However, we find it difficult to define our term *width*, because each curve extends from  $t = 0$  to  $\infty$ ! Instead, let us consider the time that would be required for the current to drop to zero *if it continued to drop at its initial rate*.

The initial rate of decay is found by evaluating the derivative at zero time:

$$\left. \frac{d}{dt} \frac{i}{I_0} \right|_{t=0} = -\frac{R}{L} e^{-Rt/L} \Big|_{t=0} = -\frac{R}{L}$$

We designate the value of time it takes for  $i/I_0$  to drop from unity to zero, assuming a constant rate of decay, by the Greek letter  $\tau$  (tau). Thus,

$$\left( \frac{R}{L} \right) \tau = 1$$

or

$$\boxed{\tau = \frac{L}{R}}$$

[12]

The ratio  $L/R$  has the units of seconds, since the exponent  $-Rt/L$  must be dimensionless. This value of time  $\tau$  is called the **time constant** and is shown pictorially in Fig. 8.8. The time constant of a series  $RL$  circuit may be found graphically from the response curve; it is necessary only to draw the tangent to the curve at  $t = 0$  and determine the intercept of this tangent line with the time axis. This is often a convenient way of approximating the time constant from the display on an oscilloscope.

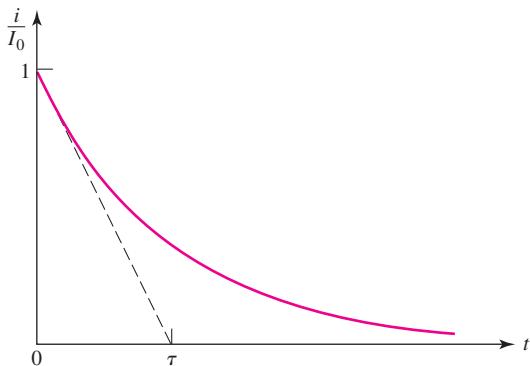


FIGURE 8.8 The time constant  $\tau$  is  $L/R$  for a series  $RL$  circuit. It is the time required for the response curve to drop to zero if it decays at a constant rate equal to its initial rate of decay.

An equally important interpretation of the time constant  $\tau$  is obtained by determining the value of  $i(t)/I_0$  at  $t = \tau$ . We have

$$\frac{i(\tau)}{I_0} = e^{-1} = 0.3679 \quad \text{or} \quad i(\tau) = 0.3679 I_0$$

Thus, in one time constant the response has dropped to 36.8 percent of its initial value; the value of  $\tau$  may also be determined graphically from this fact, as indicated by Fig. 8.9. It is convenient to measure the decay of the current at intervals of one time constant, and recourse to a hand calculator shows that  $i(t)/I_0$  is 0.3679 at  $t = \tau$ , 0.1353 at  $t = 2\tau$ , 0.04979 at  $t = 3\tau$ , 0.01832 at  $t = 4\tau$ , and 0.006738 at  $t = 5\tau$ . At some point three to five time constants after zero time, most of us would agree that the current is a

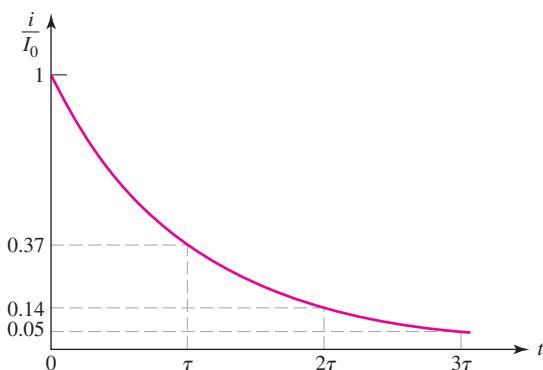


FIGURE 8.9 The current in a series  $RL$  circuit is reduced to 37 percent of its initial value at  $t = \tau$ , 14 percent at  $t = 2\tau$ , and 5 percent at  $t = 3\tau$ .



negligible fraction of its former self. Thus, if we are asked, “*How long does it take for the current to decay to zero?*” our answer might be, “*About five time constants.*” At that point, the current is less than 1 percent of its original value!

### PRACTICE

- 8.3 In a source-free series *RL* circuit, find the numerical value of the ratio: (a)  $i(2\tau)/i(\tau)$ ; (b)  $i(0.5\tau)/i(0)$ ; (c)  $t/\tau$  if  $i(t)/i(0) = 0.2$ ; (d)  $t/\tau$  if  $i(0) - i(t) = i(0) \ln 2$ .

Ans: 0.368; 0.607; 1.609; 1.181.

### COMPUTER-AIDED ANALYSIS

The transient analysis capability of PSpice is very useful when considering the response of source-free circuits. In this example, we make use of a special feature that allows us to vary a component parameter, similar to the way we varied the dc voltage in other simulations. We do this by adding the component **PARAM** to our schematic; it may be placed anywhere, as we will not wire it into the circuit. Our complete *RL* circuit is shown in Fig. 8.10, which includes an initial inductor current of 1 mA.

In order to relate our resistor value to the proposed parameter sweep, we must perform three tasks. First, we provide a name for our parameter, which we choose to call *Resistance* for the sake of simplicity. This is accomplished by double-clicking on the **PARAMETERS:** label in the schematic, which opens the Property Editor for this pseudo-component. Clicking on **New Column** results in the dialog box shown in Fig. 8.11a, in which we enter *Resistance* under **Name** and a placeholder value of 1 under **Value**. Our second task consists of linking the

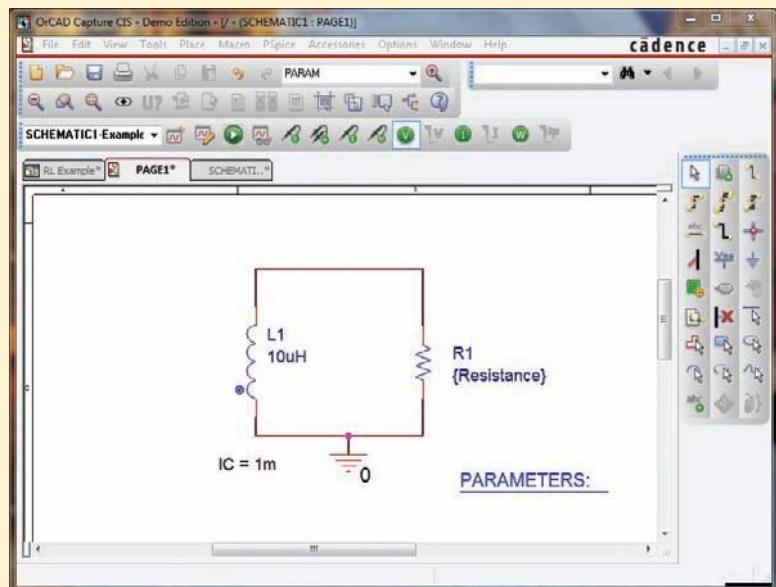
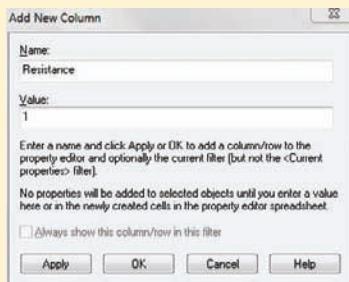
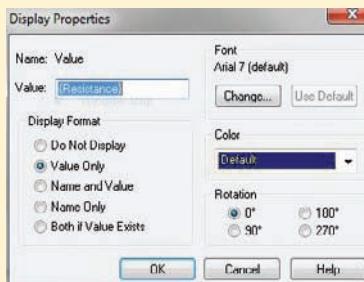


FIGURE 8.10 Simple *RL* circuit drawn using the schematic capture tool.

value of R1 to our parameter sweep, which we accomplish by double-clicking on the default value of R1 on the schematic, resulting in the dialog box of Fig. 8.11b. Under **Value**, we simply enter *{Resistance}*. (Note the curly brackets are required.)



(a)

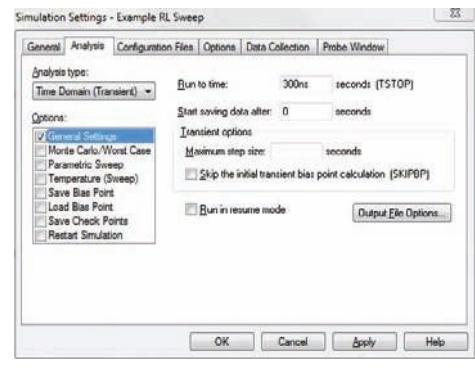


(b)

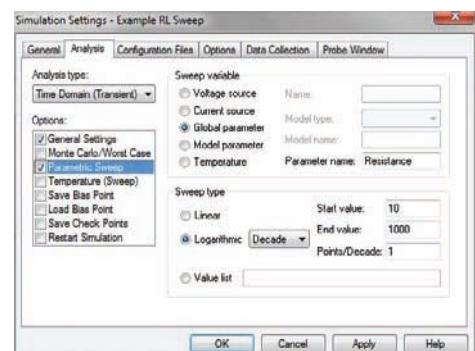
FIGURE 8.11 (a) Add New Column dialog box in the Property Editor for PARAM. (b) Resistor value dialog box.

Our third task consists of setting up the simulation, which includes setting transient analysis parameters as well as the values we desire for R1. Under PSpice we select **New Simulation Profile** (Fig. 8.12a), in which we select **Time Domain (Transient)** for **Analysis type**, 300 ns for **Run to time**, and tick the **Parametric Sweep** box under **Options**. This last action results in the dialog box shown in Fig. 8.12b, in which we select **Global parameter** for **Sweep variable** and enter *Resistance* for **Parameter name**. The final setup step required is to select **Logarithmic** under **Sweep type**, a **Start value** of 10, an **End value** of 1000, and 1 **Points/Decade**; alternatively we could list the desired resistor values using **Value list**.

After running the simulation, the notification box shown in Fig. 8.13 appears, listing the available data sets for plotting (**Resistance** = 10, 100, and 1000 in this case). A particular data set is selected by highlighting it; we select all three for this example. Upon selecting the



(a)



(b)

FIGURE 8.12 (a) Simulation dialog box.  
(b) Parameter sweep dialog box.

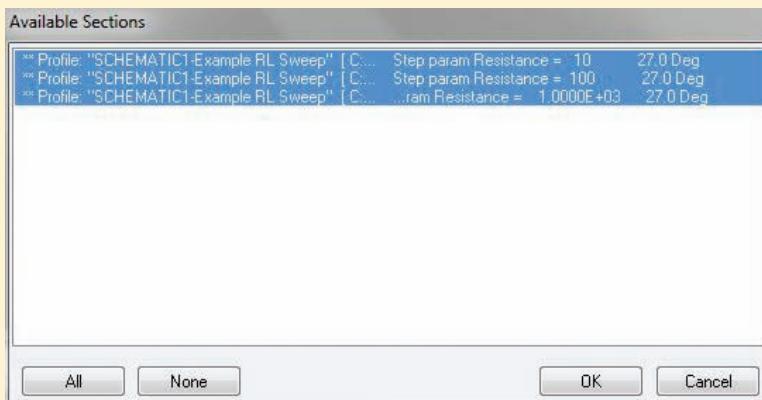
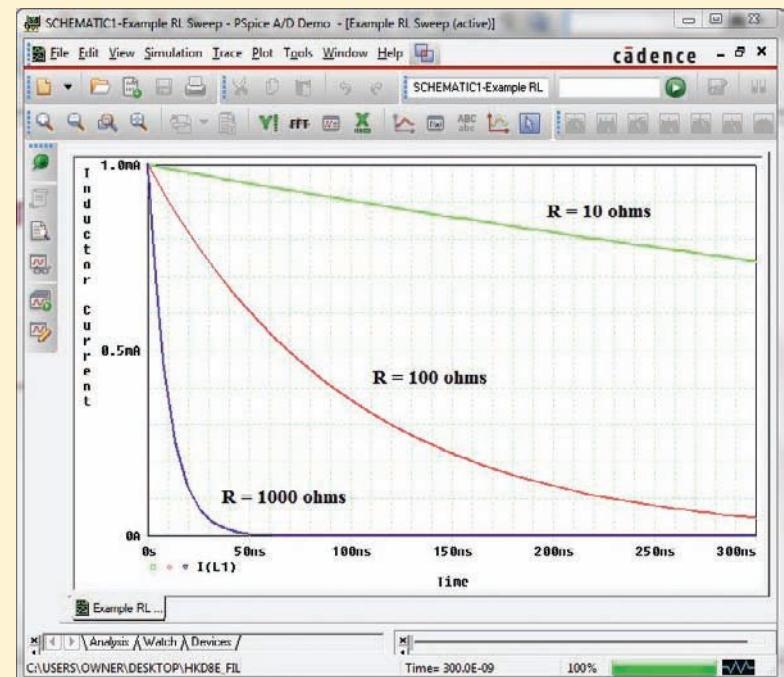


FIGURE 8.13 Available data sections dialog box.

*(Continued on next page)*

inductor current from our **Trace** variable choices in Probe, we obtain three graphs at once, as shown (after labeling by hand) in Fig. 8.14.



■ FIGURE 8.14 Probe output for the three resistances.

Why does a larger value of the time constant  $L/R$  produce a response curve that decays more slowly? Let us consider the effect of each element.

In terms of the time constant  $\tau$ , the response of the series  $RL$  circuit may be written simply as

$$i(t) = I_0 e^{-t/\tau}$$

An increase in  $L$  allows a greater energy storage for the same initial current, and this larger energy requires a longer time to be dissipated in the resistor. We may also increase  $L/R$  by reducing  $R$ . In this case, the power flowing into the resistor is less for the same initial current; again, a greater time is required to dissipate the stored energy. This effect is seen clearly in our simulation result of Fig. 8.14.

### 8.3 THE SOURCE-FREE RC CIRCUIT

Circuits based on resistor-capacitor combinations are more common than their resistor-inductor analogs. The principal reasons for this are the smaller losses present in a physical capacitor, lower cost, better agreement between the simple mathematical model and the actual device behavior, and also smaller size and lighter weight, both of which are particularly important for integrated-circuit applications.

Let us see how closely the analysis of the parallel (or is it series?)  $RC$  circuit shown in Fig. 8.15 corresponds to that of the  $RL$  circuit. We will assume an initial stored energy in the capacitor by selecting

$$v(0) = V_0$$

The total current leaving the node at the top of the circuit diagram must be zero, so we may write

$$C \frac{dv}{dt} + \frac{v}{R} = 0$$

Division by  $C$  gives us

$$\frac{dv}{dt} + \frac{v}{RC} = 0 \quad [13]$$

Equation [13] has a familiar form; comparison with Eq. [1]

$$\frac{di}{dt} + \frac{R}{L}i = 0 \quad [1]$$

shows that the replacement of  $i$  by  $v$  and  $L/R$  by  $RC$  produces the identical equation we considered previously. It should, for the  $RC$  circuit we are now analyzing is the dual of the  $RL$  circuit we considered first. This *duality* forces  $v(t)$  for the  $RC$  circuit and  $i(t)$  for the  $RL$  circuit to have identical expressions if the resistance of one circuit is equal to the reciprocal of the resistance of the other circuit, and if  $L$  is numerically equal to  $C$ . Thus, the response of the  $RL$  circuit

$$i(t) = i(0)e^{-Rt/L} = I_0e^{-Rt/L}$$

enables us to immediately write

$$v(t) = v(0)e^{-t/RC} = V_0e^{-t/RC} \quad [14]$$

for the  $RC$  circuit.

Suppose instead that we had selected the current  $i$  as our variable in the  $RC$  circuit, rather than the voltage  $v$ . Applying Kirchhoff's voltage law,

$$\frac{1}{C} \int_{t_0}^t i dt' - v_0(t_0) + Ri = 0$$

we obtain an integral equation as opposed to a differential equation. However, taking the time derivative of both sides of this equation,

$$\frac{i}{C} + R \frac{di}{dt} = 0 \quad [15]$$

and replacing  $i$  with  $v/R$ , we obtain Eq. [13] again:

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

Equation [15] could have been used as our starting point, but the application of duality principles would not have been as natural.

Let us discuss the physical nature of the voltage response of the  $RC$  circuit as expressed by Eq. [14]. At  $t = 0$  we obtain the correct initial condition, and as  $t$  becomes infinite, the voltage approaches zero. This latter result agrees with our thinking that if there were any voltage remaining across the capacitor, then energy would continue to flow into the resistor and be dissipated as heat. *Thus, a final voltage of zero is necessary.* The time constant of the  $RC$

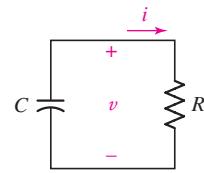


FIGURE 8.15 A parallel  $RC$  circuit for which  $v(t)$  is to be determined, subject to the initial condition that  $v(0) = V_0$ .

circuit may be found by using the duality relationships on the expression for the time constant of the  $RL$  circuit, or it may be found by simply noting the time at which the response has dropped to 37 percent of its initial value:

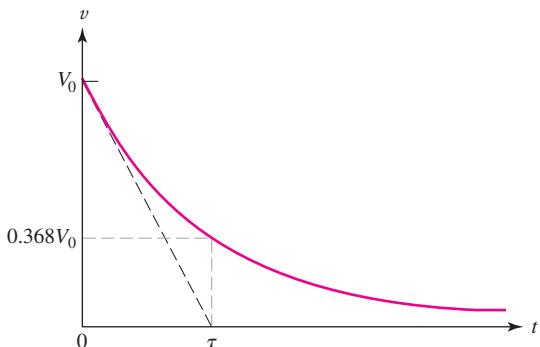
$$\frac{\tau}{RC} = 1$$

so that

$$\tau = RC$$

[16]

Our familiarity with the negative exponential and the significance of the time constant  $\tau$  enables us to sketch the response curve readily (Fig. 8.16). Larger values of  $R$  or  $C$  provide larger time constants and slower dissipation of the stored energy. A larger resistance will dissipate a smaller power with a given voltage across it, thus requiring a greater time to convert the stored energy into heat; a larger capacitance stores a larger energy with a given voltage across it, again requiring a greater time to lose this initial energy.



■ FIGURE 8.16 The capacitor voltage  $v(t)$  in the parallel  $RC$  circuit is plotted as a function of time. The initial value of  $v(t)$  is  $V_0$ .

### EXAMPLE 8.3

For the circuit of Fig. 8.17a, find the voltage labeled  $v$  at  $t = 200 \mu\text{s}$ .

To find the requested voltage, we will need to draw and analyze two separate circuits: one corresponding to before the switch is thrown (Fig. 8.17b), and one corresponding to after the switch is thrown (Fig. 8.17c).

The sole purpose of analyzing the circuit of Fig. 8.17b is to obtain an initial capacitor voltage; we assume any transients in that circuit died out long ago, leaving a purely dc circuit. With no current through either the capacitor or the  $4 \Omega$  resistor, then,

$$v(0) = 9 \text{ V} \quad [17]$$

We next turn our attention to the circuit of Fig. 8.17c, recognizing that

$$\tau = RC = (2 + 4)(10 \times 10^{-6}) = 60 \times 10^{-6} \text{ s}$$

Thus, from Eq. [14],

$$v(t) = v(0)e^{-t/RC} = v(0)e^{-t/60 \times 10^{-6}} \quad [18]$$

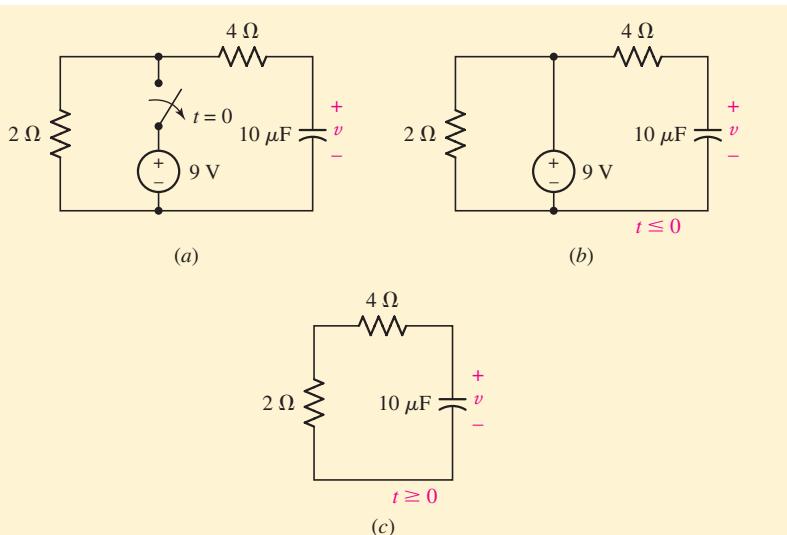


FIGURE 8.17 (a) A simple  $RC$  circuit with a switch thrown at time  $t = 0$ . (b) The circuit as it exists prior to  $t = 0$ . (c) The circuit after the switch is thrown, and the 9 V source is removed.

The capacitor voltage must be the same in both circuits at  $t = 0$ ; no such restriction is placed on any other voltage or current. Substituting Eq. [17] into Eq. [18],

$$v(t) = 9e^{-t/60 \times 10^{-6}} \text{ V}$$

so that  $v(200 \times 10^{-6}) = 321.1 \text{ mV}$  (less than 4 percent of its maximum value).

### PRACTICE

8.4 Noting carefully how the circuit changes once the switch in the circuit of Fig. 8.18 is thrown, determine  $v(t)$  at  $t = 0$  and at  $t = 160 \mu\text{s}$ .

Ans: 50 V, 18.39 V.

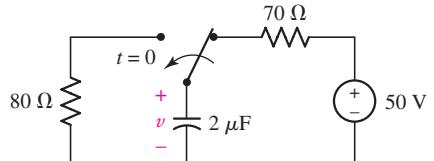
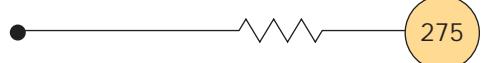


FIGURE 8.18

## 8.4 A MORE GENERAL PERSPECTIVE

As seen indirectly from Examples 8.2 and 8.3, regardless of how many resistors we have in the circuit, we obtain a single time constant (either  $\tau = L/R$  or  $\tau = RC$ ) when only one energy storage element is present. We can formalize this by realizing that the value needed for  $R$  is in fact the Thévenin equivalent resistance seen by our energy storage element. (*Strange as it may seem, it is even possible to compute a time constant for a circuit containing dependent sources!*)

### General $RL$ Circuits

As an example, consider the circuit shown in Fig. 8.19. The equivalent resistance the inductor faces is

$$R_{\text{eq}} = R_3 + R_4 + \frac{R_1 R_2}{R_1 + R_2}$$

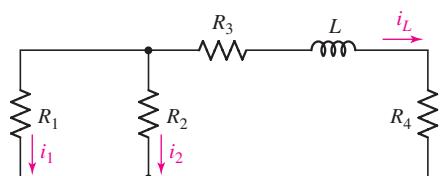


FIGURE 8.19 A source-free circuit containing one inductor and several resistors is analyzed by determining the time constant  $\tau = L/R_{\text{eq}}$ .

and the time constant is therefore

$$\tau = \frac{L}{R_{\text{eq}}} \quad [19]$$

We could also write

$$\tau = \frac{L}{R_{TH}},$$

where  $R_{TH}$  is the Thévenin equivalent resistance "seen" by the inductor  $L$ .

If several inductors are present in a circuit and can be combined using series and/or parallel combination, then Eq. [19] can be further generalized to

$$\tau = \frac{L_{\text{eq}}}{R_{\text{eq}}} \quad [20]$$

where  $L_{\text{eq}}$  represents the equivalent inductance.

### Slicing Thinly: The Distinction Between $0^+$ and $0^-$

Let's return to the circuit of Fig. 8.19, and assume that some finite amount of energy is stored in the inductor at  $t = 0$ , so that  $i_L(0) \neq 0$ .

The inductor current  $i_L$  is

$$i_L = i_L(0)e^{-t/\tau}$$

and this represents what we might call the basic solution to the problem. It is quite possible that some current or voltage other than  $i_L$  is needed, such as the current  $i_2$  in  $R_2$ . We can always apply Kirchhoff's laws and Ohm's law to the resistive portion of the circuit without any difficulty, but current division provides the quickest answer in this circuit:

$$i_2 = -\frac{R_1}{R_1 + R_2}[i_L(0)e^{-t/\tau}]$$

It may also happen that we know the initial value of some current *other* than the inductor current. Since *the current in a resistor may change instantaneously*, we will indicate the instant after any change that might have occurred at  $t = 0$  by the use of the symbol  $0^+$ ; in more mathematical language,  $i_1(0^+)$  is the limit from the right of  $i_1(t)$  as  $t$  approaches zero.<sup>1</sup> Thus, if we are given the initial value of  $i_1$  as  $i_1(0^+)$ , then the initial value of  $i_2$  is

$$i_2(0^+) = i_1(0^+) \frac{R_1}{R_2}$$

From these values, we obtain the necessary initial value of  $i_L$ :

$$i_L(0^+) = -[i_1(0^+) + i_2(0^+)] = -\frac{R_1 + R_2}{R_2}i_1(0^+)$$

and the expression for  $i_2$  becomes

$$i_2 = i_1(0^+) \frac{R_1}{R_2} e^{-t/\tau}$$

Let us see if we can obtain this last expression more directly. Since the inductor current decays exponentially as  $e^{-t/\tau}$ , *every current throughout the circuit must follow the same functional behavior*. This is made clear by considering the inductor current as a source current that is being applied to a resistive network. Every current and voltage in the resistive network must have the same time dependence. Using these ideas, we therefore express  $i_2$  as

$$i_2 = Ae^{-t/\tau}$$

where

$$\tau = \frac{L}{R_{\text{eq}}}$$

(1) Note that this is a notational convenience *only*. When faced with  $t = 0^+$  or its companion  $t = 0^-$  in an equation, we simply use the value zero. This notation allows us to clearly differentiate between the time before and after an event, such as a switch opening or closing, or a power supply being turned on or off.



Note that  $i_L(0^+)$  is *always* equal to  $i_L(0^-)$ . This is not necessarily true for the inductor voltage or any resistor voltage or current, since they may change in zero time.

and  $A$  must be determined from a knowledge of the initial value of  $i_2$ . Since  $i_1(0^+)$  is known, the voltage across  $R_1$  and  $R_2$  is known, and

$$R_2 i_2(0^+) = R_1 i_1(0^+)$$

leads to

$$i_2(0^+) = i_1(0^+) \frac{R_1}{R_2}$$

Therefore,

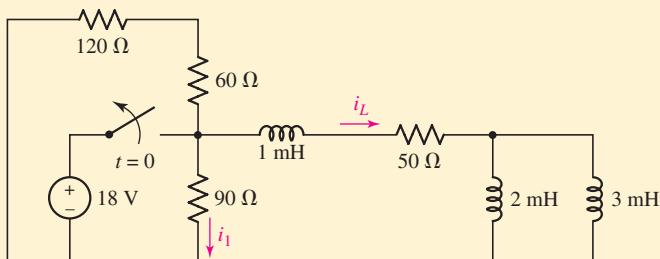
$$i_2(t) = i_1(0^+) \frac{R_1}{R_2} e^{-t/\tau}$$

A similar sequence of steps will provide a rapid solution to a large number of problems. We first recognize the time dependence of the response as an exponential decay, determine the appropriate time constant by combining resistances, write the solution with an unknown amplitude, and then determine the amplitude from a given initial condition.

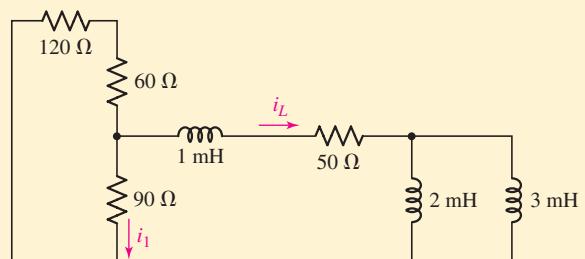
This same technique can be applied to any circuit with one inductor and any number of resistors, as well as to those special circuits containing two or more inductors and also two or more resistors that may be simplified by resistance or inductance combination to one inductor and one resistor.

## EXAMPLE 8.4

Determine both  $i_1$  and  $i_L$  in the circuit shown in Fig. 8.20a for  $t > 0$ .



(a)



(b)

FIGURE 8.20 (a) A circuit with multiple resistors and inductors. (b) After  $t = 0$ , the circuit simplifies to an equivalent resistance of  $110 \Omega$  in series with  $L_{eq} = 2.2 \text{ mH}$ .

After  $t = 0$ , when the voltage source is disconnected as shown in Fig. 8.20b, we easily calculate an equivalent inductance,

$$L_{eq} = \frac{2 \times 3}{2 + 3} + 1 = 2.2 \text{ mH}$$

an equivalent resistance, in series with the equivalent inductance,

$$R_{eq} = \frac{90(60 + 120)}{90 + 180} + 50 = 110 \Omega$$

(Continued on next page)

and the time constant,

$$\tau = \frac{L_{\text{eq}}}{R_{\text{eq}}} = \frac{2.2 \times 10^{-3}}{110} = 20 \mu\text{s}$$

Thus, the form of the natural response is  $Ke^{-50,000t}$ , where  $K$  is an unknown constant. Considering the circuit just prior to the switch opening ( $t = 0^-$ ),  $i_L = 18/50 \text{ A}$ . Since  $i_L(0^+) = i_L(0^-)$ , we know that  $i_L = 18/50 \text{ A}$  or  $360 \text{ mA}$  at  $t = 0^+$  and so

$$i_L = \begin{cases} 360 \text{ mA} & t < 0 \\ 360e^{-50,000t} \text{ mA} & t \geq 0 \end{cases}$$

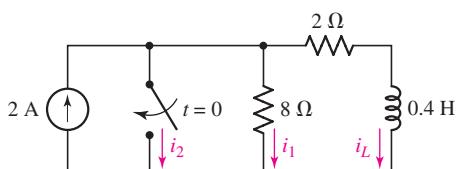
There is no restriction on  $i_1$  changing instantaneously at  $t = 0$ , so its value at  $t = 0^-$  ( $18/90 \text{ A}$  or  $200 \text{ mA}$ ) is not relevant to finding  $i_1$  for  $t > 0$ . Instead, we must find  $i_1(0^+)$  through our knowledge of  $i_L(0^+)$ . Using current division,

$$i_1(0^+) = -i_L(0^+) \frac{120 + 60}{120 + 60 + 90} = -240 \text{ mA}$$

Hence,

$$i_1 = \begin{cases} 200 \text{ mA} & t < 0 \\ -240e^{-50,000t} \text{ mA} & t \geq 0 \end{cases}$$

We can verify our analysis using PSpice and the switch model **Sw\_tOpen**, although it should be remembered that this part is actually just two resistance values: one corresponding to before the switch opens at the specified time (the default value is  $10 \text{ m}\Omega$ ), and one for after the switch opens (the default value is  $1 \text{ M}\Omega$ ). If the equivalent resistance of the remainder of the circuit is comparable to either value, the values should be edited by double-clicking on the switch symbol in the circuit schematic. Note that there is also a switch model that closes at a specified time: **Sw\_tClos**.



■ FIGURE 8.21

### PRACTICE

- 8.5 At  $t = 0.15 \text{ s}$  in the circuit of Fig. 8.21, find the value of (a)  $i_L$ ; (b)  $i_1$ ; (c)  $i_2$ .

Ans: 0.756 A; 0; 1.244 A.

We have now considered the task of finding the natural response of any circuit which can be represented by an equivalent inductor in series with an equivalent resistor. A circuit containing several resistors and several inductors does not always possess a form which allows either the resistors or the inductors to be combined into single equivalent elements. In such instances, there is no single negative exponential term or single time constant associated with the circuit. Rather, there will, in general, be several negative

exponential terms, the number of terms being equal to the number of inductors that remain after all possible inductor combinations have been made.

## General RC Circuits

Many of the *RC* circuits for which we would like to find the natural response contain more than a single resistor and capacitor. Just as we did for the *RL* circuits, we first consider those cases in which the given circuit may be reduced to an equivalent circuit consisting of only one resistor and one capacitor.

Let us suppose first that we are faced with a circuit containing a single capacitor, but any number of resistors. It is possible to replace the two-terminal resistive network which is across the capacitor terminals with an equivalent resistor, and we may then write down the expression for the capacitor voltage immediately. In such instances, the circuit has an effective time constant given by

$$\tau = R_{\text{eq}}C$$

where  $R_{\text{eq}}$  is the equivalent resistance of the network. An alternative perspective is that  $R_{\text{eq}}$  is in fact the Thévenin equivalent resistance “seen” by the capacitor.

If the circuit has more than one capacitor, but they may be replaced somehow using series and/or parallel combinations with an equivalent capacitance  $C_{\text{eq}}$ , then the circuit has an effective time constant given by

$$\tau = RC_{\text{eq}}$$

with the general case expressed as

$$\tau = R_{\text{eq}}C_{\text{eq}}$$

It is worth noting, however, that parallel capacitors replaced by an equivalent capacitance would have to have identical initial conditions.



## EXAMPLE 8.5

**Find  $v(0^+)$  and  $i_1(0^+)$  for the circuit shown in Fig. 8.22a if  $v(0^-) = V_0$ .**

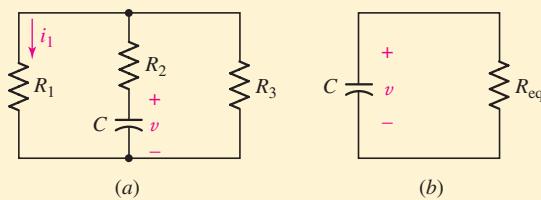


FIGURE 8.22 (a) A given circuit containing one capacitor and several resistors. (b) The resistors have been replaced by a single equivalent resistor; the time constant is simply  $\tau = R_{\text{eq}}C$ .

We first simplify the circuit of Fig. 8.22a to that of Fig. 8.22b, enabling us to write

$$v = V_0 e^{-t/R_{\text{eq}}C}$$

(Continued on next page)

where

$$v(0^+) = v(0^-) = V_0 \quad \text{and} \quad R_{\text{eq}} = R_2 + \frac{R_1 R_3}{R_1 + R_3}$$

Every current and voltage in the resistive portion of the network must have the form  $A e^{-t/R_{\text{eq}}C}$ , where  $A$  is the initial value of that current or voltage. Thus, the current in  $R_1$ , for example, may be expressed as

$$i_1 = i_1(0^+) e^{-t/\tau}$$

where

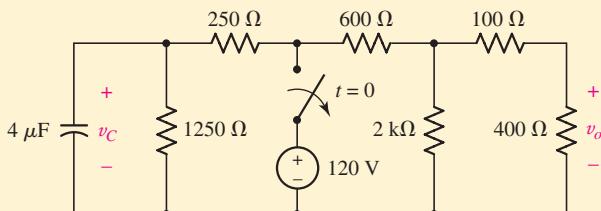
$$\tau = \left( R_2 + \frac{R_1 R_3}{R_1 + R_3} \right) C$$

and  $i_1(0^+)$  remains to be determined from the initial condition. Any current flowing in the circuit at  $t = 0^+$  must come from the capacitor. Therefore, since  $v$  cannot change instantaneously,  $v(0^+) = v(0^-) = V_0$  and

$$i_1(0^+) = \frac{V_0}{R_2 + R_1 R_3 / (R_1 + R_3)} \frac{R_3}{R_1 + R_3}$$

### PRACTICE

- 8.6 Find values of  $v_C$  and  $v_o$  in the circuit of Fig. 8.23 at  $t$  equal to  
(a)  $0^-$ ; (b)  $0^+$ ; (c) 1.3 ms.



■ FIGURE 8.23

Ans: 100 V; 38.4 V; 100 V; 25.6 V; 59.5 V; 15.22 V.

Our method can be applied to circuits with one energy storage element and one or more dependent sources as well. In such instances, we may write an appropriate KCL or KVL equation along with any necessary supporting equations, distill this down into a single differential equation, and extract the characteristic equation to find the time constant. Alternatively, we may begin by finding the Thévenin equivalent resistance of the network connected to the capacitor or inductor, and use this in calculating the appropriate  $RL$  or  $RC$  time constant—unless the dependent source is controlled by a voltage or current associated with the energy storage element, in which case the Thévenin approach cannot be used.

## EXAMPLE 8.6

For the circuit of Fig. 8.24a, find the voltage labeled  $v_C$  for  $t > 0$  if  $v_C(0^-) = 2 \text{ V}$ .

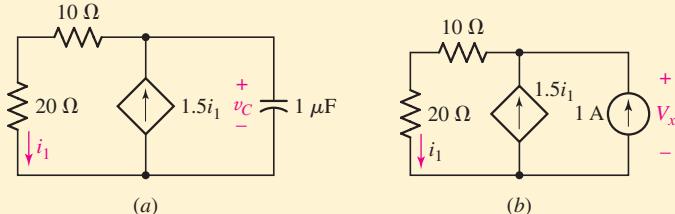


FIGURE 8.24 (a) A simple  $RC$  circuit containing a dependent source not controlled by a capacitor voltage or current. (b) Circuit for finding the Thévenin equivalent of the network connected to the capacitor.

The dependent source is not controlled by a capacitor voltage or current, so we can start by finding the Thévenin equivalent of the network to the left of the capacitor. Connecting a 1 A test source as in Fig. 8.24b,

$$V_x = (1 + 1.5i_1)(30)$$

where

$$i_1 = \left(\frac{1}{20}\right) \frac{20}{10+20} V_x = \frac{V_x}{30}$$

Performing a little algebra, we find that  $V_x = -60 \text{ V}$ , so the network has a Thévenin equivalent resistance of  $-60 \Omega$  (unusual, but not impossible when dealing with a dependent source). Our circuit therefore has a *negative* time constant

$$\tau = -60(1 \times 10^{-6}) = -60 \mu\text{s}$$

The capacitor voltage is therefore

$$v_C(t) = Ae^{t/60 \times 10^{-6}} \quad \text{V}$$

where  $A = v_C(0^+) = v_C(0^-) = 2 \text{ V}$ . Thus,

$$v_C(t) = 2e^{t/60 \times 10^{-6}} \quad \text{V} \quad [21]$$

which, interestingly enough is unstable: it grows exponentially with time. This cannot continue indefinitely; one or more elements in the circuit will eventually fail.

Alternatively, we could write a simple KCL equation for the top node of Fig. 8.24a

$$v_C = 30 \left( 1.5i_1 - 10^{-6} \frac{dv_C}{dt} \right) \quad [22]$$

where

$$i_1 = \frac{v_C}{30} \quad [23]$$

(Continued on next page)

Substituting Eq. [23] into Eq. [22] and performing some algebra, we obtain

$$\frac{dv_C}{dt} - \frac{1}{60 \times 10^{-6}} v_C = 0$$

which has the characteristic equation

$$s - \frac{1}{60 \times 10^{-6}} = 0$$

Thus,

$$s = \frac{1}{60 \times 10^{-6}}$$

and so

$$v_C(t) = Ae^{t/60 \times 10^{-6}} \quad \text{V}$$

as we found before. Substitution of  $A = v_C(0^+) = 2$  results in Eq. [21], our expression for the capacitor voltage for  $t > 0$ .

### PRACTICE

- 8.7 (a) Regarding the circuit of Fig. 8.25, determine the voltage  $v_C(t)$  for  $t > 0$  if  $v_C(0^-) = 11 \text{ V}$ . (b) Is the circuit “stable”?

Ans: (a)  $v_C(t) = 11e^{-2 \times 10^3 t/3} \text{ V}$ ,  $t > 0$ . (b) Yes; it decays (exponentially) rather than grows with time.

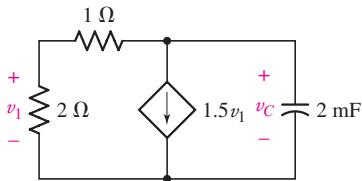


FIGURE 8.25 Circuit for Practice Problem 8.7.

Some circuits containing a number of both resistors and capacitors may be replaced by an equivalent circuit containing only one resistor and one capacitor; it is necessary that the original circuit be one which can be broken into two parts, one containing all resistors and the other containing all capacitors, such that the two parts are connected by only two ideal conductors. Otherwise, multiple time constants and multiple exponential terms will be required to describe the behavior of the circuit (one time constant for each energy storage element remaining in the circuit after it is reduced as much as possible).

As a parting comment, we should be wary of certain situations involving only ideal elements which are suddenly connected together. For example, we may imagine connecting two ideal capacitors in series having unequal voltages prior to  $t = 0$ . This poses a problem using our mathematical model of an ideal capacitor; however, real capacitors have resistances associated with them through which energy can be dissipated.



## 8.5 THE UNIT-STEP FUNCTION

We have been studying the response of *RL* and *RC* circuits when no sources or forcing functions were present. We termed this response the *natural response*, because its form depends only on the nature of the circuit. The reason that any response at all is obtained arises from the presence of initial

energy storage within the inductive or capacitive elements in the circuit. In some cases we were confronted with circuits containing sources and switches; we were informed that certain switching operations were performed at  $t = 0$  in order to remove all the sources from the circuit, while leaving known amounts of energy stored here and there. In other words, we have been solving problems in which energy sources are suddenly *removed* from the circuit; now we must consider that type of response which results when energy sources are suddenly *applied* to a circuit.

We will focus on the response which occurs when the energy sources suddenly applied are dc sources. Since every electrical device is intended to be energized at least once, and since most devices are turned on and off many times in the course of their lifetimes, our study applies to many practical cases. Even though we are now restricting ourselves to dc sources, there are still many cases in which these simpler examples correspond to the operation of physical devices. For example, the first circuit we will analyze could represent the buildup of the current when a dc motor is started. The generation and use of the rectangular voltage pulses needed to represent a number or a command in a microprocessor provide many examples in the field of electronic or transistor circuitry. Similar circuits are found in the synchronization and sweep circuits of television receivers, in communication systems using pulse modulation, and in radar systems, to name but a few examples.

We have been speaking of the “sudden application” of an energy source, and by this phrase we imply its application in zero time.<sup>2</sup> The operation of a switch in series with a battery is thus equivalent to a forcing function which is zero up to the instant that the switch is closed and is equal to the battery voltage thereafter. The forcing function has a break, or discontinuity, at the instant the switch is closed. Certain special forcing functions which are discontinuous or have discontinuous derivatives are called *singularity functions*, the two most important of these singularity functions being the *unit-step function* and the *unit-impulse function*.

We define the unit-step forcing function as a function of time which is zero for all values of its argument less than zero and which is unity for all positive values of its argument. If we let  $(t - t_0)$  be the argument and represent the unit-step function by  $u$ , then  $u(t - t_0)$  must be zero for all values of  $t$  less than  $t_0$ , and it must be unity for all values of  $t$  greater than  $t_0$ . At  $t = t_0$ ,  $u(t - t_0)$  changes abruptly from 0 to 1. Its value at  $t = t_0$  is not defined, but its value is known for all instants of time that are arbitrarily close to  $t = t_0$ . We often indicate this by writing  $u(t_0^-) = 0$  and  $u(t_0^+) = 1$ . The concise mathematical definition of the unit-step forcing function is

$$u(t - t_0) = \begin{cases} 0 & t < t_0 \\ 1 & t > t_0 \end{cases}$$

and the function is shown graphically in Fig. 8.26. Note that a vertical line of unit length is shown at  $t = t_0$ . Although this “riser” is not strictly a part of the definition of the unit step, it is usually shown in each drawing.

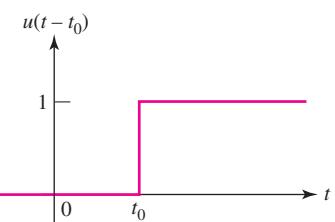
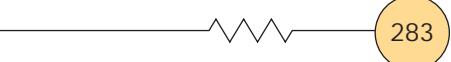


FIGURE 8.26 The unit-step forcing function,  $u(t - t_0)$ .

(2) Of course, this is not physically possible. However, if the time scale over which such an event occurs is very short compared to all other relevant time scales that describe the operation of a circuit, this is approximately true, and mathematically convenient.

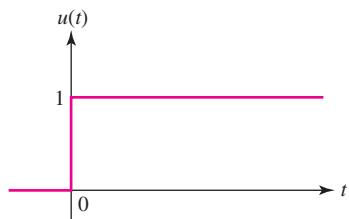


FIGURE 8.27 The unit-step forcing function  $u(t)$  is shown as a function of  $t$ .

We also note that the unit step need not be a time function. For example,  $u(x - x_0)$  could be used to denote a unit-step function where  $x$  might be a distance in meters, for example, or a frequency.

Very often in circuit analysis a discontinuity or a switching action takes place at an instant that is defined as  $t = 0$ . In that case  $t_0 = 0$ , and we then represent the corresponding unit-step forcing function by  $u(t - 0)$ , or more simply  $u(t)$ . This is shown in Fig. 8.27. Thus

$$u(t) = \begin{cases} 0 & t < 0 \\ 1 & t > 0 \end{cases}$$

The unit-step forcing function is in itself dimensionless. If we wish it to represent a voltage, it is necessary to multiply  $u(t - t_0)$  by some constant voltage, such as 5 V. Thus,  $v(t) = 5u(t - 0.2)$  V is an ideal voltage source which is zero before  $t = 0.2$  s and a constant 5 V after  $t = 0.2$  s. This forcing function is shown connected to a general network in Fig. 8.28a.

## Physical Sources and the Unit-Step Function

Perhaps we should ask what physical source is the equivalent of this discontinuous forcing function. By equivalent, we mean simply that the voltage-current characteristics of the two networks are identical. For the step-voltage source of Fig. 8.28a, the voltage-current characteristic is simple: the voltage is zero prior to  $t = 0.2$  s, it is 5 V after  $t = 0.2$  s, and the current may be any (finite) value in either time interval. Our first thoughts might produce the attempt at an equivalent shown in Fig. 8.28b, a 5 V dc source in series with a switch which closes at  $t = 0.2$  s. This network is not equivalent for  $t < 0.2$  s, however, because the voltage across the battery and switch is completely unspecified in this time interval. The “equivalent” source is an open circuit, and the voltage across it *may be anything*. After  $t = 0.2$  s, the networks are equivalent, and if this is the only time interval in which we are interested, and if the initial currents which flow from the two networks are identical at  $t = 0.2$  s, then Fig. 8.28b becomes a useful equivalent of Fig. 8.28a.

In order to obtain an exact equivalent for the voltage-step forcing function, we may provide a single-pole double-throw switch. Before  $t = 0.2$  s, the switch serves to ensure zero voltage across the input terminals of the general network. After  $t = 0.2$  s, the switch is thrown to provide a constant input voltage of 5 V. At  $t = 0.2$  s, the voltage is indeterminate (as is the step forcing function), and the battery is momentarily short-circuited (it is

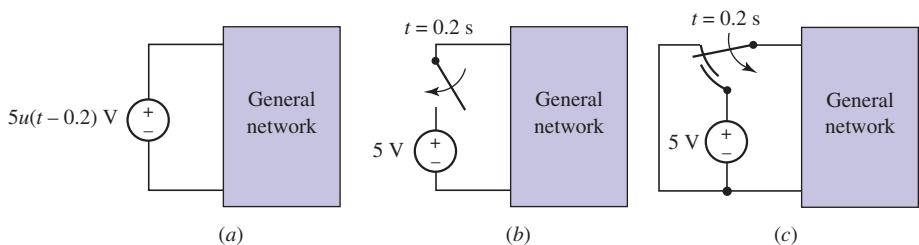


FIGURE 8.28 (a) A voltage-step forcing function is shown as the source driving a general network. (b) A simple circuit which, although not the exact equivalent of part (a), may be used as its equivalent in many cases. (c) An exact equivalent of part (a).

fortunate that we are dealing with mathematical models!). This exact equivalent of Fig. 8.28a is shown in Fig. 8.28c.

Figure 8.29a shows a current-step forcing function driving a general network. If we attempt to replace this circuit by a dc source in parallel with a switch (which opens at  $t = t_0$ ), we must realize that the circuits are equivalent after  $t = t_0$  but that the responses after  $t = t_0$  are alike only if the initial conditions are the same. The circuit in Fig. 8.29b implies no voltage exists across the current source terminals for  $t < t_0$ . This is not the case for the circuit of Fig. 8.29a. However, we may often use the circuits of Fig. 8.29a and b interchangeably. The exact equivalent of Fig. 8.29a is the dual of the circuit of Fig. 8.28c; the exact equivalent of Fig. 8.29b cannot be constructed with current- and voltage-step forcing functions alone.<sup>3</sup>

## The Rectangular Pulse Function

Some very useful forcing functions may be obtained by manipulating the unit-step forcing function. Let us define a rectangular voltage pulse by the following conditions:

$$v(t) = \begin{cases} 0 & t < t_0 \\ V_0 & t_0 < t < t_1 \\ 0 & t > t_1 \end{cases}$$

The pulse is drawn in Fig. 8.30. Can this pulse be represented in terms of the unit-step forcing function? Let us consider the difference of the two unit steps,  $u(t - t_0) - u(t - t_1)$ . The two step functions are shown in Fig. 8.31a, and their difference is a rectangular pulse. The source  $V_0u(t - t_0) - V_0u(t - t_1)$  which provides us with the desired voltage is indicated in Fig. 8.31b.

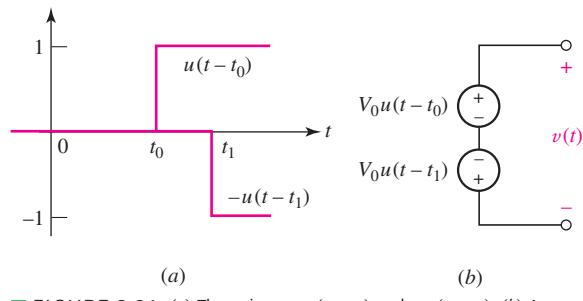


FIGURE 8.31 (a) The unit steps  $u(t - t_0)$  and  $-u(t - t_1)$ . (b) A source which yields the rectangular voltage pulse of Fig. 8.30.

If we have a sinusoidal voltage source  $V_m \sin \omega t$  which is suddenly connected to a network at  $t = t_0$ , then an appropriate voltage forcing function would be  $v(t) = V_m u(t - t_0) \sin \omega t$ . If we wish to represent one burst of energy from the transmitter for a radio-controlled car operating at 47 MHz (295 Mrad/s), we may turn the sinusoidal source off 70 ns later by a second unit-step forcing function.<sup>4</sup> The voltage pulse is thus

$$v(t) = V_m [u(t - t_0) - u(t - t_0 - 7 \times 10^{-8})] \sin(295 \times 10^6 t)$$

This forcing function is sketched in Fig. 8.32.

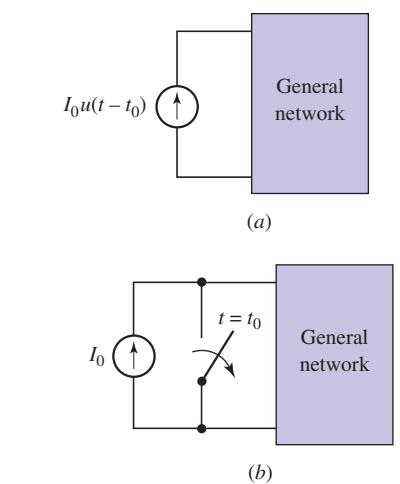


FIGURE 8.29 (a) A current-step forcing function is applied to a general network. (b) A simple circuit which, although not the exact equivalent of part (a), may be used as its equivalent in many cases.

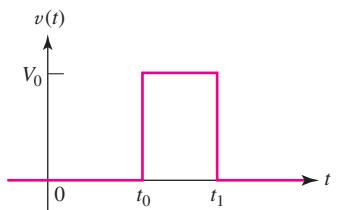
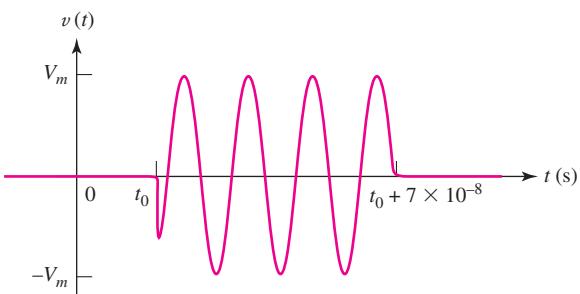


FIGURE 8.30 A useful forcing function, the rectangular voltage pulse.

(3) The equivalent can be drawn if the current through the switch prior to  $t = t_0$  is known.

(4) Apparently, we're pretty good at the controls of this car. A reaction time of 70 ns?



■ FIGURE 8.32 A 47 MHz radio-frequency pulse, described by  $v(t) = V_m[u(t - t_0) - u(t - t_0 - 7 \times 10^{-8})] \sin(259 \times 10^6 t)$ .

### PRACTICE

- 8.8 Evaluate each of the following at  $t = 0.8$ : (a)  $3u(t) - 2u(-t) + 0.8u(1-t)$ ; (b)  $[4u(t)]u(-t)$ ; (c)  $2u(t) \sin \pi t$ .

Ans: 3.8; 0; 1.176.

## 8.6 DRIVEN RL CIRCUITS

We are now ready to subject a simple network to the sudden application of a dc source. The circuit consists of a battery whose voltage is  $V_0$  in series with a switch, a resistor  $R$ , and an inductor  $L$ . The switch is closed at  $t = 0$ , as indicated on the circuit diagram of Fig. 8.33a. It is evident that the current  $i(t)$  is zero before  $t = 0$ , and we are therefore able to replace the battery and switch by a voltage-step forcing function  $V_0u(t)$ , which also produces no response prior to  $t = 0$ . After  $t = 0$ , the two circuits are clearly identical. Hence, we seek the current  $i(t)$  either in the given circuit of Fig. 8.33a or in the equivalent circuit of Fig. 8.33b.

We will find  $i(t)$  at this time by writing the appropriate circuit equation and then solving it by separation of the variables and integration. After we obtain the answer and investigate the two parts of which it is composed, we will see that there is physical significance to each of these two terms. With a more intuitive understanding of how each term originates, we will be able to produce more rapid and more meaningful solutions to every problem involving the sudden application of any source.

Applying Kirchhoff's voltage law to the circuit of Fig. 8.33b, we have

$$Ri + L \frac{di}{dt} = V_0u(t)$$

Since the unit-step forcing function is discontinuous at  $t = 0$ , we will first consider the solution for  $t < 0$  and then for  $t > 0$ . The application of zero voltage since  $t = -\infty$  forces a zero response, so that

$$i(t) = 0 \quad t < 0$$

For positive time, however,  $u(t)$  is unity and we must solve the equation

$$Ri + L \frac{di}{dt} = V_0 \quad t > 0$$

The variables may be separated in several simple algebraic steps, yielding

$$\frac{L di}{V_0 - Ri} = dt$$

and each side may be integrated directly:

$$-\frac{L}{R} \ln(V_0 - Ri) = t + k$$

In order to evaluate  $k$ , an initial condition must be invoked. Prior to  $t = 0$ ,  $i(t)$  is zero, and thus  $i(0^-) = 0$ . Since the current in an inductor cannot change by a finite amount in zero time without being associated with an infinite voltage, we thus have  $i(0^+) = 0$ . Setting  $i = 0$  at  $t = 0$ , we obtain

$$-\frac{L}{R} \ln V_0 = k$$

and, hence,

$$-\frac{L}{R} [\ln(V_0 - Ri) - \ln V_0] = t$$

Rearranging,

$$\frac{V_0 - Ri}{V_0} = e^{-Rt/L}$$

or

$$i = \frac{V_0}{R} - \frac{V_0}{R} e^{-Rt/L} \quad t > 0 \quad [24]$$

Thus, an expression for the response valid for all  $t$  would be

$$i = \left( \frac{V_0}{R} - \frac{V_0}{R} e^{-Rt/L} \right) u(t) \quad [25]$$

## A More Direct Procedure

This is the desired solution, but it has not been obtained in the simplest manner. In order to establish a more direct procedure, let us try to interpret the two terms appearing in Eq. [25]. The exponential term has the functional form of the natural response of the  $RL$  circuit; it is a negative exponential, it approaches zero as time increases, and it is characterized by the time constant  $L/R$ . The functional form of this part of the response is thus identical with that which is obtained in the source-free circuit. However, the amplitude of this exponential term depends on the source voltage  $V_0$ . We might generalize, then, that the response will be the sum of two terms, where one term has a functional form identical to that of the source-free response, but has an amplitude that depends on the forcing function. But what of the other term?

Equation [25] also contains a constant term,  $V_0/R$ . Why is it present? The answer is simple: the natural response approaches zero as the energy is gradually dissipated, but the total response must not approach zero. Eventually the circuit behaves as a resistor and an inductor in series with a battery. Since the inductor looks like a short circuit to dc, the only current now

flowing is  $V_0/R$ . This current is a part of the response that is directly attributable to the forcing function, and we call it the *forced response*. It is the response that is present a long time after the switch is closed.



The complete response is composed of two parts, the *natural response* and the *forced response*. The natural response is a characteristic of the circuit and not of the sources. Its form may be found by considering the source-free circuit, and it has an amplitude that depends on both the initial amplitude of the source and the initial energy storage. The forced response has the characteristics of the forcing function; it is found by pretending that all switches were thrown a long time ago. Since we are presently concerned only with switches and dc sources, the forced response is merely the solution of a simple dc circuit problem.

### EXAMPLE 8.7

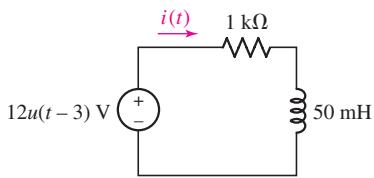


FIGURE 8.34 A simple  $RL$  circuit driven by a voltage-step forcing function.

For the circuit of Fig. 8.34, find  $i(t)$  for  $t = \infty, 3^-, 3^+$ , and  $100 \mu\text{s}$  after the source changes value.

Long after any transients have died out ( $t \rightarrow \infty$ ), the circuit is a simple dc circuit driven by a 12 V voltage source. The inductor appears as a short circuit, so

$$i(\infty) = \frac{12}{1000} = 12 \text{ mA}$$

What is meant by  $i(3^-)$ ? This is simply a notational convenience to indicate the instant before the voltage source changes value. For  $t < 3$ ,  $u(t - 3) = 0$ . Thus,  $i(3^-) = 0$  as well.

At  $t = 3^+$ , the forcing function  $12u(t - 3) = 12 \text{ V}$ . However, since the inductor current cannot change in zero time,  $i(3^+) = i(3^-) = 0$ .

The most straightforward approach to analyzing the circuit for  $t > 3 \text{ s}$  is to rewrite Eq. [25] as

$$i(t') = \left( \frac{V_0}{R} - \frac{V_0}{R} e^{-Rt'/L} \right) u(t')$$

and note that this equation applies to our circuit as well if we shift the time axis such that

$$t' = t - 3$$

Therefore, with  $V_0/R = 12 \text{ mA}$  and  $R/L = 20,000 \text{ s}^{-1}$ ,

$$i(t - 3) = (12 - 12e^{-20,000(t-3)}) u(t - 3) \quad \text{mA} \quad [26]$$

which can be written more simply as

$$i(t) = (12 - 12e^{-20,000(t-3)}) u(t - 3) \quad \text{mA} \quad [27]$$

since the unit-step function forces a zero value for  $t < 3$ , as required.

Substituting  $t = 3.0001 \text{ s}$  into Eq. [26] or [27], we find that

$i = 10.38 \text{ mA}$  at a time  $100 \mu\text{s}$  after the source changes value.

**PRACTICE**

8.9 The voltage source  $60 - 40u(t)$  V is in series with a  $10 \Omega$  resistor and a  $50 \text{ mH}$  inductor. Find the magnitudes of the inductor current and voltage at  $t$  equal to (a)  $0^-$ ; (b)  $0^+$ ; (c)  $\infty$ ; (d) 3 ms.

Ans: 6 A, 0 V; 6 A, 40 V; 2 A, 0 V; 4.20 A, 22.0 V.

## Developing an Intuitive Understanding

The reason for the two responses, forced and natural, may be seen from physical arguments. We know that our circuit will eventually assume the forced response. However, at the instant the switches are thrown, the initial inductor currents (or, in  $RC$  circuits, the voltages across the capacitors) will have values that depend only on the energy stored in these elements. These currents or voltages cannot be expected to be the same as the currents and voltages demanded by the forced response. Hence, there must be a transient period during which the currents and voltages change from their given initial values to their required final values. The portion of the response that provides the transition from initial to final values is the natural response (often called the *transient* response, as we found earlier). If we describe the response of the simple source-free  $RL$  circuit in these terms, then we should say that the forced response is zero and that the natural response serves to connect the initial response dictated by the stored energy with the zero value of the forced response.

This description is appropriate only for those circuits in which the natural response eventually dies out. This always occurs in physical circuits where some resistance is associated with every element, but there are a number of “pathologic” circuits in which the natural response is nonvanishing as time becomes infinite. Those circuits in which trapped currents circulate around inductive loops, or voltages are trapped in series strings of capacitors, are examples.

## 8.7 NATURAL AND FORCED RESPONSE

There is also an excellent mathematical reason for considering the complete response to be composed of two parts—the forced response and the natural response. The reason is based on the fact that the solution of any linear differential equation may be expressed as the sum of two parts: the *complementary solution* (natural response) and the *particular solution* (forced response). Without delving into the general theory of differential equations, let us consider a general equation of the type met in the previous section:

$$\frac{di}{dt} + Pi = Q$$

or

$$di + Pi dt = Q dt \quad [28]$$

We may identify  $Q$  as a forcing function and express it as  $Q(t)$  to emphasize its general time dependence. Let us simplify the discussion by

assuming that  $P$  is a positive constant. Later, we will also assume that  $Q$  is constant, thus restricting ourselves to dc forcing functions.

In any standard text on elementary differential equations, it is shown that if both sides of Eq. [28] are multiplied by a suitable “integrating factor,” then each side becomes an exact differential that can be integrated directly to obtain the solution. We are not separating the variables, but merely arranging them in such a way that integration is possible. For this equation, the integrating factor is  $e^{\int P dt}$  or simply  $e^{Pt}$ , since  $P$  is a constant. We multiply each side of the equation by this integrating factor and obtain

$$e^{Pt} di + iPe^{Pt} dt = Qe^{Pt} dt \quad [29]$$

The form of the left side may be simplified by recognizing it as the exact differential of  $ie^{Pt}$ :

$$d(ie^{Pt}) = e^{Pt} di + iPe^{Pt} dt$$

so that Eq. [29] becomes

$$d(ie^{Pt}) = Qe^{Pt} dt$$

Integrating each side,

$$ie^{Pt} = \int Qe^{Pt} dt + A$$

where  $A$  is a constant of integration. Multiplication by  $e^{-Pt}$  produces the solution for  $i(t)$ ,

$$i = e^{-Pt} \int Qe^{Pt} dt + Ae^{-Pt} \quad [30]$$

If our forcing function  $Q(t)$  is known, then we can obtain the functional form of  $i(t)$  by evaluating the integral. We will not evaluate such an integral for each problem, however; instead, we are interested in using Eq. [30] to draw several very general conclusions.

## The Natural Response

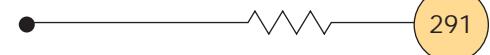
We note first that, for a source-free circuit,  $Q$  must be zero, and the solution is the natural response

$$i_n = Ae^{-Pt} \quad [31]$$

We will find that the constant  $P$  is never negative for a circuit with only resistors, inductors, and capacitors; its value depends only on the passive circuit elements<sup>5</sup> and their interconnection in the circuit. *The natural response therefore approaches zero as time increases without limit.* This must be the case for the simple *RL* circuit, because the initial energy is gradually dissipated in the resistor, leaving the circuit in the form of heat. There are also idealized circuits in which  $P$  is zero; in these circuits the natural response does not die out.

We therefore find that one of the two terms making up the complete response has the form of the natural response; it has an amplitude which will

(5) If the circuit contains a dependent source or a negative resistance,  $P$  may be negative.



depend on (but *not* always be equal to) the initial value of the complete response and thus on the initial value of the forcing function also.

## The Forced Response

We next observe that the first term of Eq. [30] depends on the functional form of  $Q(t)$ , the forcing function. Whenever we have a circuit in which the natural response dies out as  $t$  becomes infinite, this first term must completely describe the form of the response after the natural response has disappeared. This term is typically called the forced response; it is also called the *steady-state response*, the *particular solution*, or the *particular integral*.

For the present, we have elected to consider only those problems involving the sudden application of dc sources, and  $Q(t)$  will therefore be a constant for all values of time. If we wish, we can now evaluate the integral in Eq. [30], obtaining the forced response

$$i_f = \frac{Q}{P} \quad [32]$$

and the complete response

$$i(t) = \frac{Q}{P} + Ae^{-Pt} \quad [33]$$

For the  $RL$  series circuit,  $Q/P$  is the constant current  $V_0/R$  and  $1/P$  is the time constant  $\tau$ . We should see that the forced response might have been obtained without evaluating the integral, because it must be the complete response at infinite time; it is merely the source voltage divided by the series resistance. The forced response is thus obtained by inspection of the final circuit.

## Determination of the Complete Response

Let us use the simple  $RL$  series circuit to illustrate how to determine the complete response by the addition of the natural and forced responses. The circuit shown in Fig. 8.35 was analyzed earlier, but by a longer method. The desired response is the current  $i(t)$ , and we first express this current as the sum of the natural and forced current,

$$i = i_n + i_f$$

The functional form of the natural response must be the same as that obtained without any sources. We therefore replace the step-voltage source by a short circuit and recognize the old  $RL$  series loop. Thus,

$$i_n = Ae^{-Rt/L}$$

where the amplitude  $A$  is yet to be determined; since the initial condition applies to the *complete* response, we cannot simply assume  $A = i(0)$ .

We next consider the forced response. In this particular problem the forced response must be constant, because the source is a constant  $V_0$  for all positive values of time. After the natural response has died out, there can be no voltage across the inductor; hence, a voltage  $V_0$  appears across  $R$ , and the forced response is simply

$$i_f = \frac{V_0}{R}$$

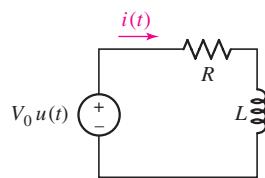


FIGURE 8.35 A series  $RL$  circuit that is used to illustrate the method by which the complete response is obtained as the sum of the natural and forced responses.



Note that the forced response is determined completely; there is no unknown amplitude. We next combine the two responses to obtain

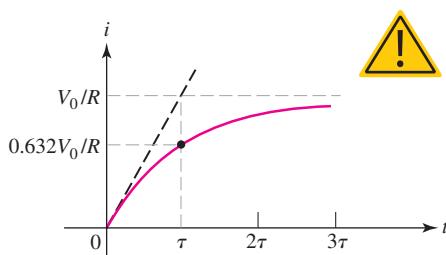
$$i = Ae^{-Rt/L} + \frac{V_0}{R}$$

and apply the initial condition to evaluate  $A$ . The current is zero prior to  $t = 0$ , and it cannot change value instantaneously since it is the current flowing through an inductor. Thus, the current is zero immediately after  $t = 0$ , and

$$0 = A + \frac{V_0}{R}$$

so

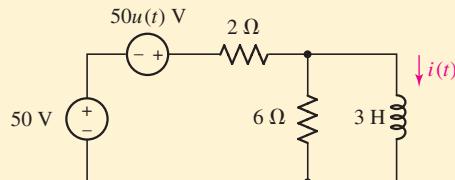
$$i = \frac{V_0}{R}(1 - e^{-Rt/L}) \quad [34]$$



**FIGURE 8.36** The current flowing through the inductor of Fig. 8.35 is shown graphically. A line extending the initial slope meets the constant forced response at  $t = \tau$ .

### EXAMPLE 8.8

Determine  $i(t)$  for all values of time in the circuit of Fig. 8.37.



**FIGURE 8.37** The circuit of Example 8.8.

The circuit contains a dc voltage source as well as a step-voltage source. We might choose to replace everything to the left of the inductor by the Thévenin equivalent, but instead let us merely recognize the form of that equivalent as a resistor in series with some voltage source. The circuit contains only one energy storage element, the inductor. We first note that

$$\tau = \frac{L}{R_{\text{eq}}} = \frac{3}{1.5} = 2 \text{ s}$$

and recall that

$$i = i_f + i_n$$

The natural response is therefore a negative exponential as before:

$$i_n = Ke^{-t/2} \quad \text{A} \quad t > 0$$

Since the forcing function is a dc source, the forced response will be a constant current. The inductor acts like a short circuit to dc, so that

$$i_f = \frac{100}{2} = 50 \text{ A}$$

Thus,

$$i = 50 + Ke^{-0.5t} \quad \text{A} \quad t > 0$$

In order to evaluate  $K$ , we must establish the initial value of the inductor current. Prior to  $t = 0$ , this current is 25 A, and it cannot change instantaneously. Thus,

$$25 = 50 + K$$

or

$$K = -25$$

Hence,

$$i = 50 - 25e^{-0.5t} \quad \text{A} \quad t > 0$$

We complete the solution by also stating

$$i = 25 \text{ A} \quad t < 0$$

or by writing a single expression valid for all  $t$ ,

$$i = 25 + 25(1 - e^{-0.5t})u(t) \quad \text{A}$$

The complete response is sketched in Fig. 8.38. Note how the natural response serves to connect the response for  $t < 0$  with the constant forced response.

### PRACTICE

- 8.10 A voltage source,  $v_s = 20u(t)$  V, is in series with a  $200 \Omega$  resistor and a  $4 \text{ H}$  inductor. Find the magnitude of the inductor current at  $t$  equal to (a)  $0^-$ ; (b)  $0^+$ ; (c) 8 ms; (d) 15 ms.

Ans: 0; 0; 33.0 mA; 52.8 mA.

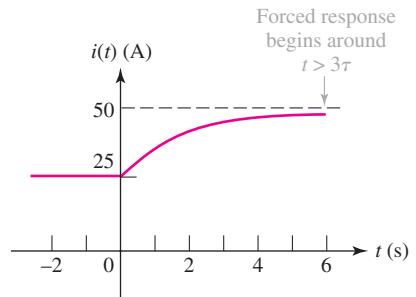
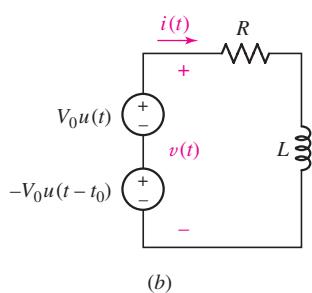
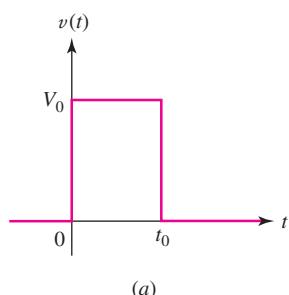


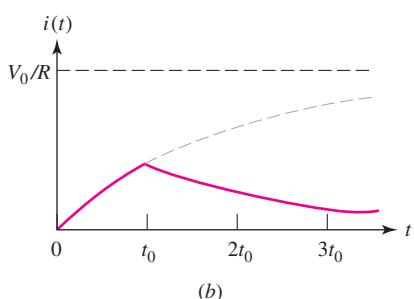
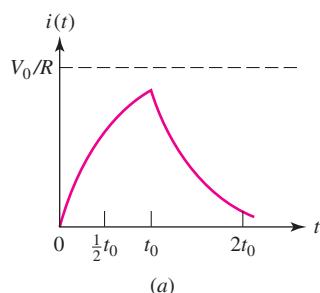
FIGURE 8.38 The response  $i(t)$  of the circuit shown in Fig. 8.37 is sketched for values of times less and greater than zero.

As a final example of this method by which the complete response of any circuit subjected to a transient may be written down *almost by inspection*, let us again consider the simple  $RL$  series circuit, but subjected to a voltage pulse.

## EXAMPLE 8.9



**FIGURE 8.39** (a) A rectangular voltage pulse which is to be used as the forcing function in a simple series  $RL$  circuit. (b) The series  $RL$  circuit, showing the representation of the forcing function by the series combination of two independent voltage-step sources. The current  $i(t)$  is desired.



**FIGURE 8.40** Two possible response curves are shown for the circuit of Fig. 8.39b. (a)  $\tau$  is selected as  $t_0/2$ . (b)  $\tau$  is selected as  $2t_0$ .

**Find the current response in a simple series  $RL$  circuit when the forcing function is a rectangular voltage pulse of amplitude  $V_0$  and duration  $t_0$ .**

We represent the forcing function as the sum of two step-voltage sources  $V_0 u(t)$  and  $-V_0 u(t - t_0)$ , as indicated in Fig. 8.39a and b, and we plan to obtain the response by using superposition. Let  $i_1(t)$  designate that part of  $i(t)$  which is due to the upper source  $V_0 u(t)$  acting alone, and let  $i_2(t)$  represent that part due to  $-V_0 u(t - t_0)$  acting alone. Then,

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

Our object is now to write each of the partial responses  $i_1$  and  $i_2$  as the sum of a natural and a forced response. The response  $i_1(t)$  is familiar; this problem was solved in Eq. [34]:

$$i_1(t) = \frac{V_0}{R} (1 - e^{-Rt/L}) \quad t > 0$$

Note that this solution is only valid for  $t > 0$  as indicated;  $i_1 = 0$  for  $t < 0$ .

We now turn our attention to the other source and its response  $i_2(t)$ . Only the polarity of the source and the time of its application are different. There is no need therefore to determine the form of the natural response and the forced response; the solution for  $i_1(t)$  enables us to write

$$i_2(t) = -\frac{V_0}{R} [1 - e^{-R(t-t_0)/L}] \quad t > t_0$$

where the applicable range of  $t$ ,  $t > t_0$ , must again be indicated; and  $i_2 = 0$  for  $t < t_0$ .

We now add the two solutions, but do so carefully, since each is valid over a different interval of time. Thus,

$$i(t) = 0 \quad t < 0 \quad [35]$$

$$i(t) = \frac{V_0}{R} (1 - e^{-Rt/L}) \quad 0 < t < t_0 \quad [36]$$

and

$$i(t) = \frac{V_0}{R} (1 - e^{-Rt/L}) - \frac{V_0}{R} (1 - e^{-R(t-t_0)/L}) \quad t > t_0$$

or more compactly,

$$i(t) = \frac{V_0}{R} e^{-Rt/L} (e^{Rt_0/L} - 1) \quad t > t_0 \quad [37]$$

Although Eqs. [35] through [37] completely describe the response of the circuit in Fig. 8.39b to the pulse waveform of Fig. 8.39a, the current waveform itself is sensitive to both the circuit time constant  $\tau$  and the voltage pulse duration  $t_0$ . Two possible curves are shown in Fig. 8.40.

The left curve is drawn for the case where the time constant is only one-half as large as the length of the applied pulse; the rising portion of the exponential has therefore almost reached  $V_0/R$  before the decaying exponential begins. The opposite situation is shown to the right; there, the time constant is twice  $t_0$  and the response never has a chance to reach the larger amplitudes.

The procedure we have been using to find the response of an *RL* circuit after dc sources have been switched on or off (or in or out of the circuit) at some instant of time is summarized in the following. We assume that the circuit is reducible to a single equivalent resistance  $R_{\text{eq}}$  in series with a single equivalent inductance  $L_{\text{eq}}$  when all independent sources are set equal to zero. The response we seek is represented by  $f(t)$ .

- With all independent sources zeroed out, simplify the circuit to determine  $R_{\text{eq}}$ ,  $L_{\text{eq}}$ , and the time constant  $\tau = L_{\text{eq}}/R_{\text{eq}}$ .
- Viewing  $L_{\text{eq}}$  as a short circuit, use dc analysis methods to find  $i_L(0^-)$ , the inductor current just prior to the discontinuity.
- Again viewing  $L_{\text{eq}}$  as a short circuit, use dc analysis methods to find the forced response. This is the value approached by  $f(t)$  as  $t \rightarrow \infty$ ; we represent it by  $f(\infty)$ .
- Write the total response as the sum of the forced and natural responses:  $f(t) = f(\infty) + Ae^{-t/\tau}$ .
- Find  $f(0^+)$  by using the condition that  $i_L(0^+) = i_L(0^-)$ . If desired,  $L_{\text{eq}}$  may be replaced by a current source  $i_L(0^+)$  [an open circuit if  $i_L(0^+) = 0$ ] for this calculation. With the exception of inductor currents (and capacitor voltages), other currents and voltages in the circuit may change abruptly.
- $f(0^+) = f(\infty) + A$  and  $f(t) = f(\infty) + [f(0^+) - f(\infty)] e^{-t/\tau}$ , or total response = final value + (initial value – final value)  $e^{-t/\tau}$ .

### PRACTICE

8.11 The circuit shown in Fig. 8.41 has been in the form shown for a very long time. The switch opens at  $t = 0$ . Find  $i_R$  at  $t$  equal to (a)  $0^-$ ; (b)  $0^+$ ; (c)  $\infty$ ; (d) 1.5 ms.

Ans: 0; 10 mA; 4 mA; 5.34 mA.

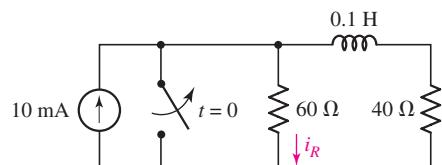


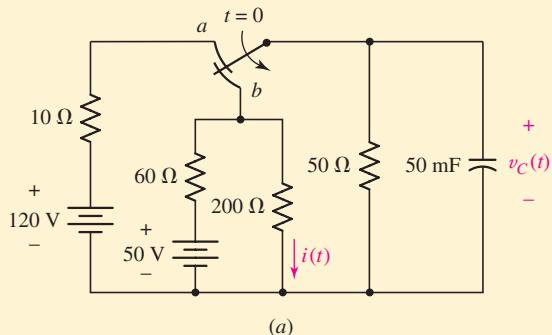
FIGURE 8.41

### 8.8 DRIVEN RC CIRCUITS

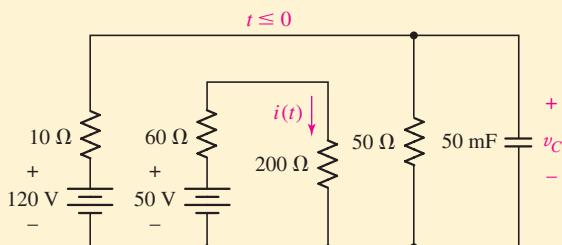
The complete response of any *RC* circuit may also be obtained as the sum of the natural and the forced response. Since the procedure is virtually identical to what we have already discussed in detail for *RL* circuits, the best approach at this stage is to illustrate it by working a relevant example completely, where the goal is not just a capacitor-related quantity but the current associated with a resistor as well.

## EXAMPLE 8.10

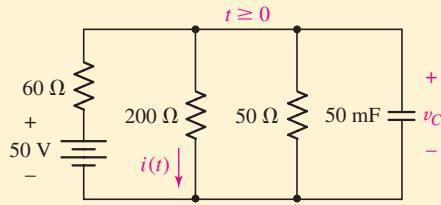
Find the capacitor voltage  $v_C(t)$  and the current  $i(t)$  in the  $200 \Omega$  resistor of Fig. 8.42 for all time.



(a)



(b)



(c)

■ FIGURE 8.42 (a) An RC circuit in which the complete responses  $v_C$  and  $i$  are obtained by adding a forced response and a natural response. (b) Circuit for  $t \leq 0$ . (c) Circuit for  $t \geq 0$ .

We begin by considering the state of the circuit at  $t < 0$ , corresponding to the switch at position  $a$  as represented in Fig. 8.42b. As usual, we assume no transients are present, so that only a forced response due to the 120 V source is relevant to finding  $v_C(0^-)$ . Simple voltage division then gives us the initial voltage,

$$v_C(0) = \frac{50}{50 + 10}(120) = 100 \text{ V}$$

Since the capacitor voltage cannot change instantaneously, this voltage is equally valid at  $t = 0^-$  and  $t = 0^+$ .

The switch is now thrown to  $b$ , and the complete response is

$$v_C = v_{Cf} + v_{Cn}$$

The corresponding circuit has been redrawn in Fig. 8.42c for convenience. The form of the natural response is obtained by replacing the 50 V source by a short circuit and evaluating the equivalent resistance to find the time constant (in other words, we are finding the Thévenin equivalent resistance “seen” by the capacitor):

$$R_{\text{eq}} = \frac{1}{\frac{1}{50} + \frac{1}{200} + \frac{1}{60}} = 24 \Omega$$

Thus,

$$v_{Cn} = Ae^{-t/R_{\text{eq}}C} = Ae^{-t/1.2}$$

In order to evaluate the forced response with the switch at *b*, we wait until all the voltages and currents have stopped changing, thus treating the capacitor as an open circuit, and use voltage division once more:

$$\begin{aligned} v_{Cf} &= 50 \left( \frac{200 \parallel 50}{60 + 200 \parallel 50} \right) \\ &= 50 \left( \frac{(50)(200)/250}{60 + (50)(200)/250} \right) = 20 \text{ V} \end{aligned}$$

Consequently,

$$v_C = 20 + Ae^{-t/1.2} \quad \text{V}$$

and from the initial condition already obtained,

$$100 = 20 + A$$

or

$$v_C = 20 + 80e^{-t/1.2} \quad \text{V} \quad t \geq 0$$

and

$$v_C = 100 \text{ V} \quad t < 0$$

This response is sketched in Fig. 8.43a; again the natural response is seen to form a transition from the initial to the final response.

Next we attack  $i(t)$ . This response need not remain constant during the instant of switching. With the contact at *a*, it is evident that  $i = 50/260 = 192.3$  milliamperes. When the switch moves to position *b*, the forced response for this current becomes

$$i_f = \frac{50}{60 + (50)(200)/(50 + 200)} \left( \frac{50}{50 + 200} \right) = 0.1 \text{ ampere}$$

The form of the natural response is the same as that which we already determined for the capacitor voltage:

$$i_n = Ae^{-t/1.2}$$

Combining the forced and natural responses, we obtain

$$i = 0.1 + Ae^{-t/1.2} \quad \text{amperes}$$

(Continued on next page)

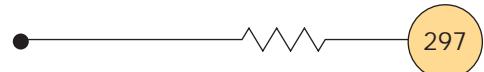
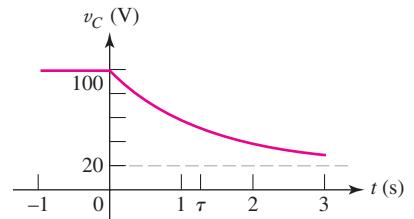
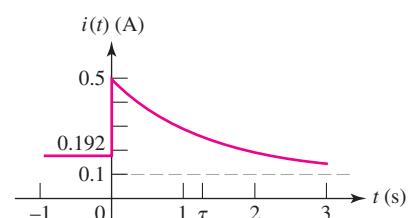


Figure 8.42 shows a circuit diagram with a 50 V DC voltage source, two resistors of 200 Ω and 60 Ω, and a 10 μF capacitor in series. A switch is positioned at node 'a'.



(a)



(b)

FIGURE 8.43 The responses (a)  $v_C$  and (b)  $i$  are plotted as functions of time for the circuit of Fig. 8.42.

To evaluate  $A$ , we need to know  $i(0^+)$ . This is found by fixing our attention on the energy-storage element (the capacitor). The fact that  $v_C$  must remain 100 V during the switching interval is the governing condition which establishes the other currents and voltages at  $t = 0^+$ . Since  $v_C(0^+) = 100$  V, and since the capacitor is in parallel with the  $200 \Omega$  resistor, we find  $i(0^+) = 0.5$  ampere,  $A = 0.4$  ampere, and thus

$$i(t) = 0.1923 \text{ ampere} \quad t < 0$$

$$i(t) = 0.1 + 0.4e^{-t/1.2} \text{ ampere} \quad t > 0$$

or

$$i(t) = 0.1923 + (-0.0923 + 0.4e^{-t/1.2})u(t) \text{ amperes}$$

where the last expression is correct for all  $t$ .

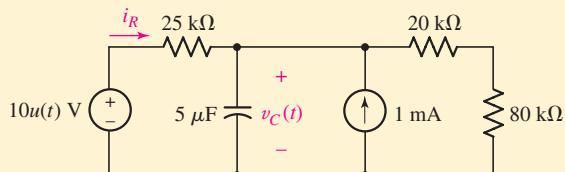
The complete response for all  $t$  may also be written concisely by using  $u(-t)$ , which is unity for  $t < 0$  and 0 for  $t > 0$ . Thus,

$$i(t) = 0.1923u(-t) + (0.1 + 0.4e^{-t/1.2})u(t) \text{ amperes}$$

This response is sketched in Fig. 8.43b. Note that only four numbers are needed to write the functional form of the response for this single-energy-storage-element circuit, or to prepare the sketch: the constant value prior to switching (0.1923 ampere), the instantaneous value just after switching (0.5 ampere), the constant forced response (0.1 ampere), and the time constant (1.2 s). The appropriate negative exponential function is then easily written or drawn.

### PRACTICE

- 8.12 For the circuit of Fig. 8.44, find  $v_C(t)$  at  $t$  equal to (a)  $0^-$ ; (b)  $0^+$ ; (c)  $\infty$ ; (d) 0.08 s.



■ FIGURE 8.44

Ans: 20 V; 20 V; 28 V; 24.4 V.

We conclude by listing the duals of the statements given at the end of Sec. 8.7.

The procedure we have been using to find the response of an  $RC$  circuit after dc sources have been switched on or off, or in or out of the circuit, at some instant of time, say  $t = 0$ , is summarized in the following. We assume that the circuit is reducible to a single equivalent resistance  $R_{\text{eq}}$  in parallel with a single equivalent capacitance  $C_{\text{eq}}$  when all independent sources are set equal to zero. The response we seek is represented by  $f(t)$ .

- With all independent sources zeroed out, simplify the circuit to determine  $R_{\text{eq}}$ ,  $C_{\text{eq}}$ , and the time constant  $\tau = R_{\text{eq}}C_{\text{eq}}$ .
- Viewing  $C_{\text{eq}}$  as an open circuit, use dc analysis methods to find  $v_C(0^-)$ , the capacitor voltage just prior to the discontinuity.
- Again viewing  $C_{\text{eq}}$  as an open circuit, use dc analysis methods to find the forced response. This is the value approached by  $f(t)$  as  $t \rightarrow \infty$ ; we represent it by  $f(\infty)$ .
- Write the total response as the sum of the forced and natural responses:  $f(t) = f(\infty) + Ae^{-t/\tau}$ .
- Find  $f(0^+)$  by using the condition that  $v_C(0^+) = v_C(0^-)$ . If desired,  $C_{\text{eq}}$  may be replaced by a voltage source  $v_C(0^+)$  [a short circuit if  $v_C(0^+) = 0$ ] for this calculation. With the exception of capacitor voltages (and inductor currents), other voltages and currents in the circuit may change abruptly.
- $f(0^+) = f(\infty) + A$  and  $f(t) = f(\infty) + [f(0^+) - f(\infty)]e^{-t/\tau}$ , or total response = final value + (initial value – final value)  $e^{-t/\tau}$ .

As we have just seen, the same basic steps that apply to the analysis of  $RL$  circuits can be applied to  $RC$  circuits as well. Up to now, we have confined ourselves to the analysis of circuits with dc forcing functions only, despite the fact that Eq. [30] holds for more general functions such as  $Q(t) = 9 \cos(5t - 7^\circ)$  or  $Q(t) = 2e^{-5t}$ . Before concluding this section, we explore one such non-dc scenario.

### EXAMPLE 8.11

Determine an expression for  $v(t)$  in the circuit of Fig. 8.45 valid for  $t > 0$ .

Based on experience, we expect a complete response of the form

$$v(t) = v_f + v_n$$

where  $v_f$  will likely resemble our forcing function and  $v_n$  will have the form  $Ae^{-t/\tau}$ .

What is the circuit time constant  $\tau$ ? We replace our source with an open circuit and find the Thévenin equivalent resistance in parallel with the capacitor:

$$R_{\text{eq}} = 4.7 + 10 = 14.7 \Omega$$

Thus, our time constant is  $\tau = R_{\text{eq}}C = 323.4 \mu\text{s}$ , or equivalently  $1/\tau = 3.092 \times 10^3 \text{ s}^{-1}$ .

There are several ways to proceed, although perhaps the most straightforward is to perform a source transformation, resulting in a voltage source  $23.5e^{-2000t}u(t) \text{ V}$  in series with  $14.7 \Omega$  and  $22 \mu\text{F}$ . (Note this does not change the time constant.)

Writing a simple KVL equation for  $t > 0$ , we find that

$$23.5e^{-2000t} = (14.7)(22 \times 10^{-6}) \frac{dv}{dt} + v$$

(Continued on next page)

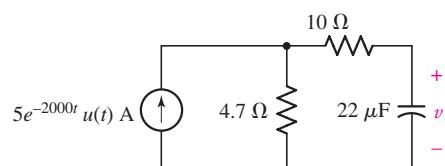


FIGURE 8.45 A simple  $RC$  circuit driven by an exponentially decaying forcing function.

A little rearranging results in

$$\frac{dv}{dt} + 3.092 \times 10^3 v = 72.67 \times 10^3 e^{-2000t}$$

which, upon comparison with Eqs. [28] and [30], allows us to write the complete response as

$$v(t) = e^{-Pt} \int Q e^{Pt} dt + A e^{-Pt}$$

where in our case  $P = 1/\tau = 3.092 \times 10^3$  and  $Q(t) = 72.67 \times 10^3 e^{-2000t}$ . We therefore find that

$$v(t) = e^{-3092t} \int 72.67 \times 10^3 e^{-2000t} e^{3092t} dt + A e^{-3092t} \quad \text{V}$$

Performing the indicated integration,

$$v(t) = 66.55 e^{-2000t} + A e^{-3092t} \quad \text{V} \quad [38]$$

Our only source is controlled by a step function with zero value for  $t < 0$ , so we know that  $v(0^-) = 0$ . Since  $v$  is a capacitor voltage,  $v(0^+) = v(0^-)$ , and we therefore find our initial condition  $v(0) = 0$  easily enough. Substituting this into Eq. [38], we find  $A = -66.55$  V and so

$$v(t) = 66.55(e^{-2000t} - e^{-3092t}) \text{ V} \quad t > 0$$

#### PRACTICE

8.13 Determine the capacitor voltage  $v$  in the circuit of Fig. 8.46 for  $t > 0$ .

Ans:  $23.5 \cos 3t + 22.8 \times 10^{-3} \sin 3t - 23.5e^{-3092t}$  V.

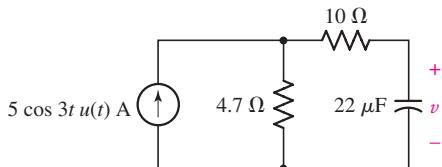
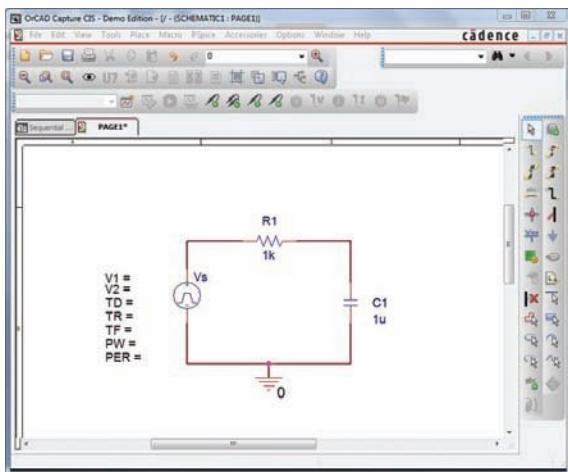


FIGURE 8.46 A simple RC circuit driven by a sinusoidal forcing function.

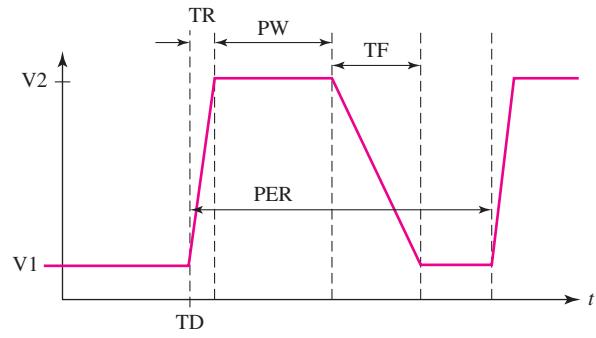
## 8.9 PREDICTING THE RESPONSE OF SEQUENTIALLY SWITCHED CIRCUITS

In Example 8.9 we briefly considered the response of an *RL* circuit to a pulse waveform, in which a source was effectively switched into and subsequently switched out of the circuit. This type of situation is common in practice, as few circuits are designed to be energized only once (passenger vehicle airbag triggering circuits, for example). In predicting the response of simple *RL* and *RC* circuits subjected to pulses and series of pulses—sometimes referred to as *sequentially switched circuits*—the key is the relative size of the circuit time constant to the various times that define the pulse sequence. The underlying principle behind the analysis will be whether the energy storage element has time to fully charge before the pulse ends, and whether it has time to fully discharge before the next pulse begins.

Consider the circuit shown in Fig. 8.47a, which is connected to a pulsed voltage source described by seven separate parameters defined in Fig. 8.47b. The waveform is bounded by two values,  $V_1$  and  $V_2$ . The time  $t_r$  required to change from  $V_1$  to  $V_2$  is called the *rise time (TR)*, and the time  $t_f$  required to change from  $V_2$  to  $V_1$  is called the *fall time (TF)*. The duration  $W_p$  of the pulse is referred to as the *pulse width (PW)*, and the *period T* of the waveform (**PER**) is the time it takes for the pulse to repeat. Note also that SPICE allows a time delay (**TD**) before the pulse train



(a)



(b)

FIGURE 8.47 (a) Schematic of a simple  $RC$  circuit connected to a pulsed voltage waveform.  
 (b) Diagram of the SPICE  $\text{VPULSE}$  parameter definitions.

begins, which can be useful in allowing initial transient responses to decay for some circuit configurations.

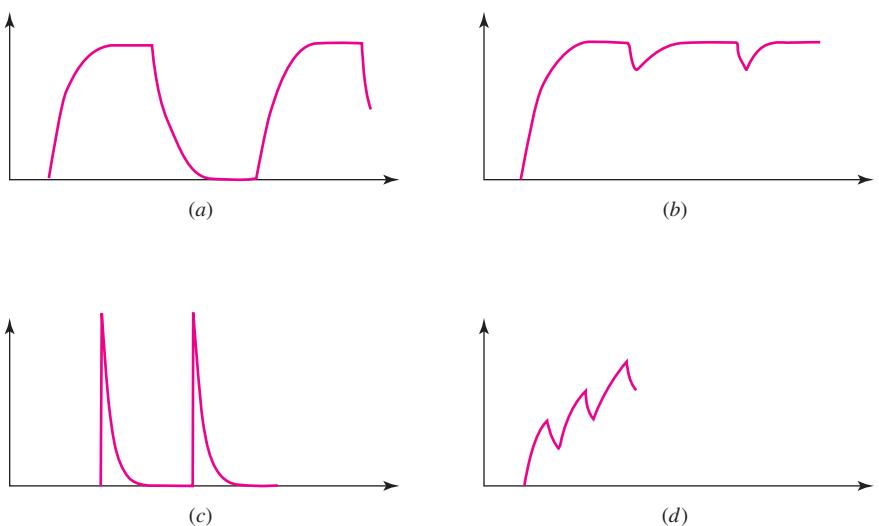
For the purposes of this discussion, we set a zero time delay,  $V1 = 0$ , and  $V2 = 9\text{ V}$ . The circuit time constant is  $\tau = RC = 1\text{ ms}$ , so we set the rise and fall times to be  $1\text{ ns}$ . Although SPICE will not allow a voltage to change in zero time since it solves the differential equations using discrete time intervals, compared to our circuit time constant  $1\text{ ns}$  is a reasonable approximation to “instantaneous.”

We will consider four basic cases, summarized in Table 8.1. In the first two cases, the pulse width  $W_p$  is much longer than the circuit time constant  $\tau$ , so we expect the transients resulting from the beginning of the pulse to die out before the pulse is over. In the latter two cases, the opposite is true: the pulse width is so short that the capacitor does not have time to fully charge before the pulse ends. A similar issue arises when we consider the response of the circuit when the time between pulses ( $T - W_p$ ) is either short (Case II) or long (Case III) compared to the circuit time constant.

TABLE 8.1 Four Separate Cases of Pulse Width and Period Relative to the Circuit Time Constant of  $1\text{ ms}$

Case	Pulse Width $W_p$	Period $T$
I	$10\text{ ms} (\tau \ll W_p)$	$20\text{ ms} (\tau \ll T - W_p)$
II	$10\text{ ms} (\tau \ll W_p)$	$10.1\text{ ms} (\tau \gg T - W_p)$
III	$0.1\text{ ms} (\tau \gg W_p)$	$10.1\text{ ms} (\tau \ll T - W_p)$
IV	$0.1\text{ ms} (\tau \gg W_p)$	$0.2\text{ ms} (\tau \gg T - W_p)$

We qualitatively sketch the circuit response for each of the four cases in Fig. 8.48, arbitrarily selecting the capacitor voltage as the quantity of interest as any voltage or current is expected to have the same time dependence.



■ FIGURE 8.48 Capacitor voltage for the  $RC$  circuit, with pulse width and period as in (a) Case I; (b) Case II; (c) Case III; and (d) Case IV.

In Case I, the capacitor has time to both fully charge and fully discharge (Fig. 8.48a), whereas in Case II (Fig. 8.48b), when the time between pulses is reduced, it no longer has time to fully discharge. In contrast, the capacitor does not have time to fully charge in either Case III (Fig. 8.48c) or Case IV (Fig. 8.48d).

### Case I: Time Enough to Fully Charge and Fully Discharge

We can obtain exact values for the response in each case, of course, by performing a series of analyses. We consider Case I first. Since the capacitor has time to fully charge, the forced response will correspond to the 9 V dc driving voltage. The complete response to the first pulse is therefore

$$v_C(t) = 9 + Ae^{-1000t} \quad \text{V}$$

With  $v_C(0) = 0$ ,  $A = -9$  V and so

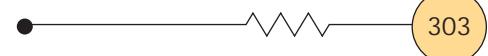
$$v_C(t) = 9(1 - e^{-1000t}) \quad \text{V} \quad [39]$$

in the interval of  $0 < t < 10$  ms. At  $t = 10$  ms, the source drops suddenly to 0 V, and the capacitor begins to discharge through the resistor. In this time interval we are faced with a simple “source-free”  $RC$  circuit, and we can write the response as

$$v_C(t) = Be^{-1000(t-0.01)} \quad 10 < t < 20 \text{ ms} \quad [40]$$

where  $B = 8.99959$  V is found by substituting  $t = 10$  ms in Eq. [39]; we will be pragmatic here and round this to 9 V, noting that the value calculated is consistent with our assumption that the initial transient dissipates before the pulse ends.

At  $t = 20$  ms, the voltage source jumps immediately back to 9 V. The capacitor voltage just prior to this event is given by substituting  $t = 20$  ms in Eq. [40], leading to  $v_C(20 \text{ ms}) = 408.6 \mu\text{V}$ , essentially zero compared to the peak value of 9 V.



If we keep to our convention of rounding to four significant digits, the capacitor voltage at the beginning of the second pulse is zero, which is the same as our starting point. Thus, Eqs. [39] and [40] form the basis of the response for all subsequent pulses, and we may write

$$v_C(t) = \begin{cases} 9(1 - e^{-1000t}) \text{ V} & 0 \leq t \leq 10 \text{ ms} \\ 9e^{-1000(t-0.01)} \text{ V} & 10 < t \leq 20 \text{ ms} \\ 9(1 - e^{-1000(t-0.02)}) \text{ V} & 20 < t \leq 30 \text{ ms} \\ 9e^{-1000(t-0.03)} \text{ V} & 30 < t \leq 40 \text{ ms} \end{cases}$$

and so on.

## Case II: Time Enough to Fully Charge But Not Fully Discharge

Next we consider what happens if the capacitor is not allowed to completely discharge (Case II). Equation [39] still describes the situation in the interval of  $0 < t < 10$  ms, and Eq. [40] describes the capacitor voltage in the interval between pulses, which has been reduced to  $10 < t < 10.1$  ms.

Just prior to the onset of the second pulse at  $t = 10.1$  ms,  $v_C$  is now 8.144 V; the capacitor has only had 0.1 ms to discharge, and therefore still retains 82 percent of its maximum energy when the next pulse begins. Thus, in the next interval,

$$v_C(t) = 9 + Ce^{-1000(t-10.1 \times 10^{-3})} \text{ V} \quad 10.1 < t < 20.1 \text{ ms}$$

where  $v_C(10.1 \text{ ms}) = 9 + C = 8.144 \text{ V}$ , so  $C = -0.856 \text{ V}$  and

$$v_C(t) = 9 - 0.856e^{-1000(t-10.1 \times 10^{-3})} \text{ V} \quad 10.1 < t < 20.1 \text{ ms}$$

which reaches the peak value of 9 V much more quickly than for the previous pulse.

## Case III: No Time to Fully Charge But Time to Fully Discharge

What if it isn't clear that the transient will dissipate before the end of the voltage pulse? In fact, this situation arises in Case III. Just as we wrote for Case I,

$$v_C(t) = 9 + Ae^{-1000t} \text{ V} \quad [41]$$

still applies to this situation, but now only in the interval  $0 < t < 0.1$  ms. Our initial condition has not changed, so  $A = -9 \text{ V}$  as before. Now, however, just before this first pulse ends at  $t = 0.1$  ms, we find that  $v_C = 0.8565 \text{ V}$ . This is a far cry from the maximum of 9 V possible if we allow the capacitor time to fully charge, and is a direct result of the pulse lasting only one-tenth of the circuit time constant.

The capacitor now begins to discharge, so that

$$v_C(t) = Be^{-1000(t-1 \times 10^{-4})} \text{ V} \quad 0.1 < t < 10.1 \text{ ms} \quad [42]$$

We have already determined that  $v_C(0.1^- \text{ ms}) = 0.8565 \text{ V}$ , so  $v_C(0.1^+ \text{ ms}) = 0.8565 \text{ V}$  and substitution into Eq. [42] yields  $B = 0.8565 \text{ V}$ . Just prior to the onset of the second pulse at  $t = 10.1$  ms, the capacitor voltage has decayed to essentially 0 V; this is the initial condition at the start of the second pulse and so Eq. [41] can be rewritten as

$$v_C(t) = 9 - 9e^{-1000(t-10.1 \times 10^{-3})} \text{ V} \quad 10.1 < t < 10.2 \text{ ms} \quad [43]$$

to describe the corresponding response.

## Case IV: No Time to Fully Charge or Even Fully Discharge

In the last case, we consider the situation where the pulse width and period are so short that the capacitor can neither fully charge nor fully discharge in any one period. Based on experience, we can write

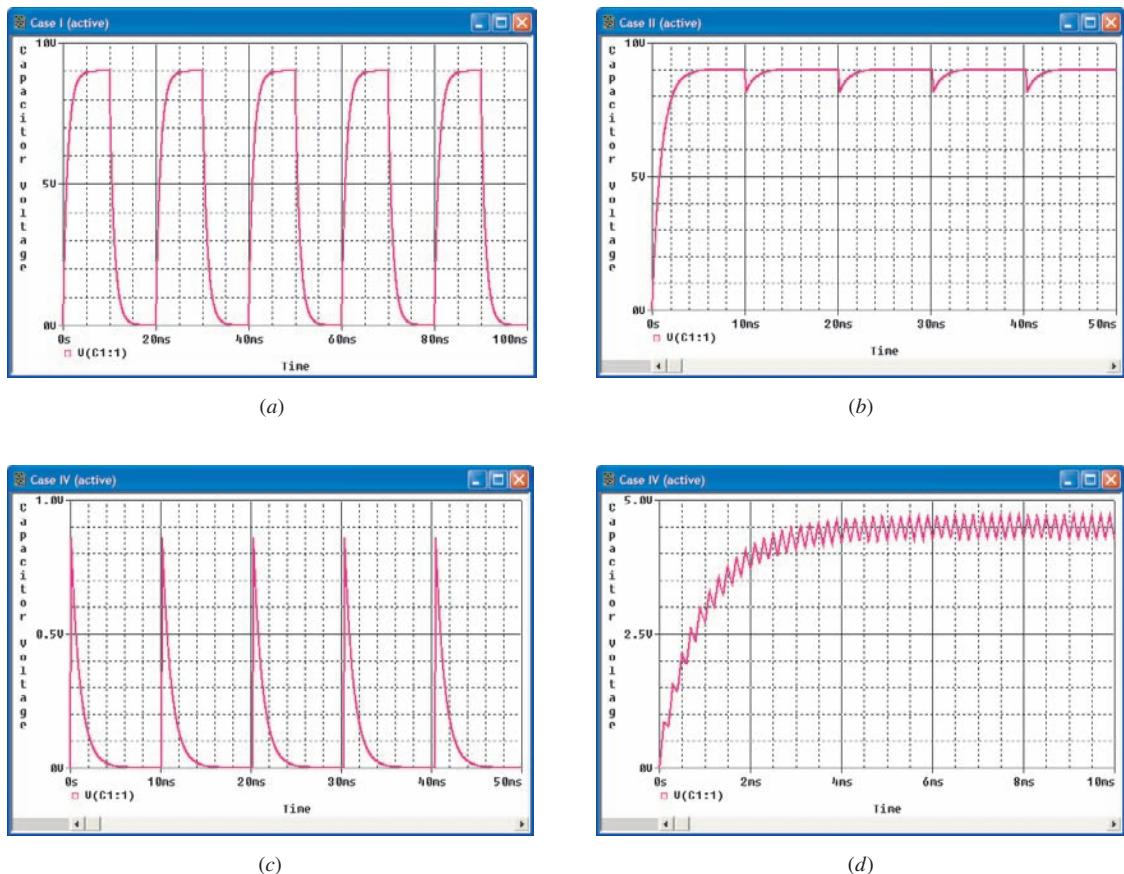
$$v_C(t) = 9 - 9e^{-1000t} \quad V \quad 0 < t < 0.1 \text{ ms} \quad [44]$$

$$v_C(t) = 0.8565e^{-1000(t-1 \times 10^{-4})} \quad V \quad 0.1 < t < 0.2 \text{ ms} \quad [45]$$

$$v_C(t) = 9 + Ce^{-1000(t-2 \times 10^{-4})} \quad V \quad 0.2 < t < 0.3 \text{ ms} \quad [46]$$

$$v_C(t) = De^{-1000(t-3 \times 10^{-4})} \quad V \quad 0.3 < t < 0.4 \text{ ms} \quad [47]$$

Just prior to the onset of the second pulse at  $t = 0.2$  ms, the capacitor voltage has decayed to  $v_C = 0.7750$  V; with insufficient time to fully discharge, it retains a large fraction of the little energy it had time to store initially. For the interval of  $0.2 < t < 0.3$  ms, substitution of  $v_C(0.2^+) = v_C(0.2^-) = 0.7750$  V into Eq. [46] yields  $C = -8.225$  V. Continuing, we evaluate Eq. [46] at  $t = 0.3$  ms and calculate  $v_C = 1.558$  V just prior to the end of the second pulse. Thus,  $D = 1.558$  V and our



■ FIGURE 8.49 PSpice simulation results corresponding to (a) Case I; (b) Case II; (c) Case III; (d) Case IV.

capacitor is slowly charging to ever increase voltage levels over several pulses. At this stage it might be useful if we plot the detailed responses, so we show the PSpice simulation results of Cases I through IV in Fig. 8.49. Note in particular that in Fig. 8.49d, the small charge/discharge transient response similar in shape to that shown in Fig. 8.49a–c is superimposed on a charging-type response of the form  $(1 - e^{-t/\tau})$ . Thus, it takes about 3 to 5 circuit time constants for the capacitor to charge to its maximum value in situations where a single period does not allow it to fully charge or discharge!

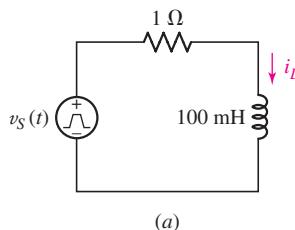
What we have not yet done is predict the behavior of the response for  $t \gg 5\tau$ , although we would be interested in doing so, especially if it was not necessary to consider a very long sequence of pulses one at a time. We note that the response of Fig. 8.49d has an *average* value of 4.50 V from about 4 ms onward. This is exactly half the value we would expect if the voltage source pulse width allowed the capacitor to fully charge. In fact, this long-term average value can be computed by multiplying the dc capacitor voltage by the ratio of the pulse width to the period.



### PRACTICE

- 8.14 With regard to Fig. 8.50a, sketch  $i_L(t)$  in the range of  $0 < t < 6$  s for  
 (a)  $v_S(t) = 3u(t) - 3u(t-2) + 3u(t-4) - 3u(t-6) + \dots$ ; (b)  $v_S(t) = 3u(t) - 3u(t-2) + 3u(t-2.1) - 3u(t-4.1) + \dots$ .

Ans: See Fig. 8.50b; see Fig. 8.50c.



(a)

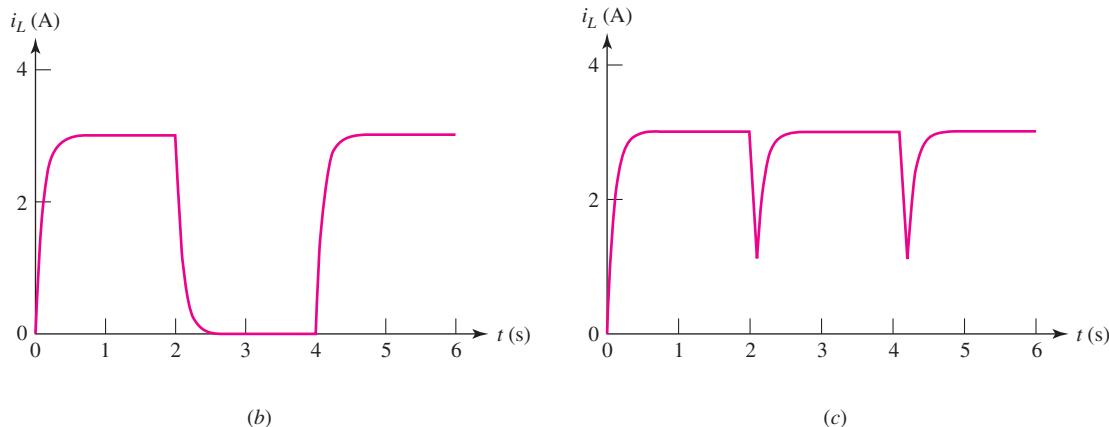
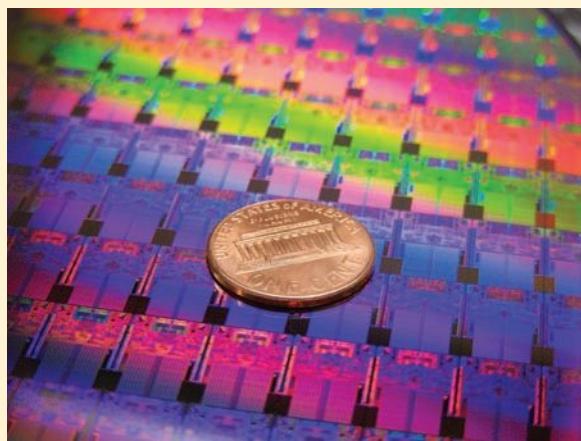


FIGURE 8.50 (a) Circuit for Practice Problem 8.14; (b) solution to part (a); (c) solution to part (b).

## PRACTICAL APPLICATION

### Frequency Limits in Digital Integrated Circuits

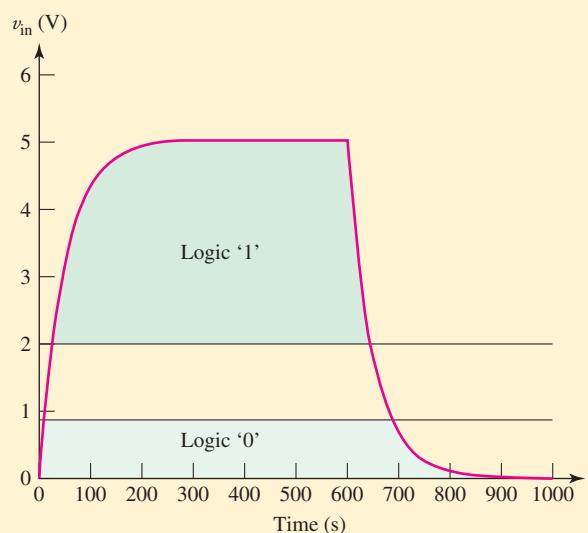
Modern digital integrated circuits such as programmable array logic (PALs) and microprocessors (Fig. 8.51) are composed of interconnected transistor circuits known as *gates*.



■ FIGURE 8.51 A silicon wafer with multiple, identical integrated circuit dies. Each die is smaller than a US 1 cent coin. Reprinted with permission of Intel Corporation.

Digital signals are represented symbolically by combinations of ones and zeros, and can be either data or instructions (such as “add” or “subtract”). Electrically, we represent a logic “1” by a “high” voltage, and a logic “0” by a “low” voltage. In practice, there is a range of voltages that correspond to each; for example, in the 7400 series of TTL logic integrated circuits, any voltage

between 2 and 5 V will be interpreted as a logic “1,” and any voltage between 0 and 0.8 V will be interpreted as a logic “0.” Voltages between 0.8 and 2 V do not correspond to either logic state, as shown in Fig. 8.52.



■ FIGURE 8.52 Charge/discharge characteristic of a pathway capacitance identifying the TTL voltage ranges for logic “1” and logic “0,” respectively.

A key parameter in digital circuits is the speed at which we can effectively use them. In this sense, “speed” refers to how quickly we can switch a gate from one logic state to another (either logic “0” to logic “1” or vice

### SUMMARY AND REVIEW

In this chapter we learned that circuits containing a single energy storage element (either an inductor or a capacitor) can be described by a characteristic time scale, namely, the *circuit time constant* ( $\tau = L/R$ , or  $\tau = RC$ , respectively). If we attempt to change the amount of energy stored in the element (either charging or discharging), *every* voltage and current in the circuit will include an exponential term of the form  $e^{-t/\tau}$ . After approximately 5 time constants from the moment we attempted to alter the amount of stored energy, the *transient* response has *essentially disappeared* and we are left simply with a *forced* response which arises from the independent sources driving the circuit at times  $t > 0$ . When determining the forced response in a purely dc circuit, we may treat inductors as short circuits and capacitors as open circuits.

versa), and the time delay required to convey the output of one gate to the input of the next gate. Although transistors contain “built-in” capacitances that affect their switching speed, it is the *interconnect pathways* that presently limit the speed of the fastest digital integrated circuits. We can model the interconnect pathway between two logic gates using a simple *RC* circuit (although as feature sizes continue to decrease in modern designs, more detailed models are required to accurately predict circuit performance). For example, consider a 2000  $\mu\text{m}$  long pathway 2  $\mu\text{m}$  wide. We can model this pathway in a typical silicon-based integrated circuit as having a capacitance of 0.5 pF and a resistance of 100  $\Omega$ , shown schematically in Fig. 8.53.

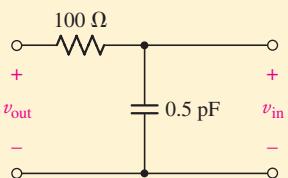


FIGURE 8.53 Circuit model for an integrated circuit pathway.

Let's assume the voltage  $v_{\text{out}}$  represents the output voltage of a gate that is changing from a logic “0” state to a logic “1” state. The voltage  $v_{\text{in}}$  appears across the input of a second gate, and we are interested in how long it takes  $v_{\text{in}}$  to reach the same value as  $v_{\text{out}}$ .

Assuming the 0.5 pF capacitance that characterizes the interconnect pathway is initially discharged [i.e.,  $v_{\text{in}}(0) = 0$ ], calculating the *RC* time constant for our

pathway as  $\tau = RC = 50 \text{ ps}$ , and defining  $t = 0$  as when  $v_{\text{out}}$  changes, we obtain the expression

$$v_{\text{in}}(t) = Ae^{-t/\tau} + v_{\text{out}}(0)$$

Setting  $v_{\text{in}}(0) = 0$ , we find that  $A = -v_{\text{out}}(0)$  so that

$$v_{\text{in}}(t) = v_{\text{out}}(0)[1 - e^{-t/\tau}]$$

Upon examining this equation, we see that  $v_{\text{in}}$  will reach the value  $v_{\text{out}}(0)$  after  $\sim 5\tau$  or 250 ps. If the voltage  $v_{\text{out}}$  changes again before this transient time period is over, then the capacitance does not have sufficient time to fully charge. In such situations,  $v_{\text{in}}$  will be less than  $v_{\text{out}}(0)$ . Assuming that  $v_{\text{out}}(0)$  equals the minimum logic “1” voltage, for example, this means that  $v_{\text{in}}$  will not correspond to a logic “1.” If  $v_{\text{out}}$  now suddenly changes to 0 V (logic “0”), the capacitance will begin to discharge so that  $v_{\text{in}}$  decreases further. Thus, by switching our logic states too quickly, we are unable to transfer the information from one gate to another.

The fastest speed at which we can change logic states is therefore  $(5\tau)^{-1}$ . This can be expressed in terms of the maximum operating frequency:

$$f_{\text{max}} = \frac{1}{2(5\tau)} = 2 \text{ GHz}$$

where the factor of 2 represents a charge/discharge period. If we desire to operate our integrated circuit at a higher frequency so that calculations can be performed faster, we need to reduce the interconnect capacitance and/or the interconnect resistance.

We started our analysis with so-called source-free circuits to introduce the idea of time constants without unnecessary distractions; such circuits have zero forced response and a transient response derived entirely from the energy stored at  $t = 0$ . We reasoned that a capacitor cannot change its voltage in zero time (or an infinite current results), and indicated this by introducing the notation  $v_C(0^+) = v_C(0^-)$ . Similarly, the current through an inductor cannot change in zero time, or  $i_L(0^+) = i_L(0^-)$ . The *complete* response is always the sum of the transient response and the forced response. Applying the initial condition to the complete response allows us to determine the unknown constant which multiplies the transient term.

We spent a little time discussing modeling switches, both analytically and within the context of PSpice. A common mathematical representation makes use of the unit-step function  $u(t - t_0)$ , which has zero value for  $t < t_0$ , unity value for  $t > t_0$ , and is indeterminate for  $t = t_0$ . Unit-step

functions can “activate” a circuit (connecting sources so current can flow) for values of  $t$  preceding a specific time as well as after. Combinations of step functions can be used to create pulses and more complex waveforms. In the case of sequentially switched circuits, where sources are connected and disconnected repeatedly, we found the behavior of the circuits to depend strongly on both period and pulse width as they compare to the circuit time constant.

This is a good time to highlight some key points worth reviewing, along with relevant example(s).

- ❑ The response of a circuit having sources suddenly switched in or out of a circuit containing capacitors and inductors will always be composed of two parts: a *natural* response and a *forced* response.
- ❑ The form of the natural response (also referred to as the *transient response*) depends only on the component values and the way they are wired together. (Examples 8.1, 8.2)
- ❑ A circuit reduced to a single equivalent capacitance  $C$  and a single equivalent resistance  $R$  will have a natural response given by  $v(t) = V_0 e^{-t/\tau}$ , where  $\tau = RC$  is the circuit time constant. (Examples 8.3, 8.5)
- ❑ A circuit reduced to a single equivalent inductance  $L$  and a single equivalent resistance  $R$  will have a natural response given by  $i(t) = I_0 e^{-t/\tau}$ , where  $\tau = L/R$  is the circuit time constant. (Example 8.4)
- ❑ Circuits with dependent sources can be represented by a resistance using Thévenin procedures.
- ❑ The unit-step function is a useful way to model the closing or opening of a switch, provided we are careful to keep an eye on the initial conditions. (Examples 8.7, 8.9)
- ❑ The form of the forced response mirrors the form of the forcing function. Therefore, a dc forcing function always leads to a constant forced response. (Examples 8.7, 8.8)
- ❑ The *complete* response of an  $RL$  or  $RC$  circuit excited by a dc source will have the form  $f(0^+) = f(\infty) + A$  and  $f(t) = f(\infty) + [f(0^+) - f(\infty)]e^{-t/\tau}$ , or total response = final value + (initial value – final value) $e^{-t/\tau}$ . (Examples 8.9, 8.10, 8.11)
- ❑ The complete response for an  $RL$  or  $RC$  circuit may also be determined by writing a single differential equation for the quantity of interest and solving. (Examples 8.2, 8.11)
- ❑ When dealing with sequentially switched circuits, or circuits connected to pulsed waveforms, the relevant issue is whether the energy storage element has sufficient time to fully charge and to fully discharge, as measured relative to the circuit time constant.

## READING FURTHER

A guide to solution techniques for differential equations can be found in:

W. E. Boyce and R. C. DiPrima, *Elementary Differential Equations and Boundary Value Problems*, 7th ed. New York: Wiley, 2002.

A detailed description of transients in electric circuits is given in:

E. Weber, *Linear Transient Analysis Volume I*. New York: Wiley, 1954.  
(Out of print, but in many university libraries.)

## EXERCISES

### 8.1 The Source-Free *RL* Circuit

- Setting  $R = 1 \text{ k}\Omega$  and  $L = 1 \text{ nH}$  for the circuit represented in Fig. 8.1, and with the knowledge that  $i(0) = -3 \text{ mA}$ , (a) write an expression for  $i(t)$  valid for all  $t \geq 0$ ; (b) compute  $i(t)$  at  $t = 0, t = 1 \text{ ps}, 2 \text{ ps}$ , and  $5 \text{ ps}$ ; and (c) calculate the energy stored in the inductor at  $t = 0, t = 1 \text{ ps}$ , and  $t = 5 \text{ ps}$ .
- If  $i(0) = 1 \text{ A}$  and  $R = 100 \Omega$  for the circuit of Fig. 8.1, (a) select  $L$  such that  $i(50 \text{ ms}) = 368 \text{ mA}$ ; (b) compute the energy stored in the inductor at  $t = 0, 50 \text{ ms}, 100 \text{ ms}$ , and  $150 \text{ ms}$ .
- Referring to the circuit shown in Fig. 8.1, select values for both elements such that  $L/R = 1$  and (a) calculate  $v_R(t)$  at  $t = 0, 1, 2, 3, 4$ , and  $5 \text{ s}$ ; (b) compute the power dissipated in the resistor at  $t = 0, 1 \text{ s}$ , and  $5 \text{ s}$ . (c) At  $t = 5 \text{ s}$ , what is the percentage of the initial energy still stored in the inductor?
- The circuit depicted in Fig. 8.1 is constructed from components whose value is unknown. If a current  $i(0)$  of  $6 \mu\text{A}$  initially flows through the inductor, and it is determined that  $i(1 \text{ ms}) = 2.207 \mu\text{A}$ , calculate the ratio of  $R$  to  $L$ .
- Determine the characteristic equation of each of the following differential equations:
  - $5v + 14 \frac{dv}{dt} = 0$ ; (b)  $-9 \frac{di}{dt} - 18i = 0$ ;
  - $\frac{di}{dt} + 18i + \frac{R}{B}i = 0$ ; (d)  $\frac{d^2f}{dt^2} + 8 \frac{df}{dt} + 2f = 0$ .
- For the following characteristic equations, write corresponding differential equations and find all roots, whether real, imaginary, or complex:
  - $4s + 9 = 0$ ; (b)  $2s - 4 = 0$ ; (c)  $s^2 + 7s + 1 = 0$ ; (d)  $5s^2 + 8s + 18 = 0$ .
- With the assumption that the switch in the circuit of Fig. 8.54 has been closed a long, long, long time, calculate  $i_L(t)$  at (a) the instant just before the switch opens; (b) the instant just after the switch opens; (c)  $t = 15.8 \mu\text{s}$ ; (d)  $t = 31.5 \mu\text{s}$ ; (e)  $t = 78.8 \mu\text{s}$ .

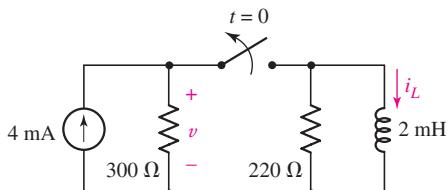


FIGURE 8.54

- The switch in Fig. 8.54 has been closed since Catfish Hunter last pitched for the New York Yankees. Calculate the voltage labeled  $v$  as well as the energy stored in the inductor at (a) the instant just prior to the switch being thrown open; (b) the instant just after the switch is opened; (c)  $t = 8 \mu\text{s}$ ; (d)  $t = 80 \mu\text{s}$ .
- The switch in the circuit of Fig. 8.55 has been closed a ridiculously long time before suddenly being thrown open at  $t = 0$ . (a) Obtain expressions for  $i_L$  and  $v$  in the circuit of Fig. 8.55 which are valid for all  $t \geq 0$ . (b) Calculate  $i_L(t)$  and

$v(t)$  at the instant just prior to the switch opening, at the instant just after the switch opening, and at  $t = 470 \mu\text{s}$ .

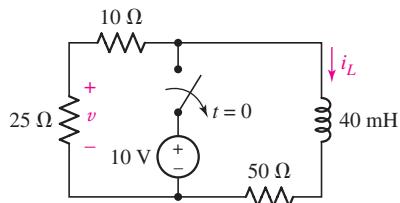


FIGURE 8.55

10. Assuming the switch initially has been open for a really, really long time,  
(a) obtain an expression for  $i_W$  in the circuit of Fig. 8.56 which is valid for all  $t \geq 0$ ; (b) calculate  $i_W$  at  $t = 0$  and  $t = 1.3 \text{ ns}$ .

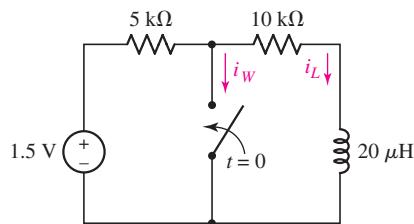


FIGURE 8.56

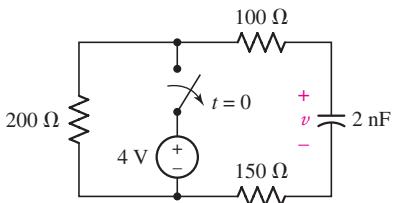
## 8.2 Properties of the Exponential Response

11. (a) Graph the function  $f(t) = 10e^{-2t}$  over the range of  $0 \leq t \leq 2.5 \text{ s}$  using linear scales for both  $y$  and  $x$  axes. (b) Replot with a logarithmic scale for the  $y$  axis. [Hint: the function `semilogy()` can be helpful here.] (c) What are the units of the 2 in the argument of the exponential? (d) At what time does the function reach a value of 9? 8? 1?
12. The current  $i(t)$  flowing through a  $1 \Omega$  resistor is given by  $i(t) = 5e^{-10t} \text{ mA}$ ,  $t \geq 0$ . (a) Determine the values of  $t$  for which the resistor voltage magnitude is equal to 5 V, 2.5 V, 0.5 V, and 5 mV. (b) Graph the function over the range of  $0 \leq t \leq 1 \text{ s}$  using linear scales for both axes. (c) Draw a tangent to your curve at  $t = 100 \text{ ms}$ , and determine where the tangent intersects the time axis.
13. The thickness of a solar cell must be chosen carefully to ensure photons are properly absorbed; even metals can be partly transparent when rolled out into very thin foils. If the incident light flux (number of photons per unit area per unit time) at the solar cell surface ( $x = 0$ ) is given by  $\Phi_0$ , and the intensity of light a distance  $x$  inside the solar cell is given by  $\Phi(x)$ , the behavior of  $\Phi(x)$  is described by the equation  $d\Phi/dx + \alpha\Phi = 0$ . Here,  $\alpha$ , known as the absorption coefficient, is a constant specific to a given semiconductor material. (a) What is the SI unit for  $\alpha$ ? (b) Obtain an expression for  $\Phi(x)$  in terms of  $\Phi_0$ ,  $\alpha$ , and  $x$ . (c) How thick should the solar cell be made in order to absorb at least 38% of the incident light? Express your answer in terms of  $\alpha$ . (d) What happens to the light which enters the solar cell at  $x = 0$  but is not absorbed?
14. For the circuit of Fig. 8.5, compute the time constant if the  $10 \Omega$  resistor is replaced with (a) a short circuit; (b) a  $1 \Omega$  resistor; (c) a series connection of two  $5 \Omega$  resistors; (d) a  $100 \Omega$  resistor. (e) Verify your answers with a suitable parameter sweep simulation. (Hint: the cursor tool might come in handy, and the answer does not depend on the initial current you choose for the inductor.)
15. Design a circuit which will produce a voltage of 1 V at some initial time, and a voltage of 368 mV at a time 5 seconds later. You may specify an initial inductor current without showing how it arises.



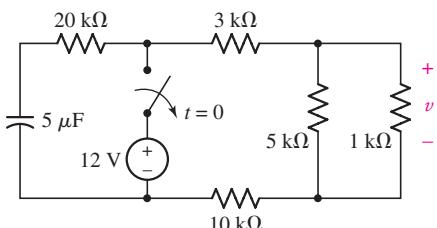
### 8.3 The Source-Free RC Circuit

16. The resistor in the circuit of Fig. 8.57 has been included to model the dielectric layer separating the plates of the  $3.1 \text{ nF}$  capacitor, and has a value of  $55 \text{ M}\Omega$ . The capacitor is storing  $200 \text{ mJ}$  of energy just prior to  $t = 0$ . (a) Write an expression for  $v(t)$  valid for  $t \geq 0$ . (b) Compute the energy remaining in the capacitor at  $t = 170 \text{ ms}$ . (c) Graph  $v(t)$  over the range of  $0 < t < 850 \text{ ms}$ , and identify the value of  $v(t)$  when  $t = 2\tau$ .
17. The resistor in the circuit of Fig. 8.57 has a value of  $1 \Omega$  and is connected to a  $22 \text{ mF}$  capacitor. The capacitor dielectric has infinite resistance, and the device is storing  $891 \text{ mJ}$  of energy just prior to  $t = 0$ . (a) Write an expression for  $v(t)$  valid for  $t \geq 0$ . (b) Compute the energy remaining in the capacitor at  $t = 11 \text{ ms}$  and  $33 \text{ ms}$ . (c) If it is determined that the capacitor dielectric is much leakier than expected, having a resistance as low as  $100 \text{ k}\Omega$ , repeat parts (a) and (b).
-  18. Calculate the time constant of the circuit depicted in Fig. 8.57 if  $C = 10 \text{ mF}$  and  $R$  is equal to (a)  $1 \Omega$ ; (b)  $10 \Omega$ ; (c)  $100 \Omega$ . (d) Verify your answers with an appropriate parameter sweep simulation. (Hint: the cursor tool might come in handy, and the time constant does not depend on the initial voltage across the capacitor.)
-  19. Design a capacitor-based circuit that will provide (a) a voltage of  $9 \text{ V}$  at some time  $t = 0$ , and a voltage of  $1.2 \text{ V}$  at a time  $4 \text{ ms}$  later; (b) a current of  $1 \text{ mA}$  at some time  $t = 0$ , and a reduced current of  $50 \mu\text{A}$  at a time  $100 \text{ ns}$  later. (You can choose to design two separate circuits if desired, and do not need to show how the initial capacitor voltage is set.)
20. It is safe to assume that the switch drawn in the circuit of Fig. 8.58 has been closed such a long time that any transients which might have arisen from first connecting the voltage source have disappeared. (a) Determine the circuit time constant. (b) Calculate the voltage  $v(t)$  at  $t = \tau$ ,  $2\tau$ , and  $5\tau$ .

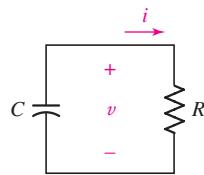


■ FIGURE 8.58

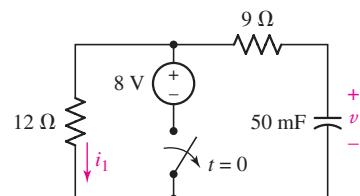
21. We can safely assume the switch in the circuit of Fig. 8.59 was closed a very long time prior to being thrown open at  $t = 0$ . (a) Determine the circuit time constant. (b) Obtain an expression for  $i_1(t)$  which is valid for  $t > 0$ . (c) Determine the power dissipated by the  $12 \Omega$  resistor at  $t = 500 \text{ ms}$ .
22. The switch above the  $12 \text{ V}$  source in the circuit of Fig. 8.60 has been closed since just after the wheel was invented. It is finally thrown open at  $t = 0$ . (a) Compute the circuit time constant. (b) Obtain an expression for  $v(t)$  valid for  $t > 0$ . (c) Calculate the energy stored in the capacitor  $170 \text{ ms}$  after the switch is opened.



■ FIGURE 8.60

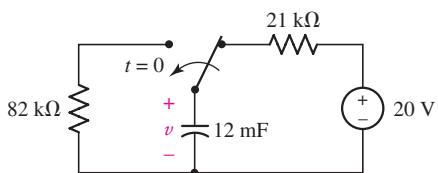


■ FIGURE 8.57



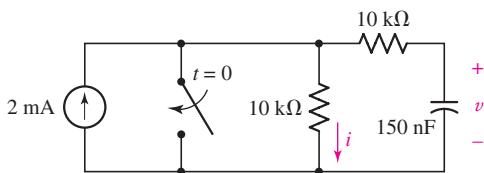
■ FIGURE 8.59

23. For the circuit represented schematically in Fig. 8.61, (a) calculate  $v(t)$  at  $t = 0$ ,  $t = 984$  s, and  $t = 1236$  s; (b) determine the energy still stored in the capacitor at  $t = 100$  s.



■ FIGURE 8.61

24. For the circuit depicted in Fig. 8.62, (a) compute the circuit time constant; (b) determine  $v$  in the instant just before the switch is closed; (c) obtain an expression for  $v(t)$  valid for  $t > 0$ ; (d) calculate  $v(3 \text{ ms})$ .

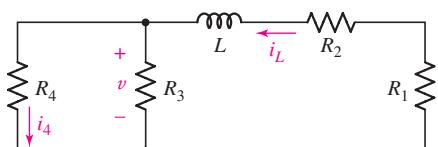


■ FIGURE 8.62

25. The switch drawn in Fig. 8.62 has been open a ponderously long time. (a) Determine the value of the current labeled  $i$  just prior to the switch being closed. (b) Obtain the value of  $i$  just after the switch is closed. (c) Compute the power dissipated in each resistor over the range of  $0 < t < 15 \text{ ms}$ . (d) Graph your answer to part (c).

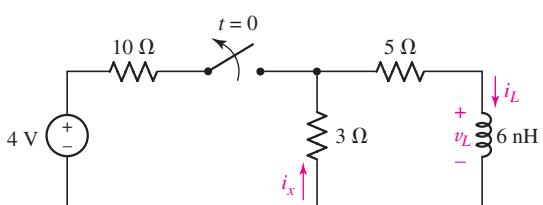
#### 8.4 A More General Perspective

26. (a) Obtain an expression for  $v(t)$ , the voltage which appears across resistor  $R_3$  in the circuit of Fig. 8.63, which is valid for  $t > 0$ . (b) If  $R_1 = 2R_2 = 3R_3 = 4R_4 = 1.2 \text{ k}\Omega$ ,  $L = 1 \text{ mH}$ , and  $i_L(0^-) = 3 \text{ mA}$ , calculate  $v(t = 500 \text{ ns})$ .



■ FIGURE 8.63

27. For the circuit of Fig. 8.64, determine  $i_x$ ,  $i_L$ , and  $v_L$  at  $t$  equal to (a)  $0^-$ ; (b)  $0^+$ .



■ FIGURE 8.64

28. The switch shown in Fig. 8.65 has been closed for 6 years prior to being flipped open at  $t = 0$ . Determine  $i_L$ ,  $v_L$ , and  $v_R$  at  $t$  equal to (a)  $0^-$ ; (b)  $0^+$ ; (c)  $1 \mu\text{s}$ ; (d)  $10 \mu\text{s}$ .

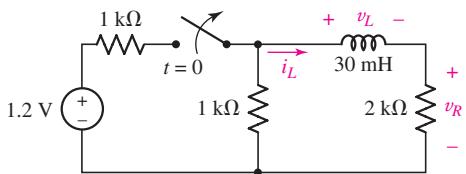


FIGURE 8.65

29. Obtain expressions for both  $i_1(t)$  and  $i_L(t)$  as labeled in Fig. 8.66, which are valid for  $t > 0$ .

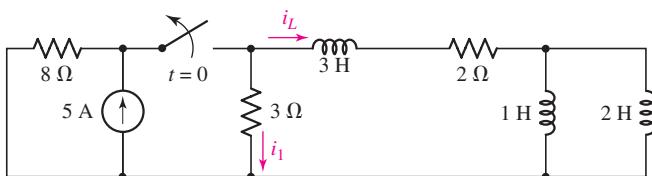


FIGURE 8.66

30. The voltage across the resistor in a simple source-free  $RL$  circuit is given by  $5e^{-90t} \text{ V}$ ,  $t > 0$ . The inductor value is not known. (a) At what time will the inductor voltage be exactly one-half of its maximum value? (b) At what time will the inductor current reach 10% of its maximum value?

31. Referring to Fig. 8.67, calculate the currents  $i_1$  and  $i_2$  at  $t$  equal to (a) 1 ms; (b) 3 ms.

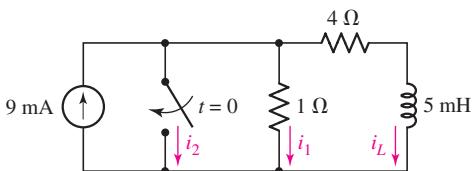


FIGURE 8.67

32. (a) Obtain an expression for  $v_x$  as labeled in the circuit of Fig. 8.68. (b) Evaluate  $v_x$  at  $t = 5 \text{ ms}$ . (c) Verify your answer with an appropriate PSpice simulation. (Hint: employ the part named `Sw_tClose`.)

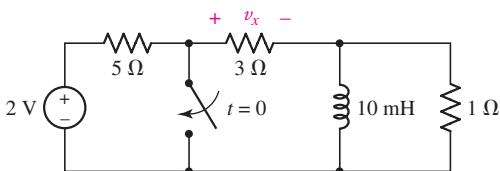


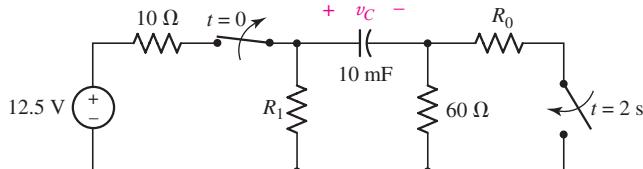
FIGURE 8.68

33. Design a complete circuit which provides a voltage  $v_{ab}$  across two terminals labeled  $a$  and  $b$ , respectively, such that  $v_{ab} = 5 \text{ V}$  at  $t = 0^-$ ,  $2 \text{ V}$  at  $t = 1 \text{ s}$ , and less than  $60 \text{ mV}$  at  $t = 5$ . Verify the operation of your circuit using an appropriate PSpice simulation. (Hint: employ the part named `Sw_tOpen` or `Sw_tClose` as appropriate.)

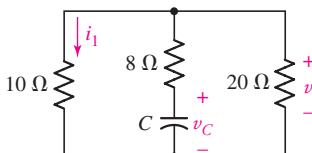


34. For the part **Sw\_tOpen**, PSpice actually employs a sequence of simulations where the part is first replaced with a resistor having value  $1 \text{ M}\Omega$ , and then replaced with a resistor having value  $10 \text{ m}\Omega$  corresponding to when the switch opens. (a) Evaluate the reliability of these default values by simulating the circuit of Fig. 8.55, and evaluating  $i_L$  at  $t = 1 \text{ ns}$ . (b) Repeat part (a) with **RCLOSED** changed to  $1 \Omega$ . Did this change your answer? (c) Repeat part (a) with **ROPEN** changed to  $100 \text{ k}\Omega$  and **RCLOSED** reset to its default value. Did this change your answer? (Hint: double-click on the part to access its attributes.)

35. Select values for the resistors  $R_0$  and  $R_1$  in the circuit of Fig. 8.69 such that  $v_C(0.65) = 5.22 \text{ V}$  and  $v_C(2.21) = 1 \text{ V}$ .



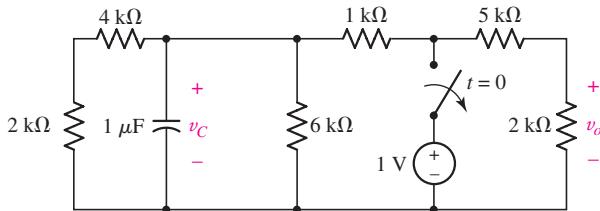
■ FIGURE 8.69



■ FIGURE 8.70

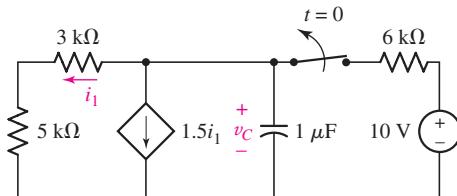
36. A quick measurement determines that the capacitor voltage  $v_C$  in the circuit of Fig. 8.70 is  $2.5 \text{ V}$  at  $t = 0^-$ . (a) Determine  $v_C(0^+)$ ,  $i_1(0^+)$ , and  $v(0^+)$ . (b) Select a value of  $C$  so that the circuit time constant is equal to  $14 \text{ s}$ .

37. Determine  $v_C(t)$  and  $v_o(t)$  as labeled in the circuit represented by Fig. 8.71 for  $t$  equal to (a)  $0^-$ ; (b)  $0^+$ ; (c)  $10 \text{ ms}$ ; (d)  $12 \text{ ms}$ .

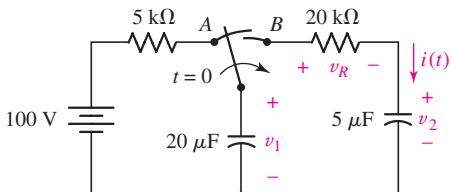


■ FIGURE 8.71

38. For the circuit shown in Fig. 8.72, determine (a)  $v_C(0^-)$ ; (b)  $v_C(0^+)$ ; (c) the circuit time constant; (d)  $v_C(3 \text{ ms})$ .



■ FIGURE 8.72



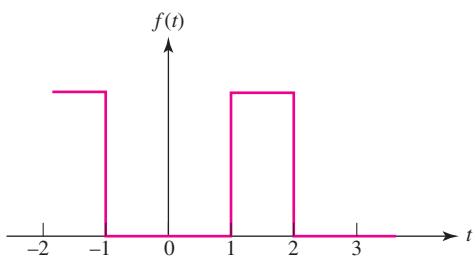
■ FIGURE 8.73

39. The switch in Fig. 8.73 is moved from  $A$  to  $B$  at  $t = 0$  after being at  $A$  for a long time. This places the two capacitors in series, thus allowing equal and opposite dc voltages to be trapped on the capacitors. (a) Determine  $v_1(0^-)$ ,  $v_2(0^-)$ , and  $v_R(0^-)$ . (b) Find  $v_1(0^+)$ ,  $v_2(0^+)$ , and  $v_R(0^+)$ . (c) Determine the time constant of  $v_R(t)$ . (d) Find  $v_R(t)$ ,  $t > 0$ . (e) Find  $i(t)$ . (f) Find  $v_1(t)$  and  $v_2(t)$  from  $i(t)$  and the initial values. (g) Show that the stored energy at  $t = \infty$  plus the total energy dissipated in the  $20 \text{ k}\Omega$  resistor is equal to the energy stored in the capacitors at  $t = 0$ .

40. The inductor in Fig. 8.74 is storing 54 nJ at  $t = 0^-$ . Compute the energy remaining at  $t$  equal to (a)  $0^+$ ; (b) 1 ms; (c) 5 ms.

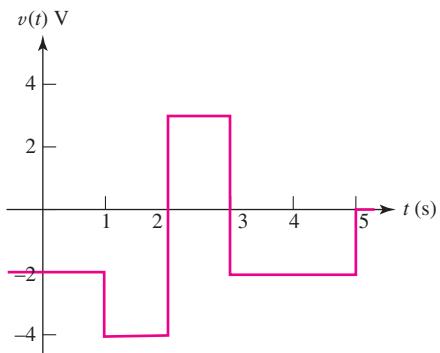
### 8.5 The Unit-Step Function

41. Evaluate the following functions at  $t = -2, 0$ , and  $+2$ : (a)  $f(t) = 3u(t)$ ; (b)  $g(t) = 5u(-t) + 3$ ; (c)  $h(t) = 5u(t - 3)$ ; (d)  $z(t) = 7u(1 - t) + 4u(t + 3)$ .
42. Evaluate the following functions at  $t = -1, 0$ , and  $+3$ : (a)  $f(t) = tu(1 - t)$ ; (b)  $g(t) = 8 + 2u(2 - t)$ ; (c)  $h(t) = u(t + 1) - u(t - 1) + u(t + 2) - u(t - 4)$ ; (d)  $z(t) = 1 + u(3 - t) + u(t - 2)$ .
43. Sketch the following functions over the range  $-3 \leq t \leq 3$ : (a)  $v(t) = 3 - u(2 - t) - 2u(t)$  V; (b)  $i(t) = u(t) - u(t - 0.5) + u(t - 1) - u(t - 1.5) + u(t - 2) - u(t - 2.5)$  A; (c)  $q(t) = 8u(-t)$  C.
44. Use step functions to construct an equation that describes the waveform sketched in Fig. 8.75.



■ FIGURE 8.75

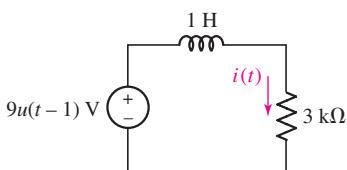
45. Employing step functions as appropriate, describe the voltage waveform graphed in Fig. 8.76.



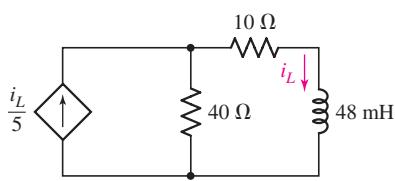
■ FIGURE 8.76

### 8.6 Driven RL Circuits

46. With reference to the simple circuit depicted in Fig. 8.77, compute  $i(t)$  for (a)  $t = 0^-$ ; (b)  $t = 0^+$ ; (c)  $t = 1^-$ ; (d)  $t = 1^+$ ; (e)  $t = 2$  ms.



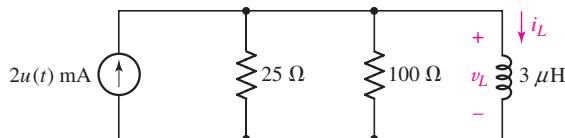
■ FIGURE 8.77



■ FIGURE 8.74

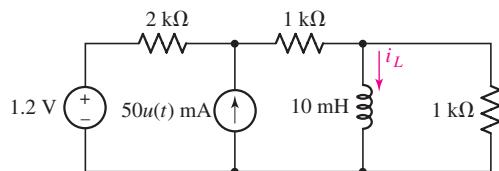


47. For the circuit given in Fig. 8.78, (a) determine  $v_L(0^-)$ ,  $v_L(0^+)$ ,  $i_L(0^-)$ , and  $i_L(0^+)$ ; (b) calculate  $i_L(150 \text{ ns})$ . (c) Verify your answer to part (b) with an appropriate PSpice simulation.

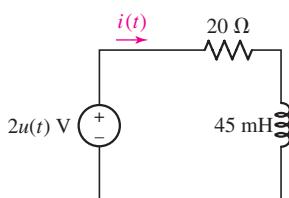


■ FIGURE 8.78

48. The circuit depicted in Fig. 8.79 contains two independent sources, one of which is only active for  $t > 0$ . (a) Obtain an expression for  $i_L(t)$  valid for all  $t$ ; (b) calculate  $i_L(t)$  at  $t = 10 \mu\text{s}$ ,  $20 \mu\text{s}$ , and  $50 \mu\text{s}$ .



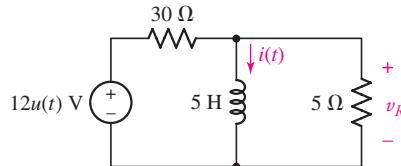
■ FIGURE 8.79



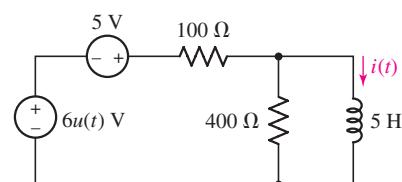
■ FIGURE 8.80

49. The circuit shown in Fig. 8.80 is powered by a source which is inactive for  $t < 0$ . (a) Obtain an expression for  $i(t)$  valid for all  $t$ . (b) Graph your answer over the range of  $-1 \text{ ms} \leq t \leq 10 \text{ ms}$ .

50. For the circuit shown in Fig. 8.81, (a) obtain an expression for  $i(t)$  valid for all time; (b) obtain an expression for  $v_R(t)$  valid for all time; and (c) graph both  $i(t)$  and  $v_R(t)$  over the range of  $-1 \text{ s} \leq t \leq 6 \text{ s}$ .



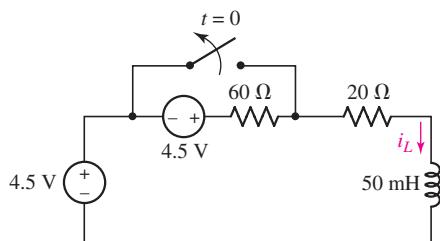
■ FIGURE 8.81



■ FIGURE 8.82

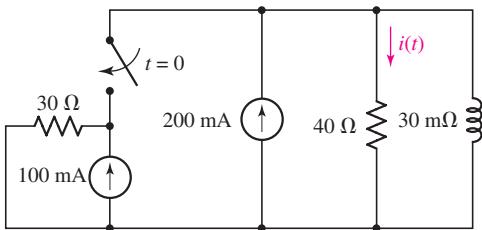
## 8.7 Natural and Forced Response

51. For the two-source circuit of Fig. 8.82, note that one source is always on. (a) Obtain an expression for  $i(t)$  valid for all  $t$ ; (b) determine at what time the energy stored in the inductor reaches 99% of its maximum value.
52. (a) Obtain an expression for  $i_L$  as labeled in Fig. 8.83 which is valid for all values of  $t$ . (b) Sketch your result over the range  $-1 \text{ ms} \leq t \leq 3 \text{ ms}$ .



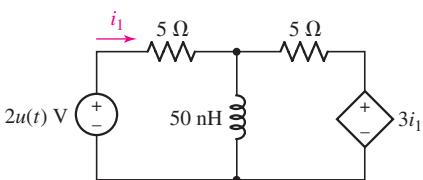
■ FIGURE 8.83

53. Obtain an expression for  $i(t)$  as labeled in the circuit diagram of Fig. 8.84, and determine the power being dissipated in the  $40\ \Omega$  resistor at  $t = 2.5\ \text{ms}$ .



■ FIGURE 8.84

54. Obtain an expression for  $i_1$  as indicated in Fig. 8.85 that is valid for all values of  $t$ .

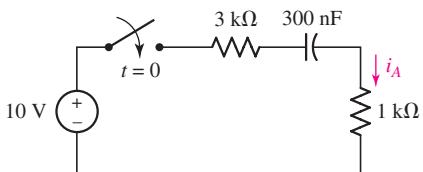


■ FIGURE 8.85

55. Plot the current  $i(t)$  in Fig. 8.86 if (a)  $R = 10\ \Omega$ ; (b)  $R = 1\ \Omega$ . In which case does the inductor (temporarily) store the most energy? Explain.

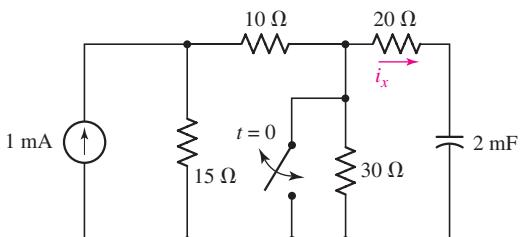
## 8.8 Driven RC Circuits

56. (a) Obtain an expression for  $v_C$  in the circuit of Fig. 8.87 valid for all values of  $t$ . (b) Sketch  $v_C(t)$  over the range  $0 \leq t \leq 4\ \mu\text{s}$ .
57. Obtain an equation which describes the behavior of  $i_A$  as labeled in Fig. 8.88 over the range of  $-1\ \text{ms} \leq t \leq 5\ \text{ms}$ .

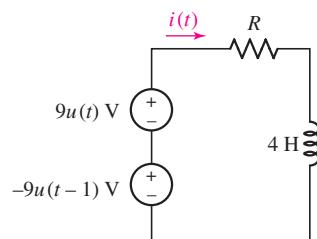


■ FIGURE 8.88

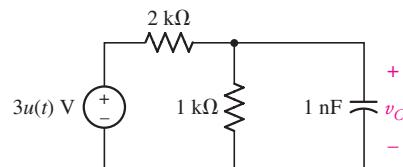
58. The switch in the circuit of Fig. 8.89 has been closed an incredibly long time, before being thrown open at  $t = 0$ . (a) Evaluate the current labeled  $i_x$  at  $t = 70\ \text{ms}$ . (b) Verify your answer with an appropriate PSpice simulation.



■ FIGURE 8.89



■ FIGURE 8.86



■ FIGURE 8.87



59. The switch in the circuit of Fig. 8.89 has been open a really, really incredibly long time, before being closed without further fanfare at  $t = 0$ . (a) Evaluate the current labeled  $i_x$  at  $t = 70$  ms. (b) Verify your answer with an appropriate PSpice simulation.
60. The “make-before-break” switch shown in Fig. 8.90 has been in position *a* since the first episode of “*Jonny Quest*” aired on television. It is moved to position *b*, finally, at time  $t = 0$ . (a) Obtain expressions for  $i(t)$  and  $v_C(t)$  valid for all values of  $t$ . (b) Determine the energy remaining in the capacitor at  $t = 33 \mu\text{s}$ .

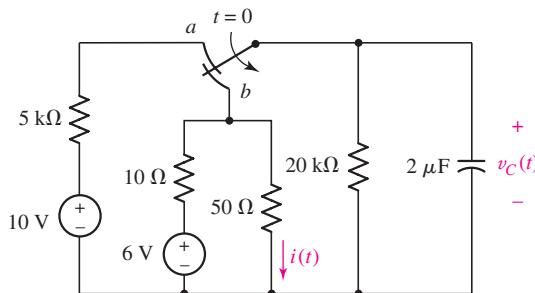


FIGURE 8.90

61. The switch in the circuit of Fig. 8.91, often called a *make-before-break* switch (since during switching it briefly makes contact to both parts of the circuit to ensure a smooth electrical transition), moves to position *b* at  $t = 0$  only after being in position *a* long enough to ensure all initial transients arising from turning on the sources have long since decayed. (a) Determine the power dissipated by the  $5 \Omega$  resistor at  $t = 0^-$ . (b) Determine the power dissipated in the  $3 \Omega$  resistor at  $t = 2 \text{ ms}$ .

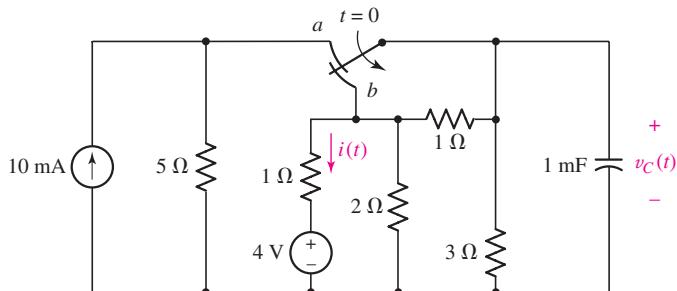


FIGURE 8.91

62. Referring to the circuit represented in Fig. 8.92, (a) obtain an equation which describes  $v_C$  valid for all values of  $t$ ; (b) determine the energy remaining in the capacitor at  $t = 0^+$ ,  $t = 25 \mu\text{s}$ , and  $t = 150 \mu\text{s}$ .

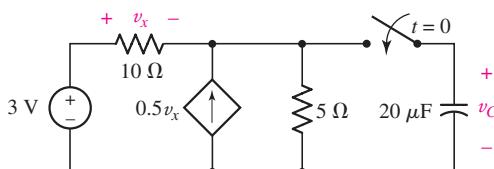
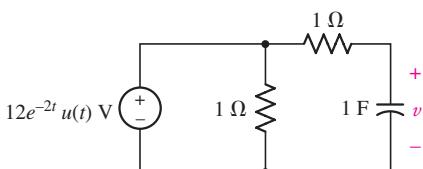


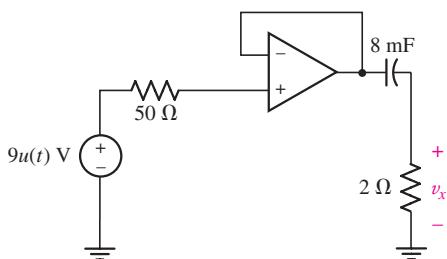
FIGURE 8.92

63. The dependent source shown in Fig. 8.92 is unfortunately installed upside down during manufacturing, so that the terminal corresponding to the arrowhead is actually wired to the negative reference terminal of the voltage source. This is not detected by the quality assurance team so the unit ships out wired improperly. The capacitor is initially discharged. If the  $5\ \Omega$  resistor is only rated to 2 W, after what time  $t$  is the circuit likely to fail?
64. For the circuit represented in Fig. 8.93, (a) obtain an expression for  $v$  which is valid for all values of  $t$ ; (b) sketch your result for  $0 \leq t \leq 3$  s.



■ FIGURE 8.93

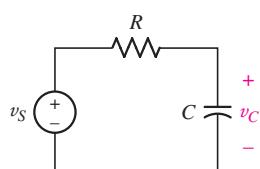
65. Obtain an expression for the voltage  $v_x$  as labeled in the op amp circuit of Fig. 8.94.



■ FIGURE 8.94

## 8.9 Predicting the Response of Sequentially Switched Circuits

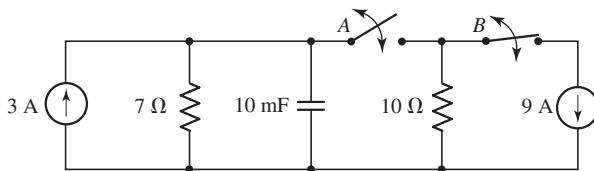
66. Sketch the current  $i_L$  of the circuit in Fig. 8.50a if the 100 mH inductor is replaced by a 1 nH inductor, and is subjected to the waveform  $v_s(t)$  equal to  
 (a)  $5u(t) - 5u(t - 10^{-9}) + 5u(t - 2 \times 10^{-9})$  V,  $0 \leq t \leq 4$  ns;  
 (b)  $9u(t) - 5u(t - 10^{-8}) + 5u(t - 2 \times 10^{-8})$  V,  $0 \leq t \leq 40$  ns.
67. The 100 mH inductor in the circuit of Fig. 8.50a is replaced with a 1 H inductor. Sketch the inductor current  $i_L$  if the source  $v_s(t)$  is equal to  
 (a)  $5u(t) - 5u(t - 0.01) + 5u(t - 0.02)$  V,  $0 \leq t \leq 40$  ms;  
 (b)  $5u(t) - 5u(t - 10) + 5u(t - 10.1)$  V,  $0 \leq t \leq 11$  s.
68. Sketch the voltage  $v_C$  across the capacitor of Fig. 8.95 for at least 3 periods if  $R = 1\ \Omega$ ,  $C = 1\text{ F}$ , and  $v_s(t)$  is a pulsed waveform having (a) minimum of 0 V, maximum of 2 V, rise and fall times of 1 ms, pulse width of 10 s, and period of 10 s; (b) minimum of 0 V, maximum of 2 V, rise and fall times of 1 ms, pulse width of 10 ms, and period of 10 ms. (c) Verify your answers with appropriate PSpice simulations.
69. Sketch the voltage  $v_C$  across the capacitor of Fig. 8.95 for at least 3 periods if  $R = 1\ \Omega$ ,  $C = 1\text{ F}$ , and  $v_s(t)$  is a pulsed waveform having (a) minimum of 0 V, maximum of 2 V, rise and fall times of 1 ms, pulse width of 10 s, and period of 10 ms; (b) minimum of 0 V, maximum of 2 V, rise and fall times of 1 ms, pulse width of 10 ms, and period of 10 s. (c) Verify your answers with appropriate PSpice simulations.



■ FIGURE 8.95

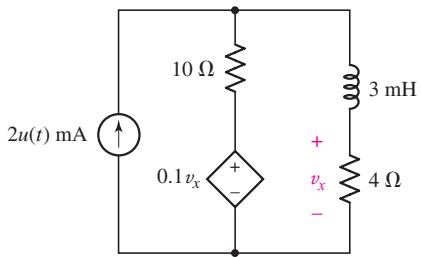
### Chapter-Integrating Exercises

70. The circuit in Fig. 8.96 contains two switches that always move in perfect synchronization. However, when switch A opens, switch B closes, and vice versa. Switch A is initially open, while switch B is initially closed; they change positions every 40 ms. Using the bottom node as the reference node, determine the voltage across the capacitor at  $t$  equal to (a)  $0^-$ ; (b)  $0^+$ ; (c)  $40^-$  ms; (d)  $40^+$  ms; (e) 50 ms.

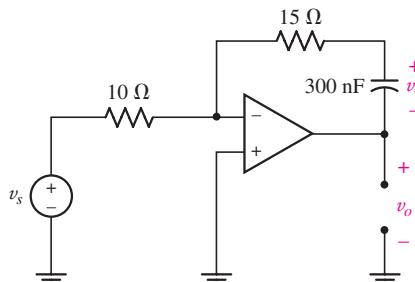


■ FIGURE 8.96

71. In the circuit of Fig. 8.96, when switch A opens, switch B closes, and vice versa. Switch A is initially open, while switch B is initially closed; they change positions every 400 ms. Determine the energy in the capacitor at  $t$  equal to (a)  $0^-$ ; (b)  $0^+$ ; (c) 200 ms; (d)  $400^-$  ms; (e)  $400^+$  ms; (f) 700 ms.
72. Refer to the circuit of Fig. 8.97, which contains a voltage-controlled dependent voltage source in addition to two resistors. (a) Compute the circuit time constant. (b) Obtain an expression for  $v_x$  valid for all  $t$ . (c) Plot the power dissipated in the resistor over the range of 6 time constants. (d) Repeat parts (a) to (c) if the dependent source is installed in the circuit upside down. (e) Are both circuit configurations “stable”? Explain.
73. In the circuit of Fig. 8.97, a 3 mH inductor is accidentally installed instead of the capacitor. Unfortunately, that’s not the end of the problems, as it’s later determined that the real capacitor is not really well modeled by an ideal capacitor, and the dielectric has a resistance of  $10 \text{ k}\Omega$  (which should be viewed as connected in parallel to the ideal capacitor). (a) Compute the circuit time constant with and without taking the dielectric resistance into account. By how much does the dielectric change your answer? (b) Calculate  $v_x$  at  $t = 200$  ms. Does the dielectric resistance affect your answer significantly? Explain.
74. For the circuit of Fig. 8.98, assuming an ideal op amp, derive an expression for  $v_o(t)$  if  $v_s$  is equal to (a)  $4u(t)$  V; (b)  $4e^{-130,000t}u(t)$  V.



■ FIGURE 8.97



■ FIGURE 8.98

# The *RLC* Circuit

## INTRODUCTION

In Chap. 8 we studied circuits which contained only **one** energy storage element, combined with a passive network which partly determined how long it took either the capacitor or the inductor to charge/discharge. The differential equations which resulted from analysis were always first-order. In this chapter, we consider more complex circuits which contain **both** an inductor **and** a capacitor. The result is a **second-order** differential equation for any voltage or current of interest. What we learned in Chap. 8 is easily extended to the study of these so-called *RLC* circuits, although now we need **two** initial conditions to solve each differential equation. Such circuits occur routinely in a wide variety of applications, including oscillators and frequency filters. They are also very useful in modeling a number of practical situations, such as automobile suspension systems, temperature controllers, and even the response of an airplane to changes in elevator and aileron positions.

### 9.1 THE SOURCE-FREE PARALLEL CIRCUIT

There are two basic types of *RLC* circuits: *parallel connected*, and *series connected*. We could start with either, but somewhat arbitrarily choose to begin by considering parallel *RLC* circuits. This particular combination of ideal elements is a reasonable model for portions of many communication networks. It represents, for example, an important part of certain electronic amplifiers found in radios, and enables the amplifiers to produce a large voltage amplification over a narrow band of signal frequencies (with almost zero amplification outside this band).

Just as we did with *RL* and *RC* circuits, we first consider the natural response of a parallel *RLC* circuit, where one or both of the

## KEY CONCEPTS

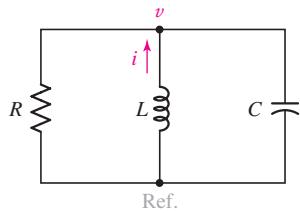
- Resonant Frequency and Damping Factor of Series and Parallel *RLC* Circuits
- Overdamped Response
- Critically Damped Response
- Underdamped Response
- Making Use of Two Initial Conditions
- Complete (Natural + Forced) Response of *RLC* Circuits
- Representing Differential Equations Using Op Amp Circuits



energy storage elements have some nonzero initial energy (the origin of which for now is unimportant). This is represented by the inductor current and the capacitor voltage, both specified at  $t = 0^+$ . Once we're comfortable with this part of *RLC* circuit analysis, we can easily include dc sources, switches, or step sources in the circuit. Then we find the total response, which will be the sum of the natural response and the forced response.

Frequency selectivity of this kind enables us to listen to the transmission of one station while rejecting the transmission of any other station. Other applications include the use of parallel *RLC* circuits in frequency multiplexing and harmonic-suppression filters. However, even a simple discussion of these principles requires an understanding of such terms as *resonance*, *frequency response*, and *impedance*, which we have not yet discussed. Let it suffice to say, therefore, that an understanding of the natural behavior of the parallel *RLC* circuit is fundamentally important to future studies of communications networks and filter design, as well as many other applications.

When a *physical* capacitor is connected in parallel with an inductor and the capacitor has associated with it a finite resistance, the resulting network can be shown to have an equivalent circuit model like that shown in Fig. 9.1. The presence of this resistance can be used to model energy loss in the capacitor; over time, all real capacitors will eventually discharge, even if disconnected from a circuit. Energy losses in the physical inductor can also be taken into account by adding an ideal resistor (in series with the ideal inductor). For simplicity, however, we restrict our discussion to the case of an essentially ideal inductor in parallel with a "leaky" capacitor.



■ FIGURE 9.1 The source-free parallel *RLC* circuit.

## Obtaining the Differential Equation for a Parallel *RLC* Circuit

In the following analysis we will assume that energy may be stored initially in both the inductor and the capacitor; in other words, nonzero initial values of both inductor current and capacitor voltage may be present. With reference to the circuit of Fig. 9.1, we may then write the single nodal equation

$$\frac{v}{R} + \frac{1}{L} \int_{t_0}^t v dt' - i(t_0) + C \frac{dv}{dt} = 0 \quad [1]$$

Note that the minus sign is a consequence of the assumed direction for  $i$ . We must solve Eq. [1] subject to the initial conditions

$$i(0^+) = I_0 \quad [2]$$

and

$$v(0^+) = V_0 \quad [3]$$

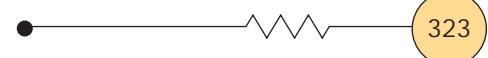
When both sides of Eq. [1] are differentiated once with respect to time, the result is the linear second-order homogeneous differential equation

$$C \frac{d^2v}{dt^2} + \frac{1}{R} \frac{dv}{dt} + \frac{1}{L} v = 0 \quad [4]$$

whose solution  $v(t)$  is the desired natural response.

## Solution of the Differential Equation

There are a number of interesting ways to solve Eq. [4]. Most of these methods we will leave to a course in differential equations, selecting only the quickest and simplest method to use now. We will assume a solution, relying upon our



intuition and modest experience to select one of the several possible forms that are suitable. Our experience with first-order equations might suggest that we at least try the exponential form once more. Thus, we *assume*

$$v = Ae^{st} \quad [5]$$

being as general as possible by allowing  $A$  and  $s$  to be complex numbers if necessary. Substituting Eq. [5] in Eq. [4], we obtain

$$CAs^2e^{st} + \frac{1}{R}Ase^{st} + \frac{1}{L}Ae^{st} = 0$$

or

$$Ae^{st} \left( Cs^2 + \frac{1}{R}s + \frac{1}{L} \right) = 0$$

In order for this equation to be satisfied for all time, at least one of the three factors must be zero. If either of the first two factors is set equal to zero, then  $v(t) = 0$ . This is a trivial solution of the differential equation which cannot satisfy our given initial conditions. We therefore equate the remaining factor to zero:

$$Cs^2 + \frac{1}{R}s + \frac{1}{L} = 0 \quad [6]$$

This equation is usually called the *auxiliary equation* or the *characteristic equation*, as we discussed in Sec. 8.1. If it can be satisfied, then our assumed solution is correct. Since Eq. [6] is a quadratic equation, there are two solutions, identified as  $s_1$  and  $s_2$ :

$$s_1 = -\frac{1}{2RC} + \sqrt{\left(\frac{1}{2RC}\right)^2 - \frac{1}{LC}} \quad [7]$$

and

$$s_2 = -\frac{1}{2RC} - \sqrt{\left(\frac{1}{2RC}\right)^2 - \frac{1}{LC}} \quad [8]$$

If *either* of these two values is used for  $s$  in the assumed solution, then that solution satisfies the given differential equation; it thus becomes a valid solution of the differential equation.

Let us assume that we replace  $s$  by  $s_1$  in Eq. [5], obtaining

$$v_1 = A_1 e^{s_1 t}$$

and, similarly,

$$v_2 = A_2 e^{s_2 t}$$

The former satisfies the differential equation

$$C \frac{d^2 v_1}{dt^2} + \frac{1}{R} \frac{dv_1}{dt} + \frac{1}{L} v_1 = 0$$

and the latter satisfies

$$C \frac{d^2 v_2}{dt^2} + \frac{1}{R} \frac{dv_2}{dt} + \frac{1}{L} v_2 = 0$$

Adding these two differential equations and combining similar terms, we have

$$C \frac{d^2(v_1 + v_2)}{dt^2} + \frac{1}{R} \frac{d(v_1 + v_2)}{dt} + \frac{1}{L}(v_1 + v_2) = 0$$

Linearity triumphs, and it is seen that the *sum* of the two solutions is also a solution. We thus have the general form of the natural response

$$v(t) = A_1 e^{s_1 t} + A_2 e^{s_2 t} \quad [9]$$

where  $s_1$  and  $s_2$  are given by Eqs. [7] and [8];  $A_1$  and  $A_2$  are two arbitrary constants which are to be selected to satisfy the two specified initial conditions.

## Definition of Frequency Terms

The form of the natural response as given in Eq. [9] offers little insight into the nature of the curve we might obtain if  $v(t)$  were plotted as a function of time. The relative amplitudes of  $A_1$  and  $A_2$ , for example, will certainly be important in determining the shape of the response curve. Furthermore, the constants  $s_1$  and  $s_2$  can be real numbers or conjugate complex numbers, depending upon the values of  $R$ ,  $L$ , and  $C$  in the given network. These two cases will produce fundamentally different response forms. Therefore, it will be helpful to make some simplifying substitutions in Eq. [9].

Since the exponents  $s_1 t$  and  $s_2 t$  must be dimensionless,  $s_1$  and  $s_2$  must have the unit of some dimensionless quantity “per second.” From Eqs. [7] and [8] we therefore see that the units of  $1/2RC$  and  $1/\sqrt{LC}$  must also be  $s^{-1}$  (i.e.,  $\text{seconds}^{-1}$ ). Units of this type are called **frequencies**.

Let us define a new term,  $\omega_0$  (omega-sub-zero, or just omega-zero):

$$\omega_0 = \frac{1}{\sqrt{LC}} \quad [10]$$

and reserve the term **resonant frequency** for it. On the other hand, we will call  $1/2RC$  the **neper frequency**, or the **exponential damping coefficient**, and represent it by the symbol  $\alpha$  (alpha):

$$\alpha = \frac{1}{2RC} \quad [11]$$

This latter descriptive expression is used because  $\alpha$  is a measure of how rapidly the natural response decays or damps out to its steady, final value (usually zero). Finally,  $s$ ,  $s_1$ , and  $s_2$ , which are quantities that will form the basis for some of our later work, are called **complex frequencies**.

We should note that  $s_1$ ,  $s_2$ ,  $\alpha$ , and  $\omega_0$  are merely symbols used to simplify the discussion of *RLC* circuits; they are not mysterious new properties of any kind. It is easier, for example, to say “*alpha*” than it is to say “*the reciprocal of 2RC*.”

Let us collect these results. The natural response of the parallel *RLC* circuit is

$$v(t) = A_1 e^{s_1 t} + A_2 e^{s_2 t} \quad [9]$$

where

$$s_1 = -\alpha + \sqrt{\alpha^2 - \omega_0^2} \quad [12]$$

$$s_2 = -\alpha - \sqrt{\alpha^2 - \omega_0^2} \quad [13]$$

$$\alpha = \frac{1}{2RC} \quad [11]$$

$$\omega_0 = \frac{1}{\sqrt{LC}} \quad [10]$$

and  $A_1$  and  $A_2$  must be found by applying the given initial conditions.

We note two basic scenarios possible with Eqs. [12] and [13] depending on the relative sizes of  $\alpha$  and  $\omega_0$  (dictated by the values of  $R$ ,  $L$ , and  $C$ ). If  $\alpha > \omega_0$ ,  $s_1$  and  $s_2$  will both be real numbers, leading to what is referred to as an **overdamped response**. In the opposite case, where  $\alpha < \omega_0$ , both  $s_1$  and  $s_2$  will have nonzero imaginary components, leading to what is known as an **underdamped response**. Both of these situations are considered separately in the following sections, along with the special case of  $\alpha = \omega_0$ , which leads to what is called a **critically damped response**. We should also note that the general response comprised by Eqs. [9] through [13] describes not only the voltage but all three branch currents in the parallel RLC circuit; the constants  $A_1$  and  $A_2$  will be different for each, of course.

The ratio of  $\alpha$  to  $\omega_0$  is called the *damping ratio* by control system engineers and is designated by  $\zeta$  (zeta).

Overdamped:  $\alpha > \omega_0$

Critically damped:  $\alpha = \omega_0$

Underdamped:  $\alpha < \omega_0$



## EXAMPLE 9.1

Consider a parallel RLC circuit having an inductance of 10 mH and a capacitance of 100  $\mu$ F. Determine the resistor values that would lead to overdamped and underdamped responses.

We first calculate the resonant frequency of the circuit:

$$\omega_0 = \sqrt{\frac{1}{LC}} = \sqrt{\frac{1}{(10 \times 10^{-3})(100 \times 10^{-6})}} = 10^3 \text{ rad/s}$$

An *overdamped* response will result if  $\alpha > \omega_0$ ; an *underdamped* response will result if  $\alpha < \omega_0$ . Thus,

$$\frac{1}{2RC} > 10^3$$

and so

$$R < \frac{1}{(2000)(100 \times 10^{-6})}$$

or

$$R < 5 \Omega$$

leads to an overdamped response;  $R > 5 \Omega$  leads to an underdamped response.

### PRACTICE

- 9.1 A parallel RLC circuit contains a 100  $\Omega$  resistor and has the parameter values  $\alpha = 1000 \text{ s}^{-1}$  and  $\omega_0 = 800 \text{ rad/s}$ . Find (a)  $C$ ; (b)  $L$ ; (c)  $s_1$ ; (d)  $s_2$ .

Ans: 5  $\mu$ F; 312.5 mH;  $-400 \text{ s}^{-1}$ ;  $-1600 \text{ s}^{-1}$ .

## 9.2 THE OVERDAMPED PARALLEL RLC CIRCUIT

A comparison of Eqs. [10] and [11] shows that  $\alpha$  will be greater than  $\omega_0$  if  $LC > 4R^2C^2$ . In this case the radical used in calculating  $s_1$  and  $s_2$  will be real, and both  $s_1$  and  $s_2$  will be real. Moreover, the following inequalities

$$\begin{aligned}\sqrt{\alpha^2 - \omega_0^2} &< \alpha \\ \left(-\alpha - \sqrt{\alpha^2 - \omega_0^2}\right) &< \left(-\alpha + \sqrt{\alpha^2 - \omega_0^2}\right) < 0\end{aligned}$$

may be applied to Eqs. [12] and [13] to show that both  $s_1$  and  $s_2$  are *negative* real numbers. Thus, the response  $v(t)$  can be expressed as the (algebraic) sum of two decreasing exponential terms, both of which approach zero as time increases. In fact, since the absolute value of  $s_2$  is larger than that of  $s_1$ , the term containing  $s_2$  has the more rapid rate of decrease, and, for large values of time, we may write the limiting expression

$$v(t) \rightarrow A_1 e^{s_1 t} \rightarrow 0 \quad \text{as } t \rightarrow \infty$$

The next step is to determine the arbitrary constants  $A_1$  and  $A_2$  in conformance with the initial conditions. We select a parallel *RLC* circuit with  $R = 6 \Omega$ ,  $L = 7 \text{ H}$ , and, for ease of computation,  $C = \frac{1}{42} \text{ F}$ . The initial energy storage is specified by choosing an initial voltage across the circuit  $v(0) = 0$  and an initial inductor current  $i(0) = 10 \text{ A}$ , where  $v$  and  $i$  are defined in Fig. 9.2.

We may easily determine the values of the several parameters

$$\begin{array}{lll}\alpha = 3.5 & \omega_0 = \sqrt{6} & (\text{all } s^{-1}) \\ s_1 = -1 & s_2 = -6 &\end{array}$$

and immediately write the general form of the natural response

$$v(t) = A_1 e^{-t} + A_2 e^{-6t} \quad [14]$$

### Finding Values for $A_1$ and $A_2$

Only the evaluation of the two constants  $A_1$  and  $A_2$  remains. If we knew the response  $v(t)$  at two different values of time, these two values could be substituted in Eq. [14] and  $A_1$  and  $A_2$  easily found. However, we know only one instantaneous value of  $v(t)$ ,

$$v(0) = 0$$

and, therefore,

$$0 = A_1 + A_2 \quad [15]$$

We can obtain a second equation relating  $A_1$  and  $A_2$  by taking the derivative of  $v(t)$  with respect to time in Eq. [14], determining the initial value of this derivative through the use of the remaining initial condition  $i(0) = 10$ , and equating the results. So, taking the derivative of both sides of Eq. [14],

$$\frac{dv}{dt} = -A_1 e^{-t} - 6A_2 e^{-6t}$$

and evaluating the derivative at  $t = 0$ ,

$$\left. \frac{dv}{dt} \right|_{t=0} = -A_1 - 6A_2$$



we obtain a second equation. Although this may appear to be helpful, we do not have a numerical value for the initial value of the derivative, so we do not yet have two equations in two unknowns . . . Or do we? The expression  $dv/dt$  suggests a capacitor current, since

$$i_C = C \frac{dv}{dt}$$

Kirchhoff's current law must hold at any instant in time, as it is based on conservation of electrons. Thus, we may write

$$-i_C(0) + i(0) + i_R(0) = 0$$

Substituting our expression for capacitor current and dividing by  $C$ ,

$$\left. \frac{dv}{dt} \right|_{t=0} = \frac{i_C(0)}{C} = \frac{i(0) + i_R(0)}{C} = \frac{i(0)}{C} = 420 \text{ V/s}$$

since zero initial voltage across the resistor requires zero initial current through it. We thus have our second equation,

$$420 = -A_1 - 6A_2 \quad [16]$$

and simultaneous solution of Eqs. [15] and [16] provides the two amplitudes  $A_1 = 84$  and  $A_2 = -84$ . Therefore, the final numerical solution for the natural response of this circuit is

$$v(t) = 84(e^{-t} - e^{-6t}) \quad \text{V} \quad [17]$$

For the remainder of our discussions concerning  $RLC$  circuits, we will always require two initial conditions in order to completely specify the response. One condition will usually be very easy to apply—either a voltage or current at  $t = 0$ . It is the second condition that usually requires a little effort. Although we will often have both an initial current and an initial voltage at our disposal, one of these will need to be applied indirectly through the derivative of our assumed solution.

## EXAMPLE 9.2

Find an expression for  $v_C(t)$  valid for  $t > 0$  in the circuit of Fig. 9.3a.

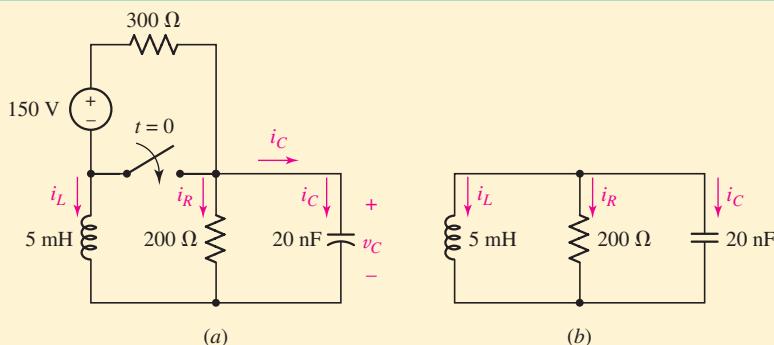


FIGURE 9.3 (a) An  $RLC$  circuit that becomes source-free at  $t = 0$ . (b) The circuit for  $t > 0$ , in which the 150 V source and the 300  $\Omega$  resistor have been shorted out by the switch, and so are of no further relevance to  $v_C$ .

### ► Identify the goal of the problem.

We are asked to find the capacitor voltage after the switch is thrown. This action leads to no sources remaining connected to either the inductor or the capacitor.

(Continued on next page)

► **Collect the known information.**

After the switch is thrown, the capacitor is left in parallel with a  $200\ \Omega$  resistor and a  $5\text{ mH}$  inductor (Fig. 9.3b). Thus,  $\alpha = 1/2RC = 125,000\text{ s}^{-1}$ ,  $\omega_0 = 1/\sqrt{LC} = 100,000\text{ rad/s}$ ,  $s_1 = -\alpha + \sqrt{\alpha^2 - \omega_0^2} = -50,000\text{ s}^{-1}$  and  $s_2 = -\alpha - \sqrt{\alpha^2 - \omega_0^2} = -200,000\text{ s}^{-1}$ .

► **Devise a plan.**

Since  $\alpha > \omega_0$ , the circuit is overdamped and so we expect a capacitor voltage of the form

$$v_C(t) = A_1 e^{s_1 t} + A_2 e^{s_2 t}$$

We know  $s_1$  and  $s_2$ ; we need to obtain and invoke two initial conditions to determine  $A_1$  and  $A_2$ . To do this, we will analyze the circuit at  $t = 0^-$  (Fig. 9.4a) to find  $i_L(0^-)$  and  $v_C(0^-)$ . We will then analyze the circuit at  $t = 0^+$  with the assumption that neither value changes.

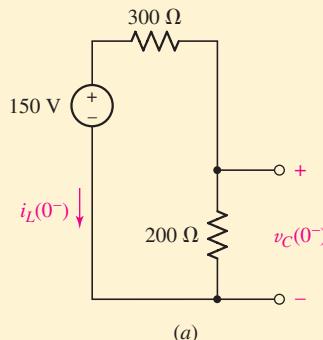
► **Construct an appropriate set of equations.**

From Fig. 9.4a, in which the inductor has been replaced with a short circuit and the capacitor with an open circuit, we see that

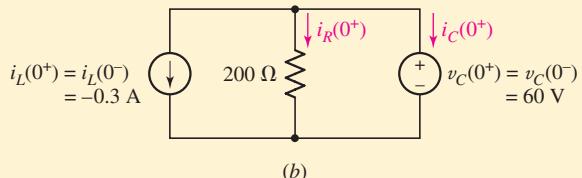
$$i_L(0^-) = -\frac{150}{200 + 300} = -300\text{ mA}$$

and

$$v_C(0^-) = 150 \frac{200}{200 + 300} = 60\text{ V}$$



(a)



(b)

■ FIGURE 9.4 (a) The equivalent circuit at  $t = 0^-$ ; (b) equivalent circuit at  $t = 0^+$ , drawn using ideal sources to represent the initial inductor current and initial capacitor voltage.

In Fig. 9.4b, we draw the circuit at  $t = 0^+$ , representing the inductor current and capacitor voltage by ideal sources for simplicity. Since neither can change in zero time, we know that  $v_C(0^+) = 60$  V.

► **Determine if additional information is required.**

We have an equation for the capacitor voltage:  $v_C(t) = A_1 e^{-50,000t} + A_2 e^{-200,000t}$ . We now know  $v_C(0) = 60$  V, but a third equation is still required. Differentiating our capacitor voltage equation, we find

$$\frac{dv_C}{dt} = -50,000A_1 e^{-50,000t} - 200,000A_2 e^{-200,000t}$$

which can be related to the capacitor current as  $i_C = C(dv_C/dt)$ .

Returning to Fig. 9.4b, KCL yields

$$i_C(0^+) = -i_L(0^+) - i_R(0^+) = 0.3 - [v_C(0^+)/200] = 0$$

► **Attempt a solution.**

Application of our first initial condition yields

$$v_C(0) = A_1 + A_2 = 60$$

and application of our second initial condition yields

$$i_C(0) = -20 \times 10^{-9}(50,000A_1 + 200,000A_2) = 0$$

Solving,  $A_1 = 80$  V and  $A_2 = -20$  V, so that

$$v_C(t) = 80e^{-50,000t} - 20e^{-200,000t} \text{ V}, \quad t > 0$$

► **Verify the solution. Is it reasonable or expected?**

At the very least, we can check our solution at  $t = 0$ , verifying that  $v_C(0) = 60$  V. Differentiating and multiplying by  $20 \times 10^{-9}$ , we can also verify that  $i_C(0) = 0$ . Also, since we have a source-free circuit for  $t > 0$ , we expect that  $v_C(t)$  must eventually decay to zero as  $t$  approaches  $\infty$ , which our solution does.

**PRACTICE**

- 9.2 After being open for a long time, the switch in Fig. 9.5 closes at  $t = 0$ . Find (a)  $i_L(0^-)$ ; (b)  $v_C(0^-)$ ; (c)  $i_R(0^+)$ ; (d)  $i_C(0^+)$ ; (e)  $v_C(0.2)$ .

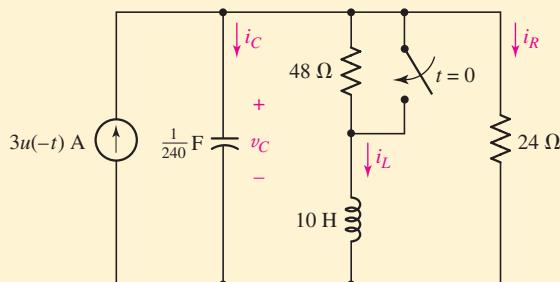


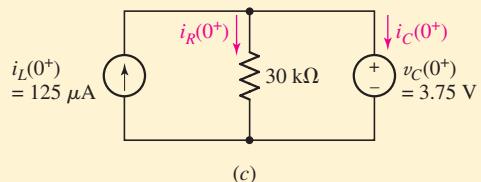
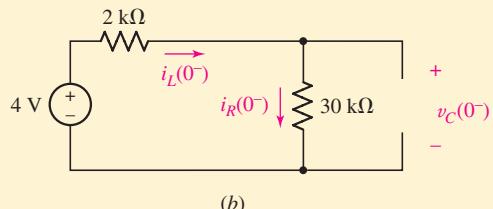
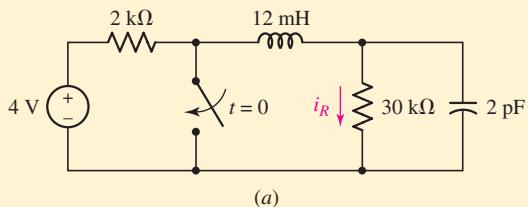
FIGURE 9.5

Ans: 1 A; 48 V; 2 A; -3 A; -17.54 V.

As noted previously, the form of the overdamped response applies to any voltage or current quantity, as we explore in the following example.

### EXAMPLE 9.3

The circuit of Fig. 9.6a reduces to a simple parallel RLC circuit after  $t = 0$ . Determine an expression for the resistor current  $i_R$  valid for all time.



■ FIGURE 9.6 (a) Circuit for which  $i_R$  is required. (b) Equivalent circuit for  $t = 0^-$ . (c) Equivalent circuit for  $t = 0^+$ .

For  $t > 0$ , we have a parallel RLC circuit with  $R = 30 \text{ k}\Omega$ ,  $L = 12 \text{ mH}$ , and  $C = 2 \text{ pF}$ . Thus,  $\alpha = 8.333 \times 10^6 \text{ s}^{-1}$  and  $\omega_0 = 6.455 \times 10^6 \text{ rad/s}$ . We therefore expect an overdamped response, with  $s_1 = -3.063 \times 10^6 \text{ s}^{-1}$  and  $s_2 = -13.60 \times 10^6 \text{ s}^{-1}$ , so that

$$i_R(t) = A_1 e^{s_1 t} + A_2 e^{s_2 t}, \quad t > 0 \quad [18]$$

To determine numerical values for  $A_1$  and  $A_2$ , we first analyze the circuit at  $t = 0^-$ , as drawn in Fig. 9.6b. We see that  $i_L(0^-) = i_R(0^-) = 4/32 \times 10^3 = 125 \mu\text{A}$ , and  $v_C(0^-) = 4 \times 30/32 = 3.75 \text{ V}$ .

In drawing the circuit at  $t = 0^+$  (Fig. 9.6c), we only know that  $i_L(0^+) = 125 \mu\text{A}$  and  $v_C(0^+) = 3.75 \text{ V}$ . However, by Ohm's law we can calculate that  $i_R(0^+) = 3.75/30 \times 10^3 = 125 \mu\text{A}$ , our first initial condition. Thus,

$$i_R(0) = A_1 + A_2 = 125 \times 10^{-6} \quad [19]$$

How do we obtain a *second* initial condition? If we multiply Eq. [18] by  $30 \times 10^3$ , we obtain an expression for  $v_C(t)$ . Taking the derivative and multiplying by 2 pF yield an expression for  $i_C(t)$ :

$$i_C = C \frac{dv_C}{dt} = (2 \times 10^{-12})(30 \times 10^3)(A_1 s_1 e^{s_1 t} + A_2 s_2 e^{s_2 t})$$

By KCL,

$$i_C(0^+) = i_L(0^+) - i_R(0^+) = 0$$

Thus,

$$-(2 \times 10^{-12})(30 \times 10^3)(3.063 \times 10^6 A_1 + 13.60 \times 10^6 A_2) = 0 \quad [20]$$

Solving Eqs. [19] and [20], we find that  $A_1 = 161.3 \mu\text{A}$  and  $A_2 = -36.34 \mu\text{A}$ . Thus,

$$i_R = \begin{cases} 125 \mu\text{A} & t < 0 \\ 161.3e^{-3.063 \times 10^6 t} - 36.34e^{-13.6 \times 10^6 t} \mu\text{A} & t > 0 \end{cases}$$

### PRACTICE

9.3 Determine the current  $i_R$  through the resistor of Fig. 9.7 for  $t > 0$  if  $i_L(0^-) = 6 \text{ A}$  and  $v_C(0^+) = 0 \text{ V}$ . The configuration of the circuit prior to  $t = 0$  is not known.

Ans:  $i_R(t) = 2.437(e^{-7.823 \times 10^{10} t} - e^{-0.511 \times 10^{10} t}) \text{ A}$ .

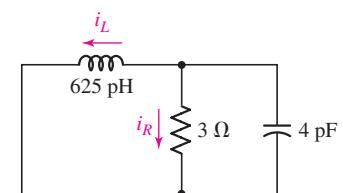


FIGURE 9.7 Circuit for Practice Problem 9.3.

## Graphical Representation of the Overdamped Response

Now let us return to Eq. [17] and see what additional information we can determine about this circuit. We may interpret the first exponential term as having a time constant of 1 s and the other exponential, a time constant of  $\frac{1}{6}$  s. Each starts with unity amplitude, but the latter decays more rapidly;  $v(t)$  is never negative. As time becomes infinite, each term approaches zero, and the response itself dies out as it should. We therefore have a response curve which is zero at  $t = 0$ , is zero at  $t = \infty$ , and is never negative; since it is not everywhere zero, it must possess at least one maximum, and this is not a difficult point to determine exactly. We differentiate the response

$$\frac{dv}{dt} = 84(-e^{-t} + 6e^{-6t})$$

set the derivative equal to zero to determine the time  $t_m$  at which the voltage becomes maximum,

$$0 = -e^{-t_m} + 6e^{-6t_m}$$

manipulate once,

$$e^{5t_m} = 6$$

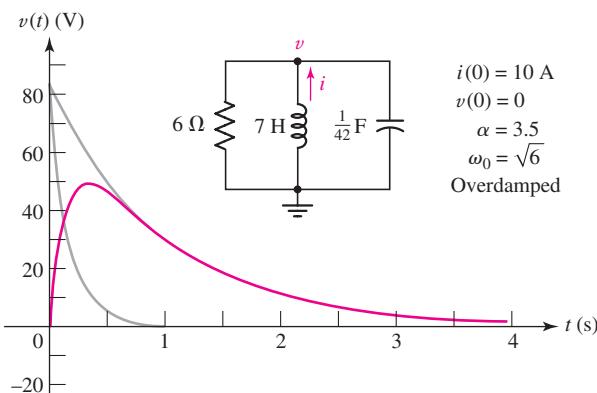
and obtain

$$t_m = 0.358 \text{ s}$$

and

$$v(t_m) = 48.9 \text{ V}$$

A reasonable sketch of the response may be made by plotting the two exponential terms  $84e^{-t}$  and  $84e^{-6t}$  and then taking their difference. This technique is illustrated by the curves of Fig. 9.8; the two exponentials are shown lightly, and their difference, the total response  $v(t)$ , is drawn as a colored line. The curves also verify our previous prediction that the functional behavior of  $v(t)$  for very large  $t$  is  $84e^{-t}$ , the exponential term containing the smaller magnitude of  $s_1$  and  $s_2$ .



■ FIGURE 9.8 The response  $v(t) = 84(e^{-t} - e^{-6t})$  of the network shown in Fig. 9.2.

A frequently asked question is the length of time it actually takes for the transient part of the response to disappear (or “*damp out*”). In practice, it is often desirable to have this transient response approach zero as rapidly as possible, that is, to minimize the **settling time**  $t_s$ . Theoretically, of course,  $t_s$  is infinite, because  $v(t)$  never settles to zero in a finite time. However, a negligible response is present after the magnitude of  $v(t)$  has settled to values that remain less than 1 percent of its maximum absolute value  $|v_m|$ . The time that is required for this to occur we define as the settling time. Since  $|v_m| = v_m = 48.9 \text{ V}$  for our example, the settling time is the time required for the response to drop to 0.489 V. Substituting this value for  $v(t)$  in Eq. [17] and neglecting the second exponential term, known to be negligible here, the settling time is found to be 5.15 s.



### EXAMPLE 9.4

For  $t > 0$ , the capacitor current of a certain source-free parallel RLC circuit is given by  $i_C(t) = 2e^{-2t} - 4e^{-t}$  A. Sketch the current in the range  $0 < t < 5$  s, and determine the settling time.

We first sketch the two terms as shown in Fig. 9.9, then subtract them to find  $i_C(t)$ . The maximum value is clearly  $|-2| = 2 \text{ A}$ . We therefore need to find the time at which  $|i_C|$  has decreased to 20 mA, or

$$2e^{-2t_s} - 4e^{-t_s} = -0.02 \quad [21]$$

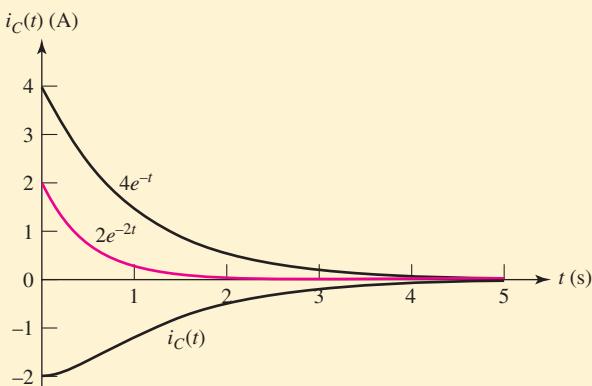


FIGURE 9.9 The current response  $i_C(t) = 2e^{-2t} - 4e^{-t}$  A, sketched alongside its two components.

This equation can be solved using an iterative solver routine on a scientific calculator, which returns the solution  $t_s = 5.296$  s. If such an option is not available, however, we can approximate Eq. [21] for  $t \geq t_s$  as

$$-4e^{-t_s} = -0.02 \quad [22]$$

Solving,

$$t_s = -\ln\left(\frac{0.02}{4}\right) = 5.298 \text{ s} \quad [23]$$

which is reasonably close (better than 0.1% accuracy) to the exact solution.

### PRACTICE

- 9.4 (a) Sketch the voltage  $v_R(t) = 2e^{-t} - 4e^{-3t}$  V in the range  $0 < t < 5$  s. (b) Estimate the settling time. (c) Calculate the maximum positive value and the time at which it occurs.

Ans: See Fig. 9.10; 5.9 s; 544 mV, 896 ms.

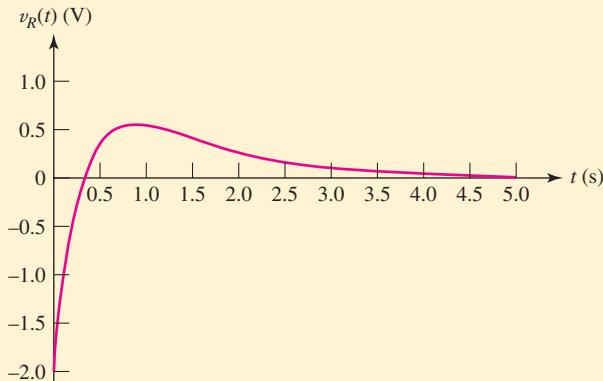


FIGURE 9.10 Response sketched for Practice Problem 9.4a.

### 9.3 CRITICAL DAMPING

The overdamped case is characterized by

$$\alpha > \omega_0$$

or

$$LC > 4R^2C^2$$

and leads to negative real values for  $s_1$  and  $s_2$  and to a response expressed as the algebraic sum of two negative exponentials.

Now let us adjust the element values until  $\alpha$  and  $\omega_0$  are equal. This is a very special case which is termed ***critical damping***. If we were to attempt to construct a parallel RLC circuit that is critically damped, we would be attempting an essentially impossible task, for we could never make  $\alpha$  exactly equal to  $\omega_0$ . For completeness, however, we will discuss the critically damped circuit here, because it shows an interesting transition between overdamping and underdamping.

Critical damping is achieved when

$$\left. \begin{array}{l} \alpha = \omega_0 \\ LC = 4R^2C^2 \\ L = 4R^2C \end{array} \right\} \text{critical damping}$$

We can produce critical damping by changing the value of any of the three elements in the numerical example discussed at the end of Sec. 9.1. We will select  $R$ , increasing its value until critical damping is obtained, and thus leave  $\omega_0$  unchanged. The necessary value of  $R$  is  $7\sqrt{6}/2 \Omega$ ;  $L$  is still 7 H, and  $C$  remains  $\frac{1}{42}$  F. We thus find

$$\begin{aligned} \alpha &= \omega_0 = \sqrt{6} \text{ s}^{-1} \\ s_1 &= s_2 = -\sqrt{6} \text{ s}^{-1} \end{aligned}$$

and recall the initial conditions that were specified,  $v(0) = 0$  and  $i(0) = 10 \text{ A}$ .

### Form of a Critically Damped Response

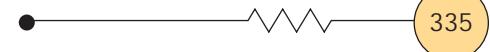
We proceed to attempt to construct a response as the sum of two exponentials,

$$v(t) \stackrel{?}{=} A_1 e^{-\sqrt{6}t} + A_2 e^{-\sqrt{6}t}$$

which may be written as

$$v(t) \stackrel{?}{=} A_3 e^{-\sqrt{6}t}$$

At this point, some of us might be feeling that something's wrong. We have a response that contains only one arbitrary constant, but there are two initial conditions,  $v(0) = 0$  and  $i(0) = 10 \text{ amperes}$ , *both of which* must be satisfied by this single constant. If we select  $A_3 = 0$ , then  $v(t) = 0$ , which is consistent with our initial capacitor voltage. However, although there is no energy stored in the capacitor at  $t = 0^+$ , we have 350 J of energy initially stored in the inductor. This energy will lead to a transient current flowing out of the inductor, giving rise to a nonzero voltage across all three elements. This seems to be in direct conflict with our proposed solution.



If a mistake has not led to our difficulties, we must have begun with an incorrect assumption, and only one assumption has been made. We originally hypothesized that the differential equation could be solved by assuming an exponential solution, and this turns out to be incorrect for this single special case of critical damping. When  $\alpha = \omega_0$ , the differential equation, Eq. [4], becomes

$$\frac{d^2v}{dt^2} + 2\alpha \frac{dv}{dt} + \alpha^2 v = 0$$

The solution of this equation is not a tremendously difficult process, but we will avoid developing it here, since the equation is a standard type found in the usual differential-equation texts. The solution is

$$v = e^{-\alpha t}(A_1 t + A_2) \quad [24]$$

It should be noted that the solution is still expressed as the sum of two terms, where one term is the familiar negative exponential and the second is  $t$  times a negative exponential. We should also note that the solution contains the *two* expected arbitrary constants.

## Finding Values for $A_1$ and $A_2$

Let us now complete our numerical example. After we substitute the known value of  $\alpha$  in Eq. [24], obtaining

$$v = A_1 t e^{-\sqrt{6}t} + A_2 e^{-\sqrt{6}t}$$

we establish the values of  $A_1$  and  $A_2$  by first imposing the initial condition on  $v(t)$  itself,  $v(0) = 0$ . Thus,  $A_2 = 0$ . This simple result occurs because the initial value of the response  $v(t)$  was selected as zero; the more general case will require the solution of two equations simultaneously. The second initial condition must be applied to the derivative  $dv/dt$  just as in the overdamped case. We therefore differentiate, remembering that  $A_2 = 0$ :

$$\frac{dv}{dt} = A_1 t (-\sqrt{6}) e^{-\sqrt{6}t} + A_1 e^{-\sqrt{6}t}$$

evaluate at  $t = 0$ :

$$\left. \frac{dv}{dt} \right|_{t=0} = A_1$$

and express the derivative in terms of the initial capacitor current:

$$\left. \frac{dv}{dt} \right|_{t=0} = \frac{i_C(0)}{C} = \frac{i_R(0)}{C} + \frac{i(0)}{C}$$

where reference directions for  $i_C$ ,  $i_R$ , and  $i$  are defined in Fig. 9.2. Thus,

$$A_1 = 420 \text{ V}$$

The response is, therefore,

$$v(t) = 420 t e^{-2.45t} \quad \text{V} \quad [25]$$

## Graphical Representation of the Critically Damped Response

Before plotting this response in detail, let us again try to anticipate its form by qualitative reasoning. The specified initial value is zero, and Eq. [25] concurs. It is not immediately apparent that the response also approaches zero as  $t$  becomes infinitely large, because  $te^{-2.45t}$  is an indeterminate form. However, this obstacle is easily overcome by use of L'Hôpital's rule, which yields

$$\lim_{t \rightarrow \infty} v(t) = 420 \lim_{t \rightarrow \infty} \frac{t}{e^{2.45t}} = 420 \lim_{t \rightarrow \infty} \frac{1}{2.45e^{2.45t}} = 0$$

and once again we have a response that begins and ends at zero and has positive values at all other times. A maximum value  $v_m$  again occurs at time  $t_m$ ; for our example,

$$t_m = 0.408 \text{ s} \quad \text{and} \quad v_m = 63.1 \text{ V}$$

This maximum is larger than that obtained in the overdamped case, and is a result of the smaller losses that occur in the larger resistor; the time of the maximum response is slightly later than it was with overdamping. The settling time may also be determined by solving

$$\frac{v_m}{100} = 420t_s e^{-2.45t_s}$$

for  $t_s$  (by trial-and-error methods or a calculator's SOLVE routine):

$$t_s = 3.12 \text{ s}$$



which is a considerably smaller value than that which arose in the overdamped case (5.15 s). As a matter of fact, it can be shown that, for given values of  $L$  and  $C$ , the selection of that value of  $R$  which provides critical damping will always give a shorter settling time than any choice of  $R$  that produces an overdamped response. However, a slight improvement (reduction) in settling time may be obtained by a further slight increase in resistance; a slightly underdamped response that will undershoot the zero axis before it dies out will yield the shortest settling time.

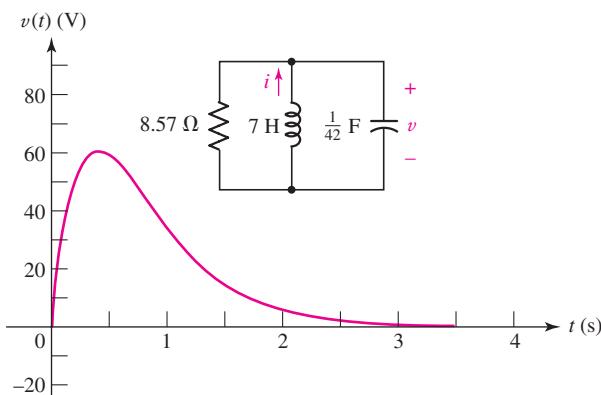


FIGURE 9.11 The response  $v(t) = 420te^{-2.45t}$  of the network shown in Fig. 9.2 with  $R$  changed to provide critical damping.

The response curve for critical damping is drawn in Fig. 9.11; it may be compared with the overdamped (and underdamped) case by reference to Fig. 9.16.

## EXAMPLE 9.5

Select a value for  $R_1$  such that the circuit of Fig. 9.12 will be characterized by a critically damped response for  $t > 0$ , and a value for  $R_2$  such that  $v(0) = 2$  V.

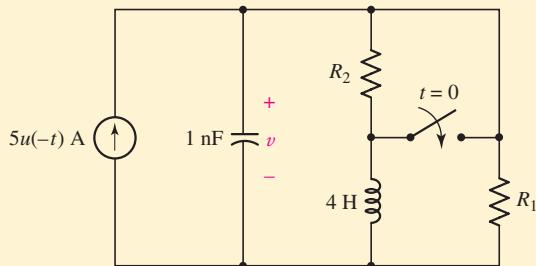


FIGURE 9.12 A circuit that reduces to a parallel  $RLC$  circuit after the switch is thrown.

We note that at  $t = 0^-$ , the current source is on, and the inductor can be treated as a short circuit. Thus,  $v(0^-)$  appears across  $R_2$ , and is given by

$$v(0^-) = 5R_2$$

and a value of  $400 \text{ m}\Omega$  should be selected for  $R_2$  to obtain  $v(0) = 2 \text{ V}$ .

After the switch is thrown, the current source has turned itself off and  $R_2$  is shorted. We are left with a parallel  $RLC$  circuit comprised of  $R_1$ , a  $4 \text{ H}$  inductor, and a  $1 \text{ nF}$  capacitor.

We may now calculate (for  $t > 0$ )

$$\begin{aligned} \alpha &= \frac{1}{2RC} \\ &= \frac{1}{2 \times 10^{-9}R_1} \end{aligned}$$

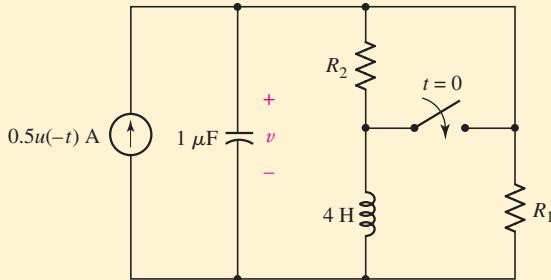
and

$$\begin{aligned} \omega_0 &= \frac{1}{\sqrt{LC}} \\ &= \frac{1}{\sqrt{4 \times 10^{-9}}} \\ &= 15,810 \text{ rad/s} \end{aligned}$$

Therefore, to establish a critically damped response in the circuit for  $t > 0$ , we need to set  $R_1 = 31.63 \text{ k}\Omega$ . (Note: since we have rounded to four significant figures, the pedantic can rightly argue that this is still not exactly a critically damped response—a difficult situation to create.)

**PRACTICE**

9.5 (a) Choose  $R_1$  in the circuit of Fig. 9.13 so that the response after  $t = 0$  will be critically damped. (b) Now select  $R_2$  to obtain  $v(0) = 100$  V. (c) Find  $v(t)$  at  $t = 1$  ms.



■ FIGURE 9.13

Ans: 1 kΩ; 250 Ω; -212 V.

## 9.4 THE UNDERDAMPED PARALLEL RLC CIRCUIT

Let us continue the process begun in Sec. 9.3 by increasing  $R$  once more to obtain what we will refer to as an ***underdamped*** response. Thus, the damping coefficient  $\alpha$  decreases while  $\omega_0$  remains constant,  $\alpha^2$  becomes smaller than  $\omega_0^2$ , and the radicand appearing in the expressions for  $s_1$  and  $s_2$  becomes negative. This causes the response to take on a much different character, but it is fortunately not necessary to return to the basic differential equation again. By using complex numbers, the exponential response turns into a ***damped sinusoidal response***; this response is composed entirely of real quantities, the complex quantities being necessary only for the derivation.<sup>1</sup>

### The Form of the Underdamped Response

We begin with the exponential form

$$v(t) = A_1 e^{s_1 t} + A_2 e^{s_2 t}$$

where

$$s_{1,2} = -\alpha \pm \sqrt{\alpha^2 - \omega_0^2}$$

and then let

$$\sqrt{\alpha^2 - \omega_0^2} = \sqrt{-1} \sqrt{\omega_0^2 - \alpha^2} = j \sqrt{\omega_0^2 - \alpha^2}$$

where  $j \equiv \sqrt{-1}$ .

We now take the new radical, which is real for the underdamped case, and call it  $\omega_d$ , the ***natural resonant frequency***:

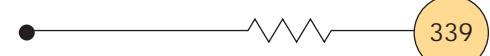
$$\omega_d = \sqrt{\omega_0^2 - \alpha^2}$$

The response may now be written as

$$v(t) = e^{-\alpha t} (A_1 e^{j\omega_d t} + A_2 e^{-j\omega_d t}) \quad [26]$$

Electrical engineers use "j" instead of "i" to represent  $\sqrt{-1}$  to avoid confusion with currents.

(1) A review of complex numbers is presented in Appendix 5.



or, in the longer but equivalent form,

$$v(t) = e^{-\alpha t} \left\{ (A_1 + A_2) \left[ \frac{e^{j\omega_d t} + e^{-j\omega_d t}}{2} \right] + j(A_1 - A_2) \left[ \frac{e^{j\omega_d t} - e^{-j\omega_d t}}{j2} \right] \right\}$$

Applying identities described in Appendix 5, the term in the first square brackets in the preceding equation is identically equal to  $\cos \omega_d t$ , and the second is identically  $\sin \omega_d t$ . Hence,

$$v(t) = e^{-\alpha t} [(A_1 + A_2) \cos \omega_d t + j(A_1 - A_2) \sin \omega_d t]$$

and the multiplying factors may be assigned new symbols:

$$v(t) = e^{-\alpha t} (B_1 \cos \omega_d t + B_2 \sin \omega_d t) \quad [27]$$

where Eqs. [26] and [27] are identical.

It may seem a little odd that our expression originally appeared to have a complex component, and now is purely real. However, we should remember that we originally allowed for  $A_1$  and  $A_2$  to be complex as well as  $s_1$  and  $s_2$ . In any event, if we are dealing with the underdamped case, we have now left complex numbers behind. This must be true since  $\alpha$ ,  $\omega_d$ , and  $t$  are real quantities, so that  $v(t)$  itself must be a real quantity (which might be presented on an oscilloscope, a voltmeter, or a sheet of graph paper). Equation [27] is the desired functional form for the underdamped response, and its validity may be checked by direct substitution in the original differential equation; this exercise is left to the doubters. The two real constants  $B_1$  and  $B_2$  are again selected to fit the given initial conditions.

We return to our simple parallel RLC circuit of Fig. 9.2 with  $R = 6 \Omega$ ,  $C = 1/42 \text{ F}$ , and  $L = 7 \text{ H}$ , but now increase the resistance further to  $10.5 \Omega$ . Thus,

$$\alpha = \frac{1}{2RC} = 2 \text{ s}^{-1}$$

$$\omega_0 = \frac{1}{\sqrt{LC}} = \sqrt{6} \text{ s}^{-1}$$

and

$$\omega_d = \sqrt{\omega_0^2 - \alpha^2} = \sqrt{2} \text{ rad/s}$$

Except for the evaluation of the arbitrary constants, the response is now known:

$$v(t) = e^{-2t} (B_1 \cos \sqrt{2}t + B_2 \sin \sqrt{2}t)$$

## Finding Values for $B_1$ and $B_2$

The determination of the two constants proceeds as before. If we still assume that  $v(0) = 0$  and  $i(0) = 10$ , then  $B_1$  must be zero. Hence

$$v(t) = B_2 e^{-2t} \sin \sqrt{2}t$$

The derivative is

$$\frac{dv}{dt} = \sqrt{2} B_2 e^{-2t} \cos \sqrt{2}t - 2B_2 e^{-2t} \sin \sqrt{2}t$$



and at  $t = 0$  it becomes

$$\left. \frac{dv}{dt} \right|_{t=0} = \sqrt{2}B_2 = \frac{i_C(0)}{C} = 420$$

where  $i_C$  is defined in Fig. 9.2. Therefore,

$$v(t) = 210\sqrt{2}e^{-2t} \sin \sqrt{2}t$$

## Graphical Representation of the Underdamped Response

Notice that, as before, this response function has an initial value of zero because of the initial voltage condition we imposed, and a final value of zero because the exponential term vanishes for large values of  $t$ . As  $t$  increases from zero through small positive values,  $v(t)$  increases as  $210\sqrt{2} \sin \sqrt{2}t$ , because the exponential term remains essentially equal to unity. But, at some time  $t_m$ , the exponential function begins to decrease more rapidly than  $\sin \sqrt{2}t$  is increasing; thus  $v(t)$  reaches a maximum  $v_m$  and begins to decrease. We should note that  $t_m$  is not the value of  $t$  for which  $\sin \sqrt{2}t$  is a maximum, but must occur somewhat before  $\sin \sqrt{2}t$  reaches its maximum.



When  $t = \pi/\sqrt{2}$ ,  $v(t)$  is zero. Thus, in the interval  $\pi/\sqrt{2} < t < \sqrt{2}\pi$ , the response is negative, becoming zero again at  $t = \sqrt{2}\pi$ . Hence,  $v(t)$  is an oscillatory function of time and crosses the time axis an infinite number of times at  $t = n\pi/\sqrt{2}$ , where  $n$  is any positive integer. In our example, however, the response is only slightly underdamped, and the exponential term causes the function to die out so rapidly that most of the zero crossings will not be evident in a sketch.

The oscillatory nature of the response becomes more noticeable as  $\alpha$  decreases. If  $\alpha$  is zero, which corresponds to an infinitely large resistance, then  $v(t)$  is an undamped sinusoid that oscillates with constant amplitude. There is never a time at which  $v(t)$  drops and stays below 1 percent of its maximum value; the settling time is therefore infinite. This is not perpetual motion; we have merely assumed an initial energy in the circuit and have not provided any means to dissipate this energy. It is transferred from its initial location in the inductor to the capacitor, then returns to the inductor, and so on, forever.

## The Role of Finite Resistance

A finite  $R$  in the parallel  $RLC$  circuit acts as a kind of electrical transfer agent. Every time energy is transferred from  $L$  to  $C$  or from  $C$  to  $L$ , the agent exacts a commission. Before long, the agent has taken all the energy, wantonly dissipating every last joule. The  $L$  and  $C$  are left without a joule of their own, without voltage and without current. Actual parallel  $RLC$  circuits can be made to have effective values of  $R$  so large that a natural undamped sinusoidal response can be maintained for years without supplying any additional energy.

Returning to our specific numerical problem, differentiation locates the first maximum of  $v(t)$ ,

$$v_{m_1} = 71.8 \text{ V} \quad \text{at} \quad t_{m_1} = 0.435 \text{ s}$$

the succeeding minimum,

$$v_{m_2} = -0.845 \text{ V} \quad \text{at} \quad t_{m_2} = 2.66 \text{ s}$$

and so on. The response curve is shown in Fig. 9.14. Additional response curves for increasingly more underdamped circuits are shown in Fig. 9.15.

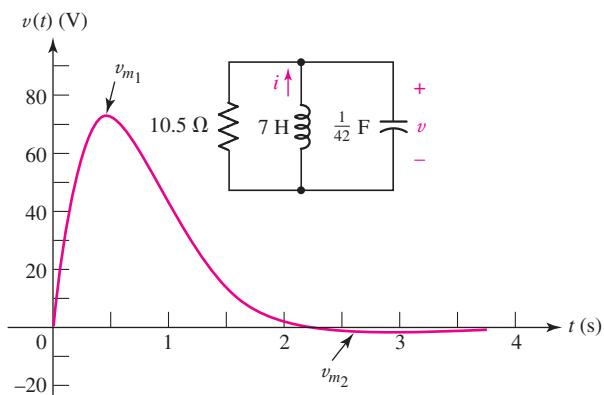


FIGURE 9.14 The response  $v(t) = 210\sqrt{2}e^{-2t} \sin \sqrt{2}t$  of the network shown in Fig. 9.2 with  $R$  increased to produce an underdamped response.

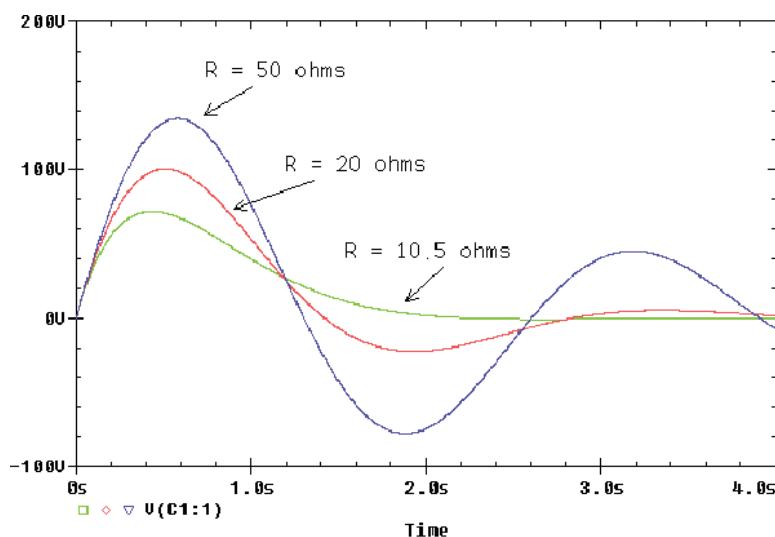


FIGURE 9.15 Simulated underdamped voltage response of the network for three different resistance values, showing an increase in the oscillatory behavior as  $R$  is increased.

The settling time may be obtained by a trial-and-error solution, and for  $R = 10.5 \Omega$ , it turns out to be 2.92 s, somewhat smaller than for critical damping. Note that  $t_s$  is *greater* than  $t_{m_2}$  because the magnitude of  $v_{m_2}$  is greater than 1 percent of the magnitude of  $v_{m_1}$ . This suggests that a slight decrease in  $R$  would reduce the magnitude of the undershoot and permit  $t_s$  to be less than  $t_{m_2}$ .

The overdamped, critically damped, and underdamped responses for this network as simulated by PSpice are shown on the same graph in Fig. 9.16. A comparison of the three curves makes the following general

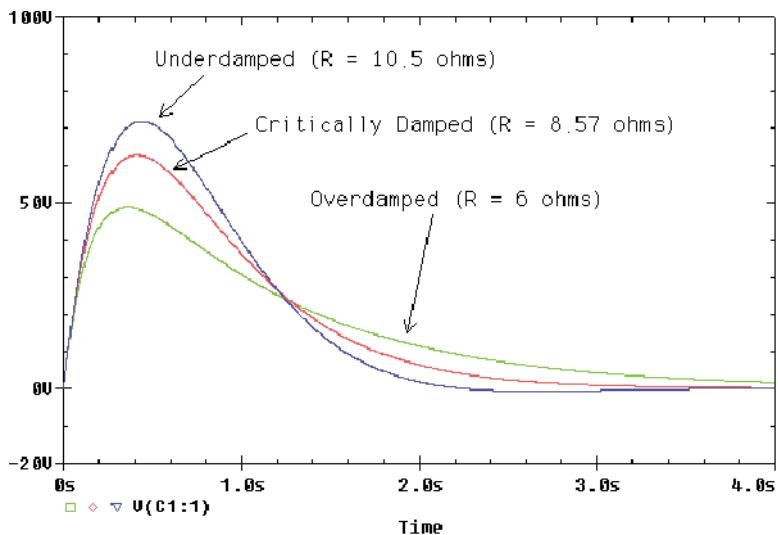


FIGURE 9.16 Simulated overdamped, critically damped, and underdamped voltage response for the example network, obtained by varying the value of the parallel resistance  $R$ .

conclusions plausible:

- When the damping is changed by increasing the size of the parallel resistance, the maximum magnitude of the response is greater and the amount of damping is smaller.
- The response becomes oscillatory when underdamping is present, and the minimum settling time is obtained for slight underdamping.

## EXAMPLE 9.6

Determine  $i_L(t)$  for the circuit of Fig. 9.17a, and plot the waveform.

At  $t = 0$ , both the 3 A source and the  $48 \Omega$  resistor are removed, leaving the circuit shown in Fig. 9.17b. Thus,  $\alpha = 1.2 \text{ s}^{-1}$  and  $\omega_0 = 4.899 \text{ rad/s}$ . Since  $\alpha < \omega_0$ , the circuit is *underdamped*, and we therefore expect a response of the form

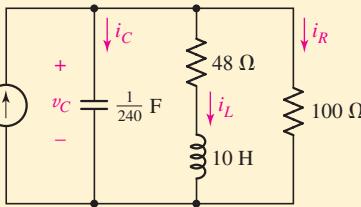
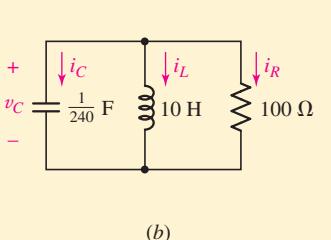
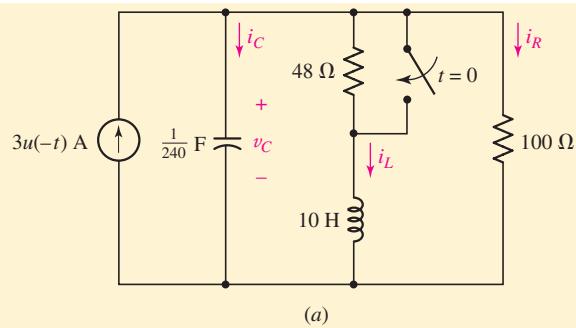
$$i_L(t) = e^{-\alpha t}(B_1 \cos \omega_d t + B_2 \sin \omega_d t) \quad [28]$$

where  $\omega_d = \sqrt{\omega_0^2 - \alpha^2} = 4.750 \text{ rad/s}$ . The only remaining step is to find  $B_1$  and  $B_2$ .

Figure 9.17c shows the circuit as it exists at  $t = 0^-$ . We may replace the inductor with a short circuit and the capacitor with an open circuit; the result is  $v_C(0^-) = 97.30 \text{ V}$  and  $i_L(0^-) = 2.027 \text{ A}$ . Since neither quantity can change in zero time,  $v_C(0^+) = 97.30 \text{ V}$  and  $i_L(0^+) = 2.027 \text{ A}$ .

Substituting  $i_L(0^-) = 2.027$  into Eq. [28] yields  $B_1 = 2.027 \text{ A}$ . To determine the other constant, we first differentiate Eq. [28]:

$$\frac{di_L}{dt} = e^{-\alpha t}(-B_1 \omega_d \sin \omega_d t + B_2 \omega_d \cos \omega_d t) - \alpha e^{-\alpha t}(B_1 \cos \omega_d t + B_2 \sin \omega_d t) \quad [29]$$



■ FIGURE 9.17 (a) A parallel RLC circuit for which the current  $i_L(t)$  is desired.  
 (b) Circuit for  $t \geq 0$ . (c) Circuit for determining the initial conditions.

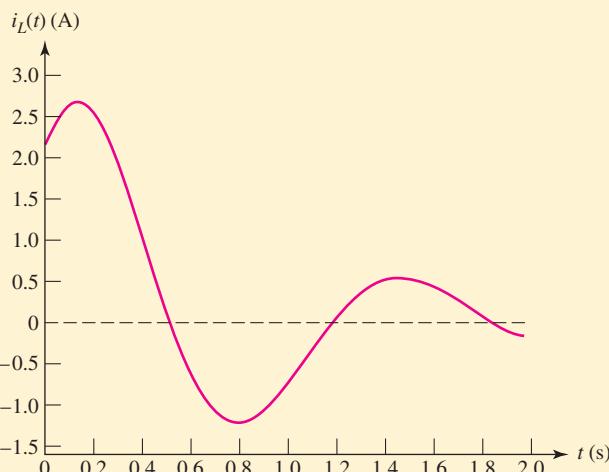
and note that  $v_L(t) = L(di_L/dt)$ . Referring to the circuit of Fig. 9.17b, we see that  $v_L(0^+) = v_C(0^+) = 97.3$  V. Thus, multiplying Eq. [29] by  $L = 10$  H and setting  $t = 0$ , we find that

$$v_L(0) = 10(B_2\omega_d) - 10\alpha B_1 = 97.3$$

Solving,  $B_2 = 2.561$  A, so that

$$i_L = e^{-1.2t}(2.027 \cos 4.75t + 2.561 \sin 4.75t) \quad \text{A}$$

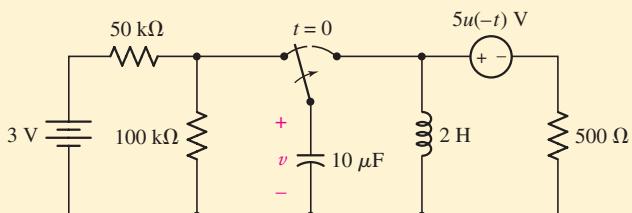
which we have plotted in Fig. 9.18.



■ FIGURE 9.18 Plot of  $i_L(t)$ , showing obvious signs of being an underdamped response.

**PRACTICE**

9.6 The switch in the circuit of Fig. 9.19 has been in the left position for a long time; it is moved to the right at  $t = 0$ . Find (a)  $dv/dt$  at  $t = 0^+$ ; (b)  $v$  at  $t = 1$  ms; (c)  $t_0$ , the first value of  $t$  greater than zero at which  $v = 0$ .



■ FIGURE 9.19

Ans:  $-1400 \text{ V/s}$ ;  $0.695 \text{ V}$ ;  $1.609 \text{ ms}$ .

**COMPUTER-AIDED ANALYSIS**

One useful feature in Probe is the ability to perform mathematical operations on the voltages and currents that result from a simulation. In this example, we will make use of that ability to show the transfer of energy in a parallel *RLC* circuit from a capacitor that initially stores a specific amount of energy ( $1.25 \mu\text{J}$ ) to an inductor that initially stores no energy.

We choose a  $100 \text{ nF}$  capacitor and a  $7 \mu\text{H}$  inductor, which immediately enables us to calculate  $\omega_0 = 1.195 \times 10^6 \text{ s}^{-1}$ . In order to consider overdamped, critically damped, and underdamped cases, we need to select the parallel resistance in such a way as to obtain  $\alpha > \omega_0$  (*over-damped*),  $\alpha = \omega_0$  (*critically damped*), and  $\alpha < \omega_0$  (*underdamped*). From our previous discussions, we know that for a parallel *RLC* circuit  $\alpha = (2RC)^{-1}$ . We select  $R = 4.1833 \Omega$  as a close approximation to the critically damped case; obtaining  $\alpha$  precisely equal to  $\omega_0$  is effectively impossible. If we increase the resistance, the energy stored in the other two elements is dissipated more slowly, resulting in an underdamped response. We select  $R = 100 \Omega$  so that we are well into this regime, and use  $R = 1 \Omega$  (a very small resistance) to obtain an overdamped response.

We therefore plan to run three separate simulations, varying only the resistance  $R$  between them. The  $1.25 \mu\text{J}$  of energy initially stored in the capacitor equates to an initial voltage of  $5 \text{ V}$ , and so we set the initial condition of our capacitor accordingly.

Once Probe is launched, we select **Add** under the **Trace** menu. We wish to plot the energy stored in both the inductor and the capacitor as a function of time. For the capacitor,  $w = \frac{1}{2}Cv^2$ , so we click in the **Trace Expression** window, type in “ $0.5*100E-9*$ ” (without the quotes), click on  $V(C1:1)$ , return to the **Trace Expression** window and enter “\*”, click on  $V(C1:1)$  once again, and then select **Ok**. We repeat the sequence to obtain the energy stored in the inductor, using  $7E-6$  instead of  $100E-9$ , and clicking on  $I(L1:1)$  instead of  $V(C1:1)$ .

The Probe output plots for three separate simulations are provided in Fig. 9.20. In Fig. 9.20a, we see that the energy remaining in the circuit is continuously transferred back and forth between the capacitor and the inductor until it is (eventually) completely dissipated by the resistor. Decreasing the resistance to  $4.1833 \Omega$  yields a critically damped circuit, resulting in the energy plot of Fig. 9.20b. The oscillatory energy transfer between the capacitor and the inductor has been dramatically reduced. We see that the energy transferred to the inductor peaks at approximately  $0.8 \mu\text{s}$ , and then drops to zero. The overdamped response is plotted in Fig. 9.20c. We note that the energy is dissipated much more quickly in the case of the overdamped response, and that very little energy is transferred to the inductor, since most of it is now quickly dissipated in the resistor.

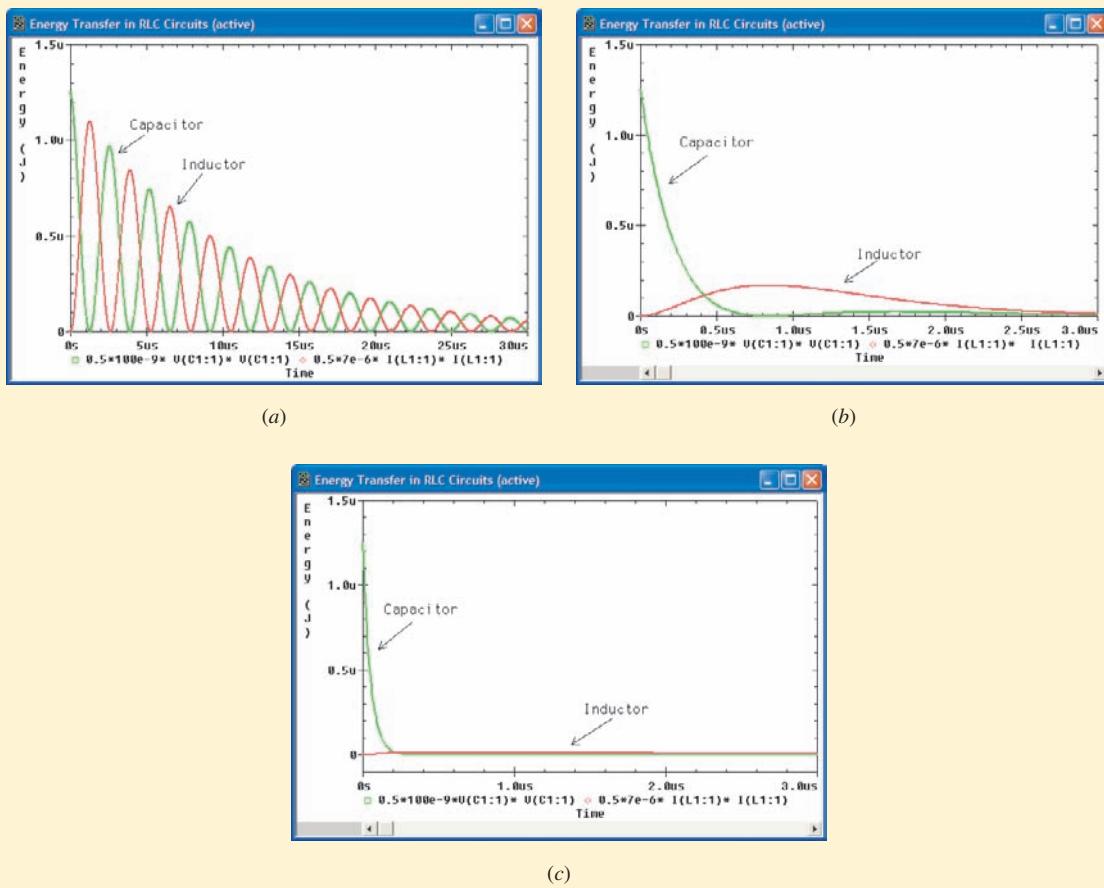
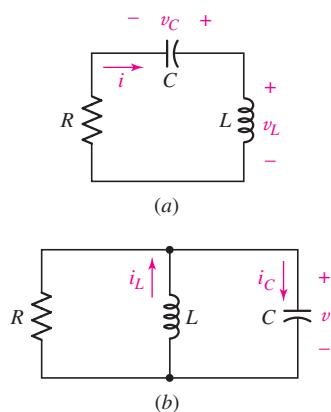


FIGURE 9.20 Energy transfer in a parallel RLC circuit with (a)  $R = 100 \Omega$  (underdamped); (b)  $R = 4.1833 \Omega$  (critically damped); and (c)  $R = 1 \Omega$  (overdamped).

## 9.5 THE SOURCE-FREE SERIES RLC CIRCUIT

We now wish to determine the natural response of a circuit model composed of an ideal resistor, an ideal inductor, and an ideal capacitor connected in series. The ideal resistor may represent a physical resistor connected into a series LC or RLC circuit; it may represent the ohmic losses and the losses in the ferromagnetic core of the inductor; or it may be used to represent all these and other energy-absorbing devices.



**FIGURE 9.21** (a) The series RLC circuit which is the dual of (b) a parallel RLC circuit. Element values are, of course, not identical in the two circuits.

The series RLC circuit is the *dual* of the parallel RLC circuit, and this single fact is sufficient to make its analysis a trivial affair. Figure 9.21a shows the series circuit. The fundamental integrodifferential equation is

$$L \frac{di}{dt} + Ri + \frac{1}{C} \int_{t_0}^t i dt' - v_C(t_0) = 0$$

and should be compared with the analogous equation for the parallel RLC circuit, drawn again in Fig. 9.21b,

$$C \frac{dv}{dt} + \frac{1}{R} v + \frac{1}{L} \int_{t_0}^t v dt' - i_L(t_0) = 0$$

The respective second-order equations obtained by differentiating these two equations with respect to time are also duals:

$$L \frac{d^2i}{dt^2} + R \frac{di}{dt} + \frac{1}{C} i = 0 \quad [30]$$

$$C \frac{d^2v}{dt^2} + \frac{1}{R} \frac{dv}{dt} + \frac{1}{L} v = 0 \quad [31]$$

Our complete discussion of the parallel RLC circuit is directly applicable to the series RLC circuit; the initial conditions on capacitor voltage and inductor current are equivalent to the initial conditions on inductor current and capacitor voltage; the *voltage* response becomes a *current* response. It is therefore possible to reread the previous four sections using dual language and thereby obtain a complete description of the series RLC circuit. This process, however, is apt to induce a mild neurosis after the first few paragraphs and does not really seem to be necessary.

## A Brief Résumé of the Series Circuit Response

In terms of the circuit shown in Fig. 9.21a, the *overdamped response* is

$$i(t) = A_1 e^{s_1 t} + A_2 e^{s_2 t}$$

where

$$s_{1,2} = -\frac{R}{2L} \pm \sqrt{\left(\frac{R}{2L}\right)^2 - \frac{1}{LC}} = -\alpha \pm \sqrt{\alpha^2 - \omega_0^2}$$

and thus

$$\alpha = \frac{R}{2L}$$

$$\omega_0 = \frac{1}{\sqrt{LC}}$$

The form of the *critically damped response* is

$$i(t) = e^{-\alpha t}(A_1 t + A_2)$$

and the *underdamped response* may be written

$$i(t) = e^{-\alpha t}(B_1 \cos \omega_d t + B_2 \sin \omega_d t)$$

**TABLE 9.1** Summary of Relevant Equations for Source-Free RLC Circuits

Type	Condition	Criteria	$\alpha$	$\omega_0$	Response
Parallel Series	Overdamped $\alpha > \omega_0$		$\frac{1}{2RC}$	$\frac{1}{\sqrt{LC}}$	$A_1 e^{s_1 t} + A_2 e^{s_2 t}$ , where $s_{1,2} = -\alpha \pm \sqrt{\alpha^2 - \omega_0^2}$
			$\frac{R}{2L}$		
Parallel Series	Critically damped $\alpha = \omega_0$		$\frac{1}{2RC}$	$\frac{1}{\sqrt{LC}}$	$e^{-\alpha t} (A_1 t + A_2)$
			$\frac{R}{2L}$		
Parallel Series	Underdamped $\alpha < \omega_0$		$\frac{1}{2RC}$	$\frac{1}{\sqrt{LC}}$	$e^{-\alpha t} (B_1 \cos \omega_d t + B_2 \sin \omega_d t)$ , where $\omega_d = \sqrt{\omega_0^2 - \alpha^2}$
			$\frac{R}{2L}$		

where

$$\omega_d = \sqrt{\omega_0^2 - \alpha^2}$$

It is evident that if we work in terms of the parameters  $\alpha$ ,  $\omega_0$ , and  $\omega_d$ , the mathematical forms of the responses for the dual situations are identical. An increase in  $\alpha$  in either the series or parallel circuit, while keeping  $\omega_0$  constant, tends toward an overdamped response. The only caution that we need exert is in the computation of  $\alpha$ , which is  $1/2RC$  for the parallel circuit and  $R/2L$  for the series circuit; thus,  $\alpha$  is increased by increasing the series resistance or decreasing the parallel resistance. The key equations for parallel and series RLC circuits are summarized in Table 9.1 for convenience.



### EXAMPLE 9.7

Given the series RLC circuit of Fig. 9.22 in which  $L = 1 \text{ H}$ ,  $R = 2 \text{ k}\Omega$ ,  $C = 1/401 \mu\text{F}$ ,  $i(0) = 2 \text{ mA}$ , and  $v_C(0) = 2 \text{ V}$ , find and sketch  $i(t)$ ,  $t > 0$ .

We find that  $\alpha = R/2L = 1000 \text{ s}^{-1}$  and  $\omega_0 = 1/\sqrt{LC} = 20,025 \text{ rad/s}$ . This indicates an *underdamped* response; we therefore calculate the value of  $\omega_d$  and obtain  $20,000 \text{ rad/s}$ . Except for the evaluation of the two arbitrary constants, the response is now known:

$$i(t) = e^{-1000t} (B_1 \cos 20,000t + B_2 \sin 20,000t)$$

Since we know that  $i(0) = 2 \text{ mA}$ , we may substitute this value into our equation for  $i(t)$  to obtain

$$B_1 = 0.002 \text{ A}$$

and thus

$$i(t) = e^{-1000t} (0.002 \cos 20,000t + B_2 \sin 20,000t) \quad \text{A}$$

(Continued on next page)

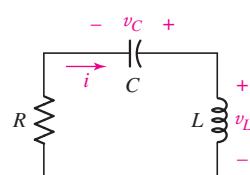


FIGURE 9.22 A simple source-free RLC circuit with energy stored in both the inductor and the capacitor at  $t = 0$ .

The remaining initial condition must be applied to the derivative; thus,

$$\begin{aligned}\frac{di}{dt} &= e^{-1000t}(-40 \sin 20,000t + 20,000B_2 \cos 20,000t \\ &\quad - 2 \cos 20,000t - 1000B_2 \sin 20,000t)\end{aligned}$$

and

$$\begin{aligned}\left. \frac{di}{dt} \right|_{t=0} &= 20,000B_2 - 2 = \frac{v_L(0)}{L} \\ &= \frac{v_C(0) - Ri(0)}{L} \\ &= \frac{2 - 2000(0.002)}{1} = -2 \text{ A/s}\end{aligned}$$

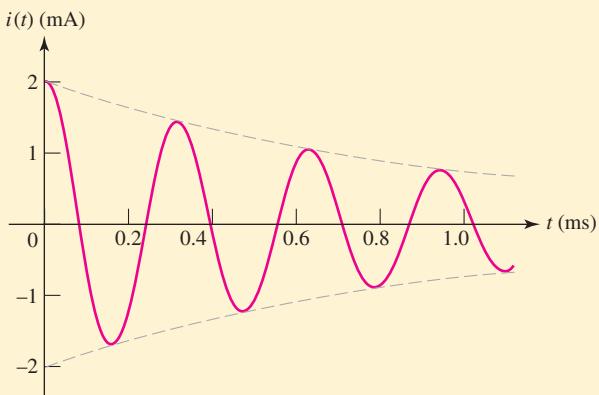
so that

$$B_2 = 0$$

The desired response is therefore

$$i(t) = 2e^{-1000t} \cos 20,000t \text{ mA}$$

A good sketch may be made by first drawing in the two portions of the exponential envelope,  $2e^{-1000t}$  and  $-2e^{-1000t}$  mA, as shown by the broken lines in Fig. 9.23. The location of the quarter-cycle points of the sinusoidal wave at  $20,000t = 0, \pi/2, \pi$ , etc., or  $t = 0.07854k$  ms,  $k = 0, 1, 2, \dots$ , by light marks on the time axis then permits the oscillatory curve to be sketched in quickly.

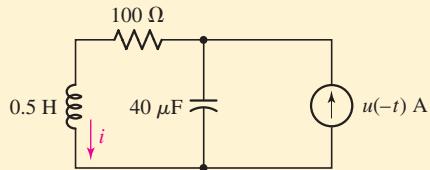


**FIGURE 9.23** The current response in an underdamped series RLC circuit for which  $\alpha = 1000 \text{ s}^{-1}$ ,  $\omega_0 = 20,000 \text{ s}^{-1}$ ,  $i(0) = 2 \text{ mA}$ , and  $v_C(0) = 2 \text{ V}$ . The graphical construction is simplified by drawing in the envelope, shown as a pair of broken lines.

The settling time can be determined easily here by using the upper portion of the envelope. That is, we set  $2e^{-1000t_s}$  mA equal to 1 percent of its maximum value, 2 mA. Thus,  $e^{-1000t_s} = 0.01$ , and  $t_s = 4.61 \text{ ms}$  is the approximate value that is usually used.

**PRACTICE**

- 9.7 With reference to the circuit shown in Fig. 9.24, find (a)  $\alpha$ ; (b)  $\omega_0$ ; (c)  $i(0^+)$ ; (d)  $di/dt|_{t=0^+}$ ; (e)  $i(12 \text{ ms})$ .



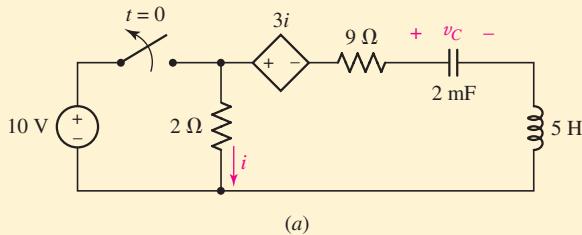
■ FIGURE 9.24

Ans:  $100 \text{ s}^{-1}$ ;  $224 \text{ rad/s}$ ;  $1 \text{ A}$ ;  $0$ ;  $-0.1204 \text{ A}$ .

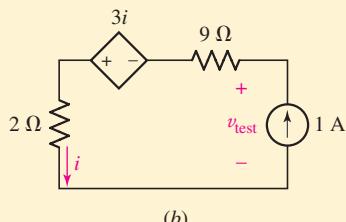
As a final example, we pause to consider situations where the circuit includes a dependent source. If no controlling current or voltage associated with the dependent source is of interest, we may simply find the Thévenin equivalent connected to the inductor and capacitor. Otherwise, we are likely faced with having to write an appropriate integrodifferential equation, take the indicated derivative, and solve the resulting differential equation as best we can.

**EXAMPLE 9.8**

- Find an expression for  $v_C(t)$  in the circuit of Fig. 9.25a, valid for  $t > 0$ .



(a)



(b)

- FIGURE 9.25 (a) An RLC circuit containing a dependent source.  
(b) Circuit for finding  $R_{\text{eq}}$ .

As we are interested only in  $v_C(t)$ , it is perfectly acceptable to begin by finding the Thévenin equivalent resistance connected in series with the

*(Continued on next page)*

inductor and capacitor at  $t = 0^+$ . We do this by connecting a 1 A source as shown in Fig. 9.25b, from which we deduce that

$$v_{\text{test}} = 11i - 3i = 8i = 8(1) = 8 \text{ V}$$

Thus,  $R_{\text{eq}} = 8 \Omega$ , so  $\alpha = R/2L = 0.8 \text{ s}^{-1}$  and  $\omega_0 = 1/\sqrt{LC} = 10 \text{ rad/s}$ , meaning that we expect an underdamped response with  $\omega_d = 9.968 \text{ rad/s}$  and the form

$$v_C(t) = e^{-0.8t}(B_1 \cos 9.968t + B_2 \sin 9.968t) \quad [32]$$

In considering the circuit at  $t = 0^-$ , we note that  $i_L(0^-) = 0$  due to the presence of the capacitor. By Ohm's law,  $i(0^-) = 5 \text{ A}$ , so

$$v_C(0^+) = v_C(0^-) = 10 - 3i = 10 - 15 = -5 \text{ V}$$

This last condition substituted into Eq. [32] yields  $B_1 = -5 \text{ V}$ . Taking the derivative of Eq. [32] and evaluating at  $t = 0$  yield

$$\left. \frac{dv_C}{dt} \right|_{t=0} = -0.8B_1 + 9.968B_2 = 4 + 9.968B_2 \quad [33]$$

We see from Fig. 9.25a that

$$i = -C \frac{dv_C}{dt}$$

Thus, making use of the fact that  $i(0^+) = i_L(0^-) = 0$  in Eq. [33] yields  $B_2 = -0.4013 \text{ V}$ , and we may write

$$v_C(t) = -e^{-0.8t}(5 \cos 9.968t + 0.4013 \sin 9.968t) \quad \text{V} \quad t > 0$$

The PSpice simulation of this circuit, shown in Fig. 9.26, confirms our analysis.

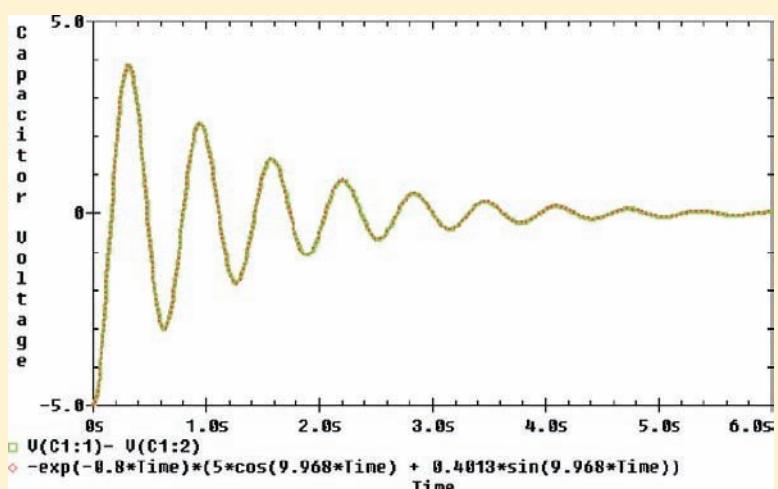


FIGURE 9.26 PSpice simulation of the circuit shown in Fig. 9.25a. The analytical result is plotted using a dashed red line.

**PRACTICE**

9.8 Find an expression for  $i_L(t)$  in the circuit of Fig. 9.27, valid for  $t > 0$ , if  $v_C(0^-) = 10 \text{ V}$  and  $i_L(0^-) = 0$ . Note that although it is not helpful to apply Thévenin techniques in this instance, the action of the dependent source links  $v_C$  and  $i_L$  such that a first-order linear differential equation results.

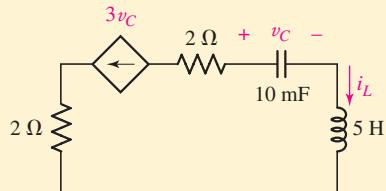


FIGURE 9.27 Circuit for Practice Problem 9.8.

Ans:  $i_L(t) = -30e^{-300t} \text{ A}, t > 0$ .

## 9.6 THE COMPLETE RESPONSE OF THE RLC CIRCUIT

We now consider those RLC circuits in which dc sources are switched into the network and produce forced responses that do not necessarily vanish as time becomes infinite.

The general solution is obtained by the same procedure that was followed for RL and RC circuits. The basic steps are (not necessarily in this order) as follows:

1. Determine the initial conditions.
2. Obtain a numerical value for the forced response.
3. Write the appropriate form of the natural response with the necessary number of arbitrary constants.
4. Add the forced response and natural response to form the complete response.
5. Evaluate the response and its derivative at  $t = 0$ , and employ the initial conditions to solve for the values of the unknown constants.

We note that *it is generally this last step that causes the most trouble for students*, as the circuit must be carefully evaluated at  $t = 0$  to make full use of the initial conditions. Consequently, although the determination of the initial conditions is basically no different for a circuit containing dc sources from what it is for the source-free circuits that we have already covered in some detail, this topic will receive particular emphasis in the examples that follow.

Most of the confusion in determining and applying the initial conditions arises for the simple reason that we do not have a rigorous set of rules laid down for us to follow. At some point in each analysis, a situation usually arises in which some thinking is involved that is more or less unique to that particular problem. This is almost always the source of the difficulty.



### The Easy Part

The *complete* response (arbitrarily assumed to be a voltage response) of a second-order system consists of a *forced* response,

$$v_f(t) = V_f$$

which is a constant for dc excitation, and a *natural* response,

$$v_n(t) = Ae^{s_1 t} + Be^{s_2 t}$$

Thus,

$$v(t) = V_f + Ae^{s_1 t} + Be^{s_2 t}$$

We assume that  $s_1$ ,  $s_2$ , and  $V_f$  have already been determined from the circuit and the given forcing functions;  $A$  and  $B$  remain to be found. The last equation shows the functional interdependence of  $A$ ,  $B$ ,  $v$ , and  $t$ ; and substitution of the known value of  $v$  at  $t = 0^+$  thus provides us with a single equation relating  $A$  and  $B$ ,  $v(0^+) = V_f + A + B$ . This is the easy part.

## The Other Part

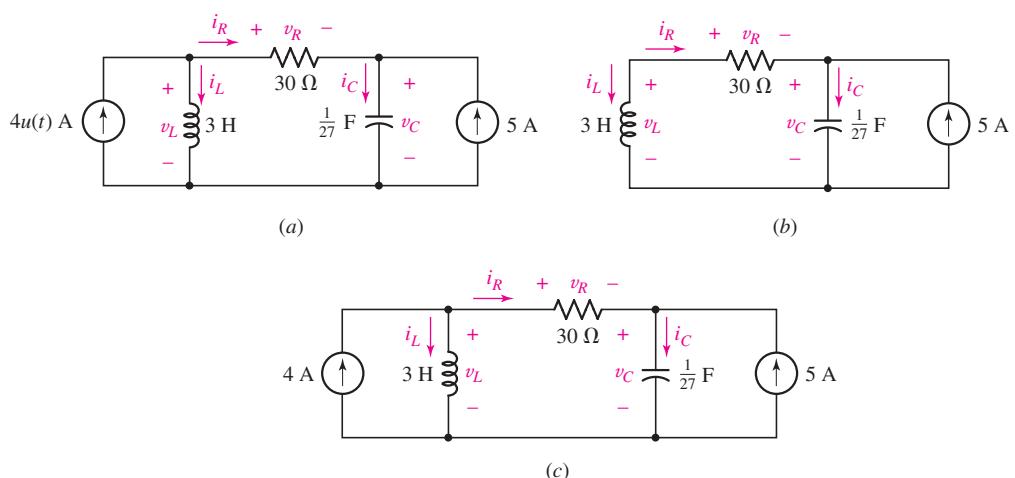
Another relationship between  $A$  and  $B$  is necessary, unfortunately, and this is normally obtained by taking the derivative of the response,

$$\frac{dv}{dt} = 0 + s_1 A e^{s_1 t} + s_2 B e^{s_2 t}$$

and inserting the known value of  $dv/dt$  at  $t = 0^+$ . We thus have two equations relating  $A$  and  $B$ , and these may be solved simultaneously to evaluate the two constants.

The only remaining problem is that of determining the values of  $v$  and  $dv/dt$  at  $t = 0^+$ . Let us suppose that  $v$  is a capacitor voltage,  $v_C$ . Since  $i_C = C dv_C/dt$ , we should recognize the relationship between the initial value of  $dv/dt$  and the initial value of some capacitor current. If we can establish a value for this initial capacitor current, then we will automatically establish the value of  $dv/dt$ . Students are usually able to get  $v(0^+)$  very easily, but are inclined to stumble a bit in finding the initial value of  $dv/dt$ . If we had selected an inductor current  $i_L$  as our response, then the initial value of  $di_L/dt$  would be intimately related to the initial value of some inductor voltage. Variables other than capacitor voltages and inductor currents are determined by expressing their initial values and the initial values of their derivatives in terms of the corresponding values for  $v_C$  and  $i_L$ .

We will illustrate the procedure and find all these values by the careful analysis of the circuit shown in Fig. 9.28. To simplify the analysis, an unusual value of capacitance is used again.



■ FIGURE 9.28 (a) An RLC circuit that is used to illustrate several procedures by which the initial conditions may be obtained. The desired response is nominally taken to be  $v_C(t)$ . (b)  $t = 0^-$ . (c)  $t > 0$ .

## EXAMPLE 9.9

**There are three passive elements in the circuit shown in Fig. 9.28a, and a voltage and a current are defined for each. Find the values of these six quantities at both  $t = 0^-$  and  $t = 0^+$ .**

Our object is to find the value of each current and voltage at both  $t = 0^-$  and  $t = 0^+$ . Once these quantities are known, the initial values of the derivatives may be found easily.

**1.  $t = 0^-$**  At  $t = 0^-$ , only the right-hand current source is active as depicted in Fig. 9.28b. The circuit is assumed to have been in this state forever, so all currents and voltages are constant. Thus, a dc current through the inductor requires zero voltage across it:

$$v_L(0^-) = 0$$

and a dc voltage across the capacitor ( $-v_R$ ) requires zero current through it:

$$i_C(0^-) = 0$$

We next apply Kirchhoff's current law to the right-hand node to obtain

$$i_R(0^-) = -5 \text{ A}$$

which also yields

$$v_R(0^-) = -150 \text{ V}$$

We may now use Kirchhoff's voltage law around the left-hand mesh, finding

$$v_C(0^-) = 150 \text{ V}$$

while KCL enables us to find the inductor current,

$$i_L(0^-) = 5 \text{ A}$$

**2.  $t = 0^+$**  During the interval from  $t = 0^-$  to  $t = 0^+$ , the left-hand current source becomes active and many of the voltage and current values at  $t = 0^-$  will change abruptly. The corresponding circuit is shown in Fig. 9.28c. However, we should begin by focusing our attention on those quantities which cannot change, namely, the inductor current and the capacitor voltage. Both of these must remain constant during the switching interval. Thus,

$$i_L(0^+) = 5 \text{ A} \quad \text{and} \quad v_C(0^+) = 150 \text{ V}$$

Since two currents are now known at the left node, we next obtain

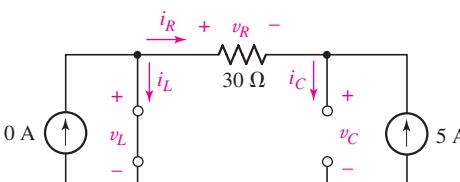
$$i_R(0^+) = -1 \text{ A} \quad \text{and} \quad v_R(0^+) = -30 \text{ V}$$

so that

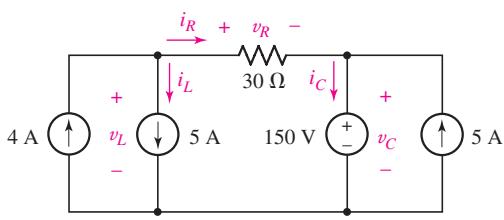
$$i_C(0^+) = 4 \text{ A} \quad \text{and} \quad v_L(0^+) = 120 \text{ V}$$

and we have our six initial values at  $t = 0^-$  and six more at  $t = 0^+$ . Among these last six values, only the capacitor voltage and the inductor current are unchanged from the  $t = 0^-$  values.

We could have employed a slightly different method to evaluate these currents and voltages at  $t = 0^-$  and  $t = 0^+$ . Prior to the switching operation, only direct currents and voltages exist in the circuit. The inductor may therefore be replaced by a short circuit, its dc equivalent, while the capacitor is replaced by an open circuit. Redrawn in this manner, the circuit of Fig. 9.28a appears as shown in Fig. 9.29a. Only the current source at the right is active, and its 5 A flow through the resistor and the inductor. We therefore have  $i_R(0^-) = -5$  A and  $v_R(0^-) = -150$  V,  $i_L(0^-) = 5$  A and  $v_L(0^-) = 0$ , and  $i_C(0^-) = 0$  and  $v_C(0^-) = 150$  V, as before.



(a)



(b)

■ FIGURE 9.29 (a) A simple circuit equivalent to the circuit of Fig. 9.28a for  $t = 0^-$ . (b) Equivalent circuit with labeled voltages and currents valid at the instant defined by  $t = 0^+$ .

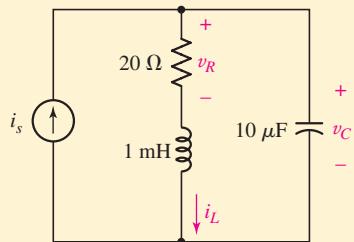
We now turn to the problem of drawing an equivalent circuit that will assist us in determining the several voltages and currents at  $t = 0^+$ . *Each capacitor voltage and each inductor current must remain constant during the switching interval.* These conditions are ensured by replacing the inductor with a current source and the capacitor with a voltage source. Each source serves to maintain a constant response during the discontinuity. The equivalent circuit of Fig. 9.29b results. It should be noted that the circuit shown in Fig. 9.29b is valid *only for the interval between  $0^-$  and  $0^+$* .

The voltages and currents at  $t = 0^+$  are obtained by analyzing this dc circuit. The solution is not difficult, but the relatively large number of sources present in the network does produce a somewhat strange sight. However, problems of this type were solved in Chap. 3, and nothing new is involved. Attacking the currents first, we begin at the upper left node and see that  $i_R(0^+) = 4 - 5 = -1$  A. Moving to the upper right node, we find that  $i_C(0^+) = -1 + 5 = 4$  A. And, of course,  $i_L(0^+) = 5$  A.

Next we consider the voltages. Using Ohm's law, we see that  $v_R(0^+) = 30(-1) = -30$  V. For the inductor, KVL gives us  $v_L(0^+) = -30 + 150 = 120$  V. Finally, including  $v_C(0^+) = 150$  V, we have all the values at  $t = 0^+$ .

**PRACTICE**

- 9.9 Let  $i_s = 10u(-t) - 20u(t)$  A in Fig. 9.30. Find (a)  $i_L(0^-)$ ; (b)  $v_C(0^+)$ ; (c)  $v_R(0^+)$ ; (d)  $i_L(\infty)$ ; (e)  $i_L(0.1$  ms).

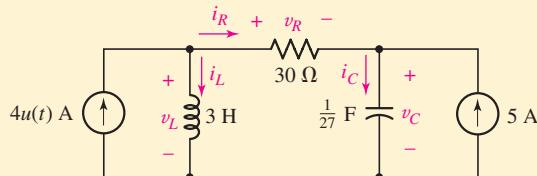


■ FIGURE 9.30

Ans: 10 A; 200 V; 200 V; -20 A; 2.07 A.

**EXAMPLE 9.10**

Complete the determination of the initial conditions in the circuit of Fig. 9.28, repeated in Fig. 9.31, by finding values at  $t = 0^+$  for the first derivatives of the three voltage and three current variables defined on the circuit diagram.



■ FIGURE 9.31 Circuit of Fig. 9.28, repeated for Example 9.10.

We begin with the two energy storage elements. For the inductor,

$$v_L = L \frac{di_L}{dt}$$

and, specifically,

$$v_L(0^+) = L \frac{di_L}{dt} \Big|_{t=0^+}$$

Thus,

$$\frac{di_L}{dt} \Big|_{t=0^+} = \frac{v_L(0^+)}{L} = \frac{120}{3} = 40 \text{ A/s}$$

Similarly,

$$\frac{dv_C}{dt} \Big|_{t=0^+} = \frac{i_C(0^+)}{C} = \frac{4}{1/27} = 108 \text{ V/s}$$

(Continued on next page)

The other four derivatives may be determined by realizing that KCL and KVL are both satisfied by the derivatives also. For example, at the left-hand node in Fig. 9.31,

$$4 - i_L - i_R = 0 \quad t > 0$$

and thus,

$$0 - \frac{di_L}{dt} - \frac{di_R}{dt} = 0 \quad t > 0$$

and therefore,

$$\left. \frac{di_R}{dt} \right|_{t=0^+} = -40 \text{ A/s}$$

The three remaining initial values of the derivatives are found to be

$$\left. \frac{dv_R}{dt} \right|_{t=0^+} = -1200 \text{ V/s}$$

$$\left. \frac{dv_L}{dt} \right|_{t=0^+} = -1092 \text{ V/s}$$

and

$$\left. \frac{di_C}{dt} \right|_{t=0^+} = -40 \text{ A/s}$$

Before leaving this problem of the determination of the necessary initial values, we should point out that at least one other powerful method of determining them has been omitted: we could have written general nodal or loop equations for the original circuit. Then the substitution of the known zero values of inductor voltage and capacitor current at  $t = 0^-$  would uncover several other response values at  $t = 0^-$  and enable the remainder to be found easily. A similar analysis at  $t = 0^+$  must then be made. This is an important method, and it becomes a necessary one in more complicated circuits which cannot be analyzed by our simpler step-by-step procedures.

Now let us briefly complete the determination of the response  $v_C(t)$  for the original circuit of Fig. 9.31. With both sources dead, the circuit appears as a series RLC circuit and  $s_1$  and  $s_2$  are easily found to be  $-1$  and  $-9$ , respectively. The forced response may be found by inspection or, if necessary, by drawing the dc equivalent, which is similar to Fig. 9.29a, with the addition of a 4 A current source. The forced response is 150 V. Thus,

$$v_C(t) = 150 + Ae^{-t} + Be^{-9t}$$

and

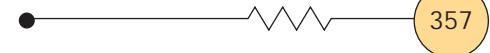
$$v_C(0^+) = 150 = 150 + A + B$$

or

$$A + B = 0$$

Then,

$$\frac{dv_C}{dt} = -Ae^{-t} - 9Be^{-9t}$$



and

$$\left. \frac{dv_C}{dt} \right|_{t=0^+} = 108 = -A - 9B$$

Finally,

$$A = 13.5 \quad B = -13.5$$

and

$$v_C(t) = 150 + 13.5(e^{-t} - e^{-9t}) \quad \text{V}$$

## A Quick Summary of the Solution Process

In summary, then, whenever we wish to determine the transient behavior of a simple three-element *RLC* circuit, we must first decide whether we are confronted with a series or a parallel circuit, so that we may use the correct relationship for  $\alpha$ . The two equations are

$$\alpha = \frac{1}{2RC} \quad (\text{parallel } RLC)$$

$$\alpha = \frac{R}{2L} \quad (\text{series } RLC)$$

Our second decision is made after comparing  $\alpha$  with  $\omega_0$ , which is given for either circuit by

$$\omega_0 = \frac{1}{\sqrt{LC}}$$

If  $\alpha > \omega_0$ , the circuit is *overdamped*, and the natural response has the form

$$f_n(t) = A_1 e^{s_1 t} + A_2 e^{s_2 t}$$

where

$$s_{1,2} = -\alpha \pm \sqrt{\alpha^2 - \omega_0^2}$$

If  $\alpha = \omega_0$ , then the circuit is *critically damped* and

$$f_n(t) = e^{-\alpha t}(A_1 t + A_2)$$

And finally, if  $\alpha < \omega_0$ , then we are faced with the *underdamped* response,

$$f_n(t) = e^{-\alpha t}(A_1 \cos \omega_d t + A_2 \sin \omega_d t)$$

where

$$\omega_d = \sqrt{\omega_0^2 - \alpha^2}$$

Our last decision depends on the independent sources. If there are none acting in the circuit after the switching or discontinuity is completed, then the circuit is source-free and the natural response accounts for the complete response. If independent sources are still present, then the circuit is driven and a forced response must be determined. The complete response is then the sum

$$f(t) = f_f(t) + f_n(t)$$

This is applicable to any current or voltage in the circuit. Our final step is to solve for unknown constants given the initial conditions.

## PRACTICAL APPLICATION

### Modeling Automotive Suspension Systems

Earlier, we alluded to the fact that the concepts investigated in this chapter actually extend beyond the analysis of electric circuits. In fact, the general form of the differential equations we have been working with appears in many fields—we need only learn how to “translate” new parameter terminology. For example, consider a simple automotive suspension, as shown in Fig. 9.32. The piston is not attached to the cylinder, but is attached to both the spring and the wheel. The moving parts therefore are the spring, the piston, and the wheel.

We will model this physical system by first determining the forces in play. Defining a position function  $p(t)$  which describes where the piston lies within the cylinder, we may write  $F_S$ , the force on the spring, as

$$F_S = Kp(t)$$

where  $K$  is known as the spring constant and has units of lb/ft. The force on the wheel  $F_W$  is equal to the mass of the wheel times its acceleration, or

$$F_W = m \frac{d^2 p(t)}{dt^2}$$

where  $m$  is measured in lb · s<sup>2</sup>/ft. Last but not least is the force of friction  $F_f$  acting on the piston

$$F_f = \mu_f \frac{dp(t)}{dt}$$



FIGURE 9.32 Typical automotive suspension system.  
© Transtock Inc./Alamy

where  $\mu_f$  is the coefficient of friction, with units of lb · s/ft.

From our basic physics courses we know that all forces acting in our system must sum to zero, so that

$$m \frac{d^2 p(t)}{dt^2} + \mu_f \frac{dp(t)}{dt} + Kp(t) = 0 \quad [34]$$

This equation most likely had the potential to give us nightmares at one point in our academic career, but no longer. We compare Eq. [32] to Eqs. [30] and [31] and immediately see a distinct resemblance, at least in the general form. Choosing Eq. [30], the differential equation describing the inductor current of a series-connected *RLC* circuit, we observe the following correspondences:

Mass	$m$	→	inductance	$L$
Coefficient of friction	$\mu_f$	→	resistance	$R$
Spring constant	$K$	→	inverse of the	$C^{-1}$
Position variable	$p(t)$	→	capacitance	$i(t)$

So, if we are willing to talk about feet instead of amperes, lb · s<sup>2</sup>/ft instead of H, ft/lb instead of F, and lb · s/ft instead of  $\Omega$ , we can apply our newly found skills at modeling *RLC* circuits to the task of evaluating automotive shock absorbers.

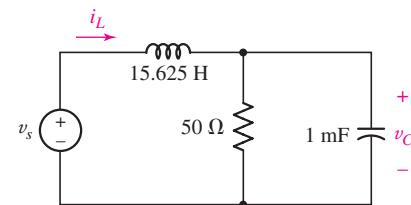
Take a typical car wheel of 70 lb. The mass is found by dividing the weight by the earth’s gravitational acceleration (32.17 ft/s<sup>2</sup>), resulting in  $m = 2.176$  lb · s<sup>2</sup>/ft. The curb weight of our car is 1985 lb, and the static displacement of the spring is 4 inches (no passengers). The spring constant is obtained by dividing the weight on each shock absorber by the static displacement, so that we have  $K = (\frac{1}{4})(1985)(3 \text{ ft}^{-1}) = 1489$  lb/ft. We are also told that the coefficient of friction for our piston-cylinder assembly is 65 lb · s/ft. Thus, we can simulate our shock absorber by modeling it with a series *RLC* circuit having  $R = 65 \Omega$ ,  $L = 2.176 \text{ H}$ , and  $C = K^{-1} = 671.6 \mu\text{F}$ .

The resonant frequency of our shock absorber is  $\omega_0 = (LC)^{-1/2} = 26.16$  rad/s, and the damping coefficient is  $\alpha = R/2L = 14.94 \text{ s}^{-1}$ . Since  $\alpha < \omega_0$ , our shock absorber represents an underdamped system; this means that we expect a bounce or two after we run over a pothole. A stiffer shock (larger coefficient of friction, or a larger resistance in our circuit model) is typically desirable when curves are taken at high speeds—at some point this corresponds to an overdamped response. However, if most of our driving is over unpaved roads, a slightly underdamped response is preferable.

**PRACTICE**

- 9.10 Let  $v_s = 10 + 20u(t)$  V in the circuit of Fig. 9.33. Find (a)  $i_L(0)$ ; (b)  $v_C(0)$ ; (c)  $i_{L,f}$ ; (d)  $i_L(0.1\text{ s})$ .

Ans: 0.2 A; 10 V; 0.6 A; 0.319 A.



■ FIGURE 9.33

## 9.7 THE LOSSLESS LC CIRCUIT

When we considered the source-free *RLC* circuit, it became apparent that the resistor served to dissipate any initial energy stored in the circuit. At some point it might occur to us to ask what would happen if we could remove the resistor. If the value of the resistance in a parallel *RLC* circuit becomes infinite, or zero in the case of a series *RLC* circuit, we have a simple *LC* loop in which an oscillatory response can be maintained forever. Let us look briefly at an example of such a circuit, and then discuss another means of obtaining an identical response without the need of supplying any inductance.

Consider the source-free circuit of Fig. 9.34, in which the large values  $L = 4\text{ H}$  and  $C = \frac{1}{36}\text{ F}$  are used so that the calculations will be simple. We let  $i(0) = -\frac{1}{6}\text{ A}$  and  $v(0) = 0$ . We find that  $\alpha = 0$  and  $\omega_0^2 = 9\text{ s}^{-2}$ , so that  $\omega_d = 3\text{ rad/s}$ . In the absence of exponential damping, the voltage  $v$  is simply

$$v = A \cos 3t + B \sin 3t$$

Since  $v(0) = 0$ , we see that  $A = 0$ . Next,

$$\left. \frac{dv}{dt} \right|_{t=0} = 3B = -\frac{i(0)}{1/36}$$

But  $i(0) = -\frac{1}{6}\text{ A}$ , and therefore  $dv/dt = 6\text{ V/s}$  at  $t = 0$ . We must have  $B = 2\text{ V}$  and so

$$v = 2 \sin 3t \quad \text{V}$$

which is an undamped sinusoidal response; in other words, our voltage response does not decay.

Now let us see how we might obtain this voltage without using an *LC* circuit. Our intentions are to write the differential equation that  $v$  satisfies and then to develop a configuration of op amps that will yield the solution of the equation. Although we are working with a specific example, the technique is a general one that can be used to solve any linear homogeneous differential equation.

For the *LC* circuit of Fig. 9.34, we select  $v$  as our variable and set the sum of the downward inductor and capacitor currents equal to zero:

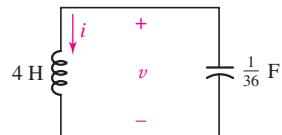
$$\frac{1}{4} \int_{t_0}^t v dt' - \frac{1}{6} + \frac{1}{36} \frac{dv}{dt} = 0$$

Differentiating once, we have

$$\frac{1}{4}v + \frac{1}{36} \frac{d^2v}{dt^2} = 0$$

or

$$\frac{d^2v}{dt^2} = -9v$$



■ FIGURE 9.34 This circuit is lossless, and it provides the undamped response  $v = 2 \sin 3t$  V, if  $v(0) = 0$  and  $i(0) = -\frac{1}{6}\text{ A}$ .

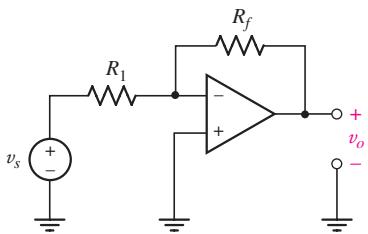


FIGURE 9.35 The inverting operational amplifier provides a gain  $v_o/v_s = -R_f/R_1$ , assuming an ideal op amp.

In order to solve this equation, we plan to make use of the operational amplifier as an integrator. We assume that the highest-order derivative appearing in the differential equation here,  $d^2v/dt^2$ , is available in our configuration of op amps at an arbitrary point A. We now make use of the integrator, with  $RC = 1$ , as discussed in Sec. 7.5. The input is  $d^2v/dt^2$ , and the output must be  $-dv/dt$ , where the sign change results from using an inverting op amp configuration for the integrator. The initial value of  $dv/dt$  is 6 V/s, as we showed when we first analyzed the circuit, and thus an initial value of  $-6$  V must be set in the integrator. The negative of the first derivative now forms the input to a second integrator. Its output is therefore  $v(t)$ , and the initial value is  $v(0) = 0$ . Now it only remains to multiply  $v$  by  $-9$  to obtain the second derivative we assumed at point A. This is amplification by 9 with a sign change, and it is easily accomplished by using the op amp as an inverting amplifier.

Figure 9.35 shows the circuit of an inverting amplifier. For an ideal op amp, both the input current and the input voltage are zero. Thus, the current going “east” through  $R_1$  is  $v_s/R_1$ , while that traveling west through  $R_f$  is  $v_o/R_f$ . Since their sum is zero, we have

$$\frac{v_o}{v_s} = -\frac{R_f}{R_1}$$

Thus, we can design for a gain of  $-9$  by setting  $R_f = 90 \text{ k}\Omega$  and  $R_1 = 10 \text{ k}\Omega$ , for example.

If we let  $R$  be  $1 \text{ M}\Omega$  and  $C$  be  $1 \mu\text{F}$  in each of the integrators, then

$$v_o = - \int_0^t v_s dt' + v_o(0)$$

in each case. The output of the inverting amplifier now forms the assumed input at point A, leading to the configuration of op amps shown in Fig. 9.36. If the left switch is closed at  $t = 0$  while the two initial-condition switches are opened at the same time, the output of the second integrator will be the undamped sine wave  $v = 2 \sin 3t$  V.

Note that both the LC circuit of Fig. 9.34 and the op amp circuit of Fig. 9.36 have the same output, but the op amp circuit does not contain a

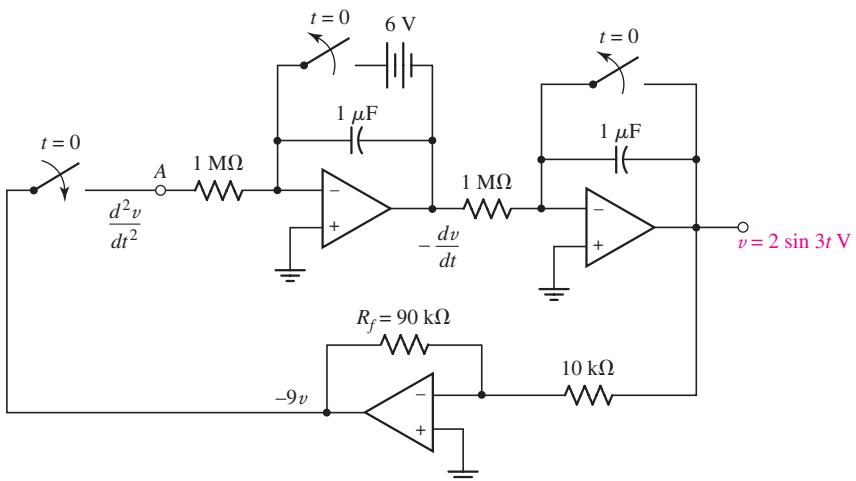


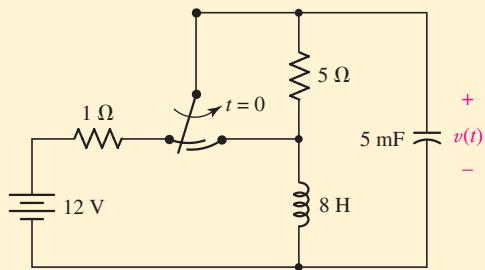
FIGURE 9.36 Two integrators and an inverting amplifier are connected to provide the solution of the differential equation  $d^2v/dt^2 = -9v$ .



single inductor. It simply *acts* as though it contained an inductor, providing the appropriate sinusoidal voltage between its output terminal and ground. This can be a considerable practical or economic advantage in circuit design, as inductors are typically bulky, more costly than capacitors, and have more losses associated with them (and therefore are not as well approximated by the “ideal” model).

### PRACTICE

- 9.11 Give new values for  $R_f$  and the two initial voltages in the circuit of Fig. 9.36 if the output represents the voltage  $v(t)$  in the circuit of Fig. 9.37.



■ FIGURE 9.37

Ans:  $250 \text{ k}\Omega$ ;  $400 \text{ V}$ ;  $10 \text{ V}$ .

### SUMMARY AND REVIEW

The simple *RL* and *RC* circuits examined in Chap. 8 essentially did one of two things as the result of throwing a switch: *charge* or *discharge*. Which one happened was determined by the initial charge state of the energy storage element. In this chapter, we considered circuits that had two energy storage elements (a capacitor and an inductor), and found that things could get pretty interesting. There are two basic configurations of such *RLC* circuits: *parallel* connected and *series* connected. Analysis of such a circuit yields a *second-order* partial differential equation, consistent with the number of distinct energy storage elements (if we construct a circuit using only resistors and capacitors such that the capacitors cannot be combined using series/parallel techniques, we also obtain—eventually—a second-order partial differential equation).

Depending on the value of the resistance connected to our energy storage elements, we found the transient response of an *RLC* circuit could be either *overdamped* (decaying exponentially) or *underdamped* (decaying, but oscillatory), with a “special case” of *critically damped* which is difficult to achieve in practice. Oscillations can be useful (for example, in transmitting information over a wireless network) and not so useful (for example, in accidental feedback situations between an amplifier and a microphone at a concert). Although the oscillations are not sustained in the circuits we examined, we have at least seen one way to create them at will,

and design for a specific frequency of operation if so desired. We didn't end up spending a great deal of time with the series connected *RLC* circuit because with the exception of  $\alpha$ , the equations are the same; only a minor adjustment in how we employ initial conditions to find the two unknown constants characterizing the transient response is needed. Along those lines, there were two "tricks," if you will, that we encountered. One is that to employ the second initial condition, we need to take the derivative of our response equation. The second is that whether we're employing KCL or KVL to make use of that initial condition, we're doing so at the instant that  $t = 0$ ; appreciating this fact can simplify equations dramatically by setting  $t = 0$  early.

We wrapped up the chapter by considering the *complete response*, and our approach to this did not differ significantly from what we did in Chap. 8. We closed with a brief section on a topic that might have occurred to us at some point—what happens when we remove the resistive losses completely (by setting parallel resistance to  $\infty$ , or series resistance to 0)? We end up with an *LC* circuit, and we saw that we can approximate such an animal with an op amp circuit.

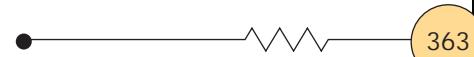
By now the reader is likely ready to finish reviewing key concepts of the chapter, so we'll stop here and list them, along with corresponding examples in the text.

- ❑ Circuits that contain two energy storage devices that cannot be combined using series-parallel combination techniques are described by a second-order differential equation.
- ❑ Series and parallel *RLC* circuits fall into one of three categories, depending on the relative values of  $R$ ,  $L$ , and  $C$ :

Overdamped	$\alpha > \omega_0$
Critically damped	$\alpha = \omega_0$
Underdamped	$\alpha < \omega_0$

(Example 9.1)

- ❑ For series *RLC* circuits,  $\alpha = R/2L$  and  $\omega_0 = 1/\sqrt{LC}$ . (Example 9.7)
- ❑ For parallel *RLC* circuits,  $\alpha = 1/2RC$  and  $\omega_0 = 1/\sqrt{LC}$ . (Example 9.1)
- ❑ The typical form of an overdamped response is the sum of two exponential terms, one of which decays more quickly than the other: e.g.,  $A_1e^{-t} + A_2e^{-6t}$ . (Examples 9.2, 9.3, 9.4)
- ❑ The typical form of a critically damped response is  $e^{-\alpha t}(A_1t + A_2)$ . (Example 9.5)
- ❑ The typical form of an underdamped response is an exponentially damped sinusoid:  $e^{-\alpha t}(B_1 \cos \omega_d t + B_2 \sin \omega_d t)$ . (Examples 9.6, 9.7, 9.8)
- ❑ During the transient response of an *RLC* circuit, energy is transferred between energy storage elements to the extent allowed by the resistive component of the circuit, which acts to dissipate the energy initially stored. (See Computer-Aided Analysis section.)
- ❑ The complete response is the sum of the forced and natural responses. In this case the total response must be determined before solving for the constants. (Examples 9.9, 9.10)



## READING FURTHER

An excellent discussion of employing PSpice in the modeling of automotive suspension systems can be found in

R.W. Goody, *MicroSim PSpice for Windows*, vol. I, 2nd ed. Englewood Cliffs, N.J.: Prentice-Hall, 1998.

Many detailed descriptions of analogous networks can be found in Chap. 3 of

E. Weber, *Linear Transient Analysis Volume I*. New York: Wiley, 1954.  
(Out of print, but in many university libraries.)

## EXERCISES

### 9.1 The Source-Free Parallel Circuit

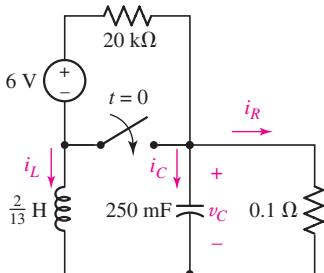
- For a certain source-free parallel *RLC* circuit,  $R = 1 \text{ k}\Omega$ ,  $C = 3 \mu\text{F}$ , and  $L$  is such that the circuit response is overdamped. (a) Determine the value of  $L$ .  
(b) Write the equation for the voltage  $v$  across the resistor if it is known that  $v(0^-) = 9 \text{ V}$  and  $dv/dt|_{t=0^+} = 2 \text{ V/s}$ .
- Element values of  $10 \text{ mF}$  and  $2 \text{ nH}$  are employed in the construction of a simple source-free parallel *RLC* circuit. (a) Select  $R$  so that the circuit is just barely overdamped. (b) Write the equation for the resistor current if its initial value is  $i_R(0^+) = 13 \text{ pA}$  and  $di_E/dt|_{t=0^+} = 1 \text{ nA/s}$ .
- If a parallel *RLC* circuit is constructed from component values  $C = 16 \text{ mF}$  and  $L = 1 \text{ mH}$ , choose  $R$  such that the circuit is (a) just barely overdamped; (b) just barely underdamped; (c) critically damped. (d) Does your answer for part (a) change if the resistor tolerance is 1%? 10%? (e) Increase the exponential damping coefficient for part (c) by 20%. Is the circuit now underdamped, over-damped, or still critically damped? *Explain*.
- Calculate  $\alpha$ ,  $\omega_0$ ,  $s_1$ , and  $s_2$  for a source-free parallel *RLC* circuit if (a)  $R = 4 \Omega$ ,  $L = 2.22 \text{ H}$ , and  $C = 12.5 \text{ mF}$ ; (b)  $L = 1 \text{ nH}$ ,  $C = 1 \text{ pF}$ , and  $R$  is 1% of the value required to make the circuit underdamped. (c) Calculate the damping ratio for the circuits of parts (a) and (b).
- You go to construct the circuit in Exercise 1, only to find no  $1 \text{ k}\Omega$  resistors. In fact, all you are able to locate in addition to the capacitor and inductor is a 1 meter long piece of 24 AWG soft solid copper wire. Connecting it in parallel to the two components you did find, compute the value of  $\alpha$ ,  $\omega_0$ ,  $s_1$ , and  $s_2$ , and verify that the circuit is still overdamped.
- Consider a source-free parallel *RLC* circuit having  $\alpha = 10^8 \text{ s}^{-1}$ ,  $\omega_0 = 10^3 \text{ rad/s}$ , and  $\omega_0 L = 5 \Omega$ . (a) Show that the stated units of  $\omega_0 L$  are correct. (b) Compute  $s_1$  and  $s_2$ . (c) Write the general form of the natural response for the capacitor voltage. (d) By appropriate substitution, verify that your answer to part (c) is indeed a solution to Eq. [1] if the inductor and capacitor each initially store 1 mJ of energy, respectively.
- A parallel *RLC* circuit is constructed with  $R = 500 \Omega$ ,  $C = 10 \mu\text{F}$ , and  $L$  such that it is critically damped. (a) Determine  $L$ . Is this value large or small for a printed-circuit board mounted component? (b) Add a resistor in parallel to the existing components such that the damping ratio is equal to 10. (c) Does increasing the damping ratio further lead to an overdamped, critically damped, or underdamped circuit? *Explain*.

### 9.2 The Overdamped Parallel *RLC* Circuit

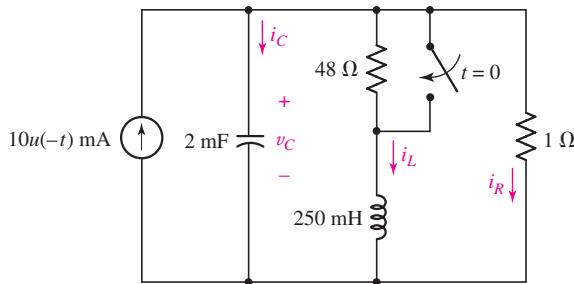
- The circuit of Fig. 9.2 is modified substantially, with the resistor being replaced with a  $1 \text{ k}\Omega$  resistor, the inductor swapped out for a smaller  $7 \text{ mH}$  version, the capacitor replaced with a  $1 \text{nF}$  alternative, and now the inductor is initially discharged while the capacitor is storing  $7.2 \text{ mJ}$ . (a) Compute  $\alpha$ ,  $\omega_0$ ,  $s_1$ , and  $s_2$ , and verify that the circuit is still overdamped. (b) Obtain an expression

for the current flowing through the resistor which is valid for  $t > 0$ . (c) Calculate the magnitude of the resistor current at  $t = 10 \mu\text{s}$ .

9. The voltage across a capacitor is found to be given by  $v_C(t) = 10e^{-10t} - 5e^{-4t} \text{ V}$ .
  - (a) Sketch each of the two components over the range of  $0 \leq t \leq 1.5 \text{ s}$ .
  - (b) Graph the capacitor voltage over the same time range.
10. The current flowing through a certain inductor is found to be given by  $i_L(t) = 0.20e^{-2t} - 0.6e^{-3t} \text{ A}$ .
  - (a) Sketch each of the two components over the range of  $0 \leq t \leq 1.5 \text{ s}$ .
  - (b) Graph the inductor current over the same time range.
  - (c) Graph the energy remaining in the inductor over  $0 \leq t \leq 1.5 \text{ s}$ .
11. The current flowing through a  $5 \Omega$  resistor in a source-free parallel RLC circuit is determined to be  $i_R(t) = 2e^{-t} - 3e^{-8t} \text{ A}$ ,  $t > 0$ . Determine (a) the maximum current and the time at which it occurs; (b) the settling time; (c) the time  $t$  corresponding to the resistor absorbing  $2.5 \text{ W}$  of power.
12. For the circuit of Fig. 9.38, obtain an expression for  $v_C(t)$  valid for all  $t > 0$ .
13. Consider the circuit depicted in Fig. 9.38. (a) Obtain an expression for  $i_L(t)$  valid for all  $t > 0$ . (b) Obtain an expression for  $i_R(t)$  valid for all  $t > 0$ . (c) Determine the settling time for both  $i_L$  and  $i_R$ .
14. With regard to the circuit represented in Fig. 9.39, determine (a)  $i_C(0^-)$ ; (b)  $i_L(0^-)$ ; (c)  $i_R(0^-)$ ; (d)  $v_C(0^-)$ ; (e)  $i_C(0^+)$ ; (f)  $i_L(0^+)$ ; (g)  $i_R(0^+)$ ; (h)  $v_C(0^+)$ .

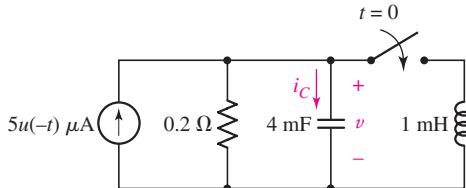


■ FIGURE 9.38



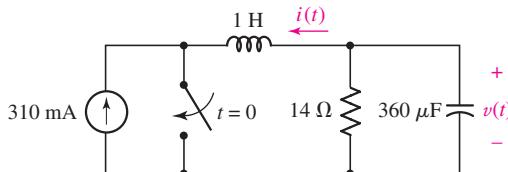
■ FIGURE 9.39

15. (a) Assuming the passive sign convention, obtain an expression for the voltage across the  $1 \Omega$  resistor in the circuit of Fig. 9.39 which is valid for all  $t > 0$ . (b) Determine the settling time of the resistor voltage.
16. With regard to the circuit presented in Fig. 9.40, (a) obtain an expression for  $v(t)$  which is valid for all  $t > 0$ ; (b) calculate the maximum inductor current and identify the time at which it occurs; (c) determine the settling time.



■ FIGURE 9.40

17. Obtain expressions for the current  $i(t)$  and voltage  $v(t)$  as labeled in the circuit of Fig. 9.41 which are valid for all  $t > 0$ .



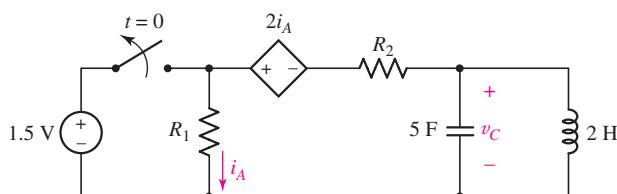
■ FIGURE 9.41



18. Replace the  $14\ \Omega$  resistor in the circuit of Fig. 9.41 with a  $1\ \Omega$  resistor.  
 (a) Obtain an expression for the energy stored in the capacitor as a function of time, valid for  $t > 0$ . (b) Determine the time at which the energy in the capacitor has been reduced to one-half its maximum value. (c) Verify your answer with an appropriate PSpice simulation.



19. Design a complete source-free parallel *RLC* circuit which exhibits an overdamped response, has a settling time of 1 s, and has a damping ratio of 15.
20. For the circuit represented by Fig. 9.42, the two resistor values are  $R_1 = 0.752\ \Omega$  and  $R_2 = 1.268\ \Omega$ , respectively. (a) Obtain an expression for the energy stored in the capacitor, valid for all  $t > 0$ ; (b) determine the settling time of the current labeled  $i_A$ .



■ FIGURE 9.42

### 9.3 Critical Damping

21. A motor coil having an inductance of  $8\text{ H}$  is in parallel with a  $2\ \mu\text{F}$  capacitor and a resistor of unknown value. The response of the parallel combination is determined to be critically damped. (a) Determine the value of the resistor.  
 (b) Compute  $\alpha$ . (c) Write the equation for the current flowing into the resistor if the top node is labeled  $v$ , the bottom node is grounded, and  $v = Ri_r$ . (d) Verify that your equation is a solution to the circuit differential equation,

$$\frac{di_r}{dt} + 2\alpha \frac{di_r}{dt} + \alpha^2 i_r = 0$$

22. The condition for critical damping in an *RLC* circuit is that the resonant frequency  $\omega_0$  and the exponential damping factor  $\alpha$  are equal. This leads to the relationship  $L = 4R^2C$ , which implies that  $1\text{ H} = 1\ \Omega^2 \cdot \text{F}$ . Verify this equivalence by breaking down each of the three units to fundamental SI units (see Chap. 2).
23. A critically damped parallel *RLC* circuit is constructed from component values  $40\ \Omega$ ,  $8\text{ nF}$ , and  $51.2\ \mu\text{H}$ , respectively. (a) Verify that the circuit is indeed critically damped. (b) Explain why, in practice, the circuit once fabricated is unlikely to be truly critically damped. (c) The inductor initially stores  $1\text{ mJ}$  of energy while the capacitor is initially discharged. Determine the magnitude of the capacitor voltage at  $t = 500\text{ ns}$ , the maximum absolute capacitor voltage, and the settling time.
24. Design a complete (i.e., with all necessary switches or step function sources) parallel *RLC* circuit which has a critically damped response such that the capacitor voltage at  $t = 1\text{ s}$  is equal to  $9\text{ V}$  and the circuit is source-free for all  $t > 0$ .
25. A critically damped parallel *RLC* circuit is constructed from component values  $40\ \Omega$  and  $2\text{ pF}$ . (a) Determine the value of  $L$ , taking care not to overround.  
 (b) Explain why, in practice, the circuit once fabricated is unlikely to be truly critically damped. (c) The inductor initially stores no energy while the capacitor is initially storing  $10\text{ pJ}$ . Determine the power absorbed by the resistor at  $t = 2\text{ ns}$ , the maximum absolute inductor current  $|i_L|$ , and the settling time.

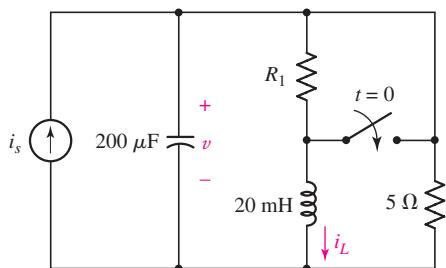


FIGURE 9.43

26. For the circuit of Fig. 9.43,  $i_s(t) = 30u(-t)$  mA. (a) Select  $R_1$  so that  $v(0^+) = 6$  V. (b) Compute  $v(2$  ms). (c) Determine the settling time of the capacitor voltage. (d) Is the inductor current settling time the same as your answer to part (c)?

27. The current source in Fig. 9.43 is  $i_s(t) = 10u(1 - t)$  μA. (a) Select  $R_1$  such that  $i_L(0^+) = 2$  μA. Compute  $i_L$  at  $t = 500$  ms and  $t = 1.002$  ms.

28. The inductor in the circuit of Fig. 9.41 is changed such that the circuit response is now critically damped. (a) Determine the new inductor value. (b) Calculate the energy stored in both the inductor and the capacitor at  $t = 10$  ms.

29. The circuit of Fig. 9.42 is rebuilt such that the quantity controlling the dependent source is now  $100i_A$ , a  $2\text{ }\mu\text{F}$  capacitor is used instead, and  $R_1 = R_2 = 10\text{ }\Omega$ . (a) Calculate the inductor value required to obtain a critically damped response. (b) Determine the power being absorbed by  $R_2$  at  $t = 300\text{ }\mu\text{s}$ .

#### 9.4 The Underdamped Parallel RLC Circuit

30. (a) With respect to the parallel RLC circuit, derive an expression for  $R$  in terms of  $C$  and  $L$  to ensure the response is underdamped. (b) If  $C = 1\text{ nF}$  and  $L = 10\text{ mH}$ , select  $R$  such that an underdamped response is (just barely) achieved. (c) If the damping ratio is increased, does the circuit become more or less underdamped? Explain. (d) Compute  $\alpha$  and  $\omega_d$  for the value of  $R$  you selected in part (b).

31. The circuit of Fig. 9.1 is constructed using component values  $10\text{ k}\Omega$ ,  $72\text{ }\mu\text{H}$ , and  $18\text{ pF}$ . (a) Compute  $\alpha$ ,  $\omega_d$ , and  $\omega_0$ . Is the circuit overdamped, critically damped, or underdamped? (b) Write the form of the natural capacitor voltage response  $v(t)$ . (c) If the capacitor initially stores  $1\text{ nJ}$  of energy, compute  $v$  at  $t = 300\text{ ns}$ .

32. The source-free circuit depicted in Fig. 9.1 is constructed using a  $10\text{ mH}$  inductor, a  $1\text{ mF}$  capacitor, and a  $1.5\text{ k}\Omega$  resistor. (a) Calculate  $\alpha$ ,  $\omega_d$ , and  $\omega_0$ . (b) Write the equation which describes the current  $i$  for  $t > 0$ . (c) Determine the maximum value of  $i$ , and the time at which it occurs, if the inductor initially stores no energy and  $v(0^-) = 9\text{ V}$ .

33. (a) Graph the current  $i$  for the circuit described in Exercise 32 for resistor values  $1.5\text{ k}\Omega$ ,  $15\text{ k}\Omega$ , and  $150\text{ k}\Omega$ . Make three separate graphs and be sure to extend the corresponding time axis to  $6\pi/\omega_d$  in each case. (b) Determine the corresponding settling times.



34. Analyze the circuit described in Exercise 32 to find  $v(t)$ ,  $t > 0$ , if  $R$  is equal to (a)  $2\text{ k}\Omega$ ; (b)  $2\text{ }\Omega$ . (c) Graph both responses over the range of  $0 \leq t \leq 60\text{ ms}$ . (d) Verify your answers with appropriate PSpice simulations.

35. For the circuit of Fig. 9.44, determine (a)  $i_C(0^-)$ ; (b)  $i_L(0^-)$ ; (c)  $i_R(0^-)$ ; (d)  $v_C(0^-)$ ; (e)  $i_C(0^+)$ ; (f)  $i_L(0^+)$ ; (g)  $i_R(0^+)$ ; (h)  $v_C(0^+)$ .

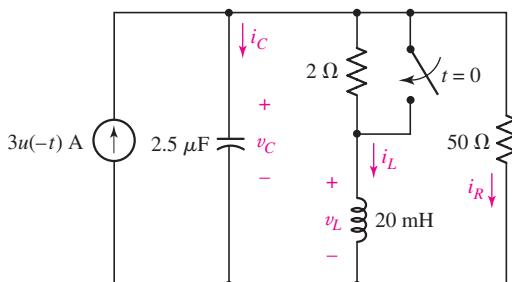


FIGURE 9.44

36. Obtain an expression for  $v_L(t)$ ,  $t > 0$ , for the circuit shown in Fig. 9.44. Plot the waveform for at least two periods of oscillation.

37. For the circuit of Fig. 9.45, determine (a) the first time  $t > 0$  when  $v(t) = 0$ ; (b) the settling time.

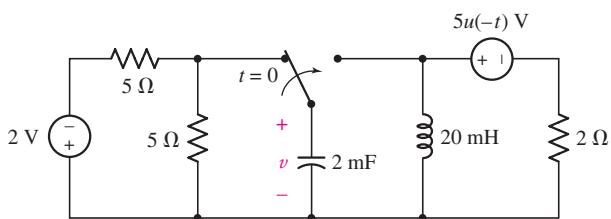


FIGURE 9.45

-  38. (a) Design a parallel RLC circuit that provides a capacitor voltage which oscillates with a frequency of 100 rad/s, with a maximum value of 10 V occurring at  $t = 0$ , and the second and third maxima both in excess of 6 V. (b) Verify your design with an appropriate PSpice simulation.

-  39. The circuit depicted in Fig. 9.46 is just barely underdamped. (a) Compute  $\alpha$  and  $\omega_d$ . (b) Obtain an expression for  $i_L(t)$  valid for  $t > 0$ . (c) Determine how much energy is stored in the capacitor, and in the inductor, at  $t = 200$  ms.

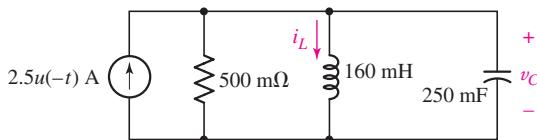


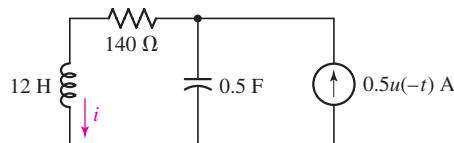
FIGURE 9.46

40. When constructing the circuit of Fig. 9.46, you inadvertently install a  $500 \text{ M}\Omega$  resistor by mistake. (a) Compute  $\alpha$  and  $\omega_d$ . (b) Obtain an expression for  $i_L(t)$  valid for  $t > 0$ . (c) Determine how long it takes for the energy stored in the inductor to reach 10% of its maximum value.

## 9.5 The Source-Free Series RLC Circuit

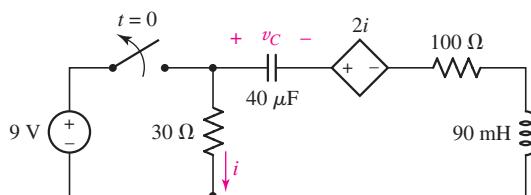
41. The circuit of Fig. 9.21a is constructed with a 160 mF capacitor and a 250 mH inductor. Determine the value of  $R$  needed to obtain (a) a critically damped response; (b) a “just barely” underdamped response. (c) Compare your answers to parts (a) and (b) if the circuit was a parallel RLC circuit.
42. Component values of  $R = 2 \Omega$ ,  $C = 1 \text{ mF}$ , and  $L = 2 \text{ mH}$  are used to construct the circuit represented in Fig. 9.21a. If  $v_C(0^-) = 1 \text{ V}$  and no current initially flows through the inductor, calculate  $i(t)$  at  $t = 1 \text{ ms}$ ,  $2 \text{ ms}$ , and  $3 \text{ ms}$ .
43. The series RLC circuit described in Exercise 42 is modified slightly by adding a  $2 \Omega$  resistor in parallel to the existing resistor. The initial capacitor voltage remains 1 V, and there is still no current flowing in the inductor prior to  $t = 0$ . (a) Calculate  $v_C(t)$  at 4 ms. (b) Sketch  $v_C(t)$  over the interval  $0 \leq t \leq 10 \text{ s}$ .
-  44. The simple three-element series RLC circuit of Exercise 42 is constructed with the same component values, but the initial capacitor voltage  $v_C(0^-) = 2 \text{ V}$  and the initial inductor current  $i(0^-) = 1 \text{ mA}$ . (a) Obtain an expression for  $i(t)$  valid for all  $t > 0$ . (b) Verify your solution with an appropriate simulation.
45. The series RLC circuit of Fig. 9.22 is constructed using  $R = 1 \text{ k}\Omega$ ,  $C = 2 \text{ mF}$ , and  $L = 1 \text{ mH}$ . The initial capacitor voltage  $v_C$  is  $-4 \text{ V}$  at  $t = 0^-$ . There is no current initially flowing through the inductor. (a) Obtain an expression for  $v_C(t)$  valid for  $t > 0$ . (b) Sketch over  $0 \leq t \leq 6 \mu\text{s}$ .

46. With reference to the circuit depicted in Fig. 9.47, calculate (a)  $\alpha$ ; (b)  $\omega_0$ ; (c)  $i(0^+)$ ; (d)  $di/dt|_{0^+}$ ; (e)  $i(t)$  at  $t = 6$  s.



■ FIGURE 9.47

47. Obtain an equation for  $v_C$  as labeled in the circuit of Fig. 9.48 valid for all  $t > 0$ .

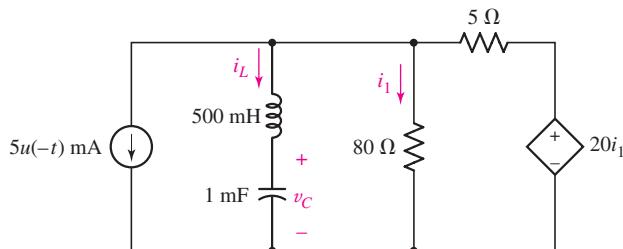


■ FIGURE 9.48



48. With reference to the series RLC circuit of Fig. 9.48, (a) obtain an expression for  $i$ , valid for  $t > 0$ ; (b) calculate  $i(0.8 \text{ ms})$  and  $i(4 \text{ ms})$ ; (c) verify your answers to part (b) with an appropriate PSpice simulation.

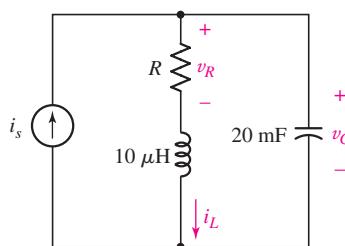
49. Obtain an expression for  $i_1$  as labeled in Fig. 9.49 which is valid for all  $t > 0$ .



■ FIGURE 9.49

## 9.6 The Complete Response of the RLC Circuit

50. In the series circuit of Fig. 9.50, set  $R = 1 \Omega$ . (a) Compute  $\alpha$  and  $\omega_0$ . (b) If  $i_s = 3u(-t) + 2u(t) \text{ mA}$ , determine  $v_R(0^-)$ ,  $i_L(0^-)$ ,  $v_C(0^-)$ ,  $v_R(0^+)$ ,  $i_L(0^+)$ ,  $v_C(0^+)$ ,  $i_L(\infty)$ , and  $v_C(\infty)$ .



■ FIGURE 9.50

51. Evaluate the derivative of each current and voltage variable labeled in Fig. 9.51 at  $t = 0^+$ .

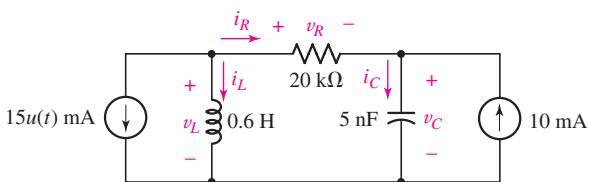


FIGURE 9.51

52. Consider the circuit depicted in Fig. 9.52. If  $v_s(t) = -8 + 2u(t)$  V, determine (a)  $v_C(0^+)$ ; (b)  $i_L(0^+)$ ; (c)  $v_C(\infty)$ ; (d)  $v_C(t = 150 \text{ ms})$ .
53. The  $15 \Omega$  resistor in the circuit of Fig. 9.52 is replaced with a  $500 \text{ m}\Omega$  alternative. If the source voltage is given by  $v_s = 1 - 2u(t)$  V, determine (a)  $i_L(0^+)$ ; (b)  $v_C(0^+)$ ; (c)  $i_L(\infty)$ ; (d)  $v_C(4 \text{ ms})$ .
54. In the circuit shown in Fig. 9.53, obtain an expression for  $i_L$  valid for all  $t > 0$  if  $i_1 = 8 - 10u(t)$  mA.
55. The  $10 \Omega$  resistor in the series RLC circuit of Fig. 9.53 is replaced with a  $1 \text{ k}\Omega$  resistor. The source voltage is given by  $v_s = 1 - 2u(t)$  V. Obtain an expression for  $i_L$  valid for all  $t > 0$ .
56. For the circuit represented in Fig. 9.54, (a) obtain an expression for  $v_C(t)$  valid for all  $t > 0$ . (b) Determine  $v_C$  at  $t = 10 \text{ ms}$  and  $t = 600 \text{ ms}$ . (c) Verify your answers to part (b) with an appropriate PSpice simulation.

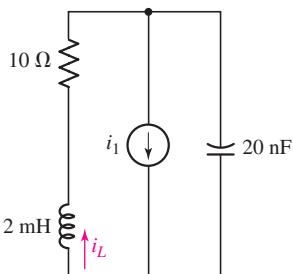


FIGURE 9.53

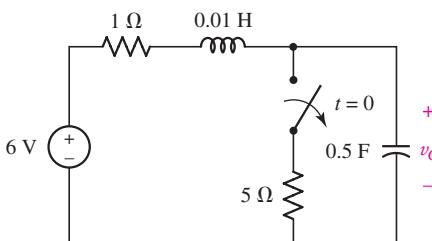


FIGURE 9.54

57. Replace the  $1 \Omega$  resistor in Fig. 9.54 with a  $100 \text{ m}\Omega$  resistor, and the  $5 \Omega$  resistor with a  $200 \text{ m}\Omega$  resistor. Assuming the passive sign convention, obtain an expression for the capacitor current which is valid for  $t > 0$ .
58. With regard to the circuit of Fig. 9.55, obtain an expression for  $v_C$  valid for  $t \geq 0$  if  $i_s(t) = 3u(-t) + 5u(t)$  mA.
59. (a) Adjust the value of the  $3 \Omega$  resistor in the circuit represented in Fig. 9.55 to obtain a “just barely” overdamped response. (b) Determine the first instant ( $t > 0$ ) at which an equal (and nonzero) amount of energy is stored in the capacitor and the inductor if  $i_s(t) = 2u(t)$  A. (c) Calculate the corresponding energy. (d) At what subsequent time will the energy stored in the inductor be twice that stored in the capacitor at the same instant?

## 9.7 The Lossless LC Circuit

60. Design an op amp circuit to model the voltage response of the LC circuit shown in Fig. 9.56. Verify your design by simulating both the circuit of Fig. 9.56 and your circuit using an LF 411 op amp, assuming  $v(0) = 0$  and  $i(0) = 1 \text{ mA}$ .

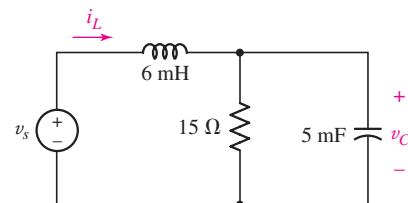


FIGURE 9.52

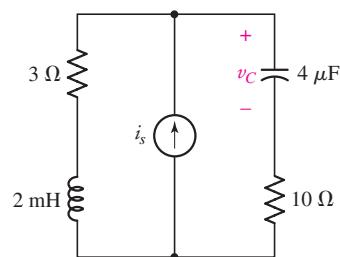


FIGURE 9.55

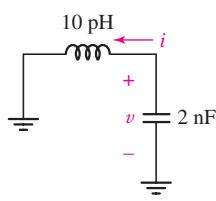


FIGURE 9.56

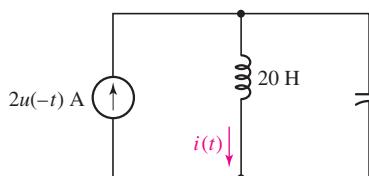


FIGURE 9.57



61. Refer to Fig. 9.57, and design an op amp circuit whose output will be  $i(t)$  for  $t > 0$ .
62. Replace the capacitor in the circuit of Fig. 9.56 with a 20 H inductor in parallel with a  $5 \mu\text{F}$  capacitor. Design an op amp circuit whose output will be  $i(t)$  for  $t > 0$ . Verify your design by simulating both the capacitor-inductor circuit and your op amp circuit. Use an LM111 op amp in the PSpice simulation.
63. A source-free  $RC$  circuit is constructed using a  $1 \text{k}\Omega$  resistor and a  $3.3 \text{ mF}$  capacitor. The initial voltage across the capacitor is  $1.2 \text{ V}$ . (a) Write the differential equation for  $v$ , the voltage across the capacitor, for  $t > 0$ . (b) Design an op amp circuit that provides  $v(t)$  as the output.
64. A source-free  $RL$  circuit contains a  $20 \Omega$  resistor and a  $5 \text{ H}$  inductor. If the initial value of the inductor current is  $2 \text{ A}$ : (a) write the differential equation for  $i$  for  $t > 0$ , and (b) design an op amp integrator to provide  $i(t)$  as the output, using  $R_1 = 1 \text{ M}\Omega$  and  $C_f = 1 \mu\text{F}$ .

### Chapter-Integrating Exercises

65. The capacitor in the circuit of Fig. 9.58 is set to  $1 \text{ F}$ . Determine  $v_C(t)$  at (a)  $t = -1 \text{ s}$ ; (b)  $t = 0^+$ ; (c)  $t = 20 \text{ s}$ .

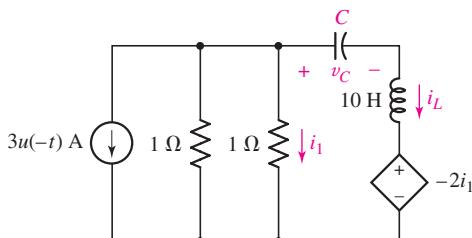


FIGURE 9.58

66. (a) What value of  $C$  for the circuit of Fig. 9.59 will result in an overdamped response? (b) Set  $C = 1 \text{ F}$  and obtain an expression for  $i_L(t)$  valid for  $t > 0$ .

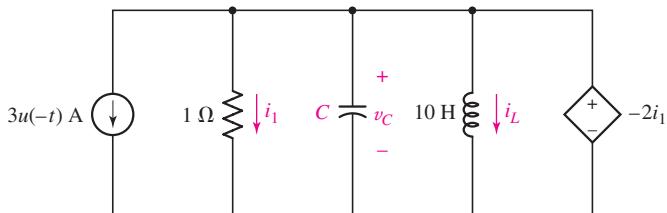


FIGURE 9.59



67. Obtain an expression for the current labeled  $i_1$  in the circuit of Fig. 9.58 which is valid for  $t > 0$ , if the current source is replaced with a source  $5u(t + 1) \text{ A}$ .
68. Design a parallel  $RLC$  circuit which produces an exponentially damped sinusoidal pulse with a peak voltage of  $1.5 \text{ V}$  and at least two additional peaks with voltage magnitude greater than  $0.8 \text{ V}$ . Verify your design with an appropriate PSpice simulation.
69. Design a series  $RLC$  circuit which produces an exponentially damped sinusoidal pulse with a peak voltage of  $1.5 \text{ V}$  and at least two additional peaks with voltage magnitude greater than  $0.8 \text{ V}$ . Verify your design with an appropriate PSpice simulation.

# Sinusoidal Steady-State Analysis

## INTRODUCTION

The complete response of a linear electric circuit is composed of two parts, the *natural* response and the *forced* response. The natural response is the short-lived transient response of a circuit to a sudden change in its condition. The forced response is the long-term steady-state response of a circuit to any independent sources present. Up to this point, the only forced response we have considered is that due to dc sources. Another very common forcing function is the sinusoidal waveform. This function describes the voltage available at household electrical sockets as well as the voltage of power lines connected to residential and industrial areas.

In this chapter, we assume that the transient response is of little interest, and the steady-state response of a circuit (a television set, a toaster, or a power distribution network) to a sinusoidal voltage or current is needed. We will analyze such circuits using a powerful technique that transforms integrodifferential equations into algebraic equations. Before we see how that works, it's useful to quickly review a few important attributes of general sinusoids, which will describe pretty much all currents and voltages throughout the chapter.

### 10.1 CHARACTERISTICS OF SINUSOIDS

Consider a sinusoidally varying voltage

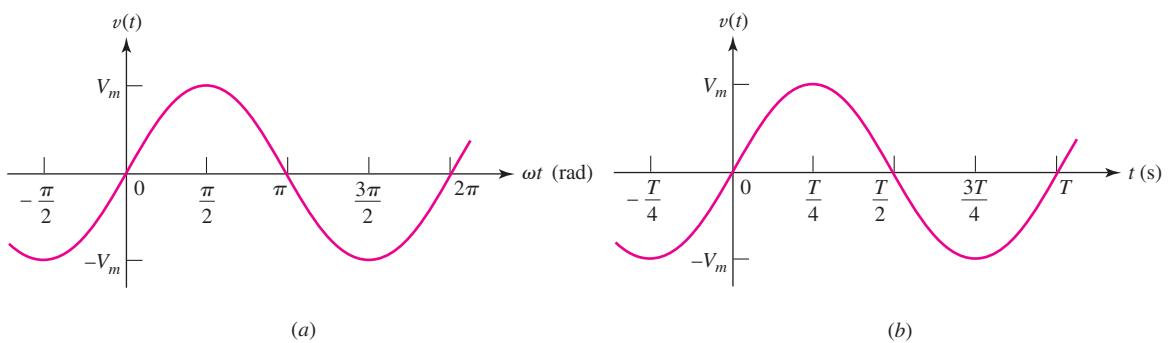
$$v(t) = V_m \sin \omega t$$

shown graphically in Figs. 10.1a and b. The *amplitude* of the sine wave is  $V_m$ , and the *argument* is  $\omega t$ . The *radian frequency*, or *angular frequency*, is  $\omega$ . In Fig. 10.1a,  $V_m \sin \omega t$  is plotted as a function of the argument  $\omega t$ , and the periodic nature of the sine wave is evident.

## KEY CONCEPTS

- Characteristics of Sinusoidal Functions
- Phasor Representation of Sinusoids
- Converting Between the Time and Frequency Domains
- Impedance and Admittance
- Reactance and Susceptance
- Parallel and Series Combinations in the Frequency Domain
- Determination of Forced Response Using Phasors
- Application of Circuit Analysis Techniques in the Frequency Domain





■ FIGURE 10.1 The sinusoidal function  $v(t) = V_m \sin \omega t$  is plotted (a) versus  $\omega t$  and (b) versus  $t$ .

The function repeats itself every  $2\pi$  radians, and its **period** is therefore  $2\pi$  radians. In Fig. 10.1b,  $V_m \sin \omega t$  is plotted as a function of  $t$  and the **period** is now  $T$ . A sine wave having a period  $T$  must execute  $1/T$  periods each second; its **frequency**  $f$  is  $1/T$  hertz, abbreviated Hz. Thus,

$$f = \frac{1}{T}$$

and since

$$\omega T = 2\pi$$

we obtain the common relationship between frequency and radian frequency,

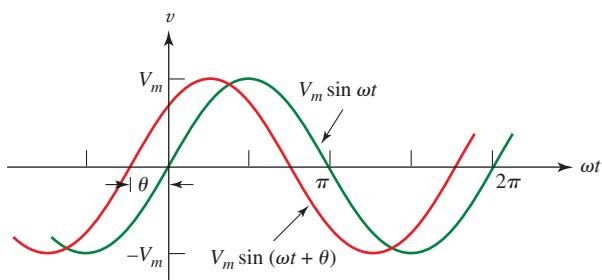
$$\boxed{\omega = 2\pi f}$$

## Lagging and Leading

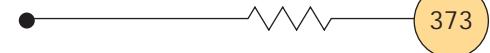
A more general form of the sinusoid,

$$v(t) = V_m \sin(\omega t + \theta) \quad [1]$$

includes a *phase angle*  $\theta$  in its argument. Equation [1] is plotted in Fig. 10.2 as a function of  $\omega t$ , and the phase angle appears as the number of radians by which the original sine wave (shown in green color in the sketch) is shifted to the left, or earlier in time. Since corresponding points on the sinusoid  $V_m \sin(\omega t + \theta)$  occur  $\theta$  rad, or  $\theta/\omega$  seconds, earlier, we say that  $V_m \sin(\omega t + \theta)$  *leads*  $V_m \sin \omega t$  by  $\theta$  rad. Therefore, it is correct to describe



■ FIGURE 10.2 The sine wave  $V_m \sin(\omega t + \theta)$  leads  $V_m \sin \omega t$  by  $\theta$  rad.



$\sin \omega t$  as **lagging**  $\sin(\omega t + \theta)$  by  $\theta$  rad, as **leading**  $\sin(\omega t + \theta)$  by  $-\theta$  rad, or as leading  $\sin(\omega t - \theta)$  by  $\theta$  rad.

In either case, leading or lagging, we say that the sinusoids are *out of phase*. If the phase angles are equal, the sinusoids are said to be *in phase*.

In electrical engineering, the phase angle is commonly given in degrees, rather than radians; to avoid confusion we should be sure to always use the degree symbol. Thus, instead of writing

$$v = 100 \sin\left(2\pi 1000t - \frac{\pi}{6}\right)$$

we customarily use

$$v = 100 \sin(2\pi 1000t - 30^\circ)$$

In evaluating this expression at a specific instant of time, e.g.,  $t = 10^{-4}$  s,  $2\pi 1000t$  becomes  $0.2\pi$  radian, and this should be expressed as  $36^\circ$  before  $30^\circ$  is subtracted from it. Don't confuse your apples with your oranges.

Recall that to convert radians to degrees, we simply multiply the angle by  $180/\pi$ .



*Two sinusoidal waves whose phases are to be compared must:*

1. Both be written as sine waves, or both as cosine waves.
2. Both be written with positive amplitudes.
3. Each have the same frequency.

## Converting Sines to Cosines

The sine and cosine are essentially the same function, but with a  $90^\circ$  phase difference. Thus,  $\sin \omega t = \cos(\omega t - 90^\circ)$ . Multiples of  $360^\circ$  may be added to or subtracted from the argument of any sinusoidal function without changing the value of the function. Hence, we may say that

$$\begin{aligned} v_1 &= V_{m_1} \cos(5t + 10^\circ) \\ &= V_{m_1} \sin(5t + 90^\circ + 10^\circ) \\ &= V_{m_1} \sin(5t + 100^\circ) \end{aligned}$$

leads

$$v_2 = V_{m_2} \sin(5t - 30^\circ)$$

by  $130^\circ$ . It is also correct to say that  $v_1$  lags  $v_2$  by  $230^\circ$ , since  $v_1$  may be written as

$$v_1 = V_{m_1} \sin(5t - 260^\circ)$$

We assume that  $V_{m_1}$  and  $V_{m_2}$  are both positive quantities. A graphical representation is provided in Fig. 10.3; note that the frequency of both sinusoids (5 rad/s in this case) must be the same, or the comparison is meaningless. Normally, the difference in phase between two sinusoids is expressed by that angle which is less than or equal to  $180^\circ$  in magnitude.

The concept of a leading or lagging relationship between two sinusoids will be used extensively, and the relationship is recognizable both mathematically and graphically.

Note that:

$$\begin{aligned} -\sin \omega t &= \sin(\omega t \pm 180^\circ) \\ -\cos \omega t &= \cos(\omega t \pm 180^\circ) \\ \mp \sin \omega t &= \cos(\omega t \pm 90^\circ) \\ \pm \cos \omega t &= \sin(\omega t \pm 90^\circ) \end{aligned}$$

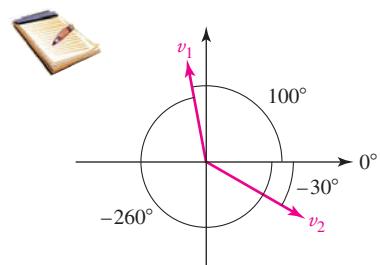


FIGURE 10.3 A graphical representation of the two sinusoids  $v_1$  and  $v_2$ . The magnitude of each sine function is represented by the length of the corresponding arrow, and the phase angle by the orientation with respect to the positive x axis. In this diagram,  $v_1$  leads  $v_2$  by  $100^\circ + 30^\circ = 130^\circ$ , although it could also be argued that  $v_2$  leads  $v_1$  by  $230^\circ$ . It is customary, however, to express the phase difference by an angle less than or equal to  $180^\circ$  in magnitude.

**PRACTICE**

10.1 Find the angle by which  $i_1$  lags  $v_1$  if  $v_1 = 120 \cos(120\pi t - 40^\circ)$  V and  $i_1$  equals (a)  $2.5 \cos(120\pi t + 20^\circ)$  A; (b)  $1.4 \sin(120\pi t - 70^\circ)$  A; (c)  $-0.8 \cos(120\pi t - 110^\circ)$  A.

10.2 Find  $A$ ,  $B$ ,  $C$ , and  $\phi$  if  $40 \cos(100t - 40^\circ) - 20 \sin(100t + 170^\circ) = A \cos 100t + B \sin 100t = C \cos(100t + \phi)$ .

Ans: 10.1:  $-60^\circ$ ;  $120^\circ$ ;  $-110^\circ$ . 10.2: 27.2; 45.4; 52.9;  $-59.1^\circ$ .

## **10.2 FORCED RESPONSE TO SINUSOIDAL FUNCTIONS**

Now that we are familiar with the mathematical characteristics of sinusoids, we are ready to apply a sinusoidal forcing function to a simple circuit and obtain the forced response. We will first write the differential equation that applies to the given circuit. The complete solution of this equation is composed of two parts, the complementary solution (which we call the *natural response*) and the particular integral (or *forced response*). The methods we plan to develop in this chapter assume that we are not interested in the short-lived transient or natural response of our circuit, but only in the long-term or “steady-state” response.

### **The Steady-State Response**

The term *steady-state response* is used synonymously with *forced response*, and the circuits we are about to analyze are commonly said to be in the “sinusoidal steady state.” Unfortunately, *steady state* carries the connotation of “not changing with time” in the minds of many students. This is true for dc forcing functions, but the sinusoidal steady-state response is definitely changing with time. The steady state simply refers to the condition that is reached after the transient or natural response has died out.

The forced response has the mathematical form of the forcing function, plus all its derivatives and its first integral. With this knowledge, one of the methods by which the forced response may be found is to assume a solution composed of a sum of such functions, where each function has an unknown amplitude to be determined by direct substitution in the differential equation. As we are about to see, this can be a lengthy process, so we will be sufficiently motivated to seek out a simpler alternative.

Consider the series  $RL$  circuit shown in Fig. 10.4. The sinusoidal source voltage  $v_s = V_m \cos \omega t$  has been switched into the circuit at some remote time in the past, and the natural response has died out completely. We seek the forced (or “steady-state”) response, which must satisfy the differential equation

$$L \frac{di}{dt} + Ri = V_m \cos \omega t$$

obtained by applying KVL around the simple loop. At any instant where the derivative is equal to zero, we see that the current must have the form  $i \propto \cos \omega t$ . Similarly, at an instant where the current is equal to zero, the

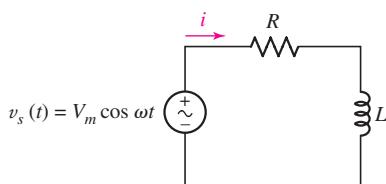
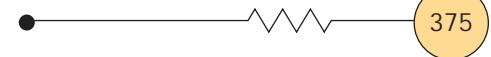


FIGURE 10.4 A series  $RL$  circuit for which the forced response is desired.



derivative must be proportional to  $\cos \omega t$ , implying a current of the form  $\sin \omega t$ . We might expect, therefore, that the forced response will have the general form

$$i(t) = I_1 \cos \omega t + I_2 \sin \omega t$$

where  $I_1$  and  $I_2$  are real constants whose values depend upon  $V_m$ ,  $R$ ,  $L$ , and  $\omega$ . No constant or exponential function can be present. Substituting the assumed form for the solution in the differential equation yields

$$L(-I_1 \omega \sin \omega t + I_2 \omega \cos \omega t) + R(I_1 \cos \omega t + I_2 \sin \omega t) = V_m \cos \omega t$$

If we collect the cosine and sine terms, we obtain

$$(-LI_1 \omega + RI_2) \sin \omega t + (LI_2 \omega + RI_1 - V_m) \cos \omega t = 0$$

This equation must be true for all values of  $t$ , which can be achieved only if the factors multiplying  $\cos \omega t$  and  $\sin \omega t$  are each zero. Thus,

$$-\omega LI_1 + RI_2 = 0 \quad \text{and} \quad \omega LI_2 + RI_1 - V_m = 0$$

and simultaneous solution for  $I_1$  and  $I_2$  leads to

$$I_1 = \frac{RV_m}{R^2 + \omega^2 L^2} \quad I_2 = \frac{\omega LV_m}{R^2 + \omega^2 L^2}$$

Thus, the forced response is obtained:

$$i(t) = \frac{RV_m}{R^2 + \omega^2 L^2} \cos \omega t + \frac{\omega LV_m}{R^2 + \omega^2 L^2} \sin \omega t \quad [2]$$

## A More Compact and User-Friendly Form

Although accurate, this expression is slightly cumbersome; a clearer picture of the response can be obtained by expressing it as a single sinusoid or cosinusoid with a phase angle. We choose to express the response as a cosine function,

$$i(t) = A \cos(\omega t - \theta) \quad [3]$$

At least two methods of obtaining the values of  $A$  and  $\theta$  suggest themselves. We might substitute Eq. [3] directly in the original differential equation, or we could simply equate the two solutions, Eqs. [2] and [3]. Selecting the latter method, and expanding the function  $\cos(\omega t - \theta)$ :

$$A \cos \theta \cos \omega t + A \sin \theta \sin \omega t = \frac{RV_m}{R^2 + \omega^2 L^2} \cos \omega t + \frac{\omega LV_m}{R^2 + \omega^2 L^2} \sin \omega t$$

All that remains is to collect terms and perform a bit of algebra, an exercise left to the reader. The result is

$$\theta = \tan^{-1} \frac{\omega L}{R}$$

and

$$A = \frac{V_m}{\sqrt{R^2 + \omega^2 L^2}}$$

and so the *alternative form* of the forced response therefore becomes

$$i(t) = \frac{V_m}{\sqrt{R^2 + \omega^2 L^2}} \cos \left( \omega t - \tan^{-1} \frac{\omega L}{R} \right) \quad [4]$$

Several useful trigonometric identities are provided on the inside cover of the book.

Once upon a time, the symbol  $E$  (for electromotive force) was used to designate voltages. Then every student learned the phase “ELI the ICE man” as a reminder that *voltage leads current* in an *inductive* circuit, while *current leads voltage* in a *capacitive* circuit. Now that we use  $V$  instead, it just isn’t the same.

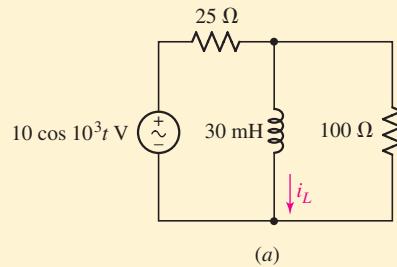


With this form, it is easy to see that the amplitude of the *response* is proportional to the amplitude of the *forcing function*; if not, the linearity concept would have to be discarded. The current is seen to lag the applied voltage by  $\tan^{-1}(\omega L/R)$ , an angle between 0 and 90°. When  $\omega = 0$  or  $L = 0$ , the current must be in phase with the voltage; since the former situation is direct current and the latter provides a resistive circuit, the result agrees with our previous experience. If  $R = 0$ , the current lags the voltage by 90°. In an inductor, then, if the passive sign convention is satisfied, the current lags the voltage by exactly 90°. In a similar manner we can show that the current through a capacitor *leads* the voltage across it by 90°.

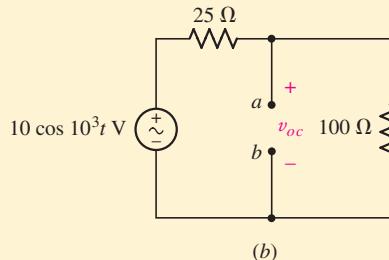
The phase difference between the current and voltage depends upon the ratio of the quantity  $\omega L$  to  $R$ . We call  $\omega L$  the *inductive reactance* of the inductor; it is measured in ohms, and it is a measure of the opposition that is offered by the inductor to the passage of a sinusoidal current.

### EXAMPLE 10.1

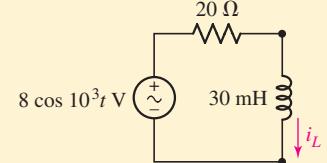
Find the current  $i_L$  in the circuit shown in Fig. 10.5a, if the transients have already died out.



(a)



(b)



(c)

FIGURE 10.5 (a) The circuit for Example 10.1, in which the current  $i_L$  is desired. (b) The Thévenin equivalent is desired at terminals  $a$  and  $b$ . (c) The simplified circuit.

Although this circuit has a sinusoidal source and a single inductor, it contains two resistors and is not a single loop. In order to apply the results of the preceding analysis, we need to seek the Thévenin equivalent as viewed from terminals  $a$  and  $b$  in Fig. 10.5b.

The open-circuit voltage  $v_{oc}$  is

$$v_{oc} = (10 \cos 10^3 t) \frac{100}{100 + 25} = 8 \cos 10^3 t \quad \text{V}$$

Since there are no dependent sources in sight, we find  $R_{th}$  by shorting out the independent source and calculating the resistance of the passive network, so  $R_{th} = (25 \times 100)/(25 + 100) = 20 \Omega$ .

Now we do have a series  $RL$  circuit, with  $L = 30 \text{ mH}$ ,  $R_{th} = 20 \Omega$ , and a source voltage of  $8 \cos 10^3 t$  V, as shown in Fig. 10.5c. Thus, applying Eq. [4], which was derived for a general  $RL$  series circuit,

$$i_L = \frac{8}{\sqrt{20^2 + (10^3 \times 30 \times 10^{-3})^2}} \cos \left( 10^3 t - \tan^{-1} \frac{30}{20} \right)$$

$$= 222 \cos(10^3 t - 56.3^\circ) \quad \text{mA}$$

The voltage and current waveforms are plotted in Fig. 10.6.

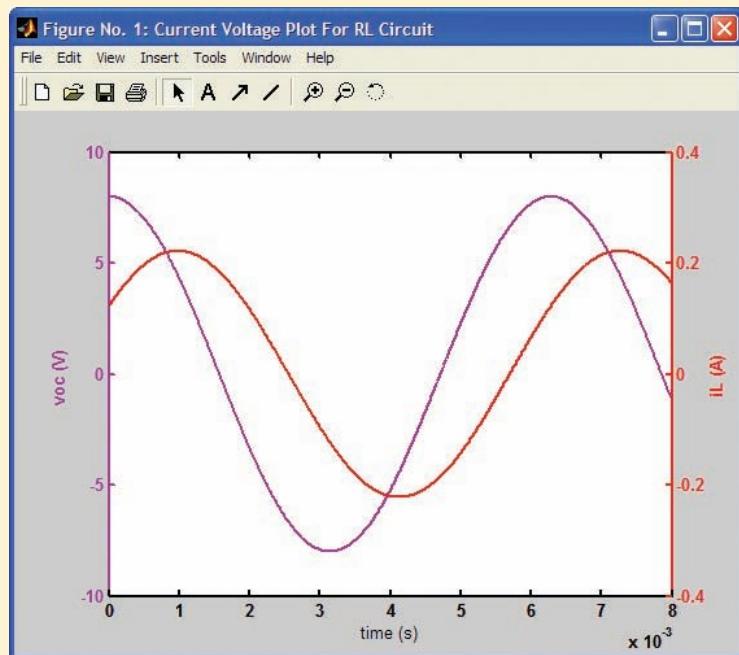


FIGURE 10.6 Voltage and current waveforms on a dual axis plot, generated using MATLAB:

```
EDU» t = linspace(0,8e-3,1000);
EDU» v = 8*cos(1000*t);
EDU» i = 0.222*cos(1000*t - 56.3*pi/180);
EDU» plotyy(t,v,t,i);
EDU» xlabel('time (s)');
```

Note that there is not a  $90^\circ$  phase difference between the current and voltage waveforms of the plot. This is because we are not plotting the inductor voltage, which is left as an exercise for the reader.

### PRACTICE

- 10.3 Let  $v_s = 40 \cos 8000t$  V in the circuit of Fig. 10.7. Use Thévenin's theorem where it will do the most good, and find the value at  $t = 0$  for (a)  $i_L$ ; (b)  $v_L$ ; (c)  $i_R$ ; (d)  $i_s$ .

Ans: 18.71 mA; 15.97 V; 5.32 mA; 24.0 mA.

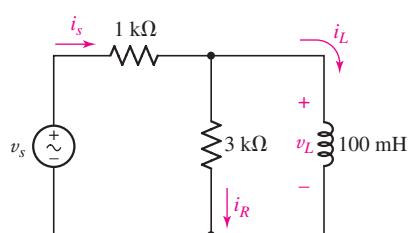


FIGURE 10.7

### 10.3 THE COMPLEX FORCING FUNCTION

The method we just employed works—the correct answer is obtained in a straightforward manner. However, it isn't particularly graceful, and after being applied to a few circuits, it remains as clunky and cumbersome as the first time we use it. The real problem isn't the time-varying source—it's the inductor (or capacitor), since a purely resistive circuit is no more difficult to analyze with sinusoidal sources than with dc sources, as only algebraic equations result. It turns out that if the *transient* response is of no interest to us, there is an alternative approach for obtaining the sinusoidal steady-state response of any linear circuit. The distinct advantage of this alternative is that it allows us to relate the current and voltage associated with any element using a simple algebraic expression.

The basic idea is that sinusoids and exponentials are related through complex numbers. Euler's identity, for example, tells us that

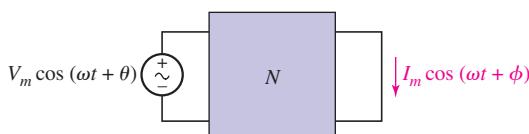
$$e^{j\theta} = \cos \theta + j \sin \theta$$

Whereas taking the derivative of a cosine function yields a (negative) sine function, the derivative of an exponential is simply a scaled version of the same exponential. If at this point the reader is thinking, “All this is great, but there are no imaginary numbers in any circuit I ever plan to build!” that may be true. What we're about to see, however, is that adding imaginary sources to our circuits leads to complex sources which (surprisingly) simplify the analysis process. It might seem like a strange idea at first, but a moment's reflection should remind us that superposition requires any *imaginary* sources we might add to cause only *imaginary* responses, and real sources can only lead to real responses. Thus, at any point, we should be able to separate the two by simply taking the real part of any complex voltage or current.

In Fig. 10.8, a sinusoidal source

$$V_m \cos(\omega t + \theta) \quad [5]$$

is connected to a general network, which we will assume to contain only passive elements (i.e., no independent sources) in order to keep things simple. A current response in some other branch of the network is to be determined, and the parameters appearing in Eq. [5] are all real quantities.



■ FIGURE 10.8 The sinusoidal forcing function  $V_m \cos(\omega t + \theta)$  produces the steady-state sinusoidal response  $I_m \cos(\omega t + \phi)$ .

We have shown that we may represent the response by the general cosine function

$$I_m \cos(\omega t + \phi) \quad [6]$$

A sinusoidal forcing function always produces a sinusoidal forced response of the same frequency in a linear circuit.



Now let us change our time reference by shifting the phase of the forcing function by  $90^\circ$ , or changing the instant that we call  $t = 0$ . Thus, the forcing function

$$V_m \cos(\omega t + \theta - 90^\circ) = V_m \sin(\omega t + \theta) \quad [7]$$

when applied to the same network will produce a corresponding response

$$I_m \cos(\omega t + \phi - 90^\circ) = I_m \sin(\omega t + \phi) \quad [8]$$

We next depart from physical reality by applying an imaginary forcing function, one that cannot be applied in the laboratory but can be applied mathematically.

## Imaginary Sources Lead to . . . Imaginary Responses

We construct an imaginary source very simply; it is only necessary to multiply Eq. [7] by  $j$ , the imaginary operator. We thus apply

$$jV_m \sin(\omega t + \theta) \quad [9]$$

Electrical engineers use " $j$ " instead of " $i$ " to represent  $\sqrt{-1}$  to avoid confusion with currents.

What is the response? If we had doubled the source, then the principle of linearity would require that we double the response; multiplication of the forcing function by a constant  $k$  would result in the multiplication of the response by the same constant  $k$ . The fact that our constant is  $\sqrt{-1}$  does not destroy this relationship. The response to the imaginary source of Eq. [9] is thus

$$jI_m \sin(\omega t + \phi) \quad [10]$$

The imaginary source and response are indicated in Fig. 10.9.

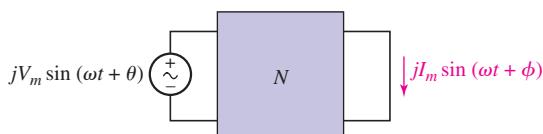


FIGURE 10.9 The imaginary sinusoidal forcing function  $jV_m \sin(\omega t + \theta)$  produces the imaginary sinusoidal response  $jI_m \sin(\omega t + \phi)$  in the network of Fig. 10.8.

## Applying a Complex Forcing Function

We have applied a *real source* and obtained a *real response*; we have also applied an *imaginary source* and obtained an *imaginary response*. Since we are dealing with a *linear circuit*, we may use the superposition theorem to find the response to a complex forcing function which is the sum of the real and imaginary forcing functions. Thus, the sum of the forcing functions of Eqs. [5] and [9],

$$V_m \cos(\omega t + \theta) + jV_m \sin(\omega t + \theta) \quad [11]$$

must produce a response that is the sum of Eqs. [6] and [10],

$$I_m \cos(\omega t + \phi) + jI_m \sin(\omega t + \phi) \quad [12]$$

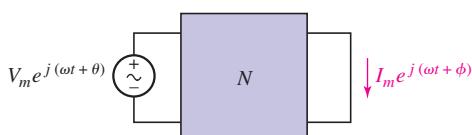
The complex source and response may be represented more simply by applying Euler's identity, i.e.,  $\cos(\omega t + \theta) + j \sin(\omega t + \theta) = e^{j(\omega t + \theta)}$ . Thus, the source of Eq. [11] may be written as

$$V_m e^{j(\omega t + \theta)} \quad [13]$$

and the response of Eq. [12] is

$$I_m e^{j(\omega t + \phi)} \quad [14]$$

The complex source and response are illustrated in Fig. 10.10.



■ FIGURE 10.10 The complex forcing function  $V_m e^{j(\omega t + \theta)}$  produces the complex response  $I_m e^{j(\omega t + \phi)}$  in the network of Fig. 10.8.

Again, linearity assures us that the *real* part of the complex response is produced by the *real* part of the complex forcing function, while the *imaginary* part of the response is caused by the *imaginary* part of the complex forcing function. Our plan is that instead of applying a *real* forcing function to obtain the desired real response, we will substitute a *complex* forcing function whose real part is the given real forcing function; we expect to obtain a complex response whose real part is the desired real response. The advantage of this procedure is that the integrodifferential equations describing the steady-state response of a circuit will now become simple algebraic equations.

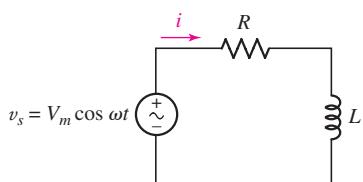
## An Algebraic Alternative to Differential Equations

Let's try out this idea on the simple *RL* series circuit shown in Fig. 10.11. The real source  $V_m \cos \omega t$  is applied; the real response  $i(t)$  is desired. Since

$$V_m \cos \omega t = \operatorname{Re}\{V_m \cos \omega t + j V_m \sin \omega t\} = \operatorname{Re}\{V_m e^{j\omega t}\}$$

the necessary complex source is

$$V_m e^{j\omega t}$$



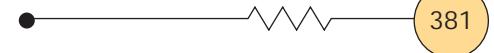
■ FIGURE 10.11 A simple circuit in the sinusoidal steady state is to be analyzed by the application of a complex forcing function.

We express the complex response that results in terms of an unknown amplitude  $I_m$  and an unknown phase angle  $\phi$ :

$$I_m e^{j(\omega t + \phi)}$$

Writing the differential equation for this particular circuit,

$$Ri + L \frac{di}{dt} = v_s$$



we insert our complex expressions for  $v_s$  and  $i$ :

$$RI_m e^{j(\omega t + \phi)} + L \frac{d}{dt} (I_m e^{j(\omega t + \phi)}) = V_m e^{j\omega t}$$

take the indicated derivative:

$$RI_m e^{j(\omega t + \phi)} + j\omega L I_m e^{j(\omega t + \phi)} = V_m e^{j\omega t}$$

and obtain an *algebraic* equation. In order to determine the values of  $I_m$  and  $\phi$ , we divide throughout by the common factor  $e^{j\omega t}$ :

$$RI_m e^{j\phi} + j\omega L I_m e^{j\phi} = V_m$$

factor the left side:

$$I_m e^{j\phi} (R + j\omega L) = V_m$$

rearrange:

$$I_m e^{j\phi} = \frac{V_m}{R + j\omega L}$$

and identify  $I_m$  and  $\phi$  by expressing the right side of the equation in exponential or polar form:

$$I_m e^{j\phi} = \frac{V_m}{\sqrt{R^2 + \omega^2 L^2}} e^{j[-\tan^{-1}(\omega L/R)]} \quad [15]$$

Thus,

$$I_m = \frac{V_m}{\sqrt{R^2 + \omega^2 L^2}}$$

and

$$\phi = -\tan^{-1} \frac{\omega L}{R}$$

In polar notation, this may be written as

$$I_m / \phi$$

or

$$V_m / \sqrt{R^2 + \omega^2 L^2} / -\tan^{-1}(\omega L/R)$$

The complex response is given by Eq. [15]. Since  $I_m$  and  $\phi$  are readily identified, we can write the expression for  $i(t)$  immediately. However, if we feel like using a more rigorous approach, we may obtain the real response  $i(t)$  by reinserting the  $e^{j\omega t}$  factor on both sides of Eq. [15] and taking the real part. Either way, we find that

$$i(t) = I_m \cos(\omega t + \phi) = \frac{V_m}{\sqrt{R^2 + \omega^2 L^2}} \cos\left(\omega t - \tan^{-1} \frac{\omega L}{R}\right)$$

which agrees with the response obtained in Eq. [4] for the same circuit.

## EXAMPLE 10.2

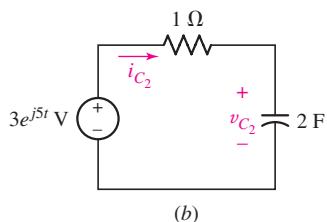
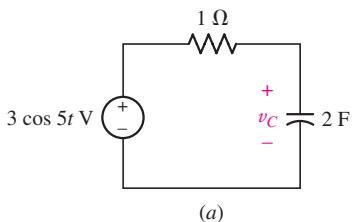


FIGURE 10.12 (a) An RC circuit for which the sinusoidal steady-state capacitor voltage is required. (b) Modified circuit, with the real source replaced with a complex source.

For the simple  $RC$  circuit of Fig. 10.12a, substitute an appropriate complex source and use it to solve for the steady-state capacitor voltage.

Since the real source is  $3 \cos 5t$ , we “replace” it with a complex source  $3e^{j5t}$  V. We’ll call the new capacitor voltage  $v_{C_2}$  and define a capacitor current  $i_{C_2}$  consistent with the passive sign convention (Fig. 10.12b).

The differential equation can be now obtained by simple application of KVL,

$$-3e^{j5t} + 1i_{C_2} + v_{C_2} = 0$$

or

$$-3e^{j5t} + 2 \frac{dv_{C_2}}{dt} + v_{C_2} = 0$$

We anticipate a steady-state response of the same form as our source; in other words,

$$v_{C_2} = V_m e^{j5t}$$

Substituting this into our differential equation and rearranging terms yields

$$j10V_m e^{j5t} + V_m e^{j5t} = 3e^{j5t}$$

Cancelling the exponential term, we find that

$$V_m = \frac{3}{1 + j10} = \frac{3}{\sqrt{1 + 10^2}} \angle -\tan^{-1}(10/1) \text{ V}$$

and our steady-state capacitor voltage is given by

$$\text{Re}\{v_{C_2}\} = \text{Re}\{29.85e^{-j84.3^\circ} e^{j5t} \text{ mV}\} = 29.85 \cos(5t - 84.3^\circ) \text{ mV}$$

## PRACTICE

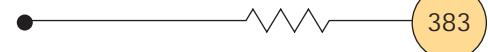
10.4 Evaluate and express the result in rectangular form:

(a)  $[(2/30^\circ)(5/-110^\circ)](1 + j2)$ ; (b)  $(5/-200^\circ) + 4/20^\circ$ . Evaluate and express the result in polar form: (c)  $(2 - j7)/(3 - j)$ ; (d)  $8 - j4 + [(5/80^\circ)/(2/20^\circ)]$ .

10.5 If the use of the passive sign convention is specified, find the (a) complex voltage that results when the complex current  $4e^{j800t}$  A is applied to the series combination of a 1 mF capacitor and a  $2 \Omega$  resistor; (b) complex current that results when the complex voltage  $100e^{j2000t}$  V is applied to the parallel combination of a 10 mH inductor and a  $50 \Omega$  resistor.

---

Ans: 10.4:  $21.4 - j6.38$ ;  $-0.940 + j3.08$ ;  $2.30/-55.6^\circ$ ;  $9.43/-11.22^\circ$ .  
 10.5:  $9.43e^{j(800t - 32.0^\circ)}$  V;  $5.39e^{j(2000t - 68.2^\circ)}$  A.



## 10.4 THE PHASOR

In the last section, we saw that the addition of an imaginary sinusoidal source led to algebraic equations which describe the sinusoidal steady-state response of a circuit. An intermediate step of our analysis was the “canceling” of the complex exponential term—once its derivative was taken, we apparently had no further use for it until the real form of the response was desired. Even then, it was possible to read the magnitude and phase angle directly from our analysis, and hence skip the step where we overtly take the real part. Another way of looking at this is that every voltage and current in our circuit contain the same factor  $e^{j\omega t}$ , and the frequency, although relevant to our analysis, *does not change* as we move through the circuit. Dragging it around, then, is a bit of a waste of time.

Looking back at Example 10.2, then, we could represent our source as

$$3e^{j0^\circ} \text{ V} \quad (\text{or even just } 3 \text{ V})$$

and our capacitor voltage as  $V_m e^{j\phi}$ , which we ultimately found was  $0.02985e^{-j84.3^\circ}$  V. Knowledge of the source frequency is implicit here; without it, we are unable to reconstruct any voltage or current.

These complex quantities are usually written in polar form rather than exponential form in order to achieve a slight additional saving of time and effort. For example, a source voltage

$$v(t) = V_m \cos \omega t = V_m \cos(\omega t + 0^\circ)$$

we now represent in complex form as

$$V_m \angle 0^\circ$$

and its current response

$$i(t) = I_m \cos(\omega t + \phi)$$

becomes

$$I_m \angle \phi$$

This abbreviated complex representation is called a **phasor**.<sup>1</sup>

Let us review the steps by which a real sinusoidal voltage or current is transformed into a phasor, and then we will be able to define a phasor more meaningfully and to assign a symbol to represent it.

A real sinusoidal current

$$i(t) = I_m \cos(\omega t + \phi)$$

is expressed as the real part of a complex quantity by invoking Euler's identity

$$i(t) = \operatorname{Re} \{ I_m e^{j(\omega t + \phi)} \}$$

We then represent the current as a complex quantity by dropping the instruction  $\operatorname{Re}\{\}$ , thus adding an imaginary component to the current without affecting the real component; further simplification is achieved by suppressing the factor  $e^{j\omega t}$ :

$$\mathbf{I} = I_m e^{j\phi}$$

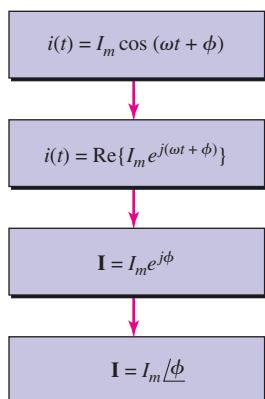
and writing the result in polar form:

$$\mathbf{I} = I_m \angle \phi$$

(1) Not to be confused with the *phaser*, an interesting device featured in a popular television series. . . .

$$e^{j0} = \cos 0 + j \sin 0 = 1$$

Remember that none of the steady-state circuits we are considering will respond at a frequency other than that of the excitation source, so that the value of  $\omega$  is always known.



The process by which we change  $i(t)$  into  $\mathbf{I}$  is called a *phasor transformation* from the time domain to the frequency domain.

This abbreviated complex representation is the *phasor representation*; phasors are complex quantities and hence are printed in boldface type. Capital letters are used for the phasor representation of an electrical quantity because the phasor is not an instantaneous function of time; it contains only amplitude and phase information. We recognize this difference in viewpoint by referring to  $i(t)$  as a *time-domain representation* and terming the phasor  $\mathbf{I}$  a *frequency-domain representation*. It should be noted that the frequency-domain expression of a current or voltage does not explicitly include the frequency. The process of returning to the time domain from the frequency domain is exactly the reverse of the previous sequence. Thus, given the phasor voltage

$$\mathbf{V} = 115/\underline{-45^\circ} \text{ volts}$$

and the knowledge that  $\omega = 500$  rad/s, we can write the time-domain equivalent directly:

$$v(t) = 115 \cos(500t - 45^\circ) \quad \text{volts}$$

If desired as a sine wave,  $v(t)$  could also be written

$$v(t) = 115 \sin(500t + 45^\circ) \quad \text{volts}$$

### PRACTICE

- 10.6 Let  $\omega = 2000$  rad/s and  $t = 1$  ms. Find the instantaneous value of each of the currents given here in phasor form: (a)  $j10$  A; (b)  $20 + j10$  A; (c)  $20 + j(10/20^\circ)$  A.

Ans:  $-9.09$  A;  $-17.42$  A;  $-15.44$  A.

## EXAMPLE 10.3

**Transform the time-domain voltage  $v(t) = 100 \cos(400t - 30^\circ)$  volts into the frequency domain.**

The time-domain expression is already in the form of a cosine wave with a phase angle. Thus, suppressing  $\omega = 400$  rad/s,

$$\mathbf{V} = 100/\underline{-30^\circ} \text{ volts}$$

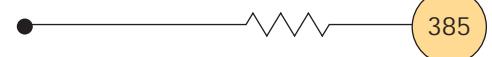
Note that we skipped several steps in writing this representation directly. Occasionally, this is a source of confusion for students, as they may forget that the phasor representation is *not* equal to the time-domain voltage  $v(t)$ . Rather, it is a simplified form of a complex function formed by adding an imaginary component to the real function  $v(t)$ .

### PRACTICE

- 10.7 Transform each of the following functions of time into phasor form:  
 (a)  $-5 \sin(580t - 110^\circ)$ ; (b)  $3 \cos 600t - 5 \sin(600t + 110^\circ)$ ;  
 (c)  $8 \cos(4t - 30^\circ) + 4 \sin(4t - 100^\circ)$ . Hint: First convert each into a single cosine function with a positive magnitude.

Ans:  $5/\underline{-20^\circ}$ ;  $2.41/\underline{-134.8^\circ}$ ;  $4.46/\underline{-47.9^\circ}$ .

Several useful trigonometric identities are provided on the inside cover for convenience.



The real power of the phasor-based analysis technique lies in the fact that it is possible to define *algebraic* relationships between the voltage and current for inductors and capacitors, just as we have always been able to do in the case of resistors. Now that we are able to transform into and out of the frequency domain, we can proceed to our simplification of sinusoidal steady-state analysis by establishing the relationship between the phasor voltage and phasor current for each of the three passive elements.

## The Resistor

The resistor provides the simplest case. In the time domain, as indicated by Fig. 10.13a, the defining equation is

$$v(t) = Ri(t)$$

Now let us apply the complex voltage

$$v(t) = V_m e^{j(\omega t + \theta)} = V_m \cos(\omega t + \theta) + jV_m \sin(\omega t + \theta) \quad [16]$$

and assume the complex current response

$$i(t) = I_m e^{j(\omega t + \phi)} = I_m \cos(\omega t + \phi) + jI_m \sin(\omega t + \phi) \quad [17]$$

so that

$$V_m e^{j(\omega t + \theta)} = Ri(t) = RI_m e^{j(\omega t + \phi)}$$

Dividing throughout by  $e^{j\omega t}$ , we find

$$V_m e^{j\theta} = RI_m e^{j\phi}$$

or, in polar form,

$$V_m / \theta = RI_m / \phi$$

But  $V_m / \theta$  and  $I_m / \phi$  merely represent the general voltage and current phasors  $\mathbf{V}$  and  $\mathbf{I}$ . Thus,

$$\mathbf{V} = R\mathbf{I} \quad [18]$$

The voltage-current relationship in phasor form for a resistor has the same form as the relationship between the time-domain voltage and current. The defining equation in phasor form is illustrated in Fig. 10.13b. The angles  $\theta$  and  $\phi$  are equal, so that the current and voltage are always in phase.

As an example of the use of both the time-domain and frequency-domain relationships, let us assume that a voltage of  $8 \cos(100t - 50^\circ)$  V is across a  $4 \Omega$  resistor. Working in the time domain, we find that the current must be

$$i(t) = \frac{v(t)}{R} = 2 \cos(100t - 50^\circ) \quad \text{A}$$

The phasor form of the same voltage is  $8/-50^\circ$  V, and therefore

$$\mathbf{I} = \frac{\mathbf{V}}{R} = 2/-50^\circ \quad \text{A}$$

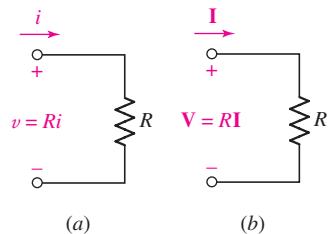


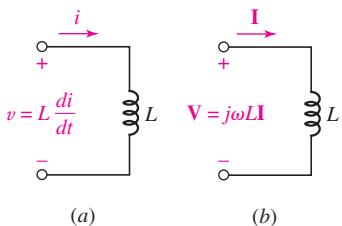
FIGURE 10.13 A resistor and its associated voltage and current in (a) the time domain,  $v = Ri$ ; and (b) the frequency domain,  $V = RI$ .

Ohm's law holds true both in the time domain and in the frequency domain. In other words, the voltage across a resistor is always given by the resistance times the current flowing through the element.



If we transform this answer back to the time domain, it is evident that the same expression for the current is obtained. We conclude that there is no saving in time or effort when a *resistive* circuit is analyzed in the frequency domain.

## The Inductor



**FIGURE 10.14** An inductor and its associated voltage and current in (a) the time domain,  $v = L di/dt$ ; and (b) the frequency domain,  $V = j\omega LI$ .

Let us now turn to the inductor. The time-domain representation is shown in Fig. 10.14a, and the defining equation, a time-domain expression, is

$$v(t) = L \frac{di(t)}{dt} \quad [19]$$

After substituting the complex voltage equation [16] and complex current equation [17] in Eq. [19], we have

$$V_m e^{j(\omega t + \theta)} = L \frac{d}{dt} I_m e^{j(\omega t + \phi)}$$

Taking the indicated derivative:

$$V_m e^{j(\omega t + \theta)} = j\omega L I_m e^{j(\omega t + \phi)}$$

and dividing through by  $e^{j\omega t}$ :

$$V_m e^{j\theta} = j\omega L I_m e^{j\phi}$$

we obtain the desired phasor relationship

$$\boxed{\mathbf{V} = j\omega L \mathbf{I}} \quad [20]$$

 The time-domain differential equation [19] has become the algebraic equation [20] in the frequency domain. The phasor relationship is indicated in Fig. 10.14b. Note that the angle of the factor  $j\omega L$  is exactly  $+90^\circ$  and that  $\mathbf{I}$  must therefore lag  $\mathbf{V}$  by  $90^\circ$  in an inductor.

### EXAMPLE 10.4

**Apply the voltage  $8/-50^\circ$  V at a frequency  $\omega = 100$  rad/s to a 4 H inductor, and determine the phasor current and the time-domain current.**

We make use of the expression we just obtained for the inductor,

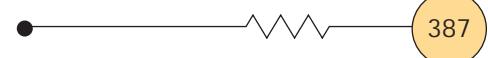
$$\mathbf{I} = \frac{\mathbf{V}}{j\omega L} = \frac{8/-50^\circ}{j100(4)} = -j0.02/-50^\circ = (1/-90^\circ)(0.02/-50^\circ)$$

or

$$\mathbf{I} = 0.02/-140^\circ \text{ A}$$

If we express this current in the time domain, it becomes

$$i(t) = 0.02 \cos(100t - 140^\circ) \text{ A} = 20 \cos(100t - 140^\circ) \text{ mA}$$



## The Capacitor

The final element to consider is the capacitor. The time-domain current-voltage relationship is

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

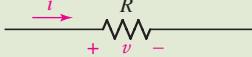
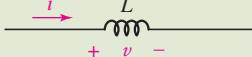
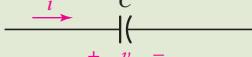
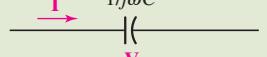
The equivalent expression in the frequency domain is obtained once more by letting  $v(t)$  and  $i(t)$  be the complex quantities of Eqs. [16] and [17], taking the indicated derivative, suppressing  $e^{j\omega t}$ , and recognizing the phasors  $\mathbf{V}$  and  $\mathbf{I}$ . Doing this, we find

$$\mathbf{I} = j\omega C \mathbf{V} \quad [21]$$

Thus,  $\mathbf{I}$  leads  $\mathbf{V}$  by  $90^\circ$  in a capacitor. This, of course, does not mean that a current response is present one-quarter of a period earlier than the voltage that caused it! We are studying steady-state response, and we find that the current maximum is caused by the increasing voltage that occurs  $90^\circ$  earlier than the voltage maximum.

The time-domain and frequency-domain representations are compared in Fig. 10.15a and b. We have now obtained the  $\mathbf{V}$ - $\mathbf{I}$  relationships for the three passive elements. These results are summarized in Table 10.1, where the time-domain  $v$ - $i$  expressions and the frequency-domain  $\mathbf{V}$ - $\mathbf{I}$  relationships are shown in adjacent columns for the three circuit elements. All the phasor equations are algebraic. Each is also linear, and the equations relating to inductance and capacitance bear a great similarity to Ohm's law. In fact, we will indeed use them as we use Ohm's law.

**TABLE 10.1 Comparison of Time-Domain and Frequency-Domain Voltage-Current Expressions**

Time Domain	Frequency Domain	
  	$v = Ri$ $v = L \frac{di}{dt}$ $v = \frac{1}{C} \int i dt$	$\mathbf{V} = R\mathbf{I}$ $\mathbf{V} = j\omega L \mathbf{I}$ $\mathbf{V} = \frac{1}{j\omega C} \mathbf{I}$
		  

## Kirchhoff's Laws Using Phasors

Kirchhoff's voltage law in the time domain is

$$v_1(t) + v_2(t) + \cdots + v_N(t) = 0$$

We now use Euler's identity to replace each real voltage  $v_i$  by a complex voltage having the same real part, suppress  $e^{j\omega t}$  throughout, and obtain

$$\mathbf{V}_1 + \mathbf{V}_2 + \cdots + \mathbf{V}_N = 0$$

Thus, we see that Kirchhoff's voltage law applies to phasor voltages just as it did in the time domain. Kirchhoff's current law can be shown to hold for phasor currents by a similar argument.

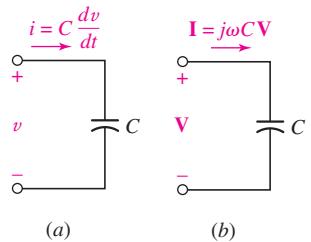


FIGURE 10.15 (a) The time-domain and (b) the frequency-domain relationships between capacitor current and voltage.

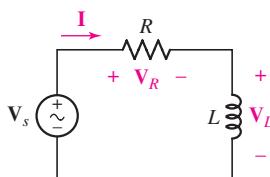


FIGURE 10.16 The series  $RL$  circuit with a phasor voltage applied.

Now let us look briefly at the series  $RL$  circuit that we have considered several times before. The circuit is shown in Fig. 10.16, and a phasor current and several phasor voltages are indicated. We may obtain the desired response, a time-domain current, by first finding the phasor current. From Kirchhoff's voltage law,

$$\mathbf{V}_R + \mathbf{V}_L = \mathbf{V}_s$$

and using the recently obtained  $\mathbf{V}$ - $\mathbf{I}$  relationships for the elements, we have

$$RI + j\omega LI = \mathbf{V}_s$$

The phasor current is then found in terms of the source voltage  $\mathbf{V}_s$ :

$$\mathbf{I} = \frac{\mathbf{V}_s}{R + j\omega L}$$

Let us select a source-voltage amplitude of  $V_m$  and phase angle of  $0^\circ$ . Thus,

$$\mathbf{I} = \frac{V_m \angle 0^\circ}{R + j\omega L}$$

The current may be transformed to the time domain by first writing it in polar form:

$$\mathbf{I} = \frac{V_m}{\sqrt{R^2 + \omega^2 L^2}} \left[ -\tan^{-1}(\omega L / R) \right]$$

and then following the familiar sequence of steps to obtain in a very simple manner the same result we obtained the “hard way” earlier in this chapter.

## EXAMPLE 10.5

For the  $RLC$  circuit of Fig. 10.17, determine  $\mathbf{I}_s$  and  $i_s(t)$  if both sources operate at  $\omega = 2$  rad/s, and  $\mathbf{I}_C = 2/28^\circ$  A.

The fact that we are given  $\mathbf{I}_C$  and asked for  $\mathbf{I}_s$  is all the prompting we need to consider applying KCL. If we label the capacitor voltage  $\mathbf{V}_C$  consistent with the passive sign convention, then

$$\mathbf{V}_C = \frac{1}{j\omega C} \mathbf{I}_C = \frac{-j}{2} \mathbf{I}_C = \frac{-j}{2} (2/28^\circ) = (0.5/-90^\circ)(2/28^\circ) = 1/-62^\circ \text{ V}$$

This voltage also appears across the  $2 \Omega$  resistor, so that the current  $\mathbf{I}_{R_2}$  flowing downward through that branch is

$$\mathbf{I}_{R_2} = \frac{1}{2} \mathbf{V}_C = \frac{1}{2} / -62^\circ \text{ A}$$

KCL then yields  $\mathbf{I}_s = \mathbf{I}_{R_2} + \mathbf{I}_C = 1/-62^\circ + \frac{1}{2}/-62^\circ = (3/2)/-62^\circ$  A. (We should note the addition of these polar quantities was trivial since the resistor and capacitor currents have the same angle, i.e., are in phase.)

Thus  $\mathbf{I}_s$  and knowledge of  $\omega$  permit us to write  $i_s(t)$  directly:

$$i_s(t) = 1.5 \cos(2t - 62^\circ) \text{ A}$$

## PRACTICE

10.8 In the circuit of Fig. 10.17, both sources operate at  $\omega = 1$  rad/s. If  $\mathbf{I}_C = 2/28^\circ$  A and  $\mathbf{I}_L = 3/53^\circ$  A, calculate (a)  $\mathbf{I}_s$ ; (b)  $\mathbf{V}_s$ ; (c)  $i_{R_1}(t)$ .

Ans: 3/-62° A; (b) 3.71/-4.5° V; (c)  $3.22 \cos(t - 4.5^\circ)$  A.

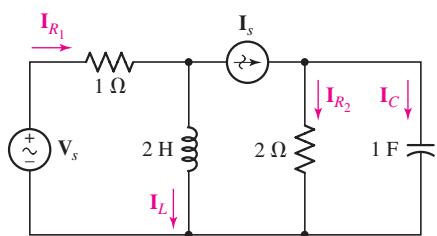


FIGURE 10.17 A three-mesh circuit. Each source operates at the same frequency.

## 10.5 IMPEDANCE AND ADMITTANCE

The current-voltage relationships for the three passive elements in the frequency domain are (assuming that the passive sign convention is satisfied)

$$\mathbf{V} = R\mathbf{I} \quad \mathbf{V} = j\omega L\mathbf{I} \quad \mathbf{V} = \frac{\mathbf{I}}{j\omega C}$$

If these equations are written as phasor voltage/phasor current ratios

$$\frac{\mathbf{V}}{\mathbf{I}} = R \quad \frac{\mathbf{V}}{\mathbf{I}} = j\omega L \quad \frac{\mathbf{V}}{\mathbf{I}} = \frac{1}{j\omega C}$$

we find that these ratios are simple quantities that depend on element values (and frequency also, in the case of inductance and capacitance). We treat these ratios in the same manner that we treat resistances, provided we remember that they are complex quantities.

Let us define the ratio of the phasor voltage to the phasor current as **impedance**, symbolized by the letter **Z**. The impedance is a complex quantity having the dimensions of ohms. Impedance is not a phasor and cannot be transformed to the time domain by multiplying by  $e^{j\omega t}$  and taking the real part. Instead, we think of an inductor as being represented in the time domain by its inductance  $L$  and in the frequency domain by its impedance  $j\omega L$ . A capacitor in the time domain has a capacitance  $C$ ; in the frequency domain, it has an impedance  $1/j\omega C$ . Impedance is a part of the frequency domain and not a concept that is a part of the time domain.



$$\begin{aligned}\mathbf{Z}_R &= R \\ \mathbf{Z}_L &= j\omega L \\ \mathbf{Z}_C &= \frac{1}{j\omega C}\end{aligned}$$

### Series Impedance Combinations

The validity of Kirchhoff's two laws in the frequency domain leads to the fact that impedances may be combined in series and parallel by the same rules we established for resistances. For example, at  $\omega = 10 \times 10^3$  rad/s, a 5 mH inductor in series with a 100  $\mu$ F capacitor may be replaced by the sum of the individual impedances. The impedance of the inductor is

$$\mathbf{Z}_L = j\omega L = j50 \Omega$$

and the impedance of the capacitor is

$$\mathbf{Z}_C = \frac{1}{j\omega C} = \frac{-j}{\omega C} = -j1 \Omega$$

The impedance of the series combination is therefore

$$\mathbf{Z}_{eq} = \mathbf{Z}_L + \mathbf{Z}_C = j50 - j1 = j49 \Omega$$

Note that  $\frac{1}{j} = -j$ .



### Parallel Impedance Combinations

The *parallel* combination of the 5 mH inductor and the 100  $\mu$ F capacitor at  $\omega = 10,000$  rad/s is calculated in exactly the same fashion in which we

calculated parallel resistances:

$$\mathbf{Z}_{\text{eq}} = \frac{(j50)(-j1)}{j50 - j1} = \frac{50}{j49} = -j1.020 \Omega$$

At  $\omega = 5000 \text{ rad/s}$ , the parallel equivalent is  $-j2.17 \Omega$ .

## Reactance

Of course, we may choose to express impedance in either *rectangular* ( $\mathbf{Z} = R + jX$ ) or *polar* ( $\mathbf{Z} = |\mathbf{Z}| \angle \theta$ ) form. In rectangular form, we can see clearly the real part which arises only from real resistances, and an imaginary component, termed the *reactance*, which arises from the energy storage elements. Both resistance and reactance have units of ohms, but reactance will always depend upon frequency. An ideal resistor has zero reactance; an ideal inductor or capacitor is purely reactive (i.e., characterized by zero resistance). Can a series or parallel combination include *both* a capacitor and an inductor, and yet have *zero reactance*? Sure! Consider the series connection of a  $1 \Omega$  resistor, a  $1 \text{ F}$  capacitor, and a  $1 \text{ H}$  inductor driven at  $\omega = 1 \text{ rad/s}$ .  $\mathbf{Z}_{\text{eq}} = 1 - j(1)(1) + j(1)(1) = 1 \Omega$ . At that particular frequency, the equivalent is a simple  $1 \Omega$  resistor. However, even small deviations from  $\omega = 1 \text{ rad/s}$  lead to nonzero reactance.

### EXAMPLE 10.6

Determine the equivalent impedance of the network shown in Fig. 10.18a, given an operating frequency of  $5 \text{ rad/s}$ .

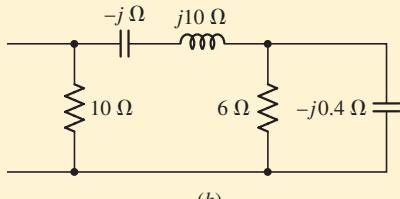
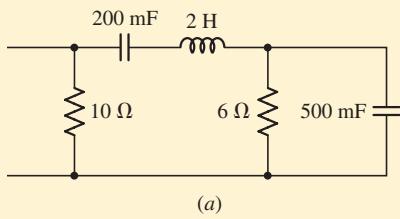


FIGURE 10.18 (a) A network that is to be replaced by a single equivalent impedance. (b) The elements are replaced by their impedances at  $\omega = 5 \text{ rad/s}$ .

We begin by converting the resistors, capacitors, and inductor into the corresponding impedances as shown in Fig. 10.18b.

Upon examining the resulting network, we observe that the  $6 \Omega$  impedance is in parallel with  $-j0.4 \Omega$ . This combination is equivalent to

$$\frac{(6)(-j0.4)}{6 - j0.4} = 0.02655 - j0.3982 \Omega$$

which is in series with both the  $-j \Omega$  and  $j10 \Omega$  impedances, so that we have

$$0.0265 - j0.3982 - j + j10 = 0.02655 + j8.602 \Omega$$

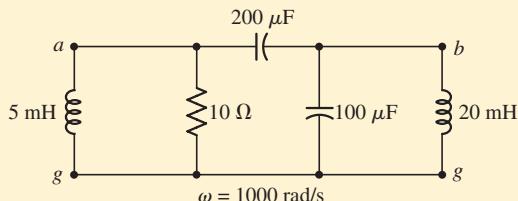
This new impedance is in parallel with  $10 \Omega$ , so that the equivalent impedance of the network is

$$\begin{aligned} 10 \parallel (0.02655 + j8.602) &= \frac{10(0.02655 + j8.602)}{10 + 0.02655 + j8.602} \\ &= 4.255 + j4.929 \Omega \end{aligned}$$

Alternatively, we can express the impedance in polar form as  $6.511 \angle 49.20^\circ \Omega$ .

### PRACTICE

- 10.9 With reference to the network shown in Fig. 10.19, find the input impedance  $Z_{in}$  that would be measured between terminals: (a)  $a$  and  $g$ ; (b)  $b$  and  $g$ ; (c)  $a$  and  $b$ .



■ FIGURE 10.19

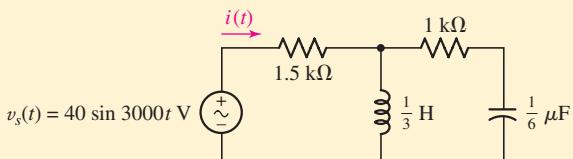
Ans:  $2.81 + j4.49 \Omega$ ;  $1.798 - j1.124 \Omega$ ;  $0.1124 - j3.82 \Omega$ .

It is important to note that the resistive component of the impedance is not necessarily equal to the resistance of the resistor that is present in the network. For example, a  $10 \Omega$  resistor and a  $5 \text{ H}$  inductor in series at  $\omega = 4 \text{ rad/s}$  have an equivalent impedance  $Z = 10 + j20 \Omega$ , or, in polar form,  $22.4 \angle 63.4^\circ \Omega$ . In this case, the resistive component of the impedance is equal to the resistance of the series resistor because the network is a simple series network. However, if these same two elements are placed in parallel, the equivalent impedance is  $10(j20)/(10 + j20) \Omega$ , or  $8 + j4 \Omega$ . The resistive component of the impedance is now  $8 \Omega$ .

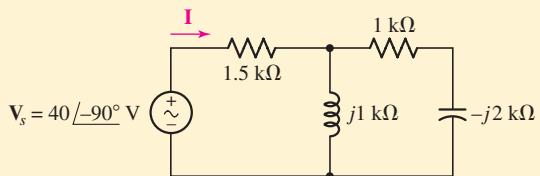


## EXAMPLE 10.7

Find the current  $i(t)$  in the circuit shown in Fig. 10.20a.



(a)



(b)

FIGURE 10.20 (a) An RLC circuit for which the sinusoidal forced response  $i(t)$  is desired. (b) The frequency-domain equivalent of the given circuit at  $\omega = 3000$  rad/s.

► **Identify the goal of the problem.**

We need to find the sinusoidal steady-state current flowing through the  $1.5\text{ k}\Omega$  resistor due to the  $3000\text{ rad/s}$  voltage source.

► **Collect the known information.**

We begin by drawing a frequency-domain circuit. The source is transformed to the frequency-domain representation  $40 / -90^\circ$  V, the frequency domain response is represented as  $\mathbf{I}$ , and the impedances of the inductor and capacitor, determined at  $\omega = 3000$  rad/s, are  $j\text{ k}\Omega$  and  $-j2\text{ k}\Omega$ , respectively. The corresponding frequency-domain circuit is shown in Fig. 10.20b.

► **Devise a plan.**

We will analyze the circuit of Fig. 10.20b to obtain  $\mathbf{I}$ ; combining impedances and invoking Ohm's law is one possible approach. We will then make use of the fact that we know  $\omega = 3000$  rad/s to convert  $\mathbf{I}$  into a time-domain expression.

► **Construct an appropriate set of equations.**

$$\begin{aligned} \mathbf{Z}_{eq} &= 1.5 + \frac{(j)(1-2j)}{j+1-2j} = 1.5 + \frac{2+j}{1-j} \\ &= 1.5 + \frac{2+j}{1-j} \frac{1+j}{1+j} = 1.5 + \frac{1+j^3}{2} \\ &= 2 + j1.5 = 2.5 / 36.87^\circ \text{ k}\Omega \end{aligned}$$

The phasor current is then simply

$$\mathbf{I} = \frac{\mathbf{V}_s}{Z_{eq}}$$

► **Determine if additional information is required.**

Substituting known values, we find that

$$\mathbf{I} = \frac{40/-90^\circ}{2.5/36.87^\circ} \text{ mA}$$

which, along with the knowledge that  $\omega = 3000 \text{ rad/s}$ , is sufficient to solve for  $i(t)$ .

► **Attempt a solution.**

This complex expression is easily simplified to a single complex number in polar form:

$$\mathbf{I} = \frac{40}{2.5} / -90^\circ - 36.87^\circ \text{ mA} = 16.00 / -126.9^\circ \text{ mA}$$

Upon transforming the current to the time domain, the desired response is obtained:

$$i(t) = 16 \cos(3000t - 126.9^\circ) \text{ mA}$$

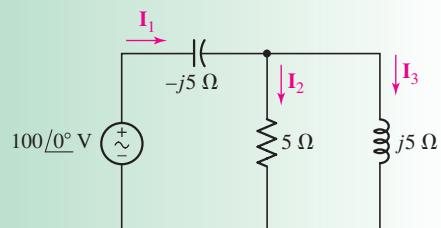
► **Verify the solution. Is it reasonable or expected?**

The effective impedance connected to the source has an angle of  $+36.87^\circ$ , indicating that it has a net inductive character, or that the current will lag the voltage. Since the voltage source has a phase angle of  $-90^\circ$  (once converted to a cosine source), we see that our answer is consistent.

**PRACTICE**

- 10.10 In the frequency-domain circuit of Fig. 10.21, find (a)  $\mathbf{I}_1$ ; (b)  $\mathbf{I}_2$ ; (c)  $\mathbf{I}_3$ .

Ans:  $28.3/45^\circ \text{ A}$ ;  $20/90^\circ \text{ A}$ ;  $20/0^\circ \text{ A}$ .



■ FIGURE 10.21

Before we begin to write great numbers of equations in the time domain or in the frequency domain, it is very important that we shun the construction of equations that are partly in the time domain, partly in the frequency domain, and wholly incorrect. One clue that a faux pas of this type has been committed is the sight of both a complex number and a  $t$  in the same equation, except in the factor  $e^{j\omega t}$ . And, since  $e^{j\omega t}$  plays a much bigger role in derivations than in applications, it is pretty safe to say that students who find they have just created an equation containing  $j$  and  $t$ , or  $\angle$  and  $t$ , have created a monster that the world would be better off without.



For example, a few equations back we saw

$$\mathbf{I} = \frac{\mathbf{V}_s}{\mathbf{Z}_{eq}} = \frac{40/-90^\circ}{2.5/36.9^\circ} = 16/-126.9^\circ \text{ mA}$$

Please do not try anything like the following:

$$i(t) \not\propto \frac{40 \sin 3000t}{2.5/36.9^\circ} \quad \text{or} \quad i(t) \not\propto \frac{40 \sin 3000t}{2 + j1.5}$$

## Admittance

Although the concept of impedance is very useful, and familiar in a way based on our experience with resistors, the reciprocal is often just as valuable. We define this quantity as the **admittance**  $\mathbf{Y}$  of a circuit element or passive network, and it is simply the ratio of current to voltage:

The real part of the admittance is the **conductance**  $G$ , and the imaginary part is the **susceptance**  $B$ . All three quantities ( $\mathbf{Y}$ ,  $G$ , and  $B$ ) are measured in siemens.

The real part of the admittance is the **conductance**  $G$ , and the imaginary part of the admittance is the **susceptance**  $B$ . Thus,

$$\mathbf{Y} = G + jB = \frac{1}{\mathbf{Z}} = \frac{1}{R + jX} \quad [22]$$

Equation [22] should be scrutinized carefully; it does *not* state that the real part of the admittance is equal to the reciprocal of the real part of the impedance, or that the imaginary part of the admittance is equal to the reciprocal of the imaginary part of the impedance!

$$\begin{aligned}\mathbf{Y}_R &= \frac{1}{R} \\ \mathbf{Y}_L &= \frac{1}{j\omega L} \\ \mathbf{Y}_C &= j\omega C\end{aligned}$$

There is a general (unitless) term for both impedance and admittance—*immitance*—which is sometimes used, but not very often.



## PRACTICE

- 10.11 Determine the admittance (in rectangular form) of (a) an impedance  $\mathbf{Z} = 1000 + j400 \Omega$ ; (b) a network consisting of the parallel combination of an  $800 \Omega$  resistor, a  $1 \text{ mH}$  inductor, and a  $2 \text{ nF}$  capacitor, if  $\omega = 1 \text{ Mrad/s}$ ; (c) a network consisting of the series combination of an  $800 \Omega$  resistor, a  $1 \text{ mH}$  inductor, and a  $2 \text{ nF}$  capacitor, if  $\omega = 1 \text{ Mrad/s}$ .

Ans:  $0.862 - j0.345 \text{ mS}$ ;  $1.25 + j1 \text{ mS}$ ;  $0.899 - j0.562 \text{ mS}$ .

## 10.6 NODAL AND MESH ANALYSIS

We previously achieved a great deal with nodal and mesh analysis techniques, and it's reasonable to ask if a similar procedure might be valid in terms of phasors and impedances for the sinusoidal steady state. We already know that both of Kirchhoff's laws are valid for phasors; also, we have an Ohm-like law for the passive elements  $\mathbf{V} = \mathbf{Z}\mathbf{I}$ . We may therefore analyze circuits by nodal techniques in the sinusoidal steady state. Using similar arguments, we can establish that mesh analysis methods are valid (and often useful) as well.

## EXAMPLE 10.8

Find the time-domain node voltages  $v_1(t)$  and  $v_2(t)$  in the circuit shown in Fig. 10.22.

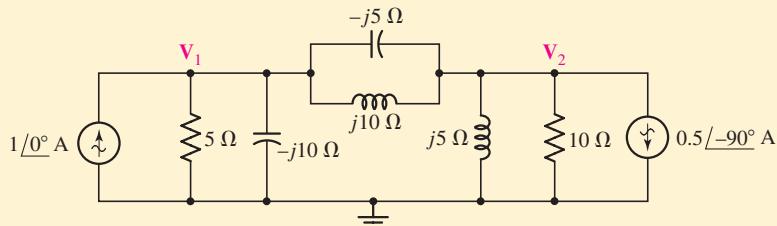


FIGURE 10.22 A frequency-domain circuit for which node voltages  $V_1$  and  $V_2$  are identified.

Two current sources are given as phasors, and phasor node voltages  $V_1$  and  $V_2$  are indicated. At the left node we apply KCL, yielding:

$$\frac{V_1}{5} + \frac{V_1}{-j10} + \frac{V_1 - V_2}{-j5} + \frac{V_1 - V_2}{j10} = 1\angle 0^\circ = 1 + j0$$

At the right node,

$$\frac{V_2 - V_1}{-j5} + \frac{V_2 - V_1}{j10} + \frac{V_2}{j5} + \frac{V_2}{10} = -(0.5\angle -90^\circ) = j0.5$$

Combining terms, we have

$$(0.2 + j0.2)V_1 - j0.1V_2 = 1$$

and

$$-j0.1V_1 + (0.1 - j0.1)V_2 = j0.5$$

These equations are easily solved on most scientific calculators, resulting in  $V_1 = 1 - j2$  V and  $V_2 = -2 + j4$  V.

The time-domain solutions are obtained by expressing  $V_1$  and  $V_2$  in polar form:

$$\begin{aligned} V_1 &= 2.24\angle -63.4^\circ \\ V_2 &= 4.47\angle 116.6^\circ \end{aligned}$$

and passing to the time domain:

$$v_1(t) = 2.24 \cos(\omega t - 63.4^\circ) \quad \text{V}$$

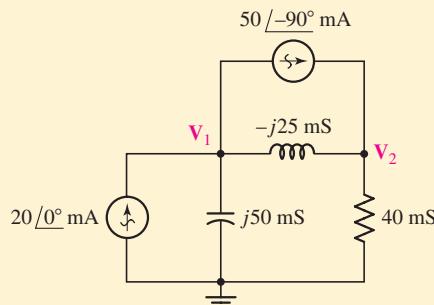
$$v_2(t) = 4.47 \cos(\omega t + 116.6^\circ) \quad \text{V}$$

Note that the value of  $\omega$  would have to be known in order to compute the impedance values given on the circuit diagram. Also, *both sources must be operating at the same frequency*.

(Continued on next page)

**PRACTICE**

10.12 Use nodal analysis on the circuit of Fig. 10.23 to find  $\mathbf{V}_1$  and  $\mathbf{V}_2$ .



■ FIGURE 10.23

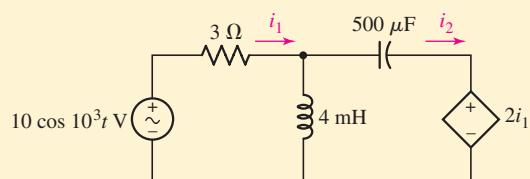
Ans:  $1.062\angle 23.3^\circ$  V;  $1.593\angle -50.0^\circ$  V.



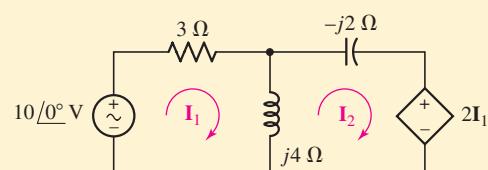
Now let us look at an example of mesh analysis, keeping in mind again that all sources must be operating at the same frequency. Otherwise, it is impossible to define a numerical value for any reactance in the circuit. As we see in the next section, the only way out of such a dilemma is to apply superposition.

**EXAMPLE 10.9**

Obtain expressions for the time-domain currents  $i_1$  and  $i_2$  in the circuit given as Fig. 10.24a.



(a)



(b)

■ FIGURE 10.24 (a) A time-domain circuit containing a dependent source. (b) The corresponding frequency-domain circuit.

Noting from the left source that  $\omega = 10^3 \text{ rad/s}$ , we draw the frequency-domain circuit of Fig. 10.24b and assign mesh currents  $\mathbf{I}_1$  and  $\mathbf{I}_2$ . Around mesh 1,

$$3\mathbf{I}_1 + j4(\mathbf{I}_1 - \mathbf{I}_2) = 10/0^\circ$$

or

$$(3 + j4)\mathbf{I}_1 - j4\mathbf{I}_2 = 10$$

while mesh 2 leads to

$$j4(\mathbf{I}_2 - \mathbf{I}_1) - j2\mathbf{I}_2 + 2\mathbf{I}_1 = 0$$

or

$$(2 - j4)\mathbf{I}_1 + j2\mathbf{I}_2 = 0$$

Solving,

$$\mathbf{I}_1 = \frac{14 + j8}{13} = 1.24/29.7^\circ \text{ A}$$

$$\mathbf{I}_2 = \frac{20 + j30}{13} = 2.77/56.3^\circ \text{ A}$$

Hence,

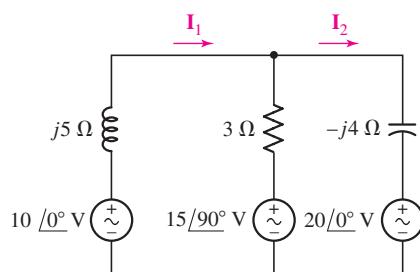
$$i_1(t) = 1.24 \cos(10^3 t + 29.7^\circ) \quad \text{A}$$

$$i_2(t) = 2.77 \cos(10^3 t + 56.3^\circ) \quad \text{A}$$

### PRACTICE

10.13 Use mesh analysis on the circuit of Fig. 10.25 to find  $\mathbf{I}_1$  and  $\mathbf{I}_2$ .

Ans:  $4.87/-164.6^\circ \text{ A}$ ;  $7.17/-144.9^\circ \text{ A}$ .



■ FIGURE 10.25

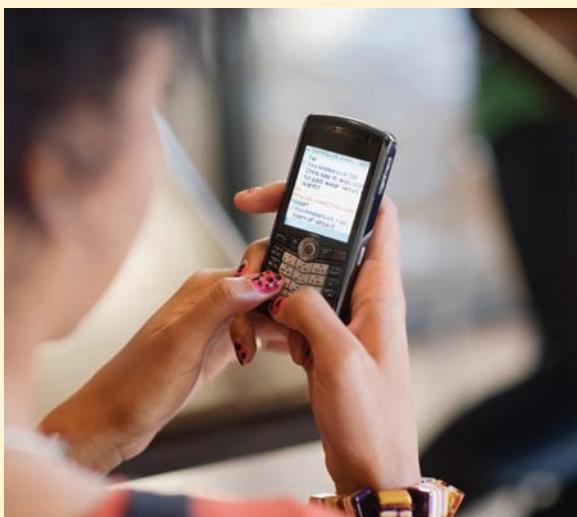
## 10.7 SUPERPOSITION, SOURCE TRANSFORMATIONS, AND THÉVENIN'S THEOREM

After inductors and capacitors were introduced in Chap. 7, we found that circuits containing these elements were still linear, and that the benefits of linearity were again available. Included among these were the superposition principle, Thévenin's and Norton's theorems, and source transformations. Thus, we know that these methods may be used on the circuits we are now considering; the fact that we happen to be applying sinusoidal sources and are seeking only the forced response is immaterial. The fact that we are analyzing the circuits in terms of phasors is also immaterial; they are still linear circuits. We might also remember that linearity and superposition were invoked when we combined real and imaginary sources to obtain a complex source.

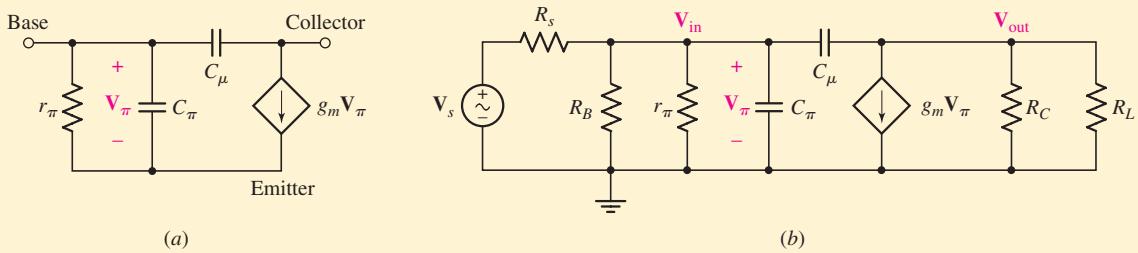
# PRACTICAL APPLICATION

## Cutoff Frequency of a Transistor Amplifier

Transistor-based amplifier circuits are an integral part of many modern electronic instruments. One common application is in mobile telephones (Fig. 10.26), where audio signals are superimposed on high-frequency carrier waves. Unfortunately, transistors have built-in capacitances that lead to limitations in the frequencies at which they can be used, and this fact must be considered when choosing a transistor for a particular application.



■ FIGURE 10.26 Transistor amplifiers are used in many devices, including mobile phones. Linear circuit models are often used to analyze their performance as a function of frequency. © PNC/Getty Images/RF.



■ FIGURE 10.27 (a) High-frequency hybrid- $\pi$  transistor model. (b) Common-emitter amplifier circuit using the hybrid- $\pi$  transistor model.

Figure 10.27a shows what is commonly referred to as a *high-frequency hybrid- $\pi$  model* for a bipolar junction transistor. In practice, although transistors are *non-linear* devices, we find that this simple *linear* circuit does a reasonably accurate job of modeling the actual device behavior. The two capacitors  $C_\pi$  and  $C_\mu$  are used to represent internal capacitances that characterize the particular transistor being used; additional capacitors as well as resistors can be added to increase the accuracy of the model as needed. Figure 10.27b shows the transistor model inserted into an amplifier circuit known as a common emitter amplifier.

Assuming a sinusoidal steady-state signal represented by its Thévenin equivalent  $V_s$  and  $R_s$ , we are interested in the ratio of the output voltage  $V_{out}$  to the input voltage  $V_{in}$ . The presence of the internal transistor capacitances leads to a reduction in amplification as the frequency of  $V_s$  is increased; this ultimately limits the frequencies at which the circuit will operate properly. Writing a single nodal equation at the output yields

$$-g_m V_\pi = \frac{V_{out} - V_{in}}{1/j\omega C_\mu} + \frac{V_{out}}{R_C \parallel R_L}$$

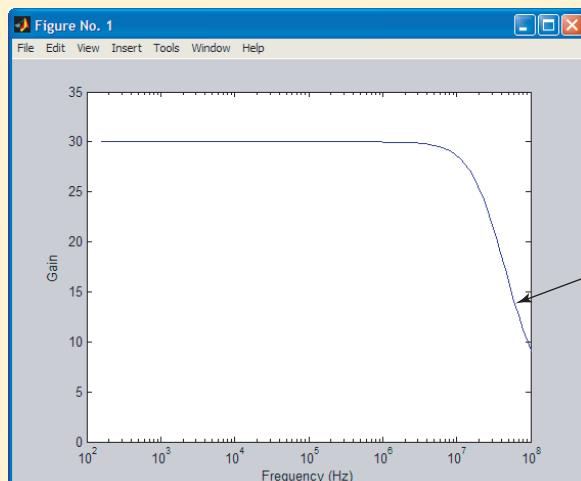


One final comment is in order. Up to this point, we have restricted ourselves to considering either single-source circuits or multiple-source circuits in which *every source operates at the exact same frequency*. This is necessary in order to define specific impedance values for inductive and capacitive elements. However, the concept of phasor analysis can be easily extended to circuits with multiple sources operating at different frequencies. In such

Solving for  $V_{out}$  in terms of  $V_{in}$ , and noting that  $V_\pi = V_{in}$ , we obtain an expression for the amplifier gain

$$\begin{aligned}\frac{V_{out}}{V_{in}} &= \frac{-g_m(R_C \parallel R_L)(1/j\omega C_\mu) + (R_C \parallel R_L)}{(R_C \parallel R_L) + (1/j\omega C_\mu)} \\ &= \frac{-g_m(R_C \parallel R_L) + j\omega(R_C \parallel R_L)C_\mu}{1 + j\omega(R_C \parallel R_L)C_\mu}\end{aligned}$$

Given the typical values  $g_m = 30 \text{ mS}$ ,  $R_C = R_L = 2 \text{ k}\Omega$ , and  $C_\mu = 5 \text{ pF}$ , we can plot the magnitude of the gain as a function of frequency (recalling that  $\omega = 2\pi f$ ). The semilogarithmic plot is shown in Fig. 10.28a, and the MATLAB script used to generate the figure is given in Fig. 10.28b. It is interesting, but maybe not totally surprising, to see that a characteristic such as the amplifier gain is dependent on frequency. In fact, we might be able to contemplate using such a circuit as a means of filtering out frequencies we aren't interested in. However, at least for relatively low frequencies, we see that the gain is essentially independent of the frequency of our input source.



(a)

When characterizing amplifiers, it is common to reference the frequency at which the gain is reduced to  $1/\sqrt{2}$  times its maximum value. From Fig. 10.28a, we see that the maximum gain magnitude is 30, and the gain magnitude is reduced to  $30/\sqrt{2} = 21$  at a frequency of approximately 30 MHz. This frequency is often called the *cutoff* or *corner* frequency of the amplifier. If operation at a higher frequency is required, either the internal capacitances must be reduced (i.e., a different transistor must be used) or the circuit must be redesigned in some way.

We should note at this point that defining the gain relative to  $V_{in}$  does not present a complete picture of the frequency-dependent behavior of the amplifier. This may be apparent if we briefly consider the capacitance  $C_\pi$ : as  $\omega \rightarrow \infty$ ,  $Z_{C_\pi} \rightarrow 0$ , so  $V_{in} \rightarrow 0$ . This effect does not manifest itself in the simple equation we derived. A more comprehensive approach is to develop an equation for  $V_{out}$  in terms of  $V_s$ , in which case both capacitances will appear in the expression; this requires a little bit more algebra.

No longer  
amplifying  
effectively

```
EDU» frequency = logspace(3,9,100);
EDU» numerator = -30e-3*1000 + i*frequency*1000*5e-12;
EDU» denominator = 1 + i*frequency*1000*5e-12;
EDU» for k = 1:100
EDU» gain(k) = abs(numerator(k)/denominator(k));
EDU» end
EDU» semilogx(frequency/2/pi,gain);
EDU» xlabel('Frequency (Hz)');
EDU» ylabel('Gain');
EDU» axis([100 1e8 0 35]);
```

(b)

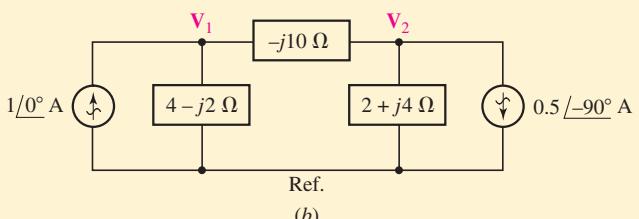
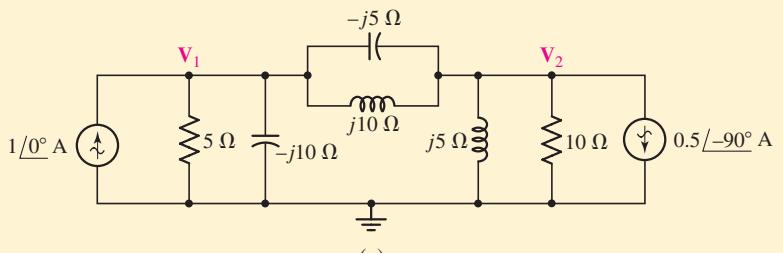
FIGURE 10.28 (a) Amplifier gain as a function of frequency. (b) MATLAB script used to create plot.

instances, we simply employ superposition to determine the voltages and currents due to each source, and then add the results *in the time domain*. If several sources are operating at the same frequency, superposition will also allow us to consider those sources at the same time, and add the resulting response to the response(s) of any other source(s) operating at a different frequency.



## EXAMPLE 10.10

Use superposition to find  $\mathbf{V}_1$  for the circuit of Fig. 10.22, repeated as Fig. 10.29a for convenience.



■ FIGURE 10.29 (a) Circuit of Fig. 10.22 for which  $\mathbf{V}_1$  is desired, (b)  $\mathbf{V}_1$  may be found by using superposition of the separate phasor responses.

First we redraw the circuit as Fig. 10.29b, where each pair of parallel impedances is replaced by a single equivalent impedance. That is,  $5 \parallel -j10 \Omega$  is  $4 - j2 \Omega$ ;  $j10 \parallel -j5 \Omega$  is  $-j10 \Omega$ ; and  $10 \parallel j5$  is equal to  $2 + j4 \Omega$ . To find  $\mathbf{V}_1$ , we first activate only the left source and find the partial response,  $\mathbf{V}_{1L}$ . The  $1\angle0^\circ$  source is in parallel with an impedance of

$$(4 - j2) \parallel (-j10 + 2 + j4)$$

so that

$$\begin{aligned}\mathbf{V}_{1L} &= 1\angle0^\circ \frac{(4 - j2)(-j10 + 2 + j4)}{4 - j2 - j10 + 2 + j4} \\ &= \frac{-4 - j28}{6 - j8} = 2 - j2 \text{ V}\end{aligned}$$

With only the right source active, current division and Ohm's law yield

$$\mathbf{V}_{1R} = (-0.5\angle-90^\circ) \left( \frac{2 + j4}{4 - j2 - j10 + 2 + j4} \right) (4 - j2) = -1 \text{ V}$$

Summing, then,

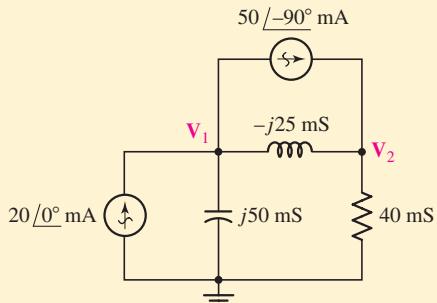
$$\mathbf{V}_1 = \mathbf{V}_{1L} + \mathbf{V}_{1R} = 2 - j2 - 1 = 1 - j2 \quad \text{V}$$

which agrees with our previous result from Example 10.8.

As we will see, superposition is also extremely useful when dealing with a circuit in which not all sources operate at the same frequency.

**PRACTICE**

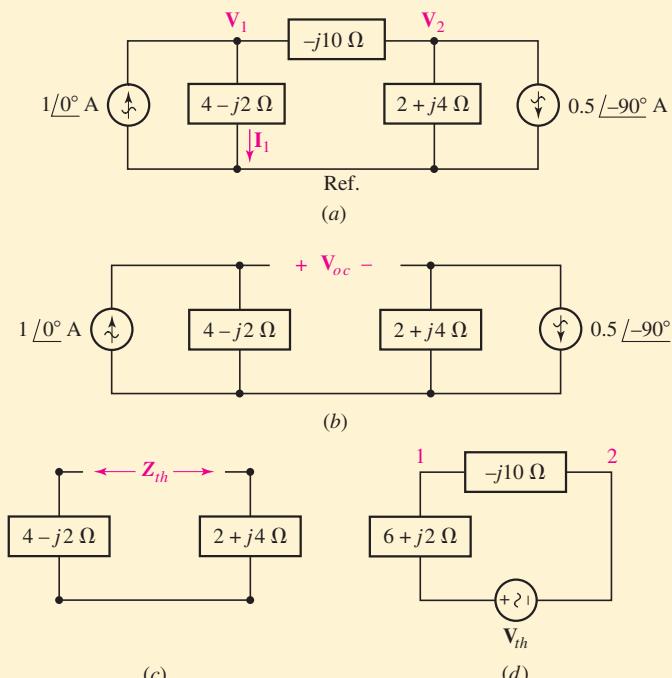
- 10.14 If superposition is used on the circuit of Fig. 10.30, find  $V_1$  with  
(a) only the  $20/0^\circ$  mA source operating; (b) only the  $50/-90^\circ$  mA  
source operating.



■ FIGURE 10.30

Ans:  $0.1951 - j0.556$  V;  $0.780 + j0.976$  V.**EXAMPLE 10.11**

Determine the Thévenin equivalent seen by the  $-j10 \Omega$  impedance of Fig. 10.31a, and use this to compute  $V_1$ .



■ FIGURE 10.31 (a) Circuit of Fig. 10.29b. The Thévenin equivalent seen by the  $-j10 \Omega$  impedance is desired. (b)  $V_{oc}$  is defined. (c)  $Z_{th}$  is defined. (d) The circuit is redrawn using the Thévenin equivalent.

(Continued on next page)

The open-circuit voltage, defined in Fig. 10.31b, is

$$\begin{aligned}\mathbf{V}_{oc} &= (1/0^\circ)(4 - j2) - (-0.5/-90^\circ)(2 + j4) \\ &= 4 - j2 + 2 - j1 = 6 - j3 \quad \text{V}\end{aligned}$$

The impedance of the inactive circuit of Fig. 10.31c as viewed from the load terminals is simply the sum of the two remaining impedances. Hence,

$$\mathbf{Z}_{th} = 6 + j2 \Omega$$

Thus, when we reconnect the circuit as in Fig. 10.31d, the current directed from node 1 toward node 2 through the  $-j10 \Omega$  load is

$$\mathbf{I}_{12} = \frac{6 - j3}{6 + j2 - j10} = 0.6 + j0.3 \text{ A}$$

We now know the current flowing through the  $-j10 \Omega$  impedance of Fig. 10.31a. Note that we are unable to compute  $\mathbf{V}_1$  using the circuit of Fig. 10.31d as the reference node no longer exists. Returning to the original circuit, then, and subtracting the  $0.6 + j0.3 \text{ A}$  current from the left source current, the downward current through the  $(4 - j2) \Omega$  branch is found:

$$\mathbf{I}_1 = 1 - 0.6 - j0.3 = 0.4 - j0.3 \quad \text{A}$$

and, thus,

$$\mathbf{V}_1 = (0.4 - j0.3)(4 - j2) = 1 - j2 \quad \text{V}$$

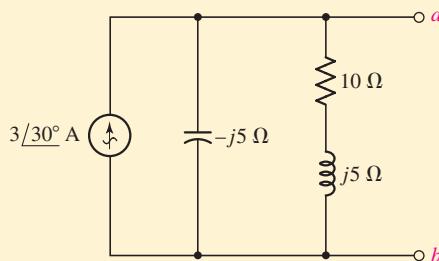
as before.

We might have been clever and used Norton's theorem on the three elements on the right of Fig. 10.31a, assuming that our chief interest is in  $\mathbf{V}_1$ . Source transformations can also be used repeatedly to simplify the circuit. Thus, all the shortcuts and tricks that arose in Chaps. 4 and 5 are available for circuit analysis in the frequency domain. The slight additional complexity that is apparent now arises from the necessity of using complex numbers and not from any more involved theoretical considerations.

### PRACTICE

- 10.15 For the circuit of Fig. 10.32, find the (a) open-circuit voltage  $\mathbf{V}_{ab}$ ; (b) downward current in a short circuit between  $a$  and  $b$ ; (c) Thévenin equivalent impedance  $\mathbf{Z}_{ab}$  in parallel with the current source.

Ans:  $16.77/-33.4^\circ \text{ V}$ ;  $2.60 + j1.500 \text{ A}$ ;  $2.5 - j5 \Omega$ .



■ FIGURE 10.32

## EXAMPLE 10.12

Determine the power dissipated by the  $10 \Omega$  resistor in the circuit of Fig. 10.33a.

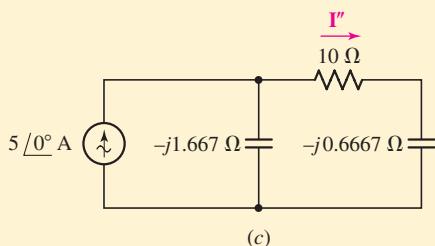
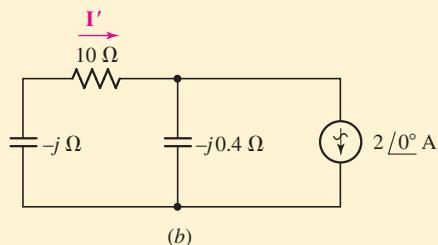
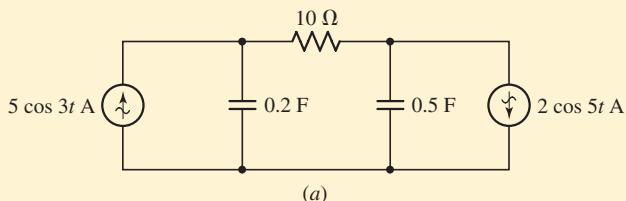


FIGURE 10.33 (a) A simple circuit having sources operating at different frequencies. (b) Circuit with the left source killed. (c) Circuit with the right source killed.

After glancing at the circuit, we might be tempted to write two quick nodal equations, or perhaps perform two sets of source transformations and launch immediately into finding the voltage across the  $10 \Omega$  resistor.

Unfortunately, this is impossible, since we have *two* sources operating at *different* frequencies. In such a situation, there is no way to compute the impedance of any capacitor or inductor in the circuit—which *would* we use?

The only way out of this dilemma is to employ superposition, grouping all sources with the same frequency in the same subcircuit, as shown in Fig. 10.33b and c.

In the subcircuit of Fig. 10.33b, we quickly compute the current  $\mathbf{I}'$  using current division:

$$\begin{aligned}\mathbf{I}' &= 2 / 0^\circ \left[ \frac{-j0.4}{10 - j - j0.4} \right] \\ &= 79.23 / -82.03^\circ \text{ mA}\end{aligned}$$

In future studies of signal processing, we will also be introduced to the method of Jean-Baptiste Joseph Fourier, a French mathematician who developed a technique for representing almost any arbitrary function by a combination of sinusoids. When working with linear circuits, once we know the response of a particular circuit to a general sinusoidal forcing function, we can easily predict the response of the circuit to an arbitrary waveform represented by a Fourier series function, simply by using superposition.

(Continued on next page)

so that

$$i' = 79.23 \cos(5t - 82.03^\circ) \text{ mA}$$

Likewise, we find that

$$\begin{aligned} \mathbf{I}'' &= 5\angle 0^\circ \left[ \frac{-j1.667}{10 - j0.6667 - j1.667} \right] \\ &= 811.7\angle -76.86^\circ \text{ mA} \end{aligned}$$

so that

$$i'' = 811.7 \cos(3t - 76.86^\circ) \text{ mA}$$

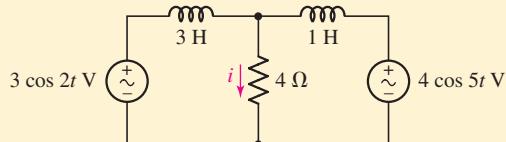


It should be noted at this point that no matter how tempted we might be to add the two phasor currents  $\mathbf{I}'$  and  $\mathbf{I}''$ , in Fig. 10.33b and c, this would be incorrect. Our next step is to add the two time-domain currents, square the result, and multiply by 10 to obtain the power absorbed by the  $10\Omega$  resistor in Fig. 10.33a:

$$\begin{aligned} p_{10} &= (i' + i'')^2 \times 10 \\ &= 10[79.23 \cos(5t - 82.03^\circ) + 811.7 \cos(3t - 76.86^\circ)]^2 \mu\text{W} \end{aligned}$$

### PRACTICE

10.16 Determine the current  $i$  through the  $4\Omega$  resistor of Fig. 10.34.



■ FIGURE 10.34

Ans:  $i = 175.6 \cos(2t - 20.55^\circ) + 547.1 \cos(5t - 43.16^\circ)$  mA.

### COMPUTER-AIDED ANALYSIS

We have several options in PSpice for the analysis of circuits in the sinusoidal steady state. Perhaps the most straightforward approach is to make use of two specially designed sources: VAC and IAC. The magnitude and phase of either source is selected by double-clicking on the part.

Let's simulate the circuit of Fig. 10.20a, shown redrawn in Fig. 10.35.

The frequency of the source is not selected through the Property Editor, but rather through the ac sweep analysis dialog box. This is accomplished by choosing AC Sweep/Noise for Analysis when presented with the Simulation Settings window. We select a Linear sweep

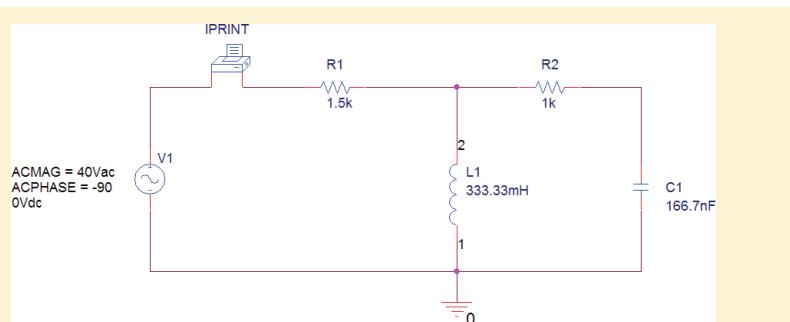


FIGURE 10.35 The circuit of Fig. 10.20a, operating at  $\omega = 3000$  rad/s. The current through the  $1.5\text{ k}\Omega$  resistor is desired.

and set **Total Points** to 1. Since we are only interested in the frequency of 3000 rad/s (477.5 Hz), we set both **Start Frequency** and **End Frequency** to 477.5 as shown in Fig. 10.36.

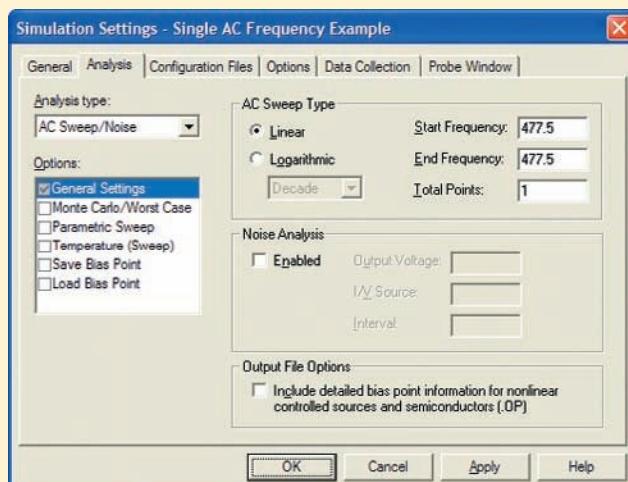


FIGURE 10.36 Dialog box for setting source frequency.

Note that an additional “component” appears in the schematic. This component is called IPRINT, and allows a variety of current parameters to be printed. In this simulation, we are interested in the **AC**, **MAG**, and **PHASE** attributes. In order for PSpice to print these quantities, double-click on the IPRINT symbol in the schematic, and enter *yes* in each of the appropriate fields.

The simulation results are obtained by choosing **View Output File** under **PSpice** in the Capture CIS window.

```
FREQ          IM(V_PRINT1)   IP(V_PRINT1)
4.775E+02    1.600E-02     -1.269E+02
```

Thus, the current magnitude is 16 mA, and the phase angle is  $-126.9^\circ$ , so that the current through the  $1.5\text{ k}\Omega$  resistor is

$$\begin{aligned} i &= 16 \cos(3000t - 126.9^\circ) \quad \text{mA} \\ &= 16 \sin(3000t - 36.9^\circ) \quad \text{mA} \end{aligned}$$

## 10.8 PHASOR DIAGRAMS

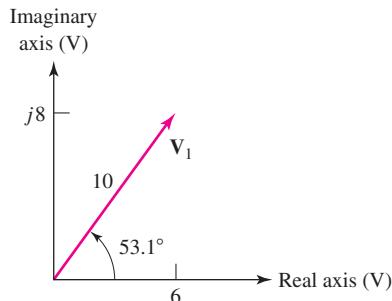


FIGURE 10.37 A simple phasor diagram shows the single voltage phasor  $V_1 = 6 + j8 = 10\angle 53.1^\circ$  V.

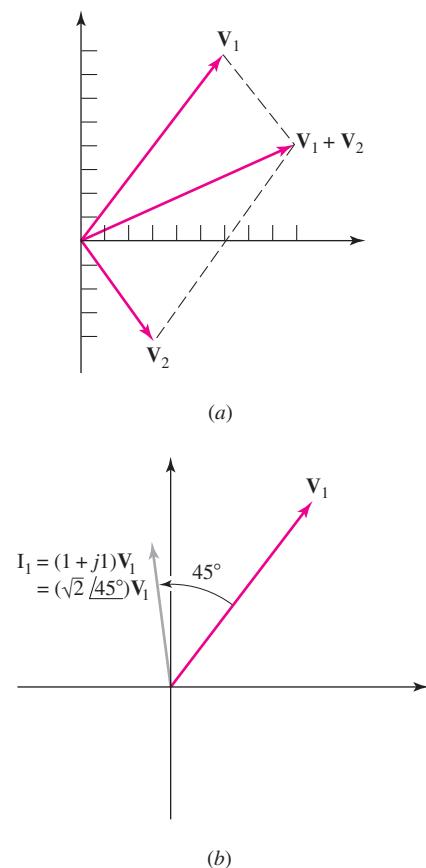


FIGURE 10.38 (a) A phasor diagram showing the sum of  $V_1 = 6 + j8$  V and  $V_2 = 3 - j4$  V,  $V_1 + V_2 = 9 + j4$  V =  $9.85\angle 24.0^\circ$  V. (b) The phasor diagram shows  $V_1$  and  $I_1$ , where  $I_1 = YV_1$  and  $Y = (1 + j1)$  S =  $\sqrt{2}\angle 45^\circ$  S. The current and voltage amplitude scales are different.

The *phasor diagram* is a name given to a sketch in the complex plane showing the relationships of the phasor voltages and phasor currents throughout a specific circuit. We are already familiar with the use of the complex plane in the graphical identification of complex numbers and in their addition and subtraction. Since phasor voltages and currents are complex numbers, they may also be identified as points in a complex plane. For example, the phasor voltage  $V_1 = 6 + j8 = 10\angle 53.1^\circ$  V is identified on the complex voltage plane shown in Fig. 10.37. The  $x$  axis is the real voltage axis, and the  $y$  axis is the imaginary voltage axis; the voltage  $V_1$  is located by an arrow drawn from the origin. Since addition and subtraction are particularly easy to perform and display on a complex plane, phasors may be easily added and subtracted in a phasor diagram. Multiplication and division result in the addition and subtraction of angles and a change of amplitude. Figure 10.38a shows the sum of  $V_1$  and a second phasor voltage  $V_2 = 3 - j4 = 5\angle -53.1^\circ$  V, and Fig. 10.38b shows the current  $I_1$ , which is the product of  $V_1$  and the admittance  $Y = 1 + j1$  S.

This last phasor diagram shows both current and voltage phasors on the same complex plane; it is understood that each will have its own amplitude scale, but a common angle scale. For example, a phasor voltage 1 cm long might represent 100 V, while a phasor current 1 cm long could indicate 3 mA. Plotting both phasors on the same diagram enables us to easily determine which waveform is leading and which is lagging.

The phasor diagram also offers an interesting interpretation of the time-domain to frequency-domain transformation, since the diagram may be interpreted from either the time- or the frequency-domain viewpoint. Up to this point, we have been using the frequency-domain interpretation, as we have been showing phasors directly on the phasor diagram. However, let us proceed to a time-domain viewpoint by first showing the phasor voltage  $V = V_m \angle \alpha$  as sketched in Fig. 10.39a. In order to transform  $V$  to the time domain, the next necessary step is the multiplication of the phasor by  $e^{j\omega t}$ ; thus we now have the complex voltage  $V_m e^{j\alpha} e^{j\omega t} = V_m \angle \omega t + \alpha$ . This voltage may also be interpreted as a phasor, one which possesses a phase angle that increases linearly with time. On a phasor diagram it therefore represents a rotating line segment, the instantaneous position being  $\omega t$  radians ahead (counterclockwise) of  $V_m \angle \alpha$ . Both  $V_m \angle \alpha$  and  $V_m \angle \omega t + \alpha$  are shown on the phasor diagram of Fig. 10.39b. The passage to the time

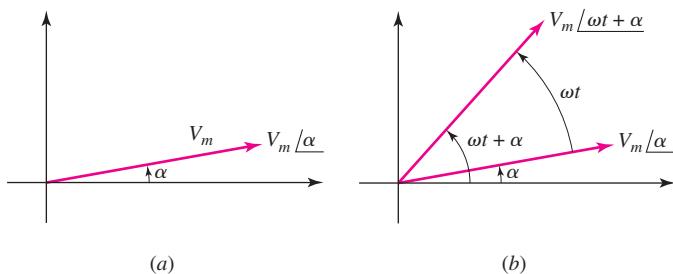


FIGURE 10.39 (a) The phasor voltage  $V_m \angle \alpha$ . (b) The complex voltage  $V_m \angle \omega t + \alpha$  is shown as a phasor at a particular instant of time. This phasor leads  $V_m \angle \alpha$  by  $\omega t$  radians.

domain is now completed by taking the real part of  $V_m/\omega t + \alpha$ . The real part of this complex quantity is the projection of  $V_m/\omega t + \alpha$  on the real axis:  $V_m \cos(\omega t + \alpha)$ .

In summary, then, the frequency-domain phasor appears on the phasor diagram, and the transformation to the time domain is accomplished by allowing the phasor to rotate in a counterclockwise direction at an angular velocity of  $\omega$  rad/s and then visualizing the projection on the real axis. It is helpful to think of the arrow representing the phasor  $\mathbf{V}$  on the phasor diagram as the photographic snapshot, taken at  $\omega t = 0$ , of a rotating arrow whose projection on the real axis is the instantaneous voltage  $v(t)$ .

Let us now construct the phasor diagrams for several simple circuits. The series RLC circuit shown in Fig. 10.40a has several different voltages associated with it, but only a single current. The phasor diagram is constructed most easily by employing the single current as the reference phasor. Let us arbitrarily select  $\mathbf{I} = I_m/0^\circ$  and place it along the real axis of the phasor diagram, Fig. 10.40b. The resistor, capacitor, and inductor voltages may then be calculated and placed on the diagram, where the  $90^\circ$  phase relationships stand out clearly. The sum of these three voltages is the source voltage, and for this circuit, which is in what we will define in a subsequent chapter as the “resonant condition” since  $Z_C = -Z_L$ , the source voltage and resistor voltage are equal. The total voltage across the resistor and inductor or resistor and capacitor is obtained from the diagram by adding the appropriate phasors as shown.

Figure 10.41a is a simple parallel circuit in which it is logical to use the single voltage between the two nodes as a reference phasor. Suppose that  $\mathbf{V} = 1/0^\circ$  V. The resistor current,  $\mathbf{I}_R = 0.2/0^\circ$  A, is in phase with this voltage, and the capacitor current,  $\mathbf{I}_C = j0.1$  A, leads the reference voltage by  $90^\circ$ . After these two currents are added to the phasor diagram, shown as Fig. 10.41b, they may be summed to obtain the source current. The result is  $\mathbf{I}_s = 0.2 + j0.1$  A.

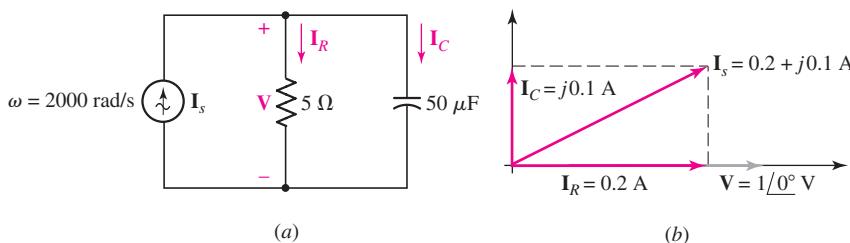


FIGURE 10.41 (a) A parallel RC circuit. (b) The phasor diagram for this circuit; the node voltage  $\mathbf{V}$  is used as a convenient reference phasor.

If the source current is specified initially as the convenient value of  $1/0^\circ$  A and the node voltage is not initially known, it is still convenient to begin construction of the phasor diagram by assuming a node voltage (for example,  $\mathbf{V} = 1/0^\circ$  V once again) and using it as the reference phasor. The diagram is then completed as before, and the source current that flows as a result of the assumed node voltage is again found to be  $0.2 + j0.1$  A. The true source current is  $1/0^\circ$  A, however, and thus the true node voltage is obtained by multiplying the assumed node voltage by  $1/0^\circ/(0.2 + j0.1)$ ;

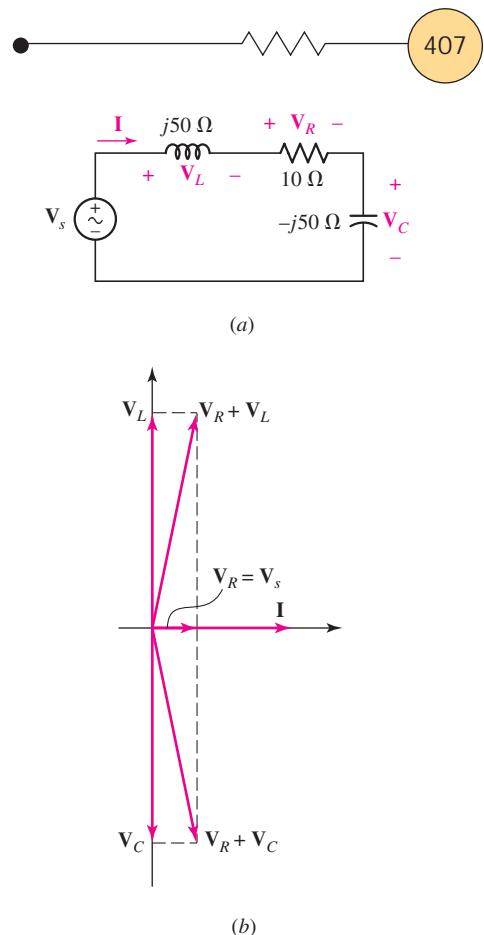


FIGURE 10.40 (a) A series RLC circuit. (b) The phasor diagram for this circuit; the current  $\mathbf{I}$  is used as a convenient reference phasor.

the true node voltage is therefore  $4 - j2 \text{ V} = \sqrt{20}/-26.6^\circ \text{ V}$ . The assumed voltage leads to a phasor diagram which differs from the true phasor diagram by a change of scale (the assumed diagram is smaller by a factor of  $1/\sqrt{20}$ ) and an angular rotation (the assumed diagram is rotated counterclockwise through  $26.6^\circ$ ).

### EXAMPLE 10.13

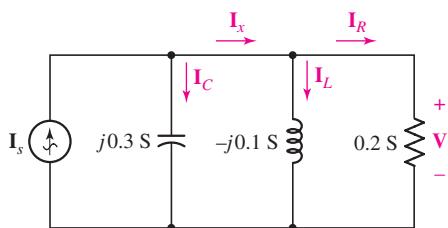


FIGURE 10.42 A simple circuit for which several currents are required.

**Construct a phasor diagram showing  $\mathbf{I}_R$ ,  $\mathbf{I}_L$ , and  $\mathbf{I}_C$  for the circuit of Fig. 10.42. Combining these currents, determine the angle by which  $\mathbf{I}_s$  leads  $\mathbf{I}_R$ ,  $\mathbf{I}_C$ , and  $\mathbf{I}_x$ .**

We begin by choosing a suitable reference phasor. Upon examining the circuit and the variables to be determined, we see that once  $\mathbf{V}$  is known,  $\mathbf{I}_R$ ,  $\mathbf{I}_L$ , and  $\mathbf{I}_C$  can be computed by simple application of Ohm's law. Thus, we select  $\mathbf{V} = 1/0^\circ \text{ V}$  for simplicity's sake, and subsequently compute

$$\mathbf{I}_R = (0.2)1/0^\circ = 0.2/0^\circ \text{ A}$$

$$\mathbf{I}_L = (-j0.1)1/0^\circ = 0.1/-90^\circ \text{ A}$$

$$\mathbf{I}_C = (j0.3)1/0^\circ = 0.3/90^\circ \text{ A}$$

The corresponding phasor diagram is shown in Fig. 10.43a. We also need to find the phasor currents  $\mathbf{I}_s$  and  $\mathbf{I}_x$ . Figure 10.43b shows the determination of  $\mathbf{I}_x = \mathbf{I}_L + \mathbf{I}_R = 0.2 - j0.1 = 0.224/-26.6^\circ \text{ A}$ , and Fig. 10.43c shows the determination of  $\mathbf{I}_s = \mathbf{I}_C + \mathbf{I}_x = 0.283/45^\circ \text{ A}$ . From Fig. 10.43c, we ascertain that  $\mathbf{I}_s$  leads  $\mathbf{I}_R$  by  $45^\circ$ ,  $\mathbf{I}_C$  by  $-45^\circ$ , and  $\mathbf{I}_x$  by  $45^\circ + 26.6^\circ = 71.6^\circ$ . These angles are only relative, however; the exact numerical values will depend on  $\mathbf{I}_s$ , upon which the actual value of  $\mathbf{V}$  (assumed here to be  $1/0^\circ \text{ V}$  for convenience) also depends.

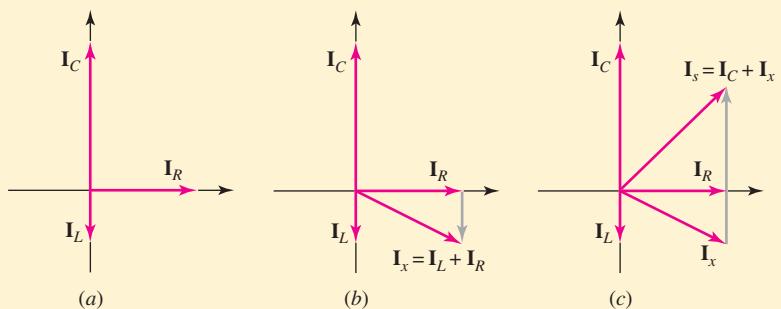


FIGURE 10.43 (a) Phasor diagram constructed using a reference value of  $\mathbf{V} = 1/0^\circ$ .  
(b) Graphical determination of  $\mathbf{I}_x = \mathbf{I}_L + \mathbf{I}_R$ . (c) Graphical determination of  $\mathbf{I}_s = \mathbf{I}_C + \mathbf{I}_x$ .

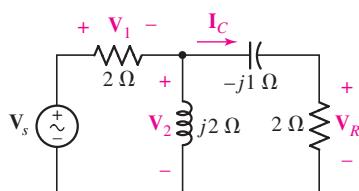


FIGURE 10.44

### PRACTICE

- 10.17 Select some convenient reference value for  $\mathbf{I}_C$  in the circuit of Fig. 10.44; draw a phasor diagram showing  $\mathbf{V}_R$ ,  $\mathbf{V}_2$ ,  $\mathbf{V}_1$ , and  $\mathbf{V}_s$ ; and measure the ratio of the lengths of (a)  $\mathbf{V}_s$  to  $\mathbf{V}_1$ ; (b)  $\mathbf{V}_1$  to  $\mathbf{V}_2$ ; (c)  $\mathbf{V}_s$  to  $\mathbf{V}_R$ .

Ans: 1.90; 1.00; 2.12

## SUMMARY AND REVIEW

This chapter dealt with the steady-state response of circuits to sinusoidal excitation. This is a limited analysis of a circuit in some respects, as the transient behavior is completely ignored. In many situations, such an approach is more than adequate, and reducing the amount of information we seek about a circuit speeds up the analysis considerably. The fundamental idea behind what we did was that an *imaginary* source was added to every *real* sinusoidal source; then Euler's identity converted the source to a complex exponential. Since the derivative of an exponential is simply another exponential, what would otherwise be integrodifferential equations arising from mesh or nodal analysis become *algebraic equations*.

A few new terms were introduced: *lagging*, *leading*, *impedance*, *admittance*, and a particularly important one *phasor*. Phasor relationships between current and voltage gave rise to the concept of impedance, where resistors are represented by a real number (resistance, as before), and inductors are represented by  $Z = j\omega L$  while capacitors are represented by  $-j/\omega C$  ( $\omega$  being the operating frequency of our sources). From that point forward, all the circuit analysis techniques learned in Chaps. 3 to 5 apply.

It might seem odd to have an imaginary number as part of our solution, but we found that recovering the time-domain solution to our analysis is straightforward once the voltage or current is expressed in polar form. The magnitude of our quantity of interest is the magnitude of the cosine function, the phase angle is the phase of the cosine term, and the frequency is obtained from the original circuit (it disappears from view during the analysis, but the circuits we are analyzing do not change it in any way). We concluded the chapter with an introduction to the concept of phasor diagrams. Prior to inexpensive scientific calculators such tools were invaluable in analyzing many sinusoidal circuits. They still find use in analysis of ac power systems, as we see in subsequent chapters.

A concise list of key concepts of the chapter is presented below for the convenience of the reader, along with the corresponding example numbers.

- ❑ If two sine waves (or two cosine waves) both have positive magnitudes and the same frequency, it is possible to determine which waveform is leading and which is lagging by comparing their phase angles.
- ❑ The forced response of a linear circuit to a sinusoidal voltage or current source can always be written as a single sinusoid having the same frequency as the sinusoidal source. (Example 10.1)
- ❑ A phasor has both a magnitude and a phase angle; the frequency is understood to be that of the sinusoidal source driving the circuit. (Example 10.2)
- ❑ A phasor transform may be performed on any sinusoidal function, and vice versa:  $V_m \cos(\omega t + \phi) \leftrightarrow V_m \angle \phi$ . (Example 10.3)
- ❑ When transforming a time-domain circuit into the corresponding frequency-domain circuit, resistors, capacitors, and inductors are replaced by impedances (or, occasionally, by admittances). (Examples 10.4, 10.6)
  - The impedance of a resistor is simply its resistance.

- The impedance of a capacitor is  $1/j\omega C \Omega$ .
  - The impedance of an inductor is  $j\omega L \Omega$ .
- Impedances combine both in series and in parallel combinations in the same manner as resistors. (Example 10.6)
- All analysis techniques previously used on resistive circuits apply to circuits with capacitors and/or inductors once all elements are replaced by their frequency-domain equivalents. (Examples 10.5, 10.7, 10.8, 10.9, 10.10, 10.11)
- Phasor analysis can only be performed on single-frequency circuits. Otherwise, superposition must be invoked, and the *time-domain* partial responses added to obtain the complete response. (Example 10.12)
- The power behind phasor diagrams is evident when a convenient forcing function is used initially, and the final result scaled appropriately. (Example 10.13)

## READING FURTHER

A good reference to phasor-based analysis techniques can be found in:

R. A. DeCarlo and P. M. Lin, *Linear Circuit Analysis*, 2nd ed. New York: Oxford University Press, 2001.

Frequency-dependent transistor models are discussed from a phasor perspective in Chap. 7 of:

W. H. Hayt, Jr., and G. W. Neudeck, *Electronic Circuit Analysis and Design*, 2nd ed. New York: Wiley, 1995.

## EXERCISES

### 10.1 Characteristics of Sinusoids

1. Evaluate the following: (a)  $5 \sin(5t - 9^\circ)$  at  $t = 0, 0.01$ , and  $0.1$  s; (b)  $4 \cos 2t$  and  $4 \sin(2t + 90^\circ)$  at  $t = 0, 1$ , and  $1.5$  s; (c)  $3.2 \cos(6t + 15^\circ)$  and  $3.2 \sin(6t + 105^\circ)$  at  $t = 0, 0.01$ , and  $0.1$  s.
2. (a) Express each of the following as a single *cosine* function:  $5 \sin 300t$ ,  $1.95 \sin(\pi t - 92^\circ)$ ,  $2.7 \sin(50t + 5^\circ) - 10 \cos 50t$ . (b) Express each of the following as a single *sine* function:  $66 \cos(9t - 10^\circ)$ ,  $4.15 \cos 10t$ ,  $10 \cos(100t - 9^\circ) + 10 \sin(100t + 19^\circ)$ .
3. Determine the angle by which  $v_1$  leads  $i_1$  if  $v_1 = 10 \cos(10t - 45^\circ)$  and  $i_1$  is equal to (a)  $5 \cos 10t$ ; (b)  $5 \cos(10t - 80^\circ)$ ; (c)  $5 \cos(10t - 40^\circ)$ ; (d)  $5 \cos(10t + 40^\circ)$ ; (e)  $5 \sin(10t - 19^\circ)$ .
4. Determine the angle by which  $v_1$  lags  $i_1$  if  $v_1 = 34 \cos(10t + 125^\circ)$  and  $i_1$  is equal to (a)  $5 \cos 10t$ ; (b)  $5 \cos(10t - 80^\circ)$ ; (c)  $5 \cos(10t - 40^\circ)$ ; (d)  $5 \cos(10t + 40^\circ)$ ; (e)  $5 \sin(10t - 19^\circ)$ .
5. Determine which waveform in each of the following pairs is lagging: (a)  $\cos 4t$ ,  $\sin 4t$ ; (b)  $\cos(4t - 80^\circ)$ ,  $\cos(4t)$ ; (c)  $\cos(4t + 80^\circ)$ ,  $\cos 4t$ ; (d)  $-\sin 5t$ ,  $\cos(5t + 2^\circ)$ ; (e)  $\sin 5t + \cos 5t$ ,  $\cos(5t - 45^\circ)$ .
6. Calculate the first three instants in time ( $t > 0$ ) for which the following functions are zero, by first converting to a single sinusoid: (a)  $\cos 3t - 7 \sin 3t$ ; (b)  $\cos(10t + 45^\circ)$ ; (c)  $\cos 5t - \sin 5t$ ; (d)  $\cos 2t + \sin 2t - \cos 5t + \sin 5t$ .
7. (a) Determine the first two instants in time ( $t > 0$ ) for which each of the functions in Exercise 6 are equal to unity, by first converting to a single sinusoid. (b) Verify your answers by plotting each waveform using an appropriate software package.



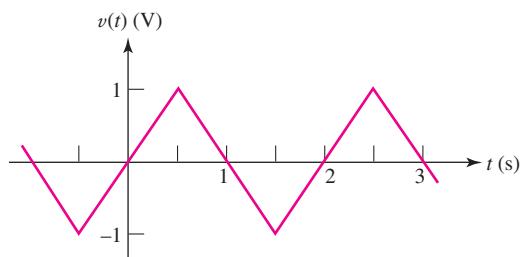


8. The concept of Fourier series is a powerful means of analyzing periodic waveforms in terms of sinusoids. For example, the triangle wave in Fig. 10.45 can be represented by the infinite sum

$$v(t) = \frac{8}{\pi^2} \left( \sin \pi t - \frac{1}{3^2} \sin 3\pi t + \frac{1}{5^2} \sin 5\pi t - \frac{1}{7^2} \sin 7\pi t + \dots \right)$$

where in practice perhaps the first several terms provide an accurate enough approximation. (a) Compute the exact value of  $v(t)$  at  $t = 0.25$  s by first obtaining an equation for the corresponding segment of the waveform.

(b) Compute the approximate value at  $t = 0.25$  s using the first term of the Fourier series only. (c) Repeat part (b) using the first three terms. (d) Plot  $v(t)$  using the first term only. (e) Plot  $v(t)$  using the first two terms only. (f) Plot  $v(t)$  using the first three terms only.



■ FIGURE 10.45

9. Household electrical voltages are typically quoted as either 110 V, 115 V, or 120 V. However, these values do not represent the peak ac voltage. Rather, they represent what is known as the root mean square of the voltage, defined as

$$V_{\text{rms}} = \sqrt{\frac{1}{T} \int_0^T V_m^2 \cos^2(\omega t) dt}$$

where  $T$  = the period of the waveform,  $V_m$  is the peak voltage, and  $\omega$  = the waveform frequency ( $f = 60$  Hz in North America).

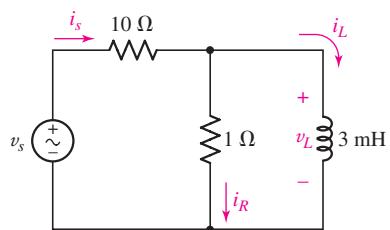
(a) Perform the indicated integration, and show that for a sinusoidal voltage,

$$V_{\text{rms}} = \frac{V_m}{\sqrt{2}}$$

(b) Compute the peak voltages corresponding to the rms voltages of 110, 115, and 120 V.

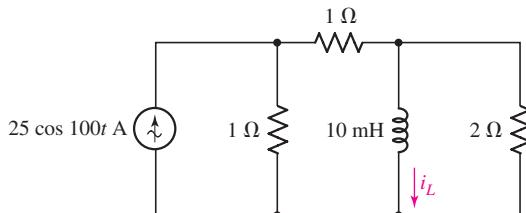
## 10.2 Forced Response to Sinusoidal Functions

10. If the source  $v_s$  in Fig. 10.46 is equal to  $4.53 \cos(0.333 \times 10^{-3}t + 30^\circ)$  V, (a) obtain  $i_s$ ,  $i_L$ , and  $i_R$  at  $t = 0$  assuming no transients are present; (b) obtain an expression for  $v_L(t)$  in terms of a single sinusoid, valid for  $t > 0$ , again assuming no transients are present.

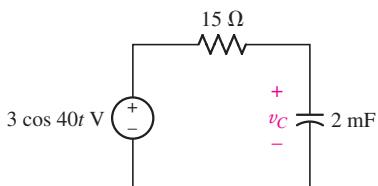


■ FIGURE 10.46

11. Assuming there are no longer any transients present, determine the current labeled  $i_L$  in the circuit of Fig. 10.47. Express your answer as a single sinusoid.

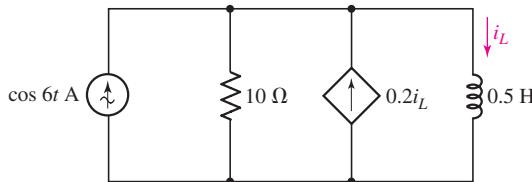


■ FIGURE 10.47



■ FIGURE 10.48

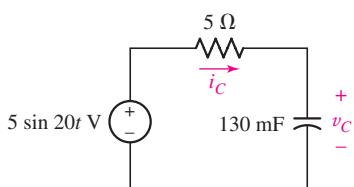
12. Calculate the power dissipated in the  $2 \Omega$  resistor of Fig. 10.47 assuming there are no transients present. Express your answer in terms of a single sinusoidal function.
13. Obtain an expression for  $v_C$  as labeled in Fig. 10.48, in terms of a single sinusoidal function. You may assume all transients have died out long before  $t = 0$ .
14. Calculate the energy stored in the capacitor of the circuit depicted in Fig. 10.48 at  $t = 10 \text{ ms}$  and  $t = 40 \text{ ms}$ .
15. Obtain an expression for the power dissipated in the  $10 \Omega$  resistor of Fig. 10.49, assuming no transients present.



■ FIGURE 10.49

### 10.3 The Complex Forcing Function

16. Express the following complex numbers in rectangular form: (a)  $50\angle-75^\circ$ ; (b)  $19e^{j30^\circ}$ ;  $2.5\angle-30^\circ + 0.5\angle45^\circ$ . Convert the following to polar form: (c)  $(2+j2)(2-j2)$ ; (d)  $(2+j2)(5\angle22^\circ)$ .
17. Express the following in polar form: (a)  $2 + e^{j35^\circ}$ ; (b)  $(j)(j)(-j)$ ; (c) 1. Express the following in rectangular form: (d)  $2 + e^{j35^\circ}$ ; (e)  $-j9 + 5\angle55^\circ$ .
18. Evaluate the following, and express your answer in polar form:  
 (a)  $4(8 - j8)$ ; (b)  $4\angle5^\circ - 2\angle15^\circ$ ; (c)  $(2 + j9) - 5\angle0^\circ$ ; (d)  $\frac{-j}{10 + 5j} - 3\angle40^\circ + 2$ ; (e)  $(10 + j5)(10 - j5)(3\angle40^\circ) + 2$ .
19. Evaluate the following, and express your answer in rectangular form:  
 (a)  $3(3\angle30^\circ)$ ; (b)  $2\angle25^\circ + 5\angle-10^\circ$ ; (c)  $(12 + j90) - 5\angle30^\circ$ ; (d)  $\frac{10 + 5j}{8 - j} + 2\angle60^\circ + 1$ ; (e)  $(10 + 5j)(10 - 5j)(3\angle40^\circ) + 2$ .
20. Perform the indicated operations, and express the answer in both rectangular and polar forms:  
 (a)  $\frac{2 + j3}{1 + 8\angle90^\circ} - 4$ ; (b)  $\left(\frac{10\angle25^\circ}{5\angle-10^\circ} + \frac{3\angle15^\circ}{3 - j5}\right)j2$ ; (c)  $\left[\frac{(1-j)(1+j) + 1\angle0^\circ}{-j}\right](3\angle-90^\circ) + \frac{j}{5\angle-45^\circ}$ .
21. Insert an appropriate complex source into the circuit represented in Fig. 10.50, and use it to determine steady-state expressions for  $i_C(t)$  and  $v_C(t)$ .



■ FIGURE 10.50

22. For the circuit of Fig. 10.51, if  $i_s = 5 \cos 10t$  A, use a suitable complex source replacement to obtain a steady-state expression for  $i_L(t)$ .
23. In the circuit depicted in Fig. 10.51,  $i_s$  is modified such that the  $2\ \Omega$  resistor is replaced by a  $20\ \Omega$  resistor. If  $i_L(t) = 62.5 \angle 31.3^\circ$  mA, determine  $i_s$ .
24. Employ a suitable complex source to determine the steady-state current  $i_L$  in the circuit of Fig. 10.52.

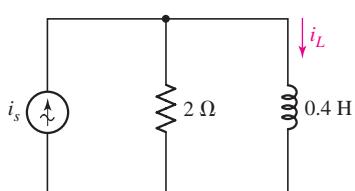


FIGURE 10.51

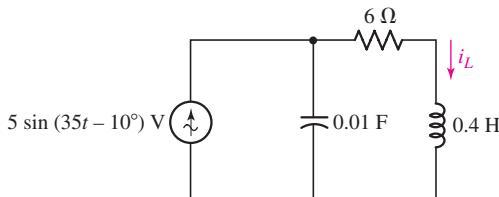


FIGURE 10.52

## 10.4 The Phasor

25. Transform each of the following into phasor form: (a)  $75.928 \cos(110.1t)$ ; (b)  $5 \cos(55t - 42^\circ)$ ; (c)  $-\sin(8000t + 14^\circ)$ ; (d)  $3 \cos 10t - 8 \cos(10t + 80^\circ)$ .
26. Transform each of the following into phasor form: (a)  $11 \sin 100t$ ; (b)  $11 \cos 100t$ ; (c)  $11 \cos(100t - 90^\circ)$ ; (d)  $3 \cos 100t - 3 \sin 100t$ .
27. Assuming an operating frequency of 1 kHz, transform the following phasor expressions into a single cosine function in the time domain: (a)  $9 \angle 65^\circ$  V; (b)  $\frac{2 \angle 31^\circ}{4 \angle 25^\circ}$  A; (c)  $22 \angle 14^\circ - 8 \angle 33^\circ$  V.
28. The following complex voltages are written in a combination of rectangular and polar form. Rewrite each, using conventional phasor notation (i.e., a magnitude and angle): (a)  $\frac{2-j}{5 \angle 45^\circ} \text{ V}$ ; (b)  $\frac{6 \angle 20^\circ}{1000} - j \text{ V}$ ; (c)  $(j)(52.5 \angle -90^\circ) \text{ V}$ .
29. Assuming an operating frequency of 50 Hz, compute the instantaneous voltage at  $t = 10$  ms and  $t = 25$  ms for each of the quantities represented in Exercise 26.
30. Assuming an operating frequency of 50 Hz, compute the instantaneous voltage at  $t = 10$  ms and  $t = 25$  ms for each of the quantities represented in Exercise 27.
31. Assuming the passive sign convention and an operating frequency of 5 rad/s, calculate the phasor voltage which develops across the following when driven by the phasor current  $\mathbf{I} = 2 \angle 0^\circ$  mA: (a) a  $1\ \text{k}\Omega$  resistor; (b) a  $1\ \text{mF}$  capacitor; (c) a  $1\ \text{nH}$  inductor.
32. (a) A series connection is formed between a  $1\ \Omega$  resistor, a  $1\ \text{F}$  capacitor, and a  $1\ \text{H}$  inductor, in that order. Assuming operation at  $\omega = 1$  rad/s, what are the magnitude and phase angle of the phasor current which yields a voltage of  $1 \angle 30^\circ$  V across the resistor (assume the passive sign convention)? (b) Compute the ratio of the phasor voltage across the resistor to the phasor voltage which appears across the capacitor-inductor combination. (c) The frequency is doubled. Calculate the new ratio of the phasor voltage across the resistor to the phasor voltage across the capacitor-inductor combination.
33. Assuming the passive sign convention and an operating frequency of  $314$  rad/s, calculate the phasor voltage  $\mathbf{V}$  which appears across each of the following when driven by the phasor current  $\mathbf{I} = 10 \angle 0^\circ$  mA: (a) a  $2\ \Omega$  resistor; (b) a  $1\ \text{F}$  capacitor; (c) a  $1\ \text{H}$  inductor; (d) a  $2\ \Omega$  resistor in series with a  $1\ \text{F}$  capacitor; (e) a  $2\ \Omega$  resistor in series with a  $1\ \text{H}$  inductor. (f) Calculate the instantaneous value of each voltage determined in parts (a) to (e) at  $t = 0$ .
34. In the circuit of Fig. 10.53, which is shown in the phasor (frequency) domain,  $\mathbf{I}_{10}$  is determined to be  $2 \angle 42^\circ$  mA. If  $\mathbf{V} = 40 \angle 132^\circ$  mV: (a) what is the likely type of element connected to the right of the  $10\ \Omega$  resistor and (b) what is its value, assuming the voltage source operates at a frequency of  $1000$  rad/s?

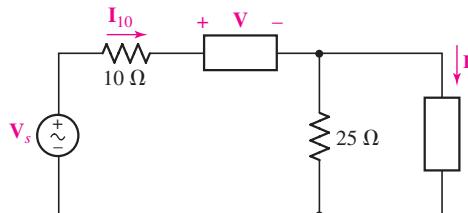


FIGURE 10.53

35. The circuit of Fig. 10.53 is shown represented in the phasor (frequency) domain. If  $I_{10} = 4\angle 35^\circ \text{ A}$ ,  $V = 10\angle 35^\circ$ , and  $I = 2\angle 35^\circ \text{ A}$ , (a) across what type of element does  $V$  appear, and what is its value? (b) Determine the value of  $V_s$ .

## 10.5 Impedance and Admittance



36. (a) Obtain an expression for the equivalent impedance  $Z_{eq}$  of a  $1 \Omega$  resistor in series with a  $10 \text{ mH}$  inductor as a function of  $\omega$ . (b) Plot the magnitude of  $Z_{eq}$  as a function of  $\omega$  over the range  $1 < \omega < 100 \text{ rad/s}$  (use a logarithmic scale for the frequency axis). (c) Plot the angle (in degrees) of  $Z_{eq}$  as a function of  $\omega$  over the range  $1 < \omega < 100 \text{ rad/s}$  (use a logarithmic scale for the frequency axis). [Hint: *semilogx* in MATLAB is a useful plotting function.]



37. Determine the equivalent impedance of the following, assuming an operating frequency of  $20 \text{ rad/s}$ : (a)  $1 \text{ k}\Omega$  in series with  $1 \text{ mF}$ ; (b)  $1 \text{ k}\Omega$  in parallel with  $1 \text{ mH}$ ; (c)  $1 \text{ k}\Omega$  in parallel with the series combination of  $1 \text{ F}$  and  $1 \text{ H}$ .



38. (a) Obtain an expression for the equivalent impedance  $Z_{eq}$  of a  $1 \Omega$  resistor in series with a  $10 \text{ mF}$  capacitor as a function of  $\omega$ . (b) Plot the magnitude of  $Z_{eq}$  as a function of  $\omega$  over the range  $1 < \omega < 100 \text{ rad/s}$  (use a logarithmic scale for the frequency axis). (c) Plot the angle (in degrees) of  $Z_{eq}$  as a function of  $\omega$  over the range  $1 < \omega < 100 \text{ rad/s}$  (use a logarithmic scale for the frequency axis). [Hint: *semilogx* in MATLAB is a useful plotting function.]

39. Determine the equivalent admittance of the following, assuming an operating frequency of  $1000 \text{ rad/s}$ : (a)  $25 \Omega$  in series with  $20 \text{ mH}$ ; (b)  $25 \Omega$  in parallel with  $20 \text{ mH}$ ; (c)  $25 \Omega$  in parallel with  $20 \text{ mH}$  in parallel with  $20 \text{ mF}$ ; (d)  $1 \Omega$  in series with  $1 \text{ F}$  in series with  $1 \text{ H}$ ; (e)  $1 \Omega$  in parallel with  $1 \text{ F}$  in parallel with  $1 \text{ H}$ .

40. Consider the network depicted in Fig. 10.54, and determine the equivalent impedance seen looking into the open terminals if (a)  $\omega = 1 \text{ rad/s}$ ; (b)  $\omega = 10 \text{ rad/s}$ ; (c)  $\omega = 100 \text{ rad/s}$ .

41. Exchange the capacitor and inductor in the network shown in Fig. 10.54, and calculate the equivalent impedance looking into the open terminals if  $\omega = 25 \text{ rad/s}$ .

42. Find  $V$  in Fig. 10.55 if the box contains (a)  $3 \Omega$  in series with  $2 \text{ mH}$ ; (b)  $3 \Omega$  in series with  $125 \mu\text{F}$ ; (c)  $3 \Omega$ ,  $2 \text{ mH}$ , and  $125 \mu\text{F}$  in series; (d)  $3 \Omega$ ,  $2 \text{ mH}$ , and  $125 \mu\text{F}$  in series, but  $\omega = 4 \text{ krad/s}$ .

43. Calculate the equivalent impedance seen at the open terminals of the network shown in Fig. 10.56 if  $f$  is equal to (a)  $1 \text{ Hz}$ ; (b)  $1 \text{ kHz}$ ; (c)  $1 \text{ MHz}$ ; (d)  $1 \text{ GHz}$ ; (e)  $1 \text{ THz}$ .

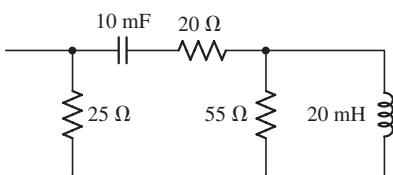


FIGURE 10.54

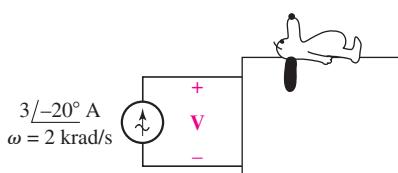


FIGURE 10.55

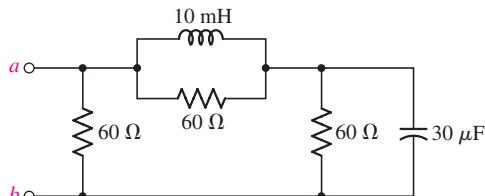


FIGURE 10.56